## SIEMENS

## Preface

## Fundamental safety instructions

## SINUMERIK

Introduction

## SINUMERIK 840D sI/828D Turning

## Operating Manual

## Valid for:

Controllers
SINUMERIK 840D sl / 840DE sl / 828D
Software version
Software CNC system for 840D sl/ 840DE sl SINUMERIK Operate for PCU/PC

V4.7 SP1 V4.7 SP1

Multi-channel machining

| Collision avoidance <br> (only 840D sl) | 11 |
| :--- | :--- | (only 840D sl)

Tool management

Managing programs
Alarm, error and system
messages messages

Working with Manual Machine

| Working with a B axis <br> (only 840 D sl) |
| :--- |

Working with two tool 17
carriers

# SINUMERIK 840D sI/828D Turning 

Teaching in a program

## Operating Manual

Ctrl-Energy 20

21
Easy Extend (828D only) 22
Edit PLC user program ..... 24

## Legal information

## Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

DANGER
indicates that death or severe personal injury will result if proper precautions are not taken.

## WARNING

indicates that death or severe personal injury may result if proper precautions are not taken.

```
\CAUTION
indicates that minor personal injury can result if proper precautions are not taken.
```


## NOTICE

indicates that property damage can result if proper precautions are not taken.
If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

## Qualified Personnel

The product/system described in this documentation may be operated only by personnel qualified for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

## Proper use of Siemens products

Note the following:

## WARNING

Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

## Trademarks

All names identified by ${ }^{\circledR}$ are registered trademarks of Siemens AG. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

## Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

## Preface

## SINUMERIK documentation

The SINUMERIK documentation is organized in the following categories:

- General documentation
- User documentation
- Manufacturer/service documentation


## Additional information

You can find information on the following topics at www.siemens.com/motioncontrol/docu:

- Ordering documentation/overview of documentation
- Additional links to download documents
- Using documentation online (find and search in manuals/information)

Please send any questions about the technical documentation (e.g. suggestions for improvement, corrections) to the following address:
docu.motioncontrol@siemens.com

## My Documentation Manager (MDM)

Under the following link you will find information to individually compile OEM-specific machine documentation based on the Siemens content:
www.siemens.com/mdm

## Training

For information about the range of training courses, refer under:

- www.siemens.com/sitrain

SITRAIN - Siemens training for products, systems and solutions in automation technology

- www.siemens.com/sinutrain

SinuTrain - training software for SINUMERIK

FAQs
You can find Frequently Asked Questions in the Service\&Support pages under Product Support. http://support.automation.siemens.com

## SINUMERIK

You can find information on SINUMERIK under the following link: www.siemens.com/sinumerik

## Target group

This documentation is intended for users of turning machines running the SINUMERIK Operate software.

## Benefits

The operating manual helps users familiarize themselves with the control elements and commands. Guided by the manual, users are capable of responding to problems and taking corrective action.

## Standard scope

This documentation describes the functionality of the standard scope. Extensions or changes made by the machine manufacturer are documented by the machine manufacturer.

Other functions not described in this documentation might be executable in the control. However, no claim can be made regarding the availability of these functions when the equipment is first supplied or in the event of servicing.

Furthermore, for the sake of clarity, this documentation does not contain all detailed information about all types of the product and cannot cover every conceivable case of installation, operation or maintenance.

## Terms

The meanings of some basic terms used in this documentation are given below.

## Program

A program is a sequence of instructions to the CNC which combine to produce a specific workpiece on the machine.

## Contour

The term contour refers generally to the outline of a workpiece. More specifically, it refers to the section of the program that defines the outline of a workpiece comprising individual elements.

## Cycle

A cycle, such as the tapping cycle, is a subprogram defined in SINUMERIK Operate for executing a frequently repeated machining operation.

## Technical Support

You will find telephone numbers for other countries for technical support in the Internet under http://www.siemens.com/automation/service\&support

## Table of contents

Preface ..... 5
1 Fundamental safety instructions ..... 23
1.1 General safety instructions ..... 23
1.2 Industrial security ..... 24
2 Introduction ..... 25
2.1 Product overview ..... 25
2.2 Operator panel fronts ..... 26
2.2.1 Overview ..... 26
2.2.2 Keys of the operator panel ..... 27
2.3 Machine control panels ..... 37
2.3.1 Overview ..... 37
2.3.2 Controls on the machine control panel ..... 37
2.4 User interface ..... 41
2.4.1 Screen layout ..... 41
2.4.2 Status display ..... 42
2.4.3 Actual value window ..... 45
2.4.4 T,F,S window ..... 47
2.4.5 Current block display ..... 49
2.4.6 Operation via softkeys and buttons ..... 49
2.4.7 Entering or selecting parameters ..... 51
2.4.8 Pocket calculator ..... 53
2.4.9 Context menu ..... 54
2.4.10 Touch operation ..... 54
2.4.11 Changing the user interface language ..... 55
2.4.12 Entering Chinese characters ..... 55
2.4.12.1 Function - input editor ..... 55
2.4.12.2 Entering Asian characters ..... 57
2.4.12.3 Editing the dictionary ..... 58
2.4.13 Entering Korean characters ..... 60
2.4.14 Protection levels ..... 62
2.4.15 Online help in SINUMERIK Operate ..... 64
3 Setting up the machine ..... 67
$3.1 \quad$ Switching on and switching off. ..... 67
3.2 Approaching a reference point ..... 68
3.2.1 Referencing axes ..... 68
3.2.2 User agreement ..... 69
3.3 Modes and mode groups ..... 70
3.3.1 General ..... 70
3.3.2 Modes groups and channels ..... 72
3.3.3 Channel switchover ..... 73
3.4 Settings for the machine ..... 74
3.4.1 Switching over the coordinate system (MCS/WCS) ..... 74
3.4.2 Switching the unit of measurement ..... 75
3.4.3 Setting the zero offset ..... 76
3.5 Measuring the tool ..... 78
3.5.1 Measuring a tool manually ..... 78
3.5.2 Measuring a tool with a tool probe ..... 80
3.5.3 Calibrating the tool probe ..... 82
3.5.4 Measuring a tool with a magnifying glass ..... 83
3.5.5 Logging tool measurement results ..... 84
3.6 Measuring the workpiece zero ..... 85
3.6.1 Measuring the workpiece zero ..... 85
3.6.2 Logging measurement results for the workpiece zero ..... 87
3.7 Settings for the measurement result log ..... 88
$3.8 \quad$ Zero offsets ..... 90
3.8.1 Display active zero offset ..... 91
3.8.2 Displaying the zero offset "overview" ..... 92
3.8.3 Displaying and editing base zero offset ..... 93
3.8.4 Displaying and editing settable zero offset ..... 94
3.8.5 Displaying and editing details of the zero offsets ..... 95
3.8.6 Deleting a zero offset ..... 97
3.8.7 Measuring the workpiece zero ..... 97
3.9 Monitoring axis and spindle data ..... 98
3.9.1 Specify working area limitations ..... 98
3.9.2 Editing spindle data ..... 99
3.9.3 Spindle chuck data ..... 100
3.10 Displaying setting data lists ..... 102
3.11 Handwheel assignment. ..... 103
3.12 MDA ..... 105
3.12.1 Loading an MDA program from the Program Manager ..... 105
3.12.2 Saving an MDA program. ..... 106
3.12.3 Editing/executing a MDI program ..... 107
3.12.4 Deleting an MDA program ..... 108
4 Working in manual mode ..... 109
4.1 General ..... 109
4.2 Selecting a tool and spindle ..... 109
4.2.1 T,S,M window. ..... 109
4.2.2 $\quad$ Selecting a tool. ..... 111
4.2.3 Starting and stopping the spindle manually ..... $\frac{112}{112}$
4.2.4 Positioning the spindle ..... 113
4.3 Traversing axes ..... 114
4.3.1 Traverse axes by a defined increment. ..... 114
4.3.2 Traversing axes by a variable increment ..... 115
$4.4 \quad$ Positioning axes ..... 116
4.5 Manual retraction ..... 117
4.6 Simple stock removal of workpiece ..... 118
$4.7 \quad$ Thread synchronizing ..... 120
4.8 Default settings for manual mode ..... 122
5 Machining the workpiece ..... 123
$5.1 \quad$ Starting and stopping machining ..... 123
5.2 Selecting a program ..... 124
5.3 Executing a trail program run ..... 125
5.4 Displaying the current program block ..... 126
5.4.1 Current block display ..... 126
5.4.2 Displaying a basic block ..... 126
5.4.3 Display program level ..... 128
5.5 Correcting a program ..... 129
5.6 Repositioning axes. ..... 130
5.7 Starting machining at a specific point ..... 131
5.7.1 Use block search ..... 131
5.7.2 Continuing program from search target ..... 133
5.7.3 Simple search target definition ..... 133
5.7.4 Defining an interruption point as search target ..... 134
5.7.5 Entering the search target via search pointer ..... 135
5.7.6 Parameters for block search in the search pointer ..... 136
5.7.7 Block search mode ..... 137
5.8 Controlling the program run ..... 139
5.8.1 Program control ..... 139
5.8.2 Skip blocks ..... 140
5.9 Overstore ..... 142
5.10 Editing a program. ..... 144
5.10.1 Searching in programs ..... 145
5.10.2 Replacing program text ..... 146
5.10.3 Copying/pasting/deleting a program block. ..... 148
5.10.4 Renumbering a program ..... 149
5.10.5 Creating a program block ..... 150
5.10.6 Opening additional programs ..... 152
5.10.7 Editor settings ..... 153
5.11 Display and edit user variables ..... 156
5.11.1 Overview ..... 156
5.11.2 R parameters ..... 157
5.11.3 Displaying global user data (GUD) ..... 158
5.11.4 Displaying channel GUDs ..... 159
5.11.5 Displaying local user data (LUD) ..... 160
5.11.6 Displaying program user data (PUD) ..... 161
5.11.7 Searching for user variables ..... 162
5.12 Displaying G functions and auxiliary functions ..... 164
5.12.1 Selected G functions ..... 164
5.12.2 $\quad$ All G functions ..... 166
5.12.3 G functions for mold making ..... 166
5.12.4 Auxiliary functions ..... 168
5.13 Mold making view. ..... 170
5.13.1 Overview ..... 170
5.13.2 Starting the mold making view ..... 173
5.13.3 Specifically jump to the program block ..... 174
5.13.4 Searching for program blocks ..... 174
5.13.5 Changing the view ..... 175
5.13.5.1 Enlarging or reducing the graphical representation ..... 175
5.13.5.2 Modifying the viewport ..... 176
5.14 Displaying the program runtime and counting workpieces ..... 177
5.15 Setting for automatic mode ..... 179
5.16 Working with DXF files ..... 181
5.16.1 Overview ..... 181
5.16.2 Displaying CAD drawings ..... 181
5.16.2.1 Open a DXF file ..... 181
5.16.2.2 Cleaning a DXF file ..... 182
5.16.2.3 Enlarging or reducing the CAD drawing ..... 183
5.16.2.4 Modifying the section ..... 183
5.16.2.5 Rotating the view. ..... 184
5.16.2.6 Displaying/editing information for the geometric data ..... 185
5.16.3 Importing and editing a DXF file in the editor ..... 186
5.16.3.1 General procedure ..... 186
5.16.3.2 Specifying a reference point ..... 186
5.16.3.3 Setting the tolerance ..... 187
5.16.3.4 Transferring the drilling positions ..... 188
5.16.3.5 Accepting contours ..... 191
6 Simulating machining ..... 195
6.1 Overview ..... 195
6.2 Simulation before machining of the workpiece ..... 200
6.3 Simultaneous recording before machining of the workpiece ..... 201
6.4 Simultaneous recording during machining of the workpiece ..... 202
6.5 Different views of the workpiece ..... 203
6.5.1 Side view. ..... 203
6.5.2 Half section ..... 203
6.5.3 Face view ..... 204
6.5.4 3D view ..... 204
6.5.5 2-window ..... 205
6.6 Graphical display ..... 205
6.7 Editing the simulation display ..... 206
6.7.1 Blank display ..... 206
6.7. Showing and hiding the tool path ..... 208
6.8 Program control during the simulation ..... 208
6.8.1 Changing the feedrate ..... 208
6.8.2 Simulating the program block by block ..... 209
6.9 Editing and adapting a simulation graphic ..... 210
6.9.1 Enlarging or reducing the graphical representation ..... 210
6.9.2 Panning a graphical representation ..... 211
6.9.3 Rotating the graphical representation ..... 211
6.9.4 Modifying the viewport ..... 212
6.9.5 Defining cutting planes ..... 213
6.10 Displaying simulation alarms ..... 214
7 Creating a G code program ..... 215
7.1 Graphical programming ..... 215
7.2 Program views ..... 215
7.3 Program structure ..... 218
7.4 Fundamentals ..... 219
7.4.1 Machining planes ..... 219
7.4.2 Current planes in cycles and input screens ..... 219
7.4.3 Programming a tool ( T ) ..... 220
7.5 Generating a G code program ..... 221
7.6 Blank input ..... 222
7.7 Machining plane, milling direction, retraction plane, safe clearance and feedrate (PL, RP, SC, F) ..... 225
7.8 Selection of the cycles via softkey ..... 226
$7.9 \quad$ Calling technology cycles ..... 231
7.9.1 Hiding cycle parameters ..... 231
7.9.2 Setting data for cycles ..... 231
7.9.3 Checking cycle parameters ..... 232
7.9.4 Programming variables ..... 232
7.9.5 Changing a cycle call ..... 233
7.9.6 Compatibility for cycle support ..... 233
7.9.7 Additional functions in the input screens ..... 234
7.10 Measuring cycle support ..... 234
8 Creating a ShopTurn program ..... 235
8.1 Graphic program control, ShopTurn programs ..... 235
8.2 Program views ..... 235
8.3 Program structure ..... 240
8.4 Fundamentals ..... 241
8.4.1 Machining planes ..... 241
8.4.2 Machining cycle, approach/retraction ..... 243
8.4.3 Absolute and incremental dimensions ..... 245
8.4.4 Polar coordinates ..... 246
8.4.5 Clamping the spindle ..... 247
8.5 Creating a ShopTurn program ..... 248
8.6 Program header ..... 250
8.7 Generating program blocks ..... 253
8.8 Tool, offset value, feedrate and spindle speed (T, D, F, S, V) ..... 254
8.9 Call work offsets ..... 256
8.10 Repeating program blocks ..... 257
8.11 Entering the number of workpieces ..... 258
8.12 Changing program blocks ..... 259
8.13 Changing program settings ..... 260
8.14 Selection of the cycles via softkey ..... 262
8.15 Calling technology functions ..... 267
8.15.1 Additional functions in the input screens ..... 267
8.15.2 Checking cycle parameters ..... 267
8.15.3 Programming variables ..... 268
8.15.4 Setting data for technological functions ..... 268
8.15.5 Changing a cycle call ..... 269
8.15.6 Compatibility for cycle support ..... 269
8.16 Programming the approach/retraction cycle ..... 270
8.17 Measuring cycle support ..... 272
8.18 Example: Standard machining ..... 272
8.18.1 Workpiece drawing ..... 273
8.18.2 Programming ..... 274
8.18.3 Results/simulation test ..... 287
8.18.4 G code machining program ..... 289
9 Programming technology functions (cycles) ..... 293
9.1 Drilling ..... 293
9.1.1 General ..... 293
9.1.2 Centering (CYCLE81) ..... 294
9.1.3 Drilling (CYCLE82) ..... 296
9.1.4 Reaming (CYCLE85) ..... 301
9.1.5 Boring (CYCLE86) ..... 303
9.1.6 Deep-hole drilling 1 (CYCLE83) ..... 306
9.1.7 Deep-hole drilling 2 (CYCLE830) ..... 312
9.1.8 Tapping (CYCLE84, 840) ..... 322
9.1.9 Drill and thread milling (CYCLE78) ..... 331
9.1.10 Positions and position patterns ..... 335
9.1.11 Arbitrary positions (CYCLE802) ..... 336
9.1.12 Row position pattern (HOLES1) ..... 341
9.1.13 Grid or frame position pattern (CYCLE801). ..... 343
9.1.14 Circle or pitch circle position pattern (HOLES2) ..... 347
9.1.15 Displaying and hiding positions ..... 352
9.1.16 Repeating positions ..... 354
9.2 Rotate ..... 355
9.2.1 General ..... 355
9.2.2 Stock removal (CYCLE951) ..... 355
9.2.3 Groove (CYCLE930) ..... 358
9.2.4 Undercut form E and F (CYCLE940) ..... 361
9.2.5 Thread undercuts (CYCLE940) ..... 363
9.2.6 Thread turning (CYCLE99) ..... 366
9.2.7 Thread chain (CYCLE98) ..... 382
9.2.8 Cut-off (CYCLE92) ..... 388
9.3 Contour turning ..... 390
9.3.1 General information ..... 390
9.3.2 Representation of the contour ..... 391
9.3.3 Creating a new contour. ..... 392
9.3.4 Creating contour elements ..... 394
9.3.5 Entering the master dimension ..... 400
9.3.6 Changing the contour ..... 402
9.3.7 Contour call (CYCLE62) - only for G code program ..... 403
9.3.8 Stock removal (CYCLE952) ..... 404
9.3.9 Stock removal rest (CYCLE952) ..... 412
9.3.10 Plunge-cutting (CYCLE952) ..... 415
9.3.11 Plunge-cutting rest (CYCLE952) ..... 420
9.3.12 Plunge-turning (CYCLE952) ..... 423
9.3.13 Plunge-turning rest (CYCLE952) ..... 429
9.4 Milling ..... 431
9.4.1 Face milling (CYCLE61) ..... 431
9.4.2 Rectangular pocket (POCKET3) ..... 435
9.4.3 Circular pocket (POCKET4) ..... 444
9.4.4 Rectangular spigot (CYCLE76) ..... 454
9.4.5 Circular spigot (CYCLE77) ..... 461
9.4.6 Multi-edge (CYCLE79) ..... 468
9.4.7 Longitudinal groove (SLOT1) ..... 473
9.4.8 Circumferential groove (SLOT2) ..... 482
9.4.9 Open groove (CYCLE899) ..... 490
9.4.10 Long hole (LONGHOLE) - only for G code program ..... 501
9.4.11 Thread milling (CYCLE70) ..... 503
9.4.12 Engraving (CYCLE60) ..... 507
9.5 Contour milling ..... 514
9.5.1 General information ..... 514
9.5.2 Representation of the contour ..... 514
9.5.3 Creating a new contour. ..... 516
9.5.4 Creating contour elements ..... 519
9.5.5 Changing the contour ..... 524
9.5.6 Contour call (CYCLE62) - only for G code program ..... 525
9.5.7 Path milling (CYCLE72) ..... 526
9.5.8 Contour pocket/contour spigot (CYCLE63/64) ..... 532
9.5.9 Predrilling contour pocket (CYCLE64) ..... 534
9.5.10 Milling contour pocket (CYCLE63) ..... 538
9.5.11 Contour pocket residual material (CYCLE63, option) ..... 544
9.5.12 Milling contour spigot (CYCLE63) ..... 547
9.5.13 Contour spigot residual material (CYCLE63, option) ..... 552
9.6 Further cycles and functions ..... 555
9.6.1 Swiveling plane / aligning tool (CYCLE800) ..... 555
9.6.2 Swiveling tool (CYCLE800) ..... 563
9.6.2.1 Aligning turning tools - only for G code program (CYCLE800) ..... 563
9.6.2.2 Aligning milling tools - only for $G$ code program (CYCLE800) ..... 566
9.6.2.3 Preloading milling tools - only for G code program (CYCLE800) ..... 567
9.6.3 High-speed settings (CYCLE832) ..... 568
9.6.4 Subroutines ..... 572
9.7 Additional cycles and functions in ShopTurn ..... 574
9.7.1 Drilling centric ..... 574
9.7.2 Thread centered ..... 578
9.7.3 Transformations ..... 581
9.7.4 Translation ..... 583
9.7.5 Rotation ..... 584
9.7.6 Scaling ..... 585
9.7.7 Mirroring ..... 586
9.7.8 Rotation C ..... 587
9.7.9 Straight and circular machining ..... 588
9.7.10 Selecting a tool and machining plane ..... 589
9.7.11 Programming a straight line ..... 591
9.7.12 Programming a circle with known center point ..... 593
9.7.13 Programming a circle with known radius ..... 595
9.7.14 Polar coordinates ..... 597
9.7.15 Straight line polar ..... 598
9.7.16 Circle polar ..... 600
9.7.17 Machining with movable counterspindle ..... 601
9.7.17.1 Programming example: Machining main spindle - Transfer workpiece - Machining counterspindle ..... 602
9.7.17.2 Programming example: Machining counter-spindle - Transfer workpiece - Machining main spindle ..... 602
9.7.17.3 Programming example: Machining, counterspindle - without previous transfer ..... 603
9.7.17.4 Programming example: Machining bar material ..... 603
9.7.18 Machining with fixed counterspindle ..... 609
10 Multi-channel machining ..... 611
10.1 Multi-channel view ..... 611
10.1.1 Multi-channel view in the "Machine" operating area ..... 611
10.1.2 Multi-channel view for large operator panels ..... 614
10.1.3 Setting the multi-channel view ..... 615
10.2 Multi-channel support ..... 617
10.2.1 Working with several channels ..... 617
10.2.2 Creating a multi-channel program. ..... 618
10.2.3 Entering multi-channel data ..... 619
10.2.4 Multi-channel functionality for large operator panels ..... 622
10.2.5 Editing the multi-channel program ..... 625
10.2.5.1 Changing the job list ..... 625
10.2.5.2 Editing a G code multi-channel program ..... 626
10.2.5.3 Editing a ShopTurn multi-channel program ..... 628
10.2.5.4 Creating a program block ..... 636
10.2.6 Setting the multi-channel function ..... 638
10.2.7 Synchronizing programs ..... 640
10.2.8 Insert WAIT marks ..... 643
10.2.9 Optimizing the machining time ..... 644
10.2.10 Automatic block building ..... 646
10.2.10.1 Creating automated program blocks ..... 646
10.2.10.2 Editing a converted program ..... 647
10.2.11 Simulating machining ..... 648
10.2.11.1 Simulation ..... 648
10.2.11.2 Different workpiece views for multi-channel support ..... 650
10.2.12 Display/edit the multi-channel functionality in the "Machine" operating area ..... 651
10.2.12.1 Running-in a program ..... 651
10.2.12.2 Block search and program control ..... 652
10.2.13 Stock removal with 2 synchronized channels ..... 654
10.2.13.1 Job list ..... 656
10.2.13.2 Stock removal ..... 658
10.2.14 Synchronizing a counterspindle ..... 659
11 Collision avoidance (only 840D sl) ..... 665
11.1 Activating collision avoidance ..... 665
11.2 Set collision avoidance ..... 667
12 Tool management ..... 669
12.1 Lists for the tool management ..... 669
12.2 Magazine management ..... 670
12.3 Tool types ..... 671
12.4 Tool dimensioning ..... 674
12.5 Tool list ..... 679
12.5.1 Additional data ..... 683
12.5.2 Creating a new tool ..... 684
12.5.3 Measuring the tool ..... 686
12.5.4 Managing several cutting edges ..... 687
12.5.5 Delete tool ..... 687
12.5.6 Loading and unloading tools ..... 688
12.5.7 Selecting a magazine ..... 690
12.5.8 Code carrier connection (only 840D sl) ..... 691
12.5.8.1 Overview ..... 691
12.5.9 Managing a tool in a file ..... 694
12.6 Tool wear ..... 696
12.6.1 Reactivate tool ..... 699
12.7 Tool data OEM ..... 700
12.8 Magazine ..... 702
12.8.1 Positioning a magazine ..... 704
12.8.2 Relocating a tool ..... 704
12.8.3 Unload/load/relocate all tools ..... 705
12.9 Tool details ..... 707
12.9.1 Displaying tool details ..... 707
12.9.2 Tool data ..... 708
12.9.3 Cutting edge data ..... 709
12.9.4 Monitoring data ..... 711
12.10 Sorting tool management lists ..... 712
12.11 Filtering the tool management lists ..... 713
12.12 Specific search in the tool management lists ..... 715
12.13 Changing the cutting edge position or tool type ..... 717
12.14 Settings for tool lists ..... 718
12.15 Working with multitool ..... 720
12.15.1 Tool list for multitool ..... 721
12.15.2 Create multitool ..... 722
12.15.3 Equipping multitool with tools ..... 724
12.15.4 Removing a tool from multitool ..... 725
12.15.5 Delete multitool ..... 725
12.15.6 Loading and unloading multitool ..... 726
12.15.7 Reactivating the multitool ..... 727
12.15.8 Relocating a multitool ..... 729
12.15.9 Positioning multitool ..... 730
13 Managing programs ..... 731
13.1 Overview ..... 731
13.1.1 NC memory ..... 734
13.1.2 Local drive ..... 735
13.1.3 USB drives ..... 736
13.1.4 FTP drive ..... 737
13.2 Opening and closing the program ..... 738
13.3 Executing a program. ..... 740
13.4 Creating a directory / program / job list / program list ..... 742
13.4.1 Creating a new directory ..... 742
13.4.2 Creating a new workpiece ..... 743
13.4.3 Creating a new G code program ..... 744
13.4.4 New ShopTurn program ..... 744
13.4.5 Storing any new file ..... 745
13.4.6 Creating a job list ..... 746
13.4.7 Creating a program list ..... 748
13.5 Creating templates ..... 749
13.6 Searching directories and files ..... 750
13.7 Displaying the program in the Preview. ..... 751
13.8 Selecting several directories/programs ..... 752
13.9 Copying and pasting a directory/program ..... 754
13.10 Deleting a directory/program ..... 756
13.11 Changing file and directory properties ..... 757
13.12 Set up drives ..... 758
13.12.1 Overview ..... 758
13.12.2 Setting up drives ..... 759
13.13 Viewing PDF documents ..... 765
13.14 EXTCALL ..... 767
13.15 Execution from External Storage (EES) ..... 769
13.15.1 Overview ..... 769
13.16 Backing up data ..... 770
13.16.1 Generating an archive in the Program Manager ..... 770
13.16.2 Generating an archive via the system data ..... 771
13.16.3 Reading in an archive in the Program Manager ..... 773
13.16.4 Read in archive from system data ..... 774
13.17 Setup data ..... 776
13.17.1 Backing up setup data ..... 776
13.17.2 Reading-in set-up data ..... 779
13.18 RS-232-C ..... 781
13.18.1 Reading-in and reading-out archives via a serial interface ..... 781
13.18.2 Setting V24 in the program manager ..... 783
14 Alarm, error and system messages ..... 785
14.1 Displaying alarms. ..... 785
14.2 Displaying an alarm log ..... 787
14.3 Displaying messages ..... 788
14.4 Sorting, alarms, faults and messages ..... 789
14.5 Creating screenshots ..... 790
14.6 PLC and NC variables ..... 791
14.6.1 Displaying and editing PLC and NC variables ..... 791
14.6.2 Saving and loading screen forms ..... 795
14.7 Version ..... 796
14.7.1 Displaying version data ..... 796
14.7.2 Save information ..... 797
14.8 Logbook ..... 798
14.8.1 Displaying and editing the logbook ..... 798
14.8.2 Making a logbook entry ..... 799
14.9 Remote diagnostics ..... 801
14.9.1 Setting remote access ..... 801
14.9.2 Permit modem ..... 803
14.9.3 Request remote diagnostics ..... 803
14.9.4 Exit remote diagnostics ..... 804
15 Working with Manual Machine ..... 805
15.1 Manual Machine ..... 805
15.2 Measuring the tool ..... 806
15.3 Setting the zero offset ..... 806
15.4 Set limit stop ..... 807
15.5 Simple workpiece machining ..... 808
15.5.1 Traversing axes ..... 808
15.5.2 Taper turning. ..... 809
15.5.3 Straight and circular machining ..... 810
15.5.3.1 Straight turning ..... 810
15.5.3.2 Circular turning ..... 812
15.6 More complex machining ..... 813
15.6.1 Drilling with Manual Machine ..... 814
15.6.2 Turning with manual machine ..... 816
15.6.3 Contour turning with Manual machine ..... 818
15.6.4 Milling with Manual Machine ..... 819
15.7 Simulation and simultaneous recording ..... 820
16 Working with a B axis (only 840D sl) ..... 821
16.1 Lathes with $B$ axis ..... 821
16.2 Tool alignment for turning ..... 824
16.3 Milling with a B axis ..... 825
16.4 Swiveling ..... 826
16.5 Approach/retraction ..... 828
16.6 Position pattern ..... 830
16.7 Tool selection for the manual mode ..... 832
16.8 Measuring a tool with the $B$ axis ..... 833
17 Working with two tool carriers. ..... 835
17.1 Programming with two tool holders ..... 835
17.2 Measure tool ..... 836
18 Teaching in a program ..... 837
18.1 Overview ..... 837
18.2 General sequence ..... 837
18.3 Inserting a block ..... 838
18.3.1 Input parameters for teach-in blocks ..... 839
18.4 Teach-in via Windows ..... 841
18.4.1 General ..... 841
18.4.2 Teach in rapid traverse G0 ..... 842
18.4.3 Teach in straight G1 ..... 842
18.4.4 Teaching in circle intermediate and circle end point CIP ..... 843
18.4.5 Teach-in A spline ..... 843
18.5 Editing a block ..... 845
18.6 Selecting a block ..... 846
18.7 Deleting a block. ..... 847
18.8 Settings for teach-in ..... 848
19 HT 8 ..... 849
19.1 HT 8 overview ..... 849
19.2 Traversing keys ..... 852
19.3 Machine control panel menu ..... 853
19.4 Virtual keyboard ..... 854
19.5 Calibrating the touch panel ..... 856
20 Ctrl-Energy ..... 857
20.1 Overview ..... 857
20.2 Displaying energy consumption ..... 858
20.3 Measuring and saving the energy consumption ..... 860
20.4 Long-term measurement of the energy consumption ..... 862
20.5 Displaying measured curves ..... 863
20.6 Using the energy-saving profile ..... 864
21 Easy Message (828D only) ..... 867
21.1 Overview ..... 867
21.2 Activating Easy Message ..... 869
21.3 Creating/editing a user profile ..... 870
21.4 Setting-up events ..... 872
21.5 Logging an active user on and off ..... 874
21.6 Displaying SMS logs ..... 875
21.7 Making settings for Easy Message ..... 876
22 Easy Extend (828D only). ..... 877
22.1 Overview ..... 877
22.2 Enabling a device ..... 878
22.3 Activating and deactivating a device ..... 879
22.4 Initial commissioning of additional devices ..... 880
23 Service Planner (828D only) ..... 881
23.1 Performing and monitoring maintenance tasks ..... 881
23.2 Set maintenance tasks ..... 883
24 Edit PLC user program (828D only) ..... 885
24.1 Introduction ..... 885
24.2 Ladder add-on tool ..... 886
24.3 Structure of the user interface ..... 887
24.4 Control options ..... 889
24.5 Displaying PLC properties ..... 891
24.6 Displaying information on the program blocks ..... 892
24.7 Displaying and editing NC/PLC variables ..... 894
24.8 Loading modified PLC user program ..... 895
24.9 Displaying local variable table ..... 896
24.10 Creating a new block ..... 897
24.11 Editing block properties subsequently ..... 898
24.12 Inserting and editing networks ..... 899
24.13 Editing network properties ..... 901
24.14 Displaying/canceling the access protection ..... 902
24.15 Displaying and editing symbol tables ..... 903
24.16 Searching for operands ..... 904
24.17 Inserting/deleting a symbol table ..... 905
24.18 Displaying the network symbol information table. ..... 906
24.19 Displaying and editing PLC signals ..... 907
24.20 Displaying cross references ..... 908
A Appendix ..... 911
A. 1 840D sl documentation overview ..... 911
Index ..... 913

## Fundamental safety instructions

### 1.1 General safety instructions

! WWARNING
Risk of death if the safety instructions and remaining risks are not carefully observed
If the safety instructions and residual risks are not observed in the associated hardware documentation, accidents involving severe injuries or death can occur.

- Observe the safety instructions given in the hardware documentation.
- Consider the residual risks for the risk evaluation.


## \. WARNING

Danger to life or malfunctions of the machine as a result of incorrect or changed parameterization

As a result of incorrect or changed parameterization, machines can malfunction, which in turn can lead to injuries or death.

- Protect the parameterization (parameter assignments) against unauthorized access.
- Respond to possible malfunctions by applying suitable measures (e.g. EMERGENCY STOP or EMERGENCY OFF).


### 1.2 Industrial security

## Note

## Industrial security

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, solutions, machines, equipment and/or networks. They are important components in a holistic industrial security concept. With this in mind, Siemens' products and solutions undergo continuous development. Siemens recommends strongly that you regularly check for product updates.
For the secure operation of Siemens products and solutions, it is necessary to take suitable preventive action (e.g. cell protection concept) and integrate each component into a holistic, state-of-the-art industrial security concept. Third-party products that may be in use should also be considered. For more information about industrial security, visit Hotspot-Text (http://www.siemens.com/industrialsecurity).
To stay informed about product updates as they occur, sign up for a product-specific newsletter. For more information, visit Hotspot-Text (http://support.automation.siemens.com).

## WARNING

Danger as a result of unsafe operating states resulting from software manipulation
Software manipulation (e.g. by viruses, Trojan horses, malware, worms) can cause unsafe operating states to develop in your installation which can result in death, severe injuries and/or material damage.

- Keep the software up to date.

You will find relevant information and newsletters at this address (http://support.automation.siemens.com).

- Incorporate the automation and drive components into a holistic, state-of-the-art industrial security concept for the installation or machine.
You will find further information at this address
(http://www.siemens.com/industrialsecurity).
- Make sure that you include all installed products into the holistic industrial security concept.


## Introduction

### 2.1 Product overview

The SINUMERIK controller is a CNC (Computerized Numerical Controller) for machine tools.
You can use the CNC to implement the following basic functions in conjunction with a machine tool:

- Creation and adaptation of part programs
- Execution of part programs
- Manual control
- Access to internal and external data media
- Editing of data for programs
- Management of tools, zero points and further user data required in programs
- Diagnostics of controller and machine


## Operating areas

The basic functions are grouped in the following operating areas in the controller:


### 2.2 Operator panel fronts

### 2.2.1 Overview

## Introduction

The display (screen) and operation (e.g. hardkeys and softkeys) of the SINUMERIK Operate user interface use the operator panel front.

In this example, the OP 010 operator panel front is used to illustrate the components that are available for operating the controller and machine tool.

## Operator controls and indicators



1 Alphabetic key group
With the <Shift> key pressed, you activate the special characters on keys with double assignments, and write in the uppercase.
Note: Depending on the particular configuration of your control system, uppercase letters are always written

2 Numerical key group
With the <Shift> key pressed, you activate the special characters on keys with double assignments.
3 Control key group
4 Hotkey group
5 Cursor key group

| 6 | USB interface |
| :--- | :--- |
| 7 | Menu select key |
| 8 | Menu forward button |
| 9 | Machine area button |
| 10 | Menu back key |
| 11 | Softkeys |

Figure 2-1 View of OP 010 operator panel front

## References

A more precise description as well as a view of the other operator panel fronts that can be used may be found in the following reference:

Manual operator components and networking; SINUMERIK 840D sl

### 2.2.2 Keys of the operator panel

The following keys and key combinations are available for operation of the control and the machine tool.

## Keys and key combinations

Key
Function


## <ALARM CANCEL>

Cancels alarms and messages that are marked with this symbol.
<CHANNEL>
Advances for several channels.
<HELP>
Calls the context-sensitive online help for the selected window.


- Moves the cursor to the beginning of a program.
- Moves the cursor to the first row of the current column.
- Selects a contiguous selection from the current cursor position up to the target position.
- Selects a contiguous selection from the current cursor position up to the beginning of the program.


## <NEXT WINDOW> *

- Toggles between the windows.
- For a multi-channel view or for a multi-channel functionality, switches within a channel gap between the upper and lower window.
- Selects the first entry in selection lists and in selection fields.
- Moves the cursor to the beginning of a text.
* on USB keyboards use the <Home> or <Pos 1> key


## <NEXT WINDOW> + <SHIFT>

- Selects the first entry in selection lists and in selection fields.
- Moves the cursor to the beginning of a text.
- Selects a contiguous selection from the current cursor position up to the target position.
- Selects a contiguous selection from the current cursor position up to the beginning of a program block.
<NEXT WINDOW> + <ALT>
- Moves the cursor to the first object.
- Moves the cursor to the first column of a table row.
- Moves the cursor to the beginning of a program block.


## <NEXT WINDOW> + <CTRL>

- Moves the cursor to the beginning of a program.
- Moves the cursor to the first row of the current column.


## <NEXT WINDOW> + <CTRL> + <SHIFT>

## <PAGE UP>

Scrolls upwards by one page in a window.

## <PAGE UP> + <SHIFT>

In the program manager and in the program editor from the cursor position, selects directories or program blocks up to the beginning of the window.
<PAGE UP> + <CTRL>
Positions the cursor to the topmost line of a window.


## <PAGE DOWN>

Scrolls downwards by one page in a window.

## <PAGE DOWN> + <SHIFT>

In the program manager and in the program editor, from the cursor position, selects directories or program blocks up to the end of the window.

## <PAGE DOWN> + <CTRL>

Positions the cursor to the lowest line of a window.

## <Cursor right>

- Editing box

Opens a directory or program (e.g. cycle) in the editor.

- Navigation

Moves the cursor further to the right by one character.

## <Cursor right> + <CTRL>

- Editing box

Moves the cursor further to the right by one word.

- Navigation

Moves the cursor in a table to the next cell to the right.

## <Cursor left>

- Editing box

Closes a directory or program (e.g. cycle) in the program editor. If you have made changes, then these are accepted.

- Navigation

Moves the cursor further to the left by one character.

## <Cursor left> + <CTRL>

- Editing box

Moves the cursor further to the left by one word.

- Navigation

Moves the cursor in a table to the next cell to the left.


## <Cursor down>

- Editing box

Moves the cursor downwards.

- Navigation
- Moves the cursor in a table to the next cell downwards.
- Moves the cursor in a window downwards.


## <Cursor up>

- Editing box

Moves the cursor into the next upper field.

- Navigation
- Moves the cursor in a table to the next cell upwards.
- Moves the cursor upwards in a menu screen.


## <Cursor up> + <CTRL>

- Moves the cursor in a table to the beginning of the table.
- Moves the cursor to the beginning of a window.


## <Cursor up> + <SHIFT>

In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

## <Cursor down> + <CTRL>

- Navigation
- Moves the cursor in a table to the end of the table.
- Moves the cursor to the end of a window.
- Simulation

Reduces the override.

## <Cursor down> + <SHIFT>

In the program manager and in the program editor, selects a contiguous selection of directories and program blocks.

## <SELECT>

Switches between several specified options in selection lists and in selection boxes.
Activates checkboxes.
In the program editor and in the program manager, selects a program block or a program.

## <SELECT> + <CTRL>

When selecting table rows, switches between selected and not selected.


## <SELECT> + <SHIFT>

Selects in selection lists and in selection boxes the previous entry or the last entry.
<END>
Moves the cursor to the last entry field in a window, to the end of a table or a program block.
Selects the last entry in selection lists and in selection boxes.
<END> + <SHIFT>
Moves the cursor to the last entry.
Selects a contiguous selection from the cursor position up to the end of a program block.

## <END> + <CTRL>

Moves the cursor to the last entry in the last line of the actual column or to the end of a program.
<END> + <CTRL> + <SHIFT>
Moves the cursor to the last entry in the last line of the actual column or to the end of a program.
Selects a contiguous selection from the cursor position up to the end of a program block.

## <BACKSPACE>

- Editing box

Deletes a character selected to the left of the cursor.

- Navigation

Deletes all of the selected characters to the left of the cursor.

## <BACKSPACE> + <CTRL>

- Editing box

Deletes a word selected to the left of the cursor.

- Navigation

Deletes all of the selected characters to the left of the cursor.
<TAB>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the right.


CTRL


## <TAB> + <SHIFT>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the left.


## <TAB> + <CTRL>

- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the right.
<TAB> + <SHIFT> + <CTRL>
- In the program editor, indents the cursor by one character.
- In the program manager, moves the cursor to the next entry to the left.


## <CTRL> + <A>

In the actual window, selects all entries (only in the program editor and program manager).

## <CTRL> + <C>

Copies the selected content.

## <CTRL> + <E>

Calls the "Ctrl Energy" function.

## <CTRL> + <F>

Opens the search dialog in the machine data and setting data lists, when loading and saving in the MDI editor as well as in the program manager and in the system data.

## <CTRL> + <G>

- Switches in the program editor for ShopMill or ShopTurn programs between the work plan and the graphic view.
- Switches in the parameter screen between the help display and the graphic view.


## <CTRL> + <L>

Scrolls the actual user interface through all installed languages one after the other.

## <CTRL> + <SHIFT> + <L>

Scrolls the actual user interface through all installed languages in the inverse sequence.

## <CTRL> + <L>

Selects the maximum feedrate of $120 \%$ during the simulation.


## <CTRL> + <P>

Generates a screenshot from the actual user interface and saves it as file.

## <CTRL> + <S>

Switches the single block in or out in the simulation.
<CTRL> + <V>

- Pastes text from the clipboard at the actual cursor position.
- Pastes text from the clipboard at the position of a selected text.


## <CTRL> + <X>

Cuts out the selected text. The text is located in the clipboard.

## <CTRL> + <Y>

Reactivates changes that were undone (only in the program editor).

## <CTRL> + <Z>

Undoes the last action (only in the program editor).
<CTRL> + <ALT> + <C>
Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sl / 828D).
Note:
The complete backup via this key combination is only suitable for diagnostic purposes.
Note:
Please refer to the machine manufacturer's specifications.

<CTRL> + <ALT> + <S>
Creates a complete standard archive (.ARC) on an external data carrier (USB-FlashDrive) (for 840D sI).
Creates a complete Easy Archive (.ARD) on an external data carrier (USB-FlashDrive) (for 828D).

## Note:

The complete backup (.ARC) via this key combination is only suitable for diagnostic purposes.
Note:
Please refer to the machine manufacturer's specifications.

<CTRL> + <ALT> + <D>
Backs up the log files on the USB-FlashDrive. If a USB-
FlashDrive is not inserted, then the files are backed-up in the manufacturer's area of the CF card.


Deletes the first character to the right of the cursor.

- Navigation

Deletes all characters.

## <DEL> + <CTRL>

- Editing box

Deletes the first word to the right of the cursor.

- Navigation

Deletes all characters.
<Spacebar>

- Editing box Inserts a space.
- Switches between several specified options in selection lists and in selection boxes.


## <Plus>

- Opens a directory which contains the element.
- Increases the size of the graphic view for simulation and traces.




## <Minus>

- Closes a directory which contains the element.
- Reduces the size of the graphic view for simulation and traces.


## <Equals>

Opens the calculator in the entry fields.

## <Asterisk>

Opens a directory with all of the subdirectories.

## <Tilde>

Changes the sign of a number between plus and minus.

## <INSERT>

- Opens an editing window in the insert mode. Pressing the key again, exits the window and the entries are undone.
- Opens a selection box and shows the selection possibilities.
- In the machining step program, enters an empty line for G code.
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.


## <INSERT> + <SHIFT>

For $G$ code programming, for a cycle call activates or deactivates the edit mode.

## <INPUT>

- Completes input of a value in the entry field.
- Opens a directory or a program.
- Inserts an empty program block if the cursor is positioned at the end of a program block.
- Inserts a character to select a new line and the program block is split up into two parts.
- In the G code, inserts a new line after the program block.
- In the machining step program, inserts a new line for G code e
- Changes into the double editor or into the multi-channel view from the edit mode into the operating mode. You can return to the edit mode by pressing the key again.
<ALARM> - only OP 010 and OP 010C
Calls the "Diagnosis" operating area.

<PROGRAM> - only OP 010 and OP 010C
Calls the "Program Manager" operating area.
<OFFSET> - only OP 010 and OP 010C
Calls the "Parameter" operating area.
<PROGRAM MANAGER> - only OP 010 and OP 010C
Calls the "Program Manager" operating area.


## Menu forward key

Advances in the extended horizontal softkey bar.

## Menu back key

Returns to the higher-level menu.
<MACHINE>
Calls the "Machine" operating area.

## <MENU SELECT>

Calls the main menu to select the operating area.

### 2.3 Machine control panels

### 2.3.1 Overview

The machine tool can be equipped with a machine control panel by Siemens or with a specific machine control panel from the machine manufacturer.

You use the machine control panel to initiate actions on the machine tool such as traversing an axis or starting the machining of a workpiece.

### 2.3.2 Controls on the machine control panel

In this example, the MCP 483C IE machine control panel is used to illustrate the operator controls and displays of a Siemens machine control panel.

## Overview


(1) Emergency Off button
(2) Installation locations for control devices (d = 16 mm )
(3) RESET
(4) Program control
(5) Operating modes, machine functions
(6) User keys T1 to T15
(7) Traversing axes with rapid traverse override and coordinate switchover
(8) Spindle control with override switch
(9) Feed control with override switch
(10) Keyswitch (four positions)

Figure 2-2 Front view of machine control panel (milling version)

## Operator controls

## Emergency Off button



Press the button in situations where:

- life is at risk.
- there is the danger of a machine or workpiece being damaged.

All drives will be stopped with the greatest possible braking torque.

## Machine manufacturer

For additional responses to pressing the Emergency Stop button, please refer to the machine manufacturer's instructions.

RESET


- Stop processing the current programs. The NCK control remains synchronized with the machine. It is in its initial state and ready for a new program run.
- Cancel alarm.

Program control

$\underset{\text { CYCLE }}{8}$
CYCLE
STOP
<SINGLE BLOCK>
Single block mode on/off.
<CYCLE START>
The key is also referred to as NC Start.
Execution of a program is started.
<CYCLE STOP>
The key is also referred to as NC Stop.
Execution of a program is stopped.

Operating modes，machine functions


包
TEACH IN

## $\underset{\text { 永 }}{ }$


＜JOG＞
Select＂JOG＂mode．
＜TEACH IN＞
Select＂Teach In＂submode．
＜MDI＞
Select＂MDI＂mode．
＜AUTO＞
Select＂AUTO＂mode．
＜REPOS＞
Repositions，re－approaches the contour．
＜REF POINT＞
Approach reference point．
Inc＜VAR＞（Incremental Feed Variable）
Incremental mode with variable increment size．
Inc（incremental feed）
Incremental mode with predefined increment size of 1，．．．， 10000 incre－ ments．
$\rightarrow 1$
10000

## Machine manufacturer

A machine data code defines how the increment value is in－ terpreted．

Traversing axes with rapid traverse override and coordinate switchover
$\square$

## Axis keys

x
Selects an axis.
z


## Direction keys

Select the traversing direction.
...


## <RAPID>

Traverse axis in rapid traverse while pressing the direction key.
<WCS MCS>
Switches between the workpiece coordinate system (WCS) and machine coordinate system (MCS).

Spindle control with override switch

| $-7 \mathrm{~h} \otimes \mid$ |
| :---: |
| $\substack{\text { SPNDLE } \\ \text { STOP }}$ |

<SPINDLE STOP>
Stop spindle.
<SPINDLE START>

SPINDLE
START
Spindle is enabled.

Feed control with override switch

| m 0 | <FEED STOP> <br> Stops execution of the running program and shuts down axis drives. |
| :---: | :---: |
| $\underset{\substack{\text { cro } \\ \text { stop }}}{\text { cose }}$ |  |
|  | <FEED START> |
|  | Enable for program execution in the current block and enable for ramp-up to the feedrate value specified by the program. |

### 2.4 User interface

### 2.4.1 Screen layout

Overview


1 Active operating area and mode
2 Alarm/message line
3 Program name
4 Channel state and program control
5 Channel operational messages
6 Axis position display in actual value window

### 2.4 User interface

7 Display for

- active tool T
- current feedrate $F$
- active spindle with current status (S)
- Spindle utilization rate in percent

8 Operating window with program block display
9 Display of active G functions, all G functions, H functions and input window for different functions (for example, skip blocks, program control)
10 Dialog line to provide additional user notes
11 Horizontal softkey bar
12 Vertical softkey bar
Figure 2-3 User interface

### 2.4.2 Status display

The status display includes the most important information about the current machine status and the status of the NCK. It also shows alarms as well as NC and PLC messages.

Depending on your operating area, the status display is made up of several lines:

- Large status display

The status display is made up of three lines in the "Machine" operating area.

- Small status display

In the "Parameter", "Program", "Program Manager", "Diagnosis" and "Start-up" operating areas, the status display consists of the first line from the large display.

## Status display of "Machine" operating area

First line
Ctrl-Energy - Power rating display

| Display | Description |
| :--- | :--- |
| $\underline{m}$ | The machine is not productive. |
| $\underline{m}$ | The machine is productive and energy is being consumed. |
| $\underline{m}$ | The machine is feeding energy back into the grid. |
| The power rating display must be switched on in the status line. |  |
| Note |  |
| Information about configuration is available in the following reference: <br> System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D |  |

## Active operating area

| Display | Description |
| :--- | :--- |
| $\square$ | "Machine" operating area |
| With touch operation, you can change the operating area here. |  |
| $\square$ | "Parameter" operating area |
| $\square$ | "Program" operating area |
| $\square$ | "Diagnosis" operating area |
| $\square$ | "Start-up" operating area |
| n |  |

Active mode or submode

| Display | Description |
| :---: | :---: |
| $\mathrm{NOG}_{\mathrm{Jog}}$ | "Jog" mode |
| $\frac{\square}{\text { MDA }}$ | "MDA" mode |
| $\underset{\text { RUTO }}{\rightarrow}$ | "Auto" mode |
| $\underset{\text { teach in }}{\mathbf{8}}$ | "Teach In" submode |
| REPOS | "Repos" submode |
| REFPOINT | "Ref Point" submode |

## Alarms and messages

| Display | Description |
| :---: | :--- |
| $10299 \downarrow$ Channel 1 | Alarm display <br> The alarm numbers are displayed in white lettering on a red <br> background. The associated alarm text is shown in red letter- <br> ing. <br> An arrow indicates that several alarms are active. <br> An acknowledgment symbol indicates that the alarm can be <br> acknowledged or canceled. |
| $600308 \downarrow$ | NC or PLC message <br> Message numbers and texts are shown in black lettering. <br> An arrow indicates that several messages are active. |
| 0 | Messages from NC programs do not have numbers and appear <br> in green lettering. |
| READY TO START |  |

Second line

| Display | Description |
| :--- | :--- |
| TEST_TEACHEN | Program path and program name |

The displays in the second line can be configured.


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Third line

| Display | Description |
| :--- | :--- |
| /CHAN1 RESET | Display of channel status. <br> If several channels are present on the machine, the channel <br> name is also displayed. <br> If only one channel is available, only the "Reset" channel status <br> is displayed. <br> With touch operation, you can change the channel here. |
| D | Display of channel status: <br> The program was aborted with "Reset". <br> The program is started. <br> The program has been interrupted with "Stop". |
| 1 |  |


| Display | Description |
| :---: | :---: |
| DRYPRT | Display of active program controls: <br> PRT: no axis motion <br> DRY: Dry run feedrate <br> RG0: reduced rapid traverse <br> M01: programmed stop 1 <br> M101: programmed stop 2 (name varies) <br> SB1: Single block, coarse (program stops only after blocks which perform a machine function) <br> SB2: Data block (program stops after each block) <br> SB3: Single block, fine (program also only stops after blocks which perform a machine function in cycles) |
| AFaulty NC block / user alarm Remaining dwell time: 15 Sec. | Channel operational messages: <br> Stop: An operator action is usually required. <br> Wait: No operator action is required. |

The machine manufacturer settings determine which program controls are displayed.


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

### 2.4.3 Actual value window

The actual values of the axes and their positions are displayed.

## Work/Machine

The displayed coordinates are based on either the machine coordinate system or the workpiece coordinate system. The machine coordinate system (Machine), in contrast to the workpiece coordinate system (Work), does not take any work offsets into consideration.

You can use the "Machine actual values" softkey to toggle between the machine coordinate system and the workpiece coordinate system.

The actual value display of the positions can also refer to the SZS coordinate system (settable zero system). However the positions are still output in the Work.

The ENS coordinate system corresponds to the Work coordinate system, reduced by certain components (\$P_TRAFRAME, \$P_PFRAME, \$P_ISO4FRAME, \$P_CYCFRAME), which are set by the system when machining and are then reset again. By using the ENS coordinate system, jumps into the actual value display are avoided that would otherwise be caused by the additional components.

## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Maximize display



Press the ">>" and "Zoom act. val." softkeys.

Zoom
act. val.

## Display overview

| Display | Meaning |  |
| :---: | :---: | :---: |
| Header columns |  |  |
| Work/Machine | Display of axes in selected coordinate system. |  |
| Position | Position of displayed axes. |  |
| Display of distance-to-go | The distance-to-go for the current NC block is displayed while the program is running. |  |
| Feed/override | The feed acting on the axes, as well as the override, are displayed in the full-screen version. |  |
| Repos offset | The distances traversed in manual mode are displayed. <br> This information is only displayed when you are in the "Repos" submode. |  |
| Collision monitoring (only 840D sl) | $9$ | Collision avoidance is activated for the JOG and MDA or AUTOMATIC modes. <br> Note: <br> The \$MN_JOG_MODE_MASK machine data can be set to suppress the display of the symbol. <br> Please refer to the machine manufacturer's specifications. |
|  | \$ | Collision avoidance is deactivated for the JOG and MDA or AUTOMATIC modes. |
| Footer | Display of active work offsets and transformations. <br> The T, F, S values are also displayed in the full-screen version. |  |

## See also

$\underline{\text { Zero offsets (Page 90) }}$

### 2.4.4 T,F,S window

The most important data concerning the current tool, the feedrate (path feed or axis feed in JOG) and the spindle is displayed in the T, F, S window.

In addition to the "T, F, S" window name, the following information is also displayed:

| Display | Meaning |
| :--- | :--- |
| BC (example) | Name of the tool carrier |
| Turning (example) | Name of the active kinematic transformation |
| $O_{5}$ | Active tool carrier rotated in the plane |
| STS $_{5}$ | Active tool carrier swiveled in space |

## Tool data

| Display | Meaning |
| :--- | :--- |
| T | Name of the current tool |
| Tool name | Location number of the current tool <br> LocationCutting edge of the current tool <br> The tool is displayed with the associated tool type symbol corresponding to <br> the actual coordinate system in the selected cutting edge position. <br> If the tool is swiveled, then this is taken into account in the display of the <br> cutting edge position. <br> In DIN-ISO mode the H number is displayed instead of the cutting edge <br> number. |
| D | H number (tool offset data record for DIN-ISO mode) <br> If there is a valid D number, this is also displayed. |
| H | Diameter of the current tool |
| $\varnothing$ | Radius of the current tool |
| R | Length of the actual tool |
| L | Z value of the current tool |
| Z | X value of the current tool |
| $X$ |  |

## Feed data

| Display | Meaning |
| :--- | :--- |
| F | Feed disable |
| N | Actual feed value <br> If several axes traverse, is displayed for: <br> • "JOG" mode: Axis feed for the traversing axis <br> • "MDA" and "AUTO" mode: Programmed axis feed |
|  | G0 is active |
| Rapid traverse | No feed is active |
| 0.000 | Display as a percentage |
| Override |  |

## Spindle data

| Display | Meaning |
| :--- | :--- |
| S | Spindle selection, identification with spindle number and main spindle |
| S1 | Actual value (when spindle turns, display increases) <br> Setpoint (always displayed, also during positioning) |
| Speed | Spindle status <br> Spindle not enabled <br> Spindle is turning clockwise <br> Spindle is turning counterclockwise <br> Spindle is stationary |
| Symbol | Display as a percentage |
| Override | Display between 0 and 100\% <br> The upper limit value can be greater than 100\%. <br> See machine manufacturer's specifications. |
| Spindle utilization <br> rate |  |

## Note

Display of logical spindles
If the spindle converter is active, logical spindles are displayed in the workpiece coordinate system. When switching over to the machine coordinate system, the physical spindles are displayed.

### 2.4.5 Current block display

The window of the current block display shows you the program blocks currently being executed.

## Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the title row.
- The program block which is just being processed appears colored.


## Editing a program directly

In the Reset state, you can edit the current program directly.


1. Press the <INSERT> key.
2. Place the cursor at the relevant position and edit the program block. Direct editing is only possible for G code blocks in the NC memory, not for external execution.

3. Press the <INSERT> key to exit the program and the edit mode again.

### 2.4.6 Operation via softkeys and buttons

## Operating areas/operating modes

The user interface consists of different windows featuring eight horizontal and eight vertical softkeys.

You operate the softkeys with the keys next to the softkey bars.
You can display a new window or execute functions using the softkeys.
The operating software is sub-divided into six operating areas (machine, parameter, program, program manager, diagnosis, startup) and five operating modes or submodes (JOG, MDA, AUTO, TEACH IN, REF POINT, REPOS)

## Changing the operating area

Press the <MENU SELECT> key and select the desired operating area using the horizontal softkey bar.

You can call the "Machine" operating area directly using the key on the operator panel.
Press the <MACHINE> key to select the "machine" operating area.

## Changing the operating mode

You can select a mode or submode directly using the keys on the machine control panel or using the vertical softkeys in the main menu.

## General keys and softkeys

When the $>\square$ symbol appears to the right of the dialog line on the user interface, you can change the horizontal softkey bar within an operating area. To do so, press the menu forward key.
The $\square>$ symbol indicates that you are in the expanded softkey bar.
Pressing the key again will take you back to the original horizontal softkey bar. Use the ">>" softkey to open a new vertical softkey bar.

Use the "<<" softkey to return to the previous vertical softkey bar.

Use the "Return" softkey to close an open window.
Back
Use the "Cancel" softkey to exit a window without accepting the entered val-


Cancel ues and return to the next highest window.

When you have entered all the necessary parameters in the parameter screen form correctly, you can close the window and save the parameters using the "Accept" softkey. The values you entered are applied to a program.

Use the "OK" softkey to initiate an action immediately, e.g. to rename or delete a program.

### 2.4.7 Entering or selecting parameters

When setting up the machine and during programming, you must enter various parameter values in the entry fields. The background color of the fields provides information on the status of the entry field.

Orange background The input field is selected
Light orange background The input field is in edit mode
Pink background The entered value is incorrect

## Selecting parameters

Some parameters require you to select from a number of options in the input field. Fields of this type do not allow you to type in a value.
The selection symbol is displayed in the tooltip: $U$

## Associated selection fields

There are selection fields for various parameters:

- Selection of units
- Changeover between absolute and incremental dimensions


## Procedure



1. Keep pressing the <SELECT> key until the required setting or unit is selected.

The <SELECT> key only works if there are several selection options available.

- OR -

Press the <INSERT> key.
The selection options are displayed in a list.
2. Select the required setting using the <Cursor down> and <Cursor up> keys.
3. If required, enter a value in the associated input field.
4. Press the <INPUT> key to complete the parameter input.

## Changing or calculating parameters

If you only want to change individual characters in an input field rather than overwriting the entire entry, switch to insertion mode.

In this mode, you can also enter simple calculation expressions, without having to explicitly call the calculator. You can execute the four basic calculation types, work with expressions in brackets as well as generate square roots and squares.

## Note

## Generating square roots and squares

The extract roots and generate square functions is not available in the parameter screens of the cycles and functions in the "Program" operating area.


Press the <INSERT> key.
The insert mode is activated.
You can navigate within the input field using the <Cursor left> and <Cursor right> keys.

Use the <BACKSPACE> and <DEL> key to delete individual characters.


Enter the multiplication characters using the <SHIFT> + <*> keys.


Enter the division character using the <SHIFT> + </> keys.


Enter bracket expressions using the <SHIFT> + <(> and <SHIFT> + <)> keys.

$\mathrm{R} \quad+\begin{aligned} & \text { <number> Enter "r" or "R" as well as the number } \mathrm{x} \text { from which you would like to } \\ & \text { extract the root. }\end{aligned}$
$\mathrm{R} \quad+\begin{aligned} & \text { <number> Enter "r" or "R" as well as the number } \mathrm{x} \text { from which you would like to } \\ & \text { extract the root. }\end{aligned}$
$\mathrm{S} \quad+\begin{aligned} & + \text { <number> Enter "s" or "S" as well as the number } \mathrm{x} \text { for which you would like to gen- } \\ & \text { erate the square. }\end{aligned}$

## Accepting parameters

When you have correctly entered all necessary parameters, you can close the window and save your settings.
You cannot accept the parameters if they are incomplete or obviously erroneous. In this case, you can see from the dialog line which parameters are missing or were entered incorrectly.


Close the value entry using the <INPUT> key and the result is transferred into the field.

### 2.4.8 Pocket calculator

## Procedure

1. Position the cursor on the desired entry field.
2. Press the <=> key.

The calculator is displayed.
3. Input the arithmetic statement.

You can use arithmetic symbols, numbers, and commas.
4. Press the equals symbol on the calculator.

- OR -

Press the "Calculate" softkey.

- OR -

Press the <INPUT> key.
The new value is calculated and displayed in the entry field of the calculator.
5. Press the "Accept" softkey.

The calculated value is accepted and displayed in the entry field of the window.

## Note

## Input order for functions

When using the square root or squaring functions, make sure to press the "R" or "S" function keys, respectively, before entering a number.

### 2.4.9 Context menu

When you right-click, the context menu opens and provides the following functions:

- Cut

Cut Ctrl+X

- Copy

Copy Ctrl+C

- Paste

> Paste Ctrl+V

## Program editor

Additional functions are available in the editor

- Undo the last change

Undo Ctrl+Z

- Redo the changes that were undone

Redo Ctrl+Y
Up to 50 changes can be undone.

### 2.4.10 Touch operation

If you have an operator panel with a touch screen, you can perform the following functions with touch operation:

## Operating area switchover

M. You can display the operating area menu by touching the display symbol for the active operating area in the status display.


## Channel switchover

You can switch over to the next channel by touching the channel display in the status display.

### 2.4.11 Changing the user interface language

## Procedure



1. Select the "Start-up" operating area.
2. Press the "Change language" softkey.

The "Language selection" window opens. The language set last is selected.
3. Position the cursor on the desired language.
4. Press the "OK" softkey.

- OR -

Press the <INPUT> key.

The user interface changes to the selected language.

## Note

## Changing the language directly on the input screens

You can switch between the user interface languages available on the controller directly on the user interface by pressing the key combination $<C T R L+L>$.

### 2.4.12 Entering Chinese characters

### 2.4.12.1 Function - input editor

Using the input editor IME (input method editor), you can select Asian characters where you enter the phonetic notation. These characters are transferred into the user interface.

## Note

Call the input editor with <Alt + S>
The input editor can only be called there where it is permissible to enter Asian characters.

The editor is available for the following Asian languages:

- Simplified Chinese
- Traditional Chinese


### 2.4 User interface

## Input types

| Input type | Description |
| :--- | :--- |
| Pinyin input | Latin letters are combined phonetically to denote the sound of the character. <br> The editor lists all of the characters from the dictionary that can be selected. |
| Zhuyin input <br> (only traditional Chinese) | Non-Latin letters are combined phonetically to denote the sound of the character. <br> The editor lists all of the characters from the dictionary that can be selected. |
| Entering Latin letters | The characters that are entered are directly transferred into the input field, from <br> where the editor was called. |

## Structure of the editor



Figure 2-4 Example: Pinyin input


Figure 2-5 Example: Zhuyin input

## Functions

中 Pinyin input
A Entering Latin letters

国
Editing the dictionary

## Dictionaries

The simplified Chinese and traditional Chinese dictionaries that are supplied can be expanded:

- If you enter new phonetic notations, the editor creates a new line. The entered phonetic notation is broken down into known phonetic notations. Select the associated character for each component. The compiled characters are displayed in the additional line. Accept the new word into the dictionary and into the input field by pressing the <Input> key.
- Using any Unicode editor, you can enter new phonetic notations into a text file. These phonetic notations are imported into the dictionary the next time that the input editor is started.


### 2.4.12.2 Entering Asian characters

## Precondition

The control has been switched over to Chinese.

## Procedure

## Editing characters using the Pinyin method

$+$

## S

2. Enter the desired phonetic notation using Latin letters. Use the upper input field for traditional Chinese
3. Press the <Cursor down> key to reach the dictionary.
4. Keeping the <Cursor down> key pressed, displays all the entered phonetic notations and the associated selection characters.
5. Press the <BACKSPACE> softkey to delete entered phonetic notations.
6. Press the number key to insert the associated character.

When a character is selected, the editor records the frequency with which it is selected for a specific phonetic notation and offers this character at the top of the list when the editor is next opened.

## Editing characters using the Zhuyin method (only traditional Chinese)

A
$+$
s
2. Enter the desired phonetic notation using the numerical block.

Each number is assigned a certain number of letters that can be selected by pressing the numeric key one or several times.
3. Press the <Cursor down> key to reach the dictionary.
4. Keeping the <Cursor down> key pressed, displays all the entered phonetic notations and the associated selection characters.
5. Press the <BACKSPACE> softkey to delete entered phonetic notations.

6. To select the associated character, press the <cursor right> or <cursor left> keys.
7. Press the <input> key to enter the character.

### 2.4.12.3 Editing the dictionary

## Learning function of the input editor

## Precondition:

The control has been switched over to Chinese.
An unknown phonetic notation has been entered into the input editor.

1. The editor provides a further line in which the combined characters and phonetic notations are displayed.
The first part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary. Various characters are listed for this particular phonetic notation.
2. Press the number key to insert the associated character into the additional line.
The next part of the phonetic notation is displayed in the field for selecting the phonetic notation from the dictionary.
3. Repeat step 2 until the complete phonetic notation has been compiled.

Press the＜TAB＞key to toggle between the compiled phonetic notation field and the phonetic notation input．
Compiled characters are deleted using the＜BACKSPACE＞key．

4．Press the＜input＞key to transfer the compiled phonetic notation into the dictionary and the input field．

## Importing a dictionary

A dictionary can now be generated using any Unicode editor by attaching the corresponding Chinese characters to the pinyin phonetic spelling．If the phonetic spelling contains several Chinese characters，then the line must not contain any additional match．If there are several matches for one phonetic spelling，then these must be specified in the dictionary line by line． Otherwise，several characters can be specified for each line．
The generated file should be saved in the UTF8 format under the name chs＿user．txt （simplified Chinese）or cht＿user．txt（traditional Chinese）．
Line structure：
Pinyin phonetic spelling＜TAB＞Chinese character＜LF＞
OR
Pinyin phonetic spelling＜TAB＞Chinese character1＜TAB＞Chinese character2＜TAB＞．．． ＜LF＞
＜TAB＞－tab key
＜LF＞－line break
Store the created dictionary in one of the following paths：
．．／user／sinumerik／hmi／ime／
．．／oem／sinumerik／hmi／ime／
When the Chinese editor is called the next time，this enters the content of the dictionary into the system dictionary．

## Example：

| ai | 哎 | 哀 唉 埃 挨 |  |
| :--- | :--- | :--- | :--- |
| caise | 彩色 |  |  |
| hongse | 紅色 |  |  |
| huise | 灰色 |  |  |
| heli | 河罗 |  |  |
| zuihaowan | 最好玩 |  |  |

## 2．4．13 Entering Korean characters

You can enter Korean characters in the input fields using the input editor IME（Input Method Editor）．

## Note

You require a special keyboard to enter Korean characters．If this is not available，then you can enter the characters using a matrix．

## Korean keyboard

To enter Korean characters，you will need a keyboard with the keyboard assignment shown below．In terms of key layout，this keyboard is the equivalent of an English QWERTY keyboard and individual events must be grouped together to form syllables．


## Structure of the editor

ットリ
T ᄃ ㅃㄼ
॥
ㅊ ㅋㅌ ㅊ 표랜낼【


## Functions

| Matrix | Editing characters using a matrix |
| :--- | :--- |
| Beolsik 2 | Editing characters using the keyboard |
| 한 | Entering Korean characters |
| A | Entering Latin letters |

Precondition
The control has been switched over to Korean．

## Procedure

## Editing characters using the keyboard

1. Open the screen form and position the cursor on the input field.

Press the <Alt + S> keys.
The editor is displayed.

S

2. Switch to the "Keyboard - Matrix" selection box.
3. Select the keyboard.
4. Switch to the function selection box.

5. Select Korean character input.
6. Enter the required characters.
7. Press the <input> key to enter the character into the input field.

## Editing characters using a matrix

$+$

S

2. Switch to the "Keyboard - Matrix" selection box.
3. Select the "matrix".
4. Switch to the function selection box.
5. Select Korean character input.
6. Enter the number of the line in which the required character is located. The line is highlighted in color.
7. Enter the number of the column in which the required character is located.
The character will be briefly highlighted in color and then transferred to the Character field.
Press the <BACKSPACE> softkey to delete entered phonetic notations.
8. Press the <input> key to enter the character into the input field.

### 2.4.14 Protection levels

The input and modification of data in the control system is protected by passwords at sensitive places.

## Access protection via protection levels

The input or modification of data for the following functions depends on the protection level setting:

- Tool offsets
- Work offsets
- Setting data
- Program creation / program editing

Note
Configuring access levels for softkeys
You have the option of providing softkeys with protection levels or completely hiding them.

## References

For additional information, please refer to the following documentation:
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

## Softkeys

| Machine operating area | Protection level |
| :--- | :--- |
| SYNC Synch． | End user <br> （protection level 3） |


| Parameters operating area | Protection level |
| :--- | :--- |
| Tool management lists | Keyswitch 3 <br> （protection level 4） |
| Details |  |


| Diagnostics operating area | Protection level |
| :---: | :--- |
| Logbook | Keyswitch 3 <br> （protection level 4） |
| Change | User <br> （protection level 3） |
| New <br> entry | User <br> （protection level 3） |
| Startup <br> complete | Manufacturer <br> （protection level 1） |
| Machine <br> installed | User <br> （protection level 3） |
| Add HW <br> comp． | Service <br> （protection level 2） |


| Start－up operating area |  | Protection levels |
| :---: | :---: | :---: |
| $\begin{array}{cc} \text { 品 } & \text { System } \\ \text { data } \end{array}$ |  | End user （protection level 3） |
| 圈罝 Setup |  | Keyswitch 3 （protection level 4） |
| General MD | Control Unit parameter | Keyswitch 3 （protection level 4） |
| $\begin{gathered} \text { Li- } \\ \text { censes } \end{gathered}$ |  | Keyswitch 3 （protection level 4） |

Turning

| Start-up operating area | Protection levels |
| :---: | :--- |
| Set MD <br> active (cf) | Keyswitch 3 <br> (protection level 4) |
| Reset <br> (po) | End user <br> (protection level 3) |
| Change <br> password | End user <br> (protection level 3) |
| Delete <br> password | End user <br> (protection level 3) |

### 2.4.15 Online help in SINUMERIK Operate

A comprehensive context-sensitive online help is stored in the control system.

- A brief description is provided for each window and, if required, step-by-step instructions for the operating sequences.
- A detailed help is provided in the editor for every entered G code. You can also display all G functions and take over a selected command directly from the help into the editor.
- A help page with all parameters is provided on the input screen in the cycle programming.
- Lists of the machine data
- Lists of the setting data
- Lists of the drive parameters
- List of all alarms


## Procedure

## Calling context-sensitive online help

1. You are in an arbitrary window of an operating area.


## Gurrent

topic

Fullscreen

Fullscreen
P
2. Press the <HELP> key or on an MF2 keyboard, the <F12> key. The help page of the currently selected window is opened in a subscreen.
3. Press the "Full screen" softkey to use the entire user interface for the display of the online help.

Press the "Full screen" softkey again to return to the subscreen.
4. If further helps are offered for the function or associated topics, position the cursor on the desired link and press the "Follow reference" softkey. The selected help page is displayed.
5. Press the "Back to reference" softkey to jump back to the previous help.

## Calling a topic in the table of contents

Table of contents

2. Select the desired manual with the <Cursor down> and <Cursor up> keys.

3 Press the <Cursor right> or <INPUT> key or double-click to open the manual and the section.
4. Navigate to the desired topic with the "Cursor down" key.
5. Press the <Follow reference> softkey or the <INPUT> key to display the help page for the selected topic.

Follow reference

1. Press the "Table of contents" softkey.

Depending on which technology you are using, the Operating Manuals "Operator control Milling", "Operator control Turning" or "Operator control Universal" as well as the "Programming" Programming Manual are displayed.


## Gurrent

6. Press the "Current topic" softkey to return to the original help.

## Searching for a topic

Search

1. Press the "Search" softkey.

The "Search in Help for:" window is opened.
2. Activate the "Full text " checkbox to search in all help pages.

If the checkbox is not activated, a search is performed in the table of contents and in the index.
3. Enter the desired keyword in the "Text" field and press the "OK" softkey. If you enter the search term on the operator panel, replace an umlaut (accented character) by an asterisk (*) as dummy.
All entered terms and sentences are sought with an AND operation. In this way, only documents and entries that satisfy all the search criteria are displayed.
4. Press the "Keyword index" softkey if you only want to display the index of the operating and programming manual.

## Displaying alarm descriptions and machine data

1. If messages or alarms are pending in the "Alarms", "Messages" or "Alarm Log" window, position the cursor at the appropriate display and press the <HELP> or the <F12> key. The associated alarm description is displayed.
2. If you are in the "Start-up" operating area in the windows for the display of the machine, setting and drive data, position the cursor on the desired machine data or drive parameter and press the <HELP> or the <F12> key.
The associated data description is displayed.

## Displaying and inserting a G code command in the editor

1. A program is opened in the editor.

Position the cursor on the desired $G$ code command and press the <HELP> or the <F12> key. The associated $G$ code description is displayed.

Display all
G functions

## Search

Transfer to editor

Exit Help

## Setting up the machine

### 3.1 Switching on and switching off

## Startup



When the control starts up, the main screen opens according to the operating mode specified by the machine manufacturer. In general, this is the main screen for the "REF POINT" submode.


## Machine manufacturer

Please also refer to the machine manufacturer's instructions.

### 3.2 Approaching a reference point

### 3.2.1 Referencing axes

Your machine tool can be equipped with an absolute or incremental path measuring system. An axis with incremental path measuring system must be referenced after the controller has been switched on - however, an absolute path measuring system does not have to be referenced.

For the incremental path measuring system, all the machine axes must therefore first approach a reference point, the coordinates of which are known to be relative to the machine zero-point.

## Sequence

Prior to the approach, the axes must be in a position from where they can approach the reference point without a collision.
The axes can also all approach the reference point simultaneously, depending on the manufacturer's settings.

## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## NOTICE

## Risk of collision

If the axes are not in a collision-free position, you must first traverse them to safe positions in "JOG" or "MDI" mode.

You must follow the axis motions directly on the machine!
Ignore the actual value display until the axes have been referenced!
The software limit switches are not active!

## Procedure



1. Press the <JOG> key.
2. Press the <REF. POINT> key.
3. Select the axis to be traversed.


## $+$

4. Press the <-> or <+> key.

The selected axis moves to the reference point.
If you have pressed the wrong direction key, the action is not accepted and the axes do not move.

A symbol is shown next to the axis if it has been referenced.

The axis is referenced as soon as the reference point is reached. The actual value display is set to the reference point value.

From now on, path limits, such as software limit switches, are active.
End the function via the machine control panel by selecting operating mode "AUTO" or "JOG".

### 3.2.2 User agreement

If you are using Safety Integrated (SI) on your machine, you will need to confirm that the current displayed position of an axis corresponds to its actual position on the machine when you reference an axis. Your confirmation is the requirement for the availability of other Safety Integrated functions.

You can only give your user agreement for an axis after it has approached the reference point.

The displayed axis position always refers to the machine coordinate system (Machine).

## Option

User agreement with Safety Integrated is only possible with a software option.

## Procedure




The selected axis is marked with an "x" meaning "safely referenced" in the "Acknowledgement" column.
By pressing the <SELECT> key again, you deactivate the acknowledgement again.

### 3.3 Modes and mode groups

### 3.3.1 General

You can work in three different operating modes.
"JOG" mode
"JOG" mode is used for the following preparatory actions:

- Approach reference point, i.e. the machine axis is referenced
- Preparing a machine for executing a program in automatic mode, i.e. measuring tools, measuring the workpiece and, if necessary, defining the work offsets used in the program
- Traversing axes, e.g. during a program interruption
- Positioning axes


## Select "JOG"


Press the <JOG> key.

## "REF POINT" operating mode

The "REF POINT" operating mode is used to synchronize the control and the machine. For this purpose, you approach the reference point in "JOG" mode.

## Selecting "REF POINT"

Press the <REF POINT> key.

## "REPOS" operating mode

The "REPOS" operating mode is used for repositioning to a defined position. After a program interruption (e.g. to correct tool wear values) move the tool away from the contour in "JOG" mode.

The distances traversed in "JOG" mode are displayed in the actual value window as the "Repos" offset.
"REPOS" offsets can be displayed in the machine coordinate system (MCS) or workpiece coordinate system (WCS).

## Selecting "Repos"



Press the <REPOS> key.

## "MDI" mode (Manual Data Input)

In "MDI" mode, you can enter and execute G code commands non-modally to set up the machine or to perform a single action.

## Selecting "MDI"

## "AUTO" mode

In automatic mode, you can execute a program completely or only partially.

## Select "AUTO"



Turning

## "TEACH IN" operating mode

"TEACH IN" is available in the "AUTO" and "MDI" operating modes.
There you may create, edit and execute part programs (main programs or subroutines) for motional sequences or simple workpieces by approaching and saving positions.

Selecting "Teach In"
匀
Press the <TEACH IN> key.
TEACH IN

### 3.3.2 Modes groups and channels

Every channel behaves like an independent NC. A maximum of one part program can be processed per channel.

- Control with 1channel

One mode group exists.

- Control with several channels

Channels can be grouped to form several "mode groups."

## Example

Control with 4 channels, where machining is carried out in 2 channels and 2 other channels are used to control the transport of the new workpieces.
Mode group 1 channel 1 (machining)
Channel 2 (transport)
Mode group 2 channel 3 (machining)
Channel 4 (transport)

## Mode groups (MGs)

Technologically-related channels can be combined to form a mode group.
Axes and spindles of the same mode group can be controlled by one or more channels.
An operating mode group is in one of "Automatic", "JOG" or "MDI" operating modes, i.e., several channels of an operating mode group can never assume different operating modes.

### 3.3.3 Channel switchover

It is possible to switch between channels when several are in use. Since individual channels may be assigned to different mode groups, a channel switchover command is also an implicit mode switchover command.

When a channel menu is available, all of the channels are displayed on softkeys and can be switched over.

## Changing the channel



Press the <CHANNEL> key.

The channel changes over to the next channel.

- OR -

If the channel menu is available, a softkey bar is displayed. The active channel is highlighted.
Another channel can be selected by pressing one of the other softkeys.

## References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

## Channel switchover via touch operation

On the HT 8 and when using a touch screen operator panel, you can switch to the next channel or display the channel menu via touch operation in the status display.

### 3.4 Settings for the machine

### 3.4.1 Switching over the coordinate system (MCS/WCS)

The coordinates in the actual value display are relative to either the machine coordinate system or the workpiece coordinate system.

By default, the workpiece coordinate system is set as a reference for the actual value display.

The machine coordinate system (MCS), in contrast to the workpiece coordinate system (WCS), does not take into account any zero offsets, tool offsets and coordinate rotation.

## Procedure



Machine

2. Press the <JOG> or <AUTO> key.


Act. vis. MCS

Act. vis. MGS

The machine coordinate system is selected.
The title of the actual value window changes in the MCS.

## Machine manufacturer

The softkey to changeover the coordinate system can be hidden. Please refer to the machine manufacturer's specifications.

### 3.4.2 Switching the unit of measurement

You can set millimeters or inches as the unit of measurement. Switching the unit of measurement always applies to the entire machine. All required information is automatically converted to the new unit of measurement, for example:

- Positions
- Tool offsets
- Work offsets

Machine manufacturer
Please also refer to the machine manufacturer's instructions.

## Proceed as follows



Machine

2. Press the menu forward key and the "Settings" softkey. A new vertical softkey bar appears.

Changeower inch
3. Press the "Switch to inch" softkey. A prompt asks you whether you really want to switch over the unit of measurement.

4. Press the "OK" softkey.

Changeover metric

1. Select <JOG> or <AUTO> mode in the "Machine" operating area.

> The softkey label changes to "Switch to metric".
> The unit of measurement applies to the entire machine.
> 5. Press the "Switch to metric" softkey to set the unit of measurement of the machine to metric again.

## See also

Default settings for manual mode (Page 122)

### 3.4.3 Setting the zero offset

You can enter a new position value in the actual value display for individual axes when a settable zero offset is active.
The difference between the position value in the machine coordinate system MCS and the new position value in the workpiece coordinate system WCS is saved permanently in the currently active zero offset (e.g. G54).

## Relative actual value

Further, you also have the possibility of entering position values in the relative coordinate system.

## Note

The new actual value is only displayed. The relative actual value has no effect on the axis positions and the active zero offset.

## Resetting the relative actual value

Delete REL
Press the "Delete REL" softkey.

The actual values are deleted.
The softkeys to set the zero point in the relative coordinate system are only available if the corresponding machine data is set.


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Precondition

The controller is in the workpiece coordinate system.
The actual value can be set in both the Reset and Stop state.

## Note

## Setting the WO in the Stop state

If you enter the new actual value in the Stop state, the changes made are only visible and only take effect when the program is continued.

## Procedure



1. Select the "JOG" mode in the "Machine" operating area.

## Machine

## $\mathrm{M}_{\mathrm{Jos}}^{\mathrm{M}}$



REL act. vals
$\int_{20}^{8}$ Set
20 REL

2. Press the "Set WO" softkey.

- OR -

Press the ">>", "REL act. vals" and "Set REL" softkeys to set position values in the relative coordinate system.
3. Enter the new required position value for $Z, X$ or $Y$ directly in the actual value display (you can toggle between the axes with the cursor keys) and press the <INPUT> key to confirm the entries.

- OR -

Press softkey " $Z=0$ ", " $X=0$ " or " $Y=0$ " (if there is a $Y$ axis), to set the required position to zero.

Press the "Delete active WO" softkey.
The offset is deleted permanently.

## NOTICE

Irreversible active zero offset
The current active zero offset is irreversibly deleted by this action.

### 3.5 Measuring the tool

The geometries of the machining tool must be taken into consideration when executing a part program. These are stored as tool offset data in the tool list. Each time the tool is called, the control considers the tool offset data.

When programming the part program, you only need to enter the workpiece dimensions from the production drawing. After this, the controller independently calculates the individual tool path.

Drilling and milling tools
You can determine the tool offset data, i.e. the length and radius or diameter, either manually or automatically with tool probes.

## Turning tools

You can specify the tool offset data, i.e. the length, either manually or automatically using a tool probe.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

## Logging measurement results

After you have completed the measurement, you have the option to output the displayed values to a log. You can define whether the log file that is generated is continually written to for each new measurement, or is overwritten.

## See also

Logging tool measurement results (Page 84)
Settings for the measurement result log (Page 88)

### 3.5.1 Measuring a tool manually

When measuring manually, traverse the tool manually to a known reference point in order to determine the tool dimensions in the $X$ and $Z$ directions. The control system then calculates the tool offset data from the position of the tool carrier reference point and the reference point.

## Reference point

The workpiece edge is used as the reference point when measuring length $X$ and length $Z$.
The chuck of the main or counterspindle can also be used when measuring in the $Z$ direction.

You specify the position of the workpiece edge during the measurement.

## Note

## Lathes with B axis

For lathes with a $B$ axis, execute the tool change and alignment in the $T, S, M$ window before performing the measurement.

## Procedure



Machine

2. Press the "Meas. tool" softkey.
3. Press the "Manual" softkey.

## Manual

## Select

 tool
8. Scratch the required edge using the tool.
4. Press the "Select tool" softkey. The "Tool selection" window is opened.
5. Select the tool that you wish to measure. already be entered in the tool list.
6. Press the "In manual" softkey.

The tool is accepted into the "Length Manual" window. measure. "Save position" softkey.

1. Select "JOG" mode in the "Machine" operating area. The cutting edge position and the radius or diameter of the tool must
2. Press the " X " or " $Z$ " softkey, depending on which tool length you want to
3. If you do not wish to keep the tool at the workpiece edge, then press the

The tool position is saved and the tool can be retracted from the workpiece. For instance, this can be practical if the workpiece diameter still has to be subsequently measured.
If the tool can remain at the workpiece edge, then after scratching you can directly continue with step 11.
10. Enter the position of the workpiece edge in XO or ZO .

If no value is entered for XO or ZO , the value is taken from the actual value display.
11. Press the "Set length" softkey.

The tool length is calculated automatically and entered in the tool list. Whereby the cutting edge position and tool radius or diameter are automatically taken into consideration as well.

## Note

Tool measurement is only possible with an active tool.

### 3.5.2 Measuring a tool with a tool probe

During automatic measuring, you determine the tool dimensions in the directions $X$ and $Z$ with the aid of a probe.
You have the possibility of measuring a tool using a tool holder that can be orientated (tool carrier, swivel).
The function "Measure with tool carrier that can be orientated" is implemented for lathes with a swivel axis around $Y$ and associated tool spindle. The swivel axis can be used to align the tool on the $X / Z$ level. The swivel axis can assume any position around $Y$ to measure turning tools. Multiples of $90^{\circ}$ are permitted for milling and drilling tools. Multiples of $180^{\circ}$ are possible when positioning the tool spindle.

## Note

## Lathes with B axis

For lathes with a $B$ axis, execute the tool change and alignment in the $T, S, M$ window before performing the measurement.

## Adapting the user interface to calibrating and measuring functions

The corresponding windows can be adapted to the measurement tasks in order to automatically measure tools.
The tool offset data is calculated from the known position of the tool carrier reference point and the probe.
The following selection options can be switched-in or switched-out:

- Calibration plane, measurement plane
- Probe
- Calibration feedrate (measuring feedrate)


## References

For further information about lathes with $B$ axis, please refer to the following reference:
Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

## Preconditions

- If you wish to measure your tools with a tool probe, the machine manufacturer must parameterize special measuring functions for that purpose.
- Enter the cutting edge position and the radius or diameter of the tool in the tool list before performing the actual measurement. If the tool is measured using a tool carrier that can be orientated, then the cutting edge position must be entered into the tool list corresponding to the initial tool carrier position.
- Calibrate the probe first.


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Procedure

2. Select "JOG" mode in the "Machine" operating area.

Meas. tool
3. Press the "Meas. tool" and "Automatic" softkeys.
4. Press the " X " or " $Z$ " softkey, depending on which tool length you want to measure.

Z

## Automatic

 measure.1. Insert the tool that you want to measure.

If the tool is to be measured using a tool carrier that can be orientated, then at this position the tool should be aligned in the same way that it will be subsequently measured.
5. Manually position the tool in the vicinity of the tool probe in such a way that any collisions can be avoided when the tool probe is being traversed in the corresponding direction.


### 3.5.3 Calibrating the tool probe

To be able to measure your tools automatically, you must first determine the position of the tool probe in the machine area in relation to the machine zero.


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Sequence

The calibrating tool must be a turning tool type (roughing or finishing tool). Cutting edge positions 1-4 can be used for the tool probe calibration. You must enter the length and the radius or diameter of the calibrating tool in the tool list.

Calibrate the probe in all directions in which you wish to subsequently perform measurements.

## Procedure

1. Change the calibrating tool.

2. Select the "JOG" mode in the "Machine" operating area.
3. Press the "Meas. tool" and "Calibrate probe" softkeys.
4. Press the "X" or "Z" softkey, depending on which point of the tool probe you wish to determine first.
5. Select the direction (+ or -), in which you would like to approach the tool probe.
6. Position the calibrating tool in the vicinity of the tool probe in such a way that any collisions can be avoided when the first point of the tool probe is being approached.
7. Press the <CYCLE START> key.

The calibration process is started, i.e. the calibrating tool is automatically traversed at the measurement feedrate to the probe and back again. The position of the tool probe is determined and saved in an internal data area.
8. Repeat the process for the other other points of the tool probe.

### 3.5.4 Measuring a tool with a magnifying glass

You can also use a magnifying glass to determine the tool dimensions, if this is available on the machine.

In this case, SINUMERIK Operate calculates the tool offset data from the known positions of the tool carrier reference point and the cross-hairs of the magnifying glass.

## Note

## Lathes with $B$ axis

For lathes with a $B$ axis, execute the tool change and alignment in the $T, S, M$ window before performing the measurement.

## Procedure



1. Select the "JOG" mode in the "Machine" operating area.
2. Press the "Meas. tool" softkey.
3. Press the "Zoom" soffkey.
4. Press the "Select tool" softkey.

The "Tool selection" window is opened.
5. Select the tool that you wish to measure.

The cutting edge position and the radius or diameter of the tool must already be entered in the tool list.
6. Press the "In manual" softkey.

The tool is accepted in the "Zoom" window.
7. Traverse the tool towards the magnifying glass and align the tool tip $P$ with the magnifying glass cross-hairs.

### 3.5.5 Logging tool measurement results

After measuring a tool, you have the option to output the measured values to a log.
The following data are determined and logged:

- Date/time
- Log name with path
- Measuring version
- Input values
- Correction target
- Setpoints, measured values and differences


## Note

## Logging active

The measurement results can only be entered into a log once the measurement has been fully completed.

## Procedure

4. Meas. ..... tool
5. You are in the "JOG" mode and have pressed the "Measure tool" softkey.
The "Measurement log" softkey cannot be used.
6. Insert the tool, select the measuring version and measure the tool as usual.
The tool data are displayed once the measurement has been completed.
The "Measurement log" softkey can be operated.
Measure. 3. Press the "Measurement log" softkey to save the measurement data as report log. The "Measurement log" softkey becomes inactive again.

## See also

Settings for the measurement result log (Page 88)

### 3.6 Measuring the workpiece zero

### 3.6.1 Measuring the workpiece zero

The reference point for programming a workpiece is always the workpiece zero. To determine this zero point, measure the length of the workpiece and save the position of the cylinder's face surface in the direction $Z$ in a zero offset. This means that the position is stored in the coarse offset and existing values in the fine offset are deleted.

## Calculation

When the workpiece zero / zero offset is calculated, the tool length is automatically taken into account.

## Measuring only

If you wish to measure the workpiece zero in "Measuring Only" mode, the measured values are merely displayed without any changes being made to the coordinate system.

## Adapting the user interface to the measurement functions

The following selection options can be switched-in or switched-out:

- Calibration plane, measuring plane (only 840D sl)
- Zero offset as basis for the measuring process (only 840D sl)
- Number of the probe calibration data record (only 840D sl)
- Offset target, settable zero offset
- Offset target, basis reference
- Offset target, global basic zero offset (only 840D sl)
- Offset target, channel-specific basic zero offset (only 840D sI)

Machine manufacturer
Please refer to the machine manufacturer's specifications.

## Logging the measurement result

After you have completed the measurement, you have the option to output the displayed values in a log. You can define whether the log file that is generated is continually written to for each new measurement, or is overwritten.

## Precondition

The requirement for measuring the workpiece is that a tool with known lengths is in the machining position.

## Procedure

1. Select "JOG" mode in the "Machine" operating area.

## Machine


2. Press the "Workpiece zero" softkey. The "Set Edge" window opens.
3. Select "Measuring only" if you only want to display the measured values.

- OR -

Select the desired zero offset in which you want to store the zero point (e.g. basis reference).

- OR -

Work offset

G54... G599

In manual
4. Traverse the tool in the $Z$ direction and scratch the workpiece.

Set
wo
5. Enter the position setpoint of the workpiece edge $Z 0$ and press the "Set ZO" softkey.

## Note

## Settable zero offsets

The labeling of the softkeys for the settable zero offsets varies, i.e. the settable zero offsets configured on the machine are displayed (examples: G54...G57, G54...G505, G54...G599). Please refer to the machine manufacturer's specifications.

### 3.6.2 Logging measurement results for the workpiece zero

When measuring the workpiece zero, you have the option to output the values that have been determined to a log.
The following data are determined and logged:

- Date/time
- Log name with path
- Measuring version
- Input values
- Correction target
- Setpoints, measured values and differences


## Note <br> Logging active

The measurement results can only be entered into a log once the measurement has been fully completed.

## Procedure

## ]. Meas. <br> workp.

1. You are in the "JOG" mode and have pressed the "Workpiece zero" softkey.
The "Measurement log" softkey cannot be used.
2. Select the required measurement version and measured the workpiece zero as usual.

The measured values are displayed once the measurement has been completed.

The "Measurement log" sofkey cannot be used.
2. Press the "Measurement log" softkey to save the measurement data as log.
The "Measurement log" softkey becomes inactive again.

### 3.7 Settings for the measurement result log

Make the following settings in the "Settings for measurement log" window:

- Log format
- Text format

The log in the text format is based on the display of the measurement results on the screen.

- Tabular format

When selecting the tabular format, the measurement results are saved so that the data can be imported into a spreadsheet program (e.g. Microsoft Excel). This allows the measurement result logs to be statistically processed.

- Log data
- new

The log of the actual measurement is created under the specified name. Existing logs with the same name are overwritten.

- attach

The log created is attached to the previous log.

- Where the log is saved

The log created is saved in a specified directory.

## Procedure

' ${ }^{\prime}$
Machine


Settings

Measure. report
4. Press the "Measurement log" softkey. The "Settings for measurement log" window is opened.
5. Position the cursor to the log format field and select the required entry.
6. Position the cursor to the log data field and select the required entry.
7. Position the cursor to the log archive field and press the softkey "Select directory".
8. Navigate to the desired directory for the log archive.
9. Press the "OK" softkey and enter the name for the log file.

## See also

Logging tool measurement results (Page 84)
Logging measurement results for the workpiece zero (Page 87)

## $3.8 \quad$ Zero offsets

Following reference point approach, the actual value display for the axis coordinates is based on the machine zero ( M ) of the machine coordinate system (Machine). The program for machining the workpiece, however, is based on the workpiece zero (W) of the workpiece coordinate system (Work). The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and the workpiece zero depends on the workpiece type and how it is clamped. This zero offset is taken into account during execution of the program and can be a combination of different offsets.
Following reference point approach, the actual value display for the axis coordinates is based on the machine zero of the machine coordinate system (Machine).
The actual value display of the positions can also refer to the SZS coordinate system (settable zero system). The position of the active tool relative to the workpiece zero is displayed.


Figure 3-1 Zero offsets
When the machine zero is not identical to the workpiece zero, at least one offset (base offset or zero offset) exists in which the position of the workpiece zero is saved.

## Base offset

The base offset is a zero offset that is always active. If you have not defined a base offset, its value will be zero. The base offset is specified in the "Zero Offset - Base" window.

## Coarse and fine offsets

Every zero offset (G54 to G57, G505 to G599) consists of a coarse offset and a fine offset. You can call the zero offsets from any program (coarse and fine offsets are added together).

You can save the workpiece zero, for example, in the coarse offset, and then store the offset that occurs when a new workpiece is clamped between the old and the new workpiece zero in the fine offset.

## Note

## Deselect fine offset (only 840D sl)

You have the option of deselecting the fine offset using machine data MD18600 \$MN_MM_FRAME_FINE_TRANS

## See also

Actual value window (Page $\underline{45}$ )

### 3.8.1 Display active zero offset

The following zero offsets are displayed in the "Zero Offset - Active" window:

- Zero offsets, for which active offsets are included, or for which values are entered.
- Settable zero offsets
- Total zero offset

This window is generally used only for monitoring.
The availability of the offsets depends on the setting.

## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Procedure



1. Select the "Parameter" operating area.

## Parameter

Work offset

Active

2. Press the "Zero offset" softkey.

The "Zero Offset - Active" window is opened.

## Note

## Further details on zero offsets

If you would like to see further details about the specified offsets or if you would like to change values for the rotation, scaling or mirroring, press the "Details" softkey.

### 3.8.2 Displaying the zero offset "overview"

The active offsets or system offsets are displayed for all axes that have been set up in the "Work offset - overview" window.

In addition to the offset (course and fine), the rotation, scaling and mirroring defined using this are also displayed.
This window is generally used only for monitoring.

## Display of active work offsets

| Work offsets |  |
| :--- | :--- |
| DRF | Displays the handwheel axis offset. |
| Rotary table reference | Displays the additional work offsets programmed with \$ <br> P_PARTFRAME. |
| Basic reference | Displays the additional work offsets programmed with \$P_SETFRAME. <br> Access to the system offsets is protected via a keyswitch. |
| External WO frame | Displays the additional work offsets programmed with \$P_EXTFRAME. |
| Total base WO | Displays all effective basis offsets. |
| G500 | Displays the work offsets activated with G54 - G599. <br> Under certain circumstances, you can change the data using "Set <br> WO", i.e. you can correct a zero point that has been set. |
| Tool reference | Displays the additional work offsets programmed with <br> \$P_TOOLFRAME. |
| Workpiece reference | Displays the additional work offsets programmed with \$P_WPFRAME. |
| Programmed WO | Displays the additional work offsets programmed with \$P_PFRAME. |
| Cycle reference | Displays the additional work offsets programmed with <br> \$P_CYCFRAME. |
| Total WO | Displays the active work offset, resulting from the total of all work off- <br> sets. |

## Procedure

Parameter

## Ouerview

1. Select the "Parameter" operating area.

2 Press the "Work offset" and "Overview" softkeys.
The "Work offsets - Overview" window opens.

### 3.8.3 Displaying and editing base zero offset

The defined channel-specific and global base offsets, divided into coarse and fine offsets, are displayed for all set-up axes in the "Zero offset - Base" window.


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Procedure



1. Select the "Parameter" operating area.
2. Press the "Zero offset" softkey.
3. Press the "Base" softkey.

The "Zero Offset - Base" window is opened.
4. You can edit the values directly in the table.

## Note

## Activate base offsets

The offsets specified here are immediately active.

### 3.8.4 Displaying and editing settable zero offset

All settable offsets, divided into coarse and fine offsets, are displayed in the "Zero Offset G54..G599" window.
Rotation, scaling and mirroring are displayed.

## Procedure



Parameter

654...6599

1. Select the "Parameter" operating area.
2. Press the "Zero offset" softkey.
3. Press the "G54...G599" softkey.

The "Zero Offset - G54..G599" window is opened.
Note
The labeling of the softkeys for the settable zero offsets varies, i.e. the settable zero offsets configured on the machine are displayed (examples: G54...G57, G54...G505, G54...G599).
Please observe the machine manufacturer's specifications.
4. You can edit the values directly in the table.

## Note

## Activate settable zero offsets

The settable zero offsets must first be selected in the program before they have an impact.

### 3.8.5 Displaying and editing details of the zero offsets

For each zero offset, you can display and edit all data for all axes. You can also delete zero offsets.

For every axis, values for the following data will be displayed:

- Coarse and fine offsets
- Rotation
- Scaling
- Mirroring


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Note

Settings for rotation, scaling and mirroring are specified here and can only be changed here.

## Tool details

You can display the following details for the tool and wear data for tools:

- TC
- Adapter dimension
- Length / length wear
- EC setup correction
- SC sum correction
- Total length
- Radius / radius wear

Act. uls. You can also change the display of the tool correction values between the MCS Machine Coordinate System and the Workpiece Coordinate System.

Please refer to the machine manufacturer's specifications.

## Procedure



Parameter

Work
offset

Active
654...6599

## Details

Clear
Offset

W0

.

W0 -
路

1. Select the "Parameter" operating area.
2. Press the "Zero offset" softkey.
3. Press the "Active", "Base" or "G54...G599" softkey. The corresponding window opens.
4. Place the cursor on the desired zero offset to view its details.
5. Press the "Details" softkey.

A window opens, depending on the selected zero offset, e.g. "Zero Offset - Details: G54 to G599".
6. You can edit the values directly in the table.

- OR -

Press the "Clear offset" softkey to reset all entered values.

Press the "ZO +" or "ZO -" softkey to select the next or previous offset, respectively, within the selected area ("Active", "Base", "G54 to G599") without first having to switch to the overview window.
If you have reached the end of the range (e.g. G599), you will switch automatically to the beginning of the range (e.g. G54).

These value changes are available in the part program immediately or after "Reset".

## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## << <br> Press the "Back" softkey to close the window. <br> Back

### 3.8.6 Deleting a zero offset

You have the option of deleting work offsets. This resets the entered values.

## Procedure



1. Select the "Parameter" operating area.

Ouerview
2. Press the "Work offset" softkey.
3. Press the "Overview", "Basis" or "G54...G599" softkey.

## 654...6599

Details
4. Press the "Details" softkey.
5. Position the cursor on the work offset you would like to delete.
6. Press the "Clear offset" softkey.

A confirmation prompt is displayed as to whether you really want to delete the work offset.
7. Press the "OK" softkey to confirm that you wish to delete the work offset.

### 3.8.7 Measuring the workpiece zero

## Procedure

1. Select the "Parameters" operating area and press the "Zero offset" softkey.
2. Press the "G54...G599" softkey and select the zero offset in which the zero point is to be saved.
3. Press the "Workpiece zero" softkey.

Meas. You change to the "Set Edge" window in the "JOG" mode.
4. Traverse the tool in the $Z$ direction and scratch it.
5. Enter the position setpoint of the workpiece edge $Z 0$ and press the "Set ZO" softkey.

### 3.9 Monitoring axis and spindle data

### 3.9.1 Specify working area limitations

The "Working area limitation" function can be used to limit the range within which a tool can traverse in all channel axes. These commands allow you to set up protection zones in the working area which are out of bounds for tool movements.

In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

## Requirements

You can only make changes in "AUTO" mode when in the RESET condition. These changes are then immediate.

You can make changes in "JOG" mode at any time. These changes, however, only become active at the start of a new motion.

## Procedure



Parameter


Working area limit.

1. Select the "Parameter" operating area.
2. Press the "Setting data" softkey.

The "Working Area Limitation" window appears.
3. Place the cursor in the required field and enter the new values via the numeric keyboard.
The upper or lower limit of the protection zone changes according to your inputs.
4. Click the "active" checkbox to activate the protection zone.

## Note

You will find all of the setting data in the "Start-up" operating area under "Machine data" via the menu forward key.

### 3.9.2 Editing spindle data

The speed limits set for the spindles that must not be under- or overshot are displayed in the "Spindles" window.

You can limit the spindle speeds in fields "Minimum" and "Maximum" within the limit values defined in the relevant machine data.

## Spindle speed limitation at constant cutting rate

In field "Spindle speed limitation at G96", the programmed spindle speed limitation at constant cutting speed is displayed together with the permanently active limitations.

This speed limitation, for example, prevents the spindle from accelerating to the max. spindle speed of the current gear stage (G96) when performing tapping operations or machining very small diameters.

## Note

The "Spindle data" softkey only appears if a spindle is configured.

## Procedure



Parameter
2. Press the "Setting data" and "Spindle data" softkeys. The "Spindles" window opens.
Spindle data

1. Select the "Parameter" operating area.
2. If you want to change the spindle speed, place the cursor on the "Maximum", "Minimum", or "Spindle speed limitation at G96" and enter a new value.

### 3.9.3 Spindle chuck data

You store the chuck dimensions of the spindles at your machine in the "Spindle Chuck Data" window.

## Manually measuring a tool

If you want to use the chuck of the main or counter-spindle as a reference point during manual measuring, specify the chuck dimension ZC.

## Main spindle



Dimensions, main spindle jaw type 1


Dimensions, main spindle jaw type 2

## Counter-spindle

You can measure either the forward edge or stop edge of the counter-spindle. The forward edge or stop edge automatically serves as the valid reference point when traversing the counter-spindle. This is especially important when gripping the workpiece using the counterspindle.


Dimensions, counter-spindle jaw type 1


Dimensions, counter-spindle jaw type 2

## Tailstock



Dimensioning the main spindle tailstock

## Procedure



1. Select the "Parameter" operating area.

## Parameter


2. Press the "Setting data" and "Spindle chuck data" softkeys.

The "Spindle Chuck Data" window opens.

```
Spindle chuck data
```



Dimensioning the counter-spindle tailstock

3. Enter the desired parameter.

The settings become active immediately.

## See also

Machining with movable counterspindle (Page 601)

| Parameter | Description | Unit |
| :--- | :--- | :--- |
| Main spindle | Dimensions of the forward edge or stop edge |  |
| • Jaw type 1 <br> • Jaw type 2 |  |  |
| ZC1 | Main spindle chuck dimensions (inc) | mm |
| ZS1 | Main spindle stop dimensions (inc) | mm |
| ZE1 | Jaw dimension, main spindle (inc) - only for "Jaw type 2" | mm |
| XR | Tailstock diameter - only for tailstock that has been set-up | mm |
| ZR | Taiistock length - only for tailstock that has been set-up |  |

3.10 Displaying setting data lists

| Parameter | Description | Unit |
| :---: | :---: | :---: |
| Counter-spindle |  |  |
|  | Dimensions of the forward edge or stop edge <br> - Jaw type 1 <br> - Jaw type 2 |  |
| ZC3 | Chuck dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up | mm |
| ZS3 | Stop dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up | mm |
| ZE3 | Jaw dimension, counter-spindle (inc) - only for a counter-spindle that has been set-up and "Jaw type 2" | mm |
| XR | Tailstock diameter - only for tailstock that has been set-up | mm |
| ZR | Tailstock length - only for tailstock that has been set-up | mm |

### 3.10 Displaying setting data lists

You can display lists with configured setting data.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

## Procedure



Parameter

Data lists

Select data list

1. Select the "Parameter" operating area.
2. Press the "Setting data" and "Data lists" softkeys. The "Setting Data Lists" window opens.
3. Press the "Select data list" softkey and in the "View" list, select the required list with setting data.

### 3.11 Handwheel assignment

You can traverse the axes in the machine coordinate system (Machine) or in the workpiece coordinate system (Work) via the handwheel.


## Software option

You require the "Extended operator functions" option for the handwheel offset (only for 828D).

All axes are provided in the following order for handwheel assignment:

- Geometry axes

When traversing, the geometry axes taken into account the actual machine status (e.g. rotations, transformations). All channel machine axes, which are currently assigned to the geometry axis, are in this case simultaneously traversed.

- Channel machine axes

Channel machine axes are assigned to the particular channel. They can only be individually traversed, i.e. the actual machine state has no influence.

The also applies to channel machine axes, that are declared as geometry axes.

## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Procedure

## M

## Machine




Handwheel
3. Position the cursor in the field next to the handwheel with which you wish to assign the axis (e.g. No. 1).

X

Press the <JOG>, <AUTO> or <MDI> key.
2. Press the menu forward key and the "Handwheel" softkey. The "Handwheel" window appears.
A field for axis assignment will be offered for every connected handwheel.

1. Select the "Machine" operating area.

$$
020
$$

4. Press the corresponding softkey to select the desired axis (e.g. "X").

- OR

Open the "Axis" selection box using the <INSERT> key, navigate to the desired axis, and press the <INPUT> key.
Selecting an axis also activates the handwheel (e.g., " X " is assigned to handwheel no. 1 and is activated immediately).
5. Press the "Handwheel" softkey again.

- OR -

Press the "Back" softkey.
The "Handwheel" window closes.

1. Position the cursor on the handwheel whose assignment you wish to cancel (e.g. No. 1).
2. Press the softkey for the assigned axis again (e.g. "X").

- OR -

Open the "Axis" selection box using the <INSERT> key, navigate to the empty field, and press the <INPUT> key.
Clearing an axis selection also clears the handwheel selection (e.g., "X" is cleared for handwheel no. 1 and is no longer active).

### 3.12 MDA

In "MDI" mode (Manual Data Input mode), you can enter G-code commands or standard cycles block-by-block and immediately execute them for setting up the machine.

You have the option of loading an MDI program or a standard program with the standard cycles directly into the MDI buffer from the program manager; you can subsequently then edit it.
You can save programs, generated or modified in the MDI working window, in the program manager, e.g. in a directory specifically created for the purpose.

Software option
You require the "Extended operator functions" option to load and save MDI programs (for 828D).

### 3.12.1 Loading an MDA program from the Program Manager

## Procedure



Machine

## Search

1. Select the "Machine" operating area.
2. Press the <MDI> key.

The MDI editor opens.
3. Press the "Load MDI" softkey.

A changeover is made into the Program Manager.
The "Load in MDI" window opens. It shows you a view of the program manager.
4. Position the cursor to the corresponding storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific file.
Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
5. Select the program that you would like to edit or execute in the MDI window.
6. Press the "OK" softkey.

The window closes and the program is ready for operation.

### 3.12.2 Saving an MDA program

## Procedure

' Machine

## 0

MDA

1. Select the "Machine" operating area.
2. Press the <MDI> key.

The MDI editor opens.
3. Create the MDI program by entering the G-code commands using the operator's keyboard.
4. Press the "Store MDI" softkey.

The "Save from MDI: Select storage location" window opens. It shows you a view of the program manager.
5. Select the drive to which you want to save the MDI program you created, and place the cursor on the directory in which the program is to be stored.

- OR -

Position the cursor to the required storage location, press the "Search" softkey and enter the required search term in the search dialog if you wish to search for a specific directory or subdirectory.
Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
6. Press the "OK" softkey.

When you place the cursor on a folder, a window opens which prompts you to assign a name.

- OR -

When you place the cursor on a program, you are asked whether the file should be overwritten.
7. Enter the name for the rendered program and press the "OK" softkey. The program will be saved under the specified name in the selected directory.

### 3.12.3 Editing/executing a MDI program

## Procedure

' ${ }^{\prime}$ Machine

1. Select the "Machine" operating area.
2. Press the <MDI> key.

The MDI editor opens.
3. Enter the desired G-code commands using the operator's keyboard.

- OR -

Enter a standard cycle, e.g. CYCLE62 ().

## Editing G-code commands/program blocks

4. Edit G-code commands directly in the "MDI" window.

- OR -


Select the required program block (e.g. CYCLE62) and press the <cursor right> key, enter the required value and press "OK".


When editing a cycle, either the help screen or the graphic view can be displayed.
5. Press the <CYCLE START> key.

The control executes the input blocks.
When executing G-code commands and standard cycles, you have the option of controlling the sequence as follows:

- Executing the program block-by-block
- Testing the program

Settings under program control

- Setting the test-run feedrate

Settings under program control

## See also

Program control (Page 139)

### 3.12.4 Deleting an MDA program

## Precondition

The MDA editor contains a program that you created in the MDI window or loaded from the program manager.

## Procedure

Press the "Delete blocks" softkey.

The program blocks displayed in the program window are deleted.

## Working in manual mode

## 4.1 <br> General

Always use "JOG" mode when you want to set up the machine for the execution of a program or to carry out simple traversing movements on the machine:

- Synchronize the measuring system of the controller with the machine (reference point approach)
- Set up the machine, i.e. activate manually-controlled motions on the machine using the keys and handwheels provided on the machine control panel.
- You can activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel while a part program is interrupted.


### 4.2 Selecting a tool and spindle

### 4.2.1 T,S,M window

For the preparatory actions in manual mode, tool selection and spindle control are both performed centrally in a screen form.
In addition to the main spindle (S1), there is another tool spindle (S2) for powered tools.
Your turning machine can also be equipped with a counter-spindle (S3).
In manual mode, you can select a tool on the basis of either its name or its revolver location number. If you enter a number, a search is performed for a name first, followed by a location number. This means that if you enter " 5 ", for example, and no tool with the name " 5 " exists, the tool is selected from location number " 5 ".

## Note

Using the revolver location number, therefore, you can swing around an empty space into the machining position and then comfortably install a new tool.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

4．2 Selecting a tool and spindle

| Parameter | Meaning |  | Unit |
| :---: | :---: | :---: | :---: |
| T | Input of the tool（name or location number） <br> You can select a tool from the tool list using the＂Select tool＂softkey． |  |  |
| DU | Cutting edge number of the tool（1－9） |  |  |
| ST U | Sister tool（1－99 for replacement tool strategy） |  |  |
| Spindle U | Spindle selection，identification with spindle number |  |  |
| Spindle M function U | ¢ $\times$ | Spindle off：Spindle is stopped |  |
|  | 9 | CCW rotation：Spindle rotates counterclockwise |  |
|  | 2 | CW rotation：Spindle rotates clockwise |  |
|  | ぞ「 | Spindle positioning：Spindle is moved to the desired position． |  |
| Other M functions | Input of machine functions <br> Refer to the machine manufacturer＇s table for the correlation between the meaning and number of the function． |  |  |
| Work offset G U | Selection of the work offset（basic reference，G54－57） <br> You can select work offsets from the tool list of settable work offsets via the＂Work offset＂softkey． |  |  |
| Dimension unit $\mathbf{U}$ | Selecting the measurement unit <br> The setting made here has an effect on the programming． |  | inch mm |
| Machining plane $\cup$ | Selection of the machining plane（G17（XY），G18（ZX），G19（YZ）） |  |  |
| Gear stage $\cup$ | Specification of the gear stage（auto，I－V） |  |  |
| Stop position | Entering the spindle position |  | Degrees |

## Note

## Spindle positioning

You can use this function to position the spindle at a specific angle，e．g．during a tool change．
－A stationary spindle is positioned via the shortest possible route．
－A rotating spindle is positioned as it continues to turn in the same direction．

### 4.2.2 Selecting a tool

## Procedure

## ' ${ }^{\prime}$

Machine



1. Select the "JOG" operating mode.
2. Press the "T, S, M" softkey.
3. Select as to whether you wish that the tool is identified using a name or the location number.
4. Enter the name or the number of the tool T in the entry field.

- OR -

Press the "Select tool" softkey.
The tool selection window is opened.
Place the cursor on the desired tool and press the "OK" softkey.
The tool is transferred to the "T, S, M... window" and displayed in the field of tool parameter "T".
5. Select the tool cutting edge $D$ or enter the number directly in the field.
6. Select the sister tool ST or enter the number directly in field "ST".
7. Press the <CYCLE START> key.

The tool is automatically swung into the machining position and the name of the tool displayed in the tool status bar.

### 4.2.3 Starting and stopping the spindle manually

## Procedure



1. Select the "T,S,M" softkey in the "JOG" mode.
2. Select the desired spindle (e.g. S1) and enter the desired spindle speed or cutting speed in the right-hand entry field.
3. If the machine has a gearbox for the spindle, set the gearing step.


4 Select a spindle direction of rotation (clockwise or counterclockwise) in the "Spindle M function" field.
5. Press the <CYCLE START> key. The spindle rotates.
6. Select the "Stop" setting in the "Spindle M function" field.

Press the <CYCLE START> key.
The spindle stops.

## Note

## Changing the spindle speed

If you enter the speed in the "Spindle" field while the spindle is rotating, the new speed is applied.

### 4.2.4 Positioning the spindle

## Procedure



1. Select the "T,S,M" softkey in the "JOG" mode.
$\mathrm{T}, \mathrm{S}, \mathrm{M}$

2. Select the "Stop Pos." setting in the "Spindle M function" field. The "Stop Pos." entry field appears.
3. Enter the desired spindle stop position. The spindle position is specified in degrees.
4. Press the <CYCLE START> key.

The spindle is moved to the desired position.

## Note

You can use this function to position the spindle at a specific angle, e.g. during a tool change.

- A stationary spindle is positioned via the shortest possible route.
- A rotating spindle is positioned as it continues to turn in the same direction.


### 4.3 Traversing axes

You can traverse the axes in manual mode via the Increment or Axis keys or handwheels.
During a traverse initiated from the keyboard, the selected axis moves at the programmed setup feedrate. During an incremental traverse, the selected axis traverses a specified increment.

## Set the default feedrate

Specify the feedrate to be used for axis traversal in the set-up, in the "Settings for Manual Operation" window.

### 4.3.1 Traverse axes by a defined increment

You can traverse the axes in manual mode via the Increment and Axis keys or handwheels.

## Procedure

$\mathrm{\sim}_{\mathrm{Jog}}^{\mathrm{M}}$
2. Press the <JOG> key.
3. Press keys 1,10 , etc. up to 10000 in order to move the axis in a defined increment.
The numbers on the keys indicate the traverse path in micrometers or microinches.
Example: Press the "100" button for a desired increment of $100 \mu \mathrm{~m}$ (= 0.1 mm ).
4. Select the axis to be traversed.

5. Press the <+> or <-> key.

Each time you press the key the selected axis is traversed by the defined increment.
Feedrate and rapid traverse override switches can be operative.

## Note

When the controller is switched on, the axes can be traversed right up to the limits of the machine as the reference points have not yet been approached and the axes referenced. Emergency limit switches might be triggered as a result.

The software limit switches and the working area limitation are not yet operative!
The feed enable signal must be set.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

### 4.3.2 Traversing axes by a variable increment

## Procedure

## M

Machine


Settings
$\rightarrow 1$
[VAR]

$-$

1. Select the "Machine" operating area.
2. Press the "Settings" softkey.

The "Settings for Manual Operation" window is opened.
3. Enter the desired value for the "Variable increment" parameter.

Example: Enter 500 for a desired increment of $500 \mu \mathrm{~m}(0.5 \mathrm{~mm})$.
4. Press the <Inc VAR> key.
5. Select the axis to be traversed.
6. Press the <+> or <-> key.

Each time you press the key the selected axis is traversed by the set increment

Feedrate and rapid traverse override switches can be operative.

### 4.4 Positioning axes

In order to implement simple machining sequences, you can traverse the axes to certain positions in manual mode.
The feedrate / rapid traverse override is active during traversing.

## Procedure

1. If required, select a tool.

| $\mathrm{M}_{\mathrm{JOG}}^{\mathrm{M}}$ |
| :---: |

2. Select the "JOG" operating mode.
3. Press the "Positions" softkey.
4. Enter the target position or target angle for the axis or axes to be traversed.
5. Specify the desired value for the feedrate F.

- OR -

Press the "Rapid traverse" softkey.
The rapid traverse is displayed in field "F".
6. Press the <CYCLE START> key.

The axis is traversed to the specified target position.
If target positions were specified for several axes, the axes are traversed simultaneously.

### 4.5 Manual retraction

After an interruption of a tapping operation (G33/G331/G332) or a general drilling operation (tools 200 to 299) due to power loss or a RESET at the machine control panel, you have the possibility to retract the tool in the JOG mode in the tool direction without damaging the tool or the workpiece.

The retraction function is especially useful when the coordinate system is swiveled, i.e. the infeed axis is not in the vertical position.

## Note

## Tapping

In the case of tapping, the form fit between the tap and the workpiece is taken into account and the spindle moved according to the thread.
The $Z$ axis as well as the spindle can be used for retraction at the thread.

The "Retraction" function must be set up by the machine manufacturer.


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Procedure

1. <RESET> interrupts the power supply to the machine and any running part program.
2. After a power supply interruption, switch on the controller.


Retract
5. Press the "Retract" softkey.

The "Retract Tool" window opens.
The softkey is available only when an active tool and retraction data are present.
6. Select the "WCS" coordinate system on the machine control panel.
3. Select the JOG operating mode.
4. Press the Menu forward key.
7. Use the traversing keys (e.g. $Z+$ ) to traverse the tool from the workpiece according to the retraction axis displayed in the "Retract Tool" window.
8. To exit the window, press the "Retract" softkey again when the tool is at the desired position.

### 4.6 Simple stock removal of workpiece

Some blanks have a smooth or even surface. For example, you can use the stock removal cycle to turn the face surface of the workpiece before machining actually takes place.

If you want to bore out a collet using the stock removal cycle, you can program an undercut (XF2) in the corner.

## 1 CAUTION

## Risk of collision

The tool moves along a direct path to the starting point of the stock removal. First move the tool to a safe position in order to avoid collisions during the approach.

## Retraction plane / safety clearance

The retraction plane and safety clearance are set via the machine data \$SCS_MAJOG_SAFETY_CLEARANCE or \$SCS_MAJOG_RELEASE_PLANE.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

## Direction of spindle rotation

If the "ShopMill/ShopTurn" option is activated, the direction of spindle rotation is taken from the tool parameters entered in the tool list.
If the "ShopMill/ShopTurn" option is not set, select the direction of spindle rotation in the input screen.

## Note

You cannot use the "Repos" function during simple stock removal.

## Requirement

To carry out simple stock removal of a workpiece in manual mode, a measured tool must be in the machining position.

## Procedure

## ＇${ }^{M}$

Machine


Stock rem．

3．Press the＂Stock removal＂softkey．

4．Enter desired values for the parameters．
5．Press the＂OK＂softkey．
The parameter screen is closed．
6．Press the＜CYCLE START＞key．
The＂Stock removal＂cycle is started．
You can return to the parameter screen form at any time to check and correct the inputs．

Table 4－1

| Parameter | Description | Unit |
| :---: | :---: | :---: |
| T | Tool name |  |
| D | Cutting edge number |  |
| F | Feedrate | mm／rev |
| $\begin{aligned} & \mathrm{S} / \mathrm{V} \\ & \mathrm{U} \end{aligned}$ | Spindle speed or constant cutting rate | rpm $\mathrm{m} / \mathrm{min}$ |
| Spindle M function | Direction of spindle rotation（only when ShopTurn is not active） <br> － 2 <br> －$\Omega$ |  |
| Machining <br> U | －$\nabla$（roughing） <br> －VVV（finishing） |  |
| Position <br> 0 | Machining position <br> ＂迬 <br> 恩 <br> 宸 <br> 原 |  |

4.7 Thread synchronizing

| Parameter | Description | Unit |
| :--- | :--- | :--- |
| Machining <br> direction <br> U | $\bullet$ Face |  |
| • Longitudinal |  |  |
| X0 | Reference point $\varnothing$ (abs) |  |
| Z0 | Reference point (abs) | mm |
| X1 U | End point X $\varnothing$ (abs) or end point X in relation to X0 (inc) | mm |
| Z1 U | End point Z (abs) or end point Z in relation to X0 (inc) | mm |
| FS1...FS3 or R1...R3 <br> U | Chamfer width (FS1...FS3) or rounding radius (R1...R3) | mm |
| XF2 U | Undercut (alternative to FS2 or R2) | mm |
| D | Infeed depth (inc) - (for roughing only) | mm |
| UX | Final machining allowance in X direction (inc) - (for roughing only) | mm |
| UZ | Final machining allowance in Z direction (inc) - (for roughing only) | mm |

## See also

Tool, offset value, feedrate and spindle speed (T, D, F, S, V) (Page 254)

### 4.7 Thread synchronizing

If you wish to re-machine a thread, it may be necessary to synchronize the spindle to the existing thread turn. This is necessary as by reclamping the blank, an angular offset can occur in the thread.

## Constraint

Thread synchronizing is not possible if a toolcarrier is used (B axis).

## Note

## Activating/deactivating thread synchronization

If a thread synchronization is active, then this is active for all of the following "Thread turning" machining steps.
Thread synchronization remains effective without deactivation even after the machining has been shutdown.

## Requirement

The spindle is stationary.
One threading tool is active.

## Procedure



Machine


Synch. thread

Teach main spin.

Teach countersp.

0

1. Select the "JOG" operating mode.
2. Press the menu forward key and the "Thread synchr." softkey.
3. Thread the thread cutting tool into the thread turn as shown in the help screen.
4. Press the "Teach-in main spindle" softkey if you are working at the main spindle.

- OR -

Press the "Teach-in counterspindle" softkey if you are working at the counterspindle.

Note:
The thread synchronization is activated by teaching in a spindle. In this case, the synchronizing positions of axes $X$ and $Z$ and the synchronizing angle of spindle (Sn) are saved in the Machine and displayed in the screen form.
The selection boxes for main spindle and counterspindle indicate whether thread synchronization is active for the particular spindle (yes = active $/$ no $=$ not active).
5. Now carry out the "thread turning" machining step.
6. For the main spindle or counterspindle, select the "no" entry to deactivate thread synchronization.

### 4.8 Default settings for manual mode

Specify the configurations for manual mode in the "Settings for manual operation" window.

## Presettings

| Settings | Description |
| :--- | :--- |
| Type of feedrate | Here, you select the type of feedrate. |
|  | $\bullet \quad$ G94: Axis feedrate/linear feedrate |
|  | $\bullet \quad$ G95: Rev. feedrate |
| Default feedrate G94 | Enter the desired feedrate in $\mathrm{mm} / \mathrm{min}$. |
| Default feedrate G95 | Enter the desired feedrate in $\mathrm{mm} / \mathrm{r}$. |
| Variable increment | Enter the desired increment for axis traversal by variable increments. |
| Spindle speed | Enter the desired spindle speed in rpm. |

## Proceed as follows

M

1. Select the "Machine" operating area.

## Machine


2. Press the <JOG> key.
3. Press the menu forward key and the "Settings" softkey. The "Settings for manual operation" window is opened.

## See also

Switching the unit of measurement (Page 75)

## Machining the workpiece

### 5.1 Starting and stopping machining

During execution of a program, the workpiece is machined in accordance with the programming on the machine. After the program is started in automatic mode, workpiece machining is performed automatically.

## Preconditions

The following requirements must be met before executing a program:

- The measuring system of the controller is referenced with the machine.
- The necessary tool offsets and work offsets have been entered.
- The necessary safety interlocks implemented by the machine manufacturer are activated.


## General sequence



1. Use the Program manager to select the desired program.
2. Select under "NC", "Local. Drive", "USB" or set-up network drives the desired program.
3. Press the "Select" softkey.

The program is selected for execution and automatically switched to the "Machine" operating area.
4. Press the <CYCLE START> key.

The program is started and executed.

## Note

Starting the program in any operating area
If the controller is in "AUTO" mode, you can also start the selected program when you are in any operating area.

## Stopping machining



Press the <CYCLE STOP> key
Machining stops immediately. Individual program blocks are not executed to the end. On the next start, machining is resumed from the point where it left off.

## Canceling machining



Press the <RESET> key.
Execution of the program is interrupted. On the next start, machining will start from the beginning.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

### 5.2 Selecting a program

## Procedure



- OR -

Press the <Cursor right> key.

The directory contents are displayed.
5. Place the cursor on the desired program.

Execute

1. Select the "Program manager" operating area.

The directory overview is opened.
2. Select the location where the program is archived (e.g. "NC") to select.
4. Press the <INPUT> key
6. Press the "Select" softkey.
3. Place the cursor on the directory containing the program that you want

The program is selected.
When the program has been successfully selected, an automatic changeover to the "Machine" operating area occurs.

### 5.3 Executing a trail program run

When testing a program, the system can interrupt the machining of the workpiece after each program block, which triggers a movement or auxiliary function on the machine. In this way, you can control the machining result block-by-block during the initial execution of a program on the machine.

## Note

## Settings for the automatic mode

Rapid traverse reduction and dry run feed rate are available to run-in or to test a program.

## Move by single block

In "Program control" you may select from among several types of block processing:

| SB mode | Scope |
| :--- | :--- |
| SB1 Single block, coarse | The machining stops after every machine block (except for cycles) |
| SB2 Data block | The machining stops after every block, i.e. also for data blocks (except <br> for cycles) |
| SB3 Single block, fine | The machining stops after every machine block (also in cycles) |

## Precondition

A program must be selected for execution in "AUTO" or "MDA" mode.

## Procedure




1. Press the "Prog. ctrl." softkey and select the desired variant in the "SBL" field.
2. Press the <SINGLE BLOCK> key.
3. Press the <CYCLE START> key.

Depending on the execution variant, the first block will be executed. Then the machining stops.
In the channel status line, the text "Stop: Block in single block ended" appears.
4. Press the <CYCLE START> key.

Depending on the mode, the program will continue executing until the next stop.
5. Press the <SINGLE BLOCK> key again, if the machining is not supposed to run block-by-block.
The key is deselected again.
If you now press the <CYCLE START> key again, the program is executed to the end without interruption.

### 5.4 Displaying the current program block

### 5.4.1 Current block display

The window of the current block display shows you the program blocks currently being executed.

## Display of current program

The following information is displayed in the running program:

- The workpiece name or program name is entered in the title row.
- The program block which is just being processed appears colored.


## Editing a program directly

In the Reset state, you can edit the current program directly.

1. Press the <INSERT> key.
2. Place the cursor at the relevant position and edit the program block. Direct editing is only possible for G code blocks in the NC memory, not for external execution.
3. Press the <INSERT> key to exit the program and the edit mode again.

### 5.4.2 Displaying a basic block

If you want precise information about axis positions and important G functions during testing or program execution, you can call up the basic block display. This is how you can check, when using cycles, for example, whether the machine is actually traversing.
Positions programmed by means of variables or R parameters are resolved in the basic block display and replaced by the variable value.
You can use the basic block display both in test mode and when machining the workpiece on the machine. All G code commands that initiate a function on the machine are displayed in the "Basic Blocks" window for the currently active program block:

- Absolute axis positions
- G functions for the first G group
- Other modal G functions
- Other programmed addresses
- M functions


## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Procedure

1. A program is selected for execution and has been opened in the "Machine" operating area.

Basic
blocks


Basic blocks
2. Press the "Basic blocks" softkey. The "Basic Blocks" window opens.
3. Press the <SINGLE BLOCK> key if you wish to execute the program block-by-block.
4. Press the <CYCLE START> key to start the program execution. The axis positions to be approached, modal G functions, etc., are displayed in the "Basic Blocks" window for the currently active program block.
5. Press the "Basic blocks" softkey once again to hide the window again.

### 5.4.3 Display program level

You can display the current program level during the execution of a large program with several subprograms.

## Several program run throughs

If you have programmed several program run throughs, i.e. subprograms are run through several times one after the other by specifying the additional parameter $P$, then during processing, the program runs still to be executed are displayed in the "Program Levels" window.

## Program example

N10 subprogram P25
If, in at least one program level, a program is run through several times, a horizontal scroll bar is displayed that allows the run through counter $P$ to be viewed in the righthand window section. The scroll bar disappears if multiple run-through is no longer applicable.

## Display of program level

The following information will be displayed:

- Level number
- Program name
- Block number, or line number
- Remain program run throughs (only for several program run throughs)


## Precondition

A program must be selected for execution in "AUTO" mode.

## Procedure

| Program <br> levels | Press the "Program levels" softkey. |
| :--- | :--- |
|  | The "Program levels" window appears. |

### 5.5 Correcting a program

As soon as a syntax error in the part program is detected by the controller, program execution is interrupted and the syntax error is displayed in the alarm line.

## Correction possibilities

Depending on the state of the control system, you can make the following corrections using the Program editing function.

- Stop mode

Only program lines that have not yet been executed can be edited.

- Reset mode

All program lines can be edited.

## Note

The "program correction" function is also available for execute from external; however, when making program changes, the NC channel must be brought into the reset state.

## Requirement

A program must be selected for execution in "AUTO" mode.

## Procedure

1. The program to be corrected is in the Stop or Reset mode.
2. Press the "Prog. corr." softkey

The program is opened in the editor.
The program preprocessing and the current block are displayed. The current block is also updated in the running program, but not the displayed program section, i.e. the current block moves out of the displayed program section.
If a subprogram is executed, it is not opened automatically.
3. Make the necessary corrections.

4. Press the "NC Execute" softkey.

The system switches back to the "Machine" operating area and selects "AUTO" mode.
5. Press the <CYCLE START> key to resume program execution.

## Note

Exit the editor using the "Close" softkey to return to the "Program manager" operating area.

### 5.6 Repositioning axes

After a program interruption in automatic mode (e.g. after a tool breaks) you can move the tool away from the contour in manual mode.

The coordinates of the interrupt position will be saved. The distances traversed in manual mode are displayed in the actual value window. This path difference is called "Repos-offset".

## Resuming program execution

Using the "Repos" function, you can return the tool to the contour in order to continue executing the program.

You cannot traverse the interrupt position, because it is blocked by the control system.
The feedrate/rapid traverse override is in effect.

## NOTICE

## Risk of collision

When repositioning, the axes move with the programmed feedrate and linear interpolation,
i.e. in a straight line from the current position to the interrupt point. Therefore, you must first move the axes to a safe position in order to avoid collisions.
If you do not use the "Repos" function and subsequently move the axes in manual mode after a program interrupt, the control automatically moves the axes during the switch to automatic mode and the subsequent start of the machining process in a straight line back to the point of interruption.

## Precondition

The following prerequisites must be met when repositioning the axes:

- The program execution was interrupted using <CYCLE STOP>.
- The axes were moved from the interrupt point to another position in manual mode.


## Procedure



1. Press the <REPOS> key.
2. Select the axes to be traversed one after the other.

3. Press the <+> or <-> key for the relevant direction. The axes are moved to the interrupt position.

### 5.7 Starting machining at a specific point

### 5.7.1 Use block search

If you would only like to perform a certain section of a program on the machine, then you need not start the program from the beginning. You can also start the program from a specified program block.

## Applications

- Stopping or interrupting program execution
- Specify a target position, e.g. during remachining


## Determining a search target

- User-friendly search target definition (search positions)
- Direct specification of the search target by positioning the cursor in the selected program (main program)
- Search target via text search
- The search target is the interruption point (main program and subprogram)

The function is only available if there is an interruption point. After a program interruption (CYCLE STOP, RESET or power off), the controller saves the coordinates of the interruption point.

- The search target is the higher program level of the interruption point (main program and subprogram)

The level can only be changed if it was previously possible to select an interruption point in a subprogram. It is then possible to change the program level up to the main program level and back to the level of the interruption point.

- Search pointer
- Direct entry of the program path


## Note

You can search for a specific point in subprograms with the search pointer if there is no interruption point.

Software option
You require the "Extended operator functions" option for the "Search pointer" function (only for 828D).

### 5.7 Starting machining at a specific point

## Cascaded search

You can start another search from the "Search target found" state. The cascading can be continued any number of times after every search target found.

## Note

Another cascaded block search can be started from the stopped program execution only if the search target has been found.

## References

Function Manual Basic Functions; Block Search

## Preconditions

- You have selected the desired program.
- The controller is in the reset state.
- The desired search mode is selected.


## NOTICE

## Risk of collision

Pay attention to a collision-free start position and appropriate active tools and other technological values.

If necessary, manually approach a collision-free start position. Select the target block considering the selected block search type.

## Toggling between search pointer and search positions

Search
pointer
<<
Back

Press the "Search pointer" softkey again to exit the "Search Pointer" window and return to the "Program" window to define search positions.

- OR -

Press the "Back" softkey.
You have now exited the block search function.

## See also

Selecting a program (Page 124)

### 5.7.2 Continuing program from search target

To continue the program at the desired position, press the <CYCLE START> key twice.

- The first CYCLE START outputs the auxiliary functions collected during the search. The program is then in the Stop state.
- Before the second CYCLE START, you can use the "Overstore" function to create states that are required, but not yet available, for the further program execution.
By changing to the JOG REPOS mode, you can also manually traverse the tool from the current position to the setpoint position, if the setpoint position is not to be automatically approached after the program start


### 5.7.3 Simple search target definition

## Requirement

> The program is selected and the controller is in Reset mode.

## Procedure

Search for text

OK
Start
search

1. Press the "Block search" softkey.
2. Place the cursor on a particular program block.

- OR -

Press the "Find text" softkey, select the search direction, enter the search text and confirm with "OK".

The search starts. Your specified search mode will be taken into account.
The current block will be displayed in the "Program" window as soon as the target is found.
4. If the located target (for example, when searching via text) does not correspond to the program block, press the "Start search" softkey again until you find your target.
Press the <CYCLE START> key twice.
Processing is continued from the defined position.

### 5.7.4 Defining an interruption point as search target

## Requirement

A program was selected in "AUTO" mode and interrupted during execution through CYCLE STOP or RESET.


## Procedure

## Software option

You require the "Extended operator functions" option (only for 828D).

Block search

Interrupt
point

## Higher <br> leuel

Lower


Start search

1. Press the "Block search" softkey.
2. Press the "Interrupt point" softkey. The interruption point is loaded.
3. If the "Higher level" and "Lower level" softkeys are available, use these to change the program level.
4. Press the "Start search" softkey.

The search starts. Your specified search mode will be taken into account.
The search screen closes.
The current block will be displayed in the "Program" window as soon as the target is found.
5. Press the <CYCLE START> key twice.

The execution will continue from the interruption point.

### 5.7.5 Entering the search target via search pointer

Enter the program point which you would like to proceed to in the "Search Pointer" window.


## Software option

You require the "Extended operator functions" option for the "Search pointer" function (only for 828D).

## Requirement

The program is selected and the controller is in the reset state.

## Screen form

Each line represents one program level. The actual number of levels in the program depends on the nesting depth of the program.

Level 1 always corresponds to the main program and all other levels correspond to subprograms.

You must enter the target in the line of the window corresponding to the program level in which the target is located.

For example, if the target is located in the subprogram called directly from the main program, you must enter the target in program level 2.

The specified target must always be unambiguous. This means, for example, that if the subprogram is called in the main program in two different places, you must also specify a target in program level 1 (main program).

## Procedure

1. Press the "Block search" softkey.
2. Press the "Search pointer" softkey.
3. Enter the full path of the program as well as the subprograms, if required, in the input fields.
4. Press the "Start search" softkey.

The search starts. Your specified search mode will be taken into account.

The Search window closes. The current block will be displayed in the "Program" window as soon as the target is found.
5. Press the <CYCLE START> key twice.

Processing is continued from the defined location.

## Note

Interruption point
You can load the interruption point in search pointer mode.

### 5.7.6 Parameters for block search in the search pointer

| Parameter | Meaning |
| :--- | :--- |
| Number of program level |  |
| Program: | The name of the main program is automatically entered |
| Ext: | File extension |
| P: | Pass counter <br> If a program section is performed several times, you can enter the number of <br> the pass here at which processing is to be continued |
| Line: | Is automatically filled for an interruption point |
| Type | " " search target is ignored on this level <br> N no. Block number <br> Label Jump label <br> Text string <br> Subprg. Subprogram call <br> Line Line number |
| Search target | Point in the program at which machining is to start |

### 5.7.7 Block search mode

Set the desired search variant in the "Search Mode" window.
The set mode is retained when the control is shut down. When you activate the "Search" function after restarting the control, the current search mode is displayed in the title row.

## Search variants

$\left.\begin{array}{|l|l|}\hline \text { Block search mode } & \text { Meaning } \\ \hline \text { With calculation } & \begin{array}{l}\text { In order to be able to approach a target position in any circumstance (e.g. } \\ \text { tool change position). } \\ \text { The end position of the target block or the next programmed position is } \\ \text { approached using the type of interpolation valid in the target block. Only the } \\ \text { axes programmed in the target block are moved. } \\ \text { Note: } \\ \text { If machine data 11450.1=1 is set, the rotary axes of the active swivel data } \\ \text { record are pre-positioned after the block search. }\end{array} \\ \hline \text { With calculation } & \begin{array}{l}\text { It is used to be able to approach the contour in any circumstance. } \\ \text { The end position of the block prior to the target block is found with <CYCLE } \\ \text { START>. The program runs in the same way as in normal program pro- } \\ \text { cessing. } \\ \text { Note: }\end{array} \\ \hline \text { With calculation } & \begin{array}{l}\text { In ShopTurn program, the search is only performed on G code-blocks. } \\ \text { programs: EXTCALL programs are not taken into account. }\end{array} \\ \hline \text {-skip extcall } & \begin{array}{l}\text { Notice: Important information, e.g. modal functions, which are located in the } \\ \text { EXTCALL program, are not taken into account. In this case, after the } \\ \text { search target has been found, the program is not able to be executed. Such } \\ \text { information should be programmed in the main program. }\end{array} \\ \hline \text { Without calculation } & \begin{array}{l}\text { For a quick search in the main program. } \\ \text { Calculations will not be performed during the block search, i.e. the calcula- } \\ \text { tion is skipped up to the target block. } \\ \text { All settings required for execution have to be programmed from the target } \\ \text { block (e.g. feedrate, spindle speed, etc.). }\end{array} \\ \hline \text { With program test } & \begin{array}{l}\text { Multi-channel block search with calculation (SERUPRO). } \\ \text { All blocks are calculated during the block search. Absolutely no axis motion } \\ \text { is executed, however, all auxiliary functions are output. }\end{array} \\ \text { The NC starts the selected program in the program test mode. If the NC } \\ \text { reaches the specified target block in the actual channel, it stops at the be- } \\ \text { ginning of the target block and deselects program test mode again. After } \\ \text { continuing the program with NC start (after REPOS motion) the auxiliary } \\ \text { functions of the target block are output. } \\ \text { For single-channel systems, the coordination is supported with events run- } \\ \text { ning in parallel, e.g. synchronized actions. } \\ \text { Note } \\ \text { The search speed depends on MD settings. }\end{array}\right\}$

## Note

## Search mode for ShopTurn programs

- The search variant for the ShopTurn machining step programs can be specified via MD 51024. This applies only to the ShopTurn single-channel view.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

## References

For additional information, please refer to the following documentation:
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

## Procedure

'M
Machine

```
->
AUTO
```

Block
search
Ble sear. mode
2. Press the <AUTO> key.
3. Press the "Block search" and "Block search mode" softkeys. The "Search Mode" window opens.

### 5.8 Controlling the program run

### 5.8.1 Program control

You can change the program sequence in the "AUTO" and "MDI" modes.

| Abbreviation/program control | Mode of operation |
| :---: | :---: |
| PRT <br> No axis motion | The program is started and executed with auxiliary function outputs and dwell times. In this mode, the axes are not traversed. <br> The programmed axis positions and the auxiliary function outputs are controlled this way. <br> Note: Program processing without axis motion can also be activated with the function "Dry run feedrate". |
| DRY <br> Dry run feedrate | The traversing velocities programmed in conjunction with G1, G2, G3, CIP and CT are replaced by a defined dry run feedrate. The dry run feedrate also applies instead of the programmed revolutional feedrate. <br> Caution: Workpieces must not be machined when "Dry run feedrate" is active because the altered feedrates might cause the permissible tool cutting rates to be exceeded and the workpiece or machine tool could be damaged. |
| RG0 <br> Reduced rapid traverse | In the rapid traverse mode, the traversing speed of the axes is reduced to the percentage value entered in RGO. <br> Note: You define the reduced rapid traverse in the settings for automatic operation. |
| M01 Programmed stop 1 | The processing of the program stops at every block in which supplementary function M01 is programmed. In this way you can check the already obtained result during the processing of a workpiece. <br> Note: In order to continue executing the program, press the <CYCLE START> key again. |
| Programmed stop 2 <br> (e.g. M101) | The processing of the program stops at every block in which the "Cycle end" is programmed (e.g. with M101). <br> Note: In order to continue executing the program, press the <CYCLE START> key again. Note: The display can be changed. Please refer to the machine manufacturer's specifications. |
| DRF <br> Handwheel offset | Enables an additional incremental work offset while processing in automatic mode with an electronic handwheel. <br> This function can be used to compensate for tool wear within a programmed block. <br> Note: You require the "Extended operator functions" option to use the handwheel offset (for 828D). |
| SB | Individual blocks are configured as follows. <br> Single block, coarse: The program stops only after blocks which perform a machine function. <br> Data block: The program stops after each block. <br> Single block, fine: The program also stops only after blocks which perform a machine function in cycles. <br> Select the desired setting using the <SELECT> key. |
| SKP | Skip blocks are skipped during machining. |
| GCC | When executing a jobshop program, it is converted into a G-code program. |
| MRD | In the program, the measurement results screen display is activated while machining. |

## Activating program control

You can control the program sequence however you wish by selecting and clearing the relevant checkboxes.

## Display / response of active program controls

If a program control is activated, the abbreviation of the corresponding function appears in the status display as response.

## Procedure

| $\bar{M}$ <br> Machine | 1. | Select the "Machine" operating area. |
| :---: | :---: | :---: |
| $\underset{\text { AUTO }}{\rightarrow}$ | 2. | Press the <AUTO> or <MDI> key. |
| $\stackrel{\text { 图 }}{\text { MDA }}$ |  |  |
|  | 3. | Press the "Prog. ctrl." softkey. <br> The "Program Control" window opens |

### 5.8.2 Skip blocks

It is possible to skip program blocks, which are not to be executed every time the program runs.

The skip blocks are identified by placing a "/" (forward slash) or "/x (x = number of skip level) character in front of the block number. Several consecutive blocks can also be skipped.

The statements in the skipped blocks are not executed, i.e. the program continues with the next block, which is not skipped.

The number of skip levels that can be used depends on a machine datum.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

## Software option

In order to have more than two skip levels, for 828D you require the "Extended operator functions" option.

## Skip levels，activate

Select the corresponding checkbox to activate the desired skip level．

## Note

The＂Program Control－Skip Blocks＂window is only available when more than one skip level is set up．

## Procedure



1．Select the＂Machine＂operating area．


2．Press the＜AUTO＞or＜MDA＞key．


| 此些自 Prog． cntrl． | 3 | Press the＂Prog．ctrl．＂and＂Skip blocks＂softkeys． <br> The＂Program Control＂window appears and show |
| :---: | :---: | :---: |

## Program control

## Skip blocks

| $\square$ Plane $/ \mathbf{1}$ | $\square$ Plane $/ \mathbf{4}$ |
| :--- | :--- |
| $\square$ Plane $/ \mathbf{1}$ | $\square$ Plane／5 |
| $\square$ Plane $/ \mathbf{2}$ | $\square$ Plane／6 |
| $\square$ Plane $/ \mathbf{3}$ |  |
|  |  |
|  |  |

## $5.9 \quad$ Overstore

With overstore, you have the option of executing technological parameters (for example, auxiliary functions, axis feed, spindle speed, programmable instructions, etc.) before the program is actually started. The program instructions act as if they are located in a normal part program. These program instructions are, however, only valid for one program run. The part program is not permanently changed. When next started, the program will be executed as originally programmed.

After a block search, the machine can be brought into another state with overstore (e.g. M function, tool, feed, speed, axis positions etc.), in which the normal part program can be successfully continued.

## Software option

You require the "Extended operator functions" option for the overstore function (for 828D).

## Precondition

The program to be corrected is in the Stop or Reset mode.

## Procedure


2. Press the "Overstore" softkey.

The "Overstore" window opens.
3. Enter the required data and NC block.
4. Press the <CYCLE START> key.

The blocks you have entered are stored. You can observe execution in the "Overstore" window.
After the entered blocks have been executed, you can append blocks again.
You cannot change the operating mode while you are in overstore mode.
5. Press the "Back" softkey.

The "Overstore" window closes.
6. Press the <CYCLE START> key again.

The program selected before overstoring continues to run.

## Note

## Block-by-block execution

The <SINGLE BLOCK> key is also active in the overstore mode. If several blocks are entered in the overstore buffer, then these are executed block-by-block after each NC start

## Deleting blocks

Delete
blocks

Press the "Delete blocks" softkey to delete program blocks you have entered.

### 5.10 Editing a program

With the editor, you are able to render, supplement, or change part programs.

## Note

Maximum block length
The maximum block length is 512 characters.

## Calling the editor

- The editor is started via the "Program correction" softkey in the "Machine" operating area. You can directly change the program by pressing the <INSERT> key.
- The editor is called via the "Open" softkey as well as with the <INPUT> or <Cursor right> key in the "Program manager" operating area.
- The editor opens in the "Program" operating area with the last executed part program, if this was not explicitly exited via the "Close" softkey.


## Note

- Please note that the changes to programs saved in the NC memory take immediate effect.
- If you are editing on a local drive or external drives, you can also exit the editor without saving, depending on the setting. Programs in the NC memory are always automatically saved.
- Exit the program correction mode using the "Close" softkey to return to the "Program manager" operating area.


## See also

## Editor settings (Page 153)

Correcting a program (Page 129)
Opening and closing the program (Page 738)
Generating a G code program (Page 221)

### 5.10.1 Searching in programs

You can use the search function to quickly arrive at points where you would like to make changes, e.g. in very large programs.
Various search options are available that enable selective searching.

## Search options

- Whole words

Activate this option and enter a search term if you want to search for texts/terms that are present as words in precisely this form.
If, for example, you enter the search term "Finishing tool", only single "Finishing tool" terms are displayed. Word combinations such as "Finishing tool_10" are not found.

- Exact expression

Activate this option if you wish to search for terms with characters, which can also be used as place holders for other characters, e.g. "?" and "*".

## Note

## Search with place holders

When searching for specific program locations, you have the option of using place holders:

- "*": Replaces any character string
- "?": Replaces any character


## Precondition

The desired program is opened in the editor.

## Procedure

## Search



1. Press the "Search" softkey.

A new vertical softkey bar appears.
The "Search" window opens at the same time.
2. Enter the desired search term in the "Text" field.
3. Select "Whole words" if you want to search for whole words only.

- OR -

Activate the "Exact expression" checkbox if, for example, you want to search for place holders ("*", "?") in program lines.
4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.


Further search options

| Softkey | Function |
| :---: | :--- |
| Go to <br> start | The cursor is set to the first character in the program. |
| Go to <br> end | The cursor is set to the last character in the program. |

### 5.10.2 Replacing program text

You can find and replace text in one step.

Precondition
The desired program is opened in the editor

## Procedure

## Search

Find + replace

1. Press the "Search" softkey.

A new vertical softkey bar appears.
2. Press the "Find and replace" softkey.

The "Find and Replace" window appears.
3. In the "Text" field, enter the term you are looking for and in the "Replace with" field, enter the text you would like to insert automatically during the search.
4. Position the cursor in the "Direction" field and choose the search direction (forward, backward) with the <SELECT> key.


OK

Replace

## Replace

 allContinue search

## $x$

Cancel
5. Press the "OK" softkey to start the search.

If the text you are searching for is found, the corresponding line is highlighted.
6. Press the "Replace" softkey to replace the text.

- OR -

Press the "Replace all" softkey to replace all text in the file that corresponds to the search term.

- OR -

Press the "Continue search" softkey if the text located during the search should not be replaced.

- OR -

Press the "Cancel" softkey when you want to cancel the search.

## Note

## Replacing texts

- Read-only lines (;*RO*)

If hits are found, the texts are not replaced.

- Contour lines (;*GP*)

If hits are found, the texts are replaced as long as the lines are not read-only.

- Hidden lines (;*HD*)

If hidden lines are displayed in the editor and hits are found, the texts are replaced as long as the lines are not read-only. Hidden lines that are not displayed, are not replaced.

## See also

Editor settings (Page 153)

### 5.10.3 Copying/pasting/deleting a program block

## Precondition

The program is opened in the editor.

## Procedure

## Mark



1. Press the "Mark" softkey.

- OR -

Press the <SELECT> key.
2. Select the desired program blocks with the cursor or mouse.
3. Press the "Copy" softkey in order to copy the selection to the buffer memory.
4. Place the cursor on the desired insertion point in the program and press the "Paste" softkey.

The content of the buffer memory is pasted.

- OR -

Press the "Cut" softkey to delete the selected program blocks and to copy them into the buffer memory.

Note: When editing a program, you cannot copy or cut more than 1024 lines. While a program that is not on the NC is opened (progress display less than 100\%), you cannot copy or cut more than 10 lines or insert more than 1024 characters.

## Numbering the program blocks

If you have selected the "Automatic numbering" option for the editor, then the newly added program blocks are allocated a block number ( N number).
The following rules apply:

- When creating a new program, the first line is allocated the "first block number".
- If, up until now, the program had no N number, then the program block inserted is allocated the starting block number defined in the "First block number" input field.
- If N numbers already exist before and after the insertion point of a new program block, then the N number before the insertion point is incremented by 1.
- If there are no N numbers before or after the insertion point, then the maximum N number in the program is increased by the "increment" defined in the settings.


## Note:

After exiting the program, you have the option of renumbering the program blocks.

## Note

The buffer memory contents are retained even after the editor is closed, enabling you to paste the contents in another program.

## Note

## Copy/cut current line

To copy and cut the current line where the cursor is positioned, it is not necessary to mark or select it. You have the option of making the "Cut" softkey only operable for marked program sections via editor settings.

## See also

Opening additional programs (Page 152)
Editor settings (Page 153)
Keys of the operator panel (Page 27)

### 5.10.4 Renumbering a program

You can modify the block numbering of programs opened in the editor at a later point in time.

## Precondition

The program is opened in the editor.

## Procedure



Renumbering


1. Press the ">>" softkey.

A new vertical softkey bar appears.
2. Press the "Renumber" softkey.

The "Renumbering" window appears.
3. Enter the values for the first block number and the increment to be used for numbering.
4. Press the "OK" softkey.

The program is renumbered.

## Note

- If you only want to renumber a section, before the function call, select the program blocks whose block numbering you want to edit.
- When you enter a value of "0" for the increment size, then all of the existing block numbers are deleted from the program and/or from the selected range.


### 5.10.5 Creating a program block

In order to structure programs to achieve a higher degree of transparency, you have the option of combining several blocks (G-code and/or ShopTurn machining steps) to form program blocks.
Program blocks can be created in two stages. This means that you can form additional blocks within a block (nesting).

You then have the option of opening and closing these blocks depending on your requirement.

| Display | Meaning |
| :---: | :---: |
| Text | Block designation |
| Spindle | - Selecting the spindle <br> Defines at which spindle a program block is to be executed. |
| Addit. run-in code | - Yes <br> If the block is not to be executed, because the specified spindle is not to be used, it is possible to temporarily activate a so-called "Addit. run-in code". <br> - No |
| Automat. retraction | - Yes <br> Block start and block end are moved to the tool change point, i.e. the tool is brought into a safe range. <br> - No |

## Structuring programs

- Before generating the actual program, generate a program frame using empty blocks.
- By forming blocks, structure existing G code or ShopTurn programs.


## Procedure

Program manager

NC

Local drive

Build group


1. Select the "Program manager" operating area.
2. Select the storage location and create a program or open a program. The program editor opens.
3. Select the required program blocks that you wish to combine to form a block.
4. Press the "Build block" softkey.

The "Build New Block" window opens.
5. Enter a designation for the block, assign the spindle, if required, select the Addit. run-in code and the automatic retraction and then press the "OK" softkey.

## Opening and closing blocks

6. Press the ">>" and "View" softkeys.

Uiew

Open
blocks

## Close

 blocks7. Press the "Open all blocks" softkey if you wish to display the program with all the blocks.
8. Press the "Close all blocks" softkey, if you wish to display the program again in a structured form.

## Remove block

9. Open the block.
10. Position the cursor at the end of the block.

Remoue
block
11. Press the "Remove block" softkey.

## Note

You can also open and close blocks using the mouse or the cursor keys:

- <Cursor right> opens the block where the cursor is positioned
- <Cursor left> closes the block if the cursor is positioned at the beginning or end of the block
- <ALT> and <Cursor left> closes the block if the cursor is positioned within the block


### 5.10.6 Opening additional programs

You have the option of viewing and editing several programs simultaneously in the editor.
For instance, you can copy program blocks or machining steps of a program and paste them into another program.

Opening several programs
You have the option of opening up to ten program blocks.

## Open



Close

1. In the program manager, select the programs that you wish to open and view in the multiple editor and then press on the "Open" softkey. The editor is opened and the first two programs are displayed.
2. Press the <NEXT WINDOW> key to change to the next opened program.
3. Press the "Close" softkey to close the actual program.

## Note

Pasting program blocks
JobShop machining steps cannot be copied into a G code program.

## Precondition

You have opened a program in the editor.

## Procedure



1. Press the ">>" and "Open additional program" softkeys.

The "Select Additional Program" window is opened.
2. Select the program or programs that you wish to display in addition to the already opened program.
3. Press the "OK" softkey.

The editor opens and displays both programs next to each another.

## See also

### 5.10.7 Editor settings

Enter the default settings in the "Settings" window that are to take effect automatically when the editor is opened.

## Defaults

| Setting | Meaning |
| :---: | :---: |
| Number automatically | - Yes: A new block number will automatically be assigned after every line change. In this case, the specifications provided under "First block number" and "Increment" are applicable. <br> - No: No automatic numbering |
| First block number | Specifies the starting block number of a newly created program. <br> The field is only visible when "Yes" is displayed under "Number automatically". |
| Increment | Defines the increment used for the block numbers. <br> The field is only visible when "Yes" is displayed under "Number automatically". |
| Display hidden lines | - Yes: Hidden lines marked with "*HD" (hidden) will be displayed. <br> - No: Lines marked with ";*HD*" will not be displayed. <br> Note: <br> Only visible program lines are taken into account with the "Search" and "Search and Replace" functions. |
| Display block end a symbol | The "LF" (line feed) symbol $\mathbb{I}$ is displayed at the block end. |
| Line break | - Yes: Long lines are broken and wrapped around. <br> - No: If the program includes long lines, then a horizontal scrollbar is displayed. You can move the section of the screen horizontally to the end of the line. |
| Line break also in standard cycles | - Yes: If the line of a cycle call becomes too long, then it is displayed over several lines. <br> - No: The cycle call is truncated. <br> The field is only visible when "Yes" is displayed under "Line break". |
| Visible programs | - 1-10 <br> Select how many programs can be displayed next to one another in the editor. <br> - Auto <br> Specifies that the number of programs entered in a job list or up to 10 selected programs will be displayed next to each other. |
| Width of the program with focus | Here, you enter the width of the program that has the input focus in the editor as a percentage of the window width. |


| Setting | Meaning |
| :---: | :---: |
| Automatic save (only local and external drives) | - Yes: The changes are saved automatically when you change to another operating area. <br> - No: You are prompted to save when changing to another operating area. Save or reject the changes with the "Yes" and "No" softkeys. |
| Cut only after selecting | - Yes: The cutting of program sections is possible only when program lines have been selected, i.e. the "Cut" softkey only then is active. <br> - No: The program line, in which the cursor is positioned, can be cut out without having to select it. |
| Determine processing times | Defines which program runtimes are determined in the simulation: <br> - Off <br> - Block by block: The runtimes are determined for each program block. <br> - Non-modal: The runtimes are determined at the NC block level. <br> After the simulation, the required processing times are displayed in the editor. |
| Saving machining times | Specifies how the machining times determined are processed. <br> - Yes <br> A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. There, the machine times determined are saved in an ini file together with the name of the program. <br> - No <br> The machining times that have been determined are only displayed in the editor. |
| Display cycles as machining step | - Yes: The cycle calls in the G code programs are displayed as plain text. <br> - No: The cycle calls in the G code programs are displayed in the NC syntax. |
| Font size | Defines the font size for the editor and displaying the program sequence. |

## Note

All entries that you make here are effective immediately.

## Precondition

You have opened a program in the editor.

## Procedure

$\square$
Program

2. Press the ">>" and "Settings" softkeys.

The "Settings" window opens.

## Settings


3. Make the desired changes here and press the "OK" softkey to confirm your settings.

### 5.11 Display and edit user variables

### 5.11.1 Overview

The defined user data may be displayed in lists.
The following variables can be defined:

- Data parameters (R parameters)
- Global user data (GUD) is valid in all programs
- Local user variables (LUD) are valid in the program where they have been defined.
- Program-global user variables (PUD) are valid in the program in which they have been defined, as well as in all of the subprograms called by this program

Channel-specific user data can be defined with a different value for each channel.

## Entering and displaying parameter values

Up to 15 positions (including decimal places) are evaluated. If you enter a number with more than 15 places, it will be written in exponential notation ( 15 places + EXXX).

## LUD or PUD

Only local or program-global user data can be displayed at one time.
Whether the user data are available as LUD or PUD depends on the current control configuration.

## Machine manufacturer

\{0\}
Please refer to the machine manufacturer's specifications.

## Note

## Reading and writing variables protected

Reading and writing of user data are protected via a keyswitch and protection levels.

## Searching for user data

You may search for user data within the lists using any character string.
Refer to the "Defining and activating user data" section to learn how to edit displayed user data.

### 5.11.2 R parameters

R parameters (arithmetic parameters) are channel-specific variables that you can use within a G code program. G code programs can read and write R parameters.

These values are retained after the controller is switched off.

## Number of channel-specific R parameters

The number of channel-specific $R$ parameters is defined in a machine data element.
Range: R0-R999 (dependent on machine data).
There are no gaps in the numbering within the range.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

## Procedure



R
variables

1. Select the "Parameter" operating area
2. Press the "User variable" softkey.
3. Press the "R variables" softkey

The "R Parameters" window appears.

## Delete R variables



Delete

1. Press the ">>" and "Delete" softkeys.

The "Delete R parameters" window appears.
2. Enter the R parameter(s) whose channel-specific values you would like to delete and press the "OK" softkey.

A value of 0 is assigned to the selected $R$ parameters or to all $R$ parameters.

### 5.11.3 Displaying global user data (GUD)

## Global user variables

Global GUDs are NC global user data (Global User Data) that remains available after switching the machine off.

GUDs apply in all programs.

## Definition

A GUD variable is defined with the following:

- Keyword DEF
- Range of validity NCK
- Data type (INT, REAL, ....)
- Variable names
- Value assignment (optional)


## Example

DEF NCK INT ZAEHLER1 = 10
GUDs are defined in files with the ending DEF. The following file names are reserved for this purpose:

| File name | Meaning |
| :--- | :--- |
| MGUD.DEF | Definitions for global machine manufacturer data |
| UGUD.DEF | Definitions for global user data |
| GUD4.DEF | User-definable data |
| GUD8.DEF, GUD9.DEF | User-definable data |

## Procedure



Parameter


Global GUD

1. Select the "Parameter" operating area.
2. Press the "User variable" softkey.
3. Press the "Global GUD" softkeys.

The "Global User Variables" window is displayed. A list of the defined UGUD variables will be displayed.

- OR -

GUD selection

GUD selection


GUD9

Press the "GUD selection" softkey and the "SGUD" to "GUD6" softkeys if you wish to display SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the global user variables.

- OR -

Press the "GUD selection" and ">>" softkeys as well as the "GUD7" to "GUD9" softkeys if you want to display GUD 7 to GUD 9 of the global user variables.

## Note

After each start-up, a list with the defined UGUD variables is displayed in the "Global User Variables" window.

### 5.11. Displaying channel GUDs

## Channel-specific user variables

Like the GUDs, channel-specific user variables are applicable in all programs for each channel. However, unlike GUDs, they have specific values.

## Definition

A channel-specific GUD variable is defined with the following:

- Keyword DEF
- Range of validity CHAN
- Data type
- Variable names
- Value assignment (optional)


## Example

$$
\text { DEF CHAN REAL X_POS = } 100.5
$$

## Procedure



Parameter

2. Press the "User variable" softkey.
3. Press the "Channel GUD" and "GUD selection" softkeys.

Channel GUDD

GUD selection

4. Press the "SGUD" ... "GUD6" softkeys if you want to display the SGUD, MGUD, UGUD as well as GUD4 to GUD 6 of the channel-specific user variables.

- OR -

Press the "Continue" softkey and the "GUD7" to "GUD9" softkeys if you want to display GUD 7 to GUD 9 of the channel-specific user variables.

### 5.11.5 Displaying local user data (LUD)

## Local user variables

LUDs are only valid in the program or subprogram in which they were defined.
The controller displays the LUDs after the start of program processing. The display is available until the end of program processing.

## Definition

A local user variable is defined with the following:

- Keyword DEF
- Data type
- Variable names
- Value assignment (optional)


## Procedure



Parameter

Local LUD

1. Select the "Parameter" operating area.
2. Press the "User variable" softkey.
3. Press the "Local LUD" softkey.

### 5.11.6 Displaying program user data (PUD)

## Program-global user variables

PUDs are global part program variables (Program User Data). PUDs are valid in all main programs and subprograms, where they can also be written and read.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

## Procedure



### 5.11.7 Searching for user variables

You can search for $R$ parameters and user variables.

## Procedure



Parameter

| R |
| :---: |
| variables |$|$| Local |
| :---: |
| LUD |

Search


1. Select the "Parameter" operating area.
2. Press the "R parameters", "Global GUD", "Channel GUD", "Local GUD" or "Program PUD" softkeys to select the list in which you would like to search for user variables.
3. Press the "Search" softkey.

The "Search for R Parameters" or "Search for User Variables" window opens.
4. Enter the desired search term and press "OK".

The cursor is automatically positioned on the $R$ parameter or user variables you are searching for, if they exist.

By editing a DEF/MAC file, you can alter or delete existing definition/macro files or add new ones.

## Procedure


3. In the data tree, select the "NC data" folder and then open the "Definitions" folder.
4. Select the file you want to edit.
5. Double-click the file.

- OR -

Press the "Open" softkey.

- OR -

Press the <INPUT> key.

- OR -

Press the <Cursor right> key.
The selected file is opened in the editor and can be edited there.
6. Define the desired user variable.
7. Press the "Exit" softkey to close the editor.

1. Select the "Start-up" operating area.
2. Press the "System data" softkey.

## Activating user variables

## Activate

OK

1. Press the "Activate" softkey.

A prompt is displayed.
2. Select whether the current values in the definition files should be retained

- OR -

Select whether the current values in the definition files should be deleted.

This will overwrite the definition files with the initial values.
3. Press the "OK" softkey to continue the process.

### 5.12 Displaying G functions and auxiliary functions

### 5.12.1 Selected G functions

16 selected G groups are displayed in the "G Function" window.
Within a G group, the G function currently active in the controller is displayed.
Some G codes (e.g. G17, G18, G19) are immediately active after switching the machine control on.

Which G codes are always active depends on the settings.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

## G groups displayed by default

| Group | Meaning |
| :--- | :--- |
| G group 1 | Modally active motion commands (e.g. G0, G1, G2, G3) |
| G group 2 | Non-modally active motion commands, dwell time (e.g. G4, G74, G75) |
| G group 3 | Programmable offsets, working area limitations and pole programming (e.g. <br> TRANS, ROT, G25, G110) |
| G group 6 | Plane selection (e.g. G17, G18) |
| G group 7 | Tool radius compensation (e.g. G40, G42) |
| G group 8 | Settable work offset (e.g. G54, G57, G500) |
| G group 9 | Offset suppression (e.g. SUPA, G53) |
| G group 10 | Exact stop - continuous-path mode (e.g. G60, G641) |
| G group 13 | Workpiece dimensioning inches/metric (e.g. G70, G700) |
| G group 14 | Workpiece dimensioning absolute/incremental (G90) |
| G group 15 | Feedrate type (e.g. G93, G961, G972) |
| G group 16 | Feedrate override on inside and outside curvature (e.g. CFC) |
| G group 21 | Acceleration profile (e.g. SOFT, DRIVE) |
| G group 22 | Tool offset types (e.g. CUT2D, CUT2DF) |
| G group 29 | Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF) |
| G group 30 | Compressor ON/OFF (e.g. COMPOF) |

## G groups displayed by default (ISO code)

| Group | Meaning |
| :--- | :--- |
| G group 1 | Modally active motion commands (e.g. G0, G1, G2, G3) |
| G group 2 | Non-modally active motion commands, dwell time (e.g. G4, G74, G75) |
| G group 3 | Programmable offsets, working area limitations and pole programming (e.g. <br> TRANS, ROT, G25, G110) |
| G group 6 | Plane selection (e.g. G17, G18) |
| G group 7 | Tool radius compensation (e.g. G40, G42) |
| G group 8 | Settable work offset (e.g. G54, G57, G500) |
| G group 9 | Offset suppression (e.g. SUPA, G53) |
| G group 10 | Exact stop - continuous-path mode (e.g. G60, G641) |
| G group 13 | Workpiece dimensioning inches/metric (e.g. G70, G700) |
| G group 14 | Workpiece dimensioning absolute/incremental (G90) |
| G group 15 | Feedrate type (e.g. G93, G961, G972) |
| G group 16 | Feedrate override on inside and outside curvature (e.g. CFC) |
| G group 21 | Acceleration profile (e.g. SOFT, DRIVE) |
| G group 22 | Tool offset types (e.g. CUT2D, CUT2DF) |
| G group 29 | Radius/diameter programming (e.g. DIAMOF, DIAMCYCOF) |
| G group 30 | Compressor ON/OFF (e.g. COMPOF) |

## Procedure



1. Select the "Machine" operating area.

Machine

...

2. Press the <JOG>, <MDA> or <AUTO> key.
3. Press the "G functions" softkey.

The "G Functions" window is opened.
4. Press the "G functions" softkey again to hide the window.

The $G$ groups selection displayed in the "G Functions" window may differ.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

## References

For more information about configuring the displayed $G$ groups, refer to the following document:

SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

### 5.12.2 All G functions

All G groups and their group numbers are listed in the "G Functions" window.
Within a G group, only the G function currently active in the controller is displayed.

## Additional information in the footer

The following additional information is displayed in the footer:

- Actual transformations

| Display | Meaning |
| :--- | :--- |
| TRANSMIT | Polar transformation active |
| TRACYL | Cylinder surface transformation active |
| TRAORI | Orientation transformation active |
| TRAANG | Inclined axis transformation active |
| TRACON | Cascaded transformation active <br> For TRACON, two transformations (TRAANG and TRACYL or TRAANG and <br> TRANSMIT) are activated in succession. |

- Current work offsets
- Spindle speed
- Path feedrate
- Active tool


### 5.12.3 G functions for mold making

In the window "G functions", important information for machining free-form surfaces can be displayed using the "High Speed Settings" function (CYCLE832).

## Software option

You require the "Advanced Surface" software option in order to use this function.

## High-speed cutting information

In addition to the information that is provided in the "All G functions" window, the following programmed values of the following specific information is also displayed:

- CTOL
- OTOL
- STOLF

The tolerances for G0 are only displayed if they are active.
Particularly important $G$ groups are highlighted.
You have the option to configure which $G$ functions are highlighted.

## References

- Additional information is available in the following references:

Function Manual, Basic Functions; Chapter, "Contour/orientation tolerance"

- For information about configuring the displayed G groups, refer to the following document:

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

## Procedure

'

## Machine


2. Press the <JOG>, <MDI> or <AUTO> key.
3. Press the ">>" and "All G functions" softkeys.

The "G Functions" window is opened.

## See also

High-speed settings (CYCLE832) (Page 568)

Turning
5.12 Displaying $G$ functions and auxiliary functions

### 5.12.4 Auxiliary functions

Auxiliary functions include M and H functions preprogrammed by the machine manufacturer, which transfer parameters to the PLC to trigger reactions defined by the manufacturer.

## Displayed auxiliary functions

Up to five current M functions and three H functions are displayed in the "Auxiliary Functions" window.

## Procedure


2. Press the <JOG>, <MDA> or <AUTO> key.

1. Select the "Machine" operating area.


Auxiliary functions

Auxiliary functions
3. Press the "H functions" softkey. The "Auxiliary Functions" window opens.
4. Press the "H functions" softkey again to hide the window again.

You can display status information for diagnosing synchronized actions in the "Synchronized Actions" window.

You get a list with all currently active synchronized actions.
In this list, the synchronized action programming is displayed in the same form as in the part program.

## References

Programming Guide Job Planning (PGA) Chapter: Motion-synchronous actions

## Status of synchronized actions

You can see the status of the synchronized actions in the "Status" column.

- Waiting
- Active
- Blocked

Non-modal synchronized actions can only be identified by their status display. They are only displayed during execution.

## Synchronization types

| Synchronization types | Meaning |
| :--- | :--- |
| ID=n | Modal synchronized actions in the automatic mode up to the end of pro- <br> gram, local to program; $n=1 \ldots 254$ |
| IDS=n | Static synchronized actions, modally effective in every operating type, <br> also beyond the end of program; $n=1 \ldots 254$ |
| Without ID/IDS | Non-modal synchronized actions in automatic mode |

## Note

Numbers ranging from 1 to 254 can only be assigned once, irrespective of the identification number.

## Display of synchronized actions

Using softkeys, you have the option of restricting the display to activated synchronized actions.

## Procedure



Machine

2. Press the <AUTO>, <MDA> or <JOG> key.


ID
4. Press the "ID" softkey if you wish to hide the modal synchronized actions in the automatic mode.

- AND / OR -

Press the "IDS" softkey if you wish to hide static synchronized actions.

- AND / OR -

Press the "Blockwise" softkey if you wish to hide the non-modal synchronized actions in the automatic mode.
5. Press the "ID", "IDS" or "Blockwise" softkeys to re-display the corresponding synchronized actions.

### 5.13 Mold making view

### 5.13.1 Overview

For large mold making programs, as provided by CAD systems, you have the option, using a fast view, to display the machining paths. This allows you to obtain a fast overview of the program and possibly correct it.


## Machine manufacturer

The mold making view can be hidden. Please refer to the machine manufacturer's instructions.

## Checking the program

For instance, you can check whether

- The programmed workpiece has the correct shape
- There are large traversing errors
- A block must be corrected
- How the approach and retraction are performed


## Simultaneous view of the program and mold making view

In the editor, next to the program block display, switch-in the graphic view.
At the left in the editor, if you place the cursor on an NC block with position data, then this NC block is selected in the graphic view.

If you select a point to the right in the graphic view - then the NC block is selected in the lefthand part of the editor - i.e. in the other direction. This is how you jump directly to a position in the program in order to edit a program block for example.

| USB／4＿F＿Finish |
| :---: |
| N1 ；Start of Path\＃ |
| H2 ；${ }^{\text {¢ }}$ |
| N3 ；TECHNOLOGY：MILL＿FINIS |
| H\｜ |
| N4 ；TOOL NAME ：RADIUSFRÄS |
| ER＿D89 |
| H5 ；T00L TYPE ：Milling To ol－Ball Milla |
| N6 ；${ }^{\text {d }}$ |
| N7 ；Intol ：0．0850日8® |
| N8；Outtol ：0．日858日里 |
|  |
| N18 ；Cantolerance＝0．01『 |
| N11 ； 9 |
| N12 ；Operation ：FINISH＿0＿ |
| F9I |
| H13 ；Second Tool『 |
| N14 T＝＂BALL＿D8＿R＂D1『 |
| N15 M69 |
| N1R monoga moat |

NC blocks that can be interpreted
The following NC blocks are supported for the mold making view：
－Types
－Lines
G0，G1 with X Y Z
－Circles
G2，G3 with center point I，J，K or radius CR，depending on the working plane G17， G18，G19，CIP with circular point I1，J1，K1 or radius CR
－Polynomials
POLY with $\mathrm{X}, \mathrm{Y}, \mathrm{Z}$ or $\mathrm{PO}[\mathrm{X}] \mathrm{PO}[\mathrm{Y}] \mathrm{PO}[Z]$
－B－splines
BSPLINE with degree $\operatorname{SD}(S D<6)$ ，node PL，weights PW
－Absolute data AC and incremental data IC are possible
－For G2，G3 and different radii at the start and end，an Archimedes spiral is used

- Orientation
- Rotary axis programming with ORIAXES or ORIVECT using ABC for G0, G1, G2, G3, CIP, POLY
- Rotary axis programming with ORIAXES or ORIVECT using PO[A] POS[b] PO[C] for POLY
- Orientation vector programming with ORIVECT using A3, B3, C3 for G0, G1, G2, G3, CIP
- Orientation curve with ORICURVE using XH, YH, ZH, for G0, G1, G2, G3, CIP, POLY, BSPLINE
- Orientation curve with ORICURVE using PO[XH] PO[YH] PO[ZH] for POLY
- Rotary axes can be specified using DC
- G codes
- Working planes (for circle definition G2, G3): G17 G18 G19
- Incremental or absolute data: G90 G91

The following NC blocks are not supported for the mold making view:

- Helix programming
- Rational polynomials
- Other G codes or language commands

All blocks that cannot be interpreted are simply ignored

## Changing and adapting the mold making view

Like simulation and simultaneous recording, you have the option of changing and adapting the simulation graphical representation in order to achieve the optimum view.

- Increasing or reducing the size of the graphic
- Moving the graphic
- Rotating the graphic
- Changing the section


### 5.13.2 Starting the mold making view

## Procedure

Open


Mold mak view

NE blocks

## Graphic

Hide $G 1$ lines

Hide
G0 lines

Hide
points

## Graphic

1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the program that you would like to display in the mold making view.
3. Press the "Open" softkey.

The program is opened in the editor.
4. Press the ">>" and "Mold making view" softkeys.

The editor splits up into two areas.

The G code blocks are displayed in the left half of the editor. The workpiece graphic is displayed in the right half of the editor. All of the points and paths programmed in the part program are shown in the graphical representation.
5. Press softkey "Lines G1" or "Lines G0", to hide the corresponding line elements of the graphic.

- OR -

Press softkey "Hide points" to hide the points of the graphic.

## Note:

You have the option of simultaneously hiding G1 and G0 lines. In this case softkey "Hide points" is deactivated.
6. Press the "Graphic" softkey in order to hide the graphic and to view the program as usual in the editor

- OR -

Press the "NC blocks" softkey in order to hide the G code blocks and therefore only to display the graphic.

### 5.13.3 Specifically jump to the program block

If you notice anything peculiar in the graphic or identify an error, then from this location, you can directly jump to the program block involved to possibly edit the program.

## Requirements

- The required program is opened in the mold making view.
- The "Graphic" softkey is active.


## Procedure



1. Press the ">>" and "Select point" softkeys.

Cross-hairs for selecting a point are shown in the diagram.

2. Using the cursor keys, move the cross-hairs to the appropriate position in the graphic.

Select
NC block
3. Press the "Select NC block" softkey. The cursor jumps to the associated program block in the editor.

### 5.13.4 Searching for program blocks

Using the "Search" function, you can search for specific program blocks as well as edit programs; you can do this in one step by replacing the text that you are searching for by a new text.

Precondition

- The required program is opened in the mold making view.
- The "NC blocks" softkey is active.


## Procedure

1. Press the "Search" softkey.

A new vertical softkey bar appears.

## See also

Replacing program text (Page 146)

### 5.13.5 Changing the view

### 5.13.5.1 Enlarging or reducing the graphical representation

## Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.


## Procedure



- OR -

Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.

Zoom -

- OR -

Press the "Details" and "Auto zoom" softkeys if you wish to automatical-

Details

Autozoom ly adapt the segment to the size of the window.
The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

## Note

## Selected section

The selected sections and size changes are kept as long as the program is selected.

### 5.13.5.2 Modifying the viewport

Use the magnifying glass if you would like to move, increase or reduce the size of the section of the mold making view, e.g. to view details or display the complete workpiece.

Using the magnifying glass, you can define your own segment and then increase or decrease its size.

## Precondition

- The mold making view has been started.
- The "Graphic" softkey is active.


## Procedure

## Details

```
Zoom
```

Zoom +

## Zoom -

- OR -

Press the "Magnify -" or <-> softkey to reduce the frame.

- OR -


Press one of the cursor keys to move the frame up, down, left or right.

4. Press the "Accept" softkey to accept the section.

### 5.14 Displaying the program runtime and counting workpieces

To gain an overview of the program runtime and the number of machined workpieces, open the "Times, Counter" window.

## Machine manufacturer

Please refer to the machine manufacturer's specifications.

## Displayed times

## - Program

Pressing the softkey the first time shows how long the program has already been running.
At every further start of the program, the time required to run the entire program the first time is displayed.
If the program or the feedrate is changed, the new program runtime is corrected after the first run.

- Program remainder

Here you can see how long the current program still has to run. In addition, you can track how much of the current program has been completed as a percentage by using the progress bar.
The display only appears when the program is run a second time.
If you are executing the program from an external location, the program loading progress is displayed here.

- Influencing the time measurement

The time measurement is started with the start of the program and ends with the end of the program (M30) or with an agreed M function.

When the program is running, the time measurement is interrupted with CYCLE STOP and continued with CYCLE START.
The time measurement starts at the beginning with RESET and subsequent CYCLE START.
The time measurement stops with CYCLE STOP or a feedrate override $=0$.

## Counting workpieces

You can also display program repetitions and the number of completed workpieces. For the worpiece count, enter the actual and planned workpiece numbers.

## Workpiece count

Completed workpieces can be counted via the end of program command (M30) or an M command.
5.14 Displaying the program runtime and counting workpieces

## Procedure

| $\bar{M}$ | 1. | Select the "Machine" operating area. |
| :---: | :---: | :---: |
| $\underset{\text { AUTO }}{\rightarrow}$ | 2. | Press the <AUTO> key. |
| Time counter | 3. | Press the "Times, Counter" softkey. <br> The "Times, Counter" window opens. |
| $\bigcup_{\text {SELECT }}$ | 4. | Select "Yes" under "Count workpieces" if you want to count completed workpieces. |
|  | 5. | Enter the number of workpieces needed in the "Desired workpieces" field. |
|  |  | The number of workpieces already finished is displayed in "Actual workpieces". This value can be corrected if necessary. |
|  |  | After the defined number of workpieces is reached, the current workpieces display is automatically reset to zero. |

## See also

Entering the number of workpieces (Page 258)

### 5.15 Setting for automatic mode

Before machining a workpiece, you can test the program in order to identify programming errors early on. Use the dry run feedrate for this purpose.

In addition, you have the option of additionally limiting the traversing speed for rapid traverse so that when running-in a new program with rapid traverse, no undesirable high traversing speeds occur.

## Dry run feedrate

The feedrate defined here replaces the programmed feedrate during execution if you have selected "DRY run feedrate" under program control.

## Reduced rapid traverse

This value entered here reduces the rapid traverse to the entered percentage value if you selected "RG0 reduced rapid traverse" under program control.

## Displaying measurement results

Using an MMC command, you can display measurement results in a part program:
You can define the following settings:

- When it reaches the command, the control automatically jumps to the "Machine" operating area and the window with the measurement results is displayed
- The window with the measurement results is opened by pressing the "Measurement result" softkey.


## Recording machining times

To provide support when creating and optimizing a program, you have the option of displaying the machining times.

You define whether the time is determined while the workpiece is being machined (i.e. if the function is energized).

- Off

When the machining a workpiece the time determination function is deactivated, i.e. the machining times are not determined.

- Non-modal

The machining times are determined for each traversing block of a main program.

- Block by block

Machining times are determined for all blocks.

## Note

## Utilization of resources

The more machining times are displayed, the more resources are utilized.
More machining times are determined and saved with the non-modal setting as with the block-by-block setting.

## Saving machining times

Here, you specify how the machining times determined are processed.

- Yes

A subdirectory with the name "GEN_DATA.WPD" is created in the directory of the part program. There, the machine times determined are saved in an ini file together with the name of the program.

- No

The machining times that have been determined are only displayed in the program block display.

## Procedure



Machine

2. Press the <AUTO> key.
3. Press the menu forward key and the "Settings" softkey.

The "Settings for Automatic Operation" window is opened.

## Settings

4. In "DRY run feedrate," enter the desired dry run speed.
5. Enter the desired percentage in the "Reduced rapid traverse RG0" field. RG0 has not effect if you do not change the specified amount of $100 \%$.
6. Select the required entry in the "Display measurement result" field:

- "automatic", if the measurement result window is to be automatically opened.
- "manual", if the measurement result window is to be opened by pressing the "Measurement result" softkey.

7. Select the required entry in the "Record machining times" and where relevant, in the "Save machining times" field.

## References

Programming Manual Measuring Cycles / 840D sl/828D

## Note

The feedrate can be changed while the operation is running.

### 5.16 Working with DXF files

### 5.16.1 Overview

The "DXF-Reader" function can be used to open files created in the SINUMERIK Operate editor directly in a CAD system as well as contours and drilling positions to be transferred and stored directly in G code and ShopTurn programs.

The DXF file can be displayed in the Program Manager.


## Software option

You require the "DXF-Reader" software option in order to use this function.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

### 5.16.2 Displaying CAD drawings

### 5.16.2.1 Open a DXF file

## Procedure



1. Select the "Program Manager" operating area.
2. Choose the desired storage location and position the cursor on the DFX file that you want to display.
3. Press the "Open" softkey.

The selected CAD drawing will be displayed with all its layers, i.e. with all graphic levels.

Exit

## Open

an
Press the "Close" softkey to close the CAD drawing and to return to the Program Manager.

### 5.16.2.2 Cleaning a DXF file

All contained layers are shown when a DXF file is opened.
Layers that do not contain any contour- or position-relevant data can be shown or hidden.

## Procedure

1. The DXF file is opened in the Program Manager or in the editor.

Clear

Layer selection
2. Press the "Clean" and "Layer selection" softkeys if you want to hide specific layers.

The "Layer Selection" window opens.

3. Deactivate the required layers and press the "OK" softkey.

- OR -

Clear
autom.

Clear
autom.

OK
6. Press the "Clean automat." softkey to hide all non-relevant layers.
7. Press the "Clean automat." softkey to redisplay the layers.

### 5.16.2.3 Enlarging or reducing the CAD drawing

## Precondition

The DXF file is opened in the Program Manager.

## Procedure



### 5.16.2.4 Modifying the section

If you want to move or change the size of a section of the drawing, for example, to view details or redisplay the complete drawing later, use the magnifying glass.
You can use the magnifying glass to determine the section and then change its size.

## Precondition

The DXF file is opened in the Program Manager or in the editor.

## Procedure

## Details

```
Zoom
Zoom
```

2. Press the <+> key to enlarge the frame.

- OR -

Press the <-> key to reduce the frame.

- OR -


Press a cursor key to move the frame up, down, left or right.

1. Press the "Details" and "Magnifying glass" softkeys.

A magnifying glass in the shape of a rectangular frame appears.
$+$


OK
3. Press the "OK" softkey to accept the section.

### 5.16.2.5 Rotating the view

You can change the orientation of the drawing.

## Precondition

The DXF file is opened in the Program Manager or in the editor.

## Procedure

## Details

Rotate
image

...


1. Press the "Details" and "Rotate figure" softkeys.
2. Press the "Arrow right", "Arrow left", "Arrow up", "Arrow down", "Arrow clockwise" and "Arrow counter-clockwise" softkeys to change the position of the drawing.

### 5.16.2.6 Displaying/editing information for the geometric data

## Precondition

The DXF file is opened in the Program Manager or in the editor.

## Procedure

## Details

Geometry

Info/edit

Element

Info

## Edit

 element<<
Back
Geometry

1. Press the "Details" and "Geometry info" softkeys.

The cursor takes the form of a question mark.
2. Position the cursor on the element for which you want to display its geometric data and press the "Element info" softkey.

If, for example, you have selected a straight line, the following window opens, "Straight line on layer: ...". You are shown the coordinates appropriate for the current zero point in the selected layer: Start point for $X$ and $Y$, end point for $X$ and $Y$ as well as the length.
4. If you are currently in the editor, press the "Element edit" softkey.

The coordinate values can be edited.
3. Press the "Back" softkey to close the display window.

## Note

## Editing a geometric element

You can use this function to make smaller changes to the geometry, e.g. for missing intersections.

You should make larger changes in the input screen of the editor.
Changes that you make with "Element edit" cannot be undone.

### 5.16.3 Importing and editing a DXF file in the editor

### 5.16.3.1 General procedure

- Create/open a G code or ShopTurn program
- Call the "Turn contour" cycles and create a "New contour"
- OR -
- Call from "Drill" cycle "Position / position pattern"
- Import the DXF file
- Select the contour or drilling positions in the DXF file or CAD drawing and click "OK" to accept the cycle
- Add the program record with "Accept" to the G code or ShopTurn program


### 5.16.3.2 Specifying a reference point

Because the zero point of the DXF file normally differs from the zero point of the CAD drawing, specify a reference point.

## Procedure



1. The DXF file is opened in the editor.
2. Press the ">>" and "Specify reference point" softkeys.
Specify ref-
erence point
Element
start start
Element
center

Element end
3. Press the "Element start" softkey to place the zero point at the start of the selected element.

- OR -

Press the "Element center" softkey to place the zero point at the center of the selected element.

- OR -

Press the "Element end" softkey to place the zero point at the end of the selected element.

- OR -


## Arc center point

## Cursor

```
Free
input
```

Press the "Arc center" softkey to place the zero point at the center of an arc.

- OR -

Press the "Cursor" softkey to define the zero point at any cursor position.

- OR -

Press the "Free input" softkey to open the "Reference Point Input" window and enter the values for the positions ( $\mathrm{X}, \mathrm{Y}$ ) there.

### 5.16.3.3 Setting the tolerance

To allow even inaccurately created drawings to be used, i.e. to compensate for gaps in the geometry, you can enter a snap radius in millimeters. This relates elements.

## Note

## Large snap radius

The larger that the snap radius is set, the larger the number of available following elements.

## Procedure

1. The DXF file is opened in the editor.

Details

Tolerance
2. Press the "Details" and "Tolerance" softkeys.

The "Tolerance Input" window opens.
3. Enter the desired value and press the "OK" softkey.

### 5.16.3.4 Transferring the drilling positions

## Calling the cycles



1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Positions" softkey.
4. Press the "Arbitary positions" softkey.

The "Positions" input window opens.

- OR -

Press the "Line" softkey.
The "Position Row" input window opens.

- OR -

Press the "Grid" softkey.
The "Position Grid" input window opens.

- OR

Press the "Frame" softkey.
The "Position Frame" input window opens.

- OR -

Press the "Circle" softkey.
The "Position Circle" input window opens.

- OR -

Press the "Partial circle" softkey.
The "Position Partial Circle" input window opens.

## Selecting the drilling positions

## Precondition

You have selected a position pattern.

## Procedure

## Open a DXF file

Import from 1. Press the "Import from DXF" softkey.
DXF

Search
ok
. Press the "OK" softkey.
The CAD drawing opens and can be edited to select drilling positions. The cursor takes the shape of a cross.

Cleaning a file
4. Prior to selecting the drilling positions, you can select a layer and clean the file.

Specifying the reference point
5. If required, specify a zero point.

## Specify clearance(s) (for position pattern "Row" / "Arbitrary positions" and "Circle" / "Partial

 circle"Select element

Accept element
6. Press the "Select element" softkey and navigate the orange selection symbol by repeatedly pressing the desired drilling position.
7. Press the "Accept element" softkey to transfer the position.

Repeat steps 6 and 7 to specify other drilling positions for "Arbitrary positions".

Specify clearance with second clearance (for position pattern "Frame", "Grid")

Select element

Accept element

Select element

Accept element
8. Once the reference point has been specified, press the "Select element" softkey repeatedly to navigate to the desired drilling position in order to specify the clearance.
9. Press the "Accept element" softkey.

A rectangular cross-hair is displayed.
10. Press the "Select element" softkey and press it repeatedly to navigate to the desired drilling position on the displayed line.
To determine the second clearance, the drilling position must be located on the line.
11. Press the "Accept element" softkey.

A frame or grid is displayed.

## Size (for position pattern "Row", "Frame", "Grid")

Select 12. Once the reference point and clearances have been specified, press the element "Select element" softkey repeatedly.
All expansions of the frame or the grid are displayed.
Accept element
13. Press the "Accept element" softkey to confirm the selected frame or grid. If all elements for the position row or position frame and position grid are valid, the drilling positions are displayed with blue points.

## Circle direction (for circle and partial circle)

Select Once the reference point and clearance have been specified, press the element "Select element" softkey repeatedly.

The displayed circle is shown in the possible orientations.
Accept $\quad$ Press the "Select element" softkey to confirm the selected circle or parelement tial circle.
If all elements of the circle or partial circle are valid, the drilling positions are displayed with blue points.

## Resetting actions

Revert Undo can be used to reset the last actions.

Transfer drilling positions to the cycle and to the program

4. Press the "OK" softkey in order to accept the position values.

You return to the associated parameter screen form.
Press the "Accept" softkey to transfer the drilling positions to the program.

## Operation with keyboard and mouse

In addition to the operation using the softkeys, you can also operate the functions with the keyboard and the mouse.

### 5.16.3.5 Accepting contours

## Calling the cycles

. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.

New 3. Press the "New contour" softkey. contour

## Selecting contours

The start and end point are specified for the contour line.
The start point and the direction are selected on a selected element. Beginning at the start point, the automatic contour line takes all subsequent elements of a contour until there are no following elements or intersections with other elements.

## Note

If a contour includes more elements than can be processed, you will be offered the option of transferring the contour to the program as pure G code.

This contour then can no longer be edited in the editor.

## Procedure

## Opening a DXF file

1. Enter the desired name in the "New Contour" window.

Import from
DXF

Accept

## Search

OK
2. Press the "From DXF file" and "Accept" softkeys.

The "Open DXF File" window opens.
3. Select a storage location and place the cursor on the relevant DXF file.

You can, for example, use the search function to search directly for a DXF file in comprehensive folders and directories.
4. Press the "OK" softkey.

The CAD drawing opens and can be edited for contour selection. The cursor takes the form of a cross.

Specifying a reference point
5. If required, specify a zero point.

## Contour line



Automatic

Only to 1st step
6. Press the ">>" and "Automatic" softkeys if you want to accept the largest possible number of contour elements.
This makes it fast to accept contours that consist of many individual elements.

- OR -

Press "Only until first cut" if you do not want to accept the complete contour elements at once.
The contour will be followed to the first cut of the contour element.

## Defining the start point

Select element

Accept element

Element Starting point

Element end point

Element center
7. Press the "Select element" softkey to select the desired element.
8. Press the "Accept element" softkey.
9. Press the "Element start point" softkey to place the contour start at the start point of the element.

- OR -

Press the "Element end point" softkey to place the contour start at the end point of the element.

- OR -

Press the "Element center" softkey to place the contour start at the center of the element.

- OR -

Cursor

OK
OK
Accept element

Press the "Cursor" softkey to define the start of the element with the cursor at any position.
9. Press the "OK" softkey to confirm your selection.

## Specifying the end point

11. Press the ">>" and "Specify end point" softkeys if you do not want to accept the end point of the selected element.

Specify
end point
Current
10. Press the "Accept element" softkey to accept the offered elements.

The softkey can be operated while elements are still available to be accepted.

position center

Element
end
position

## Cursor

Element Press the "Element center" softkey to place the contour end at the center
12. Press the "Current position" softkey if you want to set the currently selected position as end point.

- OR of the element.
- OR -

Press the "Element center" softkey to place the contour end at the end of the element.

- OR -

Press the "Cursor" softkey to define the start of the element with the cursor at any position.

Transferring the contour to the cycle and to the program


## Press the "OK" softkey.

The selected contour is transferred to the contour input screen of the editor.
Press the "Accept" softkey.
The program record is transferred to the program.

## Operation with mouse and keyboard

In addition to operation using softkeys, you can also operate the functions with the keyboard and with the mouse.
5.16 Working with DXF files

## Simulating machining

## 6.1 <br> Overview

During simulation, the current program is calculated in its entirety and the result displayed in graphic form. The result of programming is verified without traversing the machine axes. Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece prevented.

## Graphic display

The simulation uses the correct proportions of the workpiece, tools, chuck, counterspindle and tailstock for the screen display.
For the spindle chuck and the tailstock, dimensions are used that are entered into the "Spindle Chuck Data" window.
For non-cylindrical blanks, the chuck closes up to the contour of the cube or polygon.

## Depth display

The depth infeed is color-coded. The depth display indicates the actual depth at which machining is currently taking place. The following applies to the color-coded depth display: "The deeper, the darker".

## Definition of blank

The blank dimensions that are entered in the program editor are used for the workpiece.
The blank is clamped with reference to the coordinate system, which is valid at the time that the blank was defined. This means that before defining the blank in $G$ code programs, the required output conditions must be established, e.g. by selecting a suitable zero offset.

## Programming a blank (example)

```
G54 G17 G90
WORKPIECE(, , ,"Cylinder",112.0,-50,-80.00,155,100)
T="NC-SPOTDRILL_D16
```


## Machine references

The simulation is implemented as workpiece simulation. This means that it is not assumed that the zero offset has already been precisely scratched or is known. In spite of this, unavoidable Machine references are in the programming, such as for example, the tool change point in the Machine, the park position for the counterspindle in the Machine or the position of the counterspindle slide. Depending on the actual zero offset - in the worst case these machine references can mean that collisions are shown in the simulation that would not occur for a realistic zero offset - or vice versa, collisions are not shown, which could occur for a realistic zero offset. This is the reason that in ShopTurn programs, in the case of a simulation, the program header calculates an appropriate zero offset for the main spindle or where relevant for the counterspindle - from the specified chuck dimensions.

## Programmable frames

All frames and zero offsets are taken into account in the simulation.

## Note

## Manually swiveled axes

Note that swivel movement in simulation and during simultaneous recording is also displayed when the axes are swiveled manually at the start.

## Display of the traversing paths

The traversing paths of the tool are shown in color. Rapid traverse is red and the feedrate is green.

## Note

## Displaying the tailstock

The tailstock is only visible with the option "ShopMill/ShopTurn".

## Machine manufacturer

Please also refer to the machine manufacturer's specifications.

## References

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

## Simulation display

You can choose one of the following types of display:

- Material removal simulation

During simulation or simultaneous recording you can follow stock removal from the defined blank.

- Path display

You have the option of including the display of the path. The programmed tool path is displayed.

## Note

## Tool display in the simulation and for simultaneous recording

In order that workpiece simulation is also possible for tools that have either not been measured or have been incompletely entered, certain assumptions are made regarding the tool geometry.

For instance, the length of a miller or drill is set to a value proportional to the tool radius so that cutting can be simulated.

## Note

Thread turns are not displayed
For thread and drill and thread milling, the thread turns are not displayed in the simulation and for simultaneous recording.

## Display variants

You can choose between three variants of graphical display:

- Simulation before machining of the workpiece

Before machining the workpiece on the machine, you can perform a quick run-through in order to graphically display how the program will be executed.

- Simultaneous recording before machining of the workpiece

Before machining the workpiece on the machine, you can graphically display how the program will be executed during the program test and dry run feedrate. The machine axes do not move if you have selected "no axis motion".

- Simultaneous recording during machining of the workpiece

You can follow machining of the workpiece on the screen while the program is being executed on the machine.

## Views

The following views are available for all three variants:

- Side view
- Half section
- Front view
- 3D view
- 2-window


## Status display

The current axis coordinates, the override, the current tool with cutting edge, the current program block, the feedrate and the machining time are displayed.

In all views, a clock is displayed during graphical processing. The machining time is displayed in hours, minutes and seconds. It is approximately equal to the time that the program requires for processing including the tool change.


## Software options

You require the option "3D simulation of the finished part" for the 3D view. You require the option "Simultaneous recording (real-time simulation)" for the "Simultaneous recording" function.

## Determining the program runtime

The program runtime is determined when executing the simulation. The program runtime is temporarily displayed in the editor at the end of the program.

## Properties of simultaneous recording and simulation

## Traversing paths

For the simulation, the displayed traversing paths are saved in a ring buffer. If this buffer is full, then the oldest traversing path is deleted with each new traversing path.

## Optimum display

If simultaneous machining is stopped or has been completed, then the display is again converted into a high-resolution screen. In some cases this is not possible. In this case, the following message is output: "High-resolution image cannot be generated".

## Working zone limitation

No working zone limits and software limit switches are effective in the tool simulation.

## Start position for simulation and simultaneous recording

During simulation, the start position is converted via the zero offset to the workpiece coordinate system.

The simultaneous recording starts at the position at which the machine is currently located.

## Constraint

- Referencing: G74 from a program run does not function.
- Alarm 15110 "REORG block not possible" is not displayed.
- Compile cycles are only partly supported.
- No PLC support.
- Axis containers are not supported.
- Swivel tables with non-swiveling offset vectors are not supported.


## Supplementary conditions

- All of the existing data records (toolcarrier / TRAORI, TRANSMIT, TRACYL) are evaluated and must be correctly commissioned for correct simulation.
- Transformations with swiveled linear axis (TRAORI 64-69) as well as OEM transformations (TRAORI 4096-4098) are not supported.
- Changes to the toolcarrier or transformation data only become effective after Power On.
- Transformation change and swivel data record change are supported. However, a real kinematic change is not supported, where a swivel head is physically changed.
- The simulation of mold making programs with extremely short block change times can take longer than machining, as the computation time distribution for this application is dimensioned in favor of the machining and to the detriment of simulation.


## Example

An example for supported kinematics is a lathe with $B$ axis:


Lathe with $B$ axis

## See also

Spindle chuck data (Page 100)

### 6.2 Simulation before machining of the workpiece

Before machining the workpiece on the machine, you have the option of performing a quick run-through in order to graphically display how the program will be executed. This provides a simple way of checking the result of the programming.

## Feedrate override

The rotary switch (override) on the control panel only influences the functions of the "Machine" operating area.
Press the "Program control" softkey to change the simulation feedrate. You can select the simulation feedrate in the range of $0-120 \%$.

## See also

Changing the feedrate (Page 208)
Simulating the program block by block (Page 209)

## Procedure



- OR -

Double-click the required program.
The selected program is opened in the "Program" operating area in the editor.
4. Press the "Simulation" softkey.

The program execution is displayed graphically on the screen. The machine axes do not move.
5. Press the "Stop" softkey if you wish to stop the simulation.

- OR -

Press the "Reset" softkey to cancel the simulation.
6. Press the "Start" softkey to restart or continue the simulation.

## Note

## Operating area switchover

The simulation is exited if you switch into another operating area. If you restart the simulation, then this starts again at the beginning of the program.


## Software option

You require the option "3D simulation of the finished part" for the 3D view.

## 6.3 <br> Simultaneous recording before machining of the workpiece

Before machining the workpiece on the machine, you can graphically display the execution of the program on the screen to monitor the result of the programming.
You can replace the programmed feedrate with a dry run feedrate to influence the speed of execution and select the program test to disable axis motion.

If you would like to view the current program blocks again instead of the graphical display, you can switch to the program view.


## Software option

You require the option "Simultaneous recording (real-time simulation)" for the simultaneous recording.

## Procedure

1. Load a program in the "AUTO" mode.
2. Press the "Prog. ctrl." softkey and activate the checkboxes "PRT no axis movement" and "DRY run feedrate".

The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate.
3. Press the "Sim. rec." softkey.
4. Press the <CYCLE START> key.

The program execution is displayed graphically on the screen.
5. Press the "Sim. rec." softkey again to stop the recording.

### 6.4 Simultaneous recording during machining of the workpiece

If the view of the work space is blocked by coolant, for example, while the workpiece is being machined, you can also track the program execution on the screen.


## Software option

You require the option "Simultaneous recording (real-time simulation)" for the simultaneous recording.

## Procedure

1. Load a program in the "AUTO" mode.
d Simult. $=$ record.
```
\)
CYCLE
```

Simult. record.
2. Press the "Sim. rec." softkey.
3. Press the <CYCLE START> key.

The machining of the workpiece is started and graphically displayed on the screen.
4. Press the "Sim. rec." softkey again to stop the recording.

## Note

- If you switch-on simultaneous recording after the unmachined part information has already been processed in the program, only traversing paths and tool are displayed.
- If you switch-off simultaneous recording during machining and then switch-on the function again at a later time, then the traversing paths generated in the intermediate time will not be displayed.


### 6.5 Different views of the workpiece

In the graphical display, you can choose between different views so that you constantly have the best view of the current workpiece machining, or in order to display details or the overall view of the finished workpiece.

The following views are available:

- Side view
- Half section
- Front view
- 3D view
- 2-window


### 6.5.1 Side view

## Side view

1. Start the simulation.

The side view shows the workpiece in the Z-X plane

## Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

### 6.5.2 Half section

Further
views

Half cut
view

1. Start the simulation.
2. Press the "Other views" and "Half section" softkeys. The half view shows the workpiece cut in the Z-X plane.

## Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

### 6.5.3 Face view

Further views

Face
view

1. Start the simulation.
2. Press the "Other views" and "Face view" softkeys.

Changing the display
You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

### 6.5.4 3D view

1. Start the simulation.


## Software option

$\square$
You require the option "3D simulation (finished part)" for the simulation.

## Changing the display

You can increase or decrease the size of the simulation graphic, move it, turn it, or change the segment.

Displaying and moving cutting planes
You can display and move cutting planes $\mathrm{X}, \mathrm{Y}$, and Z .

## See also

Defining cutting planes (Page 213)

### 6.5.5 2-window

2 windows

1. Start the simulation.
2. Press the "Additional views" and "2-window view" softkeys.

## Changing the display

You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

### 6.6 Graphical display



Figure 6-1 2-window view

## Active window

The currently active window has a lighter background than the other view windows.
Switch over the active window using the <Next Window> key.
You can change the workpiece display here, e.g. increase or decrease the size, turn it and move it.

Some of the actions that you perform in the active window also have a simultaneous effect in other view windows.

## Display of the traversing paths

- Rapid traverse = red
- Feed = green


### 6.7 Editing the simulation display

### 6.7.1 Blank display

You have the option of replacing the blank defined in the program or to define a blank for programs in which a blank definition cannot be inserted.

## Note

The unmachined part can only be entered if simulation or simultaneous recording is in the reset state.

| Parameter | Description | Unit |
| :---: | :---: | :---: |
| Main spindle |  |  |
| Mirroring Z | Mirroring of the $Z$ axis - (only for "data for counterspindle") <br> - Yes <br> Mirroring is used when machining on the Z axis <br> - No <br> Mirroring is not used when machining on the $Z$ axis |  |
| Blank U | Selecting the blank <br> - Centered cuboid <br> - Tube <br> - Cylinder <br> - Polygon <br> - without |  |
| Work offset | Selecting the work offset |  |
| XA | Outside diameter $\varnothing$ - (only for tube and cylinder) | mm |
| $\begin{aligned} & \mathrm{XI} \\ & \mathrm{U} \end{aligned}$ | Inside diameter (abs) or wall thickness (inc) - (only for tube) | mm |
| W | Width of the blank - (only for centered cuboid) | mm |
| L | Length of the blank - (only for centered cuboid) | mm |
| N | Number of edges - (only for polygon) |  |
| SW or L U | Width across flats or edge length - (only for polygon) | mm |
| ZA | Initial dimension |  |
| $\begin{gathered} \mathrm{ZI} \\ \mathrm{U} \\ \hline \end{gathered}$ | Final dimension (abs) or final dimension in relation to ZA (inc) |  |
| $\begin{aligned} & \mathrm{ZB} \\ & \mathrm{U} \end{aligned}$ | Machining dimension (abs) or machining dimension in relation to ZA (inc) |  |


| Parameter | Description | Unit |
| :---: | :---: | :---: |
| Counterspindle |  |  |
| Mirroring Z | - Yes <br> Mirroring is used when machining on the $Z$ axis <br> - No <br> Mirroring is not used when machining on the $Z$ axis |  |
| Blank <br> U | Selecting the blank <br> - Centered cuboid <br> - Tube <br> - Cylinder <br> - Polygon <br> - without |  |
| XA | Outside diameter $\varnothing$ - (only for tube and cylinder) |  |
| $\begin{array}{\|l\|} \hline X I \\ U \\ \hline \end{array}$ | Inside diameter (abs) or wall thickness (inc) - (only for tube) | mm |
| W | Width of the blank - (only for centered cuboid) |  |
| N | Number of edges - (only for polygon) |  |
| L | Length of the blank - (only for centered cuboid) | mm |
| SW or L $U$ | Width across flats or edge length - (only for polygon) | mm |
| ZI | Blank length (inc) | mm |
| ZB | Machining dimension (ink) | mm |

## Procedure

1. The simulation or the simultaneous recording is started.
2. Press the ">>" and "Blank" softkeys.

The "Blank Input" windows opens and displays the pre-assigned values.
Blank
3. Enter the desired values for the dimensions.
4. Press the "Accept" softkey to confirm your entries. The newly defined workpiece is displayed.

### 6.7.2 Showing and hiding the tool path

The path display follows the programmed tool path of the selected program. The path is continuously updated as a function of the tool movement. The tool paths can be shown or hidden as required.

## Procedure



Show tool path

Delete tool path

1. The simulation or the simultaneous recording is started.
2. Press the ">>" softkey.

The tool paths are displayed in the active view.
3. Press the softkey to hide the tool paths.

The tool paths are still generated in the background and can be shown again by pressing the softkey again.
4. Press the "'Delete tool path" softkey.

All of the tool paths recorded up until now are deleted.

### 6.8 Program control during the simulation

### 6.8.1 Changing the feedrate

You can change the feedrate at any time during the simulation.
You can track the changes in the status line.

## Note

If you are working with the "Simultaneous recording" function, the rotary switch (override) on the control panel is used.

## Procedure

## Program

 controlOverride
Override
-

```
100\% override
```

1. Simulation is started.
2. Press the "Program control" softkey.
3. Press the "Override + " or "Override -" softkey to increase or decrease the feedrate by $5 \%$.

- OR -

Press the "Override $100 \%$ " softkey to set the feedrate to $100 \%$.

- OR -

Press the "<<" softkey to return to the main screen and perform the simulation with changed feedrate.

## Toggling between "Override +" and "Override -"

$\square$ Press the <CTRL> and <Cursor down> or <Cursor up> keys to toggle
CTRL $\quad+\quad$ between the "Override + " and "Override - " softkeys.


Selecting the maximum feedrate


### 6.8.2 Simulating the program block by block

You can control the program execution during simulation, i.e. execute a program block by block, as when executing a program.

## Procedure

1. Simulation is started.

## Program

control
Single
block
<<
Back
(1) SBL

## (1) SBL

Program
control
Single
block
3. Press the "Back" and "Start SBL" softkeys.

The pending block of the program is simulated and then stops.
4. Press "Start SBL" as many times as you want to simulate a single program block.
5. Press the "Program control" and the "Single block" softkeys to exist the single block mode.

## Note

## Enabling/disabling single block



### 6.9 Editing and adapting a simulation graphic

### 6.9.1 Enlarging or reducing the graphical representation

## Precondition

The simulation or the simultaneous recording is started.

## Procedure


...


## Details

Zoom +

## Details

## Zoom -

## Details

Autozoom

1. Press the <+> and <-> keys if you wish to enlarge or reduce the graphic display.
The graphic display enlarged or reduced from the center.

- OR -

Press the "Details" and "Zoom +" softkeys if you wish to increase the size of the segment.

- OR -

Press the "Details" and "Zoom -" softkeys if you wish to decrease the size of the segment.

- OR -

Press the "Details" and "Auto zoom" softkeys if you wish to automatically adapt the segment to the size of the window.
The automatic scaling function "Fit to size" takes account of the largest expansion of the workpiece in the individual axes.

## Note

## Selected section

The selected sections and size changes are kept as long as the program is selected.

### 6.9.2 Panning a graphical representation

## Precondition

The simulation or the simultaneous recording is started.

## Procedure



### 6.9.3 Rotating the graphical representation

In the 3D view you can rotate the position of the workpiece to view it from all sides.

## Requirement

 Simulation has been started and the 3D view is selected.
## Procedure

## Details

Rotate view


```
    \uparrow
``` - OR -

Turning


\subsection*{6.9.4 Modifying the viewport}

If you would like to move, enlarge or decrease the size of the segment of the graphical display, e.g. to view details or display the complete workpiece, use the magnifying glass.
Using the magnifying glass, you can define your own section and then enlarge or reduce its size.

\section*{Precondition}

The simulation or the simultaneous recording is started.

\section*{Procedure}

\section*{Details}

Zoom

Zoom +
3. Press the "Magnify + " or <+> softkey to enlarge the frame.
- OR -

Press the "Magnify -" or <-> softkey to reduce the frame.
- OR -


Press one of the cursor keys to move the frame up, down, left or right.

4. Press the "Accept" softkey to accept the selected section.

\subsection*{6.9.5 Defining cutting planes}

In the 3D view, you have the option of "cutting" the workpiece and therefore displaying certain views in order to show hidden contours.

\section*{Precondition}

The simulation or the simultaneous recording is started.

\section*{Procedure}
```

    Details
    ```
```

    Cut
    ```
    Gut
    active

X +
... \(2-\)
1. Press the "Details" softkey.
2. Press the "Cut" softkey.

The workpiece is displayed in the cut state.
3. Press the corresponding softkey to shift the cutting plane in the required direction.

\subsection*{6.10 Displaying simulation alarms}

Alarms might occur during simulation. If an alarm occurs during a simulation run, a window opens in the operating window to display it.

The alarm overview contains the following information:
- Date and time
- Deletion criterion

Specifies with which softkey the alarm is acknowledged
- Alarm number
- Alarm text

Precondition
Simulation is running and an alarm is active.

\section*{Procedure}

\section*{Program control}

Alarm

\section*{Acknowl.} alarm
1. Press the "Program control" and "Alarm" softkeys.

The "Simulation Alarms" window is opened and a list of all pending alarms is displayed.

Press the "Acknowledge alarm" softkey to reset the simulation alarms indicated by the Reset or Cancel symbol.
The simulation can be continued.
- OR -

Press the "Simulation Power On" softkey to reset a simulation alarm indicated by the Power On symbol.

\section*{Creating a G code program}

\subsection*{7.1 Graphical programming}

\section*{Functions}

The following functionality is available:
- Technology-oriented program step selection (cycles) using softkeys
- Input windows for parameter assignment with animated help screens
- Context-sensitive online help for every input window
- Support with contour input (geometry processor)

\section*{Call and return conditions}
- The G functions active before the cycle call and the programmable frame remain active beyond the cycle.
- The starting position must be approached in the higher-level program before the cycle is called. The coordinates are programmed in a clockwise coordinate system.

\subsection*{7.2 Program views}

You can display a \(G\) code program in various ways.
- Program view
- Parameter screen, either with help screen or graphic view

\section*{Note}

Help screens / animations
Please note that not all the conceivable kinematics can be displayed in help screens and animations of the cyclic support.

\section*{Program view}

The program view in the editor provides an overview of the individual machining steps of a program.


Figure 7-1 Program view of a G code program

\section*{Note}

In the program editor settings you define as to whether cycle calls are to be displayed as plain text or in NC syntax.


In the program view, you can move between the program blocks by pressing the <Cursor up> and <Cursor down> keys.


\section*{Parameter screen with help display}


Press the <Cursor right> key to open a selected program block or cycle in the program view.
The associated parameter screen with help display is then displayed.

\section*{Note}

Switching between the help screen and the graphic view
The key combination <CTRL> + <G> is also available for the switchover between the help screen and the graphic view.


Figure 7-2 Parameter screen with help display
The animated help displays are always displayed with the correct orientation to the selected coordinate system. The parameters are dynamically displayed in the graphic. The selected parameter is displayed highlighted in the graphic.

\section*{The color symbols}

Red arrow = tool traverses in rapid traverse
Green arrow = tool traverses with the machining feedrate

\section*{Parameter screen with graphic view}

> Graphic view \(\quad \begin{aligned} & \text { Press the "Graphic view" softkey to toggle between the help screen and } \\ & \text { the graphic view in the screen. }\end{aligned}\)


Figure 7-3 Parameter screen with a graphical view of a G code program block

\subsection*{7.3 Program structure}

G_code programs can always be freely programmed. The most important commands that are included in the rule:
- Set a machining plane
- Call a tool (T and D)
- Call a work offset
- Technology values such as feedrate (F), feedrate type (G94, G95,....), speed and direction of rotation of the spindle ( S and M )
- Positions and calls, technology functions (cycles)
- End of program

For G code programs, before calling cycles, a tool must be selected and the required technology values F , S programmed.
A blank can be specified for simulation.

\section*{See also}

\subsection*{7.4 Fundamentals}

\subsection*{7.4.1 Machining planes}

A plane is defined by means of two coordinate axes. The third coordinate axis (tool axis) is perpendicular to this plane and determines the infeed direction of the tool (e.g. for \(21 / 2-\mathrm{D}\) machining).

When programming, it is necessary to specify the working plane so that the control system can calculate the tool offset values correctly. The plane is also relevant to certain types of circular programming and polar coordinates.


\section*{Working planes}

Working planes are defined as follows:
\begin{tabular}{lll} 
Plane & & Tool axis \\
X/Y & G17 & Z \\
Z/X & G18 & Y \\
Y/Z & G19 & X
\end{tabular}

\subsection*{7.4.2 Current planes in cycles and input screens}

Each input screen has a selection box for the planes, if the planes have not been specified by NC machine data.
- Empty (for compatibility reasons to screen forms without plane)
- G17 (XY)
- G18 (ZX)
- G19 (YZ)

There are parameters in the cycle screens whose names depend on this plane setting. These are usually parameters that refer to positions of the axes, such as reference point of a position pattern in the plane or depth specification when drilling in the tool axis.

For G17, reference points in the plane are called X0 Y0, for G18 they are called Z0 X0 - and for G19, they are called Y0 Z0. The depth specification in the tool axis for G17 is called Z1, for G18, Y1 and for G19, X1.

If the entry field remains empty, the parameters, the help screens and the broken-line graphics are displayed in the default plane (can be set via machine data):
- Turning: G18 (ZX)
- Milling: G17 (XY)

The plane is transferred to the cycles as new parameter. The plane is output in the cycle, i.e. the cycle runs in the entered plane. It is also possible to leave the plane fields empty and thus create a plane-independent program.
The entered plane only applies for this cycle (not modal)! At the end of the cycle, the plane from the main program applies again. In this way, a new cycle can be inserted in a program without having to change the plane for the remaining program.

\subsection*{7.4.3 Programming a tool \((T)\)}

\section*{Calling a tool}
1. You are in a part program

Select
tool
2. Press the "Select tool" softkey.

The "Tool selection" window is opened.
To program
3. Position the cursor on the desired tool and press the "To program" softkey.
The selected tool is loaded into the G code editor. Text such as the following is displayed at the current cursor position in the G code editor: T="ROUGHINGTOOL100"
- OR -
4. Press the "Tool list" and "New tool" softkeys.

New tool program
5. Then select the required tool using the softkeys on the vertical softkey bar, parameterize it and then press the softkey "To program". The selected tool is loaded into the G code editor.
6. Then program the tool change (M6), the spindle direction (M3/M4), the spindle speed (S...), the feedrate (F), the feedrate type (G94, G95,...), the coolant (M7/M8) and, if required, further tool-specific functions.

\subsection*{7.5 Generating a G code program}

Create a separate program for each new workpiece that you would like to produce. The program contains the individual machining steps that must be performed to produce the workpiece.

Part programs in the G code can be created under the "Workpieces" folder or under the "Part programs" folder.

\section*{Procedure}
1. Select the "Program Manager" operating area.

\section*{Creating a new part program}

New
programGUIDE
f code

3. Position the cursor on the folder "Part programs" and press the "New" softkey.

The "New G Code Program" window opens.
4. Enter the required name and press the "OK" softkey.

The name can contain up to 28 characters (name + dot + 3-character extension). You can use any letters (except accented), digits or the underscore symbol (_).
The program type (MPF) is set by default.
The project is created and opened in the Editor.

\section*{Creating a new part program for a workpiece}

New
programGuIDE
f code

5. Position the cursor on the folder "Workpieces" and press the "New" softkey.

The "New G Code Program" window opens.
6. Select the file type (MPF or SPF), enter the desired name of the program and press the "OK" softkey.
The project is created and opened in the Editor.
7. Enter the desired G code commands.

\section*{See also}

\subsection*{7.6 Blank input}

\section*{Function}

The blank is used for the simulation and the simultaneous recording. A useful simulation can only be achieved with a blank that is as close as possible to the real blank.
Create a separate program for each new workpiece that you would like to produce. The program contains the individual machining steps that are performed to produce the workpiece.
For the blank of the workpiece, define the shape (tube, cylinder, polygon or centered cuboid) and your dimensions.

\section*{Manually reclamping the blank}

If the blank is to be manually reclamped from the main spindle to the counterspindle for example, then delete the blank.

\section*{Example}
- Blank, main spindle, cylinder
- Machining
- MO : Manually reclamping the blank
- Blank, main spindle, delete
- Blank, counterspindle, cylinder
- Machining

The blank entry always refers to the work offset currently effective at the position in the program.

\section*{Note}

\section*{Swiveling}

For programs that use "Swiveling", a 0 swivel must first be made and then the blank defined.

\section*{Procedure}


Program

Uari-
ous
1. Select the "Program" operating area.
2. Press the "Misc." and "Blank" softkeys. The "Blank Input" window opens.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Data for & \begin{tabular}{l}
Selection of the spindle for the blank \\
- Main spindle \\
- Counterspindle \\
Note: \\
If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.
\end{tabular} & \\
\hline Mirroring Z & \begin{tabular}{l}
Mirroring of the \(Z\) axis - (only for "data for counterspindle") \\
- Yes \\
Mirroring is used when machining on the \(Z\) axis \\
- No \\
Mirroring is not used when machining on the \(Z\) axis
\end{tabular} & \\
\hline Blank
\[
0
\] & \begin{tabular}{l}
Selecting the blank \\
- Centered cuboid \\
- Tube \\
- Cylinder \\
- Polygon \\
- Delete
\end{tabular} & \\
\hline ZA & Initial dimension & mm \\
\hline ZIU & Final dimension (abs) or final dimension in relation to ZA (inc) & mm \\
\hline ZB U & Machining dimension (abs) or machining dimension in relation to ZA (inc) & mm \\
\hline Spindle chuck data & \begin{tabular}{l}
- Yes \\
You enter spindle chuck data in the program. \\
- No \\
Spindle chuck data are transferred from the setting data. \\
Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline Spindle chuck data & \begin{tabular}{l}
- Only chuck \\
You enter spindle chuck data in the program. \\
- Complete \\
You enter tailstock data in the program. \\
Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline Jaw type & \begin{tabular}{l}
Selecting the jaw type of the counterspindle. Dimensions of the front edge or stop edge (only if spindle chuck data "yes") \\
- Jaw type 1 \\
- Jaw type 2
\end{tabular} & \\
\hline ZC4 & The main spindle chuck dimensions - (only for spindle chuck data "yes") & mm \\
\hline ZS4 & Stop dimension of the main spindle - (only for spindle chuck data "yes") & mm \\
\hline ZE4 & Jaw dimension of the main spindle for jaw type 2 - (only for spindle chuck data "yes") & mm \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline ZC3 & \begin{tabular}{l} 
Chuck dimension of the counterspindle - (only for spindle chuck data "yes" and for a \\
counterspindle that has been set up)
\end{tabular} & mm \\
\hline ZS3 & \begin{tabular}{l} 
Stop dimension of the counterspindle - (only for spindle chuck data "yes" and for a coun- \\
terspindle that has been set up)
\end{tabular} & mm \\
\hline ZE3 & \begin{tabular}{l} 
Jaw dimension of the counterspindle with jaw type 2 - (only for spindle chuck data "yes" \\
and for a counterspindle that has been set up)
\end{tabular} & mm \\
\hline XR3 & \begin{tabular}{l} 
Tailstock diameter - (only for spindle chuck data "complete" and tailstock that has been \\
set up)
\end{tabular} & mm \\
\hline ZR3 & \begin{tabular}{l} 
Tailstock length - (only for spindle chuck data "complete" and tailstock that has been set \\
up)
\end{tabular} & mm \\
\hline XA & Outside diameter - (only for tube and cylinder) & mm \\
\hline XI U & Inside diameter (abs) or wall thickness (inc) - (only for tube) & mm \\
\hline N & Number of edges - (only for polygon) & mm \\
\hline SW or L U & Width across flats or edge length - (only for polygon) & mm \\
\hline W & Width of the blank - (only for centered cuboid) & mm \\
\hline L & Length of the blank - (only for centered cuboid) & \\
\hline
\end{tabular}

\subsection*{7.7 Machining plane, milling direction, retraction plane, safe clearance and feedrate (PL, RP, SC, F)}

In the program header, cycle input screens have general parameters that always repeat.
You will find the following parameters in every input screen for a cycle in a G code program.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline PL & \begin{tabular}{l} 
Each input screen has a selection box for the planes, if the planes have not been speci- \\
fied by NC machine data. \\
Machining plane: \\
- G17 (XY) \\
- G18 (ZX) \\
- G19 (YZ)
\end{tabular} & \\
\hline \begin{tabular}{l} 
Milling direction \\
U - only for mill- \\
ing
\end{tabular} & \begin{tabular}{l} 
When machining a pocket, a longitudinal slot or a spigot, the machining direction (climb- \\
ing or conventional) and the spindle direction are taken into account in the tool list. The \\
pocket is then machined in a clockwise or counter-clockwise direction. \\
During path milling, the programmed contour direction determines the machining direc- \\
tion.
\end{tabular} & \\
\hline RP & \begin{tabular}{l} 
Retraction plane (abs) \\
During machining the tool traverses in rapid traverse from the tool change point to the \\
return plane and then to the safety clearance. The machining feedrate is activated at \\
this level. When the machining operation is finished, the tool traverse st the machining \\
feedrate away from the workpiece to the safety clearance level. It traverses from the \\
safety clearance to the retraction plane and then to the tool change point in rapid trav- \\
erse. \\
The retraction plane is entered as an absolute value. \\
Normally, reference point zo and retraction plane RP have different values. The cycle \\
assumes that the retraction plane is in front of the reference point.
\end{tabular} & mm \\
\hline \begin{tabular}{l} 
Safety clearance (inc) \\
The safety clearance specifies from which clearance to the material rapid traverse is no \\
longer used. \\
The direction in which the safety clearance is active is automatically determined by the \\
cycle. Generally, it is effective in several directions. The safety clearance must be en- \\
tered as an incremental value (without sign).
\end{tabular} & mm \\
\hline U & \begin{tabular}{l} 
Feedrate \\
The feedrate F (also referred to as the machining feedrate) specifies the speed at which \\
the axes move when machining the workpiece. The unit of the feedrate (mm/min, \\
mm/rev, mm/tooth etc. ) always refers to the feedrate type programmed before the cycle \\
call. \\
The maximum feedrate is determined via machine data.
\end{tabular} & \\
\hline F
\end{tabular}

\subsection*{7.8 Selection of the cycles via softkey}

\section*{Overview of the machining steps}

The following machining steps are available.
All of the cycles/functions available in the control are shown in this display. However, at a specific system, only the steps possible corresponding to the selected technology can be selected.



Contour
call
Stock
remoual

Cut resid
stock

Groouing

Groove
resid.

Part

Part
resid.

\begin{tabular}{|c|c|c|}
\hline Pocket & \(\Rightarrow\) & \begin{tabular}{c} 
Rectang. \\
pocket
\end{tabular} \\
& \begin{tabular}{c} 
Circular \\
pocket
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|}
\hline \begin{tabular}{c} 
Multi- edge \\
spigot
\end{tabular} & \\
& \begin{tabular}{c} 
Rectang. \\
spigot
\end{tabular} \\
\hline & \begin{tabular}{c} 
Circular \\
spigot
\end{tabular} \\
& \begin{tabular}{c} 
Multi- \\
edge
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Groove & \(\Rightarrow\) & \begin{tabular}{c} 
Longit. \\
groove
\end{tabular} \\
\hline & \begin{tabular}{c} 
Circular \\
groove
\end{tabular} \\
\hline
\end{tabular}
```

Thread
milling

```

Engraving


\section*{Pocket}

Pocket res.mat.

Spigot

Spigot
res. mat.


Blank


Orient
milling tool



A menu tree with all of the available measuring versions of the measuring cycle function "Measure workpiece" can be found in the following reference:
Programming Manual Measuring cycles / SINUMERIK 840D sI/828D
A menu tree with all of the available measuring versions of the measuring cycle function "Measure tool" can be found in the following reference:
Programming Manual Measuring cycles / SINUMERIK 840D sI/828D

\section*{See also}

General (Page 293)
Generating a G code program (Page 221)

\subsection*{7.9 Calling technology cycles}

\subsection*{7.9.1 Hiding cycle parameters}

The documentation describes all the possible input parameters for each cycle. Depending on the settings of the machine manufacturer, certain parameters can be hidden in the screens, i.e. not displayed. These are then generated with the appropriate default values when the cycles are called.

For additional information, please refer to the following references:
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

\section*{Cycle support}
\begin{tabular}{ll} 
Example & Use the softkeys to select whether you want support for programming \\
contours, turning, drilling or milling cycles.
\end{tabular}

\subsection*{7.9.2 Setting data for cycles}

Cycle functions can be influenced and configured using machine and setting data.
For additional information, please refer to the following references:
Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

\subsection*{7.9.3 Checking cycle parameters}

The entered parameters are already checked during the program creation in order to avoid faulty entries.
If a parameter is assigned an illegal value, this is indicated in the input screen and is designated as follows:
- The entry field has a colored background (background color, pink).
- A note is displayed in the comment line.
- If the parameter input field is selected using the cursor, the not is also displayed as tooltip.

The programming can only be completed after the incorrect value has been corrected.
Faulty parameter values are also monitored with alarms during the cycle runtime.

\subsection*{7.9.4 Programming variables}

In principle, variables or expressions can also be used in the input fields of the screen forms instead of specific numeric values. In this way, programs can be created very flexibly.

\section*{Input of variables}

Please note the following points when using variables:
- Values of variables and expressions are not checked since the values are not known at the time of programming.
- Variables and expressions cannot be used in fields in which a text is expected (e.g. tool name).

An exception is the "Engraving" function, in which you can assign the desired text in the text field via a variable as "Variable text".
- Selection fields generally cannot be programmed with variables.

\section*{Examples}

VAR_A
VAR_A+2*VAR_B
SIN(VAR_C)

\subsection*{7.9.5 Changing a cycle call}

You have called the desired cycle via softkey in the program editor, entered the parameters and confirmed with "Accept".

\section*{Procedure}

1. Select the desired cycle call and press the <Cursor right> key. The associated input screen of the selected cycle call is opened.
- OR -


Press the <SHIFT + INSERT> key combination.
This starts the edit mode for this cycle call and you can edit it like a normal NC block. This means that it is possible to generate an empty block before the cycle is called. For instance, to insert something before a cycle that is located at the beginning of the program.
Note: In edit mode, the cycle call can be changed in such a way that it can no longer be recompiled in the parameter screen.


You exit the edit mode by pressing the <SHIFT + INSERT> key combination.
- OR -

You are in the edit mode and press the <INPUT> key. A new line is created after the cursor position.

\section*{See also}

Generating a G code program (Page 221)

\subsection*{7.9.6 Compatibility for cycle support}

The cycle support is generally upwards compatible. This means that cycle calls in NC programs can always be recompiled with a higher software version, changed and then run again.
When transferring NC programs to a machine with a lower software version, it cannot be guaranteed, however, that the program will be able to be changed by recompiling cycle calls.

\subsection*{7.9.7 \(\quad\) Additional functions in the input screens}

\section*{Selection of units}

U If, for example, the unit can be switched in a field, this is highlighted as soon as the cursor is positioned on the element. In this way, the operator recognizes the dependency.
The selection symbol is also displayed in the tooltip.

\section*{Display of abs or inc}

The abbreviations "abs" and "inc" for absolute and incremental values are displayed behind the entry fields when a switchover is possible for the field.

\section*{Help screens}

2D and 3D graphics or sectional views are displayed for the parameterization of the cycles.

\section*{Online help}

If you wish to obtain more detailed information about certain \(G\) code commands or cycle parameters, then you can call a context-sensitive online help.

\subsection*{7.10 Measuring cycle support}

Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be adapted to specific problems via parameter settings.


\section*{Software option}

You require the "Measuring cycles" option to use "Measuring cycles".

\section*{References}

You will find a more detailed description on how to use measuring cycles in:
Programming Manual Measuring cycles / SINUMERIK 840D sI/828D

\section*{Creating a ShopTurn program}

\subsection*{8.1 Graphic program control, ShopTurn programs}

The program editor offers graphic programming to generate machining step programs that you can directly generate at the machine.


\section*{Software option}

You require the "ShopMill/ShopTurn" option to generate ShopTurn machining step programs.

\section*{Functions}

The following functionality is available:
- Technology-oriented program step selection (cycles) using softkeys
- Input windows for parameter assignment with animated help screens
- Context-sensitive online help for every input window
- Support with contour input (geometry processor)

\subsection*{8.2 Program views}

You can display a ShopTurn program in various views:
- Work plan
- Graphic view
- Parameter screen, either with help screen or graphic view

\section*{Note}

Help screens / animations
Please note that not all the conceivable kinematics can be displayed in help screens and animations of the cyclic support.

\section*{Work plan}

The work plan in the editor provides an overview of the individual machining steps of a program.


Figure 8-1 Machining schedule of a ShopTurn program

2. Press the ">>" and "Graphic view" softkeys to display the graphic view.

\section*{Graphic view}

\section*{Note}

\section*{Switching between the help screen and the graphic view}

The key combination <CTRL>+ <G> is also available for the switchover between the help screen and the graphic view.

\section*{Graphic view}

The graphic view shows the contour of the workpiece as a dynamic graphic with broken lines. The program block selected in the work plan is highlighted in color in the graphic view.


Figure 8-2 Graphic view of a ShopTurn program

\section*{Parameter screen with help display and graphic view}


Graphic view
1. Press the <Cursor right> key to open a selected program block or cycle in the work plan.
The associated parameter screen with help display is then displayed.
2. Press the "Graphic view" softkey.

The graphic view of the selected program block is displayed.

\section*{Note}

\section*{Switching between the help screen and the graphic view}

The key combination <CTRL> + <G> is also available for the switchover between the help screen and the graphic view.


Figure 8-3 Parameter screen with dynamic help display
The animated help displays are always displayed with the correct orientation to the selected coordinate system. The parameters are dynamically displayed in the graphic. The selected parameter is displayed highlighted in the graphic.
```

Graphic view

```

Press the "Graphic view" softkey to toggle between the help display and the graphic view in the screen.

\section*{Note}

\section*{Switching between the help screen and the graphic view}

The key combination <CTRL> + <G> is also available for the switchover between the help screen and the graphic view.


Figure 8-4 Parameter screen with graphic view

\subsection*{8.3 Program structure}

A machining step program is divided into three sub-areas:
- Program header
- Program blocks
- End of program

These sub-areas form a process plan.

\section*{Program header}

The program header contains parameters that affect the entire program, such as blank dimensions or retraction planes.

\section*{Program blocks}

You determine the individual machining steps in the program blocks. In doing this, you specify the technology data and positions, among other things.

\section*{Linked blocks}

For the "Contour turning", "Contour milling", "Milling", and "Drilling" functions, program the technology blocks and contours or positioning blocks separately. These program blocks are automatically linked by the control and connected by brackets in the process plan.
In the technology blocks, specify how and in what form the machining should take place, e.g. centering first, and then drilling. In the positioning blocks, determine the positions for the drilling or milling machining, e.g. position the drill-holes in a full circle on the face surface.

\section*{End of program}

End of program signals to the machine that the machining of the workpiece has ended.
Further, here you set whether program execute should be repeated.

\section*{Note}

Number of workpieces
You can enter the number of required workpieces using the "Times, counters" window.

\section*{See also}

Entering the number of workpieces (Page 258)

\subsection*{8.4 Fundamentals}

\subsection*{8.4.1 Machining planes}

A workpiece can be machined on different planes. Two coordinate axes define a machining plane. On lathes with \(X, Z\), and \(C\) axes, three planes are available:
- Turning
- Face
- Peripheral

\section*{Machining planes, face and peripheral}

The face and peripheral machining planes require the CNC-ISO functions "Front surface machining" (Transmit) and "Cylindrical peripheral transformation" (Tracyl) to be set up.
These functions are a software option.

\section*{Additional Y axis}

For lathes with an additional Y axis, the machining planes are expanded to include two more planes:
- Face \(Y\)
- Peripheral Y

Therefore, the face and peripheral planes are called Face C and Peripheral C.

\section*{Inclined axis}

If the Y axis is an oblique axis (i.e. the axis is not perpendicular to the others), you can also select the "Face \(Y\) " and "Peripheral \(Y\) " machining planes and program the traversing movements in Cartesian coordinates. The control system then automatically transforms the programmed traversing movements of the Cartesian coordinate system into the traversing movements of the oblique axis.
The CNC-ISO function "Oblique Axis" (Traang) is required for the purpose of transforming the programmed traversing movements.

This function is a software option.

\section*{Selecting the machining plane}

The machining plane selection is integrated into the parameter screen forms of the individual drilling and milling cycles. For turning cycles and for "axial drilling" and "axial threads", the turning plane is automatically selected. For the "straight" and "circle" functions, you must specify the machining plane separately.

The settings for the machining plane always act modally, i.e. until you select another plane.
The machining planes are defined as follows:

\section*{Turning}

The turning machining plane corresponds to the \(\mathrm{X} / \mathrm{Z}\) plane (G18).

\section*{Face/Face C}

The Face/Face C machining plane corresponds to the X/Y plane (G17). For machines without a Y axis, however, the tools can only move in the \(\mathrm{X} / \mathrm{Z}\) plane. The \(\mathrm{X} / \mathrm{Y}\) coordinates that have been entered are automatically transformed into a movement in the \(X\) and \(C\) axis.

You can use face surface machining with a C axis for drilling and milling if, for instance, you want to mill a pocket on the face surface. You can choose between the forward or rear face surface for this purpose.

\section*{Peripheral/Peripheral C}

The Peripheral/Peripheral C machining plane corresponds to the Y/Z plane (G19). For machines without a \(Y\) axis, however, the tools can only move in the Z/X plane. The Y/Z coordinates that you entered are automatically transformed into a movement in the C and Z axis.

You can use peripheral surface machining with a C axis for drilling and milling if, for instance, you want to mill a slot with constant depth on the peripheral surface. You can choose between the inner or outer surface for this purpose.

\section*{Face \(Y\)}

The face \(Y\) machining plane corresponds to the \(X / Y\) plane (G17). You can use the face surface machining with a Y axis for drilling and milling if, for instance, you want to mill a pocket on the face surface. You can choose between the forward or rear face surface for this purpose.

\section*{Peripheral Y}

The peripheral Y machining plane corresponds to the \(\mathrm{Y} / \mathrm{Z}\) plane (G19). You can use peripheral surface machining with a Y axis for drilling and milling if, for instance, you want to mill a pocket with a flat base on the peripheral surface or drill holes that do not point to the center. You can choose between the inner or outer surface for this purpose.

\subsection*{8.4.2 Machining cycle, approach/retraction}

Approaching and retracting during the machining cycle always follows the same pattern if you have not defined a special approach/retraction cycle.
If your machine has a tailstock, you can also take this into consideration when traversing.
The retraction for a cycle ends at the safety clearance. Only the subsequent cycle moves to the retraction plane. This enables a special approach/retraction cycle to be used.

\section*{Note}

When selecting the traversing paths, the tool tip is always taken into account; i.e. the tool expansion is not considered. Therefore, you should ensure that the retraction planes are an appropriate distance away from the workpiece.

\section*{Approach/retraction sequence in a machining cycle}


Figure 8-5 Machining cycle, approach/retraction
- The tool traverses in rapid traverse along the shortest path from the tool change point to the retraction plane, which runs parallel to the machining plane.
- After this, the tool traverses in rapid traverse to the safety clearance.
- Following this, the workpiece is then machined with the programmed machining feedrate.
- After machining, the tool retracts with rapid traverse to the safety clearance.
- The tool then continues to traverse vertically in rapid traverse to the retraction plane.
- From there, the tool traverses in rapid traverse along the shortest path to the tool change point. If the tool does not need to be changed between two machining processes, the tool traverses from the retraction plane to the next machining cycle.

The spindle (main, tool, or counter-spindle) begins to rotate immediately after the tool change.
You define the tool change point, the retraction plane, and the safety clearance in the program header.

\section*{Taking into account the tailstock}


Figure 8-6 Approach/retraction taking into account the tailstock
- The tool traverses in rapid traverse from the tool change point along the shortest path to the retraction plane XRR from the tailstock.
- After this, the tool traverses in rapid traverse on the retraction plane in the \(X\) direction.
- After this, the tool traverses in rapid traverse to the safety clearance.
- Following this, the workpiece is then machined with the programmed machining feedrate.
- After machining, the tool retracts with rapid traverse to the safety clearance.
- The tool then continues to traverse vertically in rapid traverse to the retraction plane.
- After this, the tool traverses in the X direction to the retraction plane XRR from the tailstock.
- From there, the tool traverses in rapid traverse along the shortest path to the tool change point. If the tool does not need to be changed between two machining processes, the tool traverses from the retraction plane to the next machining cycle.
You define the tool change point, the retraction plane, the safety clearance, and the retraction plane for the tailstock in the program header.

\section*{See also}
\[
\text { Programming the approach/retraction cycle (Page } \underline{270)}
\]

\footnotetext{
Program header (Page 250)
}

\subsection*{8.4.3 Absolute and incremental dimensions}

When generating a machining step program, you can input positions in absolute or incremental dimensions, depending on how the workpiece drawing is dimensioned.
You can also use a combination of absolute and incremental dimensions, i.e. one coordinate as an absolute dimension and the other as an incremental dimension.

For the face axis (the \(X\) axis, in this case), in the machine data it is established whether the diameter or radius is programmed in absolute or incremental dimensions.

Please refer to the machine manufacturer's specifications.

\section*{Absolute dimensions (ABS)}

With absolute dimensions, all position specifications refer to the zero point of the active coordinate system.


Figure 8-7 Absolute dimensions
The position specifications for the points P 1 to P 4 in absolute dimensions refer to the zero point:
P1: X25 Z-7.5
P2: X40 Z-15
P3: X40 Z-25
P4: X60 Z-35

\section*{Incremental dimensions (INC)}

With incremental dimensions (also referred to as sequential dimensions) a position specification refers to the previously programmed point, i.e. the input value corresponds to the path to be traversed. As a rule, the plus/minus sign does not matter when entering the incremental value, only the absolute value of the increment is evaluated. For some parameters, the plus/minus sign specifies the traversing direction. These exceptions are identified in the parameter table of the individual functions.


Figure 8-8 Incremental dimensions
The position specifications for points P1 to P 4 in incremental dimensions are as follows:
P1: X12.5 Z-7.5 (relative to the zero point)
P2: X7.5 Z-7.5 (relative to P1)
P3:X0 Z-10 (relative to P2)
P4: X10 Z-10 (relative to P3)

\subsection*{8.4.4 Polar coordinates}

You can specify positions using right-angled coordinates or polar coordinates.
If a point in a workpiece drawing is defined by a value for each coordinate axis, you can easily input the position into the parameter screen form using right-angled coordinates. For workpieces that are dimensioned with arcs or angular data, it is often easier if you input the positions using polar coordinates.
You can only program polar coordinates for the functions "Straight circle" and "Contour milling."
The point from which dimensioning starts in polar coordinates is called the "pole".


Figure 8-9 Polar coordinates
The position specifications for the pole and points P1 to P3 in polar coordinates are:
Pole: X30 Z30 (relative to the zero point)
P1: L30 \(\alpha 30^{\circ}\) (relative to the pole)
P2: L30 \(\alpha 60^{\circ}\) (relative to the pole)
P3: L30 \(\alpha 90^{\circ}\) (relative to the pole)

\subsection*{8.4.5 Clamping the spindle}

The "Clamp spindle" function must be set up by the machine manufacturer.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Note for selecting the clamp spindle function under ShopTurn}

The machine manufacturer also specifies whether ShopTurn will clamp the spindle automatically if this would facilitate machining, or if you can decide the types of machining for which the spindle should be clamped.
If you are to decide the types of machining for which the spindle is to be clamped, the following applies:
You should note that when machining in the end face/end face \(C\) and peripheral surface/peripheral surface \(C\) planes, clamping only remains active for contour milling and drilling operations. On the other hand, when machining in the end face Y/end face B and peripheral surface \(Y\) planes, clamping is modal, i.e. it remains active until the machining plane is changed.

\section*{8．5 Creating a ShopTurn program}

Create a separate program for each new workpiece that you would like to produce．The program contains the individual machining steps that must be performed to produce the workpiece．

If you create a new program，a program header and program end are automatically defined．
ShopTurn programs can be created in a new workpiece or under the folder＂Part programs＂．

\section*{Procedure}

\section*{Creating a ShopTurn program}

1．Select the＂Program Manager＂operating area．

\section*{霛正 NG}

\section*{New}

\section*{ShopTurn}


4．Enter the required name and press the＂OK＂softkey．
The name can contain up to 28 characters（name + dot +3 －character extension）．You can use any letters（except accented），digits or the underscore symbol（＿）．The＂ShopTurn＂program type is selected． The editor is opened and the＂Program header＂parameter screen is displayed．

\section*{Filling out the program header}


5．Select a work offset．

6．Enter the dimensions of the blank and the parameter，which are effec－ tive over the complete program，e．g．dimension units in mm or inch，tool axis，retraction plane，safety clearance and machining direction．
Press the＂Teach TC position＂softkey if you want to set the actual posi－ tion of the tool as a tool change point．
The tool＇s coordinates are transferred into parameters XT and ZT ．
Teaching in the tool change point is only possible if you have selected the machine coordinate system（Machine）．
7．Press the＂Accept＂softkey．
The work plan is displayed．Program header and end of program are created as program blocks．
The end of program is automatically defined．

The retraction for a cycle ends at the safety clearance. Only the subsequent cycle moves to the retraction plane. This enables a special approach/retraction cycle to be used.

Changes to the retraction plane therefore take effect when retracting from the previous machining operation.
When selecting the traversing paths, the tool tip is always taken into account; i.e. the tool expansion is not considered. Therefore, you should ensure that the retraction planes are an appropriate distance away from the workpiece. The retraction planes refer to the workpiece. As a consequence, they are not influenced by a programmable offset.

\section*{See also}

Changing program settings (Page 260)
Programming the approach/retraction cycle (Page 270)
Creating a new workpiece (Page 743)

\subsection*{8.6 Program header}

\subsection*{8.6 Program header}

In the program header, set the following parameters, which are effective for the complete program.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Measurement unit U & \begin{tabular}{l}
The setting of the measurement unit in the program header only refers to the position data in the actual program. \\
All other data, such as feedrate or tool offsets, are entered in the unit of measure that you have set for the entire machine.
\end{tabular} & \begin{tabular}{l}
mm \\
inch
\end{tabular} \\
\hline Work offset U & Selection of the work offset in which the zero point of the workpiece is saved. You can also delete the default value of the parameter if you do not want to specify a work offset. & \\
\hline \multirow[t]{3}{*}{Write to the work offset U} & Enter the work offset in the program & \\
\hline & \begin{tabular}{l}
- No \\
The actual \(Z\) value of the selected work offset is used.
\end{tabular} & \\
\hline & \begin{tabular}{l}
- Yes \\
Enter the work offset in the ZV parameter \\
The actual \(Z\) value of the selected work offset is overwritten with the \(Z V\) value.
\end{tabular} & \\
\hline ZV & \(Z\) value of the work offset of the workpiece & mm \\
\hline \multirow[t]{2}{*}{Blank U} & Define the form and dimensions of the workpiece: & \\
\hline & - Cylinder & \\
\hline \multirow[t]{2}{*}{XA} & Outer diameter \(\varnothing\) & mm \\
\hline & - Polygon & \\
\hline N & Number of edges & \\
\hline \multirow[t]{2}{*}{SW /L
\[
0
\]} & Width across flats Edge length & mm \\
\hline & - Centered cuboid & \\
\hline W & Width of blank & mm \\
\hline \multirow[t]{2}{*}{L} & Length of blank & mm \\
\hline & - Tube & \\
\hline XA & Outer diameter \(\varnothing\) & mm \\
\hline XIU & Inner diameter \(\varnothing\) (abs) or wall thickness (inc) & mm \\
\hline ZA & Initial dimension & mm \\
\hline ZIU & Final dimension (abs) or final dimension in relation to ZA (inc) & mm \\
\hline ZB & Machining dimension (abs) or machining dimension in relation to ZA (inc) & mm \\
\hline \multirow[t]{2}{*}{Retraction \(U\)} & The retraction area indicates the area outside of which collision-free traversing of the axes must be possible. & \\
\hline & - simple & \\
\hline XRAU & Retraction plane \(X\) external \(\varnothing\) (abs) or retraction plane \(X\) referred to \(X A\) (inc) & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline XRIU & \begin{tabular}{l}
- only for "pipe" blank \\
Retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc)
\end{tabular} & mm \\
\hline ZRAU & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline & - extended - not for a "pipe" blank & \\
\hline XRAU & Retraction plane \(X\) external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & mm \\
\hline XRIU & Retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc) & mm \\
\hline ZRAU & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline & - all & \\
\hline XRAU & Retraction plane \(X\) external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & mm \\
\hline XRIU & Retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc) & mm \\
\hline ZRAU & Retraction plane \(Z\) front (abs) or retraction plane Z referred to ZA (inc) & mm \\
\hline ZRIU & Retraction plant Z rear & mm \\
\hline Tailstock U & \begin{tabular}{l}
- Yes \\
- No
\end{tabular} & \\
\hline XRR & Retraction plane tailstock - (only "Yes" for tailstock) & mm \\
\hline Tool change point \(U\) & \begin{tabular}{l}
Tool change point, which must be approached by the revolver with its zero point. \\
- WCS (Workpiece Coordinate System) \\
- MCS (Machine Coordinate System) \\
Notes \\
- The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. \\
- Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip.
\end{tabular} & \\
\hline XT & Tool change point \(\mathrm{X} \varnothing\) & mm \\
\hline ZT & Tool change point \(Z\) & mm \\
\hline Spindle chuck data & \begin{tabular}{l}
- Yes \\
You enter spindle chuck data in the program. \\
- No \\
Spindle chuck data are transferred from the setting data. \\
Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{8.6 Program header}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline Spindle chuck data & \begin{tabular}{l} 
- \begin{tabular}{l} 
Only chuck \\
You enter spindle chuck data in the program. \\
Complete \\
You enter tailstock data in the program. \\
Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} \\
\hline
\end{tabular} \begin{tabular}{l} 
- Selecting the jaw type of the counterspindle. Dimensions of the front edge or \\
stop edge - (only if spindle chuck data "yes") \\
- Jaw type 1 \\
- Jaw type 2
\end{tabular} & \\
\hline & The main spindle chuck dimensions - (only for spindle chuck data "yes") & mm \\
\hline ZC4 & Stop dimension of the main spindle - (only for spindle chuck data "yes") & mm \\
\hline ZS4 & \begin{tabular}{l} 
Jaw dimension of the main spindle for jaw type 2 - (only for spindle chuck data \\
"yes")
\end{tabular} & mm \\
\hline ZE4 & \begin{tabular}{l} 
Chuck dimension of the counterspindle - (only for spindle chuck data "yes" and for \\
a counterspindle that has been set up)
\end{tabular} & mm \\
\hline ZC3 & \begin{tabular}{l} 
Stop dimension of the counterspindle - (only for spindle chuck data "yes" and for a a \\
counterspindle that has been set up)
\end{tabular} & mm \\
\hline ZS3 & \begin{tabular}{l} 
Jaw dimension of the counterspindle with jaw type 2 - (only for spindle chuck data \\
"yes" and for a counterspindle that has been set up)
\end{tabular} & mm \\
\hline ZE3 & \begin{tabular}{l} 
Tailstock diameter - (only for spindle chuck data "complete" and tailstock that has \\
been set up)
\end{tabular} & mm \\
\hline XR3 & \begin{tabular}{l} 
Tailstock length - (only for spindle chuck data "complete" and tailstock that has \\
been set up)
\end{tabular} & mm \\
\hline ZR3 & \begin{tabular}{l} 
The safety clearance defines how close the tool can approach the workpiece in \\
rapid traverse. \\
Note \\
Enter the safety clearance without sign into the incremental dimension.
\end{tabular} & \\
\hline SC & \begin{tabular}{l} 
Spindle speed (maximum main spindle speed) \\
If you want to machine the workpiece with a constant cutting rate, the spindle \\
speed must be increased as soon as the workpiece diameter becomes smaller. \\
Since the speed cannot be increased at will, you can set a speed limit for the main \\
spindle (S1) and for the counter-spindle (S3), depending on the shape, size, and \\
material of the workpiece or collet. \\
The machine manufacturer only sets one speed limit for the machine, i.e. none \\
that are dependent on the workpiece. \\
Please refer to the machine manufacturer's specifications.
\end{tabular} & rev/min \\
\hline Z3W & \begin{tabular}{l} 
Machining position of the counter spindle in the MCs.
\end{tabular} \\
\hline \begin{tabular}{l} 
Mach. direction of rota- \\
\(i\) Milling direction \\
- \begin{tabular}{l} 
Conventional milling \\
- Climbing
\end{tabular}
\end{tabular} & mm \\
\hline
\end{tabular}

\subsection*{8.7 Generating program blocks}

After a new program is created and the program header is filled out, define the individual machining steps in program blocks that are necessary to machine the workpiece.

You can only create the program blocks between the program header and the program end.

\section*{Procedure}

\section*{Selecting a technological function}
1. Position the cursor in the work plan on the line behind which a new program block is to be inserted.
(1) \begin{tabular}{c} 
Strght \\
Circle
\end{tabular}
2. Using the softkeys, select the desired function. The associated parameter screen is displayed.
```

Mill.

```
3. First, program the tool, correction (offset) value, feedrate and spindle speed ( \(T, D, F, S, V\) ) and then enter the values for the other parameters.

\section*{Selecting a tool from the tool list}

Select tool

New
tool
4. Press the "Select tools" softkey if you wish to select the tool for parameter "T".
The "Tool selection" window is opened.
5. Position the cursor in the tool list on the tool that you wish to use for machining and press the "To program" softkey.
The selected tool is accepted into the parameter screen form.
- OR -

Press the "Tool list" and "New tool" softkeys.
The "Tool selection" window is opened.

Using the softkeys on the vertical softkey bar, select the required tool with the data and press the "To program" softkey.
The selected tool is accepted into the parameter screen form.
The process plan is displayed and the newly generated program block is marked.

\section*{\(8.8 \quad\) Tool, offset value, feedrate and spindle speed (T, D, F, S, V)}

The following parameters should be entered for every program block.

\section*{Tool (T)}

Each time a workpiece is machined, you must program a tool. Tools are selected by name, and the selection is integrated in all parameter screen forms of the machining cycles (with the exception for straight line/circle).
The tool length offsets become active as soon as the tool is changed.
Tool selection is modal for the straight line/circle, i.e. if the same tool is used to perform several machining steps in succession, you only have to program one tool for the first straight line/circle.

\section*{Cutting edge (D)}

In the case of tools with several cutting edges, there is a separate set of individual tool offset data for each edge. For these tools, you must select or specify the number of the cutting edge that you would like to use for machining.

\section*{NOTICE}

\section*{Risk of collision}

If, for tools with several cutting edges, you specify the incorrect cutting edge number and move the tool, then collisions can occur. Always ensure that you enter the correct cutting edge number.

\section*{Radius compensation}

The tool radius compensation is automatically taken into account during all machining cycles, with the exception of path milling and straight.

For path milling and straight lines, you have the option of programming the machining with or without radius compensation. The tool radius compensation is modal for straight lines, i.e. you have to deselect the radius compensation again if you want to traverse without radius compensation.

新
Radius compensation to right of contour
㗔
Radius compensation to left of contour
\%
Radius compensation offRadius compensation remains as previously set

\section*{Feedrate (F)}

The feedrate \(F\) (also referred to as the machining feedrate) specifies the speed at which the axes move when machining the workpiece. The machining feedrate is entered in \(\mathrm{mm} / \mathrm{min}\), \(\mathrm{mm} / \mathrm{rev}\) or in mm/tooth.

It is only possible to enter the feedrate in mm/tooth during milling; this ensures that each cutting edge of the milling cutter is cutting under the best possible conditions. The feedrate per tooth corresponds to the linear path traversed by the milling cutter when a tooth is engaged.

For milling and turning cycles, the feedrate during roughing is relative to the milling or cutting center point. This is also applies to finishing, with the exception of contours with inner curves. In this case, the feedrate is relative to the contact point between the tool and the workpiece.

The maximum feedrate is determined via machine data.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Converting the feedrate (F) for drilling and milling}

The feedrate entered for drilling cycles is automatically converted when switching from \(\mathrm{mm} / \mathrm{min}\) to \(\mathrm{mm} / \mathrm{rev}\) and vice versa using the selected tool diameter.

The feedrate entered for milling cycles is automatically converted when switching from mm/Z to \(\mathrm{mm} / \mathrm{min}\) and vice versa using the selected tool diameter.

\section*{Spindle speed (S)}

The spindle speed \(S\) specifies the number of spindle revolutions per minute (rpm) and is programmed along with a tool. The speed specified relates to the main spindle (S1) or counter-spindle (S3) when turning and axial drilling, and to the tool spindle (S2) when drilling and milling.

The spindle starts immediately after the tool change. The spindle stops upon reset, program end, or a tool change. The spindle's direction of rotation is specified in the tool list for each tool.

\section*{Cutting rate (V)}

The cutting rate V is a circumferential velocity ( \(\mathrm{m} / \mathrm{min}\) ) and is programmed together with a tool, as an alternative to the spindle speed. The cutting rate is relative to the main spindle (V1) or the counter-spindle (V3) for turning and axial drilling, and corresponds to the circumferential velocity of the workpiece at the point that is currently being machined.

However, for drilling and milling, the cutting rate is relative to the tool spindle (V2) and corresponds to the peripheral speed at which the cutting edge of the tool machines the workpiece.

\section*{Converting the spindle speed ( S ) / cutting rate ( V ) when milling}

As an alternative to the cutting rate, you can also program the spindle speed.
For the milling cycles, the cutting rate ( \(\mathrm{m} / \mathrm{min}\) ) that is entered is automatically converted into the spindle speed (rpm) using the tool diameter - and vice versa.

\section*{Machining}

When machining some cycles, you can choose between roughing, finishing, or complete machining. For certain milling cycles, finishing edge or finishing base are possible.
- Roughing

One or more machining operations with depth infeed
- Finishing

Single machining operation
- Edge finishing

Only the edge of the object is finished
- Base finishing

Only the base of the object is finished
- Complete machining

Roughing and finishing with one tool in a
- machining step

If you want to rough and finish using two different tools, you must call the machining cycle twice (1st block = roughing; 2nd block = finishing). The programmed parameters are retained when the cycle is called for the second time.

\subsection*{8.9 Call work offsets}

You can call work offsets (G54, etc.) from any program.
You define work offsets in work offset lists. You can also view the coordinates of the selected offset here.

\section*{Procedure}

1. Press the "Various", "Transformations" and "Work offset" softkeys. The "Work offset" window opens.
2. Select the desired work offset (e.g. G54).
3. Press the "Accept" softkey.

The work offset is transferred into the work plan.

\subsection*{8.10 Repeating program blocks}

If certain steps when machining a workpiece have to be executed more than once, it is only necessary to program these steps once. You have the option of repeating program blocks.

\section*{Note}

Machining several workpieces
The program repeat function is not suitable to program repeat machining of parts.
In order to repeatedly machine the same workpieces, program this using "end of program".

\section*{Start and end marker}

You must mark the program blocks that you want to repeat with a start and end marker. You can then call these program blocks up to 200 times within a program. The markers must be unique, i.e. they must have different names. No names used in the NCK can be used.
You can also set markers and repeats after creating the program, but not within linked program blocks.

\section*{Note}

You can use one and the same marker as end marker for preceding program blocks and as start marker for following program blocks.

\section*{Procedure}


Repeat program

Set mark Accept
4. Press the "Set marker" and "Accept" softkeys. A start marker is inserted behind the actual block.
5. Enter the program blocks that you want to repeat later.
6. Press the "Set marker" and "Accept" softkeys again. An end marker is inserted after the actual block.


Repeat program
7. Continue programming up to the point where you want to repeat the program blocks.
8. Press the "Various" and "Repeat progr." softkeys.
9. Enter the names of the start and end markers and the number of times the blocks are to be repeated.
10. Press the "Accept" softkey.

The marked program blocks are repeated.

\subsection*{8.11 Entering the number of workpieces}

If you wish to produce a certain quantity of the same workpiece, then at the end of the program, specify that you wish to repeat the program.
If your machine has a bar loader for example, you can program the reloading of the workpiece and then the actual machining at the beginning of the program. At the end, cut off the completed workpiece.
Control the numbers of times that the program is repeated using the "Times, counters" window. Enter the number of required workpieces using the target number. You can track the number of machined and completed workpieces in the actual counter window.
Workpieces can be completely automatically produced in this fashion.

\section*{Controlling program repetition}
\begin{tabular}{|l|l|l|}
\hline \begin{tabular}{l} 
End of program: \\
Repeat
\end{tabular} & \begin{tabular}{l} 
Times, counter: \\
Counts the work- \\
pieces
\end{tabular} & No \\
\hline No & Yes & A CYCLE START is required for each workpiece. \\
\hline No & Yes & \begin{tabular}{l} 
A CYCLE START is required for each workpiece. \\
The workpieces are counted.
\end{tabular} \\
\hline Yes & No & \begin{tabular}{l} 
The program is repeated without a new CYCLE START \\
until the required number of workpieces have been ma- \\
chined.
\end{tabular} \\
\hline Yes & \begin{tabular}{l} 
Without a new CYCLE START, the program is repeated \\
an infinite number of times. \\
You can interrupt program execution with <RESET>.
\end{tabular} \\
\hline
\end{tabular}

\section*{Procedure}
1. Open the "Program end" program block, if you want to machine more than one workpiece.
2. In the "Repeat" field, enter "Yes".
3. Press the "Accept" softkey.

If you start the program later, program execution is repeated.
Depending on the settings in the "Times, counters" window, the program is repeated until the set number of workpieces has been machined.

See also
Displaying the program runtime and counting workpieces (Page 177)

\subsection*{8.12 Changing program blocks}

You can subsequently optimize the parameters in the programmed blocks or adapt them to new situations, e.g. if you want to increase the feedrate or shift a position. In this case, you can directly change all the parameters in every program block in the associated parameter screen form.

\section*{Procedure}

1. Select the program that you wish to change in the "Program Manager" operating area.
2. Press the <Cursor right> or <INPUT> key.

The work plan of the program is displayed.
3. Position the cursor in the work plan at the desired program block and press the <Cursor right> key.
The parameter screen for the selected program block is displayed.
4. Make the desired changes.
5. Press the "Accept" softkey.
- OR -

Press the <Cursor left> key.

The changes are accepted in the program.

\subsection*{8.13 Changing program settings}

\section*{Function}

All parameters specified in the program header with the exception of the blank shape and the unit of measurement can be changed at any point in the program. It is also possible to change the basic setting for the direction of rotation of machining in the case of milling.

The settings in the program header are modal, i.e. they remain active until they are changed.

\section*{Retraction}

A changed retraction plane takes effect starting with the safety clearance of the last cycle, because the additional retraction from the next cycle is accepted.

\section*{Machining direction}

The machining direction of rotation (climbing or conventional) is defined as the direction of movement of the milling tooth with respect to the workpiece, i.e. ShopTurn evaluates this parameter in conjunction with the direction of rotation of the spindle for milling, with the exception of path milling.

The basic setting for the machining direction is programmed in a machine data.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}

1. Select the "Program" operating area.

ous
2. Press the "Various" and "Settings" softkeys. The "Settings" input window opens.

\section*{Parameters}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Retraction U & \begin{tabular}{l}
Lift mode \\
- simple \\
- Extended \\
- all
\end{tabular} & \\
\hline XRAU & Retraction plane X external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & mm \\
\hline XRI U & Retraction plane \(X\) internal \(\varnothing\) (abs) or retraction plane \(X\) referred to XI (inc) - (only for retraction "extended" and "all") & mm \\
\hline ZRAU & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline ZRI & Retraction plane Z rear - (only for retraction "all") & mm \\
\hline Tailstock \(\cup\) & \begin{tabular}{l}
Yes \\
- Tailstock is displayed for simulation / simultaneous recording \\
- When approaching/retracting, the retraction logic is taken into account No
\end{tabular} & \\
\hline XRR & Retraction plane - (only "Yes" for tailstock) & mm \\
\hline Tool change point U & \begin{tabular}{l}
Tool change point \\
- Work (Workpiece Coordinate System) \\
- Machine (Machine Coordinate System)
\end{tabular} & \\
\hline XT & Tool change point \(X\) & mm \\
\hline ZT & Tool change point \(Z\) & mm \\
\hline SC & \begin{tabular}{l}
Safety clearance (inc) \\
Acts in relation to the reference point. The direction in which the safety clearance is active is automatically determined by the cycle.
\end{tabular} & mm \\
\hline S1 & Maximum speed, main spindle & \(\mathrm{rev} / \mathrm{min}\) \\
\hline Machining direction & \begin{tabular}{l}
Milling direction: \\
- Climbing \\
- Conventional
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{8.14 Selection of the cycles via softkey}

\section*{Overview of the machining steps}

The following machining steps are available.
All of the cycles/functions available in the control are shown in this display. However, at a specific system, only the steps possible corresponding to the selected technology can be selected.


Drilling cycles only for turning/milling machine
\begin{tabular}{|c|c|}
\hline Centering & \\
\hline \begin{tabular}{c} 
Drilling \\
Reaming
\end{tabular} & \(\Rightarrow\) \\
\hline
\end{tabular}


\section*{:}




\section*{Scaling}

\section*{Mirroring}
Rotation
\(C\) axis

\begin{tabular}{|c|c|c|}
\(\substack{\text { Repeat } \\
\text { program }}\) & & Set \\
mark \\
\hline
\end{tabular}

Repeat
program

\(\Rightarrow \quad\)\begin{tabular}{l} 
A menu tree with all of the available measuring versions of the measuring \\
cycle function "Measure workpiece" can be found in the following refer- \\
ence:
\end{tabular}
Programming Manual Measuring cycles / SINUMERIK 840D sI/828D

\subsection*{8.15 Calling technology functions}

\subsection*{8.15.1 Additional functions in the input screens}

\section*{Selection of units}

0 If, for example, the unit can be switched in a field, this is highlighted as soon as the cursor is positioned on the element. In this way, the operator recognizes the dependency.
The selection symbol is also displayed in the tooltip.

\section*{Display of abs or inc}

The abbreviations "abs" and "inc" for absolute and incremental values are displayed behind the entry fields when a switchover is possible for the field.

\section*{Help screens}

2D and 3D graphics or sectional views are displayed for the parameterization of the cycles.

\section*{Online help}

If you wish to obtain more detailed information about certain \(G\) code commands or cycle parameters, then you can call a context-sensitive online help.

\subsection*{8.15.2 Checking cycle parameters}

The entered parameters are already checked during the program creation in order to avoid faulty entries.

If a parameter is assigned an illegal value, this is indicated in the input screen as follows:
- The entry field is displayed with a colored background (orange).
- The comment line displays a note.
- If the parameter entry field is selected with the cursor, the note is also displayed as a tool tip.
The programming can only be completed after the incorrect value has been corrected.
Faulty parameter values are also monitored with alarms during the cycle runtime.

\subsection*{8.15.3 Programming variables}

In principle, variables or expressions can also be used in the input fields of the screen forms instead of specific numeric values. In this way, programs can be created very flexibly.

\section*{Input of variables}

Please note the following points when using variables:
- Values of variables and expressions are not checked since the values are not known at the time of programming.
- Variables and expressions cannot be used in fields in which a text is expected (e.g. tool name).

An exception is the "Engraving" function, in which you can assign the desired text in the text field via a variable as "Variable text".
- Selection fields generally cannot be programmed with variables.

\section*{Examples}
```

VAR_A
VAR_A+2*VAR_B
SIN(VAR_C)

```

\subsection*{8.15.4 Setting data for technological functions}

Technological functions can be influenced and corrected using machine or setting data.
For additional information, please refer to the following documentation:
Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl

\subsection*{8.15.5 Changing a cycle call}

You have called the desired cycle via softkey in the program editor, entered the parameters and confirmed with "Accept".

\section*{Procedure}

1. Select the desired cycle call and press the <Cursor right> key. The associated input screen of the selected cycle call is opened.
- OR -


Press the <SHIFT + INSERT> key combination.
This starts the edit mode for this cycle call and you can edit it like a normal NC block. This means that it is possible to generate an empty block before the cycle is called. For instance, to insert something before a cycle that is located at the beginning of the program.
Note: In edit mode, the cycle call can be changed in such a way that it can no longer be recompiled in the parameter screen.


You exit the edit mode by pressing the <SHIFT + INSERT> key combination.
- OR -

You are in the edit mode and press the <INPUT> key. A new line is created after the cursor position.

\subsection*{8.15.6 Compatibility for cycle support}

The cycle support is generally upwards compatible. This means that cycle calls in NC programs can always be recompiled with a higher software version, changed and then run again.

When transferring NC programs to a machine with a lower software version, it cannot be guaranteed, however, that the program will be able to be changed by recompiling cycle calls.

\subsection*{8.16 Programming the approach/retraction cycle}

If you wish to shorten the approach/retraction for a machining cycle or solve a complex geometrical situation when approaching/retracting, you can generate a special cycle. In this case, the approach/retraction strategy normally used is not taken into account.
You can insert the approach/retraction cycle between any machining step program blocks, but not within linked program blocks.

\section*{Starting point}

The starting point for the approach/retraction cycle is the safety clearance approached after the last machining operation.

\section*{Tool change}

If you want to perform a tool change, you can move the tool through a total of 3 positions (P1 to P3) to the tool change point and through a maximum of 3 additional positions ( P 4 to P 6 ) to the next starting point. If the tool does not need to be changed, however, you have a total of 6 positions available for the approach to the next starting position.

If 3 or 6 positions are not sufficient for the approach/retraction, you can call the cycle several times in succession to program further positions.

\section*{\(\triangle\) CAUTION}

\section*{Risk of collision}

Note that the tool will move from the last position programmed in the approach/retraction cycle directly to the starting point for the next machining operation.

\section*{See also}

Machining cycle, approach/retraction (Page 243)

\section*{Procedure}


Press the menu forward key and the "Straight Circle" softkey.

Retract/ Approach

Table 8- 1
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \[
\begin{gathered}
\mathrm{F} 1 \\
\mathrm{U}
\end{gathered}
\] & Feedrate to approach the first position Alternatively, rapid traverse & \(\mathrm{mm} / \mathrm{min}\) \\
\hline X1 & 1. position \(\varnothing\) (abs) or 1st position (inc) & mm (in) \\
\hline Z1 & 1. position (abs or inc) & mm \\
\hline \[
\begin{aligned}
& \mathrm{F} 2 \\
& \mathrm{U}
\end{aligned}
\] & Feedrate for approach to the second position Alternatively, rapid traverse & \(\mathrm{mm} / \mathrm{min}\) \\
\hline X2 & 2. position \(\varnothing\) (abs) or 2nd position (inc) & mm \\
\hline Z2 & 2. position (abs or inc) & mm \\
\hline \[
\begin{aligned}
& \mathrm{F} 3 \\
& \mathrm{U}
\end{aligned}
\] & Feedrate to approach the third position Alternatively, rapid traverse & \(\mathrm{mm} / \mathrm{min}\) \\
\hline X3 & 3. position \(\varnothing\) (abs) or 3rd position (inc) & mm \\
\hline Z3 & 3. position (abs or inc) & mm \\
\hline Tool change U & \begin{tabular}{l}
TIChngPt: Approach the tool change point from the last programmed position and carry out a tool change \\
direct: Tool is not changed at the tool change position, but at the last programmed position \\
No: Tool is not changed
\end{tabular} & \\
\hline T & Tool name - (only for "direct" tool change) & \\
\hline D & Cutting edge number - (only for "direct" tool change) & \\
\hline \[
\begin{gathered}
\mathrm{F} 4 \\
\mathrm{U}
\end{gathered}
\] & Feedrate for approach to the fourth position Alternatively, rapid traverse & \(\mathrm{mm} / \mathrm{min}\) \\
\hline X4 & 4. position \(\varnothing\) (abs) or 4th position (inc) & mm \\
\hline Z4 & 4. position (abs or inc) & mm \\
\hline \[
\begin{aligned}
& \text { F5 } \\
& \mathrm{U}
\end{aligned}
\] & Feedrate to approach the fifth position Alternatively, rapid traverse & \(\mathrm{mm} / \mathrm{min}\) \\
\hline X5 & 5. position \(\varnothing\) (abs) or 5th position (inc) & mm \\
\hline Z5 & 5. position (abs or inc) & mm \\
\hline \[
\begin{aligned}
& \mathrm{F} 6 \\
& \mathrm{O}
\end{aligned}
\] & Feedrate to approach the sixth position Alternatively, rapid traverse & \(\mathrm{mm} / \mathrm{min}\) \\
\hline X6 & 6. position \(\varnothing\) (abs) or 6th position (inc) & mm \\
\hline Z6 & 6. position (abs or inc) & mm \\
\hline
\end{tabular}

\subsection*{8.17 Measuring cycle support}

Measuring cycles are general subroutines designed to solve specific measurement tasks. They can be adapted to specific problems via parameter settings.


\section*{Software option}

You require the "Measuring cycles" option to use "Measuring cycles".

\section*{References}

You will find a more detailed description on how to use measuring cycles in:
Programming Manual Measuring cycles / SINUMERIK 840D sl/828D

\subsection*{8.18 Example: Standard machining}

\section*{General information}

The following example is described in detail as ShopTurn program. A G code program is generated in the same way; however, some differences must be observed.
If you copy the G code program listed below, read it into the control and open it in the editor, then you can track the individual program steps.


\section*{Machine manufacturer}

Under all circumstances, observe the machine manufacturer's instructions.

\section*{Tools}

The following tools are saved in the tool manager:
\begin{tabular}{ll} 
Roughing tool_80 & \(80^{\circ}\), R0.6 \\
Roughing tool_55 & \(55^{\circ}\), R0.4 \\
Finishing tool & \(35^{\circ}\), R0.4 \\
Plunge cutter & Plate width 4 \\
Threading tool_2 & \\
Drill_D5 & \(\varnothing 5\) \\
Miller_D8 & \(\varnothing 8\)
\end{tabular}

Adapt the cutting data to the tools used and the specific application conditions at the machine.

\section*{Blank}

Dimensions: \(\varnothing 90 \times 120\)
Material: Aluminum

\subsection*{8.18.1 Workpiece drawing}


\subsection*{8.18.2 Programming}

\section*{1. Program header}
1. Specify the blank.

Measurement unit mm
\begin{tabular}{lr} 
Blank & \multicolumn{1}{c}{ Cylinder } \\
XA & 90abs \\
ZA & \(+1.0 a b s\) \\
ZI & -120 abs \\
ZB & -100 abs \\
Retraction & \\
XRA & simple \\
ZRA & Machine \\
Tool change point & 160 abs \\
XT & \(409 a b s\) \\
ZT & 1 \\
SC & \(4000 \mathrm{rev} / \mathrm{min}\) \\
S1 & Climbing
\end{tabular}
2. Press the "Accept" softkey.

The work plan is displayed. Program header and end of program are created as program blocks.
The end of program is automatically defined.

\section*{2. Stock removal cycle for facing}

Stock
remoual
2. Select the machining strategy.
3. Enter the following technology parameters:

T Roughing D1 F \(0.300 \mathrm{~mm} / \mathrm{rev} \quad\) V \(350 \mathrm{~m} / \mathrm{min}\) tool_80
4. Enter the following parameters:
\begin{tabular}{lc} 
Machining & Roughing ( \(\nabla\) ) \\
Position & S.s. \\
Direction & Face (parallel to the \(X\) axis) \\
X0 & \(90 a b s\) \\
Z0 & \(2 a b s\)
\end{tabular}
\begin{tabular}{|c|c|c|c|}
\hline \multirow[t]{5}{*}{} & & X1 & -1.6abs \\
\hline & & Z1 & Oabs \\
\hline & & D & 2inc \\
\hline & & UX & Oinc \\
\hline & & UZ & 0.1 inc \\
\hline Accept & 5. & & \\
\hline
\end{tabular}

\section*{3. Input of blank contour with contour computer}


New contour
1. Press the "Cont. turn." and "New contour" softkeys. The "New Contour" input window opens.
2. Enter the contour name (in this case: Cont_1).

The contour calculated as NC code is written as internal subprogram between a start and an end marker containing the entered name.

3. Press the "Accept" softkey. The "Starting point" entry field opens.
4. Enter the starting point of the contour.

X 60abs Z 0abs
5. Press the "Accept" softkey.

Accept
6. Enter the following contour elements and acknowledge using the "Accept" softkey.

6.1 Z -40abs

6.2 X

80abs
Z
-45abs

6.3
\(-65 a b s\)
6.4 X

90abs
Z
-70abs
6.5 Z
-95abs
6.6 X

Oabs


Blank contour

\section*{4. Input of finished part with contour computer}

New contour

1. Press the "Cont. turn." and "New contour" softkeys.

The "New Contour" input window opens.
2. Enter the contour name (in this case: Cont_2).

The contour calculated as NC code is written as internal subprogram between a start and an end marker containing the entered name.
3. Press the "Accept" softkey.

The "Starting point" entry field opens.
4. Specify the contour starting point of the contour.
X
Oabs
Z
Oabs
5. Press the "Accept" softkey.

6. Enter the following contour elements and acknowledge using the "Accept" softkey.
6.1 X 48abs FS 3

\subsection*{6.3 Direction of rotation}
\(\mathbf{R} \quad\) 23abs X 60abs K \(-35 a b s \quad 180\) abs Afterwards, entry fields are inactive.

Dialog select

Dialog accept


FS 2
6.5 Z
\(6.6 \quad X\)
90abs
Z
-85abs
FS3

6.7 Z -95abs
7. Press the "Accept" softkey.


Finished-part contour

\section*{5. Stock removal (roughing)}

Stock removal
1. Press the "Cont. turn." and "Stock removal" softkeys.

The "Stock Removal" input window opens.
2. Enter the following technology parameters:

T Roughing tool 80 D1 F \(0.350 \mathrm{~mm} / \mathrm{rev} \quad\) V \(400 \mathrm{~m} / \mathrm{min}\)
3. Enter the following parameters:

Machining Roughing ( \(\overline{\text { })}\)
Machining direction Longitudinal
Position outside
Machining direction
\(\longleftarrow\)
(from the face to the rear side)
D 4.000inc
Cutting depth
\begin{tabular}{cc} 
UX & 0.4 inc \\
UZ & 0.2 inc \\
DI & 0
\end{tabular}

BL Cylinder
XD Oinc
ZD Oinc
Relief cuts No
Set machining No area limits
4. Press the "Accept" softkey.


Stock removal contour

\section*{6. Solid machine residual material}

1. Press the "Cont. turn." and "St. remov. resid." softkeys. The "Stock removal residual material" input window opens.
2. Enter the following technology parameters:

T Roughing tool_55 D1 F \(0.35 \mathrm{~mm} / \mathrm{rev} \quad\) V \(400 \mathrm{~m} / \mathrm{min}\)
3. Enter the following parameters:
\begin{tabular}{ll} 
Machining & Roughing \((\nabla)\) \\
Machining direction & Longitudinal \\
Position & outside \\
Machining direction & \(\leftarrow\)
\end{tabular}

D 2inc
Cutting depth

UX
UZ
DI
Relief cuts
FR
\(\leftleftarrows\)

Set machining area limits No
4. Press the "Accept" softkey.

\section*{7. Stock removal (finishing)}

1. Press the "Cont. turn." and "Stock removal" softkeys.

The "Stock Removal" input window opens.
Stock
remoual
2. Enter the following technology parameters:

T Finishing tool_D1 F \(0.1 \mathrm{~mm} / \mathrm{rev} \quad\) V \(450 \mathrm{~m} / \mathrm{min}\)
3. Enter the following parameters:

Machining Finishing ( \(\nabla \nabla \nabla\) )
Machining direction Longitudinal
Position outside
Machining direction
\(\leftarrow\)
(from the face to the rear side)

\section*{Allowance \\ No}

Relief cuts Yes
Set machining area limits No
4. Press the "Accept" softkey.
8. Groove (roughing)

1. Press the "Turning", "Groove" and "Groove with inclines" softkeys. The "Groove 1" entry field opens.
2. Enter the following technology parameters:
T Grooving D1 F \(0.150 \mathrm{~mm} / \mathrm{rev} \quad\) V \(220 \mathrm{~m} / \mathrm{min}\) tool
3. Enter the following parameters:
\begin{tabular}{|c|c|}
\hline Machining & Roughing ( \(\nabla\) ) \\
\hline Groove position & 颜 \\
\hline Reference point & 미 \\
\hline X0 & 60abs \\
\hline Z0 & -70 \\
\hline B2 & 8inc \\
\hline T1 & 4 inc \\
\hline
\end{tabular}
\begin{tabular}{lc}
\(\boldsymbol{\alpha} 1\) & 15degrees \\
a2 & 15degrees \\
FS1 & 1 \\
R2 & 1 \\
R3 & 1 \\
FS4 & 1 \\
D & 2 inc \\
UX & 0.4 inc \\
UZ & 0.2 inc \\
N & 1
\end{tabular}
4. Press the "Accept" softkey.


Contour, groove

\section*{9. Groove (finishing)}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \[
\begin{gathered}
\text { Turn- } \\
\text { ing }
\end{gathered}
\] & \multirow[t]{2}{*}{1.} & \multicolumn{4}{|l|}{Press the "Turning", "Groove" and "Groove with inclines" softkeys. The "Groove 2" entry field opens.} \\
\hline Groove & & & & & \\
\hline \multicolumn{6}{|l|}{[]} \\
\hline & \multirow[t]{2}{*}{2.} & \multicolumn{4}{|l|}{Enter the following technology parameters:} \\
\hline & & T Grooving tool & D1 & F 0.1 mm/rev & V \(220 \mathrm{~m} / \mathrm{min}\) \\
\hline & \multirow[t]{15}{*}{3.} & \multicolumn{4}{|l|}{Enter the following parameters:} \\
\hline & & Machining & & Finishing ( \(\bar{\nabla} \nabla\) ) & \\
\hline & & Groove position & & 榡 & \\
\hline & & Reference point & & 回 & \\
\hline & & X0 & & 60abs & \\
\hline & & Z0 & & -70 & \\
\hline & & B1 & & 5.856inc & \\
\hline & & T1 & & 4 inc & \\
\hline & & a1 & & 15degrees & \\
\hline & & a2 & & 15degrees & \\
\hline & & FS1 & & 1 & \\
\hline & & R2 & & 1 & \\
\hline & & R3 & & 1 & \\
\hline & & FS4 & & 1 & \\
\hline & & N & & 1 & \\
\hline Accept & 4. & Press the "Accep & softk & & \\
\hline
\end{tabular}

\section*{10. Longitudinal threads M48 x2 (roughing)}


Thread
Thread long.
1. Press the "Turning", "Thread" and "Thread longitudinal" softkeys. The "Longitudinal thread" entry field opens.
2. Enter the following parameters:
\begin{tabular}{lc} 
T & Threading tool_2 \\
Table & D1 \\
P & \(2 \mathrm{~mm} / \mathrm{rev}\) \\
G & 0 \\
S & \(995 \mathrm{rev} / \mathrm{min}\) \\
Machining type & Roughing ( \(\nabla\) ) \\
Infeed: Constant cutting & Diminishing \\
cross-section & \\
Thread & External thread \\
X0 & 48 abs \\
Z0 & 0 abs \\
Z1 & -25 abs \\
LW & 4 inc \\
LR & 4 inc \\
H1 & 1.227 inc \\
aP & 30 degrees \\
Infeed & \\
ND & 5 \\
U & 0.150 inc \\
VR & 1 inc \\
Multiple threads & No \\
a0 & Odegrees
\end{tabular}
3. Press the "Accept" softkey.

\section*{11. Longitudinal threads M48 x 2 (finishing)}
\begin{tabular}{|c|c|c|c|c|}
\hline \[
\begin{gathered}
\text { Turn- } \\
\text { Ing } \\
\text { ing }
\end{gathered}
\] & \multirow[t]{3}{*}{1.} & \multicolumn{3}{|l|}{Press the "Turning", "Thread" and "Thread longitudinal" softkeys. The "Longitudinal thread" entry field opens.} \\
\hline Thread & & & & \\
\hline \multirow[t]{21}{*}{\[
\begin{aligned}
& \text { Thread } \\
& \text { long. }
\end{aligned}
\]} & & & & \\
\hline & \multirow[t]{20}{*}{2.} & Enter the followi & ters: & \\
\hline & & T & Threading tool_2 & D \\
\hline & & Table & without & \\
\hline & & P & \(2 \mathrm{~mm} / \mathrm{rev}\) & \\
\hline & & G & 0 & \\
\hline & & S & 995rev/min & \\
\hline & & Machining type & Finishing ( \(\nabla \nabla \nabla\) ) & \\
\hline & & Thread & External thread & \\
\hline & & X0 & 48abs & \\
\hline & & Z0 & Oabs & \\
\hline & & Z1 & -25abs & \\
\hline & & LW & 4 inc & \\
\hline & & LR & 4inc & \\
\hline & & H1 & 1.227 inc & \\
\hline & & aP & 30degrees & \\
\hline & & Infeed & \(凶\) & \\
\hline & & NN & 2 & \\
\hline & & VR & 1 inc & \\
\hline & & Multiple threads & No & \\
\hline & & a0 & Odegrees & \\
\hline \[
\underset{\text { Accept }}{\checkmark}
\] & 3. & Press the "Acce & & \\
\hline
\end{tabular}

\section*{12. Drilling}

Drill.
Drilling Reaming

Drilling
1. Press the "Drilling", "Drilling reaming" and "Drilling" softkeys. The "Drilling" input window opens.
2. Enter the following technology parameters:
T Drill_D5 D1 F \(0.1 \mathrm{~mm} / \mathrm{rev} \quad\) V \(50 \mathrm{~m} / \mathrm{min}\)
3. Enter the following parameters:
\begin{tabular}{ll} 
Machined surface & Face C \\
Drilling depth & Tip \\
Z1 & \multicolumn{2}{c}{ 10inc } \\
DT & 0s
\end{tabular}
4. Press the "Accept" softkey.
13. Positioning

1. Press the "Drilling", "Positions" and "Freely Programmable Positions" softkeys.
The "Positions" input window opens.
2. Enter the following parameters:
\begin{tabular}{lr} 
Machined surface & \multicolumn{1}{l}{\begin{tabular}{l} 
Face C \\
Coordinate system
\end{tabular}} \\
Z0 & Polar \\
C0 & Oabs \\
L0 & 0abs \\
C1 & 16 abs \\
L1 & 90 abs \\
C2 & 16 abs \\
L2 & 180 abs \\
C3 & 16 abs \\
L3 & 270 abs \\
& 16 abs
\end{tabular}
3. Press the "Accept" softkey.

\section*{14. Milling the rectangular pocket}

Pocket
Rectang. pocket
2. Enter the following technology parameters:

T Miller_D8 D1 F \(0.030 \mathrm{~mm} /\) tooth V \(200 \mathrm{~m} / \mathrm{min}\)
3. Enter the following parameters:
\begin{tabular}{ll} 
Machined surface & Face C \\
Machining type & Roughing ( \(\nabla\) ) \\
Machining position & Single position \\
XO & Oabs \\
YO & Oabs \\
ZO & Oabs
\end{tabular}

W 23
L 23
R 8
a0
Z1
4Degrees
5inc
DXY 50\%
DZ 3
UXY 0.1mm
UZ
0
Insertion
FZ
Vertical
\(0.015 \mathrm{~mm} /\) tooth
4. Press the "Accept" softkey.

\section*{8．18．3 Results／simulation test}


Figure 8－10 Programming graphics
\begin{tabular}{|c|c|c|}
\hline P Program header & & Work offset 654 \\
\hline Stw Stock remoual & \(\nabla\) & T＝Schrupper＿80 F0．3／reu U＝350m Face X0＝90 \\
\hline \(\checkmark\) Contour & \multicolumn{2}{|r|}{KONT＿1} \\
\hline \(\checkmark\)－Contour & \multicolumn{2}{|r|}{KONT＿2} \\
\hline \％Stock remoual & \(\nabla\) & \(\mathrm{T}=\) Schrupper \(80 \mathrm{FO} .35 / \mathrm{reu} \mathrm{J}=400 \mathrm{~m}\) \\
\hline 亥，Residual cutting & \(\nabla\) & \(\mathrm{T}=\) Schrupper＿55 F0．35／reu U＝400m \\
\hline \％Stock removal & \(\nabla \nabla \nabla\) & \(\mathrm{I}=\) Schlichter \(\mathrm{F} 0.1 / \mathrm{reu} \mathrm{U}=450 \mathrm{~m}\) \\
\hline 弱 Groove & \(\nabla\) & \(\mathrm{I}=\) Stecher \(\mathrm{F} 0.15 / \mathrm{reu} \mathrm{U}=220 \mathrm{~m} \mathrm{X}=\mathbf{6 0 ~ 2 0}=-70\) \\
\hline 揈 Groove & \(\nabla \nabla \nabla\) & \(\mathrm{T}=\) Stecher \(\mathrm{F} 0.1 / \mathrm{reu} \mathrm{U}=220 \mathrm{~m} \mathrm{X0}=60 \mathrm{ZO}=-70\) \\
\hline Thread long． & \(\nabla\) & T＝Gewindestahl＿2 P2mm／reu S＝995reu Outside \\
\hline NiNTHread long． & \(\nabla \nabla \nabla\) & \(\mathrm{T}=\) Gewindestahl＿2 P2mm／reu S＝995reu Outside \\
\hline 氛发］Drilling & & T＝Bohrer＿D5 F0．1／min U＝50m \(21=10 \mathrm{inc}\) \\
\hline v．001：Positions & & 20＝0 C0＝0 L0＝16 C1＝90 L1＝16 C2＝180 L2＝16 \\
\hline \％em．Rectang．pocket & \(\nabla\) & \(\mathrm{T}=\) Fräser＿08 F0．03／ \(\mathrm{min} \mathrm{U}=\mathbf{2 0 0 m ~ X ~} 0=0\) Y0＝0 \\
\hline END End of program & & \\
\hline
\end{tabular}

Figure 8－11 Process plan
Program test by means of simulation
During simulation，the current program is calculated in its entirety and the result displayed in graphic form．


Figure 8-12 3D view

\subsection*{8.18.4 G code machining program}
```

N1 G54
N2 WORKPIECE (,,"","CYLINDER",192,2,-120,-100,90)
N3 GO X200 Z200 Y0
;*****************************************
N4 T="ROUGHING TOOL_80" D1
N5 M06
N6 G96 S350 M04
N7 CYCLE951(90,2,-1.6,0,-1.6,0,1,2,0,0.1,12,0,0,0,1,0.3,0,2,1110000)
N8 G96 S400
N9 CYCLE62(,2,"E_LAB_A_CONT_2","E_LAB_E_CONT_2")
N10 CYCLE952("STOCK
REMOVAL_1", "BLANK_1",2301311,0.35,0.15,0,4,0.1,0.1,0.4,0.2,0.1,0,1,0,0, ,, , 2,2, , 0, 1, ,0,12,1110110)
N11 G0 X200 Z200
;*******************************************
N12 T="ROUGHING TOOL_55" D1
N13 M06
N14 G96 S400 M04
N15 CYCLE952("STOCK
REMOVAL_2", "BLANK_1", "Blank_1",1301311,0.35,0.2,0,2,0.1,0.1,0.4,0.2,0.1,0,1,0, ,, , ,2,2,, ,0,1, ,0,112,1
100110)
N16 G0 X200 Z200
;******************************************
N17 T="FINISHING TOOL" D1
N18 M06
N19 G96 S450 M04
N20 CYCLE952("STOCK
REMOVAL_3", ,"",1301321,0.1,0.5,0,1.9,0.1,0.1,0.2,0.1,0.1,0,1,0,0, , , ,2,2, , 0,1, 0, 12,1000110)
N21 G0 X200 Z200
;*****************************************
N22 T="GROOVING TOOL" D1
N23 M06
N24 G96 S220 M04
N25 CYCLE930 (60,-70,5.856406,8,4, 0,15,15,1,1,1,1,0.2,2,1,10110, 1,30,0.15,1,0.4,0.2,2,1001010)
N26 CYCLE930 (60, -70,5.856406, 8,4, 0,15,15,1,1,1,1,0.2,2,1,10120, 1, 30,0.1,1,0.1,0.1,2,1001110)
N27 G0 X200 Z200
N28 T="THREADING TOOL_2" D1
N29 M06
N30 G97 S995 M03
N31 CYCLE99(0,48,-25, 4,4,1.226,0.1,30,0,5,0,2,1100103,4,1,0.2815,0.5,0,0,1,0,0.707831,1, , , ,2,0)
N32 CYCLE99(0,48,-25, 4,4,1.226,0.02,30,0,3,2,2,1210103,4,1,0.5,0.5,0,0,1,0,0.707831,1, , , 2,0)
N33 G0 X200 Z200

```

\section*{Turning}

\subsection*{8.18 Example: Standard machining}
```

;******************************************
N34 T="DRILL_D5" D1
N35 M06
N36 SPOS=0
N37 SETMS (2)
N38 M24 ; couple-in driven tool, machine-specific
N39 G97 S3183 M3
N40 G94 F318
N41 TRANSMIT
N42 MCALL CYCLE82(1,0,1, 10,0,0,1,11)
N43 HOLES2 (0,0,16,0,30,4,1010,0, ,,1)
N44 MCALL
N45 M25 ; couple out driven tool, machine-specific
N46 SETMS(1)
N47 TRAFOOF
N48 G0 X200 Z200
;*******************************************
N49 T="MILLER_D8"
N50 M6
N51 SPOS=0
N52 SETMS (2)
N53 M24
N54 G97 S1989 M03
N55 G95 FZ=0.15
N56 TRANSMIT
N57 POCKET3 (20,0,1,5,23,23,8,0,0,4,3,0,0,0.12,0.08,0,11,50,8,3,15,0,2,0,1,2,11100,11,111)
N58 M25
N59 TRAFOOF
N60 DIAMON
N61 SETMS (1)
N62 G0 X200 Z200
N63 M30
;**********************************************
N64 E_LAB_A_CONT_1: ;\#SM Z:3
;\#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAMOF;*GP*
G0 Z0 X30 ;*GP*
G1 Z-40 ;*GP*
Z-45 X40 ;*GP*
Z-65 ;*GP*
Z-70 X45 ;*GP*
Z-95 ;*GP*
X0 ;*GP*
Z0 ;*GP*

```
```

X30 ;*GP*
;CON,2,0.0000,1,1,MST:0,0,AX:Z,X,K,I;*GP*;*RO*;*HD*
; S,EX:0,EY:30;*GP*;*RO*;*HD*
;LL,EX:-40;*GP*;*RO*;*HD*
;LA,EX:-45,EY:40;*GP* ; *RO* ; *HD*
;LL,EX:-65;*GP*;*RO*;*HD*
;LA,EX:-70,EY:45;*GP*;*RO*;*HD*
;LL,EX:-95;*GP*;*RO*;*HD*
;LD,EY:0;*GP* ; *RO* ; *HD*
;LR,EX:0;*GP*;*RO*;*HD*
; LA,EX:0,EY:30;*GP*; *RO*; *HD*
;\#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_CONT_1:
N65 E_LAB_A_CONT_2: ; \#SM Z:4
;\#7__DlgK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAMOF;*GP*
G0 Z0 X0 ; *GP*
G1 X24 CHR=3 ;*GP*
Z-18.477 ; *GP*
G2 Z-55.712 X30 K=AC(-35) I=AC(40) ;*GP*
G1 Z-80 RND=6 ;*GP*
Z-85 X45 CHR=3 ;*GP*
Z-95 ;*GP*
;CON,V64,2,0.0000,0,0,MST:0,0,AX:Z,X,K,I;*GP*;*RO*;*HD*
;S,EX:0,EY:0,ASE:90;*GP*;*RO* ; *HD*
;LU,EY:24;*GP*;*RO*;*HD*
; F,LFASE:3;*GP*;*RO*;*HD*
;LL,DIA:225/0,AT:90;*GP*;*RO*;*HD*
;ACW,DIA:210/0,EY:30,CX:-35,CY:40,RAD:23;*GP*;*RO*;*HD*
;LL,EX:-80;*GP*;*RO*;*HD*
;R,RROUND: 6; *GP* ; *RO* ; *HD*
;LA,EX:-85,EY:45;*GP*;*RO*;*HD*
;F,LFASE:3;*GP*;*RO*;*HD*
;LL,EX:-95;*GP*;*RO*;*HD*
;\#End contour definition end - Don't change!;*GP*;*RO*;*HD*
E_LAB_E_CONT_2:

```

\title{
Programming technology functions (cycles)
}

\section*{\(9.1 \quad\) Drilling}

\subsection*{9.1.1 General}

\section*{General geometry parameters}
- Retraction plane RP and reference point ZO

Normally, reference point Z0 and retraction plane RP have different values. The cycle assumes that the retraction plane is in front of the reference point.

\section*{Note}

If the values for reference point and retraction planes are identical, a relative depth specification is not permitted. Error message "Reference plane defined incorrectly" is output and the cycle is not executed.
This error message is also output if the retraction plane is located after the reference point, i.e. its distance to the final drilling depth is smaller.
- Safety clearance SC

Acts in relation to the reference point. The direction in which the safety clearance is active is automatically determined by the cycle.
- Drilling depth

Depending on the selection of the drill shank or drill tip or the centering diameter, the programmed drilling depth refers to the following for cycles with a selection field:
- Tip (drilling depth in relation to the tip)

The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1.
- Shank (drilling depth in relation to the shank)

The drill is inserted into the workpiece until the drill shank reaches the value programmed for \(\mathrm{Z1}\). The angle entered in the tool list is taken into account.
- Diameter (centering in relation to the diameter, only for CYCLE81)

The diameter of the centering hole is programmed at Z 1 . In this case, the tip angle of the tool must be specified in the tool list. The drill is inserted into the workpiece until the specified diameter is reached.

\section*{Drilling positions}

The cycle assumes the tested hole coordinates of the plane.

The hole centers should therefore be programmed before or after the cycle call as follows (see also Section, Cycles on single position or position pattern (MCALL)):
- A single position should be programmed before the cycle call
- Position patterns (MCALL) should be programmed after the cycle call
- as drilling pattern cycle (line, circle, etc.) or
- as a sequence of positioning blocks for the hole centers

\section*{See also}

Selection of the cycles via softkey (Page 226)

\subsection*{9.1.2 Centering (CYCLE81)}

\section*{Function}

With the "Centering" function, the tool drills with the programmed spindle speed and feedrate either:
- Down to the programmed final drilling depth or
- So deep until the programmed diameter of the centering is reached

The tool is retracted after a programmed dwell time has elapsed.

\section*{Clamping the spindle}

For ShopTurn, "Clamp spindle" function can be set up by the machine manufacturer.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Approach/retraction}
1. The tool moves with G 0 to safety clearance of the reference point.
2. Inserted into the workpiece with G 1 and the programmed feedrate F until the depth or the centering diameter is reached.
3. On expiry of a dwell time DT , the tool is retracted at rapid traverse GO to the retraction plane.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Drill.

Centering
2. Press the "Drilling" softkey.
3. Press the "Centering" softkey.

The "Centering" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & mm/min \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining position U (only for G code) & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline Z0 (only for G code) & Reference point Z & mm \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Position U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
D \\
(0) \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Centering \\
0
\end{tabular} & \begin{tabular}{l}
- Diameter (centered with reference to the diameter) \\
The angle for the center drill entered in the tool list is applied. \\
- Tip (centered with reference to the depth) \\
The drill is inserted into the workpiece until the programmed insertion depth is reached.
\end{tabular} & \\
\hline \(\varnothing\) & It is inserted into the workpiece until the diameter is correct. - (for diameter centering only) & mm \\
\hline & \begin{tabular}{l}
Drilling depth (abs) or drilling depth in relation to Z0 (inc) \\
It is inserted into the workpiece until it reaches Z1. - (for tip centering only)
\end{tabular} & mm \\
\hline & \begin{tabular}{l}
- Dwell time (at final drilling depth) in seconds \\
- Dwell time (at final drilling depth) in revolutions
\end{tabular} & \[
\begin{aligned}
& \mathrm{s} \\
& \mathrm{rev}
\end{aligned}
\] \\
\hline
\end{tabular}

\subsection*{9.1.3 Drilling (CYCLE82)}

\section*{Function}

With the "Drilling" function, the tool drills with the programmed spindle speed and feedrate down to the specified final drilling depth (shank or tip).
The tool is retracted after a programmed dwell time has elapsed.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{See also}

Clamping the spindle (Page 247)

\section*{Approach/retraction}
1. The tool moves with G 0 to safety clearance of the reference point.
2. The tool is inserted into the workpiece with G 1 and the programmed feedrate F until it reaches the programmed final depth \(\mathrm{Z1}\).
3. When a dwell time DT expires, the tool is retracted at rapid traverse G0 to the retraction plane.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.

Drill.

Drilling
Reaming

Drilling
3. Press the "Drilling Reaming" softkey.
4. Press the "Drilling" softkey.

The "Drilling" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{InputU} & \multicolumn{4}{|l|}{- Complete} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}

\subsection*{9.1 Drilling}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
Machining posi- \\
tion (only for G \\
code)
\end{tabular} & \begin{tabular}{l} 
- Single position \\
Drill hole at programmed position \\
Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline \begin{tabular}{l} 
Z0 (only for G \\
code)
\end{tabular} & Reference point Z
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Through drilling & \begin{tabular}{l}
- Yes \\
Through drilling with feedrate FD \\
- No
\end{tabular} & \\
\hline ZD - (only for through drilling "yes") & Depth for feedrate reduction (abs) or depth for feedrate reduction in relation to Z1 (inc) & mm \\
\hline \multirow[t]{3}{*}{FD - (only for through drilling "yes")} & Reduced feedrate for through drilling referred to drilling feedrate F & \% \\
\hline & Feedrate for through drilling (ShopTurn) & \(\mathrm{mm} / \mathrm{min}\) or \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & Feedrate for through drilling (G code) & Distance/min or distance/rev \\
\hline DT - (only for through drilling "no") & \begin{tabular}{l}
- Dwell time (at final drilling depth) in seconds \\
- Dwell time (at final drilling depth) in revolutions
\end{tabular} & S rev \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \begin{tabular}{l}
Input \\
U
\end{tabular} & & \multicolumn{4}{|l|}{- Simple} \\
\hline RP & Retraction plane & mm & T & Tool name & \\
\hline & & & D & Cutting edge number & \\
\hline & & & F U & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& 0
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l} 
Machining \\
position (only for G \\
code)
\end{tabular} & \begin{tabular}{l} 
Single position \\
Drill hole at programmed position. \\
\(\mathbf{U}\)
\end{tabular} & \begin{tabular}{l} 
Position pattern \\
Position with MCALL
\end{tabular} \\
\hline \begin{tabular}{l} 
Machining \\
surface
\end{tabular} & - Face C & \\
\(\mathbf{U}\) & Pace \(Y\) \\
(only for ShopTurn) & - Peripheral surface C & \\
\hline
\end{tabular}

\subsection*{9.1 Drilling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l}
Position \\
0 (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline ( 0 (only for ShopTurn) & \begin{tabular}{l}
Clamp/release spindle \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline \begin{tabular}{l}
Drilling depth \\
O
\end{tabular} & \begin{tabular}{l}
- Shank (drilling depth in relation to the shank) \\
The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z 1 . The angle entered in the tool list is taken into account. \\
- Tip (drilling depth in relation to the tip) \\
The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1. \\
Note: If it is not possible to define an angle for the drill in the tool management, it will not be possible to select tip or shank (always tip, 0 field).
\end{tabular} & \\
\hline Z0 (only for G code) & Reference point Z & mm \\
\hline & Drilling depth (abs) or drilling depth in relation to ZO (inc). It is inserted into the workpiece until it reaches Z 1 . & \\
\hline DTU & Dwell time at final drilling depth & \[
\begin{array}{|l|l|}
\hline \text { spov }
\end{array}
\] \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline Predrilling & & & \\
\hline ZA & Predrilling depth & & \\
\hline FA & Reduced predrilling feedrate & & \\
\hline Through drilling & & & \\
\hline ZD & Depth for reduced feedrate & & \\
\hline FD & Reduced through drilling feedrate & & \\
\hline
\end{tabular}

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.1.4 Reaming (CYCLE85)}

\section*{Function}

With the "Reaming" cycle, the tool is inserted in the workpiece with the programmed spindle speed and the feedrate programmed at \(F\).
If Z 1 has been reached and the dwell time expired, the reamer is retracted at the programmed retraction feedrate to the retraction plane.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Approach/retraction}
1. The tool moves with G 0 to safety clearance of the reference point.
2. The tool is inserted into the workpiece with the programmed feedrate \(F\) until it reaches the final depth Z1.
3. Dwell time DT at final drilling depth.
4. Retraction to retraction plane with programmed retraction feedrate FR.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Drill.

Drilling Reaming Reaming
2. Press the "Drilling" softkey.
3. Press the "Drilling Reaming" softkey.
4. Press the "Reaming" softkey.

The "Reaming" input window opens.

Turning

\subsection*{9.1 Drilling}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline F & Feedrate & * & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining position U (only for G code) & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline Z0 (only for G code) & Reference point \(Z\) & mm \\
\hline FR (only for G code) & Feedrate during retraction & * \\
\hline FR (only for ShopTurn) & Feedrate during retraction & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
a \\
(0) \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline Z1 & Drilling depth (abs) or drilling depth in relation to ZO (inc) It is inserted into the workpiece until it reaches Z 1 . & mm \\
\hline \[
\begin{aligned}
& \text { DT } \\
& \mathrm{U}
\end{aligned}
\] & \begin{tabular}{l}
- Dwell time (at final drilling depth) in seconds \\
- Dwell time (at final drilling depth) in revolutions
\end{tabular} & \begin{tabular}{l}
s \\
rev
\end{tabular} \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\subsection*{9.1.5 Boring (CYCLE86)}

\section*{Function}

With the "Boring" cycle, the tool approaches the programmed position in rapid traverse, allowing for the retraction plane and safety clearance. It is then inserted into the workpiece at the feedrate programmed under \(F\) until it reaches the programmed depth (Z1). There is an oriented spindle stop with the SPOS command. After the dwell time has elapsed, the tool is retracted either with or without lift of the tool.

\section*{Note}

If, for example, swiveling or mirroring has been performed with CYCLE800 before machining, the SPOS command must be adapted so that the spindle position acts synchronously with DX and DY.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

Lift
When lifting, define the amount of lift \(D\) and the tool orientation angle \(\alpha\).

\section*{Note}

The "Boring" cycle can be used if the spindle to be used for the boring operation is technically able to go into position-controlled spindle operation.

\section*{See also}

\section*{Approach/retraction}
1. The tool moves with G 0 to safety clearance of the reference point.
2. Travel to the final drilling depth with G 1 and the speed and feedrate programmed before the cycle call.
3. Dwell time at final drilling depth.
4. Oriented spindle hold at the spindle position programmed under SPOS.
5. With the "Lift" selection, the cutting edge retracts from the hole edge with GO in up to three axes.
6. Retraction with \(G 0\) to the safety clearance of the reference point.
7. Retraction to retraction plane with G 0 to drilling position in the two axes of the plane (coordinates of the hole center point).

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Boring

Drilling
Reaming

Boring
2. Press the "Drilling" softkey.
3. Press the softkey "Boring" for G code.
- OR -
3. Press the softkeys "Drilling Reaming" and "Boring" for ShopTurn

The "Boring" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multirow[t]{2}{*}{\[
\begin{aligned}
& \hline \text { PL } \\
& \mathrm{O}
\end{aligned}
\]} & \multirow[t]{2}{*}{Machining plane} & & \multirow[t]{2}{*}{T} & \multirow[t]{2}{*}{Tool name} & \\
\hline & & & & & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline \multirow[t]{2}{*}{SC} & Safety clearance & mm & \[
\begin{aligned}
& F \\
& \hline
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{array}{|l}
\hline \mathrm{S} / \mathrm{V} \\
\mathrm{U}
\end{array}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining position \\
U \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position. \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline \begin{tabular}{l}
DIR \\
0 \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
Direction of rotation \\
- \(\quad \checkmark\) \\
- \(\quad 2\)
\end{tabular} & \\
\hline Z0 (only for G code) & Reference point Z & mm \\
\hline \begin{tabular}{l}
Machining surface \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Position U \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
C \\
(0) \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline Z1 U & Drilling depth (abs) or drilling depth in relation to Z0 (inc) & mm \\
\hline \[
\begin{aligned}
& \text { DT } \\
& \mathrm{U}
\end{aligned}
\] & \begin{tabular}{l}
- Dwell time at final drilling depth in seconds \\
- Dwell time at final drilling depth in revolutions
\end{tabular} & \[
\begin{array}{|l}
\mathrm{s} \\
\text { rev }
\end{array}
\] \\
\hline SPOS & Spindle stop position & Degrees \\
\hline \begin{tabular}{l}
Lift mode \\
0
\end{tabular} & \begin{tabular}{l}
- Do not lift off contour \\
The cutting edge is not fully retracted, but traverses back to the safety clearance in rapid traverse. \\
- Lift \\
The cutting edge is retracted from the hole edge and then moved back to the retraction plane.
\end{tabular} & \\
\hline DX (only G Code) & Retraction distance in the X direction (incremental) - (for lift-off only) & mm \\
\hline DY (only G code) & Retraction distance in the Y direction (incremental) - (for lift-off only) & mm \\
\hline DZ (only G code) & Retraction distance in the Z direction (incremental) - (for lift-off only) & mm \\
\hline D (only ShopTurn) & Retraction distance (incremental) - (for lift only) & mm \\
\hline
\end{tabular}

\subsection*{9.1.6 Deep-hole drilling 1 (CYCLE83)}

\section*{Function}

With the "Deep-hole drilling 1" cycle, the tool is inserted in the workpiece with the programmed spindle speed and feedrate in several infeed steps until the depth Z1 is reached. The following can be specified:
- Number of infeed steps constant or decreasing (via programmable degression factor)
- Chip breaking without lifting or swarf removal with tool retraction
- Feedrate factor for 1 st infeed to reduce the feedrate or increase the feedrate (e.g. if a hole has already be predrilled)
- Dwell times
- Depth in relation to drill shank of drill tip

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction during chip breaking}
1. The tool moves with G 0 to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate F = F • FD1 [\%] up to the first infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool is retracted by retraction distance \(V 2\) for chip breaking and drills up to the next infeed depth with programmed feedrate \(F\).
5. Step 4 is repeated until the final drilling depth Z 1 is reached.
6. Dwell time at final drilling depth DT.
7. The tool retracts to the retraction plane at rapid traverse.

\section*{Approach/retraction during stock removal}
1. The tool moves with G 0 to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate F = F • FD1 [\%] up to the first infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool retracts from the workpiece for the stock removal with rapid traverse to the safety clearance.
5. Dwell time at starting point DTS.
6. Approach of the last drilling depth with G0, reduced by the clearance distance V3.
7. Drilling is then continued to the next drilling depth.
8. Steps 4 to 7 are repeated until the programmed final drilling depth Z 1 is reached.
9. Dwell time at final drilling depth.
10. The tool retracts to the retraction plane at rapid traverse.

\section*{Procedure}

Deep hole drilling 1
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Deep-hole drilling" and "Deep-hole drilling 1" softkeys. The "Deep-hole drilling 1" input window opens.

\subsection*{9.1 Drilling}

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{O}
\end{aligned}
\] & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{s} / \mathrm{V} \\
& 0
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining position U (only for G code) & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline Z0 (only for G code) & Reference point Z & mm \\
\hline Machining surface 0 (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
(0) \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline Machining U & \begin{tabular}{l}
- Swarf removal \\
The drill is retracted from the workpiece for swarf removal. \\
- Chip breaking \\
The drill is retracted by the retraction distance V2 for chip breaking
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Drilling depth 0 & \begin{tabular}{l}
- Shank (drilling depth in relation to the shank) \\
The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z 1 . The angle entered in the tool list is taken into account. \\
- Tip (drilling depth in relation to the tip) \\
The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z 1 . \\
Note: If it is not possible to define an angle for the drill in the tool management, it will not be possible to select tip or shank (always tip, 0 field).
\end{tabular} & \\
\hline \[
\begin{aligned}
& \mathrm{Z} 1 \\
& \mathrm{U}
\end{aligned}
\] & Drilling depth (abs) or drilling depth in relation to \(\mathrm{Z0}\) (inc). It is inserted into the workpiece until it reaches Z 1 . & mm \\
\hline \[
\begin{array}{|l|}
\hline \text { D U } \\
\text { (only for G code) }
\end{array}
\] & First drilling depth (abs) or first drilling depth in relation to Z0 (inc). & mm \\
\hline D (only for ShopTurn) & Maximum depth infeed. & mm \\
\hline FD1 & Percentage for the feedrate at the first infeed. & \% \\
\hline \[
\begin{aligned}
& \mathrm{DF} \\
& \mathrm{U}
\end{aligned}
\] & \begin{tabular}{l}
Infeed: \\
- Degression amount by which each additional infeed is reduced. \\
- Percentage for each additional infeed. \\
DF \(=100 \%\) : Infeed increment remains constant. \\
DF < 100\%: Infeed increment is reduced in direction of final drilling depth. \\
Example: Last infeed was 4 mm ; DF is \(80 \%\) \\
Next infeed \(=4 \times 80 \%=3.2 \mathrm{~mm}\) \\
Next infeed \(=3.2 \times 80 \%=2.56 \mathrm{~mm}\), etc.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline V1 & \begin{tabular}{l}
Minimum infeed - (only for DF in \%) \\
Parameter V1 is only available if \(\mathrm{DF}<100\) has been programmed. \\
If the infeed increment becomes very small, a minimum infeed can be programmed in parameter " V 1 ". \\
V1 < Infeed increment: The tool is inserted by the infeed increment. \\
\(\mathrm{V} 1>\) Infeed increment: The tool is inserted by the infeed value programmed under V1.
\end{tabular} & mm \\
\hline V2 & Retraction distance after each machining step - (for chip breaking only). Distance by which the drill is retracted for chip breaking. \(\mathrm{V} 2=0\) : The tool is not retracted but is left in place for one revolution. & mm \\
\hline Clearance distance (for swarf removal only) & \begin{tabular}{l}
- Manual \\
The clearance distance must be entered manually. \\
- Automatic \\
The clearance distance is calculated by the cycle.
\end{tabular} & \\
\hline V3 & \begin{tabular}{l}
Clearance distance - (for swarf removal only and manual limit distance) \\
Distance to the last infeed depth that the drill approaches in rapid traverse after swarf removal.
\end{tabular} & mm \\
\hline
\end{tabular}

\subsection*{9.1 Drilling}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
DTB (only for G \\
code) \\
\(\mathbf{U}\)
\end{tabular} & \begin{tabular}{l} 
- Dwell time at drilling depth in seconds \\
- Dwell time at drilling depth in revolutions \\
Note: \\
DT > 0: The programmed value is effective \\
DT \(=0:\) The same value is effective as programmed under DTB (DT = DTB)
\end{tabular} & \begin{tabular}{l}
s \\
rev
\end{tabular} \\
\hline \begin{tabular}{lll} 
DT \\
\(\boldsymbol{U}\)
\end{tabular} & \begin{tabular}{l} 
Dwell time at final drilling depth in seconds
\end{tabular} & \\
\hline \begin{tabular}{l} 
DTS (only for G \\
code) \(\mathbf{U}\)
\end{tabular} & \begin{tabular}{l} 
- Dwell time at final drilling depth in revolutions
\end{tabular} & \begin{tabular}{l}
s \\
rev
\end{tabular} \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parmeters} \\
\hline \multicolumn{2}{|l|}{Input U} & \multicolumn{4}{|l|}{- simple} \\
\hline RP & Retraction plane & mm & T & Tool name & \\
\hline & & & D & Cutting edge number & \\
\hline & & & FO & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U} \\
& \hline
\end{aligned}
\] & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l} 
Machining \\
position \\
\(\mathbf{U}\)
\end{tabular} & \begin{tabular}{l} 
- Single position \\
Drill hole at programmed position. \\
Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline \begin{tabular}{l} 
Machining \\
\(\mathbf{U}\)
\end{tabular} & \begin{tabular}{l} 
- Swarf removal \\
The drill is retracted from the workpiece for swarf removal. \\
Chipbreaking \\
The drill is retracted by the retraction distance V2 for chipbreaking.
\end{tabular} & \\
\hline \begin{tabular}{l} 
Zo (only for G \\
code)
\end{tabular} & \begin{tabular}{l} 
Reference point \(Z\)
\end{tabular} & \\
\hline \begin{tabular}{l} 
Machining \\
surface (only for \\
ShopTurn) \\
\(\mathbf{U}\)
\end{tabular} & \begin{tabular}{l} 
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & mm \\
\hline
\end{tabular}
\(\left.\begin{array}{|l|l|l|}\hline \text { Parameter } & \text { Description } & \\ \hline \begin{array}{l}\text { Position (only for } \\ \text { ShopTurn) U }\end{array} & \bullet \quad \text { At the front (face) } & \text { At the rear (face) } \\ & \bullet \quad \text { Outside (peripheral surface) } & \text { Inside (peripheral surface) }\end{array}\right)\)

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline Drilling depth & Drilling depth in relation to the tip & Tip & \\
\hline FD1 & Percentage for the feedrate for the first infeed & \(90 \%\) & x \\
\hline PF & \begin{tabular}{l} 
Percentage for each additional infeed (for swarf removal \\
only)
\end{tabular} & \(90 \%\) & x \\
\hline V1 & Minimum indeed & 1.2 mm & x \\
\hline V2 & Retraction distance after each machining step & 1.4 mm & x \\
\hline Clearance distance & The clearance distance is calculated by the cycle & Automatic & x \\
\hline \begin{tabular}{l} 
DBT (only for G \\
code)
\end{tabular} & Dwell time at drilling depth & 0.6 s & x \\
\hline PT & Dwell time at final drilling depth & 0.6 s & x \\
\hline \begin{tabular}{l} 
DTS (only for G \\
code)
\end{tabular} & Dwell time for swarf removal (for swarf removal only) & 0.6 s & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.1.7 Deep-hole drilling 2 (CYCLE830)}

\section*{Function}

The cycle "Deep-hole drilling 2" covers the complete functionality of "Deep-hole drilling 1". in addition, the cycle provides the following functions:
- Predrilling with reduced feedrate
- Taking into account a pilot hole
- Soft first cut when entering the material
- Drilling to the final depth in one cut
- Through drilling with reduced feedrate
- Control for switching-in and switching-out the coolant

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction during chipbreaking}
1. The tool moves with GO to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate F = F • FD1 [\%] up to the 1st infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool is retracted by retraction distance V 2 for chipbreaking and drills up to the next infeed depth with programmed feedrate F.
5. Step 4 is repeated until the final drilling depth Z 1 is reached.
6. Dwell time at final drilling depth DT.
7. The tool retracts to the retraction plane at rapid traverse.

\section*{Approach/retraction during stock removal}
1. The tool moves with G 0 to safety clearance of the reference point.
2. The tool drills with the programmed spindle speed and feedrate \(F=F \cdot F D 1\) [\%] up to the 1st infeed depth.
3. Dwell time at drilling depth DTB.
4. The tool retracts from the workpiece for the stock removal with rapid traverse to the safety clearance.
5. Dwell time at starting point DTS.
6. Approach of the last drilling depth with G0, reduced by the clearance distance V3.
7. Drilling is then continued to the next drilling depth.
8. Steps 4 to 7 are repeated until the programmed final drilling depth Z 1 is reached.
9. The tool retracts to the retraction plane at rapid traverse.

\section*{Deep-hole drilling at the entrance to the hole}

The following versions are available for deep-hole drilling 2 :
- Deep-hole drilling with/without predrilling
- Deep-hole drilling with pilot hole

\section*{Note}

Predrilling or pilot hole mutually exclude one another.

\section*{Predrilling}

For predrilling, the reduced feedrate (FA) is used up to the predrilling depth (ZA) and then the drilling feedrate is used. When drilling with several infeed steps, the predrilling depth must be located between the reference point and the 1st drilling depth.

\section*{Through drilling}

For a through-hole, starting from the remaining drilling depth (ZD), a reduced feedrate (FD) is used.

\section*{Pilot hole}

The cycle optionally takes into account the depth of a pilot hole. This can be programmed with abs/inc - or a multiple of the hole diameter ( 1.5 to \(5^{*} \mathrm{D}\) is typical, for example) - and is assumed that it is available.

If a pilot hole exists, then the 1st drilling depth must be located between the pilot hole and the final drilling depth. The tool enters the pilot hole with reduced feedrate and reduced speed; these values can be programmed.

\section*{Direction of spindle rotation}

The direction of rotation of the spindle, with which the tool enters and withdraws from the pilot hole can be programmed as follows:
- with stationary spindle
- with clockwise rotating spindle
- with counterclockwise rotating spindle

This avoids long or thin drills from being broken.

\section*{Horizontal drilling}

For horizontal drilling using spiral drills, entering the pilot hole is improved if the cutting edges of the drill are also in the horizontal position. To support this, the alignment of the drill in the spindle can be programmed for a specific position (SPOS).

The feedrate is stopped before reaching the pilot hole depth, the speed increased to the drilling speed and the coolant switched in.

\section*{Soft first cut into the material}

The entry into the material can be influenced, depending on the tool and the material.
- The predrilling feed rate is maintained to a programmable first feed distance.
- An additional programmable feed distance is used to continuously increase the feedrate (with FLIN) to the drilling feed rate.
- For chip breaking/swarf removal, this mechanism is effective at each first cut. This means that when removing swarf, no clearance distance (V3) applies - and for chip breaking, the retraction distance is not effective (V2). These parameters are not displayed in the screen form. In these cases, the depth (ZS1) of the soft first cut is effective as "clearance distance" or as "retraction amount".

\section*{Deep-hole drilling at the exit from the hole}

It makes sense to reduce the feedrate when for through drilling the exit is inclined with respect to the tool axis.
- Through drilling "no"

The machining feedrate is used when drilling to the final drilling depth. You then have the option of programming a dwell time at the drilling depth.
- Through drilling "yes"

Up to the remaining drilling depth, program drilling with the drilling feedrate; from that point onwards, with a reduced feedrate.
The feedrate is not stopped, instead FLIN is employed (linear feed rate) in order to prevent the drill motion from being excited.

\section*{Retraction}

Retraction can be realized at the pilot hole depth or the retraction plane.
- Retraction to the retraction plane is realized with G0 or feedrate, programmable speed as well as direction of rotation respectively stationary spindle.
- For retraction at the pilot hole depth, subsequent retraction and insertion are realized with the same data.

\section*{Note}

\section*{Direction of spindle rotation}

The direction of spindle rotation is not reversed; however, where necessary, can be stopped.

\section*{Coolant}

The technology and tools require that also in the G code, the control for the coolant is supported.
- Coolant on

Switch on at ZO + safety clearance or at the pilot hole depth (if a pilot hole is being used)
- Coolant off

Always switch off at the final drilling depth
- Programming in the \(G\) code

An executable block ( M command or subprogram call), which can be programmed as string.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Drill.
Deep hole
drilling

Deep hole drilling 2

\subsection*{9.1 Drilling}

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{\begin{tabular}{l}
Input \\
U
\end{tabular}} & \multicolumn{4}{|l|}{- Complete} \\
\hline PL U & Machining plane & & & & \\
\hline RP & Retraction plane & mm & T & Tool name & \\
\hline SC & Safety clearance & mm & DU & Cutting edge number & \\
\hline F U & & & FO & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline \multirow[t]{2}{*}{\[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\]} & Direction of spindle rotation & & \multirow[t]{2}{*}{\[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\]} & \multirow[t]{2}{*}{Spindle speed or constant cutting rate} & \multirow[t]{2}{*}{rpm \(\mathrm{m} / \mathrm{min}\)} \\
\hline & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
Distance/min
\end{tabular} & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining position U (only G code) & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position \\
- Position pattern with MCALL
\end{tabular} & \\
\hline Z0 (only G code) & Reference point \(Z\) & mm \\
\hline Machining surface U (only ShopTurn) & \begin{tabular}{l}
- Face \\
- Face B \\
- Peripheral
\end{tabular} & \\
\hline Drilling depth U & \begin{tabular}{l}
- Shank (drilling depth in relation to the shank) \\
The drill is inserted into the workpiece until the drill shank reaches the value programmed for Z 1 . The angle entered in the tool list is taken into account. \\
- Tip (drilling depth in relation to the tip) \\
The drill is inserted into the workpiece until the drill tip reaches the value programmed for Z1.
\end{tabular} & \\
\hline \[
\begin{aligned}
& \mathrm{Z} 1 \\
& \mathrm{U}
\end{aligned}
\] & Final drilling depth (abs) or final drilling depth in relation to ZO (inc). It is inserted into the workpiece until it reaches Z 1 . & mm \\
\hline Coolant on (only G code) & M function to switch on the coolant. & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline Technology at the entrance to the hole & \multicolumn{2}{|l|}{\begin{tabular}{l}
Selecting the drilling feedrate \\
- Without predrilling Drilling with feedrate F \\
- With predrilling Drilling with feedrate FA \\
- With pilot hole Insertion in the pilot hole with feedrate FP.
\end{tabular}} & \\
\hline ZP - (only for pilot hole) U & \multicolumn{2}{|l|}{Depth of the pilot hole as a factor of the bore diameter Depth of the pilot hole in relation to ZO (inc) or depth of the pilot hole (abs)} & \[
\begin{aligned}
& * \varnothing \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \multirow[t]{3}{*}{FP - (only for pilot hole) O} & \multicolumn{2}{|l|}{First cut feedrate as a percentage of the drilling feedrate} & \% \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (ShopTurn)} & \(\mathrm{mm} /\) rev or mm/min \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (G code)} & distance/min or distance/rev \\
\hline \begin{tabular}{l}
SP \\
(only for pilot hole)
\end{tabular} & \({ }^{8} \times\) & Spindle position during approach (spindle off) & Degrees \\
\hline ZA - (only for predrilling) U & \multicolumn{2}{|l|}{Predrilling depth (abs) or predrilling depth in relation to Z0 (inc)} & mm \\
\hline \multirow[t]{3}{*}{FA - (only for predrilling) U} & \multicolumn{2}{|l|}{Predrilling feedrate as a percentage of the drilling feedrate} & \% \\
\hline & \multicolumn{2}{|l|}{Predrilling feedrate (ShopTurn)} & \(\mathrm{mm} / \mathrm{min}\) or mm/rev. \\
\hline & \multicolumn{2}{|l|}{Predrilling feedrate (G code)} & distance/min or distance/rev \\
\hline Soft first cut 0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Yes \\
First cut with feedrate FS \\
- No \\
First cut with drilling feedrate
\end{tabular}} & \\
\hline \begin{tabular}{l}
ZS1 \\
(only "Yes" for soft first cut)
\end{tabular} & \multicolumn{2}{|l|}{Depth of each first cut with constant first cut feedrate FS (inc)} & mm \\
\hline \begin{tabular}{l}
ZS2 \\
(only "Yes" for soft first cut)
\end{tabular} & \multicolumn{2}{|l|}{Depth of each first cut for feedrate increase (inc)} & mm \\
\hline \multirow[t]{3}{*}{FS (only "Yes" for soft first cut)} & \multicolumn{2}{|l|}{First cut feedrate as a percentage of the drilling feedrate} & \% \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (ShopTurn)} & \(\mathrm{mm} / \mathrm{min}\) or \(\mathrm{mm} / \mathrm{rev}\). \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (G code)} & distance/min or distance/rev \\
\hline Drilling interruption 0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
- One cut \\
- Chip breaking \\
- Swarf removal \\
- Chip breaking and swarf removal
\end{tabular}} & \\
\hline D U & \multicolumn{2}{|l|}{First drilling depth (abs) or first drilling depth in relation to Z0 (inc).} & mm \\
\hline
\end{tabular}

\subsection*{9.1 Drilling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline FD1 & Percentage for the feedrate at the first infeed. & \% \\
\hline \[
\begin{aligned}
& \hline \mathrm{DF} \\
& \mathrm{O}
\end{aligned}
\] & \begin{tabular}{l}
Infeed: \\
- Degression amount by which each additional infeed is reduced. \\
- Percentage for each additional infeed. \\
DF \(=100 \%\) : Infeed increment remains constant. \\
DF < 100\%: Infeed increment is reduced in direction of final drilling depth. \\
Example: Last infeed was 4 mm ; DF is \(80 \%\) \\
Next infeed \(=4 \times 80 \%=3.2 \mathrm{~mm}\) \\
Next infeed \(=3.2 \times 80 \%=2.56 \mathrm{~mm}\), etc.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline V1 & \begin{tabular}{l}
Minimum infeed - (only for DF in \%) \\
Parameter V1 is only available if \(\mathrm{DF}<100\) has been programmed. \\
If the infeed increment becomes very small, a minimum infeed can be programmed in parameter "V1". \\
V1 < Infeed increment: The tool is inserted by the infeed increment. \\
\(\mathrm{V} 1>\) Infeed increment: The tool is inserted by the infeed value programmed under V1.
\end{tabular} & mm \\
\hline \begin{tabular}{l}
V2 \\
(only for chip breaking and soft first cut "no")
\end{tabular} & \begin{tabular}{l}
Retraction distance after each machining step. \\
Distance by which the drill is retracted for chip breaking. \\
\(\mathrm{V} 2=0\) : The tool is not retracted but is left in place for one revolution.
\end{tabular} & mm \\
\hline \[
\begin{array}{|l|}
\hline \text { DTB } \\
0
\end{array}
\] & \begin{tabular}{l}
- Dwell time at drilling depth in seconds \\
- Dwell time at drilling depth in revolutions
\end{tabular} & s
rev \\
\hline Clearance distance (only for swarf removal and soft first cut "no") & \begin{tabular}{l}
- Manual \\
The clearance distance must be entered manually. \\
- Automatic \\
The clearance distance is calculated by the cycle.
\end{tabular} & \\
\hline V3 - (for "manual" clearance distance only) & Clearance distance (inc) & mm \\
\hline N - (only for "chip breaking and swarf removal") & Number of chip breaking strokes before each swarf removal operation. & \\
\hline Retraction for swarf removal U & \begin{tabular}{l}
- Swarf removal at the pilot hole depth \\
- Swarf removal at the safety clearance
\end{tabular} & \\
\hline \[
\begin{aligned}
& \mathrm{DTS} \\
& 0
\end{aligned}
\] & \begin{tabular}{l}
- Dwell time for swarf removal in seconds \\
- Dwell time for swarf removal in revolutions
\end{tabular} & s
rev \\
\hline \begin{tabular}{l}
Through drilling \\
0
\end{tabular} & \begin{tabular}{l}
- Yes \\
Through drilling with feedrate FD \\
- No
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline ZD - (only for through drilling "yes") U & Depth for feedrate reduction (abs) or depth for feedrate reduction in relation to Z 1 (inc). & mm \\
\hline \multirow[t]{3}{*}{FD - (only for through drilling "yes") U} & Feedrate for through drilling referred to drilling feedrate F. & \% \\
\hline & Feedrate for through drilling (ShopTurn). & \(\mathrm{mm} / \mathrm{min}\) or mm/rev. \\
\hline & Feedrate for through drilling (G code). & distance/min or distance/rev \\
\hline DT - (only for through drilling "no") U & \begin{tabular}{l}
- Dwell time at final depth in seconds \\
- Dwell time at final depth in revolutions
\end{tabular} & \[
\begin{aligned}
& \mathrm{s} \\
& \mathrm{U}
\end{aligned}
\] \\
\hline Retraction
\[
0
\] & \begin{tabular}{l}
- Retraction to pilot hole depth \\
- Retraction to retraction plane
\end{tabular} & \\
\hline FR & Retraction in rapid traverse & \\
\hline Direction of spindle rotation during retraction & \begin{tabular}{l}
Retraction with stationary spindle \\
- 2 \\
- S
\end{tabular} & \\
\hline SR / VR (only for selected spindle direction of rotation) & \begin{tabular}{l}
- Spindle speed for retraction referred to the drilling speed \\
- Spindle speed for retraction \\
- Constant cutting rate VR for retraction
\end{tabular} & \begin{tabular}{l}
\% \\
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline Coolant off (only G code) & M function to switch off the coolant & \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{\begin{tabular}{l}
Input \\
U
\end{tabular}} & \multicolumn{4}{|l|}{- simple} \\
\hline RP & Retraction plane & mm & T & Tool name & \\
\hline & & & DU & Cutting edge number & \\
\hline F & Feedrate & mm/min \(\mathrm{mm} / \mathrm{rev}\) & F U & Feedrate & mm/min \(\mathrm{mm} / \mathrm{rev}\) \\
\hline \[
\begin{aligned}
& S / V \\
& U
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}

\subsection*{9.1 Drilling}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline Machining position U(only G code) & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Single position \\
Drill hole at programmed position \\
- Position pattern with MCALL
\end{tabular}} & \\
\hline Z0 (only G code) & \multicolumn{2}{|l|}{Reference point Z} & mm \\
\hline \begin{tabular}{l}
Machining surface \\
U \\
(only Shop- \\
Turn)
\end{tabular} & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Face \\
- Face B \\
- Peripheral
\end{tabular}} & \\
\hline \[
\begin{aligned}
& \mathrm{Z} 1 \\
& \mathrm{U} \\
& \hline
\end{aligned}
\] & \multicolumn{2}{|l|}{Final drilling depth (abs) or final drilling depth in relation to ZO (inc) It is inserted into the workpiece until it reaches Z 1 .} & mm \\
\hline Coolant on (only G code) & \multicolumn{2}{|l|}{M function to switch on the coolant} & \\
\hline \[
\begin{aligned}
& \mathrm{ZP} \\
& \mathrm{U}
\end{aligned}
\] & \multicolumn{2}{|l|}{Depth of the pilot hole as a factor of the bore diameter Depth of the pilot hole in relation to ZO (inc) or depth of the pilot hole (abs)} & \[
\begin{aligned}
& * \varnothing \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \multirow[t]{3}{*}{\[
\begin{gathered}
\mathrm{FP} \\
\mathrm{U}
\end{gathered}
\]} & \multicolumn{2}{|l|}{First cut feedrate in percent of the drilling feedrate} & \% \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (ShopTurn)} & \(\mathrm{mm} / \mathrm{rev}\) or mm/min \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (G code)} & distance/min or distance/rev \\
\hline SP & X \(\times\) & Spindle position during approach (spindle off) & Degrees \\
\hline Soft section U & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Yes \\
First cut with feedrate FS \\
- No \\
First cut with drilling feedrate
\end{tabular}} & \\
\hline ZS1 (only "Yes" for soft first cut) & \multicolumn{2}{|l|}{Depth of each first cut with constant first cut feedrate FS (inc)} & mm \\
\hline \begin{tabular}{l}
ZS2 \\
(only "Yes" for soft first cut)
\end{tabular} & \multicolumn{2}{|l|}{Depth of each first cut for feedrate increase (inc)} & mm \\
\hline \multirow[t]{3}{*}{FS (only "Yes" for soft first cut)
\[
U
\]} & \multicolumn{2}{|l|}{First cut feedrate in percent of the drilling feedrate} & \% \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (ShopTurn)} & \(\mathrm{mm} / \mathrm{min}\) or \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & \multicolumn{2}{|l|}{First cut feedrate (G code)} & distance/min or distance/rev \\
\hline Through drilling U & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Yes \\
Through drilling with feedrate FD \\
- No
\end{tabular}} & \\
\hline ZD - (only for through drilling "yes") ( & \multicolumn{2}{|l|}{Depth for feedrate reduction (abs) or depth for feedrate reduction in relation to Z1 (inc)} & mm \\
\hline FD - (only for & \multicolumn{2}{|l|}{Feedrate for through drilling referred to drilling feed rate F} & \% \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|l|}
\hline \multicolumn{2}{|l|}{ G code program parameters } & ShopTurn program parameters \\
\hline \begin{tabular}{l} 
through drilling \\
"yes")
\end{tabular} & Feedrate for through drilling (ShopTurn) & \(\mathrm{mm} / \mathrm{min}\) or \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & Feedrate for through drilling (G code) & \begin{tabular}{l} 
distance \(/ \mathrm{min}\) or \\
distance \(/ \mathrm{rev}\)
\end{tabular} \\
\hline \begin{tabular}{l} 
Coolant off - \\
(only G code)
\end{tabular} & M function to switch off the coolant & \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|l|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline Drilling depth & Drilling depth referred to the shaft or tip & Tip & \\
\hline \begin{tabular}{l} 
Entrance to the \\
hole
\end{tabular} & Technology at the entrance to the hole & With pilot hole & \\
\hline ZA & Predrilling depth (inc) & 1 mm & \\
\hline FA & Predrilling feed & \(50 \%\) & \\
\hline \begin{tabular}{l} 
Drilling \\
interruption
\end{tabular} & \begin{tabular}{l} 
- 1 cut \\
- Chipbreaking \\
- Swarf removal \\
- Chipbreaking and swarf removal
\end{tabular} & & \\
\hline D & 1. Drilling depth referred to Z0 (inc.) & Percentage for the feedrate for the first infeed & 10 mm \\
\hline FD1 & \begin{tabular}{l} 
Percentage for the feedrate for each additional infeed \\
Infeed increment is continually reduced in the direction of \\
final drilling depth
\end{tabular} & \(90 \%\) & \\
\hline DF & \begin{tabular}{l} 
Minimum infeed \\
V1 < Infeed increment: The tool is inserted by the infeed
\end{tabular} & 2 mm & \\
\hline increment \\
V1 > Infeed increment: The tool is inserted by the infeed \\
value programmed under V1.
\end{tabular}
\begin{tabular}{|l|l|l|l|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
DT - (only for \\
through drilling \\
"no")
\end{tabular} & Dwell time at final depth in seconds & 0.6 s & \\
\hline Retraction & Retraction to pilot hole depth or retraction plane & \begin{tabular}{l} 
Pilot hole \\
depth
\end{tabular} & \\
\hline FR & Retraction in rapid traverse & M5 & \\
\hline \begin{tabular}{l} 
Direction of spindle \\
rotation during \\
retraction
\end{tabular} & & \(10 \%\) & \\
\hline \begin{tabular}{l} 
SR (only for se- \\
lected spindle \\
direction of rota- \\
tion)
\end{tabular} & Spindle speed for retraction referred to the drilling speed & & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.1.8 Tapping (CYCLE84, 840)}

\section*{Function}

You can machine an internal thread with the "tapping" cycle.
The tool moves to the safety clearance with the active speed and rapid traverse. The spindle stops, spindle and feedrate are synchronized. The tool is then inserted in the workpiece with the programmed speed (dependent on \%S).
You can choose between drilling in one cut, chipbreaking or retraction from the workpiece for swarf removal.
Depending on the selection in the "Compensating chuck mode" field, alternatively the following cycle calls are generated:
- With compensating chuck: CYCLE840
- Without compensating chuck: CYCLE84

When tapping with compensating chuck, the thread is produced in one cut. CYCLE84 enables tapping to be performed in several cuts if the spindle is equipped with a measuring system.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Input simple (only for G code)}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction - CYCLE840 - with compensating chuck}
1. The tool moves with \(G 0\) to safety clearance of the reference point.
2. The tool drills with G1 and the programmed spindle speed and direction of rotation to depth Z1. The feedrate F is calculated internally in the cycle from the speed and pitch.
3. The direction of rotation is reversed.
4. Dwell time at final drilling depth.
5. Retraction to safety clearance with G1.
6. Reversal of direction of rotation or spindle stop.
7. Retraction to retraction plane with G0.

\section*{Approach/retraction CYCLE84 - without compensating chuck in the "1 cut" mode}
1. Travel with \(G 0\) to the safety clearance of the reference point.
2. Spindle is synchronized and started with the programmed speed (dependent on \%S).
3. Tapping with spindle-feedrate synchronization to Z 1 .
4. Spindle stop and dwell time at drilling depth.
5. Spindle reverse after dwell time has elapsed.
6. Retraction with active spindle retraction speed (dependent on \%S) to safety clearance
7. Spindle stop.
8. Retraction to retraction plane with G0.

\section*{Approach/retraction CYCLE84 - without compensating chuck in the "swarf removal" mode}
1. The tool drills at the programmed spindle speed S (dependent on \(\% \mathrm{~S}\) ) as far as the 1 st infeed depth (maximum infeed depth D ).
2. Spindle stop and dwell time DT.
3. The tool retracts from the workpiece for the stock removal with spindle speed SR to the safety clearance.
4. Spindle stop and dwell time DT.
5. The tool then drills with spindle speed \(S\) as far as the next infeed depth.
6. Steps 2 to 5 are repeated until the programmed final drilling depth Z 1 is reached.
7. On expiry of dwell time DT, the tool is retracted with spindle speed SR to the safety clearance. The spindle stops and retracts to the retraction plane.

\section*{Approach/retraction CYCLE84 - without compensating chuck in the "chip breaking" mode}
1. The tool drills at the programmed spindle speed \(S\) (dependent on \(\%\) S) as far as the 1 st infeed depth (maximum infeed depth D).
2. Spindle stop and dwell time DT.
3. The tool retracts by the retraction distance V 2 for chip breaking.
4. The tool then drills to the next infeed depth at spindle speed S (dependent on \%S).
5. Steps 2 to 4 are repeated until the programmed final drilling depth Z 1 is reached.
6. On expiry of dwell time DT, the tool is retracted with spindle speed SR to the safety clearance. The spindle stops and retracts to the retraction plane.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}

\section*{Thread}

Tapping
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Thread" and "Tapping" softkeys.

The "Tapping" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input (only for G code)} & \multicolumn{2}{|l|}{- Complete} & & & \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{O}
\end{aligned}
\] & \multicolumn{2}{|l|}{Machining plane} & & T & Tool name & \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & D & Cutting edge number & \\
\hline SC & \multicolumn{2}{|l|}{Safety clearance} & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline & & & & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Compensating chuck mode U & \begin{tabular}{l}
- with compensating chuck \\
- without compensating chuck
\end{tabular} & \\
\hline Machining position \(\mathbf{U}\) (only for G code) & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline Z0 (only for G code) & Reference point \(Z\) & mm \\
\hline Machining - (with compensating chuck) & \begin{tabular}{l}
You can select the following technologies for tapping: \\
- with encoder \\
Tapping with spindle encoder \\
- without encoder \\
Tapping without spindle encoder - the following fields are displayed: \\
- Select the "pitch" parameter (only G code) \\
- Enter parameter "DT" (only ShopMill) \\
Note: \\
For ShopMill, the selection box is only displayed if tapping without encoder is enabled. \\
Please observe the information provided by your machine manufacturer.
\end{tabular} & \\
\hline SR (only for ShopTurn) & Spindle speed for retraction - (only for spindle speed "S") & \(\mathrm{rev} / \mathrm{min}\) \\
\hline VR (only for ShopTurn) & Constant cutting rate for retraction - (only for constant cutting rate "V") & \(\mathrm{m} / \mathrm{min}\) \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline  & \begin{tabular}{l}
Clamp/release spindle \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline Z1 O & \begin{tabular}{l}
End point of the thread (abs) or thread length (inc) - only for G code and ShopTurn machining surface "face") \\
It is inserted into the workpiece until it reaches Z 1 .
\end{tabular} & mm \\
\hline X1 U & \begin{tabular}{l}
End point of the thread (abs) or thread length (inc) - (only for ShopTurn machining surface, "peripheral") \\
It is inserted into the workpiece until X 1 is reached.
\end{tabular} & mm \\
\hline \begin{tabular}{l}
Pitch - (only machining without encoder) \\
U \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
- User input \\
Pitch is obtained from the input \\
- Active feedrate \\
Pitch is obtained from the feedrate
\end{tabular} & \\
\hline Thread (only for G code) U & \begin{tabular}{l}
Direction of rotation of the thread \\
- Right-hand thread \\
- Left-hand thread (only in mode "without compensating chuck")
\end{tabular} & \\
\hline \begin{tabular}{l}
Table \\
U
\end{tabular} & \begin{tabular}{l}
Thread table selection: \\
- without \\
- ISO metric \\
- Whitworth BSW \\
- Whitworth BSP \\
- UNC
\end{tabular} & \\
\hline \begin{tabular}{l}
Selection \\
0
\end{tabular} & \begin{tabular}{l}
Selection of table value: e.g. \\
- M3; M10; etc. (ISO metric) \\
- W3/4"; etc. (Whitworth BSW) \\
- G3/4"; etc. (Whitworth BSP) \\
- 1" - 8 UNC; etc. (UNC)
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
P \\
- (selection only possible for table selection "without")
\end{tabular} & \begin{tabular}{l}
Pitch ... \\
- \(\quad\) in MODULUS: MODULUS \(=\) Pitch/ \(\pi\) \\
- in turns per inch: Used with pipe threads, for example. \\
When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. \\
- in mm/rev \\
- in inch/rev \\
The pitch is determined by the tool used.
\end{tabular} & \begin{tabular}{l}
MODULUS \\
Turns/" \\
\(\mathrm{mm} / \mathrm{rev}\) in/rev
\end{tabular} \\
\hline aS (only for G code) & Starting angle offset - (only for tapping without compensating chuck) & Degrees \\
\hline \begin{tabular}{l}
S \\
(only for G code)
\end{tabular} & Spindle speed - (only for tapping without compensating chuck) & \(\mathrm{rev} / \mathrm{min}\) \\
\hline Machining (not in the "with compensating chuck" mode) & \begin{tabular}{l}
The following machining operations can be selected: \\
- 1 cut \\
The thread is drilled in one cut without interruption. \\
- Chipbreaking \\
The drill is retracted by the retraction amount V2 for chipbreaking. \\
- Swarf removal \\
The drill is retracted from the workpiece for swarf removal.
\end{tabular} & \\
\hline D & Maximum depth infeed - (only when used without compensating chuck, swarf removal or chipbreaking) & mm \\
\hline \begin{tabular}{l}
Retraction \\
U
\end{tabular} & \begin{tabular}{l}
Retraction distance - (for chipbreaking only) \\
- Manual \\
Retraction distance after each machining step (V2) \\
- Automatic \\
The tool is retracted by one revolution.
\end{tabular} & \\
\hline V2 & \begin{tabular}{l}
Retraction distance after each machining step - (only without compensating chuck, chipbreaking and manual retraction) \\
Distance by which the drill is retracted for chipbreaking.
\end{tabular} & mm \\
\hline DT (for ShopTurn, only in the mode "with compensating chuck without encoder") & \begin{tabular}{l}
Dwell time in seconds: \\
- without compensating chuck \\
- 1 cut: Dwell time at final drilling depth \\
- Chip breaking: Dwell time at drilling depth \\
- Swarf removal: Dwell time at the drilling depth and after retraction \\
- with compensating chuck \\
- with encoder: Dwell time after drilling \\
- without encoder: Dwell time at final drilling depth
\end{tabular} & S \\
\hline \begin{tabular}{l}
SR \\
(only for G code)
\end{tabular} & Spindle speed for retraction - (only for when a compensating chuck is not used) & \(\mathrm{rev} / \mathrm{min}\) \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
SDE U \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
Direction of rotation after end of cycle: \\
- \(\quad \otimes\) \\
- 2 \\
- 5
\end{tabular} & \\
\hline Technology U & \begin{tabular}{l}
Adapting the technology: \\
- Yes \\
- Exact stop \\
- Precontrol \\
- Acceleration \\
- Spindle \\
- No \\
Note: \\
The technology fields are only displayed if their display has been enabled. \\
Please observe the information provided by your machine manufacturer.
\end{tabular} & \\
\hline Exact stop (only for technology, yes) 0 & \begin{tabular}{l}
- Empty: Behavior the same as it was before the cycle was called \\
- G601: Block advance for exact stop fine \\
- G602: Block advance for exact stop coarse \\
- G603: Block advance if the setpoint is reached
\end{tabular} & \\
\hline Precontrol (only for technology, yes) \(\mathbf{U}\) & \begin{tabular}{l}
- Empty: Behavior the same as it was before the cycle was called \\
- FFWON: with precontrol \\
- FFWOF: without precontrol
\end{tabular} & \\
\hline Acceleration (only for technology, yes) \(\mathbf{U}\) & \begin{tabular}{l}
(only in mode "without compensating chuck") \\
- Empty: Behavior the same as it was before the cycle was called \\
- SOFT: Jerk-limited (soft) acceleration of the axes \\
- BRISK: Abrupt acceleration of the axes \\
- DRIVE: Reduced axis acceleration
\end{tabular} & \\
\hline Spindle (only for technology, yes) U & \begin{tabular}{l}
(only in mode "without compensating chuck") \\
- Speed controlled: Spindle for MCALL: Speed-controlled mode \\
- Position controlled: Spindle for MCALL: Position-controlled operation
\end{tabular} & \\
\hline
\end{tabular}

\section*{Parameters in the mode "Input simple" (only for G code program)}
\begin{tabular}{|l|l|l|l|l|l|}
\hline \multicolumn{9}{|l|}{ G code program parameters } & & & \\
\hline \begin{tabular}{l} 
Input \\
\(U\)
\end{tabular} & \(\bullet\) simple & \\
\hline RP & Retraction plane & mm & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline Compensating chuck mode U & \begin{tabular}{l}
- with compensating chuck \\
- Without compensating chuck
\end{tabular} & \\
\hline Machining position
\(\qquad\) & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position. \\
- Position pattern Position with MCALL
\end{tabular} & \\
\hline Z0 & Reference point Z & mm \\
\hline & End point of the thread (abs) or thread length (inc) It is inserted into the workpiece until it reaches Z 1 . & mm \\
\hline Machining - (with compensating chuck) U & \begin{tabular}{l}
- With encoder \\
Tapping with spindle encoder \\
- without encoder \\
Tapping without spindle encoder; selection: \\
- Define "Pitch" parameter
\end{tabular} & \\
\hline Pitch - (only machining without encoder) \(U\) & \begin{tabular}{l}
- User input \\
Pitch is obtained from the input \\
- Active feedrate \\
Pitch is obtained from the feedrate
\end{tabular} & \\
\hline Thread U & \begin{tabular}{l}
Direction of rotation of the thread \\
- Right-hand thread \\
- Left-hand thread \\
(only in mode "without compensating chuck")
\end{tabular} & \\
\hline & \begin{tabular}{l}
Pitch ... \\
- in MODULUS: MODULUS \(=\) Pitch/ \(\pi\) \\
- in turns per inch: Used with pipe threads, for example. \\
When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. \\
- in \(\mathrm{mm} / \mathrm{rev}\) \\
- in inch/rev \\
The pitch is determined by the tool being used
\end{tabular} & \begin{tabular}{l}
MODULUS Turns/" \\
mm/rev \\
in/rev
\end{tabular} \\
\hline S & Spindle speed - (only for tapping without compensating chuck) & \\
\hline
\end{tabular}

\subsection*{9.1 Drilling}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l} 
Machining \(\mathbf{U}\) (not \\
for "with compen- \\
sating chuck")
\end{tabular} & \begin{tabular}{l} 
The following machining operations can be selected: \\
• cut \\
The thread is drilled in one cut without interruption. \\
Chipbreaking \\
The drill is retracted by the retraction amount V2 for chipbreaking. \\
Swarf removal \\
The drill is retracted from the workpiece for swarf removal.
\end{tabular} & \\
\hline D & \begin{tabular}{l} 
Maximum depth infeed - (only for tapping without compensating chuck, swarf removal \\
or chipbreaking)
\end{tabular} & mm \\
\hline SR & Spindle speed for retraction - (only for "without compensating chuck") & \(\mathrm{rev} / \mathrm{min}\) \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline PL & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline SC & Safety clearance & 1 mm & x \\
\hline Table & Thread table selection & without & \\
\hline\(\alpha S\) ) & Starting angle offset & \(0^{\circ}\) & \\
\hline Retraction & \begin{tabular}{l} 
Without retraction distance after each machining step - (for \\
chip breaking only)
\end{tabular} & Automatic & \\
\hline DT & Dwell time at final drilling depth & 0.6 s & x \\
\hline SDE & Direction of rotation after end of cycle & \(\otimes\) & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.1.9 Drill and thread milling (CYCLE78)}

\section*{Function}

You can use a drill and thread milling cutter to manufacture an internal thread with a specific depth and pitch in one operation. This means that you can use the same tool for drilling and thread milling, a change of tool is superfluous.
The thread can be machined as a right- or left-hand thread.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Approach/retraction}
1. The tool traverses with rapid traverse to the safety clearance.
2. If pre-drilling is required, the tool traverses at a reduced drilling feedrate to the predrilling depth defined in a setting data (ShopMill/ShopTurn). When programming in G code, the predrilling depth can be programmed using an input parameter.

\section*{Machine manufacturer}

Please also refer to the machine manufacturer's instructions.
1. The tool bores at drilling feedrate F1 to the first drilling depth D. If the final drilling depth Z1 is not reached, the tool will travel back to the workpiece surface in rapid traverse for stock removal. Then the tool will traverse with rapid traverse to a position 1 mm above the drilling depth previously achieved - allowing it to continue drilling at drill feedrate F1 at the next infeed. Parameter "DF" is taken into account from the 2nd infeed and higher (refer to the table "Parameters").
2. If another feedrate \(F R\) is required for through-boring, the residual drilling depth \(Z R\) is drilled with this feedrate.
3. If required, the tool traverses back to the workpiece surface for stock removal before thread milling with rapid traverse.
4. The tool traverses to the starting position for thread milling.
5. The thread milling is carried out (climbing, conventional or conventional + climbing) with milling feedrate F2. The thread milling acceleration path and deceleration path is traversed in a semicircle with concurrent infeed in the tool axis.

\subsection*{9.1 Drilling}

\section*{Procedure}

\section*{Drill.}

Thread
Dri+thrd milling
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" softkey.
3. Press the "Thread" and "Cut thread" softkeys.

The "Drilling and thread milling" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{O}
\end{aligned}
\] & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Machining Uposition (only for G code) & \begin{tabular}{l}
- Single position \\
Drill hole at programmed position \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline \begin{tabular}{l}
F1 U \\
(only for G code)
\end{tabular} & Drilling feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline \begin{tabular}{l}
Z0 \\
(only for G code)
\end{tabular} & Reference point Z & mm \\
\hline \begin{tabular}{l}
Machining surface U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l}
D \\
(0) \\
(only for Shop- \\
Turn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Z1 U & Thread length (inc) or end point of the thread (abs) & mm \\
\hline D & Maximum depth infeed & mm \\
\hline \[
\begin{aligned}
& \mathrm{DF} \\
& \mathrm{O}
\end{aligned}
\] & \begin{tabular}{l}
- Percentage for each additional infeed \\
\(D F=100\) : Infeed increment remains constant \\
\(\mathrm{DF}<100\) : Infeed increment is reduced in direction of final drilling depth Z 1 . \\
Example: last infeed 4 mm; DF 80\% \\
next infeed \(=4 \times 80 \%=3.2 \mathrm{~mm}\) \\
next but one infeed \(=3.2 \times 80 \%=2.56 \mathrm{~mm}\) etc. \\
- Amount for each additional infeed
\end{tabular} & \begin{tabular}{l}
\% \\
mm
\end{tabular} \\
\hline V1 & \begin{tabular}{l}
Minimum infeed - (only for DF, percentage for each additional infeed) \\
Parameter V1 is only available if \(\mathrm{DF}<100\) has been programmed. \\
If the infeed increment becomes very small, a minimum infeed can be programmed in parameter " V 1 ". \\
V1 < Infeed increment: The tool is inserted by the infeed increment \\
\(\mathrm{V} 1>\) Infeed increment: The tool is inserted by the infeed value programmed under V1.
\end{tabular} & mm \\
\hline \begin{tabular}{l}
Predrilling \\
U
\end{tabular} & \begin{tabular}{l}
Predrilling with reduced feedrate \\
- Yes \\
- No \\
The reduced drilling feedrate is obtained as follows: \\
Drilling feedrate F1 \(<0.15 \mathrm{~mm} /\) rev: Predrilling feedrate \(=30 \%\) of F1 \\
Drilling feedrate F1 \(\geq 0.15 \mathrm{~mm} / \mathrm{rev}\) : Predrilling feedrate \(=0.1 \mathrm{~mm} / \mathrm{rev}\)
\end{tabular} & \\
\hline AZ (only for G code) & Predrilling depth with reduced drilling feedrate - ("yes", only for predrilling) & mm \\
\hline Through boring U & \begin{tabular}{l}
Remaining drilling depth with drilling feedrate \\
- Yes \\
- No
\end{tabular} & \\
\hline ZR & Residual drilling depth for through boring - ("yes", only for through boring) & mm \\
\hline \[
\begin{aligned}
& \mathrm{FR} \\
& \mathrm{U}
\end{aligned}
\] & Drilling feedrate for remaining drilling depth - ("yes", only for through boring) & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline Chip removal U & \begin{tabular}{l}
Stock removal before thread milling \\
- Yes \\
- No \\
Return to workpiece surface for stock removal before thread milling.
\end{tabular} & \\
\hline Thread 0 & \begin{tabular}{l}
Direction of rotation of the thread \\
- Righthand thread \\
- Lefthand thread
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.1 Drilling}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \[
\begin{aligned}
& \hline \text { F2 } \\
& \mathrm{U} \\
& \hline
\end{aligned}
\] & Feedrate for thread milling & mm/min mm/tooth \\
\hline Table U & \begin{tabular}{l}
Thread table selection: \\
- without \\
- ISO metric \\
- Whitworth BSW \\
- Whitworth BSP \\
- UNC
\end{tabular} & \\
\hline Selection - (not for table "Without") \(U\) & \begin{tabular}{l}
Selection, table value: e.g. \\
- M3; M10; etc. (ISO metric) \\
- W3/4"; etc. (Whitworth BSW) \\
- G3/4"; etc. (Whitworth BSP) \\
- N1" - 8 UNC; etc. (UNC)
\end{tabular} & \\
\hline \begin{tabular}{l}
P U \\
- (selection only possible for "Table without selection")
\end{tabular} & \begin{tabular}{l}
Pitch ... \\
- in MODULUS: MODULUS \(=\) Pitch/ \(\pi\) \\
- in turns per inch: Used with pipe threads, for example. \\
When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. \\
- in mm/rev \\
- in inch/rev \\
The pitch is determined by the tool used.
\end{tabular} & \begin{tabular}{l}
MODULUS Turns/" \\
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev
\end{tabular} \\
\hline Z2 & \begin{tabular}{l}
Retraction amount before thread milling \\
The thread depth in the direction of the tool axis is defined using \(\mathrm{Z} 2 . \mathrm{Z2}\) is relative to the tool tip
\end{tabular} & mm \\
\hline \(\varnothing\) & Nominal diameter & mm \\
\hline Milling direction U & \begin{tabular}{l}
- Climbing: Mill thread in one cycle. \\
- Conventional: Mill thread in one cycle. \\
- Climbing - conventional: Mill thread in two cycles: rough cutting is performed by conventional milling with defined allowances, then finish cutting is performed with climb milling with milling feedrate FS.
\end{tabular} & \\
\hline \[
\begin{aligned}
& \hline \text { FS } \\
& \mathrm{O}
\end{aligned}
\] & Finishing feedrate rate - (only for climbing - conventional milling) & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline
\end{tabular}

\subsection*{9.1.10 Positions and position patterns}

\section*{Function}
- Arbitrary positions
- Position on a line, on a grid or frame
- Position on a full or pitch circle

\section*{Programming a position pattern in ShopTurn}

Several position patterns can be programmed in succession (up to 20 technologies and position patterns in total). They are executed in the order in which you program them.

\section*{Note}

The number of positions that can be programmed in a "Positions" step is limited to a maximum of 600!

The programmed technologies and subsequently programmed positions are automatically linked by the control.

\section*{Displaying and hiding positions}

You can display or hide any positions (Section "Displaying and hiding positions (Page 352)").

\section*{Approach/retraction}
1. Within a position pattern, or while approaching the next position pattern, the tool is retracted to the retraction plane and the new position or position pattern is then approached at rapid traverse.
2. With subsequent technological operations (e.g. centering - drilling - tapping), the respective drilling cycle must programmed after calling the next tool (e.g. drill) and immediately afterwards the call of the position pattern to be machined.

\section*{Tool traverse path}
- ShopTurn

The programmed positions are machined with the previously programmed tool (e.g. center drill). Machining of the positions always starts at the reference point. In the case of a grid, machining is performed first in the direction of the 1st axis and then meandering back and forth. The frame and hole circle are machined counter-clockwise.
- G codes

For G code, for rows/frames/grids, a start is always made at the next corner of the frame or grid or the end of the row. The frame and circle or pitch circle are machined counterclockwise.

\subsection*{9.1.11 Arbitrary positions (CYCLE802)}

\section*{Function}

The "Arbitrary positions" function allows you to program any positions, i.e. in rectangular or polar coordinates. Individual positions are approached in the order in which you program them.

Press "Delete all" softkey to delete all positions programmed in X/Y.

\section*{Rotary axis}

\section*{ZC plane}

You program in ZC to prevent the Y axis moving during machining.
To ensure that the holes point to the center of the "Cylinder", you must first position the \(Y\) axis centrally above the "Cylinder".


Figure 9-1 Holes pointing toward the center


Figure 9-2 \(\quad \mathrm{Y}\) axis is not centered above the cylinder

\section*{YZCA plane}

You program in YZC if the \(Y\) axis should also move during machining. A value can be specified for each position. In addition to the possibilities of ZC, the following is also possible, for example.


Figure 9-3 \(\quad \mathrm{Y}\) axis is traversed \((\mathrm{YO}, \mathrm{Y} 1)\)

\section*{See also}

Positions and position patterns (Page 335)

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Drill.

Positions

2. Press the "Drilling" softkey.
3. Press the "Positions" and "Arbitrary positions" softkeys. The "Positions" input window opens.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
LAB \\
(only for G code)
\end{tabular} & Repeat jump label for position & \\
\hline \begin{tabular}{l}
PL U \\
(only for G code)
\end{tabular} & Machining plane & \\
\hline Axes (only for G code) U & \begin{tabular}{l}
Selection of the participating axes \\
- XY (1st and 2nd axis of the plane) \\
- ZC (rotary axis and assigned linear axis) \\
- YZC (rotary axis and both axes of the plane) \\
Note: \\
Rotary axes are only displayed in the selection field if they have been released for use in the position pattern. \\
Please observe the information provided by your machine manufacturer.
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline Coordinate system U (only for ShopTurn) & \begin{tabular}{l}
- Right-angled or polar \\
Dimensions in right-angled coordinates or polar coordinates (only for face \(C\) and face \(Y\) ) \\
- Right-angled or cylindrical \\
Dimensions in right-angled coordinates or cylindrical coordinates - (only for peripheral surface C)
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Y0 } \\
& \text { X1 } \\
& \text { Y...X8 } \\
& \text { Y1 } \\
& \text { Y...Y8 } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
Axes XY (at right angles) \\
X coordinate of 1st position (abs) \\
Y coordinate of 1st position (abs) \\
\(X\) coordinate for additional positions (abs or inc) \\
Y coordinate for additional positions (abs or inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Z0 } \\
& \text { C0 } \\
& \text { Z1U ... Z8 U } \\
& \text { C1 U...C8 U } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
Axes ZC (for G19) \\
Z coordinate of 1st position (abs) \\
C coordinate of 1st position (abs) \\
Z coordinates for additional positions (abs or inc) \\
C coordinates for additional positions (abs or inc)
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
Degrees
\end{tabular} \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& \text { Y0 } \\
& \text { Z0 } \\
& \text { C0 } \\
& \text { Y1 U ... Y5 U } \\
& \text { Z1 U ... Z5 U } \\
& \text { C1 U...C5 U } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
Axes YZC (for G19) \\
Y coordinate of 1st position (abs) \\
Z coordinate of 1st position (abs) \\
C coordinate of 1st position \\
Y coordinates of additional positions (abs or inc) \\
Z coordinates for additional positions (abs or inc) \\
C coordinates for additional positions (abs or inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline \begin{tabular}{l}
Z0 \\
CP \\
X0 \\
Y0 \\
X1 U... X7 U \\
Y1 U... Y7 U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face \(C\) and face \(Y\) - at right angles \\
\(Z\) coordinate of the reference point (abs) \\
Positioning angle for machining area (only for face Y ) \\
\(X\) coordinate of 1st position (abs) \\
Y coordinate of 1st position (abs) \\
\(X\) coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated Y coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Z0 } \\
& \text { CP } \\
& \text { C0 } \\
& \text { L0 } \\
& \text { C1 U...C7 U } \\
& \text { L1 U... L7 U (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face C and face Y - polar (ShopTurn: \\
\(Z\) coordinate of the reference point (abs) \\
Positioning angle for machining area (only for face Y ) \\
C coordinate of 1st position (abs) \\
1st position of hole with reference to \(Y\) axis (abs) \\
C coordinate for additional positions (abs or inc) \\
Incremental dimension: The sign is also evaluated \\
Distance to position (abs or inc) \\
Incremental dimension: The sign is also evaluated
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
Degrees \\
mm \\
Degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Y0 } \\
& \text { Z0 } \\
& \text { Y1 U...Y7 U } \\
& \text { Z1 U...Z7 U (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface \(\mathbf{C}\) - at right angles \\
Cylinder diameter \(\varnothing\) (abs) \\
Y coordinate of 1st position (abs) \\
Z coordinate of 1st position (abs) \\
Y coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated Z coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { C0 } \\
& \text { Z0 } \\
& \text { C1 U...C7 U } \\
& \text { Z1 U... Z7 U (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C - cylindrical \\
C coordinate of 1st position (abs) 1st position of hole with reference to \(Z\) axis (abs) C coordinate for additional positions (abs or inc) Incremental dimension: The sign is also evaluated Additional positions in the \(Z\) axis (abs or inc) Incremental dimension: The sign is also evaluated
\end{tabular} & \begin{tabular}{l}
Degrees mm \\
Degrees \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline & Peripheral surface Y: & \\
X0 & Reference point in X direction (abs) & mm \\
Y0 & Positioning angle for machining surface & Degrees \\
Z0 & Y coordinate of 1st position (abs) & mm \\
Y1 U...Y7 U & Z coordinate of 1st position (abs) & mm \\
Z1 U...Z7 U (only for ShopTurn & Y coordinate for additional positions (abs or inc) & mm \\
& Incremental dimension: The sign is also evaluated & \\
\hline
\end{tabular}

\subsection*{9.1.12 Row position pattern (HOLES1)}

\section*{Function}

You can program any number of positions at equal distances along a line using the "Row position pattern" function.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Drill.
2. Press the "Drilling" softkey.
3. Press the "Positions" and "Row" softkeys.
Positions

The "Position row" input window opens.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
LAB \\
(only for G code)
\end{tabular} & Repeat jump label for position & \\
\hline \begin{tabular}{l} 
PL U \\
(only for G code)
\end{tabular} & Machining plane & \\
\hline \begin{tabular}{l} 
Machining \\
surface
\end{tabular} & - Face C & \\
\begin{tabular}{l} 
U \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l} 
- Pace Y \\
- Peripheral surface C
\end{tabular} & \\
\hline
\end{tabular}

\section*{9. 1 Drilling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Position U \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
x0 \\
Yo \\
\(\alpha 0\) \\
(only for G Code)
\end{tabular} & \begin{tabular}{l}
\(X\) coordinate of the reference point \(X\) (abs) \\
This position must be programmed absolutely in the 1st call. \\
Y coordinate of the reference point Y (abs) \\
This position must be programmed absolutely in the 1st call. \\
Angle of rotation of the line referred to the X axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees
\end{tabular} \\
\hline \begin{tabular}{l}
Z0 \\
X0 \\
Y0 \\
a0 \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face C: \\
Z coordinate of the reference point (abs) \\
\(X\) coordinate of the reference point - first position (abs) \\
Y coordinate of the reference point - first position (abs) \\
Angle of rotation of line in relation to the \(X\) axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline \begin{tabular}{l}
Z0 \\
CP \\
X0 \\
Y0 \\
a0 \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face \(Y\) : \\
Z coordinate of the reference point (abs) \\
Positioning angle for machining area \\
\(X\) coordinate of the reference point - first position (abs) \\
Y coordinate of the reference point - first position (abs) \\
Angle of rotation of line in relation to the X axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Y0 } \\
& \text { zo } \\
& \text { a0 } \\
& \text { (only for Shop- } \\
& \text { Turn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : \\
Cylinder diameter \(\varnothing\) (abs) \\
Y coordinate of the reference point - first position (abs) \\
\(Z\) coordinate of the reference point - first position (abs) \\
Angle of rotation of line with reference to Y axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline ```
X0
C0
Y0
ZO
a0
(only for Shop-
Turn)
``` & \begin{tabular}{l}
Peripheral surface Y : \\
X coordinate of the reference point (abs) \\
Positioning angle for machining surface \\
Y coordinate of the reference point - first position (abs) \\
Z coordinate of the reference point - first position (abs) \\
Angle of rotation of line with reference to Y axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & mm Degrees mm mm Degrees \\
\hline \[
\begin{aligned}
& \hline \text { LO } \\
& \text { L } \\
& \mathrm{N}
\end{aligned}
\] & \begin{tabular}{l}
Distance of the 1st position to reference point \\
Distance between the positions \\
Number of positions
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline
\end{tabular}

\subsection*{9.1.13 Grid or frame position pattern (CYCLE801)}

\section*{Function}
- You can use the "Grid position pattern" function (CYCLE801) to program any number of positions that are spaced at an equal distance along one or several parallel lines.

If you want to program a rhombus-shaped grid, enter angle \(\alpha X\) or \(\alpha\) Y.
- Frame

You can use the "Frame position pattern" function (CYCLE801) to program any number of positions that are spaced at an equal distance on a frame. The spacing may be different on both axes.

If you want to program a rhombus-shaped frame, enter angle \(\alpha X\) or \(\alpha Y\).

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

\section*{Positions}

2. Press the "Drilling" softkey.
3. Press the "Positions" softkey.
4. Press the "Grid" softkey.
- OR -

Press the "Frame" softkey.

The "Grid position" or "Frame position" input window opens.

\section*{Parameters - "Grid" position pattern}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
LAB \\
(only for G code)
\end{tabular} & Repeat jump label for position & \\
\hline \begin{tabular}{l} 
PL U \\
(only for G code)
\end{tabular} & Machining plane & \\
\hline \begin{tabular}{l} 
Machining \\
surface \\
\begin{tabular}{l}
O \\
(only for ShopTurn)
\end{tabular} \\
\hline
\end{tabular} \begin{tabular}{l} 
- Face C \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline
\end{tabular}

\section*{9. 1 Drilling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
x0 \\
Yo \\
a0 \\
(only for G Code)
\end{tabular} & \begin{tabular}{l}
\(X\) coordinate of the reference point \(X\) (abs) \\
This position must be programmed absolutely in the 1st call. \\
Y coordinate of the reference point Y (abs) \\
This position must be programmed absolutely in the 1st call. \\
Angle of rotation of the line referred to the X axis Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees
\end{tabular} \\
\hline ```
Z0
xo
Y0
\alpha0
(only for ShopTurn)
``` & \begin{tabular}{l}
Face C: \\
Z coordinate of the reference point (abs) \\
X coordinate of the reference point - first position (abs) \\
Y coordinate of the reference point - first position (abs) \\
Angle of rotation of line in relation to the X axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm} \\
& \text { Degrees }
\end{aligned}
\] \\
\hline \begin{tabular}{l}
Z0 \\
CP \\
X0 \\
Yo \\
a0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face \(Y\) : \\
Z coordinate of the reference point (abs) \\
Positioning angle for machining area \\
\(X\) coordinate of the reference point - first position (abs) \\
Y coordinate of the reference point - first position (abs) \\
Angle of rotation of line in relation to the \(X\) axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Zo } \\
& \text { a0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : \\
Cylinder diameter \(\varnothing\) (abs) \\
Y coordinate of the reference point - first position (abs) \\
\(Z\) coordinate of the reference point - first position (abs) \\
Angle of rotation of line with reference to Y axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm} \\
& \text { Degrees }
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& \text { Xo } \\
& \text { co } \\
& \text { Yo } \\
& \text { zo } \\
& \text { a0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface \(Y\) : \\
\(X\) coordinate of the reference point (abs) \\
Positioning angle for machining surface \\
Y coordinate of the reference point - first position (abs) \\
\(Z\) coordinate of the reference point - first position (abs) \\
Angle of rotation of line with reference to Y axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & mm Degrees mm mm Degrees \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline\(\alpha X\) & Shear angle X & Dhear angle Y \\
\(\alpha\) Y & Distance between columns & Degrees \\
L1 & Distance between rows & Degrees \\
L2 & Number of columns & mu \\
N1 & Number of rows & mm \\
N2 & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
LAB \\
(only for G code)
\end{tabular} & Repeat jump label for position & \\
\hline \begin{tabular}{l}
PL U \\
(only for G code)
\end{tabular} & Machining plane & \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Y0 } \\
& \text { a0 } \\
& \text { (only for G Code) }
\end{aligned}
\] & \begin{tabular}{l}
\(X\) coordinate of the reference point \(X\) (abs) \\
This position must be programmed absolutely in the 1st call. \\
\(Y\) coordinate of the reference point \(Y\) (abs) \\
This position must be programmed absolutely in the 1st call. \\
Angle of rotation of the line referred to the X axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees
\end{tabular} \\
\hline Z0
X0
Y0
\(\alpha 0\)
(only for ShopTurn) & \begin{tabular}{l}
Face C: \\
Z coordinate of the reference point (abs) \\
\(X\) coordinate of the reference point - first position (abs) \\
Y coordinate of the reference point - first position (abs) \\
Angle of rotation of line in relation to the \(X\) axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline ZO
CP
X0
Y0
a0
(only for ShopTurn) & \begin{tabular}{l}
Face \(Y\) : \\
Z coordinate of the reference point (abs) \\
Positioning angle for machining area \\
\(X\) coordinate of the reference point - first position (abs) \\
Y coordinate of the reference point - first position (abs) \\
Angle of rotation of line in relation to the X axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { zo } \\
& \text { a0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : \\
Cylinder diameter \(\varnothing\) (abs) \\
Y coordinate of the reference point - first position (abs) \\
Z coordinate of the reference point - first position (abs) \\
Angle of rotation of line with reference to Y axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm Degrees
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Co } \\
& \text { Yo } \\
& \text { zo } \\
& \text { a0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface \(Y\) : \\
X coordinate of the reference point (abs) \\
Positioning angle for machining surface \\
Y coordinate of the reference point - first position (abs) \\
\(Z\) coordinate of the reference point - first position (abs) \\
Angle of rotation of line with reference to Y axis \\
Positive angle: Line is rotated counter-clockwise. \\
Negative angle: Line is rotated clockwise.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
Degrees
\end{tabular} \\
\hline \[
\begin{array}{|l}
\hline \text { L0 } \\
\text { L } \\
\mathrm{N}
\end{array}
\] & Distance of the 1st position to reference point Distance between the positions Number of positions & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& \alpha \mathrm{X} \\
& \mathrm{\alpha Y} \\
& \mathrm{~L} 1 \\
& \mathrm{~L} 2 \\
& \mathrm{~N} 1 \\
& \mathrm{~N} 2
\end{aligned}
\] & \begin{tabular}{l}
Grid or frame \\
Shear angle \(X\) \\
Shear angle \(Y\) \\
Distance between columns \\
Distance between rows \\
Number of columns \\
Number of rows
\end{tabular} & \begin{tabular}{l}
Degrees \\
Degrees \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}

\subsection*{9.1.14 Circle or pitch circle position pattern (HOLES2)}

\section*{Function}

You can program holes on a full circle or a pitch circle of a defined radius with the "Circle position pattern" and "Pitch circle position pattern" functions. The basic angle of rotation ( \(\alpha 0\) ) for the 1 st position is relative to the \(X\) axis. The control calculates the angle of the next hole position as a function of the total number of holes. The angle it calculates is identical for all positions.

The tool can approach the next position along a linear or circular path.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

2. Press the "Drilling" softkey.
3. Press the "Positions" softkey.
4. Press the "Circle" softkey.
- OR -

Press the "Pitch circle" softkey.
The "Position circle" or "Position pitch circle" input window is opened.

\section*{Parameters - "Circle" position pattern}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
LAB \\
(only for G code)
\end{tabular} & Repeat jump label for position & \\
\hline \begin{tabular}{l} 
PL \(\mathcal{U}\) \\
(only for G code)
\end{tabular} & Machining plane & \\
\hline \begin{tabular}{l} 
Axes \\
U \\
(only for G code)
\end{tabular} & \begin{tabular}{l} 
Selection of the participating axes: \\
- XY (1st and 2nd axis of the plane) \\
- ZC (rotary axis and assigned linear axis)
\end{tabular} & \begin{tabular}{l} 
Note: \\
Rotary axes are only displayed in the selection field if they have been re- \\
leased for use in the position pattern. \\
Please observe the information provided by your machine manufacturer.
\end{tabular}
\end{tabular}

Turning

\subsection*{9.1 Drilling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
X0 \\
Y0 \\
a0 \\
R \\
N \\
Positioning \(U\) (only for G code)
\end{tabular} & \begin{tabular}{l}
Axes XY (at right angles) \\
\(X\) coordinate of the reference point (abs) \\
Y coordinate of the reference point (abs) \\
Starting angle for first position referred to the X axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Radius \\
Number of positions \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { zo } \\
& \text { Co } \\
& \mathrm{N} \\
& \text { (only for G code) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
Axes ZC (G19) \\
Z coordinate of the reference point (abs) \\
Start angle of the C axis (abs) \\
Number of positions
\end{tabular} & mm Degrees \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface \(C\) \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
Selection options for the following positions - (only for face C/Y) \\
- center \\
- off-center
\end{tabular} & \\
\hline \begin{tabular}{l}
center/ \\
off-center \\
Z0 \\
X0 \\
YO \\
a0 \\
R \\
N \\
Positioning U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face C: \\
Position circle center on the face surface \\
Position circle off-center on the face surface \\
Z coordinate of the reference point (abs) \\
\(X\) coordinate of the reference point (abs) - (only for off-center) \\
Y coordinate of the reference point (abs) - (only for off-center) \\
Starting angle for first position referred to the X axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Radius \\
Number of positions \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
Degrees \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
center/ \\
off-center \\
Z0 \\
CP \\
XO or LO U \\
YO or CO U \\
a0 \\
R \\
N \\
Positioning U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face Y: \\
Position circle center on the face surface \\
Position circle off-center on the face surface \\
Z coordinate of the reference point (abs) \\
Positioning angle for machining area \\
X coordinate of the reference point (abs) or reference point length, polar \\
- (only for off-center) \\
Y coordinate of the reference point (abs) or reference point angle, polar - (only for off-center) \\
Starting angle for first position referred to the X axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Radius \\
Number of positions \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
Degrees \\
Degrees \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
X0 \\
Z0 \\
a0 \\
N \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Peripheral surface C: \\
Cylinder diameter \(\varnothing\) (abs) \\
Z coordinate of the reference point (abs) \\
Starting angle for first position referred to the Y axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Number of positions
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees
\end{tabular} \\
\hline \begin{tabular}{l}
X0 \\
C0 \\
YO \\
Z0 \\
a0 \\
N \\
R \\
Positioning U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Peripheral surface Y : \\
\(X\) coordinate of the reference point (abs) \\
Positioning angle for machining surface \\
Y coordinate of the reference point (abs) \\
\(Z\) coordinate of the reference point (abs) \\
Starting angle for first position referred to the Y axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Number of positions \\
Radius \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
Degrees \\
mm
\end{tabular} \\
\hline
\end{tabular}

\section*{Parameters - "Pitch circle" position pattern}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline LAB (only for G code) & Repeat jump label for position & \\
\hline \begin{tabular}{l}
PL U \\
(only for G code)
\end{tabular} & Machining plane & \\
\hline \begin{tabular}{l}
Axes \\
U (only for G code)
\end{tabular} & \begin{tabular}{l}
Selection of the participating axes: \\
- XY (1st and 2nd axis of the plane) \\
- ZC (rotary axis and assigned linear axis) \\
Note: \\
Rotary axes are only displayed in the selection field if they have been released for use in the position pattern. \\
Please observe the information provided by your machine manufacturer.
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Y0 } \\
& \text { a0 } \\
& \\
& \alpha 1 \\
& R \\
& \mathrm{~N} \\
& \text { Positioning } U \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
Axes XY (at right angles) \\
\(X\) coordinate of the reference point (abs) \\
Y coordinate of the reference point (abs) \\
Starting angle for first position referred to the X axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Incrementing angle \\
Radius \\
Number of positions \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees \\
Degrees \\
mm
\end{tabular} \\
\hline ZO
C0
N
(only for G code) & \begin{tabular}{l}
Axes ZC (with G19) \\
\(Z\) coordinate of the reference point (abs) \\
Start angle of the C axis (abs) \\
Number of positions
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees
\end{tabular} \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline Position U (only for ShopTurn) & \begin{tabular}{l}
Selection options for the following positions - (only for face C/Y) \\
- center \\
- off-center
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
center/ \\
off-center \\
Z0 \\
X0 \\
YO \\
a0 \\
a1 \\
R \\
N \\
Positioning U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face C: \\
Position circle center on the face surface \\
Position circle off-center on the face surface \\
Z coordinate of the reference point (abs) \\
X coordinate of the reference point (abs) - (only for off-center) \\
Y coordinate of the reference point (abs) - (only for off-center) \\
Starting angle for first position referred to the X axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Incrementing angle \\
Radius \\
Number of positions \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
Degrees \\
Degrees \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
center/ \\
off-center \\
Z0 \\
CP \\
X0 or LO U \\
YO or CO U \\
a0 \\
a1 \\
R \\
N \\
Positioning U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face Y: \\
Position circle center on the face surface \\
Position circle off-center on the face surface \\
Z coordinate of the reference point (abs) \\
Positioning angle for machining area \\
X coordinate of the reference point (abs) or reference point length, polar \\
- (only for off-center) \\
Y coordinate of the reference point (abs) or reference point angle, polar - (only for off-center) \\
Starting angle for first position referred to the X axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Incrementing angle \\
Radius \\
Number of positions \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees mm \\
mm \\
Degrees \\
Degrees \\
Degrees \\
mm
\end{tabular} \\
\hline X0
Z0
a0
a1
N
only for ShopTurn) & \begin{tabular}{l}
Peripheral surface C : \\
Cylinder diameter \(\varnothing\) (abs) \\
Z coordinate of the reference point (abs) \\
Starting angle for first position referred to the Y axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Incrementing angle \\
Number of positions
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
Degrees \\
Degrees
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
X0 \\
C0 \\
Yo \\
Z0 \\
a0 \\
a1 \\
N \\
R \\
Positioning \(U\) (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Peripheral surface \(Y\) : \\
\(X\) coordinate of the reference point (abs) \\
Positioning angle for machining surface \\
Y coordinate of the reference point (abs) \\
Z coordinate of the reference point (abs) \\
Starting angle for first position referred to the Y axis. \\
Positive angle: Circle is rotated counter-clockwise. \\
Negative angle: Circle is rotated clockwise. \\
Incrementing angle \\
Number of positions \\
Radius \\
- Straight line: Next position is approached linearly in rapid traverse. \\
- Circular: Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees \\
mm \\
mm \\
Degrees \\
Degrees \\
mm
\end{tabular} \\
\hline
\end{tabular}

\subsection*{9.1.15 Displaying and hiding positions}

\section*{Function}

You can hide any positions in the following position patterns:
- Position pattern line
- Position pattern grid
- Position pattern frame
- Full circle position pattern
- Pitch circle position pattern

The hidden positions are skipped when machining.

\section*{Display}

The programmed positions of the position pattern are shown as follows in the programming graphic:
x Position is activated \(=\) displayed (position is shown as a cross)
- Position deactivated = hidden (position shown as a circle)

\section*{Selecting positions}

You have the option of either displaying or hiding positions - by activating the checkbox in the displayed position table either using the keyboard or mouse.

\section*{Procedure:}


\section*{Positions}


Hide position
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling" and "Positions" softkeys.
3. Press the "Line/Grid/Frame" or "Full/Pitch Circle" softkeys.
4. Press the "Hide position" softkey.

The "Hide position" window opens on top of the input form of the position pattern.
The positions are displayed in a table.
The numbers of the positions, their coordinates ( \(\mathrm{X}, \mathrm{Y}\) ) as well as a checkbox with the state (activated \(=\) on \(/\) deactivated \(=\) off) are displayed.
The actual position in the graphic is highlighted in color.
5. Using the mouse, select the required position and deactivate or activate the checkbox in order to hide the position or display it again.
In the diagram, skipped positions are shown in the form of a circle and displayed (active) positions are shown in the form of a cross.
Note: You have the option of selecting individual positions using the <Cursor up> or <Cursor down> keys - and hiding and displaying using the <SELECT> key.

\section*{Display or hide all positions at once}

\section*{Hide}
1. Press the "Hide all" softkey to hide all positions.

2. Press the "Show all" softkey to display all positions again.

\subsection*{9.1.16 Repeating positions}

\section*{Function}

If you want to approach positions that you have already programmed again, you can do this quickly with the function "Repeat position".
You must specify the number of the position pattern. The cycle automatically assigns this number (for ShopTurn). You will find this position pattern number in the work plan (program view) or G-code program after the block number.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Drilling", and "Repeat position" softkeys.

The "Repeat positions" input window opens.

\section*{Position \\ repetit.}

3. After you have entered the label or the position pattern number, e.g. 1, press the "Accept" softkey. The position pattern you have selected is then approached again.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
LAB \\
(only for G code)
\end{tabular} & Repeat jump label for position & \\
\hline \begin{tabular}{l} 
Position (only for \\
ShopTurn)
\end{tabular} & Enter the number of the position pattern & \\
\hline
\end{tabular}

\subsection*{9.2 Rotate}

\subsection*{9.2.1 General}

In all turning cycles apart from contour turning (CYCLE95), in the combined roughing and finishing mode, when finishing it is possible to reduce the feedrate as a percentage.


Machine manufacturer
Please also refer to the machine manufacturer's specifications.

\subsection*{9.2.2 Stock removal (CYCLE951)}

\section*{Function}

You can use the "Stock removal" cycle for longitudinal or transverse stock removal of corners at outer or inner contours.

\section*{Note}

\section*{Removing stock from corners}

For this cycle, the safety clearance is additionally limited using setting data. The lower value is taken for machining.

Please refer to the machine manufacturer's specifications.

\section*{Machining method}
- Roughing

In roughing applications, paraxial cuts are machined to the finishing allowance that has been programmed. If no finishing allowance has been programmed, the workpiece is roughed down to the final contour.

During roughing, the cycle reduces the programmed infeed depth \(D\) if necessary so that it is possible for cuts of an equal size to be made. For example, if the overall infeed depth is 10 and you have specified an infeed depth of 3 , this would result in cuts of \(3,3,3\) and 1. The cycle would reduce the infeed depth to 2.5 to create 4 equally sized cuts.
The angle between the contour and the tool cutting edge determines whether the tool rounds the contour at the end of each cut by the infeed depth \(D\), in order to remove residual corners, or is raised immediately. The angle beyond which rounding is performed is stored in a machine data element.

\section*{Machine manufacturer}

Please also refer to the machine manufacturer's instructions.

If the tool does not round the corner at the end of the cut, it is raised by the safety distance or a value specified in the machine data at rapid traverse. The cycle always observes the lower value; otherwise, stock removal at inner contours, for example, could cause the contour to be damaged.

\section*{Machine manufacturer}

Please also refer to the machine manufacturer's instructions.
- Finishing

Finishing is performed in the same direction as roughing. The cycle automatically selects and deselects tool radius compensation during finishing.

\section*{Approach/retraction}
1. The tool first moves at rapid traverse to the starting point of the machining operation calculated internally in the cycle (reference point + safety distance).
2. The tool moves to the first infeed depth at rapid traverse.
3. The first cut is made at machining feedrate.
4. The tool rounds the contour at machining feedrate or is raised at rapid traverse (see "Roughing").
5. The tool is moved at rapid traverse to the starting point for the next infeed depth.
6. The next cut is made at machining feedrate.
7. Steps 4 to 6 are repeated until the final depth is reached.
8. The tool moves back to the safety distance at rapid traverse.

\section*{Procedure}

1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Turning" softkey.
3. Press the "Stock removal" softkey. The "Stock Removal" input window opens.
4. Select one of the three stock removal cycles via the softkeys:

Simple straight stock removal cycle.
The "Stock removal 1" input window opens.
- OR


Straight stock removal cycle with radii or chamfers.
The "Stock removal 2" input window opens.
- OR

Stock removal cycle with oblique lines, radii, or chamfers.
The "Stock Removal 3" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline SC & Safety clearance & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & F & Feedrate & mm/rev \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|c|c|c|}
\hline Parameter & \multicolumn{4}{|l|}{Description} & Unit \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \multicolumn{4}{|l|}{\begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing)
\end{tabular}} & \\
\hline Position 0 & \multicolumn{4}{|l|}{Machining posi
"/4} & \\
\hline \multirow[t]{3}{*}{Machining direction
\[
U
\]} & \multicolumn{4}{|l|}{Stock removal direction (transverse or longitudinal) in the coordinate system} & \\
\hline & Parallel to the & s (longitudinal) & Parallel to the & s (transverse) & \\
\hline & outside & inside & outside &  & \\
\hline X0 & \multicolumn{4}{|l|}{Reference point in \(\mathrm{X} \varnothing\) (abs, always diameter)} & mm \\
\hline Z0 & \multicolumn{4}{|l|}{Reference point in Z (abs)} & mm \\
\hline X1 U & \multicolumn{4}{|l|}{End point X (abs) or end point X in relation to X 0 (inc)} & \\
\hline Z1 U & \multicolumn{4}{|l|}{End point \(Z\) (abs) or end point \(Z\) in relation to \(Z 0\) (inc)} & \\
\hline D & \multicolumn{4}{|l|}{Maximum depth infeed - (not for finishing)} & mm \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline UX & Finishing allowance in \(\mathrm{X}-\) (not for finishing) & mm \\
\hline UZ & Finishing allowance in Z - (not for finishing) & mm \\
\hline FS1 ...FS3 or R1...R3 & Chamfer width (FS1...FS3) or rounding radius (R1...R3) - (not for stock removal 1) & mm \\
\hline 0 & \begin{tabular}{l}
Parameter selection of intermediate point \\
The intermediate point can be determined through position specification or angle. The following combinations are possible - (not for stock removal 1 and 2) \\
- XM ZM \\
- XM \(\alpha 1\) \\
- XM 22 \\
- \(\quad\) 1 ZM \\
- \(\alpha 2 \mathrm{ZM}\) \\
- \(\quad \alpha 1\) a2
\end{tabular} & \\
\hline XM U & Intermediate point \(\mathrm{X} \varnothing\) (abs) or intermediate point X in relation to \(\mathrm{X0}\) (inc) & mm \\
\hline ZM U & Intermediate point \(Z\) (abs or inc) & mm \\
\hline a1 & Angle of the 1st edge & Degrees \\
\hline a2 & Angle of the 2nd edge & Degrees \\
\hline
\end{tabular}

\subsection*{9.2.3 Groove (CYCLE930)}

\section*{Function}

You can use the "Groove" cycle to machine symmetrical and asymmetrical grooves on any straight contour elements.

You have the option of machining outer or inner grooves, longitudinally or transversely (face). Use the "Groove width" and "Groove depth" parameters to determine the shape of the groove. If a groove is wider than the active tool, it is machined in several cuts. The tool is moved by a maximum of \(80 \%\) of the tool width for each groove.
You can specify a finishing allowance for the groove base and the flanks; roughing is then performed down to this point.
The dwell time between recessing and retraction is stored in a setting data element.


\section*{Machine manufacturer}

Please also refer to the machine manufacturer's specifications.

\section*{Approach/retraction during roughing}

\section*{Infeed depth D > 0}
1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The tool cuts a groove in the center of infeed depth \(D\).
3. The tool moves back by \(\mathrm{D}+\) safety clearance with rapid traverse.
4. The tool cuts a groove next to the first groove with infeed depth \(2 \cdot \mathrm{D}\).
5. The tool moves back by \(\mathrm{D}+\) safety clearance with rapid traverse.
6. The tool cuts alternating in the first and second groove with the infeed depth \(2 \cdot \mathrm{D}\), until the final depth T1 is reached.

Between the individual grooves, the tool moves back by D + safety clearance with rapid traverse. After the last groove, the tool is retracted at rapid traverse to the safety distance.
7. All subsequent groove cuts are made alternating and directly down to the final depth T1. Between the individual grooves, the tool moves back to the safety distance at rapid traverse.

\section*{Approach/retraction during finishing}
1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The tool moves at the machining feedrate down one flank and then along the bottom to the center.
3. The tool moves back to the safety distance at rapid traverse.
4. The tool moves at the machining feedrate along the other flank and then along the bottom to the center.
5. The tool moves back to the safety distance at rapid traverse.

\section*{Procedure}


Grooue

1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Turning" softkey.
3. Press the "Groove" softkey.

The "Groove" input window opens.
4. Select one of the three groove cycles with the softkey:

Simple groove cycle
The "Groove 1" input window opens.
- OR

Groove cycle with inclines, radii, or chamfers.
The "Groove 2" input window opens.
- OR

Groove cycle on an incline with inclines, radii or chamfers.
The "Groove 3" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters，G code program} & \multicolumn{3}{|l|}{Parameters，ShopTurn program} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline SC & Safety clearance & mm & D & Cutting edge number & \\
\hline F & Feedrate & ＊ & F & Feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
－\(\nabla\)（roughing） \\
－\(\nabla \nabla \nabla\)（finishing） \\
－\(\quad \nabla+\nabla \nabla \nabla\)（roughing and finishing）
\end{tabular} & \\
\hline Position U & Groove position： & \\
\hline X0 & Reference point in \(\mathrm{X} \varnothing\) & mm \\
\hline Z0 & Reference point in Z & mm \\
\hline B1 & Groove width & mm \\
\hline T1 & Groove depth \(\varnothing\)（abs）or groove depth referred to X0 or Z0（inc） & mm \\
\hline D & \begin{tabular}{l}
－Maximum depth infeed for insertion－（only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ） \\
－For zero：Insertion in a cut－（only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ） \\
\(D=0\) ： 1 ．cut is made directly to final depth T1 \\
\(D>0\) ： 1 st and 2 nd cuts are made alternately to infeed depth \(D\) ，in order to achieve a better chip flow and prevent the tool from breaking，refer to approaching／retraction when roughing． \\
Alternate cutting is not possible if the tool can only reach the groove base at one position．
\end{tabular} & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and \(\mathrm{Z}-\)（only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ） & mm \\
\hline UZ & Finishing allowance in \(\mathrm{Z}-\)（for UX，only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ） & mm \\
\hline N & Number of grooves（ \(\mathrm{N}=1\) ．．．．65535） & \\
\hline DP & Distance between grooves（inc） DP is not displayed when \(\mathrm{N}=1\) & mm \\
\hline a1，\(\alpha^{2}\) & \begin{tabular}{l}
Flank angle 1 or flank angle 2 －（only for grooves 2 and 3） \\
Asymmetric grooves can be described by separate angles．The angles can be be－ tween 0 and \(<90^{\circ}\) ．
\end{tabular} & Degrees \\
\hline FS1．．．FS4 or R1．．．R4 U & Chamfer width（FS1．．．FS4）or rounding radius（R1．．．R4）－（only for grooves 2 and 3） & mm \\
\hline a0 & Angle of the incline－（only for groove 3） & Degrees \\
\hline
\end{tabular}
＊Unit of feedrate as programmed before the cycle call

\subsection*{9.2.4 Undercut form E and F (CYCLE940)}

\section*{Function}

You can use the "Undercut form E" or "Undercut form F" cycle to turn form E or F undercuts in accordance with DIN 509.

\section*{Approach/retraction}
1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The undercut is made in one cut at the machining feedrate, starting from the flank through to the cross-feed VX.
3. The tool moves back to the starting point at rapid traverse.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

2. Press the "Turning" softkey.

Undercut

\section*{Undercut} form E

Undercut form F
3. Press the "Undercut" softkey.

The "Undercut" input window opens.
4. Select one of the following undercut cycles via the softkeys:

Press the "Undercut form E" softkey. The "Undercut form E (DIN 509)" input window opens.
- OR

Press the "Undercut form F" softkey.
The "Undercut form F (DIN 509)" input window opens.
\begin{tabular}{|l|l|l|l|l|l|l|}
\hline \multicolumn{2}{|l|}{ Parameters, G code program (undercut, form E) } & & \multicolumn{3}{|l|}{ Parameters, ShopTurn program (undercut, form E) } \\
\hline PL & Machining plane & & & T & Tool name & \\
\hline SC & Safety clearance & & mm & & D & Cutting edge number
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Position 0 & Form E machining position: & \\
\hline U & Undercut size according to DIN table: E.g.: E1.0 x 0.4 (undercut form E) & \\
\hline X0 & Reference point \(\mathrm{X} \varnothing\) & mm \\
\hline Z0 & Reference point Z & mm \\
\hline \[
\begin{aligned}
& \mathrm{X} 1 \\
& \mathrm{U}
\end{aligned}
\] & Allowance in \(\mathrm{X} \varnothing\) (abs) or allowance in X (inc) & mm \\
\hline \[
\begin{aligned}
& \mathrm{VX} \\
& \mathrm{U}
\end{aligned}
\] & Cross feed \(\varnothing\) (abs) or cross feed (inc) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program (undercut, form F)} & \multicolumn{3}{|l|}{Parameters, ShopTurn program (undercut, form F)} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline SC & Safety clearance & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & F & Feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Position \(U\) & \begin{tabular}{l}
Form F machining position:㣻 \\
曻 \\
2\%
\end{tabular} & \\
\hline U & Undercut size according to DIN table: e.g.: F0.6 x 0.3 (undercut form F) & \\
\hline X0 & Reference point \(\mathrm{X} \varnothing\) & mm \\
\hline Z0 & Reference point Z & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline X1 U & Allowance in \(\mathrm{X} \varnothing\) (abs) or allowance in X (inc) & mm \\
\hline Z1 U & Allowance in Z (abs) or allowance in Z (inc) - (for undercut form F only) & mm \\
\hline VXU & Cross feed \(\varnothing\) (abs) or cross feed (inc) & mm \\
\hline
\end{tabular}

\subsection*{9.2.5 Thread undercuts (CYCLE940)}

\section*{Function}

The "Thread undercut DIN" or "Thread undercut" cycle is used to program thread undercuts to DIN 76 for workpieces with a metric ISO thread, or freely definable thread undercuts.

\section*{Approach/retraction}
1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The first cut is made at the machining feedrate, starting from the flank and traveling along the shape of the thread undercut as far as the safety distance.
3. The tool moves to the next starting position at rapid traverse.
4. Steps 2 and 3 are repeated until the thread undercut is finished.
5. The tool moves back to the starting point at rapid traverse.

During finishing, the tool travels as far as cross-feed VX.

\section*{Procedure}
\begin{tabular}{|c|c|c|}
\hline & 1. & The part program or ShopTurn program to be processed has been created and you are in the editor. \\
\hline 龃 Turn- & 2. & Press the "Turning" softkey. \\
\hline Undercut & 3. & Press the "Undercut" softkey. \\
\hline Undercut thrd DIN & 4. & \begin{tabular}{l}
Press the "Thread undercut DIN" softkey. \\
The "Thread Undercut (DIN 76)" input window opens.
\end{tabular} \\
\hline & & - OR - \\
\hline Undercut thread & & \begin{tabular}{l}
Press the "Thread undercut" softkey. \\
The "Thread Undercut" input window opens.
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|l|l|l|l|}
\hline \multicolumn{2}{|l|}{\begin{tabular}{l} 
Parameters, G code program \\
(undercut, thread DIN)
\end{tabular}} & & \multicolumn{2}{|l|}{\begin{tabular}{l} 
Parameters, ShopTurn program \\
(undercut, thread DIN)
\end{tabular}} \\
\hline PL & Machining plane & & & T & Tool name & \\
\hline SC & Safety clearance & & mm & & D & Cutting edge number
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Machining 0 & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Position U & Machining position: & \\
\hline Machining direction U & \begin{tabular}{l}
- Longitudinal \\
- Parallel to the contour
\end{tabular} & \\
\hline Form \(U\) & \begin{tabular}{l}
- Normal (form A) \\
- \(\quad\) Short (form B)
\end{tabular} & \\
\hline P U & Thread pitch (select from the preset DIN table or enter) & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline X0 & Reference point X \(\varnothing\) & mm \\
\hline Z0 & Reference point Z & mm \\
\hline a & Insertion angle & Degrees \\
\hline VXU & Cross feed \(\varnothing\) (abs) or cross feed (inc) - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline D & Maximum depth infeed - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline U or UX U & Finishing allowance in \(X\) or finishing allowance in \(X\) and \(Z\) - (only for \(\nabla\) and \(\nabla+\) \(\nabla \nabla \nabla\) ) & mm \\
\hline UZ & Finishing allowance in \(Z-\) (only for UX, \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call
\begin{tabular}{|l|l|l|l|l|l|l|}
\hline \multicolumn{2}{|l|}{ Parameters, G code program (undercut, thread) } & & \multicolumn{3}{|l|}{ Parameters, ShopTurn program (undercut, thread) } \\
\hline PL & Machining plane & & & T & Tool name & \\
\hline SC & Safety clearance & & & & \\
\hline F & Feedrate & & & Cutting edge number & \\
\hline & & & & F & Feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \(\mathrm{S} / \mathrm{V}\) & \begin{tabular}{l} 
Spindle speed or constant cut- \\
ting rate
\end{tabular} & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Machining
\[
0
\] & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Machining direction U & \begin{tabular}{l}
- Longitudinal \\
- Parallel to the contour
\end{tabular} & \\
\hline Position U & Machining position: & \\
\hline X0 & Reference point X \(\varnothing\) & mm \\
\hline Z0 & Reference point Z & mm \\
\hline X1 U & Undercut depth referred to \(\mathrm{X} \varnothing\) (abs) or undercut depth referred to X (inc) & mm \\
\hline Z1 U & Allowance Z (abs or inc) & mm \\
\hline R1 & Rounding radius 1 & mm \\
\hline R2 & Rounding radius 2 & mm \\
\hline a & Insertion angle & Degrees \\
\hline VXU & Cross feed \(\varnothing\) (abs) or cross feed (inc) - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & \\
\hline D & Maximum depth infeed - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline U or UX U & Finishing allowance in \(X\) or finishing allowance in \(X\) and \(Z\) (only for \(\nabla\) and \(\nabla+\) \(\nabla \nabla \nabla\) ) & mm \\
\hline UZ & Finishing allowance in Z - (only for UZ, \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\subsection*{9.2.6 Thread turning (CYCLE99)}

\section*{Function}

The "Longitudinal thread", "Tapered thread" or "Face thread" cycle is used to turn external or internal threads with a constant or variable pitch.
There may be single or multiple threads.
For metric threads (thread pitch P in \(\mathrm{mm} / \mathrm{rev}\) ), the cycle assigns a value (calculated on the basis of the thread pitch) to the thread depth H 1 parameter. You can change this value.

The default must be activated via setting data SD 55212
\$SCS_FUNCTION_MASK_TECH_SET.


Machine manufacturer
Please refer to the machine manufacturer's specifications.
The cycle requires a speed-controlled spindle with a position measuring system.

\section*{Interruption of thread cutting}

You have the option to interrupt thread cutting (for example if the cutting tool is broken).
1. Press the <CYCLE STOP> key.

The tool is retracted from the thread and the spindle is stopped.
2. Replace the tool and press the <CYCLE START> key.

The aborted thread cutting is started again with the interrupted cut at the same depth.

\section*{Thread re-machining}

You have the option of subsequently machining threads. To do this, change into the "JOG" operating mode and carry out a thread synchronization.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction}
1. The tool moves to the starting point calculated internally in the cycle at rapid traverse.
2. Thread with advance:

The tool moves at rapid traverse to the first starting position displaced by the thread advance LW.

Thread with run-in:
The tool moves at rapid traverse to the starting position displaced by the thread run-in LW2.
3. The first cut is made with thread pitch \(P\) as far as the thread run-out \(L R\).
4. Thread with advance:

The tool moves at rapid traverse to the return distance VR and then to the next starting position.

Thread with run-in:
The tool moves at rapid traverse to the return distance VR and then back to the starting position.
5. Steps 3 and 4 are repeated until the thread is finished.
6. The tool moves back to the retraction plane at rapid traverse.

Thread machining can be stopped at any time with the "Rapid lift" function. It ensures that the tool does not damage the thread when it is raised.

\section*{Procedure for longitudinal thread, tapered thread, or face thread}
\begin{tabular}{|c|c|c|}
\hline & 1. & The part program or ShopTurn program to be processed has been created and you are in the editor. \\
\hline \[
\begin{gathered}
\text { Turn- } \\
\text { ing }
\end{gathered}
\] & 2. & Press the "Turning" softkey. \\
\hline Thread & 3. & \begin{tabular}{l}
Press the "Thread" softkey. \\
The "Thread" input window opens.
\end{tabular} \\
\hline Thread long. & 4. & \begin{tabular}{l}
Press the "Longitudinal thread" softkey. \\
The "Longitudinal Thread" input window opens.
\end{tabular} \\
\hline & & - OR - \\
\hline \multirow[t]{2}{*}{Thread taper} & & Press the "Tapered thread" softkey. \\
\hline & & The "Tapered Thread" input window opens. \\
\hline & & - OR - \\
\hline \multirow[t]{2}{*}{Thread face} & & Press the "Face thread" softkey. \\
\hline & & The "Face Thread" input window opens. \\
\hline
\end{tabular}

\section*{Parameter "Longitudinal thread" in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G-code program parameters} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline \multirow[t]{3}{*}{PL} & Machining plane & & T & Tool name & \\
\hline & & & D & Cutting edge number & \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline & & & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Table \\
0
\end{tabular} & \begin{tabular}{l}
Thread table selection: \\
- Without \\
- ISO metric \\
- Whitworth BSW \\
- Whitworth BSP \\
- UNC
\end{tabular} & \\
\hline Selection - (not for table "Without") U & Data, table value, e.g. M10, M12, M14, ... & \\
\hline \begin{tabular}{l} 
P \\
\hline
\end{tabular} & \begin{tabular}{l}
Select the thread pitch / turns for table "Without" or specify the thread pitch/turns corresponding to the selection in the thread table: \\
- Thread pitch in mm/revolution \\
- Thread pitch in inch/revolution \\
- Thread turns per inch \\
- Thread pitch in MODULUS
\end{tabular} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev \\
turns/" \\
MODULUS
\end{tabular} \\
\hline G & \begin{tabular}{l}
Change in thread pitch per revolution - (only for \(\mathrm{P}=\mathrm{mm} / \mathrm{rev}\) or \(\mathrm{in} / \mathrm{rev}\) ) \\
\(\mathrm{G}=0\) : The thread pitch P does not change. \\
\(G>0\) : The thread pitch \(P\) increases by the value \(G\) per revolution. \\
\(G<0\) : The thread pitch \(P\) decreases by the value \(G\) per revolution. \\
If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows:
\[
\mathrm{G}=\frac{\left|\mathrm{Pe}^{2}-\mathrm{P}^{2}\right|}{2 *---------\mathrm{Z}_{1}}\left[\mathrm{~mm} / \mathrm{rev}^{2}\right]
\] \\
The meanings are as follows: \\
\(\mathrm{P}_{\mathrm{e}}\) : End pitch of thread [mm/rev] \\
\(\mathrm{Pa}_{\mathrm{a}}\) : Start pitch of thread [mm/rev] \\
\(Z_{1}\) : Thread length [mm] \\
A larger pitch results in a larger distance between the thread turns on the workpiece.
\end{tabular} & \(\mathrm{mm} / \mathrm{rev}^{2}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Infeed (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla)\)
\[
0
\] & \begin{tabular}{l}
- Linear: \\
Infeed with constant cutting depth \\
- Degressive: \\
Infeed with constant cutting cross-section
\end{tabular} & \\
\hline Thread & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline X0 & Reference point X from thread table \(\varnothing\) (abs) & mm \\
\hline Z0 & Reference point Z (abs) & mm \\
\hline Z1 U & End point of the thread (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated. & mm \\
\hline Amount of crown U & \begin{tabular}{l}
Allowance to compensate for sag ( - only for external thread and G=0) \\
- XS \\
Segment height, crowned thread \\
- RS \\
Radius crowned thread \\
Positive values: Convex \\
Negative values: Concave \\
Note: \\
the pitch change per revolution "G" must be "0".
\end{tabular} & mm mm \\
\hline LW & Thread advance (inc) & mm \\
\hline U or & \begin{tabular}{l}
The starting point for the thread is the reference point (XO,ZO) brought forward by the thread advance W . The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. \\
Thread run-in (inc)
\end{tabular} & \\
\hline \begin{tabular}{l}
LW2 \\
U \\
or
\end{tabular} & \begin{tabular}{l}
The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). \\
Thread run-in = thread run-out (inc)
\end{tabular} & mm \\
\hline \[
\begin{aligned}
& \mathrm{LW} 2=\mathrm{LR} \\
& \mathrm{U}
\end{aligned}
\] & & mm \\
\hline LR & \begin{tabular}{l}
Thread run-out (inc) \\
The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).
\end{tabular} & mm \\
\hline H1 & Thread depth from thread table (inc) & mm \\
\hline DP & \begin{tabular}{l}
Infeed slope as flank (inc) - (alternative to infeed slope as angle) \\
DP > 0: Infeed along the rear flank \\
DP < 0: Infeed along the front flank
\end{tabular} & mm \\
\hline
\end{tabular}

\subsection*{9.2 Rotate}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & Description & & Unit \\
\hline or \({ }_{\text {aP }}\) & \multicolumn{2}{|l|}{\begin{tabular}{l}
Infeed slope as angle - (alternative to infeed slope as flank) \\
\(\alpha>0\) : Infeed along the rear flank \\
\(\alpha<0\) : Infeed along the front flank \\
\(\alpha=0\) : Infeed at right angle to cutting direction \\
If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.
\end{tabular}} & Degrees \\
\hline \begin{tabular}{l}
\[
* U
\] \\
\(* *\)
\end{tabular} & \multicolumn{2}{|l|}{\begin{tabular}{l}
Infeed along the flank \\
Infeed with alternating flanks (alternative) \\
Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. \\
\(\alpha>0\) : Start at the rear flank \\
\(\alpha<0\) : Start at the front flank
\end{tabular}} & \\
\hline D0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
Initial plunge depth - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) under "Manual Machine") \\
If you want to rework some threads, input the initial plunge depth D0 (inc.). This is the depth that was reached during a previous machining. \\
By inputting the plunge depth, you avoid unnecessary idle cuts when reworking the threads.
\end{tabular}} & mm \\
\hline \begin{tabular}{l}
D1 or ND U \\
(only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla)\)
\end{tabular} & \multicolumn{2}{|l|}{\begin{tabular}{l}
First infeed depth or number of roughing cuts \\
The respective value is displayed when you switch between the number of roughing cuts and the first infeed.
\end{tabular}} & mm \\
\hline U & \multicolumn{2}{|l|}{Finishing allowance in X and Z - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) )} & mm \\
\hline NN & \multicolumn{2}{|l|}{Number of noncuts - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) )} & \\
\hline VR & \multicolumn{2}{|l|}{Return distance (inc)} & mm \\
\hline \multirow[t]{6}{*}{Multiple threads
U} & \multicolumn{2}{|l|}{No} & \\
\hline & a0 & Starting angle offset & Degrees \\
\hline & \multicolumn{2}{|l|}{Yes} & \\
\hline & N & Number of thread turns The thread turns are distributed evenly across the periphery of the turned part; the 1 st thread turn is always located at \(0^{\circ}\). & \\
\hline & DA & \begin{tabular}{l}
Thread changeover depth (inc) \\
First machine all thread turns sequentially to thread changeover depth DA, then machine all thread turns sequentially to depth 2 . DA, etc. until the final depth is reached. \\
DA \(=0\) : Thread changeover depth is not taken into account, i.e. finish machining each thread before starting the next thread.
\end{tabular} & mm \\
\hline & Machining:
\[
0
\] & \begin{tabular}{l}
- Complete, or \\
- From turn N1 \\
N1 (1...4) start thread N1 = 1...N U or \\
- Only thread NX \\
\(N X(1 \ldots 4) 1\) from \(N\) threads \(U\)
\end{tabular} & \\
\hline
\end{tabular}

\section*{Parameter "Longitudinal thread" in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|}
\hline \multicolumn{2}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline Input & - simple & & & \\
\hline & & T & Tool name & \\
\hline & & D & Cutting edge number & \\
\hline & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline & & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline & \begin{tabular}{l}
Select the thread pitch / turns for table "Without" or specify the thread pitch/turns corresponding to the selection in the thread table: \\
- Thread pitch in mm/revolution \\
- Thread pitch in inch/revolution \\
- Thread turns per inch \\
- Thread pitch in MODULUS
\end{tabular} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev \\
turns/" \\
MODULUS
\end{tabular} \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla V \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline \begin{tabular}{l}
Infeed (only for \(\nabla\) and \(\nabla\) \(+\nabla \nabla \nabla\) ) \\
O
\end{tabular} & \begin{tabular}{l}
- Linear: \\
Infeed with constant cutting depth \\
- Degressive: \\
Infeed with constant cutting cross-section
\end{tabular} & \\
\hline Thread U & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline x0 & Reference point X from thread table \(\varnothing\) (abs) & mm \\
\hline Z0 & Reference point \(Z\) (abs) & mm \\
\hline Z10 & End point of the thread (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated. & mm \\
\hline LW
\(U\)
or
LW2
\(U\)
or
LW2 \(=~ L R ~\)
\(U\) & \begin{tabular}{l}
Thread advance (inc) \\
The starting point for the thread is the reference point ( \(\mathrm{XO}, \mathrm{ZO}\) ) brought forward by the thread advance \(W\). The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. \\
Thread run-in (inc) \\
The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). \\
Thread run-in = thread run-out (inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}

\subsection*{9.2 Rotate}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline LR & \begin{tabular}{l}
Thread run-out (inc) \\
The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).
\end{tabular} & mm \\
\hline H1 & Thread depth from thread table (inc) & mm \\
\hline \[
\begin{aligned}
& \mathrm{DP} \\
& \mathrm{U}
\end{aligned}
\] & Infeed slope as flank (inc) - (alternative to infeed slope as angle) DP >0: Infeed along the rear flank DP < 0: Infeed along the front flank & mm \\
\hline aP & \begin{tabular}{l}
Infeed slope as angle - (alternative to infeed slope as flank) \\
\(\alpha>0\) : Infeed along the rear flank \\
\(\alpha<0\) : Infeed along the front flank \\
\(\alpha=0\) : Infeed at right angle to cutting direction \\
If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.
\end{tabular} & Degrees \\
\hline \[
\begin{aligned}
& * 0 \\
& *
\end{aligned}
\] & \begin{tabular}{l}
Infeed along the flank \\
Infeed with alternating flanks (alternative) \\
Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. \\
\(\alpha>0\) : Start at the rear flank \\
\(\alpha<0\) : Start at the front flank
\end{tabular} & \\
\hline D1 or ND U (only for \(\nabla\) and \(\nabla+\nabla \mathrm{V}\) ) & \begin{tabular}{l}
First infeed depth or number of roughing cuts \\
The respective value is displayed when you switch between the number of roughing cuts and the first infeed.
\end{tabular} & mm \\
\hline \(\cup\) & Finishing allowance in \(X\) and \(Z-\) (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline NN & Number of noncuts - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\(\left.\begin{array}{|l|l|l|l|}\hline \text { Parameter } & \text { Description } & \text { Value } & \text { Can be set in SD } \\ \hline \text { PL } & \text { Machining plane } & \begin{array}{l}\text { Defined in MD } \\ 52005\end{array} & \\ \hline \text { Table } & \text { Thread table selection } & \text { without } & \\ \hline \text { G } & \begin{array}{l}\text { Change in thread pitch per revolution - (only for P = mm/rev } \\ \text { or in/rev): } \\ \text { Without change in thread pitch }\end{array} & 0 & \\ \hline \begin{array}{l}\text { XS } \\ \text { RS } \\ \text { U }\end{array} & \begin{array}{l}\text { Segment height, crowned thread } \\ \text { Radius crowned thread }\end{array} & \begin{array}{l}0 \mathrm{~mm} \\ \hline \text { D0 }\end{array} & \text { Initial plunge depth for reworking the threads }\end{array}\right]\)

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

Parameter "Tapered thread" in the "Input complete" mode
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G-code program parameters} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline & & & D & Cutting edge number & \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
P \\
\hline
\end{tabular} & \begin{tabular}{l}
- Thread pitch in mm/revolution \\
- Thread pitch in inch/revolution \\
- Thread turns per inch \\
- Thread pitch in MODULUS
\end{tabular} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev \\
turns/" \\
MODULUS
\end{tabular} \\
\hline G & \begin{tabular}{l}
Change in thread pitch per revolution - (only for \(\mathrm{P}=\mathrm{mm} / \mathrm{rev}\) or \(\mathrm{in} / \mathrm{rev}\) ) \\
\(G=0\) : The thread pitch \(P\) does not change. \\
\(G>0\) : The thread pitch \(P\) increases by the value \(G\) per revolution. \\
\(\mathrm{G}<0\) : The thread pitch \(P\) decreases by the value \(G\) per revolution. \\
If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows:
\[
G=\frac{\left|\mathrm{Pe}^{2}-\mathrm{P}^{2}\right|}{2{ }^{*} \mathrm{Z}_{1}}
\] \\
The meanings are as follows: \\
\(\mathrm{P}_{\mathrm{e}}\) : End pitch of thread [mm/rev] \\
P : Start pitch of thread [mm/rev] \\
\(Z_{1}\) : Thread length [mm] \\
A larger pitch results in a larger distance between the thread turns on the workpiece.
\end{tabular} & \(\mathrm{mm} / \mathrm{rev}^{2}\) \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Infeed (only for \(\nabla\) and \(\nabla\)
\[
+\nabla \nabla \nabla)
\]
\[
U
\] & \begin{tabular}{l}
- Linear: \\
Infeed with constant cutting depth \\
- Degressive: \\
Infeed with constant cutting cross-section
\end{tabular} & \\
\hline
\end{tabular}

Turning

\subsection*{9.2 Rotate}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Thread U & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline X0 & Reference point \(\mathrm{X} \varnothing\) (abs, always diameter) & mm \\
\hline Z0 & Reference point Z (abs) & mm \\
\hline X1 or X1a U & End point \(\mathrm{X} \varnothing\) (abs) or end point in relation to XO (inc) or Thread taper Incremental dimensions: The sign is also evaluated. & mm or degrees \\
\hline Z1 U & End point \(Z\) (abs) or end point in relation to \(Z 0\) (inc) Incremental dimensions: The sign is also evaluated. & mm \\
\hline \begin{tabular}{l}
LW \\
U \\
or \\
LW2 \\
U \\
or \\
\(L W 2=L R\) \\
U
\end{tabular} & \begin{tabular}{l}
Thread advance (inc) \\
The starting point for the thread is the reference point (XO, ZO) brought forward by the thread advance W . The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. \\
Thread run-in (inc) \\
The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). \\
Thread run-in = thread run-out (inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline LR & \begin{tabular}{l}
Thread run-out (inc) \\
The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).
\end{tabular} & mm \\
\hline H1 & Thread depth (inc) & mm \\
\hline \begin{tabular}{c} 
DP \\
\hline \\
\hline
\end{tabular} & \begin{tabular}{l}
Infeed slope as flank (inc) - (alternative to infeed slope as angle) \\
DP > 0: Infeed along the rear flank \\
DP < 0: Infeed along the front flank
\end{tabular} & mm \\
\hline \(\alpha P\) & \begin{tabular}{l}
Infeed slope as angle - (alternative to infeed slope as flank) \\
\(\alpha>0\) : Infeed along the rear flank \\
\(\alpha<0\) : Infeed along the front flank \\
\(\alpha=0\) : Infeed at right angle to cutting direction \\
If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.
\end{tabular} & Degrees \\
\hline \[
\begin{aligned}
& * \\
& * \\
& 0
\end{aligned}
\] & \begin{tabular}{l}
Infeed along the flank \\
Infeed with alternating flanks (alternative) \\
Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. \\
\(\alpha>0\) : Start at the rear flank \\
\(\alpha<0\) : Start at the front flank
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline D0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
Initial plunge depth - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) under "Manual Machine") \\
If you want to rework some threads, input the initial plunge depth D0 (inc.). This is the depth that was reached during a previous machining. \\
By inputting the plunge depth, you avoid unnecessary idle cuts when reworking the threads.
\end{tabular}} & mm \\
\hline \begin{tabular}{l}
D1 or ND U \\
(only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla)\)
\end{tabular} & \multicolumn{2}{|l|}{\begin{tabular}{l}
First infeed depth or number of roughing cuts \\
The respective value is displayed when you switch between the number of roughing cuts and the first infeed.
\end{tabular}} & mm \\
\hline U & \multicolumn{2}{|l|}{Finishing allowance in \(X\) and \(Z-\) (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) )} & mm \\
\hline NN & \multicolumn{2}{|l|}{Number of noncuts - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) )} & \\
\hline VR & \multicolumn{2}{|l|}{Return distance (inc)} & mm \\
\hline \multirow[t]{6}{*}{Multiple threads 0} & \multicolumn{2}{|l|}{No} & \\
\hline & a0 & Starting angle offset & Degrees \\
\hline & \multicolumn{2}{|l|}{} & \\
\hline & N & \begin{tabular}{l}
Number of thread turns \\
The thread turns are distributed evenly across the periphery of the turned part; the 1 st thread turn is always located at \(0^{\circ}\).
\end{tabular} & \\
\hline & DA & \begin{tabular}{l}
Thread changeover depth (inc) \\
First machine all thread turns sequentially to thread changeover depth DA, then machine all thread turns sequentially to depth 2 . DA, etc. until the final depth is reached. \\
DA \(=0\) : Thread changeover depth is not taken into account, i.e. finish machining each thread before starting the next thread.
\end{tabular} & mm \\
\hline & \begin{tabular}{l}
Machining: \\
U
\end{tabular} & \begin{tabular}{l}
- Complete, or \\
- from turn N 1 \\
N1 (1...4) start thread N1 = 1...N U or \\
- only thread NX \\
NX (1...4) 1 from \(N\) threads \(U\)
\end{tabular} & \\
\hline
\end{tabular}

\section*{Parameter "Tapered thread" in the "Input simple" mode}


\subsection*{9.2 Rotate}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline P
O & \begin{tabular}{l}
Select the thread pitch / turns for table "without" or specify the thread pitch/turns corresponding to the selection in the thread table: \\
- Thread pitch in mm/revolution \\
- Thread pitch in inch/revolution \\
- Thread turns per inch \\
- Thread pitch in MODULUS
\end{tabular} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev \\
turns/" \\
MODULUS
\end{tabular} \\
\hline Machining
\[
0
\] & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Infeed (only for \(\nabla\) and \(\nabla\) \(+\nabla \nabla \nabla\) ) & \begin{tabular}{l}
- Linear: Infeed with constant cutting depth \\
- Degressive: \\
Infeed with constant cutting cross-section
\end{tabular} & \\
\hline \begin{tabular}{l}
Thread \\
U
\end{tabular} & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline X0 & Reference point X \(\varnothing\) (abs, always diameter) & mm \\
\hline Z0 & Reference point \(Z\) (abs) & mm \\
\hline \[
\begin{aligned}
& \text { X1 or } \\
& \text { X1a }
\end{aligned}
\] & End point \(\mathrm{X} \varnothing\) (abs) or end point in relation to \(\mathrm{X0}\) (inc) or thread taper incremental dimensions: The sign is also evaluated. & mm or degrees \\
\hline Z1 U & End point \(Z\) (abs) or end point in relation to \(Z 0\) (inc) Incremental dimensions: The sign is also evaluated. & mm \\
\hline \begin{tabular}{l}
or LW2 \\
U \\
or
LW2 = LR \\
0
\end{tabular} & \begin{tabular}{l}
Thread advance (inc) \\
The starting point for the thread is the reference point (XO, ZO) brought forward by the thread advance W. The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. \\
Thread run-in (inc) \\
The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). \\
Thread run-in = thread run-out (inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline LR & \begin{tabular}{l}
Thread run-out (inc) \\
The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).
\end{tabular} & mm \\
\hline H1 & Thread depth (inc) & mm \\
\hline DP
U & \begin{tabular}{l}
Infeed slope as flank (inc) - (alternative to infeed slope as angle) DP > 0: Infeed along the rear flank \\
DP \(<0\) : Infeed along the front flank
\end{tabular} & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& \text { or } \\
& \text { aP }
\end{aligned}
\] & \begin{tabular}{l}
Infeed slope as angle - (alternative to infeed slope as flank) \\
\(\alpha>0\) : Infeed along the rear flank \\
\(\alpha<0\) : Infeed along the front flank \\
\(\alpha=0\) : Infeed at right angle to cutting direction \\
If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.
\end{tabular} & Degrees \\
\hline \[
* 0
\]
\[
* *
\] & \begin{tabular}{l}
Infeed along the flank \\
Infeed with alternating flanks (alternative) \\
Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. \\
\(\alpha>0\) : Start at the rear flank \\
\(\alpha<0\) : Start at the front flank
\end{tabular} & \\
\hline \begin{tabular}{l}
D1 or ND U \\
(only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla)\)
\end{tabular} & \begin{tabular}{l}
First infeed depth or number of roughing cuts \\
The respective value is displayed when you switch between the number of roughing cuts and the first infeed.
\end{tabular} & mm \\
\hline U & Finishing allowance in X and Z - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline NN & Number of noncuts - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|l|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline PL & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline G & \begin{tabular}{l} 
Change in thread pitch per revolution - (only for \(\mathrm{P}=\mathrm{mm} / \mathrm{rev}\) \\
or in/rev): \\
Without change in thread pitch
\end{tabular} & 0 & \\
\hline D0 & Initial plunge depth for reworking the threads & 0 mm & \\
\hline VR & Return distance & 2 mm & x \\
\hline Multiple threads & 1 thread & No & \\
\hline\(\alpha 0\) & Starting angle offset & \(0^{\circ}\) & \\
\hline
\end{tabular}

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Parameter "Face thread" in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G-code program parameters} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline \multirow[t]{3}{*}{PL} & Machining plane & & T & Tool name & \\
\hline & & & D & Cutting edge number & \\
\hline & & & S/V & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
P \\
\hline
\end{tabular} & \begin{tabular}{l}
- Thread pitch in \(\mathrm{mm} /\) revolution \\
- Thread pitch in inch/revolution \\
- Thread turns per inch \\
- Thread pitch in MODULUS
\end{tabular} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev \\
turns/" \\
MODULUS
\end{tabular} \\
\hline G & \begin{tabular}{l}
Change in thread pitch per revolution - (only for \(P=\mathrm{mm} / \mathrm{rev}\) or \(\mathrm{in} / \mathrm{rev}\) ) \\
\(\mathrm{G}=0\) : The thread pitch P does not change. \\
\(G>0\) : The thread pitch \(P\) increases by the value \(G\) per revolution. \\
\(G<0\) : The thread pitch \(P\) decreases by the value \(G\) per revolution. \\
If the start and end pitch of the thread are known, the pitch change to be programmed can be calculated as follows:
\[
G=\frac{\left|\mathrm{Pe}^{2}-\mathrm{P}^{2}\right|}{2 *--------Z_{1}}
\] \\
The meanings are as follows: \\
\(\mathrm{Pe}_{\mathrm{e}}\) : End pitch of thread [mm/rev] \\
P: Start pitch of thread [mm/rev] \\
\(\mathrm{Z}_{1}\) : Thread length [mm] \\
A larger pitch results in a larger distance between the thread turns on the workpiece.
\end{tabular} & \(\mathrm{mm} / \mathrm{rev}^{2}\) \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Infeed (only for \(\nabla\) and \(\nabla\) \(+\nabla \nabla \nabla\) ) & \begin{tabular}{l}
- Linear: \\
Infeed with constant cutting depth \\
- Degressive: \\
Infeed with constant cutting cross-section
\end{tabular} & \\
\hline Thread \(\mathbf{U}\) & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline X0 & Reference point \(\mathrm{X} \varnothing\) (abs, always diameter) & mm \\
\hline Z0 & Reference point \(Z\) (abs) & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline X1 U & End point of the thread \(\varnothing\) (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated. & mm \\
\hline \begin{tabular}{l}
LW \\
U \\
or \\
LW2 \\
U \\
or \\
\(L W 2=L R\) \\
U
\end{tabular} & \begin{tabular}{l}
Thread advance (inc) \\
The starting point for the thread is the reference point (XO, ZO) brought forward by the thread advance W . The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. \\
Thread run-in (inc) \\
The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). \\
Thread run-in = thread run-out (inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline LR & \begin{tabular}{l}
Thread run-out (inc) \\
The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).
\end{tabular} & mm \\
\hline H1 & Thread depth (inc) & mm \\
\hline \begin{tabular}{l} 
DP \\
\hline \\
\hline
\end{tabular} & \begin{tabular}{l}
Infeed slope as flank (inc) - (alternative to infeed slope as angle) \\
DP > 0: Infeed along the rear flank \\
DP < 0: Infeed along the front flank
\end{tabular} & \\
\hline \(\alpha \mathrm{P}\) & \begin{tabular}{l}
Infeed slope as angle - (alternative to infeed slope as flank) \\
\(\alpha>0\) : Infeed along the rear flank \\
\(\alpha<0\) : Infeed along the front flank \\
\(\alpha=0\) : Infeed at right angle to cutting direction \\
If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.
\end{tabular} & Degrees \\
\hline \[
\begin{aligned}
& * \\
& * \\
& * \\
& 0
\end{aligned}
\] & \begin{tabular}{l}
Infeed along the flank \\
Infeed with alternating flanks (alternative) \\
Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. \\
\(\alpha>0\) : Start at the rear flank \\
\(\alpha<0\) : Start at the front flank
\end{tabular} & \\
\hline D0 & \begin{tabular}{l}
Initial plunge depth - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) under "Manual Machine") \\
If you want to rework some threads, input the initial plunge depth D0 (inc.). This is the depth that was reached during a previous machining. \\
By inputting the plunge depth, you avoid unnecessary idle cuts when reworking the threads.
\end{tabular} & mm \\
\hline \begin{tabular}{l}
D1 or ND U \\
(only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla)\)
\end{tabular} & \begin{tabular}{l}
First infeed depth or number of roughing cuts \\
The respective value is displayed when you switch between the number of roughing cuts and the first infeed.
\end{tabular} & mm \\
\hline U & Finishing allowance in X and Z - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline NN & Number of noncuts - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & \\
\hline VR & Return distance (inc) & mm \\
\hline \multirow[t]{2}{*}{Multiple threads \(\cup\)} & No & \\
\hline & a0 \(\quad\) Starting angle offset & Degrees \\
\hline
\end{tabular}

Turning

\subsection*{9.2 Rotate}


\section*{Parameter "Face thread" in the "Input simple" mode}

\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline P & \begin{tabular}{l}
Select the thread pitch / turns for table "Without" or specify the thread pitch/turns corresponding to the selection in the thread table: \\
- Thread pitch in mm/revolution \\
- Thread pitch in inch/revolution \\
- Thread turns per inch \\
- Thread pitch in MODULUS
\end{tabular} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev \\
turns/" \\
MODULUS
\end{tabular} \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline \begin{tabular}{l}
Infeed (only for \(\nabla\) and \(\nabla\) \(+\nabla \nabla \nabla\) ) \\
0
\end{tabular} & \begin{tabular}{l}
- Linear: \\
Infeed with constant cutting depth \\
- Degressive: \\
Infeed with constant cutting cross-section
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Thread & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline X0 & Reference point \(\mathrm{X} \varnothing\) (abs, always diameter) & mm \\
\hline Z0 & Reference point Z (abs) & mm \\
\hline X1 U & End point of the thread (abs) or thread length (inc) Incremental dimensions: The sign is also evaluated. & mm \\
\hline \begin{tabular}{l}
LW \\
U \\
or \\
LW2 \\
U \\
or
\[
\begin{aligned}
& \mathrm{LW} 2=\mathrm{LR} \\
& \mathrm{U}
\end{aligned}
\]
\end{tabular} & \begin{tabular}{l}
Thread advance (inc) \\
The starting point for the thread is the reference point (XO,ZO) brought forward by the thread advance W . The thread advance can be used if you wish to begin the individual cuts slightly earlier in order to also produce a precise start of thread. \\
Thread run-in (inc) \\
The thread run-in can be used if you cannot approach the thread from the side and instead have to insert the tool into the material (e.g. lubrication groove on a shaft). \\
Thread run-in = thread run-out (inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline LR & \begin{tabular}{l}
Thread run-out (inc) \\
The thread run-out can be used if you wish to retract the tool obliquely at the end of the thread (e.g. lubrication groove on a shaft).
\end{tabular} & mm \\
\hline H1 & Thread depth from thread table (inc) & mm \\
\hline \[
\begin{aligned}
& \mathrm{DP} \\
& \mathrm{O}
\end{aligned}
\] & \begin{tabular}{l}
Infeed slope as flank (inc) - (alternative to infeed slope as angle) \\
DP > 0: Infeed along the rear flank \\
DP < 0: Infeed along the front flank
\end{tabular} & mm \\
\hline & \begin{tabular}{l}
Infeed slope as angle - (alternative to infeed slope as flank) \\
\(\alpha>0\) : Infeed along the rear flank \\
\(\alpha<0\) : Infeed along the front flank \\
\(\alpha=0\) : Infeed at right angle to cutting direction \\
If you wish to infeed along the flanks, the maximum absolute value of this parameter may be half the flank angle of the tool.
\end{tabular} & Degrees \\
\hline \[
\begin{array}{|l}
* * \\
*
\end{array}
\] & \begin{tabular}{l}
Infeed along the flank \\
Infeed with alternating flanks (alternative) \\
Instead of infeed along one flank, you can infeed along alternating flanks to avoid always loading the same tool cutting edge. As a consequence you can increase the tool life. \\
\(\alpha>0\) : Start at the rear flank \\
\(\alpha<0\) : Start at the front flank
\end{tabular} & \\
\hline \begin{tabular}{l}
D1 or ND U \\
(only for \(\nabla\) and
\[
\nabla+\nabla \nabla \nabla)
\]
\end{tabular} & \begin{tabular}{l}
First infeed depth or number of roughing cuts \\
The respective value is displayed when you switch between the number of roughing cuts and the first infeed.
\end{tabular} & mm \\
\hline U & Finishing allowance in X and Z - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline NN & Number of noncuts - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|l|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline PL & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline G & \begin{tabular}{l} 
Change in thread pitch per revolution - (only for \(P=\mathrm{mm} / \mathrm{rev}\) \\
or in/rev): \\
Without change in thread pitch
\end{tabular} & 0 & \\
\hline D0 & Initial plunge depth for reworking the threads & 0 mm & \\
\hline VR & Return distance & 2 mm & \\
\hline Multiple threads & 1 Thread & No & \\
\hline\(\alpha 0\) & Starting angle offset & \(0^{\circ}\) & \\
\hline
\end{tabular}

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{9.2.7 Thread chain (CYCLE98)}

\section*{Function}

With this cycle, you can produce several concatenated cylindrical or tapered threads with a constant pitch in longitudinal and face machining, all of which can have different thread pitches.

There may be single or multiple threads. With multiple threads, the individual thread turns are machined one after the other.

You define a right or left-hand thread by the direction of spindle rotation and the feed direction.

The infeed is performed automatically with a constant infeed depth or constant cutting crosssection.
- With a constant infeed depth, the cutting cross-section increases from cut to cut. The finishing allowance is machined in one cut after roughing.
A constant infeed depth can produce better cutting conditions at small thread depths.
- With a constant cutting cross-section, the cutting pressure remains constant over all roughing cuts and the infeed depth is reduced.
The feedrate override has no effect during traversing blocks with thread. The spindle override must not be changed during the thread machining.

\section*{Interruption of thread cutting}

You have the option to interrupt thread cutting (for example if the cutting tool is broken).
1. Press the <CYCLE STOP> key.

The tool is retracted from the thread and the spindle is stopped.
2. Replace the tool and press the <CYCLE START> key.

The aborted thread cutting is started again with the interrupted cut at the same depth.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction}
1. Approach of the starting point determined in the cycle at the beginning of the run-in path for the first thread with G0
2. Infeed for roughing according to the defined infeed type.
3. Thread cutting is repeated according to the programmed number of roughing cuts.
4. The finishing allowance is removed in the following step with G33.
5. This cut is repeated according to the number of noncuts.
6. The whole sequence of motions is repeated for each further thread.

\section*{Procedure for thread chain}


Thread

Thread chain
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Turning" softkey.
3. Press the "Thread" softkey.

The "Thread" input window opens.
4. Press the "Thread chain" softkey.

The "Thread Chain" input window opens.

\subsection*{9.2 Rotate}

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline \multirow[t]{2}{*}{SC} & Safety clearance & mm & D & Cutting edge number & \\
\hline & & & \[
\begin{aligned}
& S / V \\
& U
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Infeed (only for \(\nabla\) and \(\nabla\) \(+\nabla \nabla \nabla\) ) & \begin{tabular}{l}
- Linear: \\
Constant cutting depth infeed \\
- Degressive: \\
Constant cutting cross-section infeed
\end{tabular} & \\
\hline \begin{tabular}{l}
Thread \\
U
\end{tabular} & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline X0 & Reference point \(\mathrm{X} \varnothing\) (abs, always diameter) & mm \\
\hline zo & Reference point Z (abs) & mm \\
\hline P0 0 & Thread pitch 1 & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev \\
turns/" \\
MODULUS
\end{tabular} \\
\hline X1 or X1a & \begin{tabular}{l}
- Intermediate point \(1 \times \varnothing\) (abs) or \\
- Intermediate point 1 in relation to X 0 (inc) or \\
- Thread taper 1 \\
Incremental dimensions: The sign is also evaluated.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees
\end{tabular} \\
\hline Z1 0 & \begin{tabular}{l}
- Intermediate point 1 Z (abs) or \\
- Intermediate point 1 in relation to \(\mathrm{Z0}\) (inc)
\end{tabular} & mm \\
\hline P1 & Thread pitch 2 (unit as parameterized for P0) & \(\mathrm{mm} / \mathrm{rev}\) in/rev turns/" MODULUS \\
\hline X2 or X2a & \begin{tabular}{l}
- Intermediate point \(2 \times \varnothing\) (abs) or \\
- Intermediate point 2 in relation to X1 (inc) or \\
- Thread taper 2 \\
Incremental dimensions: The sign is also evaluated.
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees
\end{tabular} \\
\hline \[
\begin{aligned}
& \mathrm{Z2} \\
& 0
\end{aligned}
\] & \begin{tabular}{l}
- Intermediate point 2 Z (abs) or \\
- Intermediate point 2 in relation to Z 1 (inc)
\end{tabular} & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline P2 & Thread pitch 3 (unit as parameterized for P0) & \(\mathrm{mm} / \mathrm{rev}\) in/rev turns/" MODULUS \\
\hline X3 U & \begin{tabular}{l}
- End point \(X \varnothing\) (abs) or \\
- End point 3 in relation to \(X 2\) (inc) or \\
- Thread taper 3
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees
\end{tabular} \\
\hline Z3 U & \begin{tabular}{l}
- End point \(Z \varnothing\) (abs) or \\
- End point with reference to Z2 (inc)
\end{tabular} & mm \\
\hline LW & Thread run-in & mm \\
\hline LR & Thread run-out & mm \\
\hline H1 & Thread depth & mm \\
\hline DP or aP U & Infeed slope (flank) or infeed slope (angle) & mm or degrees \\
\hline \[
\begin{aligned}
& * * \\
& * * \\
& U
\end{aligned}
\] & \begin{tabular}{l}
- Infeed along a flank \\
- Infeed with alternating flanks
\end{tabular} & \\
\hline D1 or ND U & First infeed depth or number of roughing cuts - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline U & Finishing allowance in X and Z - (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & mm \\
\hline NN & Number of noncuts - (only for \(\nabla \nabla \nabla\) and \(\nabla+\nabla \nabla \nabla\) ) & \\
\hline VR & Return distance & mm \\
\hline \multirow[t]{5}{*}{Multiple threads 0} & No & \\
\hline & \begin{tabular}{|l|l} 
a0 & Starting angle offset \\
\hline
\end{tabular} & Degrees \\
\hline & Yes & \\
\hline & \begin{tabular}{|l|l} 
N & Number of thread turns \\
\hline
\end{tabular} & \\
\hline & DA \(\quad\) Thread changeover depth (inc) & mm \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}

\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (roughing and finishing)
\end{tabular} & \\
\hline Infeed (only for \(\nabla\) and \(\nabla\) \(+\nabla \nabla \nabla\) ) & \begin{tabular}{l}
- Linear: Infeed with constant cutting depth \\
- Degressive: \\
Infeed with constant cutting cross-section
\end{tabular} & \\
\hline \begin{tabular}{l}
Thread \\
u
\end{tabular} & \begin{tabular}{l}
- Internal thread \\
- External thread
\end{tabular} & \\
\hline X0 & Reference point \(\mathrm{X} \varnothing\) (abs, always diameter) & mm \\
\hline Z0 & Reference point Z (abs) & mm \\
\hline & Thread pitch 1 & \(\mathrm{mm} / \mathrm{rev}\) in/rev turns/" MODULUS \\
\hline \[
\begin{aligned}
& \mathrm{X} 1 \text { or } \\
& \mathrm{X} 1 a \mathrm{O}
\end{aligned}
\] & \begin{tabular}{l}
- Intermediate point \(1 \times \varnothing\) (abs) or \\
- Intermediate point 1 in relation to XO (inc) or Thread taper 1 \\
Incremental dimensions: The sign is also evaluated
\end{tabular} & mm Degrees \\
\hline Z10 & \begin{tabular}{l}
- Intermediate point 1 Z (abs) or \\
- Intermediate point 1 in relation to ZO (inc)
\end{tabular} & mm \\
\hline P1 U & Thread pitch 2 (unit as parameterized for P0) & \(\mathrm{mm} / \mathrm{rev}\) in/rev turns/" MODULUS \\
\hline \[
\begin{aligned}
& \mathrm{X} 2 \text { or } \\
& \mathrm{x} 2 \alpha \mathrm{O}
\end{aligned}
\] & \begin{tabular}{l}
- Intermediate point \(2 \times \varnothing\) (abs) or \\
- Intermediate point 2 in relation to X1 (inc) or Thread taper 2 \\
Incremental dimensions: The sign is also evaluated
\end{tabular} & mm Degrees \\
\hline Z2 U & \begin{tabular}{l}
- Intermediate point 2 Z (abs) or \\
- Intermediate point 2 in relation to \(\mathrm{Z1}\) (inc)
\end{tabular} & mm \\
\hline P2 U & Thread pitch 3 (unit as parameterized for P0) & \(\mathrm{mm} / \mathrm{rev}\) in/rev turns/" MODULUS \\
\hline X3 0 & \begin{tabular}{l}
- End point \(X \varnothing\) (abs) or \\
- End point 3 in relation to X2 (inc) or \\
- Thread taper 3
\end{tabular} & mm Degrees \\
\hline Z3 0 & \begin{tabular}{l}
- End point \(Z \varnothing\) (abs) or \\
- End point with reference to Z2 (inc)
\end{tabular} & mm \\
\hline LW & Thread advance (inc) & mm \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline LR & Thread run-out (inc) & mm \\
\hline H1 & Thread depth (inc) & \begin{tabular}{l}
mm \\
\hline \begin{tabular}{l} 
DP or aP \\
U
\end{tabular} \\
\hline \multirow{3}{*}{}
\end{tabular} \\
\hline Inced slope (flank) or infeed slope (angle) \\
\hline D1 or ND U & \begin{tabular}{l} 
Unfed along the flank
\end{tabular} \\
\hline Unfed with alternating flanks & First indeed depth or number of roughing cuts (only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla)\) & \\
\hline ND & Finishing allowance in \(X\) and \(Z-(\) only for \(\nabla\) and \(\nabla+\nabla \nabla \nabla)\) & mm \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline PL & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline YR & Return distance & 2 mm & x \\
\hline Multiple threads & 1 Thread & No & \\
\hline an & Starting angle offset & \(0^{\circ}\) & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.2.8 Cut-off (CYCLE92)}

\section*{Function}

The "Cut-off" cycle is used when you want to cut off dynamically balanced parts (e.g. screws, bolts, or pipes).
You can program a chamfer or rounding on the edge of the machined part. You can machine at a constant cutting rate \(V\) or speed \(S\) up to a depth X 1 , from which point the workpiece is machined at a constant speed. As of depth X1, you can also program a reduced feedrate FR or a reduced speed \(S R\), in order to adapt the velocity to the smaller diameter.

Use parameter X2 to enter the final depth that you wish to reach with the cut-off. With pipes, for example, you do not need to cut-off until you reach the center; cutting off slightly more than the wall thickness of the pipe is sufficient.

\section*{Approach/retraction}
1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The chamfer or radius is machined at the machining feedrate.
3. Cut-off down to depth X 1 is performed at the machining feedrate.
4. Cut-off is continued down to depth \(X 2\) at reduced feedrate \(F R\) and reduced speed \(S R\).
5. The tool moves back to the safety distance at rapid traverse.

If your turning machine is appropriately set up, you can extend a workpiece drawer (part catcher) to accept the cut-off workpiece. Extension of the workpiece drawer must be enabled in a machine data element.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Cutoff
2. Press the "Turning" softkey.
3. Press the "Cut-off" softkey.

The "Cut-off" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline PL & Machining plane & & T & Tool name & \\
\hline SC & Safety clearance & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & F & Feedrate & mm/rev \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
DIR U \\
(only for G code)
\end{tabular} & Direction of spindle rotation & \\
\hline S & Spindle speed & \(\mathrm{rev} / \mathrm{min}\) \\
\hline V & Constant cutting rate & \(\mathrm{m} / \mathrm{min}\) \\
\hline SV & Maximum speed limit - (only for constant cutting rate V ) & \(\mathrm{rev} / \mathrm{min}\) \\
\hline X0 & Reference point in \(\mathrm{X} \varnothing\) (abs, always diameter) & mm \\
\hline Z0 & Reference point in Z (abs) & mm \\
\hline FS or R U & Chamfer width or rounding radius & mm \\
\hline X1 U & Depth for speed reduction \(\varnothing\) (abs) or depth for speed reduction in relation to X0 (inc) & mm \\
\hline \begin{tabular}{l}
FR \\
(only for ShopTurn)
\end{tabular} & Reduced feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline \[
\begin{array}{|l|}
\hline \text { FR } \\
\text { (only for G code) } \\
\hline
\end{array}
\] & & * \\
\hline SR & Reduced speed & \(\mathrm{rev} / \mathrm{min}\) \\
\hline X2 U & Final depth \(\varnothing\) (abs) or final depth in relation to X1 (inc) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\subsection*{9.3 Contour turning}

\subsection*{9.3.1 General information}

\section*{Function}

You can machine simple or complex contours with the "Contour turning" cycle. A contour comprises separate contour elements, whereby at least two and up to 250 elements result in a defined contour.

You can program chamfers, radii, undercuts or tangential transitions between the contour elements.

The integrated contour calculator calculates the intersection points of the individual contour elements taking into account the geometrical relationships, which allows you to enter incompletely dimensioned elements.
When you machine the contour, you can make allowance for a blank contour which must be entered before the finished-part contour. Then select one of the the following machining technologies:
- Stock removal
- Grooving
- Plunge-turning

You can rough, remove residual material and finish for each of the three technologies above.

\section*{Programming}

For example, the programming procedure for stock removal is as follows:

\section*{Note}

When programming in G code, it must be ensured that the contours are located after the end of program identifier!
1. Enter the blank contour

If, when removing stock along the contour, you want to take into account a blank contour (and no cylinder or no allowance) as blank shape, then you must define the contour of the blank before you define the finished-part contour. Compile the blank contour step-by-step from various contour elements.
2. Enter finished-part contour

You build up the finished-part contour gradually from a series of different contour elements.
3. Contour call - only for \(G\) code program
4. Stock removal along the contour (roughing)

The contour is machined longitudinally, transversely or parallel to the contour.
5. Remove residual material (roughing)

When removing stock along the contour, ShopTurn automatically detects residual material that has been left. For \(G\) code programming, when removing stock, it must first be decided whether to machine with residual material detection - or not. A suitable tool will allow you to remove this without having to machine the contour again.
6. Stock removal along the contour (finishing)

If you programmed a finishing allowance for roughing, the contour is machined again.

\subsection*{9.3.2 Representation of the contour}

\section*{G code program}

In the editor, the contour is represented in a program section using individual program blocks. If you open an individual block, then the contour is opened.

\section*{ShopTurn program}

The cycle represents a contour as a program block in the program. If you open this block, the individual contour elements are listed symbolically and displayed in broken-line graphics.

\section*{Symbolic representation}

The individual contour elements are represented by symbols adjacent to the graphics window. They appear in the order in which they were entered.
\begin{tabular}{|l|l|l|}
\hline Contour element & Symbol & Meaning \\
\hline Starting point & & Starting point of the contour \\
\hline Straight line up & & Straight line in \(90^{\circ}\) grid \\
\hline Straight line down & & Straight line in \(90^{\circ}\) grid \\
\hline Straight line left & & Straight line in \(90^{\circ}\) grid \\
\hline Straight line right & & Straight line with any gradient \\
\hline Straight line in any direction & & Circle \\
\hline Arc right & & Circle \\
\hline Arc left & & \begin{tabular}{l} 
Straight diagonal or circle in \\
polar coordinates
\end{tabular} \\
\hline Pole & & End of contour definition \\
\hline Finish contour & & \\
\hline
\end{tabular}

The different colors of the symbols indicate their status:
\begin{tabular}{|l|c|l|}
\hline Foreground & Background & Meaning \\
\hline Black & Blue & Cursor on active element \\
\hline Black & Orange & Cursor on current element \\
\hline Black & White & Normal element \\
\hline Red & White & \begin{tabular}{l} 
Element not currently evaluated \\
(element will only be evaluated \\
when it is selected with the \\
cursor)
\end{tabular} \\
\hline
\end{tabular}

\section*{Graphic display}

The progress of contour programming is shown in broken-line graphics while the contour elements are being entered.

When the contour element has been created, it can be displayed in different line styles and colors:
- Black: Programmed contour
- Orange: Current contour element
- Green dashed: Alternative element
- Blue dotted: Partially defined element

The scaling of the coordinate system is adjusted automatically to match the complete contour.

The position of the coordinate system is displayed in the graphics window.

\subsection*{9.3.3 Creating a new contour}

\section*{Function}

For each contour that you want to cut, you must create a new contour.
The first step in creating a contour is to specify a starting point. Enter the contour element. The contour processor then automatically defines the end of the contour.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

2. Press the "Contour turning" softkey.
3. Press the "Contour" and "New contour" softkeys. The "New Contour" input window opens.
4. Enter a name for the new contour. The contour name must be unique.
5. Press the "Accept" softkey.

The input window for the starting point of the contour appears.
Enter the individual contour elements (see Section "Creating contour elements").
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Z & Starting point Z (abs) & mm \\
\hline X & Starting point \(\mathrm{X} \varnothing\) (abs) & mm \\
\hline \begin{tabular}{l}
Transition to contour start \\
0
\end{tabular} & \begin{tabular}{l}
Type of transition \\
- Radius \\
- Chamfer \\
FS=0 or R=0: No transition element
\end{tabular} & \\
\hline R & Transition to following element - radius & mm \\
\hline FS & Transition to following element - chamfer & mm \\
\hline \begin{tabular}{l}
Direction in front of the contour \\
U
\end{tabular} & \begin{tabular}{l}
Direction of the contour element towards the starting point: \\
- In the negative direction of the horizontal axis \\
- In the positive direction of the horizontal axis
 \\
- In the negative direction of the vertical axis
 \\
- In the positive direction of the vertical axis
\end{tabular} & \\
\hline Additional commands & \begin{tabular}{l}
You can enter additional commands in the form of \(G\) code for each contour element. You can enter the additional commands (max. 40 characters) in the extended parameter screens ("All parameters" softkey). The softkey is always available at the starting point, it only has to be pressed when entering additional contour elements. \\
You can program feedrates and \(M\) commands, for example, using additional \(G\) code commands. However, carefully ensure that the additional commands do not collide with the generated G code of the contour and are compatible with the machining type required. Therefore, do not use any G code commands of group 1 (G0, G1, G2, G3), no coordinates in the plane and no \(G\) code commands that have to be programmed in a separate block. \\
The contour is finished in continuous-path mode (G64). As a result, contour transitions such as corners, chamfers or radii may not be machined precisely. \\
If you wish to avoid this, then it is possible to use additional commands when programming. \\
Example: For a contour, first program the straight X parallel and then enter "G9" (nonmodal exact stop) for the additional command parameter. Then program the Z-parallel straight line. The corner will be machined exactly, as the feedrate at the end of the X parallel straight line is briefly zero. \\
Note: \\
The additional commands are only effective for finishing!
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.3.4 Creating contour elements}

\section*{Creating contour elements}

After you have created a new contour and specified the starting point, you can define the individual elements that make up the contour.

The following contour elements are available for the definition of a contour:
- Straight vertical line
- Straight horizontal line
- Diagonal line
- Circle/arc

For each contour element, you must parameterize a separate parameter screen. Parameter entry is supported by various help screens that explain the parameters.

If you leave certain fields blank, the cycle assumes that the values are unknown and attempts to calculate them from other parameters.

Conflicts may result if you enter more parameters than are absolutely necessary for a contour. In such a case, try entering less parameters and allowing the cycle to calculate as many parameters as possible.

\section*{Contour transition elements}

As transition element between two contour elements, you can select a radius or a chamfer or, in the case of linear contour elements, an undercut. The transition element is always attached at the end of a contour element. The contour transition element is selected in the parameter screen of the respective contour element.

You can use a contour transition element whenever there is an intersection between two successive elements which can be calculated from the input values. Otherwise you must use the straight/circle contour elements.

\section*{Additional commands}

You can enter additional commands in the form of \(G\) code for each contour element. You can enter the additional commands (max. 40 characters) in the extended parameter screens ("All parameters" softkey).

You can program feedrates and M commands, for example, using additional \(G\) code commands. However, make sure that the additional commands do not collide with the generated \(G\) code of the contour. Therefore, do not use any \(G\) code commands of group 1 (G0, G1, G2, G3), no coordinates in the plane and no G code commands that have to be programmed in a separate block.

\section*{Additional functions}

The following additional functions are available for programming a contour:
- Tangent to preceding element

You can program the transition to the preceding element as tangent.
- Selecting a dialog box

If two different possible contours result from the parameters entered thus far, one of the options must be selected.
- Close contour

From the actual position, you can close the contour with a straight line to the starting point.

\section*{Producing exact contour transitions}

The axis moves in the continuous path mode (G64). As a result, contour transitions such as corners, chamfers or radii may not be machined precisely.

If you wish to avoid this, there are two different options when programming. Use the additional programs or program the special feedrate for the transition element.
- Additional command

For a contour, first program the vertical straight line and then enter "G9" (non-modal exact stop) for the additional command parameter. Then program the horizontal straight line. The corner will be machined exactly, since the feedrate at the end of the vertical straight line is briefly zero.
- Feedrate, transition element

If you have chosen a chamfer or a radius as the transition element, enter a reduced feedrate in the "FRC" parameter. The slower machining rate means that the transition element is machined more accurately.

\section*{Procedure for entering contour elements}
1. The part program is opened. Position the cursor at the required input position, this is generally at the physical end of the program after M02 or M30.
2. Contour input using contour support:

2.1 Press the "Contour turning", "Contour" and "New contour" softkeys.
2.2 In the opened input window, enter a name for the contour, e.g. contour_1.
Press the "Accept" softkey.

2.3 The input screen to enter the contour opens, in which you initially enter a starting point for the contour. This is marked in the lefthand navigation bar using the "+" symbol.
Press the "Accept" softkey.
3. Enter the individual contour elements of the machining direction.

Select a contour element via softkey.
The "Straight line (e.g. Z)" input window opens.
- OR

The "Straight line (e.g. X)" input window opens.
- OR

The "Straight line (e.g. ZX)" input window opens.
- OR

The "Circle" input window opens.
4. Enter all the data available from the workpiece drawing in the input screen (e.g. length of straight line, target position, transition to next element, angle of lead, etc.).
5. Press the "Accept" softkey.

The contour element is added to the contour.
6. When entering data for a contour element, you can program the transition to the preceding element as a tangent.
Press the "Tangent to prec. elem." softkey. The "tangential" selection appears in the parameter \(\alpha 2\) entry field.
7. Repeat the procedure until the contour is complete.
8. Press the "Accept" softkey.

The programmed contour is transferred into the process plan (program view).

\section*{Contour element "Straight line e.g. Z"}
\begin{tabular}{|c|c|c|c|c|}
\hline Parameters & \multicolumn{3}{|l|}{Description} & Unit \\
\hline ZU & \multicolumn{3}{|l|}{End point Z (abs or inc)} & mm \\
\hline a1 & \multicolumn{3}{|l|}{Starting angle to \(Z\) axis} & Degrees \\
\hline a2 & \multicolumn{3}{|l|}{Angle to the preceding element} & Degrees \\
\hline Transition to next element \(U\) & \multicolumn{3}{|l|}{\begin{tabular}{l}
Type of transition \\
- Radius \\
- Undercut \\
- Chamfer
\end{tabular}} & \\
\hline Radius & R \(\quad\) Trans & \multicolumn{2}{|l|}{Transition to following element - radius} & mm \\
\hline Undercut U & Form E & \multicolumn{2}{|l|}{Undercut size U e.g. E1.0x0.4} & \\
\hline & Form F & \multicolumn{2}{|l|}{Undercut size \(\cup\) e.g. F0.6x0.3} & \\
\hline & DIN thread & \[
\begin{aligned}
& P \\
& \alpha
\end{aligned}
\] & Thread pitch Insertion angle & \(\mathrm{mm} / \mathrm{rev}\) Degrees \\
\hline & Thread & \begin{tabular}{l} 
Z1 \\
Z2 \\
R1 \\
R2 \\
T \\
\hline
\end{tabular} & \begin{tabular}{l}
Length Z1 \\
Length Z2 \\
Radius R1 \\
Radius R2 \\
Insertion depth
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline Chamfer & FS \(\quad\) Transition to following element - chamfer & \multicolumn{2}{|l|}{Transition to following element - chamfer} & mm \\
\hline CA & \multicolumn{3}{|l|}{\begin{tabular}{l}
Grinding allowance \\
- \(\dagger\) Grinding allowance to right of contour \\
- \(\dagger\) Grinding allowance to left of contour
\end{tabular}} & mm \\
\hline Additional commands & \multicolumn{3}{|l|}{Additional G code commands} & \\
\hline
\end{tabular}

\section*{Contour element "Straight line e.g. X"}
\begin{tabular}{|c|c|c|c|}
\hline Parameters & \multicolumn{2}{|l|}{Description} & Unit \\
\hline XU & \multicolumn{2}{|l|}{End point \(\mathrm{X} \varnothing\) (abs) or end point X (inc)} & mm \\
\hline a1 & \multicolumn{2}{|l|}{Starting angle to \(Z\) axis} & Degrees \\
\hline 人2 & \multicolumn{2}{|l|}{Angle to the preceding element} & Degrees \\
\hline Transition to next element \(U\) & \multicolumn{2}{|l|}{\begin{tabular}{l}
Type of transition \\
- Radius \\
- Undercut \\
- Chamfer
\end{tabular}} & \\
\hline Radius & R \(\quad\) T & n to following element - radius & mm \\
\hline Undercut U & Form E & Undercut size \(\mathbf{U}\) e.g. E1.0x0.4 & \\
\hline
\end{tabular}

Turning

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|c|c|}
\hline Parameters & \multicolumn{3}{|l|}{Description} & Unit \\
\hline & Form F & \multicolumn{2}{|l|}{Undercut size \(\mathbf{U}\) e.g. F0.6x0.3} & \\
\hline & DIN thread & \[
\begin{aligned}
& P \\
& \alpha
\end{aligned}
\] & Thread pitch Insertion angle & \(\mathrm{mm} / \mathrm{rev}\) Degrees \\
\hline & Thread & \[
\begin{aligned}
& \mathrm{Z} 1 \\
& \text { Z2 } \\
& \mathrm{R} 1 \\
& \mathrm{R} 2 \\
& \mathrm{~T}
\end{aligned}
\] & \begin{tabular}{l}
Length Z1 \\
Length Z2 \\
Radius R1 \\
Radius R2 \\
Insertion depth
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline Chamfer & \multicolumn{3}{|r|}{Transition to following element - chamfer} & mm \\
\hline CA & \multicolumn{3}{|l|}{\begin{tabular}{l}
Grinding allowance \(U\) \\
- \(\dagger\) Grinding allowance to right of contour \\
- \(\quad 1\) Grinding allowance to left of contour
\end{tabular}} & mm \\
\hline Additional commands & \multicolumn{3}{|l|}{Additional G code commands} & \\
\hline
\end{tabular}

\section*{Contour element "Straight line e.g. ZX"}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline ZU & End point Z (abs or inc) & mm \\
\hline XU & End point \(\mathrm{X} \varnothing\) (abs) or end point X (inc) & mm \\
\hline a1 & Starting angle to Z axis & Degrees \\
\hline a2 & Angle to the preceding element & Degrees \\
\hline Transition to next element \(U\) & \begin{tabular}{l}
Type of transition \\
- Radius \\
- Chamfer
\end{tabular} & \\
\hline Radius & R \(\quad\) Transition to following element - radius & mm \\
\hline Chamfer & FS Transition to following element - chamfer & mm \\
\hline CA & \begin{tabular}{l}
Grinding allowance \(U\) \\
- \(\dagger\) Grinding allowance to right of contour \\
- IT Grinding allowance to left of contour
\end{tabular} & mm \\
\hline Additional commands & Additional G code commands & \\
\hline
\end{tabular}

\section*{Contour element "Circle"}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Direction of rotation U & \begin{tabular}{l}
- Clockwise direction of rotation \\
- Counterclockwise direction of rotation
\end{tabular} & \\
\hline ZU & End point Z (abs or inc) & mm \\
\hline XU & End point \(\mathrm{X} \varnothing\) (abs) or end point X (inc) & mm \\
\hline KU & Circle center point K (abs or inc) & mm \\
\hline 10 & Circle center point I \(\varnothing\) (abs or circle center point I (inc) & mm \\
\hline a1 & Starting angle to Z axis & Degrees \\
\hline \(\beta 1\) & End angle to \(Z\) axis & Degrees \\
\hline \(\beta 2\) & Opening angle & Degrees \\
\hline Transition to next element \(U\) & \begin{tabular}{l}
Type of transition \\
- Radius \\
- Chamfer
\end{tabular} & \\
\hline Radius & \(\mathrm{R} \quad\) Transition to following element - radius & mm \\
\hline Chamfer & FS Transition to following element - chamfer & mm \\
\hline CA & \begin{tabular}{l}
Grinding allowance \\
- \(\dagger\) Grinding allowance to right of contour \\
- IT Grinding allowance to left of contour
\end{tabular} & mm \\
\hline Additional commands & Additional G code commands & \\
\hline
\end{tabular}

\section*{Contour element "End"}

The data for the transition at the contour end of the previous contour element is displayed in the "End" parameter screen.

The values cannot be edited.

\subsection*{9.3.5 Entering the master dimension}

If you would like to finish your workpiece to an exact fit, you can input the master dimension directly into the parameter screen form during programming.
Specify the master dimension as follows:
F<Diameter/Length> <Tolerance class> <Tolerance quality>
"F" identifies that a master dimension follows, i.e. in this case, a hole.
Example: F20h7
Possible tolerance classes:
A, B, C, D, E, F, G, H, J, T, U, V, X, Y, Z
Upper-case characters: Holes
Lower case letters: Shafts
Possible tolerance qualities:
1 to 18 , if they are not restricted by DIN standard 7150 .

Fit calculator
A fit calculator supports you when making entries.

\section*{Procedure}
1. Position the cursor on the desired entry field.
2. Press the <=> key.
4. Enter the diameter or length value in the first field.
5. In the second field, select the tolerance class and in the third field, enter the tolerance quality.
6. Press the equals symbol on the calculator.

\section*{Calculate}


Press the "Calculate" softkey.
- OR -

Press the <INPUT> key.

The new value is calculated and displayed in the entry field of the calculator.
Press the "Accept" softkey.
The calculated value is accepted and displayed in the entry field of the window.

Rejecting entries
Delete
Press the "Delete" softkey to reject your entries.

\subsection*{9.3.6 Changing the contour}

\section*{Function}

You can change a previously created contour later.
Individual contour elements can be
- added,
- changed,
- inserted or
- deleted.

\section*{Procedure for changing a contour element}
1. Open the part program or ShopTurn program to be executed.
2. With the cursor, select the program block where you want to change the contour. Open the geometry processor.
The individual contour elements are listed.
3. Position the cursor at the position where a contour element is to be inserted or changed.
4. Select the desired contour element with the cursor.
5. Enter the parameters in the input screen or delete the element and select a new element.
6. Press the "Accept" softkey.

The desired contour element is inserted in the contour or changed.

\section*{Procedure for deleting a contour element}
1. Open the part program or ShopTurn program to be executed.
2. Position the cursor on the contour element that you want to delete.

Delete element
3. Press the "Delete element" softkey.
4. Press the "Delete" softkey.

\subsection*{9.3.7 Contour call (CYCLE62) - only for \(G\) code program}

\section*{Function}

The input creates a reference to the selected contour.
There are four ways to call the contour:
1. Contour name

The contour is in the calling main program.
2. Labels

The contour is in the calling main program and is limited by the labels that have been entered.
3. Subprogram

The contour is located in a subprogram in the same workpiece.
4. Labels in the subprogram

The contour is in a subprogram and is limited by the labels that have been entered.

\section*{Procedure}
1. The part program to be executed has been created and you are in the editor.

2. Press the "Contour turning" softkey.
3. Press the "Contour" and "Contour call" softkeys. The "Contour Call" input window opens.
4. Assign parameters to the contour selection.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline Contour selection & \(\bullet\) Contour name & \(\bullet\) Labels \\
U & \(\bullet\) Subprogram \\
& \(\bullet\) Labels in the subprogram & \\
\hline Contour name & CON: Contour name & \\
\hline Labels & \(\bullet\) LAB1: Label 1 \\
& \(\bullet\) LAB2: Label 2 & \\
\hline Subprogram & PRG: Subprogram & \\
\hline \begin{tabular}{l} 
Labels in the subpro- \\
gram
\end{tabular} & \(\bullet\) PRG: Subprogram & LAB1: Label 1 \\
& \(\bullet\) & LAB2: Label 2
\end{tabular}

\subsection*{9.3.8 Stock removal (CYCLE952)}

\section*{Function}

For stock removal, the cycle takes into account a blank that can comprise a cylinder, an allowance on the finished-part contour or any blank contour. You must define a blank contour as a separate closed contour in advance of the finished-part contour.

\section*{Precondition}

For a G code program, at least one CYCLE62 is required before CYCLE952.
If CYCLE62 is only present once, then this involves the finished part contour.
If CYCLE62 is present twice, then the first call is the blank contour and the second call is the finished-part contour (also see Chapter "Programming (Page 390)").

\section*{Note}

\section*{Execution from external media}

If you execute programs from an external drive (e.g. local drive or network drive), then you require the execution from external storage function (EES).

For additional information, please refer to the following documentation:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Rule}

Using the execution from external storage function (EES) you have the option of accessing any logical drive with part programs and directly executing these the same as via the NC program memory

\section*{Rounding the contour}

In order to avoid residual corners during roughing, you can enable the "Always round the contour" function. This will remove the protrusions that are always left at the end with each cut (due to the cut geometry). The "Round to the previous intersection" setting accelerates machining of the contour. However, any resulting residual corners will not be recognized or machined. Thus, it is imperative that you check the behavior before machining using the simulation.

When set to "automatic", rounding is always performed if the angle between the cutting edge and the contour exceeds a certain value. The angle is set in a machine data element.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Alternating cutting depth}

Instead of working with constant cutting depth D, you can use an alternating cutting depth to vary the load on the tool edge. As a consequence you can increase the tool life.

The percentage for the alternating cutting depth is saved in a machine data element.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Cut segmentation}

To avoid the occurrence of very thin cuts in cut segmentation due to contour edges, you can align the cut segmentation to the contour edges. During machining the contour is then divided by the edges into individual sections and cut segmentation is performed separately for each section.

\section*{Set machining area limits}

If, for example, you want to machine a certain area of the contour with a different tool, you can set machining area limits so that machining only takes place in the area of the contour you have selected. You can define between 1 and 4 limit lines.

\section*{Feedrate interruption}

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption. Parameter DI specifies the distance after which the feedrate interruption should occur.
The interruption time or retraction distance is defined in machine data.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Residual material machining / naming conventions}

\section*{G code program}

For multi-channel systems, cycles attach a "_C" and a two-digit number of the specific channel to the names of the programs to be generated, e.g. for channel 1 "_C01".

This is the reason that the name of the main program must not end with "_C" and a two-digit number. This is monitored by the cycles.

For programs with residual machining, when specifying the name for the file, which includes the updated blank contour, it must be ensured that this does not have the attached characters ("_C" and double-digit number).

For single-channel systems, cycles do not extend the name for the programs to be generated.

\section*{Note}

\section*{G code programs}

For \(G\) code programs, the programs to be generated, which do not include any path data, are saved in the directory in which the main program is located. In this case, it must be ensured that programs, which already exist in the directory and which have the same name as the programs to be generated, are overwritten.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Machining type}

You can freely select the machining type (roughing, finishing or complete machining roughing + finishing). During contour roughing, parallel cuts of maximum programmed infeed depth are created. Roughing is performed to the programmed allowance.

You can also specify a compensation allowance U1 for finishing operations, which allows you to either finish several times (positive allowance) or to shrink the contour (negative allowance). Finishing is performed in the same direction as roughing.

\section*{Procedure}

\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining
\[
0
\] & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (complete machining)
\end{tabular} & \\
\hline Machining direction \(U\) & \begin{tabular}{l}
- Face U \\
- Longitudinal U \\
- Parallel to the contour U \\
- From inside to outside \\
- From outside to inside \\
- From end face to rear side \\
- From rear side to end face
\end{tabular} & \\
\hline & The machining direction depends on the stock removal direction and choice of tool. & \\
\hline Position 0 & \begin{tabular}{l}
- Front \\
- Rear \\
- Inside \\
- Outside
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline D & Maximum depth infeed - (only for \(\overline{\text { ) }}\) & mm \\
\hline DX & Maximum depth infeed - (only for parallel to the contour, as an alternative to D). & mm \\
\hline \[
\begin{aligned}
& \|_{t} \\
& \left.\right|_{\leftarrow} \\
& i_{i} \\
& 0
\end{aligned}
\] & Always round on the contour Never round on the contour Only round to the previous intersection. & \\
\hline \[
\begin{aligned}
& \stackrel{4}{4+} \\
& \stackrel{4}{4} \\
& 0
\end{aligned}
\] & Uniform cut segmentation Round cut segmentation at the edge & \\
\hline \[
\begin{aligned}
& \leftleftarrows \\
& \leftarrow \\
& \leftarrow
\end{aligned}
\] & \begin{tabular}{l}
Constant cutting depth \\
Alternating cutting depth - (only with align cut segmentation to edge)
\end{tabular} & \\
\hline DZ & Maximum depth infeed - (only for position parallel to the contour and UX) & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and Z - (only for \(\nabla\) ) & mm \\
\hline UZ & Finishing allowance in Z - (only for UX) & mm \\
\hline DI & For zero: Continuous cut - (only for \(\nabla\) ) & mm \\
\hline BL O & \begin{tabular}{l}
Blank description (only for \(\nabla\) ) \\
- Cylinder (described using XD, ZD) \\
- Allowance (XD and ZD on the finished part contour) \\
- Contour (additional CYCLE62 call with blank contour - e.g. cast iron mold)
\end{tabular} & \\
\hline XD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension \(\varnothing\) (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline ZD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline Allowance \(\cup\) & \begin{tabular}{l}
Allowance for pre-finishing - (only for \(\nabla \nabla \nabla\) ) \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in X and Z direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is retained \\
- Negative value: Compensation allowance is removed in addition to finishing allowance
\end{tabular} & mm \\
\hline Set machining area limits \(U\) & \begin{tabular}{l}
Set machining area limits \\
- Yes \\
- No
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { XA } \\
& \text { XB } O \\
& Z A \\
& Z B C \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
With limited machining area only, yes: \\
1st limit XA \(\varnothing\) \\
2nd limit XB \(\varnothing\) (abs) or 2nd limit referred to XA (inc) \\
1st limit ZA \\
2nd limit ZB (abs) or 2nd limit referred to ZA (inc)
\end{tabular} & mm \\
\hline Relief cuts \(U\) & \begin{tabular}{l}
Machine relief cuts \\
- Yes \\
- No
\end{tabular} & \\
\hline FR & Insertion feedrate, relief cuts & \\
\hline
\end{tabular}

\subsection*{9.3 Contour turning}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{simple} \\
\hline PRG & \multicolumn{2}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline & & & D & Cutting edge number & \\
\hline RP & Retraction plane - (only for machining direction, longitudinal, inner) & mm & F & Feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline F & Feedrate & * & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline Machining U & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla+\nabla \nabla \nabla\) (complete mac
\end{tabular} & hining) & \\
\hline \multirow[t]{2}{*}{Machining direction U} & \begin{tabular}{l}
- face \(U\) \\
- longitudinal \(U\) \\
- parallel to the contour U
\end{tabular} & \begin{tabular}{l}
- from inside to outside
\(\square\) \(\dagger\) \\
- from outside to inside \\
- from end face to rear side \\
- from rear side to end face
\end{tabular} & \\
\hline & \multicolumn{2}{|l|}{The machining direction depends on the stock removal direction and choice of tool.} & \\
\hline Position \(U\) & \begin{tabular}{l}
- front \\
- back \\
- inside \\
- outside
\end{tabular} & & \\
\hline D & Maximum depth infeed - ( & only for \(\nabla\) ) & mm \\
\hline DX & Maximum depth infeed - & (only for parallel to the contour, as an alternative to D) & mm \\
\hline DZ & Maximum depth infeed - ( & (only for position parallel to the contour and UX) & mm \\
\hline UX or U U & Finishing allowance in X or & r finishing allowance in X and Z - (only for \(\nabla\) ) & mm \\
\hline UZ & Finishing allowance in Z - & (only for UX) & mm \\
\hline BL U & \begin{tabular}{l}
Blank description (only for \\
- Cylinder (described us \\
- Allowance (XD and ZD \\
- Contour (additional CY
\end{tabular} & \begin{tabular}{l}
\(\nabla\) ) \\
ing XD, ZD) \\
on the finished part contour) \\
CLE62 call with blank contour - e.g. cast iron mold)
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline XD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension \(\varnothing\) (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline ZD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline Allowance \(U\) & \begin{tabular}{l}
Allowance for pre-finishing - (only for \(\nabla \nabla \nabla\) ) \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in X and Z direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is kept \\
- Negative value: Compensation allowance is removed in addition to finishing allowance
\end{tabular} & mm \\
\hline Relief cuts & Machine relief cuts (cannot be changed) & \\
\hline FR & Insertion feedrate, relief cuts & \\
\hline
\end{tabular}

\subsection*{9.3 Contour turning}

\section*{Hidden parameters}
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline Residual material & With subsequent residual material removal & No & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & x \\
\hline Selection & \begin{tabular}{l} 
Always round on the contour \\
Uniform cut segmentation \\
Constant cutting depth
\end{tabular} & \begin{tabular}{l} 
法
\end{tabular} & \\
\hline DI & Continuous cut - (only for \(\nabla\) ) & 0 & \\
\hline \begin{tabular}{l} 
Set machining area \\
limits
\end{tabular} & Set machining area limits & No & \\
\hline Relief cuts & Machine relief cuts (grayed out) & Yes & \\
\hline
\end{tabular}

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.3.9 Stock removal rest (CYCLE952)}

\section*{Function}

Using the "Stock removal residual" function, you remove material that has remained for stock removal along the contour.
During stock removal along the contour, the cycle automatically detects any residual material and generates an updated blank contour. for ShopTurn, the updated unmachined-part contour is automatically generated. For a C code program, for stock removal residual material, "Yes" must be programmed. Material that remains as part of the finishing allowance is not residual material. Using the "Stock removal residual material" function, you can remove unwanted material with a suitable tool.

\section*{Software option}

For stock removal of residual material, you require the option "residual material detection and machining".

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Stock removal residual material" softkey.

The "Stock removal residual material" input window opens.
\begin{tabular}{|l|l|l|l|l|l|l|l|}
\hline \multicolumn{3}{|l|}{ Parameters, G code program } & & \multicolumn{2}{|l|}{ Parameters, ShopTurn program } \\
\hline PRG & Name of the program to be generated & & T & Tool name
\end{tabular}
\begin{tabular}{|c|c|c|c|}
\hline Parameters & \multicolumn{2}{|l|}{Description} & Unit \\
\hline Machining U & \multicolumn{2}{|l|}{\begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing)
\end{tabular}} & \\
\hline Machining direction 0 & \begin{tabular}{l}
- Face U \\
- Longitudinal \(\mathbf{U}\) \\
- Parallel to the contour U
\end{tabular} & \begin{tabular}{l}
- From inside to outside \\
- From outside to inside \\
- From end face to rear side \\
- From rear side to end face
\end{tabular} & \\
\hline & \multicolumn{2}{|l|}{The machining direction depends on the stock removal direction and choice of tool.} & \\
\hline
\end{tabular}

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l}
Position \\
U
\end{tabular} & \begin{tabular}{l}
- front \\
- back \\
- internal \\
- external
\end{tabular} & \\
\hline D & Maximum depth infeed - (only for \(\nabla\) ) & mm \\
\hline XDA & 1. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline XDB & 2. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline DX & Maximum depth infeed - (only for parallel to the contour, as an alternative to D) & mm \\
\hline U & Do not round contour at end of cut. Always round contour at end of cut. & \\
\hline 0 & \begin{tabular}{l}
Uniform cut segmentation \\
Round cut segmentation at the edge
\end{tabular} & \\
\hline 0 & \begin{tabular}{l}
only for align cut segmentation at the edge: \\
Constant cutting depth \\
alternating cutting depth
\end{tabular} & \\
\hline Allowance \(\mathbf{U}\) & \begin{tabular}{l}
Allowance for pre-finishing - (only for \(\nabla \nabla \nabla\) ) \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & s \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in X and Z direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is kept \\
- Negative value: Compensation allowance is removed in addition to finishing allowance
\end{tabular} & mm \\
\hline Set machining area limits \(U\) & \begin{tabular}{l}
Set machining area limits \\
- Yes \\
- No
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { XA } \\
& \text { XB } O \\
& \text { ZA } \\
& \text { ZB } O \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
with limited machining area only, yes: \\
1. Limit \(X A \varnothing\) \\
2. Limit \(X B \varnothing\) (abs) or 2nd limit referred to \(X A\) (inc) \\
1. Limit ZA \\
2. Limit ZB (abs) or 2nd limit referred to ZA (inc)
\end{tabular} & mm \\
\hline Relief cuts \(\mathbf{U}\) & \begin{tabular}{l}
Machine relief cuts \\
- Yes \\
- No
\end{tabular} & \\
\hline FR & Insertion feedrate, relief cuts & \\
\hline
\end{tabular}

\subsection*{9.3.1 Plunge-cutting (CYCLE952)}

\section*{Function}

The "Grooving" function is used to machine grooves of any shape.
Before you program the groove, you must define the groove contour.
If a groove is wider than the active tool, it is machined in several cuts. The tool is moved by a maximum of \(80 \%\) of the tool width for each groove.

\section*{Blank}

When grooving, the cycle takes into account a blank that can consist of a cylinder, an allowance on the finished-part contour or any other blank contour.

\section*{Precondition}

For a G code program, at least one CYCLE62 is required before CYCLE952.
If CYCLE62 is only present once, then this involves the finished part contour.
If CYCLE62 is present twice, then the first call is the blank contour and the second call is the finished-part contour (also see Chapter "Programming (Page 390)").

\section*{Note}

\section*{Execution from external media}

If you execute programs from an external drive (e.g. local drive or network drive), then you require the execution from external storage function (EES).
For additional information, please refer to the following documentation:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Set machining area limits}

If, for example, you want to machine a certain area of the contour with a different tool, you can set machining area limits so that machining only takes place in the area of the contour you have selected.

\section*{Feedrate interruption}

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption.

\subsection*{9.3 Contour turning}

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


Machine manufacturer
Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Machining type}

You can freely select the machining type (roughing, finishing or complete machining).
For more detailed information, please refer to section "Stock removal".

\section*{Procedure}

Grooving
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Grooving" softkey.

The "Grooving" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input -} & \multicolumn{4}{|l|}{- Complete} \\
\hline PRG & \multicolumn{2}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline PL & \multicolumn{2}{|l|}{Machining plane} & D & Cutting edge number & \\
\hline RP & Retraction plane - (only for machining direction, longitudinal, inner) & mm & F & Feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline Residual material 0 & \begin{tabular}{l}
With subsequent residual m \\
- Yes \\
- No
\end{tabular} & erial rem & & & \\
\hline CONR & Name to save the updated contour for residual materia "Yes" for residual material & machine emoval moval) & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining U & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (complete machining)
\end{tabular} & \\
\hline Machining direction U & \begin{tabular}{l}
- Face \\
- Longitudinal
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0
\end{tabular} & \begin{tabular}{l}
- front \\
- back \\
- Inside \\
- outside
\end{tabular} & \\
\hline D & Maximum depth infeed - (only for \(\nabla\) ) & mm \\
\hline XDA & 1. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline XDB & 2. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and Z - (only for \(\nabla\) ) & mm \\
\hline UZ & Finishing allowance in Z - (only for UX) & mm \\
\hline DI & For zero: Continuous cut - (only for \(\nabla\) ) & mm \\
\hline BL U & \begin{tabular}{l}
Blank description (only for \(\nabla\) ) \\
- Cylinder (described using XD, ZD) \\
- Allowance (XD and ZD on the finished part contour) \\
- Contour (additional CYCLE62 call with blank contour - e.g. cast iron mold)
\end{tabular} & \\
\hline XD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension \(\varnothing\) (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline ZD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline
\end{tabular}

Turning

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|c|}
\hline \multicolumn{2}{|l|}{G code program parameters} & \multicolumn{2}{|l|}{ShopTurn program parameters} \\
\hline Allowance \(U\) & \begin{tabular}{l}
Allowance for pre-finishing - (only for \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & & mm \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in X and Z d \\
- Positive value: Compensation allow \\
- Negative value: Compensation allo allowance
\end{tabular} & \begin{tabular}{l}
ction (inc) - (only for allowance) \\
nce is kept \\
ance is removed in addition to finishing
\end{tabular} & mm \\
\hline Set machining area limits \(U\) & \begin{tabular}{l}
Set machining area limits \\
- Yes \\
- No
\end{tabular} & & \\
\hline \[
\begin{aligned}
& \text { XA } \\
& \text { XB } O \\
& Z A \\
& Z B \quad O
\end{aligned}
\] & \begin{tabular}{l}
with limited machining area only, yes: \\
1. Limit XA \(\varnothing\) \\
2. Limit \(X B \varnothing\) (abs) or 2nd limit referre \\
1. Limit ZA \\
2. Limit ZB (abs) or 2nd limit referred
\end{tabular} & \begin{tabular}{l}
to XA (inc) \\
ZA (inc)
\end{tabular} & mm \\
\hline N & Number of grooves & & \\
\hline DP & Distance between grooves (inc) & & mm \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline Input & \multicolumn{5}{|c|}{- simple} \\
\hline PRG & \multicolumn{2}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline PL & Machining plane & & D & Cutting edge number & \\
\hline RP & Retraction plane - (only for machining direction, longitudinal, inner) & mm & F & Feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline F & Feedrate & * & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla+\nabla \nabla \nabla\) (complete machining)
\end{tabular} & \\
\hline Machining direction U & \begin{tabular}{l}
- Face \\
- Longitudinal
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0
\end{tabular} & \begin{tabular}{l}
- front \\
- back \\
- inside \\
- outside
\end{tabular} & \\
\hline D & Maximum depth infeed - (only for \(\nabla\) ) & mm \\
\hline XDA & 1. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline XDB & 2. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and Z - (only for \(\nabla\) ) & mm \\
\hline UZ & Finishing allowance in Z - (only for UX) & mm \\
\hline BL U & \begin{tabular}{l}
Blank description (only for \(\nabla\) ) \\
- Cylinder (described using XD, ZD) \\
- Allowance (XD and ZD on the finished part contour) \\
- Contour (additional CYCLE62 call with blank contour - e.g. cast iron mold)
\end{tabular} & \\
\hline XD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension \(\varnothing\) (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline ZD & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline
\end{tabular}

Turning

\subsection*{9.3 Contour turning}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline Allowance \(\mathbf{U}\) & \begin{tabular}{l} 
Allowance for pre-finishing - (only for \(\nabla \nabla \nabla\) ) \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & \begin{tabular}{l} 
Compensation allowance in X and Z direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is kept \\
- Negative value: Compensation allowance is removed in addition to finishing \\
allowance
\end{tabular}
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Hidden parameters}
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline Residual material & With subsequent residual material removal & No & \\
\hline SC & Safety clearance & & x \\
\hline DI & Continuous cut - (only for \(\nabla\) ) & 0 & \\
\hline \begin{tabular}{l} 
Set machining area \\
limits
\end{tabular} & Set machining area limits & No & \\
\hline N & Number of grooves & 1 & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.3.11 Plunge-cutting rest (CYCLE952)}

\section*{Function}

The "Grooving residual material" function is used when you want to machine the material that remained after grooving along the contour.

During grooving ShopTurn, the cycle automatically detects any residual material and generates an updated blank contour. For a G code program, the function must have been previously selected. Material that remains as part of the finishing allowance is not residual material. The "Grooving residual material" function allows you to remove unwanted material with a suitable tool.


\section*{Software option}

To machine residual material, you require the "Residual material detection and machining" option.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Groove resid.
2. Press the "Contour turning" softkey.
3. Press the "Grooving residual material" softkey.

The "Grooving residual material" input window is opened.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline PRG & Name of the program to be gen & enerated & T & Tool name & \\
\hline PL & Machining plane & & D & Cutting edge number & \\
\hline RP & Retraction plane - (only for longitudinal machining direction) & mm & F & Feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline CON & \multicolumn{2}{|l|}{Name of the updated blank contour for residual material machining (without the attached character "_C" and double-digit number)} & & & \\
\hline Residual material & \multicolumn{2}{|l|}{\begin{tabular}{l}
With subsequent residual material removal \\
- Yes \\
- No
\end{tabular}} & & & \\
\hline CONR & \multicolumn{2}{|l|}{Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal)} & & & \\
\hline
\end{tabular}

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\quad \nabla \nabla \nabla\) (finishing)
\end{tabular} & \\
\hline Machining direction U & \begin{tabular}{l}
- Face \\
- Longitudinal
\end{tabular} & \\
\hline Position U & \begin{tabular}{l}
- front \\
- back \\
- internal \\
- external
\end{tabular} & \\
\hline D & Maximum depth infeed - (only for \(\nabla\) ) & mm \\
\hline XDA & 1. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline XDB & 2. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and \(Z\) - (only for \(\nabla\) ) & mm \\
\hline UZ & Finishing allowance in \(Z\) - (only for UX) & mm \\
\hline DI & For zero: Continuous cut - (only for \(\nabla\) ) & mm \\
\hline Allowance \(\cup\) & \begin{tabular}{l}
Allowance for pre-finishing - (only for \(\nabla \nabla \nabla\) ) \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & mm \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in \(X\) and \(Z\) direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is kept \\
- Negative value: Compensation allowance is removed in addition to finishing allowance
\end{tabular} & mm \\
\hline Set machining area limits \(U\) & \begin{tabular}{l}
Set machining area limits \\
- Yes \\
- No
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { XA } \\
& \text { XB } \quad U \\
& Z A \\
& Z B \quad U \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
with limited machining area only, yes: \\
1. Limit XA \(\varnothing\) \\
2. Limit \(\mathrm{XB} \varnothing\) (abs) or 2nd limit referred to \(X A\) (inc) \\
1. Limit ZA \\
2. Limit ZB (abs) or 2nd limit referred to ZA (inc)
\end{tabular} & mm \\
\hline N & Number of grooves & \\
\hline DP & Distance between grooves (inc) & mm \\
\hline
\end{tabular}

\subsection*{9.3.12 Plunge-turning (CYCLE952)}

\section*{Function}

Using the "Plunge turning" function, you can machine any shape of groove.
Contrary to grooving, the plunge turning function removes material on the sides after the groove has been machined in order to reduce machining time. Contrary to stock removal, the plunge turning function allows you to machine contours that the tool must enter vertically.
You will need a special tool for plunge turning. Before you program the "Plunge turning" cycle, you must define the contour.

\section*{Blank}

For plunge turning, the cycle takes into account a blank that can consist of a cylinder, an allowance on the finished-part contour or any other blank contour.

\section*{Precondition}

For a G code program, at least one CYCLE62 is required before CYCLE952.
If CYCLE62 is only present once, then this involves the finished part contour.
If CYCLE62 is present twice, then the first call is the unmachined part contour and the second call is the finished-part contour (also see Chapter "Programming (Page 390)").

\section*{Note}

\section*{Execution from external media}

If you execute programs from an external drive (e.g. local drive or network drive) then you require the execution from external storage function (EES).

For additional information, please refer to the following references:
SINUMERIK Operate (IM9) / SINUMERIK 840D sI Commissioning Manual

\section*{Set machining area limits}

If, for example, you want to machine a certain area of the contour with a different tool, you can set machining area limits so that machining only takes place in the area of the contour you have selected.

\section*{Feedrate interruption}

To prevent the occurrence of excessively long chips during machining, you can program a feedrate interruption.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


Machine manufacturer
Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Machining type}

You can freely select the machining type (roughing, finishing or complete machining).
For more detailed information, please refer to section "Stock removal".

\section*{Procedure}

Part
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.
3. Press the "Plunge turning" softkey.

The "Plunge turning" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input -} & \multicolumn{4}{|l|}{- Complete} \\
\hline PRG & \multicolumn{2}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline PL & \multicolumn{2}{|l|}{Machining plane} & D & Cutting edge number & \\
\hline RP & Retraction plane - (only for machining direction, longitudinal, inner) & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline SC & Safety clearance & mm & & & \\
\hline Residual material 0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
With subsequent residual material removal \\
- Yes \\
- No
\end{tabular}} & & & \\
\hline CONR & \multicolumn{2}{|l|}{Name to save the updated unmachined-part contour for residual material removal - (only "Yes" for residual material removal)} & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline FX (only ShopTurn) & Feedrate in X direction & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline FZ (only ShopTurn) & Feedrate in Z direction & mm/rev \\
\hline FX (only G Code) & Feedrate in X direction & * \\
\hline FZ (only for G code) & Feedrate in Z direction & * \\
\hline Machining U & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (complete machining)
\end{tabular} & \\
\hline Machining direction U & \begin{tabular}{l}
- Face \\
- Longitudinal
\end{tabular} & \\
\hline Position U & \begin{tabular}{l}
- front \\
- back \\
- inside \\
- outside
\end{tabular} & \\
\hline D & Maximum depth infeed - (only for \(\nabla\) ) & mm \\
\hline XDA & 1. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline XDB & 2. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and \(Z\) - (only for \(\nabla\) ) & mm \\
\hline UZ & Finishing allowance in Z - (only for \(\nabla\) ) & mm \\
\hline DI & For zero: Continuous cut - (only for \(\nabla\) ) & mm \\
\hline BL \(\mathcal{U}\) & \begin{tabular}{l}
Blank description (only for \(\nabla\) ) \\
- Cylinder (described using XD, ZD) \\
- Allowance (XD and ZD on the finished part contour) \\
- Contour (additional CYCLE62 call with blank contour - e.g. cast iron mold)
\end{tabular} & \\
\hline XD \(\cup\) & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension \(\varnothing\) (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline
\end{tabular}

Turning

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline ZD U & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline Allowance U & \begin{tabular}{l}
Allowance for pre-finishing - (only for \(\nabla \nabla \nabla\) ) \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & mm \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in X and Z direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is kept \\
- Negative value: Compensation allowance is removed in addition to finishing allowance
\end{tabular} & mm \\
\hline Set machining area limits \(U\) & \begin{tabular}{l}
Set machining area limits \\
- Yes \\
- No
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { XA } \\
& \text { XB } O \\
& \text { ZA } \\
& \text { ZB } O
\end{aligned}
\] & \begin{tabular}{l}
with limited machining area only, yes: \\
1. Limit \(\mathrm{XA} \varnothing\) \\
2. Limit \(X B \varnothing\) (abs) or 2 nd limit referred to \(X A\) (inc) \\
1. Limit \(Z A\) \\
2. Limit ZB (abs) or 2nd limit referred to ZA (inc)
\end{tabular} & mm \\
\hline N & Number of grooves & \\
\hline DP & Distance between grooves & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline Input & \multicolumn{5}{|c|}{- simple} \\
\hline PRG & \multicolumn{2}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline PL & Machining plane & & D & Cutting edge number & \\
\hline RP & Retraction plane - (only for machining direction, longitudinal, inner) & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & \(\mathrm{m} / \mathrm{min}\) \\
\hline & & & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline FX (only ShopTurn) & - Feedrate in X direction & mm/rev \\
\hline FZ (only ShopTurn) & - Feedrate in \(Z\) direction & mm/rev \\
\hline FX (only G Code) & - Feedrate in X direction & * \\
\hline FZ (only for G code) & - Feedrate in Z direction & * \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla+\nabla \nabla \nabla\) (complete machining)
\end{tabular} & \\
\hline Machining direction u & \begin{tabular}{l}
- face U \\
- longitudinal U
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0
\end{tabular} & \begin{tabular}{l}
- front \\
- back \\
- inside \\
- outside
\end{tabular} & \\
\hline D & Maximum depth infeed - (only for \(\overline{\text { ) }}\) & mm \\
\hline XDA & 1. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline XDB & 2. Grooving limit tool (abs) - (only for face machining direction) & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and Z - (only for \(\overline{\mathrm{V}}\) ) & mm \\
\hline UZ & Finishing allowance in Z - (only for UX) & mm \\
\hline BL U & \begin{tabular}{l}
Blank description (only for \(\nabla\) ) \\
- Cylinder (described using XD, ZD) \\
- Allowance (XD and ZD on the finished part contour) \\
- Contour (additional CYCLE62 call with blank contour - e.g. cast iron mold)
\end{tabular} & \\
\hline XD U & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension \(\varnothing\) (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline
\end{tabular}

Turning

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline ZD \(\cup\) & \begin{tabular}{l}
- (only for \(\nabla\) machining) \\
- (only for blank description, cylinder and allowance) \\
- For blank description, cylinder \\
- Version, absolute: \\
Cylinder dimension (abs) \\
- Version incremental: \\
Allowance (inc) to maximum values of the CYCLE62 finished part contour \\
- For blank description, allowance \\
- Allowance on the CYCLE62 finished part contour (inc)
\end{tabular} & mm \\
\hline Allowance \(U\) & \begin{tabular}{l}
Allowance for pre-finishing - (only for \(\nabla \nabla \nabla\) ) \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in X and Z direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is kept \\
- Negative value: Compensation allowance is removed in addition to finishing allowance
\end{tabular} & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Hidden parameters}
\begin{tabular}{|l|l|l|l|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline Residual material & With subsequent residual material removal & No & \\
\hline SC & Safety clearance & & \\
\hline DI & Continuous cut - (only for \(\nabla\) ) & 0 & \\
\hline \begin{tabular}{l} 
Set machining area \\
limits
\end{tabular} & Set machining area limits & No & \\
\hline N & Number of grooves & 1 & \\
\hline
\end{tabular}

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.3.13 Plunge-turning rest (CYCLE952)}

\section*{Function}

The "Plunge turning residual material" function is used when you want to machine the material that remained after plunge turning.

For plunge turning ShopTurn, the cycle automatically detects any residual material and generates an updated blank contour. For a G code program, the function must have been previously selected in the screen. Material that remains as part of the finishing allowance is not residual material. The "Plunge turning residual material" function allows you to remove unwanted material with a suitable tool.


\section*{Software option}

To machine residual material, you require the "Residual material detection and machining" option.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.


Part resid.
2. Press the "Contour turning" softkey.
3. Press the "Plunge turning residual material" softkey.

The "Plunge turning residual material" input window opens.
\begin{tabular}{|l|l|l|l|l|l|l|}
\hline \multicolumn{2}{|l|}{ Parameters, G code program } & & \multicolumn{2}{|l|}{ Parameters, ShopTurn program } \\
\hline PRG & Name of the program to be generated & & T & Tool name
\end{tabular}\(|\)\begin{tabular}{ll} 
\\
\hline PL & Machining plane
\end{tabular}

\subsection*{9.3 Contour turning}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline FX (only ShopTurn) & Feedrate in X direction & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline FZ (only ShopTurn) & Feedrate in Z direction & mm/rev \\
\hline FX (only G Code) & Feedrate in X direction & * \\
\hline FZ (only for G code) & Feedrate in Z direction & * \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing)
\end{tabular} & \\
\hline Machining direction
\[
0
\] & \begin{tabular}{l}
- Face \\
- Longitudinal
\end{tabular} & \\
\hline Position 0 & \begin{tabular}{l}
- front \\
- back \\
- internal \\
- external
\end{tabular} & \\
\hline D & Maximum depth infeed - (only for \(\nabla\) ) & mm \\
\hline UX or U U & Finishing allowance in X or finishing allowance in X and Z - (only for \(\nabla\) ) & mm \\
\hline UZ & Finishing allowance in \(Z\) - (only for \(\nabla\) ) & mm \\
\hline XDA & 1. Grooving limit tool \(\varnothing\) (abs) - (end face or rear face only) & mm \\
\hline XDB & 2. Grooving limit tool \(\varnothing\) (abs) - (end face or rear face only) & mm \\
\hline Allowance \(U\) & \begin{tabular}{l}
Allowance for prefinishing \\
- Yes \\
U1 contour allowance \\
- No
\end{tabular} & \\
\hline DI & For zero: Continuous cut - (only for \(\nabla\) ) & mm \\
\hline U1 & \begin{tabular}{l}
Compensation allowance in \(X\) and \(Z\) direction (inc) - (only for allowance) \\
- Positive value: Compensation allowance is kept \\
- Negative value: Compensation allowance is removed in addition to finishing allowance
\end{tabular} & mm \\
\hline Set machining area limits & \begin{tabular}{l}
Set machining area limits \\
- Yes \\
- No
\end{tabular} & \\
\hline \[
\begin{array}{|l}
\text { XA } \\
\text { XB } O \\
Z A \\
Z B \quad O \\
\hline
\end{array}
\] & \begin{tabular}{l}
with limited machining area only, yes: \\
1. Limit XA \(\varnothing\) \\
2. Limit \(X B \varnothing\) (abs) or 2nd limit referred to \(X A\) (inc) \\
1. Limit ZA \\
2. Limit ZB (abs) or 2nd limit referred to ZA (inc)
\end{tabular} & mm \\
\hline N & Number of grooves & \\
\hline DP & Distance between grooves (inc) & mm \\
\hline
\end{tabular}

\section*{\(9.4 \quad\) Milling}

\subsection*{9.4.1 Face milling (CYCLE61)}

\section*{Function}

You can face mill any workpiece with the "Face milling" cycle.
A rectangular surface is always machined. The rectangle is obtained from corner points 1 and 2 - which for a ShopTurn program - are pre-assigned with the values of the blank part dimensions from the program header.
Workpieces with and without limits can be face-milled.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Approach/retraction}
1. For vertical machining, the starting point is always at the top or bottom. For horizontal machining, it is at the left or right.
The starting point is marked in the help display.
2. Machining is performed from the outside to the inside.

\section*{Machining type}

The cycle makes a distinction between roughing and finishing:
- Roughing:

Milling the surface
Tool turns above the workpiece edge
- Finishing:

Milling the surface once
Tool turns at safety distance in the \(\mathrm{X} / \mathrm{Y}\) plane
Retraction of milling cutter

Turning

Depth infeed always takes place outside the workpiece.
For a workpiece with edge breaking, select the rectangular spigot cycle.
In face milling, the effective tool diameter for a tool of type "Milling cutter" is stored in a machine data item.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Selecting the machining direction}

Toggle the machining direction in the "Direction" field until the symbol for the required machining direction appears.
- Same direction of machining
- Alternating direction of machining

\section*{Selecting limits}

Press the respective softkey for the required limit.


The selected limits are shown in the help screen and in the broken-line graphics.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.

Face
3. Press the "Face milling" softkey.
milling
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters，G code program} & \multicolumn{3}{|l|}{Parameters，ShopTurn program} \\
\hline \[
\begin{gathered}
\mathrm{PL} \\
\mathrm{U}
\end{gathered}
\] & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{U}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) mm／tooth \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cut－ ting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & ＊ & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
0 \\
（only for Shop－ \\
Turn）
\end{tabular} & \begin{tabular}{l}
－Face Y \\
－Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
（0） \\
（only for Shop－ Turn）
\end{tabular} & \begin{tabular}{l}
Clamp／release spindle \\
The function must be set up by the machine manufacturer．
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
The following machining technologies can be selected： \\
－\(\nabla\)（roughing） \\
－\(\quad \nabla \nabla \nabla\)（finishing）
\end{tabular} & \\
\hline Direction 0 & \begin{tabular}{l}
Same direction of machining \\
- 臣 \\
- 世 \\
Alternating direction of machining \\
- 写 \\
- 朴
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { (only for G code) } \\
& \text { X0 } \\
& \text { Y0 } \\
& \text { Z0 } \\
& \text { X1 U } \\
& \text { Y1 U } \\
& \text { Z1 U }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point： \\
Corner point 1 in \(X\) \\
Corner point 1 in Y \\
Height of blank \\
Corner point 2 X （abs）or corner point 2 X in relation to \(\mathrm{X0}\)（inc） \\
Corner point 2 Y （abs）or corner point 2 Y in relation to Y 0 （inc） \\
Height of blank（abs）or height of blank in relation to Z0（inc）
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline (only ShopTurn)
CP
X0
Y0
Z0
X1 U
Y1 U
Z1 U & \begin{tabular}{l}
Face Y: The positions refer to the reference point: \\
Positioning angle for machining area - only for face Y \\
Corner point 1 in \(X\) \\
Corner point 1 in Y \\
Height of blank \\
Corner point 2 in X (abs) or corner point 2 X in relation to XO (inc) \\
Corner point 2 in Y (abs) or corner point 2 Y in relation to Y 0 (inc) \\
Height of blank (abs) or height of blank in relation to \(Z 0\) (inc)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline (only ShopTurn)
C0
Y0
Z0
X0
Y1 \(U\)
Z1 \(U\)
X1 \(U\) & \begin{tabular}{l}
Peripheral surface \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining area - (only for peripheral surface Y ) \\
Corner point 1 in \(Y\) \\
Corner point 1 in Z \\
Height of blank \\
Corner point 2 in Y (abs) or corner point 2 X in relation to Y 0 (inc) \\
Corner point 2 in \(Z\) (abs) or corner point 2 Y in relation to ZO (inc) \\
Height of blank (abs) or height of blank in relation to X0 (inc)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline DXY U & \begin{tabular}{l}
Maximum plane infeed \\
Alternately, you can specify the plane infeed in \%, as a ratio \(\rightarrow\) plane infeed ( mm ) to milling cutter diameter (mm).
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed - (for roughing only) & mm \\
\hline UZ & Finishing allowance, depth & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Note}

The same finishing allowance must be entered for both roughing and finishing. The finishing allowance is used to position the tool for retraction.

\subsection*{9.4.2 Rectangular pocket (POCKET3)}

\section*{Function}

You can use the "Mill rectangular pocket" cycle to mill any rectangular pockets on the face or peripheral surface.
The following machining variants are available:
- Mill rectangular pocket from solid material.
- Predrill rectangular pocket in the center first if, for example, the milling cutter does not cut in the center (e.g. for ShopTurn, program the drilling, rectangular pocket and position program blocks in succession).
- Machine pre-machined rectangular pocket (see "Solid machining" parameter).
- Complete machining
- Post machining

Depending on the dimensions of the rectangular pocket in the workpiece drawing, you can select a corresponding reference point for the rectangular pocket.

\section*{Note}

\section*{Predrilling}

If the programmed input parameters, deviating from Pocket3, result in a longitudinal slot or a longitudinal hole, then in the cycle, from Pocket3, the corresponding cycle to machine slots (Slot1 or Longhole) is called. In these cases, the insertion points can deviate from the pocket center.
Note this peculiarity when you predrill.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction}
1. The tool approaches the center point of the rectangular pocket in rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool is inserted into the material according to the chosen strategy.
3. The rectangular pocket is always machined with the chosen machining type from inside out.
4. The tool moves back to the safety clearance at rapid traverse.

\section*{Machining type}
- Roughing

Roughing involves machining the individual planes of the pocket one after the other from the center out, until depth Z 1 or X 1 is reached.
- Finishing

During finishing, the edge is always machined first. The pocket edge is approached on the quadrant that joins the corner radius. During the last infeed, the base is finished from the center out.
- Edge finishing

Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfering

Chamfering involves edge breaking at the upper edge of the rectangular pocket.


Figure 9-4 Geometries when chamfering inside contours

\section*{Note}

The following error messages can occur when chamfering inside contours:
- Safety clearance in the program header too large

This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- Immersion depth too large

This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- Tool diameter too large

This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

\section*{Procedure}

Pocket
Rectang.
pocket
pocket
1. The part program or ShopTurn program to be processed has been cre-
ated and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Pocket" and "Rectangular pocket" softkeys.

The "Rectangular Pocket" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{InputU} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{U}
\end{aligned}
\] & Machining plane & & T & Tool name & \\
\hline 0 & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & \begin{tabular}{l} 
F \\
\hline
\end{tabular} & Feedrate & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Reference point (only for G code)
\[
U
\] & \begin{tabular}{l}
The following different reference point positions can be selected: \\
- \(\quad\) ! (center) \\
- \(\sqrt{6}\) (bottom left) \\
- \(\quad\) (bottom right) \\
- \(\quad\) : \(\quad\) (top left) \\
- : \\
The reference point (highlighted in blue) is displayed in the Help screen.
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining surface \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
aU \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining
\[
0
\] & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline Machining position U & \begin{tabular}{l}
- Single position \\
Mill rectangular pocket at the programmed position (XO, YO, ZO). \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline \[
\begin{array}{|l|}
\hline \text { X0 } \\
\text { Y0 } \\
\text { Z0 } \\
\text { (only for G code) } \\
\hline
\end{array}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) - (only for single position) \\
Reference point \(Y\) - (only for single position) \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline ```
X0 or LO U
YO or CO U
Z0
(only for Shop-
Turn)
``` & \begin{tabular}{l}
Face C: The positions refer to the reference point: \\
Reference point \(X\) or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline ```
CP
X0 or LO U
Y0 or CO U
Z0
(only for Shop-
Turn)
``` & \begin{tabular}{l}
Face Y: The positions refer to the reference point: \\
Positioning angle for machining area - (only single position) \\
Reference point \(X\) or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Y0 or CO U } \\
& \text { Z0 } \\
& \text { X0 } \\
& \text { (only for Shop- } \\
& \text { Turn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C: The positions refer to the reference point: \\
Reference point \(Y\) or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position) \\
Cylinder diameter \(\varnothing\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline \begin{tabular}{l} 
C0 \\
Y0 \\
Z0 \\
X0 \\
(only for Shop- \\
Turn) \\
\hline W
\end{tabular} & \begin{tabular}{l}
Peripheral surface Y: The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point \(Y\) - (only for single position) \\
Reference point Z - (only for single position) \\
Reference point X - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline W & Pocket width & mm \\
\hline L & Pocket length & mm \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline R & Corner radius & mm \\
\hline a0 & Angle of rotation & Degrees \\
\hline \[
\begin{aligned}
& \mathrm{Z} 1 \\
& 0
\end{aligned}
\] & Pocket depth (abs) or depth relative to ZO (inc) - (only for \(\nabla\), \(\nabla \nabla \nabla\) or \(\nabla \nabla \nabla \mathrm{edge}\) ) & mm \\
\hline \[
\begin{aligned}
& \mathrm{DXY} \\
& \mathrm{U}
\end{aligned}
\] & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter - (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ &  & mm \\
\hline UXY & Plane finishing allowance - (only for \(\bar{\nabla}, \mathrm{V} \nabla \overline{\text { or }} \bar{\nabla} \nabla \mathrm{edge}\) ) & mm \\
\hline UZ & Depth finishing allowance - (only for \(\nabla\), \(\mathrm{VV} \mathrm{\nabla}\) ) & mm \\
\hline Insertion
\[
0
\] & \begin{tabular}{l}
The following insertion modes can be selected - (only for \(\nabla, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge): \\
- Predrilled: (only for G code) \\
With GO, the pocket center point is approached at the retraction plane level, and then, from this position, also with GO, the axis travels to the reference point brought forward by the safety clearance. The machining of the rectangular pocket is then performed according to the selected insertion strategy, taking into account the programmed blank dimensions. \\
- Vertical: Insert vertically at center of pocket \\
The tool executes the calculated actual depth infeed at the pocket center in a single block. This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. \\
- Helical: Insert along helical path \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. \\
- Oscillating: Insert with oscillation along center axis of rectangular pocket(only for G code) \\
The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.
\end{tabular} & \\
\hline \begin{tabular}{l}
a \\
( 0 \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face C/peripheral surface C, if inserted vertically) The function must be set up by the machine manufacturer & \\
\hline FZ (only for G code) & Depth infeed rate - (for vertical insertion only) & * \\
\hline \begin{tabular}{l}
FZ \\
U \\
(only for Shop- \\
Turn)
\end{tabular} & Depth infeed rate - (for vertical insertion only) & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline EP & Maximum pitch of helix - (for helical insertion only) & mm/rev \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline ER & \begin{tabular}{l} 
Radius of helix - (for helical insertion only) \\
The radius cannot be any larger than the milling cutter radius; otherwise, material will \\
remain.
\end{tabular} & mm \\
\hline EW & \begin{tabular}{l} 
Maximum insertion angle - (for insertion with oscillation only)
\end{tabular} \\
\hline \begin{tabular}{l} 
Solid machining \\
(for roughing only) \\
U
\end{tabular} & \begin{tabular}{l} 
Complete machining \\
The rectangular pocket is milled from the solid material. \\
Post machining \\
The size of any existing smaller rectangular pocket or hole is increased in one or \\
more axes. You must program parameters AZ, W1 and L1 for this purpose.
\end{tabular} & Degrees \\
\hline AZ & Depth of premachining - (for post machining only) & mm \\
\hline W1 & Width of premachining - (for post machining only) & mm \\
\hline L1 & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline FS & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline ZFS & * Unit of feedrate as programmed before the cycle call & mm \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parmeters} \\
\hline \multicolumn{2}{|l|}{Input U} & \multicolumn{4}{|l|}{- simple} \\
\hline U & Milling direction & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & F U & Feedrate & mm/min \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& 0
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline MachiningU & The following machining operations can be selected: & \\
& \(\bullet \quad \nabla\) (roughing) & \\
& \(\bullet\) & \(\nabla \nabla \nabla\) (finishing) \\
& - & \(\nabla \nabla \nabla\) edge (edge finishing) \\
Chamfering & & \\
\hline Machining & - Face C & \\
surface (only for & - Face Y \\
ShopTurn) & - Peripheral surface C & \\
\(\mathbf{U}\) & Peripheral surface Y & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l}
Position (only for ShopTurn) \\
0
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline (0) (only for ShopTurn) O & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Z0 } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) \\
Reference point \(Y\) \\
Reference point \(Z\)
\end{tabular} & \[
\begin{gathered}
\mathrm{mm} \\
\mathrm{~mm} \\
\mathrm{~mm}
\end{gathered}
\] \\
\hline \begin{tabular}{l}
X0 or LO U \\
Yo or COU \\
Zo (only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point Y or reference point angle polar \\
Reference point Z
\end{tabular} & mm mm or degrees mm \\
\hline \begin{tabular}{l}
CP \\
X0 or LO U \\
Y0 or C0 U \\
Z0 (only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining area \\
Reference point \(X\) or reference point length polar \\
Reference point \(Y\) or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Y0 or CO U } \\
& \text { zo } \\
& \text { X0 } \\
& \text { (only for ShopTurn) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point length polar \\
Reference point Z \\
Cylinder diameter \(\varnothing\)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { C0 } \\
& \text { Yo } \\
& \text { Z0 } \\
& \text { X0 } \\
& \text { (only for ShopTurn) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining surface \\
Reference point \(Y\) \\
Reference point \(Z\) \\
Reference point \(X\)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline W & Pocket width & mm \\
\hline L & Pocket length & mm \\
\hline R & Corner radius & mm \\
\hline Z1 U & Depth referred to Z0 (inc) or pocket depth (abs) - (only for \(\nabla\), \(\bar{\nabla} \nabla \mathrm{\nabla}\) or \(\nabla \nabla \nabla\) edge) & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline DXY U & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter - (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\), \(\nabla \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\nabla, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge) & mm \\
\hline UZ & Depth finishing allowance - (only for \(\nabla, \nabla \nabla \nabla\) ) & mm \\
\hline Insertion 0 & \begin{tabular}{l}
The following insertion modes can be selected - (only for \(\nabla, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge): \\
- Predrilled: (only for G code) \\
With G0, the pocket center point is approached at the retraction plane level, and then, from this position, also with G0, the axis travels to the reference point brought forward by the safety clearance. The machining of the rectangular pocket is then performed according to the selected insertion strategy, taking into account the programmed blank dimensions. \\
- Vertical: Insert vertically at center of pocket \\
The tool executes the calculated actual depth infeed at the pocket center in a single block. This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. \\
- Helical: Insert along helical path \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. \\
- Oscillating: Insert with oscillation along center axis of rectangular pocket \\
The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.
\end{tabular} & \\
\hline \begin{tabular}{l}
(0) \\
(only for ShopTurn) U
\end{tabular} & Clamp/release spindle (only for face C/face C, if inserted vertically) The function must be set up by the machine manufacturer & \\
\hline \begin{tabular}{l}
FZ \\
(only for G code)
\end{tabular} & Depth infeed rate - (for vertical insertion only) & * \\
\hline FZ (only for ShopTurn) & Depth infeed rate - (for vertical insertion only) & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline EP & Maximum pitch of helix - (for helical insertion only) & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline ER & \begin{tabular}{l}
Radius of helix - (for helical insertion only) \\
The radius cannot be any larger than the milling cutter radius; otherwise, material will remain.
\end{tabular} & mm \\
\hline EW & Maximum insertion angle - (for insertion with oscillation only) & Degrees \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|l|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline Reference point & Position of the reference point: Center & a & \\
\hline \begin{tabular}{l} 
Machining \\
position
\end{tabular} & \begin{tabular}{l} 
Mill rectangular pocket at the programmed position (X0, Y0, \\
Z0).
\end{tabular} & \begin{tabular}{l} 
Single posi- \\
tion
\end{tabular} & \\
\hline a0 & Angle of rotation & \(0^{\circ}\) & \\
\hline Solid machining & \begin{tabular}{l} 
The rectangular pocket is milled from the solid material - \\
(only for roughing)
\end{tabular} & \begin{tabular}{l} 
Complete \\
machining
\end{tabular} & \\
\hline
\end{tabular}

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.4.3 Circular pocket (POCKET4)}

\section*{Function}

You can use the "Circular pocket" cycle to mill circular pockets on the face or peripheral surface.
The following machining variants are available:
- Mill circular pocket from solid material.
- Predrill circular pocket in the center first if, for example, the milling cutter does not cut in the center (program the drilling, circular pocket and position program blocks in succession).

For milling with the "Circular pocket" function two methods are available: the plane-by-plane method and the helical method.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction during plane-by-plane machining}

In plane-by-plane machining of the circular pocket, the material is removed horizontally, one layer at a time.
1. The tool approaches the center point of the pocket at rapid traverse at the height of the retraction plane and adjusts to the safety distance.
2. The tool is inserted into the material according to the method selected.
3. The circular pocket is always machined from inside out using the selected machining method.
4. The tool moves back to the safety distance at rapid traverse.

\section*{Approach/retraction during helical machining}

In helical machining, the material is removed down to pocket depth in a helical movement.
1. The tool approaches the center point of the pocket at rapid traverse at the height of the retraction plane and adjusts to the safety distance.
2. Infeed to the first machining diameter.
3. The circular pocket is machined to pocket depth using the selected machining method.
4. The tool moves back to the safety distance at rapid traverse.

\section*{Machining type: Plane by plane}

When milling circular pockets, you can select these methods for the following machining types:
- Roughing

Roughing involves machining the individual planes of the circular pocket one after the other from the center out, until depth Z1 or X1 is reached.
- Finishing

During finishing, the edge is always machined first. The pocket edge is approached on the quadrant that joins the pocket radius. During the last infeed, the base is finished from the center out.
- Edge finishing

Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.

\section*{Machining type: Helical}

When milling circular pockets, you can select these methods for the following machining types:
- Roughing

During roughing, the circular pocket is machined downward with helical movements.
A full circle is made at pocket depth to remove the residual material.
The tool is removed from the edge and base in the quadrant and retracted with rapid traverse to the safety distance.
This process is repeated layer by layer, from inside out, until the circular pocket has been completely machined.
- Finishing

In finishing mode, the edge is machined first with a helical movement down to the bottom.
A full circle is made at pocket depth to remove the residual material.
The base is milled from outside in, using a spiral movement.
The tool is retracted with rapid traverse from the center of the pocket to a safety distance.
- Edge finishing

In edge finishing, the edge is machined first with a helical movement down to the bottom.
A full circle is made at pocket depth to remove the residual material.
The tool is removed from the edge and base in the quadrant and retracted with rapid traverse to the safety distance.

\section*{Chamfering machining}

Chamfering involves edge breaking at the upper edge of the circular pocket.


Figure 9-5 Geometries when chamfering inside contours

\section*{Note}

The following error messages can occur when chamfering inside contours:
- Safety clearance in the program header too large

This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- Immersion depth too large

This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- Tool diameter too large

This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

\subsection*{9.4 Milling}

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Mill.

Pocket

\section*{Circular} pocket
2. Press the "Milling" softkey.
3. Press the "Pocket" and "Circular pocket" softkeys. The "Circular Pocket" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{InputU} & \multicolumn{4}{|l|}{- Complete} \\
\hline PLU & Machining plane & & T & Tool name & \\
\hline 0 & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & F & Feedrate & mm/min \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
0 \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face \(Y\) \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
U \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
D \\
(0) \\
(only for Shop- \\
Turn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing, plane-by-plane or helical) \\
- \(\nabla \nabla \nabla\) (finishing, plane-by-plane or helical) \\
- \(\nabla \mathbb{V}\) edge (edge finishing, plane-by-plane or helical) \\
- Chamfering
\end{tabular} & \\
\hline Machining type U & \begin{tabular}{l}
- Plane by plane \\
Solid machine circular pocket plane-by-plane \\
- Helical \\
Solid machine circular pocket helically
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining position \\
0
\end{tabular} & \begin{tabular}{l}
- Single position \\
A circular pocket is machined at the programmed position ( \(\mathrm{XO}, \mathrm{YO}, \mathrm{ZO}\) ). \\
- Position pattern \\
Several circular pockets are machined in a position pattern (e.g. full circle, pitch circle, grid, etc.).
\end{tabular} & \\
\hline \begin{tabular}{l} 
X0 \\
Y0 \\
Z0 \\
(only for G code) \\
\hline
\end{tabular} & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point X - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point \(Z\)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{array}{|l}
\text { X0 or LO U } \\
\text { YO or CO U } \\
\text { Z0 } \\
\text { (only for Shop- } \\
\text { Turn) } \\
\hline
\end{array}
\] & \begin{tabular}{l}
Face C : The positions refer to the reference point: \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point \(Z\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
CP \\
XO or LO U \\
YO or CO U \\
Z0 \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining area - (only single position) \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Yo or CO O } \\
& \text { zo } \\
& \text { X0 } \\
& \text { (only for Shop- } \\
& \text { Turn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position) \\
Cylinder diameter \(\varnothing\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
C0 \\
Y0 \\
zo \\
X0 \\
(only for Shop- \\
Turn) \\
\hline\(\varnothing\)
\end{tabular} & \begin{tabular}{l}
Peripheral surface Y : The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z - (only for single position) \\
Reference point X - (only for single position)
\end{tabular} & \[
\begin{aligned}
& \text { Degrees } \\
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \(\varnothing\) & Diameter of pocket & mm \\
\hline & Pocket depth (abs) or depth relative to Z0/X0 (inc) - (only for \(\bar{\nabla}, \mathrm{\nabla} \nabla \mathrm{\nabla}\) and \(\nabla \nabla \nabla \mathrm{edge}\) ) & mm \\
\hline \[
\begin{aligned}
& \mathrm{DXY} \\
& \mathrm{O}
\end{aligned}
\] & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter - (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & \[
\begin{array}{|l|}
\hline \text { in } \\
\%
\end{array}
\] \\
\hline DZ & Maximum depth infeed - (only for \(\nabla, \nabla \nabla \nabla\) and \(\nabla \nabla \nabla\) edge) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\bar{\nabla}, \mathrm{\nabla} \nabla \mathrm{\nabla}\) and \(\nabla \nabla \nabla \mathrm{edge}\) ) & mm \\
\hline UZ & Depth finishing allowance - (only for \(\nabla\) and \(\nabla \mathrm{V} \nabla\) ) & mm \\
\hline \begin{tabular}{l}
Insertion \\
U
\end{tabular} & \begin{tabular}{l}
Various insertion modes can be selected - (only for plane-by-plane machining method and for \(\nabla, \nabla \nabla \nabla\) and \(\nabla \nabla \nabla\) edge): \\
- Predrilled (only for G code) \\
- Vertical: Insert vertically at center of pocket \\
The tool executes the calculated depth infeed vertically at the center of the pocket. \\
Feedrate: Infeed rate as programmed under FZ \\
- Helical: Insert along helical path \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution. If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. \\
Feedrate: Machining feedrate \\
Note: The vertical insertion into pocket center method can be used only if the tool can cut across center or if the workpiece has been predrilled.
\end{tabular} & \\
\hline  & Clamp/release spindle (only for end face \(\mathrm{C} /\) peripheral surface C , if inserted vertically) The function must be set up by the machine manufacturer. & \\
\hline \[
\begin{aligned}
& \hline \text { FZ } \\
& \text { (only for G code) }
\end{aligned}
\] & Depth infeed rate - (for vertical insertion only) & * \\
\hline \begin{tabular}{l}
FZ \\
0 \\
(only for ShopTurn)
\end{tabular} & Depth infeed rate - (for vertical insertion only) & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline EP & Maximum pitch of helix - (for helical insertion only) The helix pitch may be lower due to the geometrical situation. & mm/rev \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline ER & \begin{tabular}{l} 
Radius of helix - (only for helical insertion) \\
The radius must not be larger than the milling cutter radius, otherwise material will \\
remain. Also make sure the circular pocket is not violated.
\end{tabular} & mm \\
\hline \begin{tabular}{l} 
Solid machining \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
\(\bullet \quad\) Complete machining \\
The circular pocket must be milled from a solid workpiece (e.g. casting). \\
Post machining \\
A small pocket or hole has already been machined in the workpiece, which needs \\
to be enlarged. Parameters AZ, and Ø1 must be programmed.
\end{tabular} & \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline \begin{tabular}{l} 
AZ \\
(only for G code)
\end{tabular} & Depth of premachining - (for remachining only) & mm \\
\hline \begin{tabular}{l}
\(\varnothing 1\) \\
(only for G code)
\end{tabular} & Diameter of premachining - (for remachining only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input \({ }^{\text {U }}\)} & \multicolumn{4}{|l|}{- simple} \\
\hline U & Milling direction & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & F U & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U} \\
& \hline
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l} 
Machining \\
surface (only for \\
ShopTurn) \\
\(\mathbf{U}\)
\end{tabular} & \(\bullet\) Face C & Face Y \\
\hline Position (only for & - Peripheral surface C & \\
ShopTurn) & - At the front (face) & \\
\(\mathbf{U}\) & - \begin{tabular}{l} 
Outside (peripheral surface)
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline  & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Machining U & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla \nabla \nabla\) edge (edge finishing, plane-by-plane or helical) \\
- Chamfering
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining type \\
0
\end{tabular} & \begin{tabular}{l}
- Plane by plane \\
Solid machine circular pocket plane-by-plane \\
- Helical \\
Solid machine circular pocket helically
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { zo } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) \\
Reference point \(Y\) \\
Reference point \(Z\)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\]
\[
\mathrm{mm}
\] \\
\hline \begin{tabular}{l}
X0 or LO U \\
Y0 or CO U \\
Z0 (only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point Y or reference point angle polar \\
Reference point \(Z\)
\end{tabular} & mm mm or degrees mm \\
\hline \begin{tabular}{l}
CP \\
XO or LO U \\
Y0 or C0 U \\
Z0 (only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face Y : The positions refer to the reference point: Positioning angle for machining area \\
Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Y0 or CO U } \\
& \text { zo } \\
& \text { X0 } \\
& \text { (only for ShopTurn) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point length polar \\
Reference point \(Z\) \\
Cylinder diameter \(\varnothing\)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { C0 } \\
& \text { Yo } \\
& \text { Z0 } \\
& \text { X0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface Y : The positions refer to the reference point: \\
Positioning angle for machining surface \\
Reference point \(Y\) \\
Reference point \(Z\) \\
Reference point \(X\)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \(\varnothing\) & Diameter of pocket & mm \\
\hline Z1 U & Depth referred to Z0/X0 (inc) or pocket depth (abs) - (only for \(\nabla\), \(\nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge) & mm \\
\hline DXY U & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter - (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed - (only for \(\nabla, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\bar{\nabla}, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla \nabla\) edge) & mm \\
\hline UZ & Depth finishing allowance - (only for \(\nabla, \nabla \nabla \nabla\) ) & mm \\
\hline Insertion U & \begin{tabular}{l}
The following insertion modes can be selected - (only for plane-by-plane machining method and for \(\nabla, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge): \\
- Predrilled: (only for G code) \\
- Vertical: Insert vertically at center of pocket \\
The tool executes the calculated depth infeed at the pocket center in a single block. This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. \\
- Helical: Insert along helical path \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. \\
Feedrate: Machining feedrate \\
Note: The vertical insertion into pocket center method can be used only if the tool can cut across center or if the workpiece has been predrilled.
\end{tabular} & \\
\hline \begin{tabular}{l}
(0) \\
(only for ShopTurn) U
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{C} /\) peripheral surface C , if inserted vertically) The function must be set up by the machine manufacturer & \\
\hline \begin{tabular}{l}
FZ \\
(only for G code)
\end{tabular} & Depth infeed rate - (for vertical insertion only) & * \\
\hline \begin{tabular}{l}
FZ \\
(only for ShopTurn)
\end{tabular} & Depth infeed rate - (for vertical insertion only) & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline EP & Maximum pitch of helix - (for helical insertion only) & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline ER & \begin{tabular}{l}
Radius of helix - (for helical insertion only) \\
The radius cannot be any larger than the milling cutter radius; otherwise, material will remain.
\end{tabular} & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|l|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline \begin{tabular}{l} 
Machining \\
position
\end{tabular} & Mill circular pocket at the programmed position (X0, Y0, Z0). & \begin{tabular}{l} 
Single posi- \\
tion
\end{tabular} & \\
\hline Solid machining & \begin{tabular}{l} 
The rectangular pocket is milled from the solid material - \\
(only for roughing)
\end{tabular} & \begin{tabular}{l} 
Complete \\
machining
\end{tabular} & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.4.4 Rectangular spigot (CYCLE76)}

\section*{Function}

You can mill various rectangular spigots with the "Rectangular spigot" cycle.
You can select from the following shapes with or without a corner radius:


In addition to the required rectangular spigot, you must also define a blank spigot, i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. The blank spigot must not overlap adjacent blank spigots and is automatically placed by the cycle in a central position on the finished spigot.
The spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Rectangular spigot" function several times, with a continually decreasing finishing allowance.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Function}

You can mill various rectangular spigots with the "Rectangular spigot" cycle.
You can select from the following shapes with or without a corner radius:


In addition to the required rectangular spigot, you must also define a blank spigot, i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. The blank spigot must not overlap adjacent blank spigots and is automatically placed by the cycle in a central position on the finished spigot.

The spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Rectangular spigot" function several times, with a continually decreasing finishing allowance.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Approach/retraction}
1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety distance. The starting point is on the positive \(X\) axis rotated through \(\alpha 0\).
2. The tool traverses the spigot contour sideways in a semicircle at the machining feedrate. The tool first executes the infeed at machining depth, followed by the movement in the plane. Depending on the machining direction that has been programmed (upcut/synchronism), the spigot is machined in a clockwise or counterclockwise direction.
3. When the spigot has been circumnavigated once, the tool is removed from the contour in a semicircle; the infeed to the next machining depth is then executed.
4. The spigot is approached again in a semicircle and circumnavigated once. This process is repeated until the programmed spigot depth is reached.
5. The tool moves back to the safety distance at rapid traverse.

\section*{Machining type}
- Roughing

Roughing involves moving around the rectangular spigot until the programmed finishing allowance has been reached.
- Finishing

If you have programmed a finishing allowance, the rectangular spigot is moved around until depth Z1 is reached.
- Chamfering

Chamfering involves edge breaking at the upper edge of the rectangular spigot.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

Mill.

Multi- edge spigot

Rectang. spigot
2. Press the "Milling" softkey.
3. Press the "Multi-edge spigot" and "Rectangular spigot" softkeys. The "Rectangular Spigot" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input U} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{O}
\end{aligned}
\] & Machining plane & & T & Tool name & \\
\hline 0 & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & mm/min mm/tooth \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& \text { FZ } \\
& \text { (only for G code) }
\end{aligned}
\] & Depth infeed rate (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & * \\
\hline \begin{tabular}{l}
Reference point \\
0 \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
The following different reference point positions can be selected: \\
- \(\quad \div\) (center) \\
 \\
- \(\quad\). (bottom right) \\
- \(\because\) (top left) \\
- \(\quad\) (top right)
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining surface \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- Chamfering
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining position U & \begin{tabular}{l}
- Single position \\
Mill rectangular pocket at the programmed position (XO, YO, ZO). \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Zo } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: Reference point X - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline ```
X0 or LO U
Y0 or CO U
ZO
(only for ShopTurn)
``` & \begin{tabular}{l}
Face C: The positions refer to the reference point: \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point \(Z\) - (only for single position)
\end{tabular} & \[
\begin{gathered}
\mathrm{mm} \\
\mathrm{~mm} \\
\mathrm{~mm}
\end{gathered}
\] \\
\hline \begin{tabular}{l}
CP \\
X0 or LO U \\
Y0 or C0 U \\
Z0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face Y : The positions refer to the reference point: \\
Positioning angle for machining area - (only single position) \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Yo or CO U } \\
& \text { Zo } \\
& \text { X0 } \\
& \text { (only for ShopTurn) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C: The positions refer to the reference point: \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position) \\
Cylinder diameter \(\varnothing\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{array}{|l|}
\text { C0 } \\
\text { Yo } \\
\text { zo } \\
\text { X0 } \\
\text { (only for ShopTurn) } \\
\hline
\end{array}
\] & \begin{tabular}{l}
Peripheral surface Y : The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z - (only for single position) \\
Reference point X - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline W & Width of spigot & mm \\
\hline L & Length of spigot & mm \\
\hline R & Corner radius & mm \\
\hline a0 & Angle of rotation & Degrees \\
\hline & Spigot depth (abs) or depth relative to Z0 or X0 (inc) - (only for \(\bar{\nabla}\) and \(\bar{\nabla} \mathrm{V}\) ) & mm \\
\hline DZ & Maximum depth infeed - (only for \(\bar{\nabla}\) and \(\overline{\text { VVF }}\) ) & mm \\
\hline UXY & Plane finishing allowance for the length ( L ) and width ( W ) of the rectangular spigot. Smaller rectangular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance. - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline UZ & Depth finishing allowance (tool axis) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline W1 & \begin{tabular}{l} 
Width of blank spigot (important for determining approach position) - (only for \(\nabla\) \\
and \(\nabla \nabla \nabla\) )
\end{tabular} & mm \\
\hline L1 & \begin{tabular}{l} 
Length of blank spigot (important for determining approach position) - (only for \(\nabla\) \\
and \(\nabla \nabla \nabla\) )
\end{tabular} & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline \begin{tabular}{l} 
ZFS \\
\(\mathbf{U}\)
\end{tabular} & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input U} & \multicolumn{4}{|l|}{- simple} \\
\hline U & Milling direction & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & F U & Feedrate & mm/min \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U} \\
& \hline
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline FZ & Depth infeed rate (only for \(\nabla\) and V V ) & * \\
\hline Machining surface (only for ShopTurn) U & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline Position (only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \[
\begin{aligned}
& \mathbf{d} \\
& \text { (O) } \\
& \text { (only for ShopTurn) } \\
& 0
\end{aligned}
\] & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Machining \(\mathbf{U}\) & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- Chamfering
\end{tabular} & \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Y0 } \\
& \text { Z0 } \\
& \text { (only for G code) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) \\
Reference point \(Y\) \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
X0 or LO U \\
YO or CO U \\
Z0 (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { CP } \\
& \text { X0 or LO } U \\
& \text { Y0 or CO } U \\
& \text { Z0 (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face Y : The positions refer to the reference point: Positioning angle for machining area \\
Reference point \(X\) or reference point length polar \\
Reference point Y or reference point angle polar \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or degrees \\
mm
\end{tabular} \\
\hline \begin{tabular}{l} 
C0 \\
Y0 \\
Z0 \\
X0 \\
(only for ShopTurn) \\
\hline
\end{tabular} & \begin{tabular}{l}
Peripheral surface \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining surface \\
Reference point \(Y\) \\
Reference point \(Z\) \\
Reference point X
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline W & Width of spigot & mm \\
\hline L & Length of spigot & mm \\
\hline R & Corner radius & mm \\
\hline Z1 U & Depth relative to Z0 or X0 (inc) or spigot depth (abs) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline UXY & Plane finishing allowance for the length (L) and width (W) of the rectangular spigot. Smaller rectangular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance.
\[
\text { - (only for } \nabla \text { and } \nabla \nabla \nabla \text { ) }
\] & mm \\
\hline UZ & Depth finishing allowance (tool axis) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline W1 & Width of blank spigot (important for determining approach position) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline L1 & Length of blank spigot (important for determining approach position) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS & Insertion depth of tool tip (abs and inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline Reference point & Position of the reference point: Center & \(\vdots\) \\
\hline \begin{tabular}{l} 
Machining \\
position
\end{tabular} & \begin{tabular}{l} 
Mill rectangular spigot at the programmed position (X0, Y0, \\
ZO).
\end{tabular} & \begin{tabular}{l} 
Single posi- \\
tion
\end{tabular} & \\
\hline\(\alpha 0\) & Angle of rotation & \(0^{\circ}\) & \\
\hline
\end{tabular}

\subsection*{9.4.5 Circular spigot (CYCLE77)}

\section*{Function}

You can mill various circular spigots with the "Circular spigot" function.
In addition to the required circular spigot, you must also define a blank spigot, i.e. the outer limits of the material. The tool moves at rapid traverse outside this area. The blank spigot must not overlap adjacent blank spigots and is automatically placed on the finished spigot in a centered position.

The circular spigot is machined using only one infeed. If you want to machine the spigot using multiple infeeds, you must program the "Circular spigot" function several times with a reducing finishing allowance.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction}
1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and is fed in to the safety clearance. The starting point is always on the positive X axis.
2. The tool approaches the spigot contour sideways in a semicircle at machining feedrate. The tool first executes infeed at machining depth and then moves in the plane. The circular spigot is machined depending on the programmed machining direction (up-cut/down-cut) in a clockwise or counterclockwise direction.
3. When the circular spigot has been traversed once, the tool retracts from the contour in a semicircle and then infeed to the next machining depth is performed.
4. The circular spigot is approached again in a semicircle and traversed once. This process is repeated until the programmed spigot depth is reached.
5. The tool moves back to the safety clearance at rapid traverse.

\section*{Machining type}

You can select the machining mode for milling the circular spigot as follows:
- Roughing

Roughing involves moving round the circular spigot until the programmed finishing allowance has been reached.
- Finishing

If you have programmed a finishing allowance, the circular spigot is moved around until depth Z1 is reached.
- Chamfering

Chamfering involves edge breaking at the upper edge of the circular spigot.

\section*{Procedure}

Multi- edge spigot

Circular spigot
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Multi-edge spigot" and "Circular spigot" softkeys. The "Circular Spigot" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{InputO} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{O}
\end{aligned}
\] & Machining plane & & T & Tool name & \\
\hline U & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & F
U & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
FZ \\
(only for G code)
\end{tabular} & Depth infeed rate & * \\
\hline \begin{tabular}{l}
Machining surface \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
 \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Machining 0 & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- Chamfering
\end{tabular} & \\
\hline Machining position
\[
0
\] & \begin{tabular}{l}
- Single position \\
Mill circular spigot at the programmed position (XO, YO, ZO). \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline X0
Y0
Z0
(only for G code) & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point X - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point \(Z\)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\]
\[
\mathrm{mm}
\] \\
\hline \begin{tabular}{l}
Xo or LO U \\
YO or COU \\
ZO \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face C : The positions refer to the reference point: \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point \(Z\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
CP \\
X0 or LO U \\
YO or COU \\
ZO \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face Y : The positions refer to the reference point: \\
Positioning angle for machining area - (only single position) \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point \(Z\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \begin{tabular}{|l} 
Y0 or CO O \\
Z0 \\
X0 \\
(only for Shop- \\
Turn) \\
\hline
\end{tabular} & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position) \\
Cylinder diameter \(\varnothing\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
C0 \\
Y0 \\
Z0 \\
X0 \\
(only for Shop- \\
Turn) \\
\hline\(\varnothing\)
\end{tabular} & \begin{tabular}{l}
Peripheral surface \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z - (only for single position) \\
Reference point X - (only for single position)
\end{tabular} & \[
\begin{aligned}
& \text { Degrees } \\
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \(\varnothing\) & Diameter of spigot & mm \\
\hline \[
\begin{gathered}
\mathrm{Z} 1 \\
\mathrm{U}
\end{gathered}
\] & Spigot depth (abs) or depth relative to Z0 or X0 (inc) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\) and \(\nabla \nabla \nabla \nabla\) ) & mm \\
\hline UXY & \begin{tabular}{l}
Plane finishing allowance for the length (L) and width (W) of the circular spigot. \\
Smaller circular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance. - (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & mm \\
\hline UZ & Depth finishing allowance (tool axis) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline \(\varnothing 1\) & Diameter of blank spigot (important for determining approach position) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline \[
\begin{aligned}
& \mathrm{ZFS} \\
& \mathrm{U}
\end{aligned}
\] & \begin{tabular}{l}
Insertion depth of tool tip (abs or inc) - (for chamfering only) \\
(ZFS for machining surface, face C/Y or XFS for peripheral surface \(\mathrm{C} / \mathrm{Y}\) )
\end{tabular} & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input U} & \multicolumn{4}{|l|}{- simple} \\
\hline U & Milling direction & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & FO & Feedrate & mm/min \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U} \\
& \hline
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline FZ (only for G code) & Depth infeed rate & * \\
\hline Machining surface (only for ShopTurn) 0 & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline Position (only for ShopTurn)
\[
0
\] & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline O & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Machining U & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- Chamfering
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { zo } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) \\
Reference point \(Y\) \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
X0 or Lo U \\
Yo or CO U \\
Z0 (only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point Z
\end{tabular} & mm mm or degrees mm \\
\hline \[
\begin{aligned}
& \text { CP } \\
& \text { X0 or LO U } \\
& \text { Y0 or CO U } \\
& \text { Z0 (only for Shop- } \\
& \text { Turn) }
\end{aligned}
\] & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: Positioning angle for machining area Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm mm or degrees mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l}
Y0 or C0 \\
Z0 \\
X0(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point angle polar \\
Reference point \(Z\) \\
Cylinder diameter \(\varnothing\)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline C0
Y0
Z0
X0
(only for ShopTurn) & \begin{tabular}{l}
Peripheral surface Y : The positions refer to the reference point: \\
Positioning angle for machining surface \\
Reference point \(Y\) \\
Reference point \(Z\) \\
Reference point \(X\)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline \(\varnothing 1\) & Diameter of blank spigot (important for determining approach position) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline \(\varnothing\) & Diameter of spigot & mm \\
\hline Z1 U & Depth relative to Z0 or X0 (inc) or spigot depth (abs) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline UXY & \begin{tabular}{l}
Plane finishing allowance for the length (L) and width (W) of the rectangular spigot. Smaller rectangular spigot dimensions are obtained by calling the cycle again and programming it with a lower finishing allowance. \\
- (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & mm \\
\hline UZ & Depth finishing allowance (tool axis) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline \[
\begin{aligned}
& \text { ZFS } \\
& U \\
& \hline
\end{aligned}
\] & Insertion depth of tool tip (abs and inc) - (for chamfering only) (ZFS for machining surface, face C/Y or XFS for peripheral surface \(C / Y\) ) & mm \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline \begin{tabular}{l} 
Machining \\
position
\end{tabular} & Mill circular spigot at the programmed position (X0, Y0, Z0). & \begin{tabular}{l} 
Single posi- \\
tion
\end{tabular} & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.4.6 Multi-edge (CYCLE79)}

\section*{Function}

You can mill a multi-edge with any number of edges with the "Multi-edge" cycle.
You can select from the following shapes with or without a corner radius or chamfer:


\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction}
1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and is fed in to the safety clearance.
2. The tool traverses the multi-edge in a quadrant at machining feedrate. The tool first executes infeed at machining depth and then moves in the plane. The multi-edge is machined depending on the programmed machining direction (up-cut/down-cut) in a clockwise or counterclockwise direction.
3. When the first plane has been machined, the tool retracts from the contour in a quadrant and then infeed to the next machining depth is performed.
4. The multi-edge is traversed again in a quadrant. This process is repeated until the depth of the multi-edge has been reached.
5. The tool retracts to the safety clearance at rapid traverse.

\section*{Note}

A multi-edge with more than two edges is traversed helically; with a single or double edge, each edge is machined separately.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.

Multi- edge spigot
Multiedge
3. Press the "Multi-edge spigot" and "Multi-edge" softkeys. The "Multi-edge" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{gathered}
\text { PL } \\
U
\end{gathered}
\] & Machining plane & & T & Tool name & \\
\hline U & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
0 \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0 \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
- Front \\
- back
\end{tabular} & \\
\hline \begin{tabular}{l}
d \\
(0) \\
(only for Shop- \\
Turn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle (only for face Y ) \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline Machining position U (only for G code) & \begin{tabular}{l}
- Single position \\
A multiple edge is milled at the programmed position (XO, YO, ZO). \\
- Position pattern \\
Several multiple edges are milled at the programmed position pattern (e.g. pitch circle, grid, line).
\end{tabular} & \\
\hline \begin{tabular}{l}
X0 (only G code) \\
Y0 (only G code) \\
Z0
\end{tabular} & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point X - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \(\varnothing\) & Diameter of blank spigot & mm \\
\hline N & Number of edges & \\
\hline SW or L U & Width across flats or edge length & mm \\
\hline a0 & Angle of rotation & Degrees \\
\hline R1 or FS1 U & Rounding radius or chamfer width & mm \\
\hline Z1 U & Multi-edge depth (abs) or depth in relation to Z 0 (inc) - (only for \(\nabla\), \(\nabla \nabla \nabla\) and \(\nabla \nabla \nabla \mathrm{edge}\) ) & mm \\
\hline DXY 0 & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter - (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed - (only for \(\bar{\square}\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\bar{\nabla}, \mathrm{V} \nabla \overline{\mathrm{V}}\) and \(\nabla \nabla \nabla\) edge) & mm \\
\hline UZ & Depth finishing allowance - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering only) & \begin{tabular}{l}
mm \\
\(\%\)
\end{tabular} \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{\begin{tabular}{l}
Input \\
U
\end{tabular}} & \multicolumn{4}{|l|}{- simple} \\
\hline U & Milling direction & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & FO & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline Machining surface U & \begin{tabular}{l}
- Face C \\
- Face \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Front \\
- Back
\end{tabular} & \\
\hline \begin{tabular}{l}
Cl \\
(0) \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle (only for face C) \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline Machining U & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline \begin{tabular}{l}
X0 (only G code) \\
Y0 (only G code) \\
Z0
\end{tabular} & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) \\
Reference point \(Y\) \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
mm \\
mm mm
\end{tabular} \\
\hline \(\varnothing\) & Diameter of blank spigot & mm \\
\hline N & Number of edges & \\
\hline SW or L U & Width across flats or edge length & mm \\
\hline
\end{tabular}

Turning
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline R1 and FS1 \(U\) & Rounding radius or chamfer width & \begin{tabular}{l}
mm \\
\(\%\)
\end{tabular} \\
\hline Z1 \(\mathcal{\text { DXY } U}\) & \begin{tabular}{l} 
Multi-edge depth (abs) or depth in relation to Z0 (inc) - (only for \(\nabla, \nabla \nabla \nabla\) and \(\nabla \nabla \nabla\) edge) \\
- Maximum plane infeed \\
Maximum plane infeed as a percentage of the milling cutter diameter \\
- (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & mm \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\nabla, \nabla \nabla \nabla\) and \(\nabla \nabla \nabla\) edge). & mm \\
\hline UZ & Depth finishing allowance (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS \(U\) & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline \begin{tabular}{l} 
Machining \\
position (only for G \\
code)
\end{tabular} & Mill multi-edge at the programmed position (XO, Y0, Z0). & \begin{tabular}{l} 
Single posi- \\
tion
\end{tabular} & \\
\hline\(\alpha 0\) & Angle of rotation & \(0^{\circ}\) & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.4.7 Longitudinal groove (SLOT1)}

\section*{Function}

You can use the "Longitudinal groove" function to mill any longitudinal groove.
The following machining methods are available:
- Mill longitudinal groove from solid material.

Depending on the dimensions of the longitudinal slot in the workpiece drawing, you can select a corresponding reference point for the longitudinal slot.
- First predrill longitudinal groove if, for example, the milling cutter does not cut in the center (for ShopTurn, program the drilling, longitudinal groove and position program blocks in succession).
In this case, select the predrilling position corresponding to the "Insertion", "vertical" parameter (see "Procedure").
Depending on the dimensions of the longitudinal slot in the workpiece drawing, you can select a corresponding reference point for the longitudinal slot.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Longitudinal slot with the width of the tool}

When milling a longitudinal slot, which is located in parallel with the spindle axis, and which should be machined with the width of the tool, then the clamping remains active after insertion in order to achieve more accurate results.

The cycles identify this special case and do not cancel clamping after insertion if the following secondary conditions are fulfilled.

After machining, the clamping in the cycles is canceled again.

\section*{General conditions}
- Finishing longitudinal slot with width = tool diameter
- Roughing longitudinal slot with (width -2 * finishing allowance) \(=\) tool diameter

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction}
1. The tool approaches the center point of the slot at rapid traverse at the height of the retraction plane and adjusts to the safety distance.
2. The tool is inserted into the material according to the method selected.
3. The longitudinal slot is always machined from inside out using the selected machining method.
4. The tool moves back to the safety distance at rapid traverse.

\section*{Machining type}

You can select any of the following machining types for milling the longitudinal slot:
- Roughing

Roughing involves machining the individual planes of the slot one after the other from the inside out, until depth Z 1 or X 1 is reached.
- Finishing

During finishing, the edge is always machined first. The slot edge is approached on the quadrant that joins the corner radius. During the last infeed, the base is finished from the center out.
- Edge finishing

Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfering

Chamfering involves edge breaking at the upper edge of the longitudinal slot.


Figure 9-6 Geometries when chamfering inside contours

\section*{Note}

The following error messages can occur when chamfering inside contours:
- Safety clearance in the program header too large

This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- Immersion depth too large

This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- Tool diameter too large

This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

\section*{Procedure}

\section*{Groove}

Longit. growe
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Groove" and "Longitudinal groove" softkeys. The "Longitudinal Groove (SLOT1)" input window opens.

\subsection*{9.4 Milling}

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G-code program parameters} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{gathered}
\text { PL } \\
\bar{U}
\end{gathered}
\] & Machining plane & & T & Tool name & \\
\hline U & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & \begin{tabular}{l} 
F \\
\hline
\end{tabular} & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline F & Feedrate & * & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Reference point \\
U \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
Position of the reference point: \\
- (lefthand edge) \\
- (inside left) \\
- \(\because\) (center) \\
- \\
- (righthand edge)
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining surface \\
0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
D \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining position
\[
0
\] & \begin{tabular}{l}
- Single position \\
Mill rectangular pocket at the programmed position (XO, Y0, Z0). \\
- Position pattern \\
Position with MCALL
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Y0 } \\
& \text { Z0 } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point X - (only for single position) \\
Reference point \(Y\) - (only for single position) \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
X0 or LO U \\
Y0 or C0 U \\
Z0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face C: The positions refer to the reference point: \\
Reference point \(X\) or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or degrees mm
\end{tabular} \\
\hline ```
CP
X0 or LO U
Y0 or C0 U
Z0
(only for ShopTurn)
``` & \begin{tabular}{l}
Face Y: The positions refer to the reference point: \\
Positioning angle for machining area - (only single position) \\
Reference point \(X\) or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Y0 or CO } \mathrm{U} \\
& \text { Z0 } \\
& \text { X0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C: The positions refer to the reference point: \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position) \\
Cylinder diameter \(\varnothing\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline C0
Y0
Z0
X0
(only for ShopTurn) & \begin{tabular}{l}
Peripheral surface Y: The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point \(Y\) - (only for single position) \\
Reference point \(Z\) - (only for single position) \\
Reference point X - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline W & Slot width & mm \\
\hline L & Slot length & mm \\
\hline a0 & \begin{tabular}{l}
Angle of rotation of slot \\
Face: \(\alpha 0\) refers to the \(X\) axis or to the position of \(C 0\) with a polar reference point Peripheral surface: \(\alpha 0\) refers to the Y axis
\end{tabular} & Degrees \\
\hline Z1 U & Slot depth (abs) or depth relative to Z0 or X0 (inc) - (only for \(\nabla\) and \(\bar{\nabla} \nabla\) ) & mm \\
\hline \begin{tabular}{l}
DXY \\
U (only ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter \\
- (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed - (only for \(\nabla, \nabla \nabla \nabla\) and \(\nabla \nabla \nabla\) edge) & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline UXY & \begin{tabular}{l}
Plane finishing allowance for the length (L) and width (W) of the slot. \\
- (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & mm \\
\hline UZ & Depth finishing allowance (tool axis) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline \begin{tabular}{l}
Insertion \\
0
\end{tabular} & \begin{tabular}{l}
The following insertion modes can be selected - (only for \(\nabla, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge): \\
- Predrilled (only for G code) \\
Approach reference point shifted by the amount of the safety clearance with G0. \\
- Vertical \\
ShopTurn: Depending on the effective milling tool width (milling tool diameter \(\mathrm{x} \operatorname{DXY}[\%]\) ) or DXY [mm] - at the pocket center or at the pocket edge, is moved to the infeed depth. \\
- At the edge of the longitudinal slot ("inside left"): Effective milling tool width >= half the slot width. \\
- At the longitudinal slot center: Effective milling tool width < half the slot width. \\
G code: The tool is inserted to the infeed depth at the reference point "inside left". \\
Note: This setting can be used only if the cutter can cut across center. \\
- Helical (only for G code) \\
Insertion along helical path: \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full longitudinal slot is machined to eliminate the inclined insertion path. \\
- Oscillating Insert with oscillation along center axis of longitudinal slot: \\
The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.
\end{tabular} & \\
\hline \begin{tabular}{l}
a \\
(O) \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle (only for end face C/peripheral surface C, if inserted vertically) \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { FZ } \\
& \text { (only for G code) }
\end{aligned}
\] & Depth infeed rate - (for vertical insertion only) & * \\
\hline \[
\begin{array}{|l|}
\hline \text { FZ } \\
U \\
\text { (only for ShopTurn) } \\
\hline
\end{array}
\] & Depth infeed rate - (only for insertion, predrilled and perpendicular) & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline \begin{tabular}{l}
EP \\
(only for G code)
\end{tabular} & Maximum pitch of helix - (for helical insertion only) & mm/rev \\
\hline \begin{tabular}{l}
ER \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
Radius of helix - (for helical insertion only) \\
The radius cannot be any larger than the milling cutter radius; otherwise, material will remain.
\end{tabular} & mm \\
\hline EW & Maximum insertion angle - (for insertion with oscillation only) & Degrees \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{\begin{tabular}{l}
Input \\
U
\end{tabular}} & \multicolumn{4}{|l|}{- simple} \\
\hline U & Milling direction & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline F & Feedrate & * & FU & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}

\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Zo } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) \\
Reference point \(Y\) \\
Reference point \(Z\)
\end{tabular} & mm
mm
mm \\
\hline \[
\begin{aligned}
& \text { X0 or L0 } \\
& \text { Yo or C0 } \\
& \text { zo } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or degrees mm
\end{tabular} \\
\hline \begin{tabular}{l}
CP \\
X0 or LO \\
Y0 or C0 \\
ZO \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining area \\
Reference point \(X\) or reference point length polar \\
Reference point \(Y\) or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or \\
degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Y0 or CO U } \\
& \text { zo } \\
& \text { X0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point angle polar \\
Reference point Z \\
Cylinder diameter \(\varnothing\)
\end{tabular} & \begin{tabular}{l}
mm or degrees \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { C0 } \\
& \text { Yo } \\
& \text { zo } \\
& \text { X0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface Y : The positions refer to the reference point: \\
Positioning angle for machining surface \\
Reference point \(Y\) \\
Reference point \(Z\) \\
Reference point X
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline W & Slot width & mm \\
\hline L & Slot length & mm \\
\hline Z1 U & Slot depth (abs) or depth referred to Z0 or X0 (inc) - (only for \(\nabla\) and V V ) & mm \\
\hline \begin{tabular}{l}
DXY U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter - (only for \(\nabla\) and \(\nabla \nabla \nabla\) )
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline UXY & Plane finishing allowance for the length (L) and width (W) of the slot (only for \(\nabla\) and \(\nabla \nabla \nabla\) ). & mm \\
\hline UZ & Depth finishing allowance (tool axis) - (only for \(\bar{\nabla}\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline Insertion 0 & \begin{tabular}{l}
The following insertion modes can be selected - (only for \(\nabla, \nabla \nabla \nabla\) or \(\nabla \nabla \nabla\) edge): \\
- Predrilled (only for G code) \\
Approach reference point shifted by the amount of the safety clearance with G0. \\
- Vertical \\
ShopTurn: Depending on the effective milling tool width (milling tool diameter x DXY[\%]) or DXY [mm] - at the pocket center or at the pocket edge, is moved to the infeed depth. \\
- At the edge of the longitudinal slot ("inside left"): Effective milling tool width >= half the slot width. \\
- At the longitudinal slot center: Effective milling tool width < half the slot width. \\
G code: The tool is inserted to the infeed depth at the reference point "inside left". \\
Note: This setting can be used only if the cutter can cut across center. \\
- Helical (only for G code) \\
Insertion along helical path: \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full longitudinal slot is machined to eliminate the inclined insertion path. \\
- Oscillation \\
Insert with oscillation along center axis of longitudinal slot: \\
The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.
\end{tabular} & \\
\hline \begin{tabular}{l}
a \\
(0) \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Clamp/release spindle (only for end face \(\mathrm{C} /\) peripheral surface C , if inserted vertically) \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline \begin{tabular}{l}
FZ \\
(only for G code)
\end{tabular} & Depth infeed rate - (for vertical insertion only) & * \\
\hline FZ U (only for ShopTurn) & Depth infeed rate - (only for insertion, predrilled and perpendicular) & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline \[
\begin{array}{|l|}
\hline \text { EP } \\
\text { (only for G code) } \\
\hline
\end{array}
\] & Maximum pitch of helix - (for helical insertion only) & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline \begin{tabular}{l}
ER \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
Radius of helix - (for helical insertion only) \\
The radius cannot be any larger than the milling cutter radius; otherwise, material will remain.
\end{tabular} & mm \\
\hline EW & Maximum insertion angle - (for insertion with oscillation only) & Degrees \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline PL (only for G code) & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline SC (only for G code) & Safety clearance & 1 mm & x \\
\hline \begin{tabular}{l} 
Reference point (only for G \\
code)
\end{tabular} & Position of the reference point: Center & a & \\
\hline \begin{tabular}{l} 
Machining \\
position (only for G code)
\end{tabular} & Mill slot at the programmed position (X0, Y0, Z0) & \begin{tabular}{l} 
Single posi- \\
tion
\end{tabular} & \\
\hline\(\alpha 0\) & Angle of rotation & \(0^{\circ}\) & \\
\hline
\end{tabular}

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.4.8 Circumferential groove (SLOT2)}

\section*{Function}

You can mill one or several circumferential slots of equal size on a full or pitch circle with the "circumferential slot" cycle.

\section*{Tool size}

Please note that there is a minimum size for the milling cutter used to machine the circumferential slot:
- Roughing:
\(1 / 2\) groove width W - finishing allowance \(\mathrm{UXY} \leq\) milling cutter diameter
- Finishing:

12 groove width \(\mathrm{W} \leq\) milling cutter diameter
- Edge finishing:

Finishing allowance UXY \(\leq\) milling cutter diameter

\section*{Annular groove}

To create an annular groove, you must enter the following values for the "Number N " and
"Aperture angle a1" parameters:
\(\mathrm{N}=1\)
\(\alpha 1=360^{\circ}\)

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction}
1. At the height of the retraction plane, the tool approaches the center point of the semicircle at the end of the groove at rapid traverse and adjusts to the safety distance.
2. It is then inserted into the workpiece at the machining feedrate, allowing for the maximum \(Z\) direction infeed (for face machining), \(X\) direction infeed (for peripheral machining), and the finishing allowance. Depending on the machining direction (up-cut or down-cut), the circumferential groove is machined in a clockwise or counterclockwise direction.
3. When the first circumferential groove is finished, the tool moves to the retraction plane at rapid traverse.
4. The next circumferential groove is approached along a straight line or circular path and then machined.
5. The rapid traverse feedrate for positioning on a circular path is specified in a machine data element.

\section*{Machining type}

You can select the machining mode for milling the circumferential groove as follows:
- Roughing

During roughing, the individual planes of the groove are machined one after the other from the center point of the semicircle at the end of the groove until depth Z1 is reached.
- Finishing

In "Finishing" mode, the edge is always machined first until depth Z 1 is reached. The groove edge is approached on the quadrant that joins the radius. In the last infeed, the base is finished from the center point of the semicircle to the end of the groove.
- Edge finishing

Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.
- Chamfering

Chamfering involves edge breaking at the upper edge of the circumferential groove.


Figure 9-7 Geometries when chamfering inside contours

\section*{Note}

The following error messages can occur when chamfering inside contours:
- Safety clearance in the program header too large

This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- Immersion depth too large

This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- Tool diameter too large

This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

\section*{Mill.}

\section*{Groove}

Circular groove
2. Press the "Milling" softkey.
3. Press the "Groove" and "Circumferential groove" softkeys. The "Circumferential Groove" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{O}
\end{aligned}
\] & Machining plane & & T & Tool name & \\
\hline 0 & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & * & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline Position
\(\mathbf{U}\)
(only for ShopTurn) & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
ld \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Machining
\[
0
\] & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad \nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline FZ (only for G code) & Depth infeed rate & * \\
\hline Circular pattern 0 & \begin{tabular}{l}
- Full circle \\
The circumferential slots are positioned around a full circle. The distance from one circumferential slot to the next circumferential slot is always the same and is calculated by the control. \\
- Pitch circle \\
The circumferential slots are positioned around a pitch circle. The distance from one circumferential slot to the next circumferential slot can be defined using angle \(\alpha 2\).
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Z0 } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point X - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \[
\begin{gathered}
\mathrm{mm} \\
\mathrm{~mm} \\
\mathrm{~mm}
\end{gathered}
\] \\
\hline \begin{tabular}{l}
X0 or LO U \\
YO or CO U \\
Z0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face C: The positions refer to the reference point: \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline \begin{tabular}{l}
CP \\
X0 or LO U \\
Y0 or C0 U \\
Zo \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining area - (only single position) \\
Reference point \(X\) or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees mm \\
mm or degrees mm
\end{tabular} \\
\hline ```
Y0 or CO U
zo
X0
(only for ShopTurn)
``` & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position) \\
Cylinder diameter \(\varnothing\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline  & \begin{tabular}{l}
Peripheral surface \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z - (only for single position) \\
Reference point X - (only for single position)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline N & Number of slots & \\
\hline R & Radius of circumferential slot & mm \\
\hline a0 & Starting angle & Degrees \\
\hline a1 & Opening angle of the slot & Degrees \\
\hline a2 & Advance angle - (for pitch circle only) & Degrees \\
\hline W & Slot width & mm \\
\hline Z1 0 & Slot depth (abs) or depth relative to Z0 or X0 (inc) - (only for \(\nabla\), V V V ) & mm \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\), \(\bar{\nabla} \nabla\) ) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\nabla, \nabla \nabla \nabla\) ) & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Positioning 0 & \begin{tabular}{l}
Positioning motion between the slots: \\
- Straight line: \\
Next position is approached linearly in rapid traverse. \\
- Circular: \\
Next position is approached along a circular path at the feedrate defined in a machine data code.
\end{tabular} & \\
\hline FS & Chamfer width for chamfering (inc) - (for chamfering only) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|l|l|l|l|l|l|l|}
\hline \multicolumn{3}{|l|}{ G code program parameters } & & \multicolumn{2}{l|}{\begin{tabular}{l} 
ShopTurn program parameters \\
\hline \begin{tabular}{l} 
Input \\
\(U\)
\end{tabular} \\
\hline
\end{tabular}} & \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l} 
Machining \\
surface \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l} 
- Face C \\
- Face Y \\
- Peripheral surface C
\end{tabular} & \\
\hline \begin{tabular}{l} 
Position U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l} 
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l} 
D \\
(O) \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l} 
Clamp/release spindle (only for end face Y/peripheral surface Y ) \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline Machining \(\mathbf{U}\) & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\mathrm{V} \nabla \mathrm{V}\) (finishing) \\
- \(\quad\) VVV edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline FZ (only for G code) & Depth infeed rate & * \\
\hline Circular pattern 0 & \begin{tabular}{l}
- Full circle \\
The circumferential slots are positioned around a full circle. The distance from one circumferential slot to the next circumferential slot is always the same and is calculated by the control. \\
- Pitch circle \\
The circumferential slots are positioned around a pitch circle. The distance from one circumferential slot to the next circumferential slot can be defined using angle \(\alpha 2\).
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Z0 } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point X \\
Reference point \(Y\) \\
Reference point Z
\end{tabular} & \[
\begin{gathered}
\mathrm{mm} \\
\mathrm{~mm} \\
\mathrm{~mm}
\end{gathered}
\] \\
\hline \[
\begin{aligned}
& \text { X0 or LO U } \\
& \text { Y0 or CO U } \\
& \text { z0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
mm \\
mm or degrees mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { CP } \\
& \text { X0 or LOU } \\
& \text { Y0 or COU } \\
& \text { ZO } \\
& \text { (only for ShopTurn) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: Positioning angle for machining area \\
Reference point \(X\) or reference point length polar Reference point Y or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or \\
degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Y0 or CO U } \\
& \text { zo } \\
& \text { X0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: Reference point Y or reference point angle polar \\
Reference point \(Z\) \\
Cylinder diameter \(\varnothing\)
\end{tabular} & \begin{tabular}{l}
mm or degrees \\
mm mm
\end{tabular} \\
\hline C0
Y0
Z0
X0
(only for ShopTurn) & \begin{tabular}{l}
Peripheral surface \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining surface \\
Reference point \(Y\) \\
Reference point \(Z\) \\
Reference point X
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline N & Number of slots & mm \\
\hline R & Radius of circumferential slot & Degrees \\
\hline a1 & Opening angle of the slot & Degrees \\
\hline a2 & Advance angle - (for pitch circle only) & Degrees \\
\hline W & Slot width & mm \\
\hline Z1 U & Slot depth (abs) or depth referred to Z0 or X 0 (inc) - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline Positioning
\[
0
\] & \begin{tabular}{l}
Positioning motion between the slots: \\
- Straight line: \\
Next position is approached linearly in rapid traverse. \\
- Circular: \\
Next position is approached along a circular path at the feedrate defined in the machine data.
\end{tabular} & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline PL (only for G code) & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline SC (only for G code) & Safety clearance & 1 mm & x \\
\hline\(\alpha 0\) & Angle of rotation & \(0^{\circ}\) & \\
\hline
\end{tabular}

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{9.4.9 Open groove (CYCLE899)}

\section*{Function}

Use the "Open slot" function if you want to machine open slots.
For roughing, you can choose between the following machining strategies, depending on your workpiece and machine properties.
- Vortex milling
- Plunge cutting

The following machining types are available to completely machine the slot:
- Roughing
- Rough-finishing
- Finishing
- Base finishing
- Edge finishing
- Chamfering

\section*{Vortex milling}

Particularly where hardened materials are concerned, this process is used for roughing and contour machining using coated VHM milling cutters.
Vortex milling is the preferred technique for HSC roughing, as it ensures that the tool is never completely inserted. This means that the set overlap is precisely maintained.

\section*{Plunge cutting}

Plunge cutting is the preferred method of machining slots for "unstable" machines and workpiece geometries. This method generally only exerts forces along the tool axis, i.e. perpendicular to the surface of the pocket/slot to be machined (with the XY plane in \(Z\) direction). Therefore, the tool is subject to virtually no deformation. As a result of the axial loading of the tool, there is hardly any danger of vibration occurring for unstable workpieces.
The cutting depth can be considerably increased. The plunge cutter, as it is known, ensures a longer service life due to less vibration for long overhangs.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.

If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction for vortex milling}
1. The tool approaches the starting point in front of the slot in rapid traverse and maintains the safety clearance.
2. The tool goes to the cutting depth.
3. The open slot is always machined along its entire length using the selected machining method.
4. The tool retracts to the safety clearance in rapid traverse.

\section*{Approach/retraction for plunge cutting}
1. The tool moves in rapid traverse to the starting point in front of the slot at the safety clearance.
2. The open slot is always machined along its entire length using the selected machining method.
3. The tool retracts to the safety clearance in rapid traverse.

\section*{Machining type, roughing vortex milling}

Roughing is performed by moving the milling cutter along a circular path.
While performing this motion, the milling cutter is continuously fed into the plane. Once the milling cutter has traveled along the entire slot, it returns to its starting point, while continuing to move in a circular fashion. By doing this, it removes the next layer (infeed depth) in the \(Z\) direction. This process is repeated until the set slot depth plus the finishing allowance has been reached.


Vortex milling: Down-cut or up-cut


Vortex milling: Down-cut-up-cut

\section*{Supplementary conditions for vortex milling}
- Roughing
\(1 / 2\) slot width W - finishing allowance UXY \(\leq\) milling cutter diameter
- Slot width
minimum \(1.15 \times\) milling cutter diameter + finishing allowance
maximum, \(2 \times\) milling cutter diameter \(+2 x\) finishing allowance
- Radial infeed
minimum, \(0.02 \times\) milling cutter diameter
maximum, \(0.25 \times\) milling cutter diameter
- Maximum infeed depth \(\leq\) cutting height of milling cutter

Please note that the cutting height of the milling cutter cannot be checked.
The maximum radial infeed depends on the milling cutter.
For hard materials, use a lower infeed.

\section*{Machining type, roughing plunge cutting}

Roughing of the slot takes place sequentially along the length of the groove, with the milling cutter performing vertical insertions at the machining feedrate. The milling cutter is then retracted and repositioned at the next insertion point.
The milling cutter moves along the length of the slot, at half the infeed rate, and inserts alternately at the left-hand and right-hand walls.
The first insertion motion takes place at the slot edge, with the milling cutter inserted at half the infeed, less the safety clearance (if the safety clearance is greater than the infeed, this will be on the outside). For this cycle, the maximum width of the slot must be less than double the width of the milling cutter + the finishing allowance.
Following each insertion, the milling cutter is lifted by the height of the safety clearance at the machining feedrate. As far as possible, this occurs during what is known as the retraction process, i.e. if the milling cutter's wrap angle is less than \(180^{\circ}\), it is lifted at an angle below \(45^{\circ}\) in the opposite direction to the bisector of the wrap area.

The milling cutter then traverses over the material in rapid traverse.


\section*{Supplementary conditions for plunge cutting}
- Roughing
\(1 / 2\) slot width \(W\) - finishing allowance UXY \(\leq\) milling cutter diameter
- Maximum radial infeed

The maximum infeed depends on the cutting edge width of the milling cutter.
- Increment

The lateral increment is calculated on the basis of the required slot width, milling cutter diameter and finishing allowance.
- Retraction

Retraction involves the milling cutter being retracted at a \(45^{\circ}\) angle if the wrap angle is less than \(180^{\circ}\). Otherwise, retraction is perpendicular, as is the case with drilling.
- Retraction

Retraction is performed perpendicular to the wrapped surface.
- Safety clearance

Traverse through the safety clearance beyond the end of the workpiece to prevent rounding of the slot walls at the ends.
Please note that the milling cutter's cutting edge cannot be checked for the maximum radial infeed.

\section*{Machining type, rough finishing}

If there is too much residual material on the slot walls, unwanted corners are removed to the finishing dimension.

\section*{Machining type, finishing:}

When finishing walls, the milling cutter travels along the slot walls, whereby just like for roughing, it is again fed in the \(Z\) direction, increment by increment. During this process, the milling cutter travels through the safety clearance beyond the beginning and end of the slot, so that an even slot wall surface can be guaranteed across the entire length of the slot.

\section*{Machining type, edge finishing}

Edge finishing is performed in the same way as finishing, except that the last infeed (finish base) is omitted.

\section*{Machining type, finishing base:}

When finishing the base, the milling cutter moves backwards and forwards once in the finished slot.

\section*{Machining type, chamfering}

Chamfering involves breaking the edge at the upper slot edge.


Figure 9-8 Geometries when chamfering inside contours

\section*{Note}

The following error messages can occur when chamfering inside contours:
- Safety clearance in the program header too large

This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
- Immersion depth too large

This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.
- Tool diameter too large

This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

\section*{Additional supplementary conditions}
- Finishing
\(1 / 2\) slot width \(W \leq\) milling cutter diameter
- Edge finishing

Finishing allowance UXY \(\leq\) milling cutter diameter
- Chamfering

The tip angle must be entered into the tool table.

\section*{Procedure}

\section*{Groove}

Open slot
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Slot" and "Open slot" softkeys.

The "Open slot" input window opens.

\subsection*{9.4 Milling}

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{Input} & \multicolumn{4}{|l|}{- Complete} \\
\hline \[
\begin{gathered}
\text { PL } \\
U
\end{gathered}
\] & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline F & Feedrate & * & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline & & & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face \(Y\) \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline  & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Reference point 0 & \begin{tabular}{l}
Position of the reference point: \\
- (lefthand edge) \\
- \(\quad\) (center) \\
-
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla\) (pre-finishing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\nabla \nabla \nabla\) base (base finishing) \\
- \(\quad\) VVV edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Technology \\
U
\end{tabular} & \begin{tabular}{l}
- Vortex milling \\
The milling cutter performs circular motions along the length of the slot and back again. \\
- Plunge cutting Sequential drilling motion along the tool axis.
\end{tabular} & \\
\hline U & \begin{tabular}{l}
Milling direction: - (except plunge cutting) \\
- Climbing \\
- Conventional \\
- Climbing-conventional milling
\end{tabular} & \\
\hline \begin{tabular}{l}
Machining position \\
O
\end{tabular} & \begin{tabular}{l}
- Single position \\
Mill a slot at the programmed position (XO, YO, ZO). \\
- Position pattern \\
Mill slots at a programmed position pattern (e.g. full circle or grid).
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Z0 } \\
& \text { (only for G code) } \\
& \hline
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: Reference point X - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline ```
X0 or LO U
Y0 or CO U
ZO
(only for ShopTurn)
``` & \begin{tabular}{l}
Face C: The positions refer to the reference point: \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& \text { CP } \\
& \text { X0 or LO U } \\
& \text { Y0 or CO U } \\
& \text { Z0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining area - (only single position) \\
Reference point X or reference point length polar - (only for single position) \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position)
\end{tabular} & Degrees mm mm or degrees mm \\
\hline \[
\begin{aligned}
& \text { Y0 or CO U } \\
& \text { zo } \\
& \text { X0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point angle polar - (only for single position) \\
Reference point Z - (only for single position) \\
Cylinder diameter \(\varnothing\) - (only for single position)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline \[
\begin{array}{|l|}
\hline \text { C0 } \\
\text { Yo } \\
\text { Z0 } \\
\text { X0 } \\
\text { (only for ShopTurn) } \\
\hline
\end{array}
\] & \begin{tabular}{l}
Peripheral surface \(Y\) : The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point \(Y\) - (only for single position) \\
Reference point \(Z\) - (only for single position) \\
Reference point X - (only for single position)
\end{tabular} & \[
\begin{aligned}
& \text { Degrees } \\
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline W & Slot width & mm \\
\hline L & Slot length & mm \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline\(\alpha 0\) & Angle of rotation of slot & Degrees \\
\hline \begin{tabular}{l} 
Z1 U \\
(only for G code)
\end{tabular} & \begin{tabular}{l} 
Slot depth (abs) or depth relative to Z0 (abs) - (only for \(\nabla, \nabla \nabla \nabla, \nabla \nabla \nabla\) base and \\
\(\nabla \nabla)\)
\end{tabular} & mm \\
\hline \begin{tabular}{l} 
Z1 or X1 U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l} 
Slot depth (abs) or depth relative to Z0 or X0 (abs) - (only for \(\nabla, \nabla \nabla \nabla, \nabla \nabla \nabla\) base \\
and \(\nabla \nabla\) ) \\
\((\) Z1 for machining surface, face C/Y or X1 for peripheral surface C/Y)
\end{tabular} & mm \\
\hline DXY & \begin{tabular}{l} 
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter \\
\(-(\) (only for \(\nabla\) )
\end{tabular} & \begin{tabular}{l} 
Maximum depth infeed - (only for \(\nabla, \nabla \nabla, \nabla \nabla \nabla\) and \(\nabla \nabla \nabla\) edge) \\
\(-(\) (only for vortex milling)
\end{tabular} \\
\hline DZ & Plane finishing allowance (slot edge) - (only for \(\nabla, \nabla \nabla\) and \(\nabla \nabla \nabla\) base) & mm \\
\hline UXY & Depth finishing allowance (slot base) - (only for \(\nabla, \nabla \nabla\) and \(\nabla \nabla \nabla\) edge) & mm \\
\hline UZ & Chamfer width for chamfering (inc) - (for chamfering only) & mm \\
\hline FS & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline ZFS U & * Unit of feedrate as programmed before the cycle call & mm \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{2}{|l|}{InputU} & \multicolumn{4}{|l|}{- simple} \\
\hline & & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline \multirow[t]{2}{*}{F} & Feedrate & * & FU & Feedrate & mm/min \(\mathrm{mm} / \mathrm{rev}\) \\
\hline & & & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& 0
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\(\left.\begin{array}{|l|l|l|}\hline \text { Parameter } & \text { Description } & \\ \hline \begin{array}{l}\text { Machining } \\ \text { surface U } \\ \text { (only for ShopTurn) }\end{array} & \text { - Face C } & \text { - Face Y } \\ & \text { - Peripheral surface C } \\ \text { - Peripheral surface } \mathrm{Y}\end{array}\right)\)
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline Machining U & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \mathrm{V} \nabla\) (pre-finishing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- \(\quad\) VVV base (base finishing) \\
- \(\quad\) VVV edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { Technology } \\
& 0
\end{aligned}
\] & \begin{tabular}{l}
- Vortex milling \\
The milling cutter performs circular motions along the length of the slot and back again. \\
- Plunge cutting \\
Sequential drilling motion along the tool axis.
\end{tabular} & \\
\hline 0 & \begin{tabular}{l}
Milling direction - (except plunge cutting) \\
- Climbing \\
- Conventional \\
- Climbing-conventional milling
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { zo } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) \\
Reference point \(Y\) \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { X0 or LO U } \\
& \text { Y0 or CO U } \\
& \text { Z0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point Y or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
mm \\
mm or degrees mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { CP } \\
& \text { X0 or LO U } \\
& \text { Y0 or CO U } \\
& \text { Z0 } \\
& \text { (only for ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face \(Y\) : The positions refer to the reference point: Positioning angle for machining area \\
Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or degrees \\
mm
\end{tabular} \\
\hline Y0 or CO U
Z0
X0
(only for ShopTurn) & \begin{tabular}{l}
Peripheral surface C : The positions refer to the reference point: \\
Reference point Y or reference point angle polar \\
Reference point \(Z\) \\
Cylinder diameter \(\varnothing\)
\end{tabular} & \begin{tabular}{l}
mm or degrees \\
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}
\(\left.\begin{array}{|l|l|l|}\hline \text { Parameter } & \text { Description } & \\
\hline \text { C0 } & \begin{array}{l}\text { Peripheral surface Y: The positions refer to the reference point: } \\
\text { Positioning angle for machining surface } \\
\text { Reference point } Y \\
\text { Z0 } \\
\text { R0 } \\
\text { (only for ShopTurn) } \\
\text { Reference point Z }\end{array} & \text { Slot width }\end{array} \begin{array}{l}\text { Degrees } \\
\mathrm{mm} \\
\mathrm{mm} \\
\mathrm{mm}\end{array}\right]\)\begin{tabular}{l}
\hline W \\
\hline L
\end{tabular}

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline PL (only for G code) & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline SC (only for G code) & Safety clearance & 1 mm & x \\
\hline Reference point & Position of the reference point: Center & \(\vdots\). & \\
\hline \begin{tabular}{l} 
Machining \\
position
\end{tabular} & Mill slot at the programmed position (X0, Y0, ZO). & \begin{tabular}{l} 
Single posi- \\
tion
\end{tabular} & \\
\hline\(\alpha 0\) & Angle of rotation of slot & \(0^{\circ}\) & \\
\hline
\end{tabular}

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{9.4.10 Long hole (LONGHOLE) - only for G code program}

\section*{Function}

In contrast to the groove, the width of the elongated hole is determined by the tool diameter.
Internally in the cycle, an optimum traversing path of the tool is determined, ruling out unnecessary idle passes. If several depth infeeds are required to machine an elongated hole, the infeed is carried out alternately at the end points. The path to be traversed in the plane along the longitudinal axis of the elongated hole changes its direction after each infeed. The cycle searches for the shortest path when changing to the next elongated hole.

\section*{Note}

The cycle requires a milling cutter with a "face tooth cutting over center" (DIN 844).

\section*{Approach/retraction}
1. Using G0, the starting position for the cycle is approached. In both axes of the current plane, the closest end point of the first elongated hole to be machined is approached at the level of the retraction plane in the tool axis and then lowered to the reference point shifted by the amount of the safety clearance.
2. Each elongated hole is milled in a reciprocating motion. The machining in the plane is performed using G1 and the programmed feedrate. At each reversal point, the infeed to the next machining depth calculated internally in the cycle is performed with G1 and the feedrate, until the final depth is reached.
3. Retraction to the retraction plane using G0 and approach to the next elongated hole on the shortest path.
4. After the last elongated hole has been machined, the tool at the position reached last in the machining plane is moved with G0 to the retraction plane, and the cycle terminated.

\section*{Procedure}
1. The part program to be executed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Groove" and "Elongated hole" softkeys.

The "Elongated Hole" input window opens.

\subsection*{9.4 Milling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline PL U & Machining plane & \\
\hline RP & Retraction plane (abs) & \\
\hline SC & Safety clearance (inc) & \\
\hline F & Feedrate & * \\
\hline Machining type & \begin{tabular}{l}
- Plane-by-plane \\
The tool is inserted to infeed depth in the pocket center. \\
Note: This setting can be used only if the cutter can cut across center. \\
- Oscillating Insert with oscillation along center axis of longitudinal slot: \\
The cutter center point oscillates along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.
\end{tabular} & mm \\
\hline Reference point
\[
0
\] & Position of the reference point: & \\
\hline Machining position
\[
U
\] & \begin{tabular}{l}
- Single position \\
An elongated hole is machined at the programmed position (XO, YO, ZO). \\
- Position pattern \\
Several elongated holes are machined in the programmed position pattern (e.g. pitch circle, grid, line).
\end{tabular} & \\
\hline \[
\begin{aligned}
& \mathrm{XO} \\
& \mathrm{YO} \\
& \mathrm{ZO}
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point X - (for single position only) \\
Reference point \(Y\) - (for single position only) \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline L & Elongated hole length & mm \\
\hline a0 & Angle of rotation & Degrees \\
\hline Z1 U & Elongated hole depth (abs) or depth in relation to Z0 (inc) & mm \\
\hline DZ & Maximum depth infeed & mm \\
\hline FZ & Depth infeed rate & * \\
\hline
\end{tabular}

\subsection*{9.4.11 Thread milling (CYCLE70)}

\section*{Function}

Using a thread cutter, internal or external threads can be machined with the same pitch. Threads can be machined as right-hand or left-hand threads and from top to bottom or vice versa.

For metric threads (thread pitch \(P\) in \(\mathrm{mm} / \mathrm{rev}\) ), the cycle assigns a value (calculated on the basis of the thread pitch) to the thread depth H1 parameter. You can change this value. The default selection must be activated via a machine data code.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

The entered feedrate acts on the workpiece contour, i.e. it refers to the thread diameter. However the feedrate of the cutter center point is displayed. That is why a smaller value is displayed for internal threads and a larger value is displayed for external threads than was entered.

\section*{Approach/retraction when milling internal threads}
1. Positioning on retraction plane with rapid traverse.
2. Approach of starting point of the approach circle in the current plane with rapid traverse.
3. Infeed to a starting point in the tool axis calculated internally in the controller with rapid traverse.
4. Approach motion to thread diameter on an approach circle calculated internally in the controller with the programmed feedrate, taking into account the finishing allowance and maximum plane infeed.
5. Thread cutting along a spiral path in clockwise or counter-clockwise direction (depending on whether it is left-hand/right-hand thread, for number of cutting teeth of a milling plate \((\mathrm{NT}) \geq 2\) only one rotation, offset in the \(Z\) direction).

To reach the programmed thread length, traversing is beyond the Z1 value for different distances depending on the thread parameters.
6. Exit motion along a circular path in the same rotational direction at programmed feedrate.
7. With a programmed number of threads per cutting edge NT \(>2\), the tool is fed in (offset) by the amount NT-1 in the \(Z\) direction. Points 4 to 7 are repeated until the programmed thread depth is reached.
8. If the plane infeed is less than the thread depth, points 3 to 7 are repeated until the thread depth + programmed allowance is reached.
9. Retract on the thread center point and then to retraction plane in the tool axis in rapid traverse.

Please note that when milling an internal thread the tool must not exceed the following value:
Milling cutter diameter < (nominal diameter - \(2 \cdot\) thread depth H1)

\section*{Approach/retraction when milling external threads}
1. Positioning on retraction plane with rapid traverse.
2. Approach of starting point of the approach circle in the current plane with rapid traverse.
3. Infeed to a starting point in the tool axis calculated internally in the controller with rapid traverse.
4. Approach motion to thread core diameter on an approach circle calculated internally in the controller with the programmed feedrate, taking into account the finishing allowance and maximum plane infeed.
5. Cut thread along a spiral path in clockwise or counter-clockwise direction (depending on whether it is left-hand/right-hand thread, with \(N T \geq 2\) only one rotation, offset in \(Z\) direction).
To reach the programmed thread length, traversing is beyond the Z 1 value for different distances depending on the thread parameters.
6. Exit motion along a circular path in opposite rotational direction at programmed feedrate.
7. With a programmed number of threads per cutting edge NT \(>2\), the tool is fed in (offset) by the amount NT-1 in the \(Z\) direction. Points 4 to 7 are repeated until the programmed thread depth is reached.
8. If the plane infeed is less than the thread depth, points 3 to 7 are repeated until the thread depth + programmed allowance is reached.
9. Retraction on the retraction plane in the tool axis with rapid traverse.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
```

Mill.

```

Thread
milling
2. Press the "Milling" softkey.
3. Press the "Thread milling" softkey. The "Thread Milling" input window opens.

Table 9-1

\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
D \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Machining & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing)
\end{tabular} & \\
\hline U & \begin{tabular}{l}
Machining direction: \\
- \(\mathrm{Z} 0 \rightarrow \mathrm{Z1}\) \\
Machining from top to bottom \\
- Z1 \(\rightarrow\) Z0 \\
Machining from bottom to top
\end{tabular} & \\
\hline U & \begin{tabular}{l}
Direction of rotation of the thread: \\
- Right-hand thread \\
A right-hand thread is cut. \\
- Left-hand thread \\
A left-hand thread is cut.
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.4 Milling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \(u\) & \begin{tabular}{l}
Position of the thread: \\
- Internal thread \\
An internal thread is cut. \\
- External thread \\
An external thread is cut.
\end{tabular} & \\
\hline NT & \begin{tabular}{l}
Number of teeth per cutting edge \\
Single or multiple toothed milling inserts can be used. The motions required are executed by the cycle internally, so that the tip of the bottom tooth on the milling tool cutting edge corresponds to the programmed end position when the thread end position is reached. Depending on the cutting edge geometry of the milling insert, the retraction path must be taken into account at the base of the workpiece.
\end{tabular} & \\
\hline \begin{tabular}{l}
0 \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
Machining position: \\
- Single position \\
- Position pattern (MCALL)
\end{tabular} & \\
\hline \[
\begin{aligned}
& \text { X0 } \\
& \text { Yo } \\
& \text { Zo } \\
& \text { (only for G code) }
\end{aligned}
\] & \begin{tabular}{l}
The positions refer to the center point: \\
Reference point X - (only for single position) \\
Reference point Y - (only for single position) \\
Reference point \(Z\)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline Z1 U & End point of the thread (abs) or thread length (inc) & mm \\
\hline \begin{tabular}{l}
Table \\
U
\end{tabular} & \begin{tabular}{l}
Thread table selection: \\
- Without \\
- ISO metric \\
- Whitworth BSW \\
- Whitworth BSP \\
- UNC
\end{tabular} & \\
\hline Selection - (not for table "without") & \begin{tabular}{l}
Selection, table value: e.g. \\
- M3; M10; etc. (ISO metric) \\
- W3/4"; etc. (Whitworth BSW) \\
- G3/4"; etc. (Whitworth BSP) \\
- N1" - 8 UNC; etc. (UNC)
\end{tabular} & \\
\hline P & Display of the thread pitch for the parameter input in the input field "Table" and "Selection". & \begin{tabular}{l}
MODULUS \\
turns/" \\
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
PU \\
- (selection \\
option only for table \\
selection "without")
\end{tabular} & \begin{tabular}{l}
Pitch ... \\
- In MODULUS: For example, generally used for worm gears that mesh with a gear wheel. \\
- Per inch: Used with pipe threads, for example. \\
When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. \\
- In mm/rev \\
- In inch/rev \\
The tool used depends on the thread pitch.
\end{tabular} & \begin{tabular}{l}
MODULUS \\
Turns/" \\
\(\mathrm{mm} / \mathrm{rev}\) \\
in/rev
\end{tabular} \\
\hline \(\varnothing\) & \begin{tabular}{l}
Nominal diameter \\
Example: Nominal diameter of M12 \(=12 \mathrm{~mm}\)
\end{tabular} & mm \\
\hline H1 & Thread depth & mm \\
\hline DXY & Maximum plane infeed & mm \\
\hline rev & Finishing allowance in X and Y - (only for \(\nabla\) ) & mm \\
\hline aS & Starting angle & Degrees \\
\hline
\end{tabular}

\subsection*{9.4.12 Engraving (CYCLE60)}

\section*{Function}

The "Engraving" function is used to engrave a text on a workpiece along a line or arc.
You can enter the text directly in the text field as "fixed text" or assign it via a variable as "variable text".

Engraving uses a proportional font, i.e., the individual characters are of different widths.

\section*{Approach/retraction}
1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and adjusts to the safety clearance.
2. The tool moves to the machining depth FZ at the infeed feedrate \(Z 1\) and mills the characters.
3. The tool retracts to the safety clearance at rapid traverse and moves along a straight line to the next character.
4. Steps 2 and 3 are repeated until the entire text has been milled.
5. The tool moves to the retraction plane in rapid traverse.

\section*{Procedure}

Engrauing

\section*{Entering the engraving text}

Special character


OK

Delete
text

Delete
Lower-case
letters

Variable

Date

\section*{Variable}

Time
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Engraving" softkey.

The "Engraving" input window opens.
4. Press the "Special characters" softkey if you need a character that does not appear on the input keys.
The "Special characters" window appears.
- Position the cursor on the desired character.
- Press the "OK" softkey.

The selected character is inserted into the text at the cursor position.
5. If you wish to delete the complete text, press the "Delete text" and "Delete" softkeys one after the other.
6. Press the "Lowercase" softkey to enter lowercase letters. Press it again to enter uppercase letters.
7. Press the "Variable" and "Date" softkeys if you want to engrave the current date.

The data is inserted in the European date format (<DD>.<MM>.<YYYY>).
To obtain a different date format, you must adapt the format specified in the text field. For example, to engrave the date in the American date format (month/day/year => 8/16/04), change the format to <M>/<D>/<YY>.
7. Press the "Variable" and "Time" softkeys if you want to engrave the current time.

The time is inserted in the European format (<TIME24>).
To have the time in the American format, change the format to <TIME12>.
Example:
Text entry: Time: <TIME24> Execute: Time: 16.35
Time: <TIME12> Execute: Time: 04.35 PM

\section*{Uariable \\ Quantity \\ 000123}

Uariable

Number
123.456
7. - Press the "Variable" and "Workpiece count 000123 " softkeys to engrave a workpiece count with a fixed number of digits and leading zeroes.
The format text <\#\#\#\#\#\#,_\$AC_ACTUAL_PARTS> is inserted and you return to the engraving field with the softkey bar.
- Define the number of digits by adjusting the number of place holders (\#) in the engraving field.
If the specified number of positions (e.g. \#\#) is not sufficient to represent the unit quantity, then the cycle automatically increases the number of positions.
- OR
7. - Press the "Variable" and "Workpiece count 123 " softkeys if you want to engrave a workpiece count without leading zeroes.

The format text <\#,_\$AC_ACTUAL_PARTS> is inserted and you return to the engraving field with the softkey bar.
- Define the number of digits by adjusting the number of place holders in the engraving field.

If the specified number of digits is not enough to display the workpiece count (e.g. 123), the cycle will automatically increase the number digits.
7. - Press the "Variable" and "Number 123.456 " softkeys if you want to engrave a any number in a certain format.
The format text <\#.\#\#\#,_VAR_NUM> is inserted and you return to the engraving field with the softkey bar.
- The place holders \#.\#\#\# define the digit format in which the number defined in _VAR_NUM will be engraved.
For example, if you have stored 12.35 in _VAR_NUM, you can format the variable as follows.
\begin{tabular}{lll} 
Input & Output & \begin{tabular}{l} 
Meaning \\
<\#,_VAR_NUM>
\end{tabular} \\
12 & \begin{tabular}{l} 
Places before decimal point un- \\
formatted, no places after the \\
decimal point
\end{tabular} \\
<\#\#\#\#,_VAR_NUM> & 0012 & \begin{tabular}{l} 
4 places before decimal point, \\
leading zeros, no places after the \\
decimal point
\end{tabular}
\end{tabular}
\begin{tabular}{lll} 
<\#,_VAR_NUM> & 12 & \begin{tabular}{l} 
4 places before decimal point, \\
leading blanks, no places after the \\
decimal point
\end{tabular} \\
<\#.,_VAR_NUM> & 12.35 & \begin{tabular}{l} 
Places before and after the deci- \\
mal point not formatted.
\end{tabular} \\
<\#.\#,_VAR_NUM> & 12.4 & \begin{tabular}{l} 
Places before decimal point un- \\
formatted, \\
1 place after the decimal point \\
(rounded)
\end{tabular} \\
<\#.\#\#,_VAR_NUM> & 12.35 & \begin{tabular}{l} 
Places before decimal point un- \\
formatted, \\
2 places after the decimal point \\
(rounded)
\end{tabular} \\
<\#.\#\#\#\#,_VAR_NUM> & 12.3500 & \begin{tabular}{l} 
Places before decimal point un- \\
formatted, \\
4 places after the decimal point \\
(rounded)
\end{tabular}
\end{tabular}

If there is insufficient space in front of the decimal point to display the number entered, it is automatically extended. If the specified number of digits is larger than the number to be engraved, the output format is automatically filled with the appropriate number of leading and trailing zeroes.
You can optionally use blanks to format before the decimal place. Instead of _VAR_NUM you can use any other numeric variable (e.g. RO).
7. Press the "Variable" and "Variable text" softkeys if you want to take the text to be engraved (up to 200 characters) from a variable.

The format text <Text, _VAR_TEXT> is inserted and you return to the engraving field with the softkey bar.
You can use any other text variable instead of _VAR_TEXT.

\section*{Note}

\section*{Entering the engraving text}

Only single-line entries without line break are permissible!

\section*{Variable texts}

There are various ways of defining variable text:
- Date and time

For example, you can engrave the time and date of manufacture on a workpiece. The values for date and time are read from the NCK.
- Quantity

Using the workpiece variables you can assign a consecutive number to the workpieces.
You can define the format (number of digits, leading zeroes).
The place holder (\#) is used to format the number of digits at which the workpiece counts output will begin.
If you do not want to output a count of 1 for the first workpiece, you can specify an additive value (e.g., <\#,\$AC_ACTUAL_PARTS + 100>). The workpiece count output is then incremented by this value (e. g. 101, 102, 103,...).
- Numbers

When outputting number (e. g. measurement results), you can select the output format (digits either side of the point) of the number to be engraved.
- Text

Instead of entering a fixed text in the engraving text field, you can specify the text to be engraved via a text variable (e. g., _VAR_TEXT="ABC123").

\section*{Mirror writing}

You can engrave the text mirrored on the workpiece.

\section*{Full circle}

If you want to distribute the characters evenly around a full circle, enter the arc angle \(\alpha 2=360^{\circ}\). The cycle then distributes the characters evenly around the full circle.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \[
\begin{gathered}
\mathrm{PL} \\
\mathrm{O}
\end{gathered}
\] & Machining plane & & T & Tool name & \\
\hline 0 & Milling direction & & D & Cutting edge number & \\
\hline RP & Retraction plane & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline F & Feedrate & mm/min & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline FZ (only for G code) & Depth infeed rate & * \\
\hline FZ U (only for ShopTurn) & Depth infeed rate & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline \begin{tabular}{l}
Machining surface \\
O \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline  & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline \begin{tabular}{l}
Alignment \\
0
\end{tabular} & \begin{tabular}{|l|l|l|}
\hline - & ABC & (linear alignment) \\
- & \(\mathrm{ABB}_{\mathrm{C}}\) & (curved alignment) \\
- & \(\boldsymbol{A}_{\mathrm{B}} \mathrm{C}\) & (curved alignment) \\
\hline
\end{tabular} & \\
\hline Reference point U &  & \\
\hline Mirror writing & \begin{tabular}{l}
- Yes \\
The mirrored text is engraved on the workpiece. \\
- No \\
The text is engraved on the workpiece without mirroring.
\end{tabular} & \\
\hline Engraving text & maximum 100 characters & \\
\hline \begin{tabular}{l}
X0 or R U \\
Y0 or a0 U \\
Z0 \\
(only for G code)
\end{tabular} & \begin{tabular}{l}
The positions refer to the reference point: \\
Reference point \(X\) or reference point length polar Reference point Y or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
mm \\
mm or de- \\
grees \\
mm
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& \text { X0 or L0 U } \\
& \text { Y0 or CO U } \\
& \text { Z0 } \\
& \text { (only ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Face C: The positions refer to the reference point: Reference point \(X\) or reference point length polar Reference point \(Y\) or reference point angle polar \\
Reference point \(Z\)
\end{tabular} & \begin{tabular}{l}
mm \\
mm or degrees mm
\end{tabular} \\
\hline CP
X0 or LO U
Y0 or CO U
Z0
(only ShopTurn) & \begin{tabular}{l}
Face Y: The positions refer to the reference point: Positioning angle for machining area \\
Reference point \(X\) or reference point length polar \\
Reference point Y or reference point angle polar \\
Reference point Z
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm or degrees \\
mm
\end{tabular} \\
\hline \[
\begin{aligned}
& \text { Y0 or CO U } \\
& \text { Z0 } \\
& \text { X0 } \\
& \text { (only ShopTurn) }
\end{aligned}
\] & \begin{tabular}{l}
Peripheral surface C: The positions refer to the reference point: \\
Reference point \(Y\) or reference point angle polar - (only for single position) \\
Reference point \(Z\) \\
Cylinder diameter \(\varnothing\)
\end{tabular} & \begin{tabular}{l}
mm or de- \\
grees \\
mm \\
mm
\end{tabular} \\
\hline C0
Y0
Z0
X0
(only ShopTurn) & \begin{tabular}{l}
Peripheral surface Y: The positions refer to the reference point: \\
Positioning angle for machining surface - (only for single position) \\
Reference point \(Y\) \\
Reference point \(Z\) \\
Reference point \(X\)
\end{tabular} & \begin{tabular}{l}
Degrees \\
mm \\
mm \\
mm
\end{tabular} \\
\hline Z1 U & Engraving depth (abs) or referenced depth (inc) & mm \\
\hline W & Character height & mm \\
\hline DX1 or a2 U & Distance between characters or angle of opening - (for curved alignment only) & \begin{tabular}{l}
mm or \\
Degrees
\end{tabular} \\
\hline DX1 or DX2 U & Distance between characters or total width - (for linear alignment only) & mm \\
\hline a1 & Text direction (for linear alignment only) & Degrees \\
\hline XM or LM U (only G code) & Center point X (abs) or center point length polar - (for curved alignment only) & mm \\
\hline YM or aM U (only G code) & Center point \(Y\) (abs) or center point angle polar - (for curved alignment only) & mm \\
\hline YM or CM U (only ShopTurn) & \begin{tabular}{l}
Center point Y or C (abs.) - (for curved alignment only) \\
- (only for machining surface, peripheral surface C/Y)
\end{tabular} & mm or degrees \\
\hline \begin{tabular}{l}
ZM \\
(only ShopTurn)
\end{tabular} & \begin{tabular}{l}
Center point \(Z\) (abs.) - (for curved alignment only) \\
- (only for machining surface, peripheral surface C/Y)
\end{tabular} & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\subsection*{9.5 Contour milling}

\subsection*{9.5.1 General information}

\section*{Function}

You can mill simple or complex contours with the "Contour milling" cycle. You can define open contours or closed contours (pockets, islands, spigots).

A contour comprises separate contour elements, whereby at least two and up to 250 elements result in a defined contour. Radii, chamfers and tangential transitions are available as contour transition elements.

The integrated contour calculator calculates the intersection points of the individual contour elements taking into account the geometrical relationships, which allows you to enter incompletely dimensioned elements.
With contour milling, you must always program the geometry of the contour before you program the technology.

\subsection*{9.5.2 Representation of the contour}

\section*{G code program}

In the editor, the contour is represented in a program section using individual program blocks. If you open an individual block, then the contour is opened.

\section*{ShopTurn program}

The cycle represents a contour as a program block in the program. If you open this block, the individual contour elements are listed symbolically and displayed in broken-line graphics.

\section*{Symbolic representation}

The individual contour elements are represented by symbols adjacent to the graphics window. They appear in the order in which they were entered.
\begin{tabular}{|l|c|l|}
\hline Contour element & Symbol & Meaning \\
\hline Starting point & - & Starting point of the contour \\
\hline Straight line up & & Straight line in \(90^{\circ}\) grid \\
\hline Straight line down & \(\longrightarrow\) & Straight line in \(90^{\circ}\) grid \\
\hline Straight line left & \(\longrightarrow\) & Straight line in \(90^{\circ}\) grid \\
\hline Straight line right & & Straight line in \(90^{\circ}\) grid \\
\hline
\end{tabular}
\begin{tabular}{|l|c|l|}
\hline Contour element & \multicolumn{2}{|c|}{ Symbol } \\
\hline Straight line in any direction & Meaning \\
\hline Arc right & & Straight line with any gradient \\
\hline Arc left & & Circle \\
\hline Pole & END & \begin{tabular}{l} 
Straight diagonal or circle in \\
polar coordinates
\end{tabular} \\
\hline Finish contour & & End of contour definition \\
\hline
\end{tabular}

The different colors of the symbols indicate their status:
\begin{tabular}{|l|c|l|}
\hline Foreground & Background & Meaning \\
\hline Black & Blue & Cursor on active element \\
\hline Black & Orange & Cursor on current element \\
\hline Black & White & Normal element \\
\hline Red & White & \begin{tabular}{l} 
Element not currently evaluated \\
(element will only be evaluated \\
when it is selected with the \\
cursor)
\end{tabular} \\
\hline
\end{tabular}

\section*{Graphic display}

The progress of contour programming is shown in broken-line graphics while the contour elements are being entered.

When the contour element has been created, it can be displayed in different line styles and colors:
- Black: Programmed contour
- Orange: Current contour element
- Green dashed: Alternative element
- Blue dotted: Partially defined element

The scaling of the coordinate system is adjusted automatically to match the complete contour.

The position of the coordinate system is displayed in the graphics window.

\subsection*{9.5.3 Creating a new contour}

\section*{Function}

For each contour that you want to mill, you must create a new contour.
The contours are stored at the end of the program.

\section*{Note}

When programming in the G code, it must be ensured that the contours are located after the end of program identifier!

The first step in creating a contour is to specify a starting point. Enter the contour element. The contour processor then automatically defines the end of the contour.

If you alter the tool axis, the cycle will automatically adjust the associated starting point axes. You can enter any additional commands (up to 40 characters) in \(G\) code format for the starting point.

\section*{Additional commands}

You can program feedrates and M commands, for example, using additional G code commands. You can enter the additional commands (max. 40 characters) in the extended parameter screens ("All parameters" softkey). However, make sure that the additional commands do not collide with the generated \(G\) code of the contour. Therefore, do not use any G code commands of group 1 (G0, G1, G2, G3), no coordinates in the plane and no G code commands that have to be programmed in a separate block.

\section*{Procedure}

\section*{Cont.}
mill.
New contour
4. Enter a contour name.
5. Press the "Accept" softkey.

Accept The "New Contour" input window opens.
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Contour milling" and "New contour" soffkeys.

The input screen for the starting point of the contour appears. You can enter Cartesian or polar coordinates.

\section*{Cartesian starting point}
1. Enter the starting point for the contour.
2. Enter any additional commands in G code format, as required.
3. Press the "Accept" softkey.
4. Enter the individual contour elements.

\section*{Polar starting point}

1. Press the "Pole" softkey
2. Enter the pole position in Cartesian coordinates.
3. Enter the starting point for the contour in polar coordinates.
4. Enter any additional commands in G code format, as required.
5. Press the "Accept" softkey.
6. Enter the individual contour elements.
\begin{tabular}{|c|c|c|c|}
\hline \multicolumn{2}{|l|}{parameters} & Description & Unit \\
\hline \multicolumn{2}{|l|}{\begin{tabular}{l}
Machining surface \\
U \\
(only for ShopTurn)
\end{tabular}} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface \(Y\)
\end{tabular} & \\
\hline \multicolumn{2}{|l|}{PL U (only for G code)} & \begin{tabular}{l}
Machining plane \\
- G17 (XY) \\
- G19 (YZ)
\end{tabular} & \\
\hline \multicolumn{2}{|l|}{(only ShopTurn)} & \begin{tabular}{l}
Cylinder diameter \\
(only peripheral surface C)
\end{tabular} & mm \\
\hline \[
\begin{aligned}
& \text { G17 } \\
& \text { or } \\
& \text { face C/Y/B }
\end{aligned}
\] & \begin{tabular}{l}
G19 \\
or \\
peripheral surface C/Y
\end{tabular} & & \\
\hline \[
\begin{aligned}
& X \\
& Y \\
& \hline
\end{aligned}
\] & \[
\begin{aligned}
& \mathrm{Y} \\
& \mathrm{Z}
\end{aligned}
\] & \begin{tabular}{l}
Cartesian: \\
Starting point X or Y (abs) \\
Starting point Y or Z (abs)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& \mathrm{X} \\
& \mathrm{Y}
\end{aligned}
\] & \[
\begin{aligned}
& \mathrm{Y} \\
& \mathrm{Z}
\end{aligned}
\] & \begin{tabular}{l}
Polar: \\
Position pole (abs) \\
Position pole (abs)
\end{tabular} & \[
\begin{gathered}
\mathrm{mm} \\
\mathrm{~mm}
\end{gathered}
\] \\
\hline \multicolumn{2}{|l|}{\begin{tabular}{l}
Starting point \\
L1 \\
\$1
\end{tabular}} & \begin{tabular}{l}
Distance to pole, end point (abs) \\
Polar angle to the pole, end point (abs)
\end{tabular} & \begin{tabular}{l}
mm \\
Degrees
\end{tabular} \\
\hline \multicolumn{2}{|l|}{Additional commands} & \begin{tabular}{l}
You can program feedrates and M commands, for example, using additional G code commands. However, carefully ensure that the additional commands do not collide with the generated G code of the contour and are compatible with the machining type required. Therefore, do not use any \(G\) code commands of group 1 (G0, G1, G2, G3), no coordinates in the plane and no G code commands that have to be programmed in a separate block. \\
The contour is finished in continuous-path mode (G64). As a result, contour transitions such as corners, chamfers or radii may not be machined precisely. If you wish to avoid this, then it is possible to use additional commands when programming. \\
Example: For a contour, first program the straight X parallel and then enter "G9" (non-modal exact stop) for the additional command parameter. Then program the Y -parallel straight line. The corner will be machined exactly, as the feedrate at the end of the X -parallel straight line is briefly zero. \\
Note: \\
The additional commands are only effective for path milling!
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.5.4 Creating contour elements}

After you have created a new contour and specified the starting point, you can define the individual elements that make up the contour.
The following contour elements are available for the definition of a contour:
- Straight vertical line
- Straight horizontal line
- Diagonal line
- Circle/arc
- Pole

For each contour element, you must parameterize a separate parameter screen.
The coordinates for a horizontal or vertical line are entered in Cartesian format; however, for the contour elements Diagonal line and Circle/arc you can choose between Cartesian and polar coordinates. If you wish to enter polar coordinates you must first define a pole. If you have already defined a pole for the starting point, you can also refer the polar coordinates to this pole. Therefore, in this case, you do not have to define an additional pole.

\section*{Cylinder surface transformation}

For contours (e.g. slots) on cylinders, lengths are frequently specified in the form of angles. If the "Cylinder surface transformation" function is activated, you can also define on a cylinder the length of contours (in the circumferential direction of the cylinder surface) using angles. This means instead of X, Y and I, J, you enter Xa, Ya and Ia, Ja.

\section*{Machine manufacturer}


Please refer to the machine manufacturer's specifications.

\section*{Parameter input}

Parameter entry is supported by various help screens that explain the parameters.
If you leave certain fields blank, the geometry processor assumes that the values are unknown and attempts to calculate them from other parameters.

Conflicts may result if you enter more parameters than are absolutely necessary for a contour. In such a case, try to enter fewer parameters and allow the geometry processor to calculate as many parameters as possible.

\section*{Contour transition elements}

As a transition between two contour elements, you can choose a radius or a chamfer. The transition element is always attached at the end of a contour element. The contour transition element is selected in the parameter screen of the respective contour element.

You can use a contour transition element whenever there is an intersection between two successive elements which can be calculated from the input values. Otherwise you must use the straight/circle contour elements.

The contour end is an exception. Although there is no intersection to another element, you can still define a radius or a chamfer as a transition element for the blank.

\section*{Additional functions}

The following additional functions are available for programming a contour:
- Tangent to preceding element

You can program the transition to the preceding element as tangent.
- Dialog box selection

If two different possible contours result from the parameters entered thus far, one of the options must be selected.
- Close contour

From the actual position, you can close the contour with a straight line to the starting point.

\section*{Procedure for entering or changing contour elements}
1. The part program or ShopTurn program to be executed is created.

2. Select the file type (MPF or SPF), enter the desired name of the program and press the "OK" softkey or the "Input" key.
This editor is opened.
3. Select a contour element via softkey.

The input window "Straight (e.g. X)" opens.
- OR

The input window "Straight (e.g. Y)" opens.
- OR

The input window "Straight (e.g. XY)" opens.
- OR


The "Circle" input window opens.
- OR

The "Pole Input" input window opens.

\section*{Pole}
4. Enter all the data available from the workpiece drawing in the input
screen (e.g. length of straight line, target position, transition to next element, angle of lead, etc.).
5. Press the "Accept" softkey.

The contour element is added to the contour.
6. When entering data for a contour element, you can program the transition to the preceding element as a tangent.
Press the "Tangent to prec. elem." softkey. The angle to the preceding element \(\alpha 2\) is set to \(0^{\circ}\). The "tangential" selection appears in the parameter input field.
7. Repeat the procedure until the contour is complete.
8. Press the "Accept" softkey.

The programmed contour is transferred to the machining plan (program

All parameters
Tangent. trans.
view).
9. If you want to display further parameters for certain contour elements, e.g. to enter additional commands, press the "All parameters" softkey.

\section*{Contour element "Straight line, e.g. X"}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline XU & End point X (abs or inc) & mm \\
\hline a1 & Starting angle e.g. to the X axis & Degrees \\
\hline a2 & Angle to the preceding element & Degrees \\
\hline Transition to next element \(U\) & \begin{tabular}{l}
Type of transition \\
- Radius \\
- Chamfer
\end{tabular} & \\
\hline Radius & \(\mathrm{R} \quad\) Transition to following element - radius & mm \\
\hline Chamfer & FS \(\quad\) Transition to following element - chamfer & mm \\
\hline Additional commands & Additional G code commands & \\
\hline
\end{tabular}

\subsection*{9.5 Contour milling}

\section*{Contour element "straight line, e.g. Y "}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face \(Y\) \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline YO & End point Y (abs or inc) & mm \\
\hline a1 & Starting angle to X axis & Degrees \\
\hline Transition to next element \(U\) & \begin{tabular}{l}
Type of transition \\
- Radius \\
- Chamfer
\end{tabular} & \\
\hline Radius & R Transition to following element - radius & mm \\
\hline Chamfer & FS Transition to following element - chamfer & mm \\
\hline Additional commands & Additional G code commands & \\
\hline
\end{tabular}

\section*{Contour element "Straight line e.g. XY"}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline XU & End point X (abs or inc) & mm \\
\hline YU & End point Y (abs or inc) & mm \\
\hline L & Length & mm \\
\hline a1 & Starting angle e.g. to the X axis & Degrees \\
\hline 人2 & Angle to the preceding element & Degrees \\
\hline Transition to next element \(U\) & \begin{tabular}{l}
Type of transition \\
- Radius \\
- Chamfer
\end{tabular} & \\
\hline Radius & R \(\quad\) Transition to following element - radius & mm \\
\hline Chamfer & FS Transition to following element - chamfer & mm \\
\hline Additional commands & Additional G code commands & \\
\hline
\end{tabular}

\section*{Contour element "Circle"}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l}
Machining surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face \(Y\) \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline Direction of rotation & \begin{tabular}{l}
- Clockwise direction of rotation \\
- Counterclockwise direction of rotation
\end{tabular} & \\
\hline R & Radius & mm \\
\hline e.g. XU & End point X (abs or inc) & mm \\
\hline e.g. Y U & End point Y (abs or inc) & mm \\
\hline e.g. IU & Circle center point I (abs or inc) & mm \\
\hline e.g. J U & Circle center point J (abs or inc) & mm \\
\hline a1 & Starting angle to X axis & Degrees \\
\hline a2 & Angle to the preceding element & Degrees \\
\hline \(\beta 1\) & End angle to Z axis & Degrees \\
\hline \(\beta 2\) & Opening angle & Degrees \\
\hline Transition to next element \(U\) & \begin{tabular}{l}
Type of transition \\
- Radius \\
- Chamfer
\end{tabular} & \\
\hline Radius & R \(\quad\) Transition to following element - radius & mm \\
\hline Chamfer & FS Transition to following element - chamfer & mm \\
\hline Additional commands & Additional G code commands & \\
\hline
\end{tabular}

\section*{Contour element "Pole"}
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline Machining & \(\bullet\) Face C & ( Face Y \\
surface & - Face B \\
(only for ShopTurn) & - Peripheral surface C \\
& - Peripheral surface Y & \\
\hline\(X\) & Position pole (abs) \\
Position pole (abs) & mm (in) \\
\hline
\end{tabular}

\section*{Contour element "End"}

The data for the transition at the contour end of the previous contour element is displayed in the "End" parameter screen.

The values cannot be edited.

\subsection*{9.5.5 Changing the contour}

\section*{Function}

You can change a previously created contour later.
If you want to create a contour that is similar to an existing contour, you can copy the existing one, rename it and just alter selected contour elements.
Individual contour elements can be
- added,
- changed,
- inserted or
- deleted.

\section*{Procedure for changing a contour element}
1. Open the part program or ShopTurn program to be executed.
2. With the cursor, select the program block where you want to change the contour. Open the geometry processor. The individual contour elements are listed.
3. Position the cursor at the position where a contour element is to be inserted or changed.
4. Select the desired contour element with the cursor.
5. Enter the parameters in the input screen or delete the element and select a new element.
6. Press the "Accept" softkey. The desired contour element is inserted in the contour or changed.

\section*{Procedure for deleting a contour element}

\section*{Delete} element
1. Open the part program or ShopTurn program to be executed.
2. Position the cursor on the contour element that you want to delete.
3. Press the "Delete element" softkey.
4. Press the "Delete" softkey.

\subsection*{9.5.6 Contour call (CYCLE62) - only for \(G\) code program}

\section*{Function}

The input creates a reference to the selected contour.
There are four ways to call the contour:
1. Contour name

The contour is in the calling main program.
2. Labels

The contour is in the calling main program and is limited by the labels that have been entered.
3. Subprogram

The contour is located in a subprogram in the same workpiece.
4. Labels in the subprogram

The contour is in a subprogram and is limited by the labels that have been entered.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

\section*{Cont.}
mill.

\section*{Contour}

Contour call
2. Press the "Milling" and "Contour milling" softkeys.
3. Press the "Contour" and "Contour call" softkeys.

The "Contour Call" input window opens.
4. Assign parameters to the contour selection.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l} 
Contour selection \\
\(\boldsymbol{U}\)
\end{tabular} & \begin{tabular}{l} 
- Contour name \\
- Labels \\
- Subprogram \\
- Labels in the subprogram
\end{tabular} & \\
\hline Contour name & CON: Contour name & \\
\hline Labels & \begin{tabular}{l} 
- LAB1: Label 1 \\
- LAB2: Label 2
\end{tabular} & \\
\hline Subprogram & PRG: Subprogram & \\
\hline \begin{tabular}{l} 
Labels in the subpro- \\
gram
\end{tabular} & \begin{tabular}{l} 
- PRG: Subprogram \\
- LAB1: Label 1 \\
- LAB2: Label 2
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.5.7 Path milling (CYCLE72)}

\section*{Function}

You can machine open or closed contours with the "Path milling" cycle. Before you can mill the contour, you must enter the contour. Machining can be performed in either direction, i.e. in the direction of the programmed contour or in the opposite direction.
For machining in the opposite direction, contours must not consist of more than 170 contour elements (incl. chamfers/radii). Special aspects (except for feed values) of free G code input are ignored during path milling in the opposite direction to the contour.

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Programming of arbitrary contours}

The machining of arbitrary open or closed contours is generally programmed as follows:
1. Enter contour

You build up the contour gradually from a series of different contour elements.
Define the contour in a subprogram or in the machining program, e.g. after the end of program (M02 or M30).
2. Contour call (CYCLE62)

You select the contour to be machined.
3. Path milling (roughing)

The contour is machined taking into account various approach and retract strategies.
4. Path milling (finishing)

If you programmed a finishing allowance for roughing, the contour is machined again.
5. Path milling (chamfering)

If you have planned edge breaking, chamfer the workpiece with a special tool.

\section*{Path milling on right or left of the contour}

A programmed contour can be machined with the cutter radius compensation to the right or left. You can also select various modes and strategies of approach and retraction from the contour.

\section*{Approach/retraction mode}

The tool can approach or retract from the contour along a quadrant, semi-circle or straight line.
- With a quadrant or semi-circle, you must specify the radius of the cutter center point path.
- With a straight line, you must specify the distance between the cutter outer edge and the contour starting or end point.

You can also program a mixture of modes, e.g. approach along quadrant, retract along semicircle.

\section*{Approach/retraction strategy}

You can choose between planar approach/retraction and spatial approach/retraction:
- Planar approach:

Approach is first at depth and then in the machining plane.
- Spatial approach:

Approach is at depth and in machining plane simultaneously.
- Retraction is performed in reverse order.

Mixed programming is possible, for example, approach in the machining plane, retract spatially.

\section*{Path milling along center-point path.}

A programmed contour can also be machined along the center-point path if the radius correction was switched-out. In this case, approaching and retraction is only possible along a straight line or vertical. Vertical approach/retraction can be used for closed contours, for example.

\section*{Machining type}

You can select the machining mode (roughing, finishing, or chamfer) for path milling. If you want to "rough" and then "finish", you have to call the machining cycle twice (Block 1 = roughing, Block \(2=\) finishing). The programmed parameters are retained when the cycle is called for the second time.
It is also possible to choose between machining the contour with a cutter radius offset or traversing on the center-point path.

\section*{Slot side compensation}

When you mill a contour on the peripheral surface (peripheral machining surface \(C\) ), you can work with or without a slot wall compensation.
- Slot side compensation off

ShopTurn creates slots with parallel walls when the tool diameter is equal to the slot width. If the slot width is larger than the tool diameter, the slot walls will not be parallel.
- Slot side compensation on

ShopTurn creates slots with parallel walls also when the slot width is larger than the tool diameter. If you want to work with a slot wall compensation, you must not program the contour of the slot, but instead the imagined center path of a bolt inserted in the slot whereby the bolt touches both walls. Parameter D is used to specify the slot width.

\section*{Procedure}
\(\left.\begin{array}{|c|}\hline \text { Cont. } \\
\text { mill. }\end{array}\right]\)\begin{tabular}{c} 
Path \\
milling
\end{tabular}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling" softkey.
3. Press the "Contour milling" and "Path milling" softkeys.

The "Path Milling" input window opens.
\begin{tabular}{|c|c|c|c|c|c|}
\hline \multicolumn{3}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \[
\begin{gathered}
\mathrm{PL} \\
\bar{U}
\end{gathered}
\] & Machining plane & & T & Tool name & \\
\hline RP & Retraction plane & mm & D & Cutting edge number & \\
\hline SC & Safety clearance & mm & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} /\) min mm/tooth \\
\hline F & Feedrate & * & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Peripheral surface \(C\) \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Position \\
0 \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- At the front (face) \\
- At the rear (face) \\
- Outside (peripheral surface) \\
- Inside (peripheral surface)
\end{tabular} & \\
\hline \begin{tabular}{l}
D \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} /\) peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) (finishing) \\
- Chamfering
\end{tabular} & \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining direction 0 & \begin{tabular}{l}
Machining in the programmed contour direction \\
- Forward: \\
Machining is performed in the programmed contour direction \\
- Backward: \\
Machining is performed in the opposite direction to the programmed contour
\end{tabular} & \\
\hline Radius compensation U & \begin{tabular}{l}
- Left (machining to the left of the contour) \\
- Right (machining to the right of the contour) \\
- off
\(\square\) \\
A programmed contour can also be machined on the center-point path. In this case, approaching and retraction is only possible along a straight line or vertical. Vertical approach/retraction can be used for closed contours, for example.
\end{tabular} & \\
\hline \begin{tabular}{l}
Slot side compensation \\
O \\
(only ShopTurn)
\end{tabular} & Slot side compensation on or off (only for machining surface, peripheral surface C) & \\
\hline D & \begin{tabular}{l}
Offset to programmed path \\
- (only for slot side compensation on)
\end{tabular} & \\
\hline CP & \begin{tabular}{l}
Positioning angle for machining area \\
- (only for ShopTurn, machining surface, face Y )
\end{tabular} & Degrees \\
\hline C0 & \begin{tabular}{l}
Positioning angle for machining surface \\
- (only for ShopTurn, machining surface, peripheral surface \(Y\) )
\end{tabular} & Degrees \\
\hline Z0 & Reference point \(Z\) & mm \\
\hline Z1 & Final drilling depth (abs) or final drilling depth referred to Z0 or X0 (inc) & mm \\
\hline DZ & Maximum depth infeed - (only for machining \(\nabla\) and \(\nabla \nabla \nabla\) ) & mm \\
\hline UZ & Depth finishing allowance - (only for machining \(\nabla\) ) & mm \\
\hline UXY & Finishing allowance, plane & mm \\
\hline \begin{tabular}{l}
Approach \\
U
\end{tabular} & \begin{tabular}{l}
Planar approach mode: \\
- Quadrant: \\
Part of a spiral (only with path milling left and right of the contour) \\
- Semi-circle: \\
Part of a spiral (only with path milling left and right of the contour) \\
- Straight line: \\
Slope in space \\
- Perpendicular: \\
Perpendicular to the path (only with path milling on the center-point path)
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Approach strategy \\
U
\end{tabular} & \begin{tabular}{l}
- axis-by-axis - (only for "quadrant, semi-circle or straight line" approach)
\(\square\) \\
- spatial - (only for "quadrant, semi-circle or straight line" approach)
\end{tabular} & \\
\hline R1 & Approach radius - (only for "quadrant or semi-circle" approach) & mm \\
\hline L1 & Approach distance - (only for "straight line" approach) & mm \\
\hline \[
\begin{aligned}
& \text { FZ } \\
& \text { (only for G code) }
\end{aligned}
\] & Depth infeed rate & * \\
\hline FZ (only for ShopTurn) & Depth infeed rate & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline \begin{tabular}{l}
Retraction \\
U
\end{tabular} & \begin{tabular}{l}
Planar retraction mode: \\
- Quadrant: \\
Part of a spiral (only with path milling left and right of the contour) \\
- Semi-circle: \\
Part of a spiral (only with path milling left and right of the contour) \\
- Straight line:
\end{tabular} & \\
\hline Retraction strategy & \begin{tabular}{l}
- axis-by-axis \\
- spatial
\end{tabular} & \\
\hline R2 & Retraction radius - (only for "quadrant or semi-circle" retraction) & mm \\
\hline L2 & Retraction distance - (only for "straight line" retraction) & mm \\
\hline Lift mode & \begin{tabular}{l}
If more than one depth infeed is necessary, specify the retraction height to which the tool retracts between the individual infeeds (at the transition from the end of the contour to the start). \\
Lift mode before new infeed \\
- No retraction \\
- to RP \\
- Z0 + safety clearance \\
- By the safety clearance
\end{tabular} & \\
\hline FS & Chamfer width for chamfering - (only for chamfering machining) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering machining only) & mm \\
\hline
\end{tabular}

Turning

\subsection*{9.5.8 Contour pocket/contour spigot (CYCLE63/64)}

\section*{Contours for pockets or islands}

Contours for pockets or islands must be closed, i.e. the starting point and end point of the contour are identical. You can also mill pockets that contain one or more islands. The islands can also be located partially outside the pocket or overlap each other. The first contour you specify is interpreted as the pocket contour and all the others as islands.

\section*{Automatic calculation / manual input of the starting point}

Using "Automatic starting point" you have the option of calculating the optimum plunge point.
By selecting "Manual starting point", you define the plunge point in the parameter screen.
If the islands and the miller diameter, which must be plunged at various locations, are obtained from the pocket contour, then the manual entry only defines the first plunge point; the remaining plunge points are automatically calculated.

\section*{Contours for spigots}

Contours for spigots must be closed, i.e. the starting point and end point of the contour are identical. You can define multiple spigots that can also overlap. The first contour specified is interpreted as a blank contour and all others as spigots.

\section*{Machining}

You program the machining of contour pockets with islands/blank contour with spigots, e.g. as follows:
1. Enter the pocket contour/blank contour
2. Enter the island/spigot contour
3. Call the contour for pocket contour/blank contour or island/spigot contour (only for G code program)
4. Center (this is only possible for pocket contour)
5. Predrill (this is only possible for pocket contour)
6. Solid machine/machine pocket / spigot - roughing
7. Solid machine/machine remaining material - roughing
8. Finishing (base/edge)
9. Chamfering

\section*{Note}

The following error messages can occur when chamfering inside contours:

\section*{Safety clearance in the program header too large}

This error message appears when chamfering would, in principle, be possible with the parameters entered for FS and ZFS, but the safety clearance then could not be maintained.
Immersion depth too large
This error message appears when chamfering would be possible through the reduction of the immersion depth ZFS.

\section*{Tool diameter too large}

This error message appears when the tool would already damage the edges during insertion. In this case, the chamfer FS must be reduced.

Software option
For solid machining residual material, you require the option "residual material detection and machining".

\section*{Name convention}

For multi-channel systems, cycles attach a "_C" and a two-digit number of the specific channel to the names of the programs to be generated, e.g. for channel 1 "_C01". This is the reason that the name of the main program must not end with "_C" and a two-digit number. This is monitored by the cycles.

For single-channel systems, cycles do not extend the name of the programs to be generated.

\section*{Note}

\section*{G code programs}

For \(G\) code programs, the programs to be generated, which do not include any path data, are saved in the directory in which the main program is located. In this case, it must be ensured that programs, which already exist in the directory and which have the same name as the programs to be generated, are overwritten.

\subsection*{9.5.9 Predrilling contour pocket (CYCLE64)}

\section*{Function}

In addition to predrilling, the cycle can be used for centering. The centering or predrilling program generated by the cycle is called for this purpose.
To prevent the drill slipping during drilling, you can center it first.
Before you predrill the pocket, you must enter the pocket contour. If you want to center before predrilling, you have to program the two machining steps in separate blocks.

The number and positions of the necessary predrilled holes depend on the specific circumstances (such as shape of contour, tool, plane infeed, finishing allowance) and are calculated by the cycle.

If you mill several pockets and want to avoid unnecessary tool changes, predrill all the pockets first and then remove the stock.
In this case, for centering/predrilling, you also have to enter the parameters that appear when you press the "All parameters" softkey. These parameters must correspond to the parameters from the previous stock removal step. When programming, proceed as follows:
1. Contour pocket 1
2. Centering
3. Contour pocket 2
4. Centering
5. Contour pocket 1
6. Predrilling
7. Contour pocket 2
8. Predrilling
9. Contour pocket 1
10.Stock removal

\section*{11.Contour pocket 2}

\section*{12.Stock removal}

If you are doing all the machining for the pocket at once, i.e. centering, rough-drilling and removing stock directly in sequence, and do not set the additional parameters for centering/rough-drilling, the cycle will take these parameter values from the stock removal (roughing) machining step. When programming in \(G\) code, these values must be specifically re-entered.

\section*{Note}

\section*{Execution from external media}

If you execute programs from an external drive (e.g. local drive or network drive) then you require the execution from external storage function (EES).

For additional information, please refer to the following references:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Procedure when centering}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Mill contour", "Predrilling" and "Centering" softkeys. The "Centering" input window opens.
```

    Cont.
    mill.
    Rough
    drill
    ```
Centering

\subsection*{9.5 Contour milling}
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline PRG & \multicolumn{3}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline \[
\begin{gathered}
\mathrm{PL} \\
\mathrm{U}
\end{gathered}
\] & \multicolumn{2}{|l|}{Machining plane} & & D & Cutting edge number & \\
\hline \multicolumn{2}{|l|}{Milling direction \(\cup\)} & \begin{tabular}{l}
- Climbing \\
- Conventional
\end{tabular} & & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{U}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline SC & \multicolumn{2}{|l|}{Safety clearance} & mm & & & \\
\hline F & \multicolumn{2}{|l|}{Feedrate} & \(\mathrm{mm} / \mathrm{min}\) & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline TR & Reference tool Tool, which is used in the "stock removal" machining step. This is used to determine the plunge position. & \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- End face Y (only when Y axis exists) \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y (only when Y axis exists)
\end{tabular} & \\
\hline \begin{tabular}{l}
C \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(Y / B\) and peripheral surface \(Y\) ) The function must be set up by the machine manufacturer. & \\
\hline Z0 & Reference point in the tool axis \(Z\) & mm \\
\hline Z1 & Pocket depth \(\varnothing\) (abs) or depth referred to Z0 & mm \\
\hline CP & \begin{tabular}{l}
Positioning angle for machining area \\
- (only for ShopTurn, machining surface, face Y)
\end{tabular} & Degrees \\
\hline C0 & \begin{tabular}{l}
Positioning angle for machining surface \\
- (only for ShopTurn, machining surface, peripheral surface Y)
\end{tabular} & Degrees \\
\hline DXY & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline UXY & Finishing allowance, plane & mm \\
\hline Lift mode 0 & \begin{tabular}{l}
Lift mode before new infeed \\
If the machining operation requires several points of insertion, the retraction height can be programmed: \\
- To retraction plane \\
- Z0 + safety clearance \\
When making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than Z0 in the pocket area, Z0 + safety clearance" can be selected as the lift mode.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline
\end{tabular}

\section*{Predrilling procedure}

\section*{Cont.}
mill.

\section*{Rough} drill

\section*{Rough}

\section*{drill}
2. Press the "Milling", "Contour milling", "Predrilling" and "Predrilling" soft-
keys.
The "Predrilling" input window opens.
1. The part program or ShopTurn program to be processed has been created and you are in the editor.

路

\subsection*{9.5 Contour milling}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline CP & \begin{tabular}{l}
Positioning angle for machining area \\
- (only for ShopTurn, machining surface, face Y)
\end{tabular} & Degrees \\
\hline C0 & \begin{tabular}{l}
Positioning angle for machining surface \\
- (only for ShopTurn, machining surface, peripheral surface Y)
\end{tabular} & Degrees \\
\hline DXY & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline UXY & Finishing allowance, plane & mm \\
\hline UZ & Finishing allowance, depth & mm \\
\hline Lift mode U & \begin{tabular}{l}
Lift mode before new infeed \\
If the machining operation requires several points of insertion, the retraction height can be programmed: \\
- To retraction plane \\
- Z0 + safety clearance \\
When making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than Z0 (X0) in the pocket area, then Z0 (X0) + safety clearance can be programmed as the lift mode.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline
\end{tabular}

\subsection*{9.5.10 Milling contour pocket (CYCLE63)}

\section*{Function}

You can use the "Mill pocket" function to mill a pocket on the face or peripheral surface.
Before you remove stock from the pocket, you must first enter the contour of the pocket and, if applicable, the contour of an island. Stock is removed from the pocket parallel to the contour from the inside to the outside. The direction is determined by the machining direction (climbing or conventional). If an island is located in the pocket, the cycle automatically takes this into account during stock removal.

\section*{Note}

\section*{Execution from external media}

If you execute programs from an external drive (e.g. local drive or network drive) then you require the execution from external storage function (EES).

For additional information, please refer to the following references:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Machining type}

For solid machining, you can select the machining type (roughing or finishing). If you want to rough and then finish, you have to call the machining cycle twice (block \(1=\) roughing, block 2 \(=\) finishing). The programmed parameters are retained when the cycle is called for the second time.

During insertion with oscillation, the message "Ramp path too short" will appear if the tool is less than the milling cutter diameter away from the insertion point along the ramp, or the machining depth is not reached.
- Reduce the insertion angle if the tool remains too close to the insertion point.
- Increase the insertion angle if the tool does not reach the machining depth.
- If necessary, use a tool with a smaller radius of select a different insertion mode.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Pocket" softkeys. The "Mill pocket" input window opens.

\subsection*{9.5 Contour milling}

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline Input & \multicolumn{6}{|c|}{- Complete} \\
\hline PRG & \multicolumn{3}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline \[
\begin{gathered}
\mathrm{PL} \\
\mathrm{U}
\end{gathered}
\] & \multicolumn{2}{|l|}{Machining plane} & & D & Cutting edge number & \\
\hline \multicolumn{2}{|l|}{Milling direction \(\cup\)} & \begin{tabular}{l}
- Climbing \\
- Conventional milling
\end{tabular} & & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline SC & \multicolumn{2}{|l|}{Safety clearance} & mm & & & \\
\hline F & \multicolumn{2}{|l|}{Feedrate} & \(\mathrm{mm} / \mathrm{min}\) & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Cl \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} / \mathrm{B}\) and peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) base (base finishing) \\
- \(\nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline Z0 & Reference point in the tool axis Z & mm \\
\hline Z1 & Pocket depth (abs) or depth referred to Z0 & mm \\
\hline CP & \begin{tabular}{l}
Positioning angle for machining area \\
- (only for ShopTurn, machining surface, face Y)
\end{tabular} & Degrees \\
\hline C0 & \begin{tabular}{l}
Positioning angle for machining surface \\
- (only for ShopTurn, machining surface, peripheral surface Y)
\end{tabular} & Degrees \\
\hline DXY & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed & mm \\
\hline UXY & Finishing allowance, plane & mm \\
\hline UZ & Finishing allowance, depth & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Starting point & \begin{tabular}{l}
- Manual \\
Starting point is entered \\
- Automatic \\
Starting point is automatically calculated
\end{tabular} & \\
\hline XS & Starting point X - (only for "manual" starting point) & mm \\
\hline YS & Starting point X - (only for "manual" starting point) & mm \\
\hline Insertion U & \begin{tabular}{l}
The following insertion modes can be selected - (only for \(\nabla, \nabla \nabla \nabla\) base or \(\nabla \nabla \nabla\) edge): \\
- Vertical insertion \\
The calculated actual infeed depth is executed at the calculated position for "automatic" starting point - or at the specified position for "manual" starting point. \\
- Note \\
This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. \\
- Helical insertion Insertion along a helical path. \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. \\
- Oscillating insertion \\
Oscillating insertion at the center axis of the rectangular pocket. \\
The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.
\end{tabular} & \\
\hline FZ U (only for ShopTurn) & Depth infeed rate - (for vertical insertion only) & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline FZ (only for G code) & Depth infeed rate - (for vertical insertion only) & mm/min \\
\hline EP & Maximum pitch of helix - (for helical insertion only) & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline ER & \begin{tabular}{l}
Radius of helix - (for helical insertion only) \\
The radius cannot be any larger than the milling cutter radius; otherwise, material will remain.
\end{tabular} & mm \\
\hline EW & \begin{tabular}{l}
Note: \\
During insertion with oscillation, the message "Ramp path too short" will appear if the tool is less than the milling cutter diameter away from the insertion point along the ramp. If this occurs, please reduce the angle of insertion.
\end{tabular} & Degrees \\
\hline
\end{tabular}

\subsection*{9.5 Contour milling}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Lift mode \\
O
\end{tabular} & \begin{tabular}{l}
Lift mode before new infeed \\
If the machining operation requires several points of insertion, the retraction height can be programmed: \\
- To retraction plane \\
- Z0 + safety clearance \\
When making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than \(Z 0\) (X0) in the pocket area, then Z0 (X0) + safety clearance can be programmed as the lift mode.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline FS & Chamfer width for chamfering - (only for chamfering machining) & mm \\
\hline ZFS O & Insertion depth of tool tip (abs or inc) - (for chamfering machining only) & mm \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{3}{|l|}{\begin{tabular}{l}
Input \\
U
\end{tabular}} & \multicolumn{4}{|l|}{- simple} \\
\hline PRG & \multicolumn{3}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline \multicolumn{2}{|l|}{Milling direction \(\cup\)} & \begin{tabular}{l}
- Climbing \\
- Conventional
\end{tabular} & & D & Cutting edge number & \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & F & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline F & Feedrate & & * & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline & & & & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l}
Machining surface \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face \(Y\) \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline  & Clamp/release spindle (only for end face \(\mathrm{Y} / \mathrm{B}\) and peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline Machining U & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) base (base finishing) \\
- \(\nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline Z0 & Reference point in the tool axis Z & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline Z1 & Pocket depth (abs) or depth referred to Z0 (inc) & mm \\
\hline \begin{tabular}{l}
CP \\
(only for ShopTurn)
\end{tabular} & Positioning angle for machining area - (only for machining surface, face Y ) & Degrees \\
\hline C0 (only for ShopTurn) & Positioning angle for machining area - (only for machining surface, peripheral surface Y) & Degrees \\
\hline DXY U & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed & mm \\
\hline UXY & Finishing allowance, plane & mm \\
\hline UZ & Finishing allowance, depth & mm \\
\hline Insertion & \begin{tabular}{l}
The following insertion modes can be selected - (only for \(\nabla, \nabla \nabla \nabla\) base or \(\nabla \nabla \nabla\) edge): \\
- Vertical \\
The calculated actual infeed depth is executed at the calculated position for "automatic" starting point - or at the specified position for "manual" starting point. \\
Note: \\
This setting can be used only if the cutter can cut across center or if the pocket has been predrilled. \\
- Helical \\
The cutter center point traverses along the helical path determined by the radius and depth per revolution (helical path). If the depth for one infeed has been reached, a full circle motion is executed to eliminate the inclined insertion path. \\
- Oscillation \\
The cutter center point oscillates back and forth along a linear path until it reaches the depth infeed. When the depth has been reached, the path is traversed again without depth infeed in order to eliminate the inclined insertion path.
\end{tabular} & \\
\hline FZ U (only for ShopTurn) & Depth infeed rate - (only for vertical insertion and \(\nabla\) ) & \(\mathrm{mm} / \mathrm{min}\) mm/tooth \\
\hline FZ (only for G code) & Depth infeed rate - (only for vertical insertion and \(\nabla\) ) & * \\
\hline EP & Maximum pitch of helix - (for helical insertion only) & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline ER & \begin{tabular}{l}
Radius of helix - (for helical insertion only) \\
The radius cannot be any larger than the milling cutter radius; otherwise, material will remain.
\end{tabular} & mm \\
\hline EW & \begin{tabular}{l}
Note: \\
During insertion with oscillation, the message "Ramp path too short" will appear if the tool is less than the milling cutter diameter away from the insertion point along the ramp. If this occurs, please reduce the angle of insertion.
\end{tabular} & Degrees \\
\hline FS & Chamfer width for chamfering (inc) - (for chamfering only) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

Turning

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline Starting point & \begin{tabular}{l} 
Starting point is automatically calculated - (only for \(\nabla\) and \\
\(\nabla \nabla \nabla\) base)
\end{tabular} & Automatic & \\
\hline Lift mode & \begin{tabular}{l} 
Lift mode before new infeed - (only for \(\nabla, \nabla \nabla \nabla\) base or \(\nabla \nabla \nabla\) \\
edge)
\end{tabular} & to RP & \\
\hline
\end{tabular}


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.5.11 Contour pocket residual material (CYCLE63, option)}

\section*{Function}

When you have removed stock from a pocket (with/without islands) and there is residual material, then this is automatically detected. You can use a suitable tool to remove this residual material without having to machine the whole pocket again, i.e. avoiding unnecessary non-productive motion. Material that remains as part of the finishing allowance is not residual material.

The residual material is calculated on the basis of the milling cutter used for stock removal.
If you mill several pockets and want to avoid unnecessary tool changeover, remove stock from all the pockets first and then remove the residual material. In this case, for removing the residual material, you also have to enter a value for the reference tool TR parameter, which, for the ShopTurn program, additionally appears when you press the "All parameters" softkey. When programming, proceed as follows:
1. Contour pocket 1
2. Stock removal
3. Contour pocket 2
4. Stock removal
5. Contour pocket 1
6. Removing residual stock
7. Contour pocket 2
8. Removing residual stock

\section*{Software option}

For removing residual stock, you require the option "residual stock detection and machining".

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Pocket resid. mat." softkeys.

The "Pocket Res. Mat." input window opens.

Pocket
res.mat.
All parameters
3. For the ShopTurn program, press the "All parameters" softkey if you want to enter additional parameters.
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline PRG & \multicolumn{3}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline \[
\begin{gathered}
\mathrm{PL} \\
\mathrm{U}
\end{gathered}
\] & \multicolumn{2}{|l|}{Machining plane} & & D & Cutting edge number & \\
\hline \multicolumn{2}{|l|}{Milling direction \(\cup\)} & \begin{tabular}{l}
- Climbing \\
- Conventional
\end{tabular} & & F & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline SC & \multicolumn{2}{|l|}{Safety clearance} & mm & & & \\
\hline F & \multicolumn{2}{|l|}{Feedrate} & \(\mathrm{mm} /\) min & & & \\
\hline
\end{tabular}

\subsection*{9.5 Contour milling}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face \(Y\) \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
a \\
(0) \\
(only for Shop- \\
Turn)
\end{tabular} & Clamp/release spindle (only for end face Y/B and peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline TR & Reference tool Tool, which is used in the "stock removal" machining step. This is used to determine the residual corners. & \\
\hline D U & Cutting edge number & \\
\hline Z0 & Reference point in the tool axis Z & mm \\
\hline Z1 & Pocket depth (abs) or depth referred to Z0 or X0 (inc) & mm \\
\hline CP & \begin{tabular}{l}
Positioning angle for machining area \\
- (only for ShopTurn, machining surface, face Y )
\end{tabular} & Degrees \\
\hline C0 & \begin{tabular}{l}
Positioning angle for machining surface \\
- (only for ShopTurn, machining surface, peripheral surface Y)
\end{tabular} & Degrees \\
\hline DXY & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter
\end{tabular} & \[
\begin{aligned}
& \text { mm } \\
& \%
\end{aligned}
\] \\
\hline DZ & Maximum depth infeed & \\
\hline \begin{tabular}{l}
Lift mode \\
0
\end{tabular} & \begin{tabular}{l}
Lift mode before new infeed \\
If the machining operation requires several points of insertion, the retraction height can be programmed: \\
- To retraction plane \\
- ZO + safety clearance \\
When making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than \(\mathrm{ZO}(\mathrm{XO})\) in the pocket area, then \(\mathrm{ZO}(\mathrm{XO})\) + safety clearance can be programmed as the lift mode.
\end{tabular} & \begin{tabular}{l}
mm \\
mm
\end{tabular} \\
\hline
\end{tabular}

\subsection*{9.5.12 Milling contour spigot (CYCLE63)}

\section*{Function}

You can use the "Mill spigot" function to mill any spigots on the face or peripheral surface.
Before you mill the spigot, you must first enter a blank contour and then one or more spigot contours. The blank contour defines the area, outside of which there is no material, i.e. there, the tool moves with rapid traverse. Material is then removed between the blank contour and spigot contour.

\section*{Note}

\section*{Execution from external media}

If you execute programs from an external drive (e.g. local drive or network drive) then you require the execution from external storage function (EES).

For additional information, please refer to the following references:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.


\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Machining type}

You can select the machining type (roughing, base finishing, edge finishing, chamfer) for milling. If you want to rough and then finish, you have to call the machining cycle twice (block \(1=\) roughing, block \(2=\) finishing). The programmed parameters are retained when the cycle is called for the second time.

\section*{Approach/retraction}
1. The tool approaches the starting point at rapid traverse at the height of the retraction plane and is fed in to the safety clearance. The cycle calculates the starting point.
2. The tool first infeeds to the machining depth and then approaches the spigot contour from the side in a quadrant at machining feedrate.
3. The spigot is machined in parallel with the contours from the outside in. The direction is determined by the machining direction (climb/conventional) (see "Changing program settings").
4. When the first plane of the spigot has been machined, the tool retracts from the contour in a quadrant and then infeeds to the next machining depth.
5. The spigot is again approached in a quadrant and machine in parallel with the contours from outside in.
6. Steps 4 and 5 are repeated until the programmed spigot depth is reached.
7. The tool retracts to the safety clearance at rapid traverse.

\section*{Procedure}
- Mill.

Cont.
mill.
Multi- edge spigot
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Spigot" softkeys. The "Mill spigot" input window opens.
3. Select the "Roughing" machining type.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline \multicolumn{4}{|l|}{Input - Complete} & & & \\
\hline PRG & \multicolumn{3}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline \[
\begin{gathered}
\text { PL } \\
U
\end{gathered}
\] & \multicolumn{2}{|l|}{Machining plane} & & D & Cutting edge number & \\
\hline \multicolumn{2}{|l|}{Milling direction \(\mathbf{U}\)} & \begin{tabular}{l}
- Climbing \\
- Conventional milling
\end{tabular} & & F & Feedrate & \(\mathrm{mm} /\) min \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline SC & \multicolumn{2}{|l|}{Safety clearance} & mm & & & \\
\hline F & \multicolumn{2}{|l|}{Feedrate} & \(\mathrm{mm} / \mathrm{min}\) & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining surface U (only for ShopTurn) & \begin{tabular}{l}
- Face C \\
- Face \(Y\) \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
D \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(\mathrm{Y} / \mathrm{B}\) and peripheral surface Y ) The function must be set up by the machine manufacturer. & \\
\hline \begin{tabular}{l}
Machining \\
u
\end{tabular} & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) base (base finishing) \\
- \(\nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline Z0 & Reference point in tool axis Z & mm \\
\hline Z1 & Pocket depth (abs) or depth referred to Z0 or X0 (inc) & mm \\
\hline CP & \begin{tabular}{l}
Positioning angle for machining area \\
- (only for ShopTurn, machining surface, face Y)
\end{tabular} & Degrees \\
\hline C0 & \begin{tabular}{l}
Positioning angle for machining surface \\
- (only for ShopTurn, machining surface, peripheral surface Y )
\end{tabular} & Degrees \\
\hline \[
\begin{aligned}
& U \\
& D X Y
\end{aligned}
\] & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter - (only for \(\nabla\) and \(\nabla \nabla \nabla\) base)
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \%
\end{aligned}
\] \\
\hline DZ & \begin{tabular}{l}
Maximum depth infeed \\
- (only for \(\nabla\) or \(\nabla \nabla \nabla\) edge)
\end{tabular} & mm \\
\hline UXY & \begin{tabular}{l}
Finishing allowance, plane \\
- (only for \(\nabla, \nabla \nabla \nabla\) base or \(\nabla \nabla \nabla\) edge)
\end{tabular} & mm \\
\hline UZ & Finishing allowance, depth - (only for \(\nabla\) or \(\nabla \nabla \nabla\) base) & mm \\
\hline Lift mode U & \begin{tabular}{l}
Lift mode before new infeed \\
If the machining operation requires several points of insertion, the retraction height can be programmed: \\
- To retraction plane \\
- Z0 + safety clearance \\
When making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than Z0 (X0) in the pocket area, then Z0 (X0) + safety clearance can be programmed as the lift mode.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm
\end{tabular} \\
\hline FS & Chamfer width for chamfering - (only for chamfering machining) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering machining only) & mm \\
\hline
\end{tabular}

Turning

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{G code program parameters} & \multicolumn{3}{|l|}{ShopTurn program parameters} \\
\hline \multicolumn{3}{|l|}{Input U} & \multicolumn{4}{|l|}{- simple} \\
\hline PRG & \multicolumn{3}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline \multicolumn{2}{|l|}{Milling direction \(\mathbf{U}\)} & \begin{tabular}{l}
- Climbing \\
- Conventional
\end{tabular} & & D & Cutting edge number & \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & F 0 & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline F & Feedrate & & * & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{U} \\
& \hline
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline & & & & & & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & \\
\hline \begin{tabular}{l}
Machining surface \\
U \\
(only for ShopTurn)
\end{tabular} & \begin{tabular}{l}
- Face C \\
- Face Y \\
- Face B \\
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} & \\
\hline \begin{tabular}{l}
Cl \\
(0) \\
(only for ShopTurn)
\end{tabular} & Clamp/release spindle (only for end face \(Y / B\) and peripheral surface \(Y\) ) The function must be set up by the machine manufacturer. & \\
\hline Machining U & \begin{tabular}{l}
The following machining operations can be selected: \\
- \(\nabla\) (roughing) \\
- \(\nabla \nabla \nabla\) base (base finishing) \\
- \(\nabla \nabla \nabla\) edge (edge finishing) \\
- Chamfering
\end{tabular} & \\
\hline Z0 & Reference point in the tool axis Z & mm \\
\hline Z1 & Pocket depth (abs) or depth referred to Z0 (inc) & mm \\
\hline \begin{tabular}{l}
CP \\
(only for ShopTurn)
\end{tabular} & Positioning angle for machining area - (only for machining surface, face Y ) & Degrees \\
\hline \[
\begin{array}{|l|}
\hline \text { C0 } \\
\text { (only for ShopTurn) } \\
\hline
\end{array}
\] & Positioning angle for machining area - (only for machining surface, peripheral surface Y) & Degrees \\
\hline DXY U & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter- (only for \(\nabla\) and \(\nabla \nabla \nabla\) base)
\end{tabular} & \[
\begin{array}{|l|l}
\mathrm{mm} \\
\%
\end{array}
\] \\
\hline DZ & Maximum depth infeed - (only for \(\nabla\) and \(\nabla \nabla \nabla\) edge) & mm \\
\hline UXY & Plane finishing allowance - (only for \(\nabla, \nabla \nabla \nabla\) base and \(\nabla \nabla \nabla\) edge) & mm \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline UZ & Depth finishing allowance (only for \(\nabla\) and \(\nabla \nabla \nabla\) base) & mm \\
\hline FS & Chamfer width for chamfering - (for chamfering only) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering only) & mm \\
\hline
\end{tabular}
* Unit of feedrate as programmed before the cycle call

\section*{Hidden parameters}

The following parameters are hidden. They are pre-assigned fixed values or values that can be adjusted using setting data.
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline \begin{tabular}{l} 
PL (only for G \\
code)
\end{tabular} & Machining plane & \begin{tabular}{l} 
Defined in MD \\
52005
\end{tabular} & \\
\hline \begin{tabular}{l} 
SC (only for G \\
code)
\end{tabular} & Safety clearance & 1 mm & x \\
\hline Lift mode & \begin{tabular}{l} 
Lift mode before new infeed - (only for \(\nabla, \nabla \nabla \nabla\) base or \(\nabla \nabla \nabla\) \\
edge)
\end{tabular} & to RP & \\
\hline
\end{tabular}

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{9.5.13 Contour spigot residual material (CYCLE63, option)}

\section*{Function}

When you have milled a contour spigot and residual material remains, then this is automatically detected. You can use a suitable tool to remove this residual material without having to machine the whole spigot again, i.e. avoiding unnecessary non-productive motion. Material that remains as part of the finishing allowance is not residual material.

The residual material is calculated on the basis of the milling cutter used for clearing.
If you mill several spigots and want to avoid unnecessary tool changeover, clear all the spigots first and then remove the residual material. In this case, for removing the residual material, you also have to enter a value for the reference tool TR parameter, which, for the ShopTurn program, additionally appears when you press the "All parameters" softkey. When programming, proceed as follows:
1. Contour blank 1
2. Contour spigot 1
3. Clear spigot 1
4. Contour blank 2
5. Contour spigot 2
6. Clear spigot 2
7. Contour blank 1
8. Contour spigot 1
9. Removing residual stock spigot 1
10. Contour blank 2
11.Contour spigot 2
12. Removing residual stock spigot 2


\section*{Software option}

For removing residual stock, you require the option "residual stock detection and machining".

\section*{Clamping the spindle}

For ShopTurn, the "Clamp spindle" function can be set up by the machine manufacturer.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{See also}

Clamping the spindle (Page 247)

\section*{Procedure}

The "Spigot Res. Mat." input window opens.
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Milling", "Contour milling" and "Spigot resid. mat." softkeys.
```

Cont.
mill.

```

\section*{Cont. \\ mill.}
\begin{tabular}{|c|}
\hline Spigot \\
res. mat. \\
\hline All para- \\
\hline meters \\
\hline
\end{tabular}

\section*{Spigot} res. mat.

All parameters
3. For the ShopTurn program, press the "All parameters" softkey if you want to enter additional parameters.
\begin{tabular}{|c|c|c|c|c|c|c|}
\hline \multicolumn{4}{|l|}{Parameters, G code program} & \multicolumn{3}{|l|}{Parameters, ShopTurn program} \\
\hline PRG & \multicolumn{3}{|l|}{Name of the program to be generated} & T & Tool name & \\
\hline \[
\begin{aligned}
& \mathrm{PL} \\
& \mathrm{U}
\end{aligned}
\] & \multicolumn{2}{|l|}{Machining plane} & & D & Cutting edge number & \\
\hline \multicolumn{2}{|l|}{Milling direction \(\cup\)} & \begin{tabular}{l}
- Climbing \\
- Conventional
\end{tabular} & & \[
\begin{aligned}
& \mathrm{F} \\
& \mathrm{O}
\end{aligned}
\] & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{tooth}\) \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane} & mm & \[
\begin{aligned}
& \mathrm{S} / \mathrm{V} \\
& \mathrm{O}
\end{aligned}
\] & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline SC & \multicolumn{2}{|l|}{Safety clearance} & mm & & & \\
\hline F & \multicolumn{2}{|l|}{Feedrate} & \(\mathrm{mm} / \mathrm{min}\) & & & \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline parameters & Description & Unit \\
\hline Machining & \begin{tabular}{l} 
The following machining technologies can be selected: \\
- \(\nabla\) (roughing)
\end{tabular} & \\
\hline \begin{tabular}{l} 
Machining \\
surface \\
U (only for ShopTurn)
\end{tabular} & \begin{tabular}{ll} 
- Face C \\
- Face Y \\
- Face B
\end{tabular} & \begin{tabular}{l} 
- Peripheral surface C \\
- Peripheral surface Y
\end{tabular} \\
(O) & \begin{tabular}{l} 
Clamp/release spindle (only for end face Y/B and peripheral surface Y) \\
The function must be set up by the machine manufacturer.
\end{tabular} & \\
\hline (only for ShopTurn) & & \begin{tabular}{l} 
Reference tool Tool, which is used in the "stock removal" machining step. This \\
is used to determine the residual corners.
\end{tabular} \\
\hline TR & Cutting edge number & \\
\hline D & Reference point in tool axes Z & \\
\hline Z0 & & \\
\hline
\end{tabular}

\subsection*{9.5 Contour milling}
\begin{tabular}{|c|c|c|}
\hline parameters & Description & Unit \\
\hline \[
\begin{aligned}
& \mathrm{O} \\
& \mathrm{Z} 1
\end{aligned}
\] & Pocket depth (abs) or depth referred to Z0 & mm \\
\hline CP & \begin{tabular}{l}
Positioning angle for machining area \\
- (only for ShopTurn, machining surface, face Y)
\end{tabular} & Degrees \\
\hline C0 & \begin{tabular}{l}
Positioning angle for machining surface \\
- (only for ShopTurn, machining surface, peripheral surface Y)
\end{tabular} & Degrees \\
\hline \[
\begin{aligned}
& U \\
& D X Y
\end{aligned}
\] & \begin{tabular}{l}
- Maximum plane infeed \\
- Maximum plane infeed as a percentage of the milling cutter diameter
\end{tabular} & \[
\begin{array}{|l}
\mathrm{mm} \\
\%
\end{array}
\] \\
\hline DZ & Maximum depth infeed & \\
\hline Lift mode 0 & \begin{tabular}{l}
Lift mode before new infeed \\
If the machining operation requires several points of insertion, the retraction height can be programmed: \\
- To retraction plane \\
- Z0 + safety clearance \\
When making the transition to the next insertion point, the tool returns to this height. If there are no elements larger than Z0 in the pocket area, Z0 + safety clearance" can be selected as the lift mode.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline FS & Chamfer width for chamfering - (only for chamfering machining) & mm \\
\hline ZFS U & Insertion depth of tool tip (abs or inc) - (for chamfering machining only) & mm \\
\hline
\end{tabular}

\subsection*{9.6 Further cycles and functions}

\subsection*{9.6. \(\quad\) Swiveling plane / aligning tool (CYCLE800)}

The CYCLE800 swivel cycle is used to swivel to any surface in order to either machine or measure it. In this cycle, the active workpiece zeros and the work offsets are converted to the inclined surface taking into account the kinematic chain of the machine by calling the appropriate NC functions and rotary axes (optionally) are positioned.

Swiveling can be realized:
- axis-by-axis
- via solid angle
- via projection angle
- directly

Before the rotary axes are positioned, the linear axes can be retracted if desired.
Swiveling always means three geometry axes.
In the basic version, the following functions
- \(3+2\) axes, inclined machining and
- Toolholder with orientation capability
are available.

\section*{Setting/aligning tools for a G code program}

The swivel function also includes the "Setting tool", "Align milling tool" and "Align turning tool" functions. When setting and aligning, contrary to swiveling, the coordinate system (WCS) is not rotated at the same time.

\section*{Prerequisites before calling the swivel cycle}

A tool (tool cutting edge \(D>0\) ) and the work offset (WO), with which the workpiece was scratched or measured, must be programmed before the swivel cycle is first called in the main program.

Example:
```

N1 T1D1
N2 M6
N3 G17 G54
N4 CYCLE800(1,"",0,57,0,0,0,0,0,0,0,0,0,1,0,1)) ;swivel ZERO to
;initial position of the
;machine kinematics
;blank declaration for
;simulation and
;simultaneous recording

```

For machines where swivel is set-up, each main program with a swivel should start in the initial position of the machine.

The definition of the blank (WORKPIECE) always refers to the currently effective work offset. For programs that use "swivel", a swivel to zero must be made before the blank is defined. For ShopTurn programs, the blank in the program header is automatically referred to the unswiveled state.

In the swivel cycle, the work offset (WO) as well as the shifts and rotations of the parameters of the CYCLE800 are converted to the corresponding machining plane. The work offset is kept. Shifts and rotations are saved in system frames - the swivel frames (displayed under parameter/work offsets):
- Tool reference (\$P_TOOLFRAME)
- Rotary table reference (\$P_PARTFRAME)
- Workpiece reference (\$P_WPFRAME)

The swivel cycle takes into account the actual machining plane (G17, G18, G19).
Swiveling on a machining or auxiliary surface always involves 3 steps:
- Shifting the WCS before rotation
- Rotating the WCS (axis-by-axis, ...)
- Shifting the WCS after rotation

The shifts and rotations refer to the coordinate system \(\mathrm{X}, \mathrm{Y}, \mathrm{Z}\) of the workpiece and are therefore independent of the machine (with the exception of swivel "rotary axis direct").
No programmable frames are used in the swivel cycle. The frames programmed by the user are taken into account for additive swiveling.
On the other hand, when swiveling to a new swivel plane, the programmable frames are deleted. Any type of machining operation can be performed on the swivel plane, e.g. by calling standard or measuring cycles.

The last swivel plane remains active after a program reset or when the power fails. The behavior at reset and power on can be set using machine data.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Block search when swiveling the plane / swiveling the tool}

For block search with calculation, after NC start, initially, the automatic rotary axes of the active swivel data set are pre-positioned and then the remaining machine axes are positioned. This does not apply if a type TRACYL or TRANSMIT transformation is active after the block search. In this case, all axes simultaneously move to the accumulated positions.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Aligning tools}

The purpose of the "Align turning tool" function is to support turning machines with a swivelmounted \(B\) axis. The position and orientation of the turning tool can be changed by rotating swivel axis \(B\) (around \(Y\) ) and the tool spindle.

In contrast to "Swivel plane", no rotation is operative in the active work offsets in the workpiece coordinate system in the case of "Align tool".

The maximum angular range for "Align milling tool" is limited by the traversing range of the participating rotary axes. Technological limits are also placed on the angular range depending on the tool used.

When aligning the tool, using the CUTMOD NC command, the tool data are calculated online based on the tool orientation (positions of the B axis and the tool spindle). For a turning tool, this involves the cutting edge position, the holder angle and the cut direction.

\section*{Name of swivel data set}

Selecting the swivel data set or deselecting the swivel data set.
The selection can be hidden by the machine data.
For "Swivel plane" and "Swivel tool" / "Set tool", only the swivel data sets are available for selection where no B axis kinematics, turning technology has been set.
"Swivel tool" / "Align tool", only the swivel data sets are available for selection where B axis kinematics, turning technology has been set.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Approaching a machining operation}

When approaching the programmed machining operation in the swiveled plane, under worst case conditions, the software limit switches could be violated. In this case, the system travels along the software limit switches above the retraction plane. In the event of violation below the retraction plane, for safety reasons, the program is interrupted with an alarm. To avoid this, before swiveling, e.g. move the tool in the X/Y plane and position it as close as possible to the starting point of the machining operation or define the retraction plane closer to the workpiece.

\section*{Retraction}

Before swiveling the axes you can move the tool to a safe retraction position. The retraction versions available are defined when starting up the system (commissioning).

The retraction mode is modal. When a tool is changed or after a block search, the retraction mode last set is used.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{WARNING}

Risk of collision
You must select a retraction position that avoids a collision between the tool and workpiece when swiveling.

\section*{Swivel plane (only for G code programming)}
- New

Previous swivel frames and programmed frames are deleted and a new swivel frame is formed according to the values specified in the input screen.
Every main program must begin with a swivel cycle with the new swivel plane, in order to ensure that a swivel frame from another program is not active.

\section*{- Additive}

The swivel frame is added to the swivel frame from the last swivel cycle.
If several swivel cycles are programmed in a program and programmable frames are also active between them (e.g., AROT ATRANS), these are taken into account in the swivel frame.

If the currently active WO contains rotations, e.g., due to previous workpiece measuring operations, they will be taken into account in the swivel cycle.

\section*{Swivel mode}

Swiveling can either be realized axis-by-axis, using the angle in space, using the projection angle or directly. The machine manufacturer determines when setting up the "Swivel plane/swivel tool" function which swivel methods are available.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{- Axis by axis}

In the case of axis-by-axis swiveling, the coordinate system is rotated about each axis in turn, with each rotation starting from the previous rotation. The axis sequence can be freely selected.

\section*{- Solid angle}

With the solid angle swiveling option, the tool is first rotated about the \(Z\) axis and then about the Y axis. The second rotation starts from the first.

\section*{- Projection angle}

When swiveling using the projection angle, the angle value of the swiveled surface is projected onto the first two axes of the right-angle coordinate system. The user can freely select the axis rotation sequence.

The 3rd rotation is based on the previous rotation. The active plane and the tool orientation must be taken into consideration when the projection angle is used:
- For G17 projection angle XY, 3rd rotation around Z
- For G18 projection angle ZX, 3rd rotation around Y
- For G19 projection angle YZ, 3rd rotation around \(X\)

When projection angles around \(X Y\) and \(Y X\) are programmed, the new \(X\)-axis of the swiveled coordinate system lies in the old \(Z X\) plane.

When projection angles around \(X Z\) and \(Z X\) are programmed, the new \(Z\)-axis of the swiveled coordinate system lies in the old \(\mathrm{Y}-\mathrm{Z}\) plane.

When projection angles around \(Y Z\) and \(Z Y\) are programmed, the new \(Y\)-axis of the swiveled coordinate system lies in the old \(X-Y\) plane.
- directly

For direct swiveling, the required positions of the rotary axes are specified. The HMI calculates a suitable new coordinate system based on these values. The tool axis is aligned in the \(Z\) direction. You can derive the resulting direction of the X and Y axis by traversing the axes.

\section*{Note}

\section*{Direction of rotation}

The positive direction of each rotation for the different swivel versions is shown in the help displays.

\section*{Axis sequence}

Sequence of the axes which are rotated around:
\(X Y Z\) or XZY or \(Y X Z\) or \(Y Z X\) or \(Z X Y\) or \(Z Y X\)

\section*{Direction (minus/plus)}

Direction reference of traversing direction of rotary axis 1 or 2 of the active swivel data set (machine kinematics). The NC calculates two possible solutions of the rotation / offset programmed in CYCLE800 using the angle traversing range of the rotary axes of the machine kinematics. Usually, only one of these solutions is technologically suitable. The solutions differ by 180 degrees in each case. Selecting the "minus" or "plus" direction determines which of the two possible solutions is to be applied.
- "Minus" \(\rightarrow\) Lower rotary axis value
- "Plus" \(\rightarrow\) Higher rotary axis value

Also in the basic setting (pole setting) of the machine kinematics, the NC calculates two solutions and these are approached by CYCLE800. The reference is the rotary axis that was set as direction reference when commissioning the "swivel" function.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.
If one of the two positions cannot be reached for mechanical reasons, the alternative position is automatically selected irrespective of the setting of the "Direction" parameter.

\section*{Example:}
- Machine kinematics with swivel head and swivel table.

Swivel head with rotary axis 1 (B) rotates around machine axis Y .
- Angular traversing range of rotary axis B from -90 to +90 degrees.
- Swivel table with rotary axis 2 (C) rotates around machine axis \(Z\).
- Angle traversing range of rotary axis 2 (C) from 0 to 360 degrees (modulo 360).
- Machine manufacturer has set the direction reference to rotary axis 1 (B) when he commissioned the swivel function.
- A rotation around X (WCS) of 10 degrees is programmed in the swivel cycle.

The machine in the basic setting (pole setting) of the kinematics \((B=0 C=0)\) is shown in the following diagram.

- Rotary axis B moves to -10 degrees in the negative direction (red arrow).
- Rotary axis C moves to 90 degrees (rotation around X!).
- Direction "+" (plus)
- Rotary axis B moves to +10 degrees in the positive direction (red arrow).
- Rotary axis C moves to 270 degrees.

The two "Minus" or "Plus" direction settings enable a workpiece to be machined with swiveled planes. The two solutions calculated by the NC differ by 180 degrees (see rotary axis C ).

\section*{Tool}

To avoid collisions, you can use the 5 -axis transformation (software option) to define the position of the tool tip during swiveling.
- Correct

The position of the tool tip is corrected during swiveling (tracking function).
- No correction

The position of the tool tip is not corrected during swiveling.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been reted and you are in the editor.

\section*{Uari- \\ aus}

Swivel plane

Initial setting
2. Select the "Miscellaneous" softkey.
3. Press the "Swivel plane" softkey.

The "Swivel plane" input window opens.
4. Press the "Basic setting" softkey if you wish to reestablish the initial state, i.e. you wish to set the values back to 0 .
This is done, for example, to swivel the coordinate system back to its original orientation.


\subsection*{9.6 Further cycles and functions}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline TC U & \multicolumn{2}{|l|}{Name of swivel data set} & \\
\hline \multirow[t]{5}{*}{Retract U- (only for G code)} & No & No retraction before swiveling & \\
\hline & ¢ ink & Incremental retraction in tool direction The retraction path is entered into parameter ZR When retracting in the tool direction, in the swiveled machine state, several axes can move (traverse) & \\
\hline & -1 \({ }^{\text {max }}\) & Maximum retraction in tool direction & \\
\hline & t. 2 & Retraction in the direction of machine axis \(Z\). & \\
\hline & t. 2 XY & Retract towards the machine axis \(Z\) and then in the direction X, Y & \\
\hline ZR & \multicolumn{2}{|l|}{Retraction path - (only for incremental retraction in the tool direction)} & mm \\
\hline Swivel plane U- (only for G code) & \multicolumn{2}{|l|}{\begin{tabular}{l}
- New: New swivel plane \\
- Additive: Additive swivel plane
\end{tabular}} & \\
\hline RP - (only for ShopTurn) & \multicolumn{2}{|l|}{Retraction plane for face B} & \\
\hline C0 - (only for ShopTurn) & \multicolumn{2}{|l|}{Position angle for machining surface} & Degrees \\
\hline X0 & \multicolumn{2}{|l|}{Reference point for rotation \(X\)} & \\
\hline Y0 & \multicolumn{2}{|l|}{Reference point for rotation \(Y\)} & \\
\hline Z0 & \multicolumn{2}{|l|}{Reference point for rotation \(Z\)} & \\
\hline Swivel modeU & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Axis-by-axis: Swivel coordinate system axis-by-axis \\
- Solid angle: Swivel via solid angle \\
- Proj. angle: Swiveling via projection angle \\
- Direct: Directly position rotary axes
\end{tabular}} & \\
\hline Axis sequence \(\mathbf{U}\) & \multicolumn{2}{|l|}{Sequence of the axes which are rotated around - (only for axis-by-axis swivel mode) \(X Y Z\) or \(X Z Y\) or \(Y X Z\) or \(Y Z X\) or \(Z X Y\) or \(Z Y X\)} & \\
\hline X & Rotation around \(X\) & \multirow[t]{3}{*}{- (only for axis sequence)} & Degrees \\
\hline Y & Rotation around Y & & Degrees \\
\hline Z & Rotation around \(Z\) & & Degrees \\
\hline Projection position U & \multicolumn{2}{|l|}{Position of the projection in space - (only for swivel mode, projection angle) \(X \alpha, Y \alpha, Z \beta\) or \(Y a, Z \alpha, Z \beta\) or \(Z \alpha, X \alpha, Z \beta\)} & \\
\hline X \(\alpha\) & Projection angle & \multirow[t]{3}{*}{- (only for projection position)} & Degrees \\
\hline Ya & Projection angle & & Degrees \\
\hline Z \(\beta\) & Angle of rotation in the plane & & Degrees \\
\hline Z & \multicolumn{2}{|l|}{Angle of rotation in the plane} & Degrees \\
\hline X1 & \multicolumn{2}{|l|}{Zero point of rotated surface \(X\)} & \\
\hline Y1 & \multicolumn{2}{|l|}{Zero point of rotated surface \(Y\)} & \\
\hline Z1 & \multicolumn{2}{|l|}{Zero point of rotated surface \(Z\)} & \\
\hline Direction U- (only for G code) & \multicolumn{2}{|l|}{Preferred direction, rotary axis 1 - (not for swivel mode direct)} & \\
\hline
\end{tabular}


\subsection*{9.6.2 Swiveling tool (CYCLE800)}

\subsection*{9.6.2.1 Aligning turning tools - only for G code program (CYCLE800)}

\section*{Function}

The "Align turning tool" and "Align milling tool" functions support combined turning-milling machines with a B axis that can be swiveled.

In contrast to "Swivel plane", no rotation is operative in the active work offsets in the workpiece coordinate system in the case of "Align tool". Only the offsets calculated by the NC and the corresponding tool orientation are effective.

The maximum angular range for "Align tool" is \(+/-360\) degrees or it is limited by the traversing range of the participating rotary axes. Technological limits are also placed on the angular range depending on the tool used. When aligning the tool, the data of the tool is calculated based on the tool orientation using the CUTMOD NC command. For a turning tool, the calculation involves the cutting edge position, the holder angle and the cut direction.

\section*{Definition of the \(\beta\) and \(\gamma\) angles}

The beta and gamma angles orientate the turning tools. They refer to the WCS. If the WCS corresponds to the MCS, the tool data remains unchanged for \(\beta=0^{\circ} / \gamma=0^{\circ}\) (cutter position, holder angle, ...).

The definition of angles beta and gamma depends on the particular machine. In the initial state of the machine kinematics for turning, a turning tool can be orientated according to Z or X.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

Initial state of the machine kinematics
The tool axis is aligned in the \(Z\) direction.

\(\beta=90^{\circ}\) represents a rotation of the cutting plate by +Y .


\section*{Mirroring}

A mirroring of the \(Z\) axis (e.g. on the counter-spindle) for \(\beta=0^{\circ} / \gamma=0^{\circ}\) causes the same machining in the mirrored coordinate system.
The mirroring of the \(Z\) axis must be permanently activated in a work offset.


The cutting edge position is calculated using the CUTMOD function.
If milling is to be possible on any swiveled machining plane, then the "swivel plane" function must be used.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
1. The part program to be executed has been created and you are in the editor.

2. Select the "Miscellaneous" softkey.
\begin{tabular}{|c|l|l|}
\hline \begin{tabular}{c} 
Swivel \\
tool
\end{tabular} & 3. & \begin{tabular}{l} 
Press the "Swivel tool" and "Align turning tool" softkeys. \\
The "Align turning tool" input window opens.
\end{tabular} \\
\begin{tabular}{c} 
Align \\
turning tool
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline TC U & \multicolumn{2}{|l|}{Name of swivel data set} & \\
\hline \multirow[t]{4}{*}{\begin{tabular}{l}
Retract \\
U
\end{tabular}} & No & No retraction before swiveling & \\
\hline & - \(\uparrow\) ink & \begin{tabular}{l}
Incremental retraction in tool direction \\
The retraction path is entered into parameter ZR.
\end{tabular} & \\
\hline & - \(\uparrow\) max & Maximum retraction in tool direction & \\
\hline & t 2 & Retraction in the direction of machine axis \(Z\) & \\
\hline ZR & \multicolumn{2}{|l|}{Retraction path - (only for incremental retraction in the tool direction)} & \\
\hline \(\beta\) & \multicolumn{2}{|l|}{Rotation around the 3rd geometry axis (for G18 Y)} & Degrees \\
\hline Y & \multicolumn{2}{|l|}{Rotation around the turning tool} & Degrees \\
\hline \multirow[t]{3}{*}{Tool U} & \multicolumn{2}{|l|}{Tool tip position when swiveling} & \\
\hline & \({ }^{6}{ }^{\text {a }}\) & \begin{tabular}{l}
Follow up \\
The position of the tool tip is maintained during swiveling.
\end{tabular} & \\
\hline & N & \begin{tabular}{l}
No follow up \\
The position of the tool tip changes during swiveling.
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.6.2.2 Aligning milling tools - only for G code program (CYCLE800)}

\section*{Procedure}
1. The part program to be executed has been created and you are in the editor.

2. Press the "Various" softkey.
3. Press the "Swivel tool" and "Align milling tool" softkeys. The "Align milling tool" input window opens.
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline TC U & \multicolumn{2}{|l|}{Name of the swivel data record} & \\
\hline \multirow[t]{5}{*}{Retraction U} & No & No retraction before swiveling & \\
\hline & - \(\uparrow\) ink & \begin{tabular}{l}
Incremental retraction in tool direction \\
The retraction path is entered into parameter ZR.
\end{tabular} & \\
\hline & - \(\uparrow\) max & Maximum retraction in tool direction & \\
\hline & ¢ 2 & Retraction in the direction of machine axis \(Z\) & \\
\hline & \(\dagger\) 2XY & Retract towards the machine axis Z and then in the direction \(\mathrm{X}, \mathrm{Y}\) & \\
\hline ZR & \multicolumn{2}{|l|}{Retraction path - (only for incremental retraction in the tool direction)} & \\
\hline \(\beta\) & \multicolumn{2}{|l|}{Rotation around the 3rd geometry axis (for G18 Y)} & Degrees \\
\hline \multirow[t]{3}{*}{Tool U} & \multicolumn{2}{|l|}{Tool tip position when swiveling} & \\
\hline & \({ }^{6}{ }^{\text {a }}\) & \begin{tabular}{l}
Tracking \\
The position of the tool tip is maintained during swiveling.
\end{tabular} & \\
\hline & 䍖, & No tracking The position of the tool tip changes during swiveling. & \\
\hline
\end{tabular}

\subsection*{9.6.2.3 Preloading milling tools - only for G code program (CYCLE800)}

After "Swivel plane", the tool orientation is always perpendicular on the machining plane. When milling with radial cutters, it can make technological sense to set the tool at an angle to the normal surface vector. In the swivel cycle, the setting angle is generated by an axis rotation (max. +/- 90 degrees) to the active swivel plane. When setting, the swivel plane is always "additive". With "Setting tool", only rotations are displayed on the swivel cycle input screen form. The user can freely select the rotation sequence.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.


Figure 9-9 The length up to the TCP (Tool Center Point) must be entered as tool length of the radial cutter.

\section*{Procedure}
1. The part program to be executed has been created and you are in the editor.
2. Press the "Various" softkey.

Swivel
3. Press the "Swivel tool" and "Setting milling tool" softkeys.

The "Setting tool" input window opens.
Orient milling tool

\subsection*{9.6 Further cycles and functions}
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline TC U & \multicolumn{2}{|l|}{Name of the swivel data record} & \\
\hline \multirow[t]{5}{*}{Retraction U} & No & No retraction before swiveling & \\
\hline & - \(\dagger\) ink & \begin{tabular}{l}
Incremental retraction in tool direction \\
The retraction path is entered into parameter ZR.
\end{tabular} & \\
\hline & - \(\uparrow\) max & Maximum retraction in tool direction & \\
\hline & \(t \quad 2\) & Retraction in the direction of machine axis Z & \\
\hline & ¢ \(2 \times \mathrm{XY}\) & Retract towards the machine axis Z and then in the direction \(\mathrm{X}, \mathrm{Y}\) & \\
\hline ZR & \multicolumn{2}{|l|}{Retraction path - (only for incremental retraction in the tool direction)} & \\
\hline Axis sequence \({ }^{\text {d }}\) & \multicolumn{2}{|l|}{Sequence of the axes which are rotated around XY or XZ or YX or YZ or ZX or ZY} & \\
\hline X & \multicolumn{2}{|l|}{Rotation around \(X\)} & Degrees \\
\hline Y & \multicolumn{2}{|l|}{Rotation around Y} & Degrees \\
\hline \multirow[t]{3}{*}{\[
\begin{aligned}
& \text { Tool } \\
& \mathrm{U}
\end{aligned}
\]} & \multicolumn{2}{|l|}{Tool tip position when swiveling} & \\
\hline & \({ }^{6}{ }^{\text {a }}\) & \begin{tabular}{l}
Tracking \\
The position of the tool tip is maintained during swiveling.
\end{tabular} & \\
\hline & N & \begin{tabular}{l}
No tracking \\
The position of the tool tip changes during swiveling.
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.6.3 High-speed settings (CYCLE832)}

\section*{Function}

With the "High Speed Settings" function (CYCLE832), data for the machining of free-form surfaces is pre-assigned values so that optimum machining is possible.
The call of CYCLE832 contains three parameters:
- Tolerance
- Machining type (technology)
- Input of the orientation tolerance (for 5-axis machines)

Machining of free-form surfaces involves high requirements regarding velocity, precision and surface quality.

You can achieve optimum velocity control depending on the type of machining (roughing, rough-finishing, finishing) very simply with the "High Speed Settings" cycle.
Program the cycle in the technology program before the geometry program is called.
The "High Speed Settings" cycle is also in conjunction with the "Advanced Surface" function.

\(\square\)

\section*{Software option}

You require the software option in order to use this function:
"Advanced Surface"

\section*{Machining methods}

With the "High Speed Settings" function, you can select between four technological machining types:
- "Finishing"
- "Rough-finishing"
- "Roughing"
- "Deselected" (default setting)

\section*{Note}

Plain text entry
You can enter the parameters in plain text in the "Machining" selection box.
Plain text is generated for the "Machining mode" parameter when the input screen is closed.

For CAM programs in the HSC range, the four machining types directly relate to the accuracy and speed of the path contour (see help screen).
The operator/programmer uses the tolerance value to give a corresponding weighting.
Corresponding to the appropriate G commands, the four machining types are assigned to technology G group 59:
\begin{tabular}{|l|l|}
\hline Machining type & Technology G group 59 \\
\hline Deselection & DYNNORM \\
\hline Finishing & DYNFINISH \\
\hline Rough-finishing & DYNSEMIFIN \\
\hline Roughing & DYNROUGH \\
\hline
\end{tabular}

\section*{Orientation tolerance}

You can enter the orientation tolerance for applications on machines with the dynamic multiaxis orientation transformation (TRAORI).

\section*{MD note}

Additional G commands that are available in conjunction with machining free-form surfaces, are also activated in the High Speed Settings cycle.
When deselecting CYCLE832, the G groups are programmed to the settings - during the program run time - that are declared in the machine data for the reset state.

\section*{Display of important information}

In the "Machine" operating area, you have the option of displaying important HSC information.

\section*{References}

For additional information, please refer to the following documentation:
Commissioning Manual SINUMERIK Operate / SINUMERIK 840D sl


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{See also}

G functions for mold making (Page 166)

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Select the "Miscellaneous" softkey.
3. Press the ">>" softkey.
4. Press the "High Speed Settings" softkey.

The "High Speed Settings" input window is opened.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
U
\end{tabular} & \begin{tabular}{l}
- \(\nabla\) (roughing) \\
Plain text: _ROUGH \\
- \(\quad \nabla \nabla\) (semi-finishing) \\
Plain text: _SEMIFIN \\
- \(\quad \nabla \nabla \nabla\) (finishing) \\
Plain text entry: _FINISH \\
- Deselection \\
Plain text entry: _OFF
\end{tabular} & \\
\hline & \begin{tabular}{l}
For "Multi-axis programming yes", the following plain texts are generated in accordance with the machining type: \\
- \(\nabla\) (roughing) with input of the orientation tolerance \\
Plain text: _ORI_ROUGH \\
- \(\quad \nabla \nabla\) (semi-finishing) with input of the orientation tolerance \\
Plain text entry: _ORI_SEMIFIN \\
- \(\quad \nabla \nabla \nabla\) (finishing) with input of the orientation tolerance \\
Plain text entry: _ORI_FINISH
\end{tabular} & \\
\hline Tolerance & Tolerance of the machining axis & \\
\hline Multi-axis program U & \begin{tabular}{l}
Multi-axis program for 5-axis machines \\
- Yes \\
The orientation tolerance \(>0\) degrees can be entered here \\
- No \\
The value 1 is entered automatically \\
Note \\
The field can be hidden. \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{9.6.4 Subroutines}

If you require the same machining steps when programming different workpieces, you can define these machining steps in a separate subprogram. You can then call this subprogram in any program.
Identical machining steps therefore only have to be programmed once.
A distinction is not made between the main program and subprograms. This means that you can call a "standard" ShopTurn or G code program in another ShopTurn program as a subprogram.

You can also call another subprogram in the subprogram. The maximum nesting depth is 15 subprograms.

\section*{Note}

You cannot insert subprograms in linked blocks.

If you want to call a ShopTurn program as a subprogram, the program must already have been calculated once (load or simulate program in the "Machine Auto" mode). This is not necessary for \(G\) code subprograms.

\section*{Program clipboard}

If you use the "Execution from external storage (EES)" software option, the subprogram can be stored locally or externally in an arbitrary program memory configured for EES.

Without the "Execution from external storage (EES)" software option, the subprogram must always be stored in the NCK work memory (in a separate "XYZ" directory or in the "Subprograms" directory). If you still want to call a subprogram located on another drive, you can use G code command "EXTCALL".

\section*{Program header}

Please note that when a subprogram is called, the settings in the program header of the subprogram are evaluated. These settings also remain active even after the subprogram has been exited.
If you wish to activate the settings from the program header for the main program again, you can make the settings again in the main program after calling the subprogram.

\section*{Procedure}
1. Create a ShopTurn or \(G\) code program that you would like to call as a subprogram in another program.
2. Position the cursor in the work plan or in the program view of the main program on the program block after which you wish to call the subprogram.
3. Press the "Various" and "Subprogram" softkeys.
4. Enter the path of the subprogram if the desired subprogram is not stored in the same directory as the main program.
5. Enter the name of the subprogram that you want to insert.

You only need to enter the file extension (*.mpf or *. spf) if the subprogram does not have the file extension specified for the directory in which the subprogram is stored.
6. Press the "Accept" softkey.

The subprogram call is inserted in the main program.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & \\
\hline Path/workpiece & \begin{tabular}{l} 
Path of the subprogram if the desired subprogram is not stored in the same direc- \\
tory as the main program.
\end{tabular} & \\
\hline Program name & Name of the subprogram that is to be inserted. & \\
\hline
\end{tabular}

\section*{Programming example}
\begin{tabular}{|c|c|}
\hline N10 T1 D1 & ; Load tool \\
\hline N11 M6 & \\
\hline N20 G54 G710 & ; Select work offset \\
\hline N30 M3 S12000 & ; Switch-on spindle \\
\hline N40 CYCLE832 (0.05,3,1) & ; Tolerance value 0.05 mm , machining type, roughing \\
\hline N50 EXTCALL"CAM_SCHRUPP" & Externally call subprogram CAM_SCHRUPP \\
\hline N60 T2 D1 & ;Load tool \\
\hline N61 M6 & \\
\hline N70 CYCLE832 (0.005,1,1) & ;Tolerance value 0.005 mm , machining type, finishing \\
\hline N80 EXTCALL"CAM_SCHLICHT" & ; Call subprogram CAM_SCHLICHT \\
\hline N90 M30 & ; End of program \\
\hline
\end{tabular}

The subprograms CAM_SCHRUPP.SPF, CAM_SCHLICHT.SPF contain the workpiece geometry and the technological values (feedrates). These are externally called due to the program size.

\subsection*{9.7 Additional cycles and functions in ShopTurn}

\subsection*{9.7.1 Drilling centric}

\section*{Function}

Using the "Drill centric" cycle, you can perform drilling operations at the center of a face surface.
You can choose between chip breaking during drilling or retraction from the workpiece for swarf removal. During machining, either the main spindle or counterspindle rotates. You can use a drill, rotary drill or milling cutter as the tool.
The tool is moved with rapid traverse to the programmed position, allowing for the return plane and safety clearance.

\section*{Note}

Working with rotating tool spindle
For example, if you want to drill very deep holes, you can also employ a rotating tool spindle. First specify the required tool and tool spindle speed under "Straight/Circle" \(\rightarrow\) "Tool". Then program the "Drill centered" function.

\section*{Note}

Stop tool spindle
If, during "Center drilling", the tool spindle does not rotate despite having been previously activated, then program the " M 5 " G code command before "Center drilling" in order to stop the tool spindle.

\section*{Input simple}

For simple machining operations, you have the option to reduce the wide variety of parameters to the most important parameters using the "Input" selection field. In this "Input simple" mode, the hidden parameters are allocated a fixed value that cannot be adjusted.

\section*{Machine manufacturer}

Various defined values can be pre-assigned using setting data.
Please refer to the machine manufacturer's specifications.
If the workpiece programming requires it, you can display and change all of the parameters using "Input complete".

\section*{Approach/retraction during chipbreaking}
1. The tool drills at the programmed feedrate \(F\) as far as the first infeed depth.
2. For chipbreaking, the tool retracts by the retraction value V 2 and drills as far as the next infeed depth that can be reduced by the factor DF.
3. Step 2 is repeated until final drilling depth Z 1 has been reached and dwell time DT has expired.
4. The tool retracts to the safety clearance with rapid traverse.

\section*{Approach/retraction during stock removal}
1. The tool drills at the programmed feedrate \(F\) as far as the first infeed depth.
2. The tool is retracted from the workpiece with rapid traverse to the safety clearance for stock removal and is then re-inserted at the first infeed depth in the automatic mode reduced by an anticipation distance calculated by the control system.
3. The tool then drills down to the next infeed depth that can be reduced by the factor DF and the tool retracts again to \(\mathrm{ZO}+\) safety clearance for stock removal.
4. Step 3 is repeated until final drilling depth Z 1 has been reached and dwell time DT has expired.
5. The tool retracts to the safety clearance with rapid traverse.

\section*{Procedure}

\section*{Drilling} centric

Drilling centric
1. The ShopTurn program to be edited has been created and you are in the editor.
2. Press the "Drilling" and "Drill centric" softkeys.

The "Drilling centered" input window opens.

\section*{Parameters in the "Input complete" mode}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Input & Complete & \\
\hline T & Tool name & \\
\hline D & Cutting edge number & \\
\hline FO & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline s/vo & Spindle speed or constant cutting rate & \[
\begin{aligned}
& \mathrm{rpm} \\
& \mathrm{~m} / \mathrm{min}
\end{aligned}
\] \\
\hline Machining \(U\) & \begin{tabular}{l}
- Chipbreaking \\
- Swarf removal
\end{tabular} & \\
\hline Z0 & Reference point Z (abs) & \\
\hline Drilling depth \(\mathbf{U}\) & \begin{tabular}{l}
Referred to \\
- Shank \\
The drill is inserted until the drill shank reaches the value programmed for Z 1 . The angle entered in the tool list is taken into account. \\
- Tip \\
The drill is inserted until the drill tip reaches the value programmed for Z 1 .
\end{tabular} & \\
\hline Z1 U & Final drilling depth (abs) or final drilling depth in relation to Z0 (inc) & \\
\hline D & Maximum depth infeed & \\
\hline FD1 & Percentage for the feedrate for the first infeed & \% \\
\hline DFU & \begin{tabular}{l}
- Percentage for each additional infeed or \\
- Amount for each additional infeed \\
DF = 100: Infeed increment remains constant \\
DF < 100: Infeed increment is reduced in direction of final drilling depth. \\
Example: DF \(=80\) \\
Last infeed was 4 mm ; \\
\(4 \times 80 \%=3.2\); next infeed increment is 3.2 mm \\
\(3.2 \times 80 \%=2.56\); next infeed increment is 2.56 mm , etc.
\end{tabular} & \[
\begin{gathered}
\% \\
\mathrm{~mm}
\end{gathered}
\] \\
\hline V1 & \begin{tabular}{l}
Minimum depth infeed \\
Parameter V1 is available only if \(\mathrm{DF}<100 \%\) has been programmed. \\
A minimum infeed is programmed using parameter V1.
\end{tabular} & \\
\hline V2 & Retraction distance after each machining step - (only for "chipbreaking" operation) & \\
\hline \begin{tabular}{l}
Clearance distance \\
U
\end{tabular} & \begin{tabular}{l}
- (only for "swarf removal" operation) \\
- Manual \\
- Automatic
\end{tabular} & \\
\hline V3 & Clearance distance - (for "manual" clearance distance only) & \\
\hline & \begin{tabular}{l}
- Dwell time in seconds \\
- Dwell time in revolutions
\end{tabular} & \[
\begin{aligned}
& \mathrm{s} \\
& \text { rev }
\end{aligned}
\] \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline XD & \begin{tabular}{l} 
Center offset in X direction \\
The center offset can be used for example to produce a drill hole with an exact fit. A \\
rotary drill (rotary drill type) or U drill (drill type) is required. Other drill types are not \\
suitable. \\
The maximum center offset is stored in a machine data code.
\end{tabular} & mm \\
\hline
\end{tabular}

\section*{Parameters in the "Input simple" mode}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline Input & simple & \\
\hline T & Tool name & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{min}\) \\
\(\mathrm{mm} / \mathrm{rev}\)
\end{tabular} \\
\hline D & Cutting edge number & \begin{tabular}{l}
rpm \\
\(\mathrm{m} / \mathrm{min}\)
\end{tabular} \\
\hline F U & Feedrate & \begin{tabular}{l} 
Spindle speed or \\
constant cutting rate
\end{tabular} \\
\hline S / V U & \begin{tabular}{l} 
Chipbreaking \\
Swarf removal
\end{tabular} & mm \\
\hline MachiningU & Reference point Z & \\
\hline Z0 & Final drilling depth X (abs) or final drilling depth in relation to Z0 (inc) & \\
\hline Z1 U & \begin{tabular}{l} 
Center offset in \(X\) direction \\
The center offset can be used for example to produce a drill hole with an exact fit. A \\
rotary drill (rotary drill type) or U drill (drill type) is required. Other drill types are not \\
suitable. \\
The maximum center offset is stored in a machine data code.
\end{tabular} & \\
\hline D & \begin{tabular}{l} 
XD
\end{tabular} & \\
\hline
\end{tabular}

\section*{Hidden parameters}
\begin{tabular}{|l|l|l|c|}
\hline Parameter & Description & Value & Can be set in SD \\
\hline Drilling depth & Drilling depth in relation to the tip & Tip & \\
\hline FD1 & Percentage for the feedrate for the first infeed & \(90 \%\) & x \\
\hline DF & Percentage for each additional infeed & \(90 \%\) & x \\
\hline V1 & Minimum infeed & 1.2 mm & x \\
\hline V2 & Retraction distance after each machining step & 1.4 mm & x \\
\hline Clearance distance & The clearance distance is calculated by the cycle & Automatic & x \\
\hline DBT & Dwell time at drilling depth & 0.6 s & x \\
\hline DT & Dwell time at final drilling depth & 0.6 s & \\
\hline
\end{tabular}

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{9.7.2 Thread centered}

\section*{Function}

Using the "Centered tapping" cycle, tap a righthand or lefthand thread at the center of the face surface.

During machining, either the main spindle or counterspindle rotates.You can alter the spindle speed via the spindle override; feedrate override is not operative.
You can select drilling in one cut, chipbreaking or retraction from the workpiece for stock removal.

The tool is moved with rapid traverse to the programmed position, allowing for the retraction plane and safety clearance.

\section*{Approach/retraction in one cut}
1. The tool drills in the direction of the longitudinal axis at the programmed spindle speed \(S\) or cutting rate V as far as the final drilling depth Z 1 .
2. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at the programmed spindle speed SR or cutting rate VR.

\section*{Approach/retraction for stock removal}
1. The tool drills in the direction of the longitudinal axis at the programmed spindle speed \(S\) or feedrate V as far as the first infeed depth (maximum infeed depth D ).
2. The tool retracts from the workpiece to the safety clearance at spindle speed SR or cutting rate VR for stock removal.
3. Then the tool is inserted again at spindle speed \(S\) or feedrate \(V\) and drills to the next infeed depth.
4. Steps 2 and 3 are repeated until the programmed final drilling depth Z 1 is reached.
5. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at spindle speed SR or cutting rate VR.

\section*{Approach/retraction for chipbreaking}
1. The tool drills in the direction of the longitudinal axis at the programmed spindle speed S or feedrate V as far as the first infeed depth (maximum infeed depth D ).
2. The tool retracts by the retraction clearance V 2 for chipbreaking.
3. The tool then drills to the next infeed depth at spindle speed S or feedrate V .
4. Steps 2 and 3 are repeated until the programmed final drilling depth Z 1 is reached.
5. The direction of rotation of the spindle reverses and the tool retracts to the safety clearance at spindle speed SR or cutting rate VR.
The machine manufacturer may have made specific settings for centered tapping in a machine data element.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
1. The ShopTurn program to be processed has been created and you are in the editor.

Drill.
2. Press the "Drilling" and "Drill centric" and "Thread centric" softkeys.

The "Centric tapping" input window opens.

\section*{Drilling}
centric
Thread centric
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline T & Tool name & \\
\hline D & Cutting edge number & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{min}\) \\
\(\mathrm{mm} / \mathrm{rev}\)
\end{tabular} \\
\hline FU & Feedrate & \\
\hline Table & Thread table selection: & without \\
U & • ISO metric & Whitworth BSW \\
& \(\bullet\) Whitworth BSP & \\
& UNC & \\
\hline
\end{tabular}
9.7 Additional cycles and functions in ShopTurn
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Selection
\[
0
\] & \begin{tabular}{l}
Selection, table value: \\
- M1 - M68 (ISO metric) \\
- W3/4"; etc. (Whitworth BSW) \\
- G3/4"; etc. (Whitworth BSP) \\
- 1" - 8 UNC; etc. (UNC)
\end{tabular} & \\
\hline \begin{tabular}{l}
P \\
- (selection only possible for table selection "without")
\end{tabular} & \begin{tabular}{l}
Pitch ... \\
- in MODULUS: MODULUS \(=\) Pitch/ \(\pi\) \\
- in mm/rev \\
- in inch/rev \\
- in turns per inch: Used with pipe threads, for example. \\
When entered per inch, enter the integer number in front of the decimal point in the first parameter field and the figures after the decimal point as a fraction in the second and third field. \\
The pitch is determined by the tool used.
\end{tabular} & \begin{tabular}{l}
MODULUS \\
mm/rev \\
in/rev \\
turns/"
\end{tabular} \\
\hline S/VU & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline SR & Spindle speed for retraction & \(\mathrm{rev} / \mathrm{min}\) \\
\hline VR & Constant cutting rate for retraction & \(\mathrm{m} / \mathrm{min}\) \\
\hline Machining U & \begin{tabular}{l}
- 1. Cut \\
The thread is drilled in one cut without interruption. \\
- Chipbreaking \\
The drill is retracted by the retraction distance V2 for chipbreaking. \\
- Stock removal \\
The drill is retracted from the workpiece for stock removal.
\end{tabular} & \\
\hline Z0 & Reference point Z) & mm \\
\hline Z1 U & End point of the thread (abs) or thread length (inc) & mm \\
\hline D & Maximum depth infeed - (for stock removal or chipbreaking only) & mm \\
\hline Retraction \(\cup\) & \begin{tabular}{l}
- (only for "chipbreaking" operation) \\
Retraction distance \\
- Manual \\
- Automatic
\end{tabular} & \\
\hline V2 & \begin{tabular}{l}
Retraction distance (only for "manual" retraction) \\
Distance through which the tap is retracted for chipbreaking. V2 = automatic: The tool is retracted by one revolution.
\end{tabular} & mm \\
\hline
\end{tabular}

\subsection*{9.7.3 Transformations}

To make programming easier, you can transform the coordinate system. Use this possibility, for example, to rotate the coordinate system.
Coordinate transformations only apply in the actual program.
You can define the following transformations:
- Offset
- Rotation
- Scaling
- Mirroring
- Rotation C axis

You can select between a new or an additive coordinate transformation.
In the case of a new coordinate transformation, all previously defined coordinate transformations are deselected. An additive coordinate transformation acts in addition to the currently selected coordinate transformations.

\section*{Note}

\section*{Transformations with virtual axes}

Please note that when selecting TRANSMIT or TRACYL offsets, scaling and mirroring, the real \(Y\) axis is not transferred into the virtual \(Y\) axis.

Offsets, scalings and mirroring of the virtual \(Y\) axis are deleted for TRAFOOF.

\section*{Procedure for work offset, offset, rotation, scaling, mirroring or rotation C axis.}
1. The ShopTurn program has been created and you are in the editor.

Transformations

Work offset

Off-
set

Rotation
2. Press the "Various" and "Transformation" softkeys.
3. Press the "Work offsets" softkey.

The "Work offsets" input window opens.
- OR -

Press the "Offset" softkey.
The "Offset" input window opens.
- OR -

Press the "Rotation" softkey.
The "Rotate" input window opens.
- OR -
\begin{tabular}{|c|c|}
\hline \multirow[t]{2}{*}{Scaling} & Press the "Scaling" softkey. \\
\hline & The "Scaling" input window opens. \\
\hline & - OR - \\
\hline \multirow[b]{2}{*}{Mirroring} & Press the "Mirroring" softkey. \\
\hline & The "Mirroring" input window opens. \\
\hline & - OR - \\
\hline Rotation & Press the "Rotation C axis" softkey. \\
\hline C axis & The "Rotation C axis" input window opens. \\
\hline
\end{tabular}

\subsection*{9.7.4 Translation}

For each axis, you can program an offset of the zero point.


New offset


Additive offset
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline Offset & \(\bullet\) New & \\
\(\mathbf{U}\) & New offset & Additive \\
& Additive offset & \\
\hline Z & Offset Z & \\
\hline X & Offset X & Offset Y \\
\hline Y & & mm \\
\hline
\end{tabular}

\subsection*{9.7.5 Rotation}

You can rotate every axis through a specific angle. A positive angle corresponds to counterclockwise rotation.


New rotation


Additive rotation
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Rotation U & - New & \\
\hline & - New rotation & \\
\hline Z & Rotation around \(Z\) & Degrees \\
\hline X & Rotation around \(X\) & Degrees \\
\hline Y & Rotation around \(Y\) & Degrees \\
\hline
\end{tabular}

\subsection*{9.7.6 \\ Scaling}

You can specify a scale factor for the active machining plane as well as for the tool axis. The programmed coordinates are then multiplied by this factor.


New scaling


Additive scaling
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l} 
Scaling \\
O
\end{tabular} & \begin{tabular}{l} 
New \\
New scaling \\
Additive \\
Additive scaling
\end{tabular} & \\
\hline ZX & Scale factor ZX & \\
\hline Y & Scale factor Y & \\
\hline
\end{tabular}

\subsection*{9.7.7 Mirroring}

Furthermore, you can mirror all axes. Enter the axis to be mirrored in each case.

\section*{Note}

Travel direction of the milling cutter
Note that with mirroring, the travel direction of the cutting tool (conventional/climb) is also mirrored.


Table 9- 2
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline Mirroring & \begin{tabular}{l} 
New \\
\(\mathbf{U}\)
\end{tabular} & \begin{tabular}{l} 
New mirroring \\
Additive \\
Additive mirroring
\end{tabular}
\end{tabular}

\subsection*{9.7.8 Rotation C}

You can rotate the \(C\) axis through a specific angle to enable subsequent machining operations to be performed at a particular position on the face or peripheral surface.
The direction of rotation is set in a machine data element.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline Rotation & New & \\
\(\mathbf{O}\) & New rotation & \\
& Additive & \\
\hline C & Rodatitive rotation & \\
\hline
\end{tabular}

\subsection*{9.7.9 Straight and circular machining}

If you want to perform simple, i.e. straight or circular path movements or machining without defining a complete contour, you can use the functions "Straight" or "Circle" respectively.

\section*{General sequence}

To program simple machining operations, proceed as follows:
- Specify the tool and the spindle speed
- Program the machining operations

\section*{Machining options}

The following machining options are available:
- Straight line
- Circle with known center point
- Circle with known radius
- Straight line with polar coordinates
- Circle with polar coordinates

If you want to program a straight line or a circle using polar coordinates, you must define the pole first.

\section*{NOTICE}

\section*{Risk of collision}

If you retract the tool to the retraction area defined in the program header using either
straight or circular path motion, then you must carefully ensure that a collision cannot occur as a result of the normal retraction logic.
To be on the safe side, you should also move the tool back out of the retraction area again.

\subsection*{9.7.10 Selecting a tool and machining plane}

Before you can program a line or circle, you have to select the tool, spindle, spindle speed and machining plane.
If you program a sequence of different straight or circular path motions, the settings for the tool, spindle, spindle speed and machining plane remain active until you change them again.

If you change the selected machining plane subsequently, the coordinates of the programmed path motion are automatically adjusted to the new machining plane. The originally programmed coordinates remain unchanged only for a straight motion (rightangled, not polar).

\section*{Procedure}

\section*{Edit}


\section*{Tool}

Select tool
1. The ShopTurn program to be processed has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.
3. Press the "Tool" softkey. The "Tool" window is opened.
4. Enter a tool into parameter field "T".
- OR -

Press the "Select tool" softkey if you want to select a tool from the tool list, position the cursor on the tool that you wish to use for the machining operation and press the "To program" softkey. The tool is copied into the "T" parameter field.
5. Select the tool cutting edge number \(D\) if the tool has several cutting edges.
6. In the lefthand input field of the Spindle parameter, select main spindle, tool spindle or counterspindle.
7. Enter the spindle speed or cutting rate.
8. In the selection box "Plane selection", select between the machining planes.
9. Enter the cylinder diameter if you selected the machining plane peripheral surface \(C\).
- OR -

Enter the positioning angle for the CP machining area if you selected machining plane face Y .
- OR -

Enter reference point C0 if you selected the machining plane peripheral surface \(Y\).
- OR -

Choose whether the spindle should be clamped or released or whether there should be no change (input field left blank).
Press the "Accept" softkey.
The values are saved and the window is closed. The process plan is displayed and the newly generated program block is marked.
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline T & Tool name & \\
\hline DU & Cutting edge number & \\
\hline S1 / V1 U & Spindle speed or constant cutting rate & rpm \(\mathrm{m} / \mathrm{min}\) \\
\hline Plane selection \(\mathbf{U}\) & \begin{tabular}{l}
Select between the following machining surfaces: \\
- Peripheral surface/Peripheral C \\
- Peripheral surface Y - only if there is a Y axis \\
- Face/Face C \\
- Face \(Y\) - only if there is a \(Y\) axis \\
- Turning
\end{tabular} & \\
\hline \(\varnothing\) & Diameter of the cylinder (for peripheral surface/peripheral surface C) & mm \\
\hline C0 & Positioning angle for machining area (for peripheral surface Y ) & Degrees \\
\hline CP & Positioning angle for machining area (for face Y ) & Degrees \\
\hline
\end{tabular}

\subsection*{9.7.11 Programming a straight line}

When you want to program a straight line in right-angled coordinates, you can use the "Straight" function.
The tool moves along a straight line at the programmed feedrate or at rapid traverse from its actual position to the programmed end position.

\section*{Radius compensation}

Alternately, you can implement the straight line with radius compensation. The radius compensation acts modally, therefore you must deactivate the radius compensation again when you want to traverse without radius compensation. Where several straight line blocks with radius compensation are programmed sequentially, you may select radius compensation only in the first program block.

When executing the first path motion with radius compensation, the tool traverses without compensation at the starting point and with compensation at the end point. This means that if a vertical path is programmed, the tool traverses an oblique path. The compensation is not applied over the entire traversing path until the second programmed path motion with radius compensation is executed. The reverse effect occurs when radius compensation is deactivated.


Straight line when selecting radius compensa- Straight line when deselecting radius comtion pensation
If you want to prevent deviation from the programmed path, you can program the first straight line with radius compensation or with deactivated radius compensation outside the workpiece. Programming without coordinate data is not possible.

\section*{Procedure}


Straight

Rapid
1. The ShopTurn program to be processed has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.
3. Press the "Straight" softkey.
4. Press the "Rapid traverse" softkey if you want to use rapid traverse instead of a programmed machining feedrate.


\subsection*{9.7.12 Programming a circle with known center point}

To program a circle or arc with a known center point, use the "Circle center point" function.
The tool traverses a circular path from its actual position to the programmed target position at the machining feedrate. The system calculates the radius of the circle/arc on the basis of the entered interpolation parameter settings I and K.

\section*{Procedure}


Circle center
1. The ShopTurn program to be processed has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.
3. Press the "Circle center point" softkey.
\begin{tabular}{|c|c|c|c|}
\hline Parameters & \multicolumn{2}{|l|}{Description} & Unit \\
\hline Direction of rotation & \multicolumn{2}{|l|}{Direction of rotation in which the tool travels from the circle starting point to the circle end point:} & \\
\hline \multirow[t]{2}{*}{U} & 饣 & Direction of rotation clockwise (right) & \\
\hline & \(S\) & Direction of rotation counterclockwise (left) & \\
\hline YO
\(Z O\)
\(J\)
K & \multicolumn{2}{|l|}{\begin{tabular}{l}
Machining plane, peripheral surface \(C\) \\
Target position Y (abs) or target position X referred to the last programmed position (inc) \\
Target position \(Z\) (abs) or target position \(Y\) referred to the last programmed position (inc) \\
Circle center point J (ink). \\
Circle center point K (inc). \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular}} & mm
mm
mm
mm \\
\hline YO & \multicolumn{2}{|l|}{\multirow[t]{4}{*}{\begin{tabular}{l}
Machining plane, peripheral surface \(Y\) \\
Target position \(Y\) (abs) or target position X referred to the last programmed position (inc) \\
Target position \(Z\) (abs) or target position \(Y\) referred to the last programmed position (inc) \\
Circle center point J (ink). \\
Circle center point K (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular}}} & mm \\
\hline ZU & & & mm \\
\hline \(J\) & & & mm \\
\hline K & & & mm \\
\hline
\end{tabular}
9.7 Additional cycles and functions in ShopTurn
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \(X O\)
\(Y O\)
I
J & \begin{tabular}{l}
Machining plane face C \\
Target position \(\mathrm{X} \varnothing\) (abs) or target position X referred to the last programmed position (inc) \\
Target position Y (abs) or target position Y referred to the last programmed position (inc) \\
Circle center point I (ink) \\
Circle center point J (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline \(X O\)
\(Y O\)
I
J & \begin{tabular}{l}
Machining plane face \(Y\) \\
Target position X (abs) or target position X referred to the last programmed position (inc) \\
Target position Y (abs) or target position Y referred to the last programmed position (inc) \\
Circle center point I (inc). \\
Circle center point J (inc). \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline XO
ZO
I
K & \begin{tabular}{l}
Machining plane rotation \\
Target position \(\mathrm{X} \varnothing\) (abs) or target position Y referred to the last programmed position (inc) \\
Target position \(Z\) (abs) or target position X referred to the last programmed position (inc) \\
Circle center point I (ink). \\
Circle center point K (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline FU & Machining feedrate & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
\(\mathrm{mm} / \mathrm{min}\) \\
mm/tooth
\end{tabular} \\
\hline
\end{tabular}

\subsection*{9.7.13 Programming a circle with known radius}

To program a circle or arc with a known radius, use the "Circle radius" function.
The tool traverses a circular arc with the programmed radius from its actual position to the programmed target position at the machining feedrate. To do this, the system calculates the position of the circle center point.
You can choose to traverse the arc in the clockwise or anticlockwise direction. Depending on the direction of rotation, there are two options for approaching the target position from the current position via an arc of the specified radius.

You can select the arc of your choice by entering a positive or a negative sign for the radius.


Opening angle up to \(180^{\circ}\) : +
Opening angle greater than \(180^{\circ}\) : -

Figure 9-10 Opening angle

\section*{Procedure}
1. The ShopTurn program to be processed has been created and you are in the editor.

2. Press the menu forward key and the "Straight Circle" softkey.

Circle 3. Press the "Circle radius" softkey.
radius
\begin{tabular}{|c|c|c|c|}
\hline Parameters & \multicolumn{2}{|l|}{Description} & Unit \\
\hline Direction of rotation & \multicolumn{2}{|l|}{Direction of rotation in which the tool travels from the circle starting point to the circle end point} & \\
\hline \multirow[t]{2}{*}{0} & 2 & Direction of rotation clockwise (right) & \\
\hline & 9 & Direction of rotation counterclockwise (left) & \\
\hline \(Y O\)
\(Z O\) & \multicolumn{2}{|l|}{\begin{tabular}{l}
Machining plane peripheral surface/peripheral surface \(C\) \\
Target position Y (abs) or target position X referred to the last programmed position (inc) \\
Target position \(Z\) (abs) or target position \(Y\) referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated
\end{tabular}} & mm
mm \\
\hline YO
ZO & \multicolumn{2}{|l|}{\begin{tabular}{l}
Machining plane, peripheral surface \(Y\) \\
Target position Y (abs) or target position X referred to the last programmed position (inc) \\
Target position \(Z\) (abs) or target position \(Y\) referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular}} & mm
mm \\
\hline XO
YO & \multicolumn{2}{|l|}{\begin{tabular}{l}
Machining plane face/face C \\
Target position X (abs) or target position X referred to the last programmed position (inc) \\
Target position Y (abs) or target position Y referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated
\end{tabular}} & mm
mm \\
\hline XO
YO & \multicolumn{2}{|l|}{\begin{tabular}{l}
Machining plane face \(Y\) \\
Target position X (abs) or target position X referred to the last programmed position (inc) \\
Target position Y (abs) or target position Y referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular}} & mm \\
\hline XO
ZO & \multicolumn{2}{|l|}{\begin{tabular}{l}
Machining plane rotation \\
Target position \(\mathrm{X} \varnothing\) (abs) or target position Y referred to the last programmed position (inc) \\
Target position \(Z\) (abs) or target position X referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular}} & mm
mm \\
\hline R & \multicolumn{2}{|l|}{\begin{tabular}{l}
Radius of circular arc \\
The sign determines the type of arc traversed.
\end{tabular}} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline FO & \multicolumn{2}{|l|}{Machining feedrate.} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
\(\mathrm{mm} / \mathrm{min}\) \\
\(\mathrm{mm} / \mathrm{tooth}\)
\end{tabular} \\
\hline
\end{tabular}

\subsection*{9.7.14 Polar coordinates}

If a workpiece has been dimensioned from a central point (pole) with radius and angles, you will find it helpful to program these dimensions as polar coordinates.
Before you program a straight line or circle in polar coordinates, you must define the pole, i.e. the reference point, of the polar coordinate system.

\section*{Procedure}
1. The ShopTurn program to be processed has been created and you are in the editor.

2. Press the menu forward key and the "Straight Circle" softkey.


Polar

Pole
3. Press the "Polar" and "Pole" softkeys.
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline \[
\begin{aligned}
& Y O \\
& Z O
\end{aligned}
\] & \begin{tabular}{l}
Machining plane peripheral surface/peripheral surface \(C\) \\
Pole Y (abs) \\
Pole Z (abs) or pole Z referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& Y O \\
& Z O
\end{aligned}
\] & \begin{tabular}{l}
Machining plane, peripheral surface \(Y\) \\
Pole Y (abs) \\
Pole Z (abs) or pole Z referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& X U \\
& Y U
\end{aligned}
\] & \begin{tabular}{l}
Machining plane face/face \(\mathbf{C}\) \\
Pole \(X \varnothing\) (abs) \\
Pole Y (abs) or pole Y referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& X U \\
& Y O
\end{aligned}
\] & \begin{tabular}{l}
Machining plane face \(Y\) \\
Pole X (abs) \\
Pole Y (abs) or pole Y referred to the last programmed position (inc) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline \[
\begin{aligned}
& x U \\
& Z U
\end{aligned}
\] & \begin{tabular}{l}
Machining plane rotation \\
Pole X (abs) or pole X referred to the last programmed position (inc) \\
Z position pole (abs) \\
Note: \\
Incremental dimension: The sign is also evaluated.
\end{tabular} & \[
\begin{aligned}
& \mathrm{mm} \\
& \mathrm{~mm}
\end{aligned}
\] \\
\hline
\end{tabular}

Turning

\subsection*{9.7.15 Straight line polar}

When you want to program a straight line in polar coordinates, you can use the "Straight Polar" function.

A straight line in the polar coordinate system is defined by the length \(L\) and the angle \(\alpha\). Depending on the selected machining plane, the angle refers to another axis. The direction in which a positive angle points also depends on the machining plane.
\begin{tabular}{|l|l|l|l|}
\hline Machining plane & Turning & Face & Peripheral \\
\hline Reference axis for angle & Z & X & Y \\
\hline Positive angle in direction of the axis & X & Y & Z \\
\hline
\end{tabular}

The tool traverses a straight line from its current position to the programmed end point at the machining feedrate or at rapid traverse.

The 1st line in polar coordinates entered after the pole must be programmed in absolute dimensions. You can program any additional lines or arcs also in incremental dimensions.

\section*{Radius compensation}

Alternately, you can implement the straight line with radius compensation. The radius compensation acts modally, therefore you must deactivate the radius compensation again when you want to traverse without radius compensation. Where several straight line blocks with radius compensation are programmed sequentially, you may select radius compensation only in the first program block.
For the first straight line with radius compensation, the tool approaches the starting point without radius compensation and the end point with radius compensation, i.e. if a vertical path is programmed, a slope will be traversed. The compensation does not act over the entire traverse path until the second programmed straight line with radius compensation. The reverse effect occurs when radius compensation is deactivated.


Straight line with selected radius compensation


Straight line with deselected radius compensation

If you want to prevent deviation from the programmed path, you can program the first straight line with radius compensation or with deactivated radius compensation outside the workpiece. Programming without coordinate data is not possible.

\section*{Procedure}

2. Press the menu forward key and the "Straight Circle" softkey.

\section*{Polar}

Straight
polar

Rapid
3. Press the "Polar" and "Straight Polar" softkeys.
4. Press the "Rapid traverse" softkey if you want to use rapid traverse instead of a programmed machining feedrate.
1. The ShopTurn program to be processed has been created and you are in the editor.
\begin{tabular}{|c|c|c|c|}
\hline Parameters & \multicolumn{2}{|l|}{Description} & Unit \\
\hline L & \multicolumn{2}{|l|}{Distance to the pole, end point} & mm \\
\hline aU & \multicolumn{2}{|l|}{Polar angle to the pole, end point (abs) or Polar angle change to the pole, end point (inc) The sign specifies the direction.} & Degrees \\
\hline FU & \multicolumn{2}{|l|}{Machining feedrate} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
\(\mathrm{mm} / \mathrm{min}\) \\
mm/tooth
\end{tabular} \\
\hline Radius compensation U & \multicolumn{2}{|l|}{Input defining which side of the contour the cutter travels in the traversing direction:} & \\
\hline \multirow[t]{4}{*}{} & 昣 & Radius compensation to left of contour & \\
\hline & \$6 & Radius compensation to right of contour & \\
\hline & \% & Radius compensation off & \\
\hline & - & The set radius compensation remains as previously set & \\
\hline
\end{tabular}

\subsection*{9.7.16 Circle polar}

If you want to program a circle or arc using polar coordinates, you can use the "Circle Polar" function.
A circle in the polar coordinate system is defined by the angle \(\alpha\). Depending on the selected machining plane, the angle refers to another axis. The direction in which a positive angle points also depends on the machining plane.
\begin{tabular}{|l|l|l|l|}
\hline Machining plane & Rotate & Face & Peripheral \\
\hline Reference axis for angle & Z & X & Y \\
\hline Positive angle in direction of the axis & X & Y & Z \\
\hline
\end{tabular}

The tool traverses a circular path from its actual position to the programmed end point (angle) at the machining feedrate. The radius is obtained from the distance between the actual tool position and the defined pole, i.e. the circle start and end point positions are at the same distance from the pole.

The 1st arc in polar coordinates entered after the pole must be programmed in absolute dimensions. You can program any additional lines or arcs also in incremental dimensions.

\section*{Procedure}

1. The ShopTurn program to be processed has been created and you are in the editor.
2. Press the menu forward key and the "Straight Circle" softkey.

Polar
Circle
polar
3. Press the "Polar" and "Circle Polar" softkeys.
\begin{tabular}{|l|l|l|l|}
\hline Parameters & Description & Unit \\
\hline \begin{tabular}{l} 
Direction of rotation \\
U
\end{tabular} & \begin{tabular}{l} 
Direction of rotation in which the tool travels from the circle starting point to the \\
circle end point
\end{tabular} & \\
\hline & Q & Direction of rotation clockwise (right) & \\
\hline & a & Direction of rotation counterclockwise (left) & Degrees \\
\hline FU & \begin{tabular}{l} 
Polar angle to the pole, end point (abs) or \\
Polar angle change to the pole, end point (inc) \\
The sign specifies the direction.
\end{tabular} & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{rev}\) \\
\(\mathrm{mm} / \mathrm{min}\) \\
\(\mathrm{mm} / \mathrm{tooth}\)
\end{tabular} \\
\hline
\end{tabular}

\subsection*{9.7.17 Machining with movable counterspindle}

If your lathe has a counter-spindle, you can machine workpieces using turning, drilling and milling functions on the front and rear faces without reclamping the workpiece manually.
You have the possibility to start the machining in the main spindle or in the counter-spindle. Prior to machining the associated front or rear side, the workpiece is gripped by the counterspindle or the main spindle, withdrawn from the main spindle or counter-spindle and travels to the new machining position. You can program these operations with the "Counter-spindle" function.

\section*{Operations}

The following steps are available to program the operations:
- Gripping: Gripping the workpiece with the counter-spindle or main spindle (possibly with limit stop)
- Withdrawing: Withdrawing a workpiece with the counter-spindle from the main spindle or with the main spindle from the counter-spindle
- Counter-spindle machining side: Traverse workpiece with the counter-spindle or main spindle to a new machining position; select work offset for the machining side
- Complete transfer: Gripping, withdrawing (possibly with cutting-off) and machining side
- Main spindle machining side: Work offset for machining the next front face (for bars)

If you start to execute a program containing a counter-spindle machining operation, the counter-spindle is first retracted to the return position defined in a machine data element.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Teaching in the parking position and angle offset}

Teaching the park position is possible only if you have selected the machine coordinate system (MCS).
. Manually rotate the counter-spindle chuck to the desired position, and move the tool to the desired position.
3. Select the "Gripping" or "Complete transfer" programming step.
4. Select the "MCS" tool under the park position.
\begin{tabular}{|ccl}
\begin{tabular}{|c|c|}
\hline Teach \\
park pos.
\end{tabular} & 5. & \begin{tabular}{l} 
Press the "Teach park pos." softkey. \\
The actual tool park position is saved.
\end{tabular} \\
\begin{tabular}{|ccl} 
Teach & 6. & \begin{tabular}{l} 
Press the "Teach angl. offset" softkey.
\end{tabular} \\
angle off.
\end{tabular} & \begin{tabular}{l} 
The actual angular difference between the main and counter-spindles \\
will be saved.
\end{tabular}
\end{tabular}

\subsection*{9.7.17.1 Programming example: Machining main spindle - Transfer workpiece - Machining counterspindle}

The programming for this operation might look like this:

\section*{Programming steps - alternative 1:}
- Machining, main spindle
- Gripping
- Withdrawing
- Counter-spindle machining side
- Machining, counter-spindle

\section*{Programming steps - alternative 2:}
- Machining, main spindle
- Counter-spindle complete transfer (gripping, withdrawing and machining side)
- Machining, counter-spindle

\subsection*{9.7.17.2 Programming example: Machining counter-spindle - Transfer workpiece - Machining main spindle}

The programming for this operation might look like this:

\section*{Programming steps - alternative 1:}
- Machining, counter-spindle
- Gripping
- Machining side
- Machining, main side

Programming steps - alternative 2:
- Machining, counter-spindle
- Complete transfer (gripping and machining side)
- Machining, main spindle

\subsection*{9.7.17.3 Programming example: Machining, counterspindle - without previous transfer}

\section*{Programming steps}
- Rear face
- Work offset

Work offset is only activated
- ZV:

Parameter is not evaluated.
- Machining, counterspindle

\section*{Note}

\section*{Special feature regarding "rear face":}

The work offset that you choose in the parameter screen is only activated and not calculated. This means that the workpiece zero for counterspindle machining should be stored in the work offset. In addition, parameter ZV is not evaluated.

\subsection*{9.7.17.4 Programming example: Machining bar material}

If you use bars to produce your workpieces, you can machine several workpieces on the front and rear face by starting the program just once.

\section*{Programming steps - alternative 1:}
- Program header specifying the work offset in which the workpiece zero is stored
- Machining, main spindle
- Complete transfer (withdraw blank: yes; cutting-off cycle: yes)
- Cutting-off
- Machining, counter-spindle
- End of program with number of workpieces to be machined

Programming steps - alternative 2 :
- Start marker
- Machining, main spindle
- Complete transfer (withdraw blank: yes; cutting-off cycle: yes)
- Cutting-off
- Machining, counter-spindle
- Front face
- End marker
- Repeat from start to end marker

\section*{Note}

You can withdraw the blank several times successively without parting in order to continue the machining on the same side.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Function
\[
0
\] & \begin{tabular}{l}
You can select between five different functions: \\
- Complete transfer \\
- Gripping \\
- Withdrawing \\
- Machining side
\end{tabular} & \\
\hline Workpiece transfer & \begin{tabular}{l}
- Main spindle in counter-spindle \\
- Counter-spindle in main spindle
\end{tabular} & \\
\hline Complete transfer function & Gripping & \\
\hline \begin{tabular}{l}
Coordinate system \\
0
\end{tabular} & \begin{tabular}{l}
- Machine coordinate system (MCS) \\
The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. \\
- Workpiece coordinate system (WCS) \\
The park position is specified in the workpiece coordinate system.
\end{tabular} & \\
\hline XP & Park position of tool in X direction (abs) & mm \\
\hline ZP & Park position of tool in Z direction (abs) & mm \\
\hline Flush chuck \(\mathbf{U}\) & \begin{tabular}{l}
Flush counter-spindle chuck \\
- Yes \\
- No
\end{tabular} & \\
\hline DIR U & \begin{tabular}{l}
Direction of rotation \\
- \(\quad 2 \quad\) Spindle rotates clockwise \\
- \(\quad \varsigma \quad\) Spindle rotates counter-clockwise \\
- \(\quad \otimes \quad\) Spindle does not rotate
\end{tabular} & \\
\hline Clamping & \begin{tabular}{l}
Clamping both spindles (only if spindles are not turning) \\
- (O) \\
Clamping open \\
- \(\mathbf{C}\) \\
Clamping closed
\end{tabular} & \\
\hline S & Spindle speed - (only when the spindle rotates) & \(\mathrm{rev} / \mathrm{min}\) \\
\hline a1 & Angular offset & Degrees \\
\hline Z1 & Transfer position (abs.) & \\
\hline ZR U & Position, feedrate reduction (abs or inc) Position from which a reduced feedrate is used. & \\
\hline FR & Reduced feedrate & mm/rev \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Fixed stop & \begin{tabular}{l}
Travel to fixed stop \\
- Yes \\
The counter-spindle stops at a defined distance away from transfer position Z1 and then traverses with a defined feedrate up to the fixed stop. \\
- No \\
The counter-spindle traverses to the transfer position Z1.
\end{tabular} & \\
\hline & Withdrawing & \\
\hline Withdraw blank & \begin{tabular}{l}
Withdraw complete blank: \\
- Yes \\
- No
\end{tabular} & \\
\hline F & Feed - withdraw for blank "yes" & mm/min \\
\hline Cutting-off cycle & \begin{tabular}{l}
Cutting-off cycle in the following block \\
- Yes \\
- No
\end{tabular} & \\
\hline & \begin{tabular}{l}
Rear face \\
- for main spindle in counter-spindle
\end{tabular} & \\
\hline Work offset \(\mathbf{U}\) & \begin{tabular}{l}
Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z, must be saved: \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57 \\
- ...
\end{tabular} & \\
\hline Write to the work offset U & \begin{tabular}{l}
- Yes \\
The \(Z\) value of the work offset can be directly written to the input screen form. \\
- No \\
The actual \(Z\) value of the work offset is used.
\end{tabular} & \\
\hline ZV - only for work offset write "yes" & \begin{tabular}{l}
- Offset Z = 0 (abs) \\
- Workpiece zero is offset in \(Z\) direction (inc, the sign is also evaluated)
\end{tabular} & mm \\
\hline Z4W & Machining position for special axis (abs.); machine coordinate system & mm \\
\hline & \begin{tabular}{l}
Front face \\
- for counter-spindle in main spindle
\end{tabular} & \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Work offset \(\mathbf{U}\) & \begin{tabular}{l}
Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z , must be saved: \\
Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57 \\
- ...
\end{tabular} & \\
\hline Write to the work offset U & \begin{tabular}{l}
- Yes \\
The \(Z\) value of the work offset can be directly written to the input screen form. \\
- No \\
The actual \(Z\) value of the work offset is used.
\end{tabular} & \\
\hline ZV - only for work offset write "yes" & \begin{tabular}{l}
- Offset \(Z=0\) (abs) \\
- Workpiece zero is offset in \(Z\) direction (inc, the sign is also evaluated)
\end{tabular} & mm \\
\hline Z4P & Machining position for special axis (abs.); machine coordinate system & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Function, gripping & Teaching in the park position and angular offset is possible & \\
\hline Gripping a blank & \begin{tabular}{l}
- With main spindle \\
The blank is gripped with the main spindle \\
- With counter-spindle \\
The blank is gripped with the counter-spindle
\end{tabular} & \\
\hline Work offset only for "with main spindle" & \begin{tabular}{l}
Buffer work offset: \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57 \\
- ...
\end{tabular} & \\
\hline Coordinate system 0 & \begin{tabular}{l}
- MCS \\
The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. \\
- wCS \\
The park position is specified in the workpiece coordinate system.
\end{tabular} & \\
\hline XP & Park position of tool in X direction (abs) & mm \\
\hline ZP & Park position of tool in Z direction (abs) & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Flush chuck U & \begin{tabular}{l}
Flush counter-spindle chuck \\
- Yes \\
- No
\end{tabular} & \\
\hline DIR U & \begin{tabular}{l}
Direction of rotation \\
- \(\quad\) Q Spindle rotates clockwise \\
- \(\quad \varsigma \quad\) Spindle rotates counter-clockwise \\
- \(\quad \otimes \quad\) Spindle does not rotate
\end{tabular} & \\
\hline S & Spindle speed - (only when the spindle rotates) & rev/min \\
\hline <1 & Angular offset & Degrees \\
\hline Z1 & Transfer position (abs.) & \\
\hline ZR U & \begin{tabular}{l}
Position, feedrate reduction (abs or inc) \\
Position from which a reduced feedrate is used.
\end{tabular} & \\
\hline FR & Reduced feedrate & mm/rev \\
\hline Fixed stop & \begin{tabular}{l}
Travel to fixed stop \\
- Yes \\
The counter-spindle stops at a defined distance away from transfer position Z1 and then traverses with a defined feedrate up to the fixed stop. \\
- No \\
The counter-spindle traverses to the transfer position Z1.
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Function, withdrawing & & \\
\hline Withdraw blank & \begin{tabular}{l}
- From main spindle \\
The blank is withdrawn from the main spindle \\
- From counter-spindle The blank is withdrawn from the counter-spindle
\end{tabular} & \\
\hline Also take zero pointU & \begin{tabular}{l}
Also take zero point \\
- Yes \\
- No
\end{tabular} & \\
\hline \begin{tabular}{l}
Work offset U \\
- only when for "withdraw NP" "yes"
\end{tabular} & \begin{tabular}{l}
Work offset in which the coordinate system offset by Z 1 must be saved. \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57
\end{tabular} & \\
\hline Z1 & Amount by which the workpiece is withdrawn from the main spindle (inc) & \\
\hline F & Feedrate & mm/min \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Machining side function & & \\
\hline Machining & \begin{tabular}{l}
Selection of the spindle for machining: \\
- Main spindle Machining on the main spindle \\
- Counter-spindle Machining on the counter-spindle
\end{tabular} & \\
\hline Work offset \(\mathbf{U}\) & \begin{tabular}{l}
Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z , must be saved: \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57 \\
- ...
\end{tabular} & \\
\hline Write to the work offset U & \begin{tabular}{l}
- Yes \\
The \(Z\) value of the work offset can be directly written to the input screen form. \\
- No \\
The actual \(Z\) value of the work offset is used.
\end{tabular} & \\
\hline ZV - only for work offset write "yes" & \begin{tabular}{l}
- Offset \(Z=0\) (abs) \\
- Workpiece zero is offset in \(Z\) direction (inc, the sign is also evaluated)
\end{tabular} & mm \\
\hline Park counter-spindle for machining with main spindle & \begin{tabular}{l}
- Yes \\
The counter-spindle is traversed to the park position. \\
- No \\
The counter-spindle is not traversed.
\end{tabular} & \\
\hline Z4P - for machining with main spindle & Park position of the counter-spindle (abs); MCS & mm \\
\hline Z4W - for machining with counter-spindle & Machining position of the counter-spindle (abs); MCS & mm \\
\hline
\end{tabular}

\subsection*{9.7.18 Machining with fixed counterspindle}

If your lathe is equipped with a second spindle, which is setup as a counterspindle and cannot be traversed, then the workpieces must be manually reclamped


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Machining with main spindle and counterspindle}

For instance, a new blank can be clamped in the main spindle, and a blank that has already been machined at the front can be clamped in the counterspindle. With the ShopTurn program, initially the workpiece is machined in the main spindle, and then the rear side of the workpiece, already machined at the front, is machined in the counter spindle.

\section*{Note}

\section*{Various workpieces}

You have the option of machining two different workpieces at the main spindle and counterspindle.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline Function & You can select one of the following functions: & \\
\hline & Qront face & \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline Function, front face & Work offset for machining the next front face: & \\
\hline Work offset \(\boldsymbol{U}\) & - Basic reference & \\
& - G54 & \\
& - G55 & \\
& - G56 & \\
& - G57 & \(\ldots\) \\
& \\
\hline
\end{tabular}
9.7 Additional cycles and functions in ShopTurn
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Function, rear face & & \\
\hline Work offset \(\cup\) & \begin{tabular}{l}
Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z , must be saved: \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57
\end{tabular} & \\
\hline Write to the work offset & \begin{tabular}{l}
- Yes \\
The \(Z\) value of the work offset can be directly written to the input screen form. \\
- No \\
The actual \(Z\) value of the work offset is used.
\end{tabular} & \\
\hline ZV (abs) - only for work offset. write "yes" & \(Z\) value of the work offset. & mm \\
\hline ZV (inc) & Workpiece zero is offset in Z direction (the sign is also evaluated) & mm \\
\hline
\end{tabular}

\section*{See also}

Program header (Page 250)
Program header with multi-channel data (Page 628)

\section*{Multi-channel machining}

\subsection*{10.1 Multi-channel view}

The multi-channel view allows you to simultaneously view several channels in the following operating areas:
- "Machine" operating area
- "Program" operating area

\subsection*{10.1.1 Multi-channel view in the "Machine" operating area}

With a multi-channel machine, you have the option of simultaneously monitoring and influencing the execution of several programs.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Displaying the channels in the "Machine" operating area}

In the "Machine" operating area, you can display 2-4 channels simultaneously.
Using the appropriate settings, you can define the sequence in which channels are displayed. Here, you can also select if you wish to hide a channel.

\section*{Note}

The "REF POINT" operating mode is shown only in the single-channel view.

\section*{Multi-channel view}

2-4 channels are simultaneously displayed in channel columns on the user interface.
- Two windows are displayed one above the other for each channel.
- The actual value display is always in the upper window.
- The same window is displayed for both channels in the lower window.
- You can select the display in the lower window using the vertical softkey bar.

The following exceptions apply when making a selection using the vertical softkeys:
- The "Actual values MCS" softkey switches over the coordinate systems of both channels.
- The "Zoom actual value" and "All G functions" softkeys switch into the single-channel view.

\section*{Single-channel view}

If, for your multi-channel machine, you always only wish to monitor one channel, then you can set a permanent single-channel view.

\section*{Horizontal softkeys}
- Block search

When selecting the block search, the multi-channel view is kept. The block display is displayed as search window.
- Program control

The "Program Control" window is displayed for the channels configured in the multichannel view. The data entered here applies for these channels together.
- If you press an additional horizontal softkey in the "Machine" operating area (e.g. "Overstore", "Synchronized actions"), then you change into a temporary single-channel view. If you close the window again, then you return to the multi-channel view.

\section*{Switching between single- and multi-channel view}
\begin{tabular}{|c|c|}
\hline \(\underline{M}\) & Press the <MACHINE> key in order to briefly switch between the singleand multi-channel view in the machine area. \\
\hline MACHIE & \\
\hline
\end{tabular}

Press the <NEXT WINDOW> key in order to switch between the upper and lower window within a channel column.

\section*{Editing a program in the block display}


If there is not sufficient space, you switch over into the single-channel view.

\section*{Running-in a program}

You select individual channels to run-in the program at the machine.

\section*{Requirement}
- Several channels have been set-up.
- The setting " 2 channels", " 3 channels" or "4 channels" is selected.

\section*{Displaying/hiding a multi-channel view}

1. Select the "Machine" operating area
2. Select the "JOG", "MDA" or "AUTO" mode.
3. Press the menu forward key and the "Settings" softkey.
4. Press the "Multi-channel view" softkey.
5. In the window "Settings for Multi-Channel View" in the selection box "View", select the required entry (e.g. "2 channels") and define the channels as well as the sequence in which they are to be displayed.
In the basic screen for the "AUTO", "MDA" and JOG" operating modes, the upper window of the left-hand and right-hand channel columns are occupied by the actual value window.

\section*{T,F,S}

Press the "T,F,S" softkey if you wish to view the "T,F,S" window. The "T,F,S" window is displayed in the lower window of the left- hand and right-hand channel column.
Note:
The "T,F,S" softkey is present only for smaller operator panels, i.e. up to OP012.

\section*{See also}

Setting the multi-channel view (Page 615)

\subsection*{10.1.2 Multi-channel view for large operator panels}

On the OP015 and OP019 operator panels as well as on the PC, you have the option of displaying up to four channels next to each one. This simplifies the creation and run-in for multi-channel programs.

\section*{Constraints}
- OP015 with a resolution of \(1024 \times 768\) pixels: up to three channels visible
- OP019 with a resolution of \(1280 \times 1024\) pixels: up to four channels visible
- The operation of a OP019 requires a PCU50.5

\section*{3- or 4-channel view in the "Machine" operating area}

Use the multi-channel view settings to select the channels and specify the view.
\begin{tabular}{|c|c|}
\hline Channel view & Display in the "Machine" operating area \\
\hline 3-channel view & \begin{tabular}{l}
The following windows are displayed one above the other for each channel: \\
- Actual Value window \\
- T,F,S window \\
- Block Display window \\
Selecting functions \\
- The T,F,S window is overlaid by pressing one of the vertical softkeys.
\end{tabular} \\
\hline 4-channel view & \begin{tabular}{l}
The following windows are displayed one above the other for each channel: \\
- Actual Value window \\
- G functions (the "G functions" softkey is omitted). "All G functions" is accessed with the Menu forward key. \\
- T,S,F window \\
- Block Display window \\
Selecting functions \\
- The window showing the \(G\) codes is overlaid if you press one of the vertical softkeys.
\end{tabular} \\
\hline
\end{tabular}

\section*{Toggling between the channels}

Press the <CHANNEL> key to toggle between the channels.


Press the <NEXT WINDOW> key to toggle within a channel column between the three or four windows arranged one above the other.

\section*{Note}

\section*{2-channel display}

Unlike the smaller operator panels, the T,F,S window is visible for a 2-channel view in the "Machine" operating area.

\section*{Program operating area}

You can display as many as ten programs next to each other in the editor.

\section*{Displaying a program}

You can define the width of the program in the Editor window using the settings in the editor. This means that you can distribute programs evenly - or you can widen the column with the active program .

\section*{Channel status}

When required, channel messages are displayed in the status display.
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{10.1.3 Setting the multi-channel view}
\begin{tabular}{|l|l|}
\hline Setting & Meaning \\
\hline View & \begin{tabular}{l} 
Here, you specify how many channels are displayed. \\
\\
\\
\\
\\
\\
\(\bullet\) \\
• 1 channel \\
- 2 channels
\end{tabular} \\
\hline \begin{tabular}{l} 
Channel selection and \\
sequence \\
(for "2-4 channels" \\
view)
\end{tabular} & \begin{tabular}{l} 
You specify which channels in which sequence are displayed in the multi- \\
channel view.
\end{tabular} \\
\hline \begin{tabular}{l} 
Visible \\
(for "2-4 channels" \\
view)
\end{tabular} & \begin{tabular}{l} 
Here, you specify which channels are displayed in the multi-channel view. \\
You can quickly hide channels from the view.
\end{tabular} \\
\hline
\end{tabular}

\section*{Example}

Your machine has 6 channels.
You configure channels 1-4 for the multi-channel view and define the display sequence (e.g. 1,3,4,2).

In the multi-channel view, for a channel switchover, you can only switch between the channels configured for the multi-channel view; all others are not taken into consideration. Using the <CHANNEL> key, advance the channel in the "Machine" operating area - you obtain the following views: Channels "1" and "3", channels "3" and "4", channels "4" and "2". Channels " 5 " and " 6 " are not displayed in the multi-channel view.

In the single-channel view, toggle between all of the channels (1...6) without taking into account the configured sequence for the multi-channel view.

Using the channel menu, you can always select all channels, also those not configured for multi-channel view. If you switch to another channel, which is not configured for the multichannel view, then the system automatically switches into the single-channel view. There is no automatic switchback into the multi-channel view, even if a channel is again selected, which has been configured for multi-channel view.

\section*{Procedure}

\section*{Machine}

2. Select the "JOG", "MDA" or "AUTO" mode.
3. Press the menu forward key and the "Settings" softkey.
1. Select the "Machine" operating area.
4. Press the "Multi-channel view" softkey. The "Settings for Multi-Channel View" window is opened.
5. Set the multi-channel or single-channel view and define which channels are to be seen in the "Machine" operating area - and in the editor - in which sequence.

\subsection*{10.2 Multi-channel support}

\subsection*{10.2.1 Working with several channels}

\section*{Multi-channel support}

SINUMERIK Operate supports you when generating the program, the simulation and when running-in a program on multi-channel machines.


\section*{Software options}

For the multi-channel functionality and support, i.e. for generating and editing synchronized programs in the multi-channel editor as well as the block search, you require the "programSYNC" option.


\section*{Software options}

You require the "ShopMill/ShopTurn" option to generate and edit ShopTurn machining step programs.

\section*{Note}

\section*{Execution and simulation}

The execution and simulation for multi-channel programming does not function if the programs and the job list are on an external storage medium, e.g. on the local drive.

\section*{Multi-channel view}

With the multi-channel view, you have the option of viewing several channels in parallel on the display. This means that for multi-channel machines, the execution of several programs simultaneously started - can be monitored and controlled.

\section*{View of the channels}

In the window "Settings for multi-channel view" or "Settings for multi-channel functionality", you set which channels are important for the program execution and which channels are displayed simultaneously. In so doing, you also define the channel sequence.

\section*{Note}

\section*{Hidden channels}

Hidden channels still belong to the group of channels that are handled together. They are only temporarily excluded from the multi-channel view.

In the multi-channel editor, you have the option of simultaneously opening several programs and editing them. In this case, the multi-channel editor supports you regarding program synchronization from a time perspective.

\subsection*{10.2.2 Creating a multi-channel program}

All of the programs involved in a multi-channel machining operation are combined in one workpiece.
In a job list, enter the program names, define the program type - G code or ShopTurn program - and assign these to a channel.

\section*{Machine manufacturer}

If you only program \(G\) code programs, then you can switch-out the multi-channel view.
Please refer to the machine manufacturer's specifications.

\section*{Precondition}

> - "programSYNC" option

\section*{Procedure}


\section*{New}
programSSWC
Multi-

1. Select the "Program Manager" operating area.
2. Press the "NC" softkey and select the "Workpieces" folder.
3. Press the "New" and "programSYNC multi-channel" softkeys. The "New job list" window opens.
4. Enter the required name and press the "OK" softkey.

The "Job list *.JOB" window opens.
For each channel that has been set-up, the window has one line to input or select the assigned program.
5. Position the cursor on the required channel line, enter the required program name and select the program type (G code or ShopTurn).
6. Press the "OK" softkey.

The "Multi-channel data" parameter screen opens in the editor.

\subsection*{10.2.3 Entering multi-channel data}

In the parameter screen "Multi-channel data", enter the following data, which applies for all channels for \(G\) code and ShopTurn programs:
- Measurement unit
- Work offset (e.g. G54)
- Z value of the work offset (optional)
- Blank
- Spindle chuck data (optional)
- Speed limitation
- Counter-spindle data, if required
- Counter-spindle with/without mirroring (for G code)

\section*{Machine manufacturer}

If you are working with pure \(G\) code programming, it is possible that the parameter screen "Multi-channel data" does not open.

Please refer to the machine manufacturer's instructions.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Measurement unit
\[
U
\] & Selecting the measurement unit & \begin{tabular}{l}
mm \\
inch
\end{tabular} \\
\hline \multicolumn{3}{|l|}{Main spindle} \\
\hline Work offset & Selecting the work offset & \\
\hline Write to the work offset U & \begin{tabular}{l}
- Yes \\
Parameter ZV is displayed \\
- No \\
Parameter ZV is not displayed
\end{tabular} & \\
\hline ZV & \begin{tabular}{l}
\(Z\) value of the work offset \\
For G54, the \(Z\) value is entered into the work offset. \\
Note: \\
Please refer to the machine manufacturer's instructions.
\end{tabular} & \\
\hline Blank U & \begin{tabular}{l}
- Tube \\
- Cylinder \\
- Polygon \\
- Centered cuboid
\end{tabular} & \\
\hline XA & Outside diameter \(\varnothing\) - for tube and cylinder & mm \\
\hline XI & Inside diameter (abs) or wall thickness (inc) - only for tube & mm \\
\hline ZA & Initial dimension & mm \\
\hline
\end{tabular}
10.2 Multi-channel support
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& \mathrm{zl} \\
& 0
\end{aligned}
\] & Final dimension (abs) or final dimension in relation to ZA (inc) & \\
\hline \[
\begin{aligned}
& \hline \mathrm{ZB} \\
& \mathrm{O}
\end{aligned}
\] & Machining dimension (abs) or machining dimension in relation to ZA (inc) & mm \\
\hline N & Number of edges - only for polygon & \\
\hline \[
\begin{aligned}
& \text { SW or L } \\
& \mathrm{O}
\end{aligned}
\] & Width across flats or edge length - only for polygon & mm \\
\hline W & Width of the blank - only for centered cuboid & mm \\
\hline L & Length of the blank - only for centered cuboid & mm \\
\hline S & Speed limitation of the main spindle & \(\mathrm{rev} / \mathrm{min}\) \\
\hline Spindle chuck data & \begin{tabular}{l}
- Yes \\
You enter spindle chuck data in the program. \\
- No \\
Spindle chuck data is transferred from the setting data. Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline Spindle chuck data & \begin{tabular}{l}
- Only chuck \\
You enter spindle chuck data in the program. \\
- Complete \\
You enter tailstock data in the program. \\
Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline ZC & - The main spindle chuck dimensions - (only for spindle chuck data "yes") & mm \\
\hline ZS & - Stop dimension of the main spindle - (only for spindle chuck data "yes") & mm \\
\hline ZE & - Jaw dimension of the main spindle for jaw type 2 - (only for spindle chuck data "yes") & mm \\
\hline Counter-spindle & & \\
\hline Spindle chuck data & \begin{tabular}{l}
- Yes \\
You enter spindle chuck data in the program. \\
- No \\
Spindle chuck data is transferred from the setting data. Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline Spindle chuck data & \begin{tabular}{l}
- Only chuck \\
You enter spindle chuck data in the program. \\
- Complete You enter tailstock data in the program. \\
Note: \\
- Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Jaw type & \begin{tabular}{l}
Selecting the jaw type of the counter-spindle. Dimensions of the front edge or stop edge - (only if spindle chuck data "yes") \\
- Jaw type 1 \\
- Jaw type 2
\end{tabular} & \\
\hline ZC & The counter-spindle chuck dimensions - (only for spindle chuck data "yes") & mm \\
\hline ZS & Stop dimension of the counter-spindle - (only for spindle chuck data "yes") & mm \\
\hline ZE & Jaw dimension of the counter-spindle for jaw type 2 - (only for spindle chuck data "yes") & mm \\
\hline XR & Tailstock diameter - (only for spindle chuck data "complete" and tailstock that has been set up) & mm \\
\hline ZR & Tailstock length - (only for spindle chuck data "complete" and tailstock that has been set up) & mm \\
\hline Mirroring Z & \begin{tabular}{l}
- Yes \\
Mirroring is used when machining on the \(Z\) axis \\
- No \\
Mirroring is not used when machining on the \(Z\) axis
\end{tabular} & \\
\hline Work offset & Selecting the work offset & \\
\hline Write to the work offset & \begin{tabular}{l}
- Yes \\
Parameter ZV is displayed \\
- No \\
Parameter ZV is not displayed
\end{tabular} & \\
\hline ZV & \begin{tabular}{l}
\(Z\) value of the work offset \\
The value exceeds the \(Z\) value in the selected work offset.
\end{tabular} & \\
\hline Blank & \begin{tabular}{l}
- Tube \\
- Cylinder \\
- Polygon \\
- Centered cuboid
\end{tabular} & \\
\hline ZA & Initial dimension & mm \\
\hline ZI & Final dimension (abs) or final dimension in relation to ZA (inc) & mm \\
\hline ZB & Machining dimension (abs) or machining dimension in relation to ZA (inc) & mm \\
\hline XA & Outside diameter - (only for tube and cylinder) & mm \\
\hline XI & Inside diameter (abs) or wall thickness (inc) - (only for tube) & mm \\
\hline N & Number of edges - (only for polygon) & \\
\hline SW or L
\[
0
\] & Width across flats or edge length - (only for polygon) & mm \\
\hline W & Width of the blank - (only for centered cuboid) & mm \\
\hline L & Length of the blank - (only for centered cuboid) & mm \\
\hline S & Speed limitation of the counter-spindle & \(\mathrm{rev} / \mathrm{min}\) \\
\hline
\end{tabular}

Turning

\section*{Procedure}
1. You have created programs for the multi-channel machining in the job list and the parameter screen "Multi-channel data" is open in the editor.
2. Enter the data for the cross-channel data.
3. Press the "Accept" softkey.

The multi-channel editor is opened and displays the programs that have been created.

The cursor is positioned on an empty line before the cycle for the job list (CYCLE208). You can also enter a comment.
After the cycle call, enter the required initializations for the G code program and add the program code.

\subsection*{10.2.4 Multi-channel functionality for large operator panels}

For the large OP 015, OP 019 operator panels as well as at the PC, there is more space in the "Machine", "Program" and "Parameter" operating areas - as well as in all lists - to display NC blocks, tools etc.

Further, you also have the option of simultaneously displaying more than 2 channels.
This makes it easier for you to identify the machine situation for machines with 3 and more channels. Further, it makes it simpler for you to generate and run-in three or four-channel programs.


\section*{Software options}

If you require the option "programSYNC" for the views described here.

\section*{Supplementary conditions}
- OP 015, OP 019 or PC with a display of at least \(1280 \times 1024\) pixels
- For operating an OP 019, at least one NCU720.2 or 730.2 with 1 GB of RAM or a PCU50 is required

\section*{3 / 4-channel view in the "Machine" operating area}

If you have selected 3 channels via settings, then 3 or 4 channel columns are displayed next to one another.
\begin{tabular}{|c|c|}
\hline Channel view & Display in the "Machine" operating area \\
\hline 3-channel view & \begin{tabular}{l}
The following windows are displayed one above the other for each channel: \\
- Actual value window \\
- T,F,S window \\
- Block display window
\end{tabular} \\
\hline 4-channel view & \begin{tabular}{l}
The following windows are displayed one above the other for each channel: \\
- Actual value window \\
- T,S,F window \\
- G functions (the "G functions" softkey is omitted) \\
- Block display window
\end{tabular} \\
\hline
\end{tabular}

\section*{Displaying functions}
\begin{tabular}{|l|l|}
\hline Channel view & Display in the "Machine" operating area \\
\hline & Selection using vertical softkeys: \\
\hline 3-channel view & • The T,F,S window is overlaid by pressing one of the vertical softkeys. \\
\hline 4-channel view & \begin{tabular}{l} 
• The window showing the G codes is overlaid if you press one of the verti- \\
cal softkeys.
\end{tabular} \\
\hline & Selection using horizontal softkeys: \\
\hline \begin{tabular}{l} 
3-channel view / \\
4 channel view
\end{tabular} & • The block display is overlaid if you press the "Overstore" horizontal softkey \\
\hline & - The block display is overlaid by pressing the softkey "block search". \\
\hline & • The window is shown as a pop-up if you press the "Prog. control" softkey. \\
\hline & \begin{tabular}{l} 
If you press one of the horizontal softkeys in the "JOG" operating mode \\
(e.g. "T,S,M", "Meas. tool", "Positions" etc.), then you change into the sin- \\
gle-channel view.
\end{tabular} \\
\hline
\end{tabular}

\section*{Toggling between the channels}

Press the <CHANNEL> key to toggle between the channels.


Press the <NEXT WINDOW> key to toggle within a channel column between the three or four windows arranged one above the other.

\section*{Note}

\section*{2-channel display}

Contrary to the smaller operator panels, in the "Machine" operating area, for a 2-channel view, the TFS window is visible.

\section*{Program operating area}

In the editor, just as many programs are displayed next to one another as in the "Machine" operating area.

\section*{Displaying a program}

You can define the width of the program in the editor window using the settings in the editor. This means that you can distribute programs evenly - or you can display the column with the active program wider.

\section*{Simulation}

In the simulation window, actual values are displayed for a maximum of 4 channels simultaneously as well as the actual block.

You can toggle between displaying the traversing paths and the channel zero point using the "Channel+" and "Channel-" softkeys.

Axes, which are located in several channels, are displayed grayed-out if the setpoint comes from a different channel.

\section*{Channel status}

When required, channel messages are displayed in the status display.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{10.2.5 Editing the multi-channel program}

\subsection*{10.2.5.1 Changing the job list}

You now have the option to change the composition of the programs and/or the assignment of the channel and program in a job list.

\section*{Precondition}
- "programSYNC" option

\section*{Procedure}

1. Select the "Program Manager" operating area.
2. Select where the multi-channel program should be archived
3. Position the cursor in the "Workpieces" folder on a job list and press the "Open" softkey.
The window "Job list * JOB" is opened and the program assignment to the channels is displayed.
4. Select the channel to which you wish to assign a new program and press the softkey "Select program".
The "Program" window is opened and displays all of the programs created in the workpiece.
- OR -

Press the "Open job list" softkey.

\subsection*{10.2.5.2 Editing a G code multi-channel program}

\section*{Editing a G code multi-channel program}

\section*{Precondition}
- The "programSYNC" option is set.
- In order to display the machining at the counterspindle at the correct position in the simulation, the linear axis of the counterspindle must be positioned before CYCLE208 (multi-channel data).

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Procedure}

\section*{Open}
1. Position the cursor in the "Workpieces" folder on a job list and press the "Open" softkey.
Note:
If the cursor is located on a workpiece, then a search is made for a job list with the same name.
The "Job list ..." window opens and the program assignment to the channels is displayed.
2. Press the "OK" softkey.

The programs are displayed next to one another in the editor.
3. Position the cursor on the first block of the program (multi-channel data) and press the <Cursor right> key.

The parameter screen "Multi-channel data" is opened.
4. Enter the required values if you wish to change cross-channel data.

\section*{Adding multi-channel data in a G code program}

You have the possibility of adding the multi-channel cycle (CYCLE208) subsequently.

\section*{Procedure}
1. The double editor is opened and the cursor is positioned in the \(G\) code program.
2. Press the "Misc." and "Multi-channel data" softkeys. The "Call multi-channel data" input window opens.

A field for specifying the job list appears. This field is read-only.
3. Press the "Accept job list" softkey. The name of the job list is entered in the field.
4. Press the "Accept" softkey.

CYCLE208 is taken over into the program. The name of the job list is indicated in brackets.

\section*{Modify blank}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Data for & \begin{tabular}{l}
Here, you specify the spindle selected for the blank. \\
- Main spindle \\
- Counterspindle
\end{tabular} & \\
\hline Blank U & \begin{tabular}{l}
The following blanks can be selected: \\
- Tube \\
- Cylinder \\
- Polygon \\
- Centered cuboid \\
- Delete
\end{tabular} & \\
\hline W & Width of the blank - (only for centered cuboid) & mm \\
\hline L & Length of the blank - (only for centered cuboid) & mm \\
\hline N & Number of edges - (only for polygon) & \\
\hline SW or L & Width across flats or edge length - (only for polygon) & \\
\hline ZA & Initial dimension & \\
\hline \[
\begin{gathered}
\mathrm{ZI} \\
\mathrm{U}
\end{gathered}
\] & Final dimension (abs) or final dimension in relation to ZA (inc) & \\
\hline \[
\begin{gathered}
\mathrm{ZB} \\
U
\end{gathered}
\] & Machining dimension (abs) or machining dimension in relation to ZA (inc) & \\
\hline XA & Outside diameter - (only for tube and cylinder) & mm \\
\hline \[
\begin{aligned}
& \mathrm{XI} \\
& \mathrm{U}
\end{aligned}
\] & Inside diameter (abs) or wall thickness (inc) - for tube only & mm \\
\hline
\end{tabular}

\section*{Procedure}
1. The double editor is opened and the cursor is positioned in the \(G\) code program.
2. Press the "Misc." and "Blank" softkeys.

The "Blank Input" window opens.
3. Select the desired blank and enter the corresponding values.
4. Press the "Accept" softkey.

\subsection*{10.2.5.3 Editing a ShopTurn multi-channel program}

\section*{Precondition}

The "programSYNC" option is set.

\section*{Procedure}
1. Position the cursor in the "Workpieces" folder on a job list and press the "Open" softkey.
Note:
If the cursor is located on a workpiece, then a search is made for a job list with the same name.

The "Job list ..." window opens and the program assignment to the channels is displayed.
2. Press the "OK" softkey.

The programs are displayed next to one another in the editor.
3. Open the program header if you wish to define cross-program entries.

\section*{Program header with multi-channel data}

In the program header, set the parameters, which are effective for the complete program.
You have the following options to save cross-program data:
- You can enter values in a common data set for the main and counterspindle
- You can enter values for the main and/or counterspindle
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Multi-channel data & \begin{tabular}{l}
Yes \\
Name of the job list in which the channel data are saved.
\end{tabular} & \\
\hline Data for U & \begin{tabular}{l}
- Main+counterspindle \\
All values for the main and counterspindle are saved in one data set \\
- Main spindle \\
Data set for the main spindle \\
- Counterspindle \\
Data set for the counterspindle \\
Note: \\
If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.
\end{tabular} & \\
\hline \multirow[t]{2}{*}{Retraction U} & The retraction area indicates the area outside of which collision-free traversing of the axes must be possible. & \\
\hline & \begin{tabular}{l}
- simple \\
- Extended \\
- all
\end{tabular} & \\
\hline \[
\begin{aligned}
& \mathrm{XRA} \\
& U
\end{aligned}
\] & Retraction plane \(X\) external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & \\
\hline \[
\begin{aligned}
& \text { XRI } \\
& U
\end{aligned}
\] & - not for "basic" retraction retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc) - not for "pipe" blank & mm \\
\hline \[
\begin{aligned}
& \text { ZRA } \\
& U
\end{aligned}
\] & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline \[
\begin{aligned}
& \mathrm{ZRI} \\
& 0 \\
& \hline
\end{aligned}
\] & Retraction plane Z rear - only for retraction "all" & mm \\
\hline Tailstock
\[
U
\] & \begin{tabular}{l}
- Yes \\
- No
\end{tabular} & \\
\hline XRR & \begin{tabular}{l}
Retraction plane tailstock - only "Yes" for tailstock \\
For "Main+counterspindle", the tailstock only refers to the main spindle (tailstock on the counterspindle side)
\end{tabular} & mm \\
\hline Tool change point U & \begin{tabular}{l}
Tool change point, which must be approached by the revolver with its zero point. \\
- WCS (Workpiece Coordinate System) \\
- MCS (Machine Coordinate System) \\
Notes \\
- The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. \\
- Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip.
\end{tabular} & \\
\hline XT & Tool change point \(\mathrm{X} \varnothing\) & mm \\
\hline ZT & Tool change point \(Z\) & mm \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Data for
\[
0
\] & \begin{tabular}{l}
If several spindles have been set up, the program can operate at both spindles. \\
Select the 2nd spindle \\
- Main spindle \\
- Counterspindle \\
- Empty \\
The program only operates at one spindle
\end{tabular} & \\
\hline Retraction U & \begin{tabular}{l}
The retraction area indicates the area outside of which collision-free traversing of the axes must be possible. \\
- simple \\
- extended - (not for a pipe blank) \\
- all
\end{tabular} & \\
\hline XRA U & Retraction plane \(X\) external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & \\
\hline XRI U & - only for a pipe blank retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc) & mm \\
\hline ZRA U & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & \(\mathrm{rev} / \mathrm{min}\) \\
\hline ZRI U & Retraction plane Z rear - only for retraction "all" & mm \\
\hline Tailstock U & \begin{tabular}{l}
- Yes \\
- No
\end{tabular} & \\
\hline XRR & Retraction plane tailstock - only "Yes" for tailstock & mm \\
\hline Tool change point \(\mathbf{U}\) & \begin{tabular}{l}
Tool change point, which must be approached by the revolver with its zero point. \\
- WCS (Workpiece Coordinate System) \\
- MCS (Machine Coordinate System) \\
Notes \\
- The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. \\
- Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip.
\end{tabular} & \\
\hline XT & Tool change point \(\mathrm{X} \varnothing\) & mm \\
\hline ZT & Tool change point \(Z\) & mm \\
\hline SC & \begin{tabular}{l}
The safety clearance defines how close the tool can approach the workpiece in rapid traverse. \\
Note \\
Enter the safety clearance without sign into the incremental dimension.
\end{tabular} & mm \\
\hline Mach. direction of rotation 0 & \begin{tabular}{l}
Milling direction \\
- Conventional \\
- Climbing
\end{tabular} & \\
\hline
\end{tabular}

\section*{Program header without multi-channel data}

If a program is to be executed through one channel, then deselect multi-channel data. You then have the option of entering cross-program values into the program header as usual.
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline Multi-channel data & - No This is & ly possible if you are not using a job list. & \\
\hline Measurement unit U & \multicolumn{2}{|l|}{\begin{tabular}{l}
The setting of the measurement unit in the program header only refers to the position data in the actual program. \\
All other data, such as feedrate or tool offsets, are entered in the unit of measure that you have set for the entire machine.
\end{tabular}} & \begin{tabular}{l}
mm \\
inch
\end{tabular} \\
\hline Data for 0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Main+counterspindle \\
All values for the main and counterspindle are saved in one data set \\
- Main spindle \\
Data set for the main spindle \\
- Counterspindle \\
Data set for the counterspindle \\
If the machine does not have a counterspindle, then the entry field "Data for" is not applicable.
\end{tabular}} & \\
\hline Work offset U & \multicolumn{2}{|l|}{The work offset in which the zero point of the workpiece is saved. You can also delete the default value of the parameter if you do not want to specify a work offset.} & \\
\hline write to 0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Yes \\
Parameter ZV is displayed \\
- No \\
Parameter ZV is not displayed
\end{tabular}} & \\
\hline ZV & \multicolumn{2}{|l|}{\begin{tabular}{l}
\(Z\) value of the work offset \\
For G54, the \(Z\) value is entered into the work offset. \\
Note: \\
Please observe the machine manufacturer's data
\end{tabular}} & \\
\hline Blank U & \multicolumn{2}{|l|}{Define the form and dimensions of the workpiece:} & \\
\hline & \multicolumn{2}{|l|}{- Cylinder} & \\
\hline & XA & Outer diameter \(\varnothing\) & mm \\
\hline & \multicolumn{2}{|l|}{- Polygon} & \\
\hline & N & Number of edges & \\
\hline & SW / L U & Width across flats/edge length & \\
\hline & \multicolumn{2}{|l|}{- Centered cuboid} & \\
\hline & W & Width of blank & mm \\
\hline & L & Length of blank & mm \\
\hline & \multicolumn{2}{|l|}{- Tube} & \\
\hline & XA & Outer diameter \(\varnothing\) & mm \\
\hline & U & Inner diameter \(\varnothing\) (abs) or wall thickness (inc) & mm \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline & ZA & Initial dimension & mm \\
\hline & ZIU & Final dimension (abs) or final dimension in relation to ZA (inc) & mm \\
\hline & ZB U & Machining dimension (abs) or machining dimension in relation to ZA (inc) & \\
\hline \begin{tabular}{l}
Retraction \\
U
\end{tabular} & \multicolumn{2}{|l|}{The retraction area indicates the area outside of which collision-free traversing of the axes must be possible.} & \\
\hline & \multicolumn{2}{|l|}{- simple} & \\
\hline & XRA U & Retraction plane X external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & mm \\
\hline & XRIU & \begin{tabular}{l}
- only for "pipe" blank \\
Retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc)
\end{tabular} & mm \\
\hline & ZRAU & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline & \multicolumn{2}{|l|}{- extended - not for a "pipe" blank} & mm \\
\hline & XRAU & Retraction plane X external \(\varnothing\) (abs) or retraction plane \(X\) referred to \(X A\) (inc) & mm \\
\hline & XRIU & Retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc) & mm \\
\hline & ZRA U & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline & \multicolumn{2}{|l|}{- all} & mm \\
\hline & XRAU & Retraction plane X external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & \\
\hline & XRI U & Retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc) & mm \\
\hline & ZRA U & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline & ZRI U & Retraction plant \(Z\) rear & mm \\
\hline \begin{tabular}{l}
Tailstock \\
U
\end{tabular} & \multicolumn{2}{|l|}{\begin{tabular}{l}
- Yes \\
- No
\end{tabular}} & \\
\hline XRR & \multicolumn{2}{|l|}{Retraction plane tailstock - only "Yes" for tailstock} & mm \\
\hline Tool change point U & \multicolumn{2}{|l|}{\begin{tabular}{l}
Tool change point, which must be approached by the revolver with its zero point. \\
- WCS (Workpiece Coordinate System) \\
- MCS (Machine Coordinate System) \\
Notes \\
- The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. \\
- Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip.
\end{tabular}} & \\
\hline XT & \multicolumn{2}{|l|}{Tool change point \(\mathrm{X} \varnothing\)} & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline ZT & Tool change point Z & mm \\
\hline S & Spindle speed & \(\mathrm{rev} / \mathrm{min}\) \\
\hline Spindle chuck data & \begin{tabular}{l}
- Yes \\
You enter spindle chuck data in the program. \\
- No \\
Spindle chuck data are transferred from the setting data. \\
Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline Spindle chuck data & \begin{tabular}{l}
- Only chuck \\
You enter spindle chuck data in the program. \\
- Complete \\
You enter tailstock data in the program. \\
Note: \\
Please observe the machine manufacturer's instructions.
\end{tabular} & \\
\hline Data for U & \begin{tabular}{l}
If several spindles have been set up, the program can operate at both spindles. Select the 2nd spindle \\
- Main spindle \\
- Counterspindle \\
- Empty \\
The program only operates at one spindle
\end{tabular} & \\
\hline Retraction \(\cup\) & \begin{tabular}{l}
The retraction area indicates the area outside of which collision-free traversing of the axes must be possible. \\
- simple \\
- extended - not for a "pipe" blank \\
- all
\end{tabular} & \\
\hline XRA U & Retraction plane X external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & mm \\
\hline XRI U & - for "basic" retraction, only for a "pipe" blank retraction plane X internal \(\varnothing\) (abs) or retraction plane X referred to XI (inc) & mm \\
\hline ZRA U & Retraction plane \(Z\) front (abs) or retraction plane Z referred to ZA (inc) & mm \\
\hline ZRI U & Retraction plane Z rear - only for retraction "all" & mm \\
\hline Tailstock \(\cup\) & \begin{tabular}{l}
- Yes \\
- No
\end{tabular} & \\
\hline XRR & Retraction plane tailstock - only "Yes" for tailstock & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Tool change point \(U\) & \begin{tabular}{l}
Tool change point, which must be approached by the revolver with its zero point. \\
- WCS (Workpiece Coordinate System) \\
- MCS (Machine Coordinate System) \\
Notes \\
- The tool change point must be far enough outside the retraction area that it is not possible for any tool to protrude into the retraction area while the revolver is moving. \\
- Ensure that the tool change point is relative to the zero point of the revolver and not the tool tip.
\end{tabular} & \\
\hline XT & Tool change point \(\mathrm{X} \varnothing\) & mm \\
\hline ZT & Tool change point \(Z\) & mm \\
\hline S & Spindle speed & \(\mathrm{rev} / \mathrm{min}\) \\
\hline SC & \begin{tabular}{l}
The safety clearance defines how close the tool can approach the workpiece in rapid traverse. \\
Note \\
Enter the safety clearance without sign into the incremental dimension.
\end{tabular} & mm \\
\hline Mach. direction of rotation & \begin{tabular}{l}
Milling direction \\
- Conventional \\
- Climbing
\end{tabular} & \\
\hline
\end{tabular}

\section*{Changing program settings}

Under settings, the settings for the main and/or counterspindle can be changed while the program is being executed.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline Data for & \begin{tabular}{l}
You define the spindle selection for processing the data here - (this is only available if the machine has a counterspindle) \\
- Main spindle \\
Data set for the main spindle \\
- Counterspindle \\
Data set for the counterspindle \\
- Main+counterspindle \\
All values for the main and counterspindle are saved in one data set
\end{tabular} & \\
\hline Retraction U & \begin{tabular}{l}
Lift mode \\
- simple \\
- Extended \\
- all \\
- Empty
\end{tabular} & \\
\hline XRA U & Retraction plane X external \(\varnothing\) (abs) or retraction plane X referred to XA (inc) & mm \\
\hline XRI U & \begin{tabular}{l}
Retraction plane \(X\) internal \(\varnothing\) (abs) or retraction plane \(X\) referred to XI (inc) \\
- (only for retraction "extended" and "all")
\end{tabular} & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline ZRA U & Retraction plane \(Z\) front (abs) or retraction plane \(Z\) referred to \(Z A\) (inc) & mm \\
\hline ZRI & Retraction plane Z rear - (only for retraction "all") & mm \\
\hline Tailstock & \begin{tabular}{l}
Yes \\
- Tailstock is displayed for simulation / simultaneous recording \\
- When approaching/retracting, the retraction logic is taken into account No
\end{tabular} & \\
\hline XRR & Retraction plane - (only "Yes" for tailstock) & mm \\
\hline Tool change point & \begin{tabular}{l}
Tool change point \\
- WCS (Workpiece Coordinate System) \\
- MCS (Machine Coordinate System) \\
- Empty
\end{tabular} & \\
\hline XT & Tool change point X & mm \\
\hline ZT & Tool change point \(Z\) & mm \\
\hline SC & \begin{tabular}{l}
Safety clearance (inc) \\
Acts in relation to the reference point. The direction in which the safety clearance is active is automatically determined by the cycle.
\end{tabular} & mm \\
\hline S1 & Maximum speed, main spindle & rev/min \\
\hline Machining direction & \begin{tabular}{l}
Milling direction: \\
- Climbing \\
- Conventional \\
- Empty
\end{tabular} & \\
\hline
\end{tabular}

\section*{Procedure}
1. The ShopTurn program has been created.
2. Position the cursor at the location in the program where settings must be changed.
3. Press the "Various" and "Settings" softkeys. The "Settings" input window opens.

\subsection*{10.2.5.4 Creating a program block}

In order to structure programs in order to achieve a higher degree of transparency when preparing for the synchronized view, you have the possibility of combining several blocks (G code and/or ShopTurn machining steps) to form program blocks.

\section*{Structuring programs}
- Before generating the actual program, generate a program frame using empty blocks.
- By forming blocks, structure existing G code or ShopTurn programs.

1) Cross-channel data from the "Multi-channel data" window.
2) "Multi-channel programs" program opened in channel 1 .
3) "Multi-channel programs" program opened in channel 2.
4) Actual program with block name "Stock removal".

The program block has been opened and an Addit. run-in code has been activated.
The program block is assigned to the main spindle.
5) Program block with block name "Peripheral surface".

The program block is closed. In order to identify whether an Addit. run-in code is activated or automatic retraction is activated, open the block using the <Cursor right> key.
6) Program block with block name "Face milling".

The program block is assigned to the counterspindle. The spindle assignment is color coded in order to make a distinction.

Figure 10-1 Structured programs in the multi-channel editor

\section*{Settings for a program block}
\begin{tabular}{|c|c|}
\hline Display & Meaning \\
\hline Text & Block designation \\
\hline Spindle & \begin{tabular}{l}
- S1 \\
- S2 \\
Spindle assignment. Defines at which spindle a program block is to be executed.
\end{tabular} \\
\hline Addit. run-in code & \begin{tabular}{l}
- Yes \\
For the case that the block is not executed, as the specified spindle should not be considered when running in, then it is possible to temporarily activate what is known as "Addit. run-in code". \\
- No
\end{tabular} \\
\hline Automat. retraction & \begin{tabular}{l}
- Yes \\
Block start and block end are moved to the tool change point, i.e. the tool is retracted. \\
- No
\end{tabular} \\
\hline
\end{tabular}

\section*{Procedure}
```

NC

```

Build group
1. Select the "Program manager" operating area.
2. Select the storage location and create a program or open a program. The program editor opens.
3. Select the required program blocks, which you wish to combine to form a block.
4. Press the "Build group" softkey.

The "Build group" window is opened.
5. Enter a designation for the block, assign the spindle, if required, select the Addit. run-in code and the automatic retraction and then press the "OK" softkey.

\section*{Opening and closing blocks}
1. Position the cursor on the desired program block.
2. Press the <+> key or the <Cursor right> key.
\(+\)
...

3. Press the <-> key or the <Cursor left> key.

The block is closed again.
...


\section*{Open} blocks
4. Press the "Open all blocks" softkey if you wish to display all the blocks.
5. Press the "Close all blocks" softkey if you wish to close all the blocks again.

\section*{Shifting blocks}

You have the option of using "Select", "Copy", "Cut-out" and "Paste" softkeys to move individual or several blocks within the program.

\subsection*{10.2.6 Setting the multi-channel function}
\begin{tabular}{|c|c|}
\hline Setting & Meaning \\
\hline View & \begin{tabular}{l}
Here, you specify how many channels are displayed. \\
- 1 channel \\
- 2 channels \\
- 3 channels \\
- 4 channels
\end{tabular} \\
\hline \begin{tabular}{l}
Channel selection and sequence \\
(for "2-4 channels" view)
\end{tabular} & Here, you create the channel group, i.e. you specify which channels and in which sequence are displayed in the multi-channel view. \\
\hline \begin{tabular}{l}
Visible \\
(for "2-4 channels" view)
\end{tabular} & Here, you specify which channels are displayed in the two-channel view. \\
\hline
\end{tabular}

\section*{Precondition}


\section*{Software options}

You require the "programSYNC" option to generate and edit synchronized programs in the multi-channel editor as well as for the multi-channel functions in the "Machine" operating area.

\section*{Example}

Your machine has 6 channels.
You configure channels 1-4 for the multi-channel view and define the display sequence (e.g. 1,3,4,2).

Using the <CHANNEL> key, advance the channel in the "Machine" operating area - you obtain the following views: Channels " 1 " and " 3 ", channels " 3 " and " 4 ", channels " 4 " and " 2 ". Channels " 5 " and " 6 " are not displayed in the multi-channel view.
In the single-channel view, toggle between all of the channels (1...6) without taking into account the configured sequence for the multi-channel view.

\section*{Procedure}

1. Select the "Machine" operating area.

\section*{Machine}

2. Select the operating mode "JOG", "MDA" or "AUTO".
3. Press the menu forward key and the "Settings" softkey.


Cottings

Multichan. function
4. Press the "Multi-channel function" softkey.

The "Settings for the multi-channel functionality" window is opened.

\subsection*{10.2.7 Synchronizing programs}

Using the synchronized view, you have the possibility of obtaining an overview of the time sequence of a program. In this case, program instructions are evaluated to coordinate channels and are arranged in parallel in the editor view.
As a result of the synchronized view of the programs, you can easily identify at which locations the programs are synchronized in the various channels.

\section*{Synchronizing commands}
\begin{tabular}{|l|l|}
\hline Commands & Meaning \\
\hline START & Starts another program \\
\hline WAITM & Sets a mark and waits for the specified channels (with exact stop) \\
\hline WAITMC & Sets a mark and waits for the specified channels (without exact stop) \\
\hline WAITE & Waits for the end of program of the specified channels \\
\hline SETM & Sets a mark \\
\hline CLEARM & Clears a mark \\
\hline GET & Fetches an axis \\
\hline RELEASE & Releases an axis \\
\hline
\end{tabular}

\section*{Note}

\section*{Error identification when synchronizing the programs}

The error can only be identified if the appropriate program is displayed. If a WAIT mark involves a channel to which a program is not assigned in the job list, then this is marked as having an error.

If you have enabled the synchronized view, then the following symbols are displayed at the top right in the program title bar:
\begin{tabular}{|l|l|}
\hline Symbol & Meaning \\
\hline ©! & \begin{tabular}{l} 
Synchronized view: Error \\
Errors were identified after enabling "synchronized view" or new "syn- \\
chronization" (e.g. WAIT marks not found in other programs).
\end{tabular} \\
\hline ©/ & \begin{tabular}{l} 
Synchronized view: checked \\
No error was detected after enabling "synchronized view" or new "syn- \\
chronization".
\end{tabular} \\
\hline
\end{tabular}

Wait marks can also be used within blocks.
- Closed block
- If there is a WAIT mark within a closed block, the clock of this WAIT mark is displayed in front of the block name.
In the synchronized view, the closed block is synchronized.
- If there are several WAIT marks within a closed block, then a clock is displayed in front of the block name. It is yellow, if all clocks of the WAIT marks in the block are yellow, otherwise it is red.
In the synchronized view, the closed block is synchronized to the last WAIT mark in the block.
- Opened block
- If there is a WAIT mark, the clock is displayed in front of the WAIT mark.

In front of the WAIT mark, the clock is displayed in either yellow or red. The program is synchronized at the WAIT mark.
- If there are several WAIT marks, the clocks are displayed in either yellow or red at the WAIT marks. The program is synchronized at the WAIT marks.

\section*{Determining program runtimes}

After a simulation, the required processing time for the program blocks is displayed in the editor. For multi-channel programs, the delay time is displayed at the wait points.

\section*{Spindle transfer between the channels}

If you use the spindles alternating in several channels (e.g. main and counterspindle), then it may be necessary to exit the active plane "Face C" (TRANSMIT) or "Peripheral C" (TRACYL):


1 Press the menu forward key and the "Straight/Circle" softkey.

Tool
2. Press the "Tool" softkey.
3. In the "Plane selection" field, select the "Turning" setting (TRAFOOF).

\section*{Procedure}

1. Select the required job list.
2. Press the "Open" softkey. The job list is opened in the editor.
3. Press the ">>" and "View" softkeys.
4. Press the softkey "Synchron. view".
5. Press the "Synchronizing" softkey if you wish to update the view after changes.
6. Press the "Open all blocks" softkey if you wish to view all of the program blocks in the synchronized display.
7. Press the "Close all blocks" softkey, if you wish to close the blocks to achieve a higher degree of transparency.
8. Select the required program.
9. Press the "Full screen" softkey.

The two-channel display is changed into a single-channel display and the selected program is displayed using the complete editor window.

\section*{See also}

Optimizing the machining time (Page 644)

\subsection*{10.2.8 Insert WAIT marks}

To synchronize programs via several channels, you have the option of inserting WAIT marks.

In the wait mark you define the type, and depending on the synchronizing command, the number and the channels to be synchronized.

\section*{WAIT marks}

In the "WAIT mark" window, the following synchronizing commands are available:
\begin{tabular}{|l|l|}
\hline Type & Meaning \\
\hline WAITM & Sets a mark and waits for the specified channels (with exact stop) \\
\hline WAITMC & Sets a mark and waits for the specified channels (without exact stop) \\
\hline WAITE & \begin{tabular}{l} 
Waits for the end of program of the specified channels (current channel not \\
specified) \\
Note: Numbers or variables cannot be entered.
\end{tabular} \\
\hline SETM & \begin{tabular}{l} 
Sets a mark \\
Note: Channels cannot be entered
\end{tabular} \\
\hline CLEARM & \begin{tabular}{l} 
Clear mark in own channel \\
Note: Channels cannot be entered.
\end{tabular} \\
\hline
\end{tabular}

Note
Inserting WAIT marks in additional programs
Using the "Copy" and "Paste" softkeys, you have the option to insert the blocks with WAIT marks into other programs for other channels.

\section*{Procedure}
1. The multi-channel program has been created.
2. Place the cursor at the location of the program where you want to set the WAIT mark.
3. Press the "Various", "Continue", and "WAIT mark" softkeys.

The "WAIT mark" window is opened.
4. In the "Type" selection box, select the desired WAIT mark.
5. If necessary, in the entry field, enter the desired number.
6. In the associated selection field of a channel, select "Yes" if the WAIT mark for this channel is to be valid.
7. Press the "Accept" softkey.

The WAIT mark is displayed in the program as machining step.
Using "Cursor right", as usual, open the machining step in the editor.

\section*{Editing a WAIT mark}


Press the keys <SHIFT> and <INSERT > to open the WAIT mark and edit it.

\subsection*{10.2.9 Optimizing the machining time}

After simulating a program, the machining time is displayed for the blocks.
For a multi-channel display, the wait times that occur are displayed at the wait points (wait marks). This provides you with an overview of the time sequence of the program and you can perform the first optimization runs.

\section*{Shifting blocks}

You have the option of shifting program blocks to longer wait points - assuming that the machining technology permits this - therefore reducing the machining time.
1. Select the block that you wish to shift.

Mark

Copy

Cut

Paste
2. Press the "Mark" softkey.
3. Press the "Copy" softkey if you wish to repeat the machining step at another position.
- OR -

Press the "Cut-out" softkey if you wish to execute the machining step at another position.
4. Position the cursor at the required program position and press the "Paste" softkey.

Insert the program block at the required position.

\section*{Time-related view}

In the time-related view, the wait times at the WAIT marks as well as the required machining times at the machining sections are displayed. If you change the program, time data is still displayed at the corresponding wait marks or the corresponding blocks dimmed.
The determined times are lost if you exit the editor using the "Close" softkey or you open or select another program. The times must be determined again through simulation.

Time bars


Figure 10-2 Time synchronous view

\subsection*{10.2.10 Automatic block building}

\subsection*{10.2.10.1 Creating automated program blocks}

With the "Automatic block building" function, you have a convenient option to automatically split an existing program subsequently into the desired blocks.

\section*{Rules for creating the blocks}

You define the rules for creating the blocks in the configuration file.
You can define the following program steps for better processing:
- Combine the tools used in the program into blocks
- Define cycle types
- A main block opens a new block at the top level. The block itself is entered as first block in the block. The replacement text from the configuration file is taken as block text.
- A subblock is inserted in the currently opened block of a main block. Blocks that are not classified by the configuration file are combined in one block at the second block level. This block contains the block text "Free DIN code".
- Two levels are available for the block building to structure the program.

\section*{Note}

The automatic block building can only be performed once.
If you use the function a second time, a message is displayed that no blocks can be inserted as some already exist.

In order to create blocks in the already converted program, use the "Build block" softkey in the editor.

\section*{References}

For additional information, please refer to the following references:
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

\section*{Precondition}

The configuration file seditor.ini has been installed.

\section*{Procedure}

Program manager


Automatic block form.

1. Select the "Program manager" operating area.
2. Position the cursor on the required main program (*.mpf) or on a job list (*.job).
3. Press the ">>" and "Automat." block building" softkey.

You are prompted as to whether you want to automatically insert blocks in the program.
4. Press the "OK" softkey to confirm the conversion.

\subsection*{10.2.10.2 Editing a converted program}

\section*{Precondition}

You have used the "Automatic block formation" softkey to convert a program into a structured program.

\section*{Procedure}
1. Open a converted program.

\section*{Opening and closing blocks}


Open blocks

\section*{Close} blocks
2. Press the ">>" and "View" softkeys.

Uiew
3. Press the "Open blocks" softkey.

All blocks on the first level are opened. key again.
4. Press the "Close blocks" softkey.

To open the blocks on the second level, press the "Open blocks" soft-

Opened blocks on the second level are closed.
To close the blocks on the first level, press the "Close blocks" softkey again.

Note: If all blocks on the second level are closed, the blocks on the first level are closed the first time the softkey is pressed.

\section*{Creating additional blocks manually in two planes}

Build group

Accept

Open
blocks
Build group

Accept
2. Mark the program records that you want to block subsequently to form a block and press the "Form block" softkey.
3. Enter a designation for the block in the "Form new block" window, assign the spindle, if required, select the additional run-in code and the automatic retraction, and press the "Accept" softkey.
4. If you want to group additional program records within the block to form a block, expand the block, e.g. with the "Expand blocks" softkey.
5. Mark the required program records within the block and press the "Form block" softkey.
6. Enter the required data in the "Form new block" window and press the "Accept" softkey.

\section*{Note:}

If a spindle has already been assigned to the outer block, you can no longer assign any spindle to the inner block and vice versa.

\section*{Creating a block of the first plane with a higher-level block}

Build group

Accept
2. Mark the required block that does not contain any other blocks and press the "Form block" softkey.
3. Enter the required data in the "Form new block" window and press the "Accept" softkey.

\section*{Note:}

If a spindle has already been assigned to the marked block, you cannot assign any spindle to the newly created block.

\subsection*{10.2.11 Simulating machining}

\subsection*{10.2.11.1 Simulation}

For classic lathes with main and counterspindle, up to two channels can be simultaneously simulated.

You have the option of executing the programs together before the actual machining. In this case, start, stop and reset - as well as the functions to control the program, act simultaneously on all of the simulated channels.

\section*{Correction of the zero offsets for multi-channel data}

If you use multi-channel data in the simulation, then the zero offsets are temporarily corrected so that they match the blank and the spindle chuck data.

\section*{Preconditions}

The function of the individual spindles and special axes, must be specified in the display machine data set-up for the purpose.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Tool paths}

Only the tool paths of the presently selected channel are displayed.

\section*{Procedure}
1. Start the simulation.

2. Press the "Main spindle" softkey if you wish to view the main spindle. - OR -

Contrebroche

\section*{Main} spindle

Counterspindle


Channel +

\section*{Channel -}

\subsection*{10.2.11.2 Different workpiece views for multi-channel support}

In the graphical display, you can choose between different views so that you constantly have the best view of the current workpiece machining, or in order to display details or the overall view of the finished workpiece.

The following views are available:
- Side view
- Half section
- Front view
- 3D view
- 2-window

\section*{Procedure}

Uiews

Side
view

Half cut
view

Face
view

\section*{3D}
view

2 windows
1. Start the simulation.
2. Press the "Views" softkey.
3. Press the "Side view" softkey if you wish to view the workpiece in the Z-X plane.
- OR -

Press the "Half section" softkey if you wish to view the workpiece cut in the Z-X plane.
- OR -

Press the "Front view" softkey if you wish to view the workpiece in the \(X-Y\) plane.
- OR -

Press the "3D view" softkey if you wish to view the workpiece as a three-dimensional model.
- OR -

Press the 2-window softkey if you wish to simultaneously view the side view (left-hand window) and front view (right-hand window) of the workpiece.

\section*{Note}

The 2-window view cannot be activated if you have simultaneously selected the main spindle and the counterspindle.

\subsection*{10.2.12 Display/edit the multi-channel functionality in the "Machine" operating area}

\subsection*{10.2.12.1 Running-in a program}

You have various options to run-in programs.

\section*{Running-in channel-by-channel}

Select the channels that you wish to process using the "Running-in" function in the "Program control" window. The channels not selected here are brought into the "Program test (PRT)" state. As a consequence, the channels are only calculated, but are not processed.

No M and auxiliary functions or tool functions are output. Spindle commands are only output for selected spindles.

\section*{Running-in spindle-by-spindle}

Only machining operations are executed for the spindle selected under program control / running-in. When programming, using the block formation, allocate the corresponding machining operations to a spindle.
When generating a block, a complete block can be assigned to a spindle. For the case that the block is not executed, as the specified spindle should not be considered when running in, then it is possible to temporarily activate what is known as an "Addit. run-in code".

Software options
You require the "programSYNC" option for cross-channel program control.

\section*{Precondition}
- Multi-channel machine
- You have selected the multi-channel view via "Settings for channel functionalities".

Procedure
1. Select the "AUTO" operating area.
2. Press the "Prog. ctrl." softkey.

Prog.
cntrl.

\section*{General}

Run in

The "Program control-General" window appears on the screen.
3. Press the "Run-in" softkey.

The "Program control - Running-in" window is displayed.
4. Select the channels and the associated spindles to run-in the program.

\subsection*{10.2.12.2 Block search and program control}

You define a group of channels that belong to one another from the "Settings for Multichannel Functionality" window. Here, you specify which channel numbers should be displayed for a multi-channel view.

This group results in a common behavior for a block search and for program control.

\section*{Vertical softkeys for a block search}
- The "Block search" and "Search mode" functions act on all channels, which you configured for the multi-channel view.
- All other vertical softkeys (e.g. "Search text", "Interrupt position", etc.). act on the current program.
If you selected the single-channel view in the multi-channel function settings, then all of the actions only act on the actual channel.

\section*{Software options}

You require the "programSYNC" option for the multi-channel block search and the multi-channel program control in the multi-channel editor.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}


\section*{Blk sear.} mode


Search
for text

\section*{Interrupt}
point

Search
pointer
1. Press the "Block search" softkey.

Press the "Search mode" softkey. The "Search Mode" window is displayed.
2. Select the required mode for the channel group.
3. Press the "OK" softkey to confirm the setting.
4. Press the "Search text" softkey if you wish to enter the search target using a text.
- OR -

Press the "Interrupt position" softkey if you wish to search for the search target using a program interrupt.
- OR -

Press the "Search pointer" softkey if you wish to enter a search target that you cannot enter using the editor (e.g. no interrupt position, search target is located in the subprogram) and enter the program path.
5. Press the "Start search" softkey.

The search starts.
All channels of the group are started corresponding to the search mode that has been set.

During the block search, the search states are displayed in a message window (e.g. "Block search running").
You receive a message if the search target has been reached - or an error message if the search target was not found.

\section*{Note}

\section*{Search target via "Search pointer"}

The "Search Pointer" window is displayed - as usual - over the complete lower part of the screen. The upper contents of the window remain multi-channel.

The actual channel is displayed in the title bar of the "Search Pointer" window.
The "Delete search pointer" also only acts on this channel.

\section*{Note}

\section*{Search target via "Search text"}

If, for a block search, you approach the program position using "Search text", please note that a search is only made in the selected channel column.

\subsection*{10.2.13 Stock removal with 2 synchronized channels}

With multi-channel lathes, you have the option of simultaneously machining with 2 channels (4 axes).
The tools are located in front of and behind the center of rotation, and machine the same workpiece. For sufficiently long cuts, these are simultaneously used at different depth infeeds, with a specified offset.

The advantage is that you reduce your production time. Further, when machining, the cutting forces are better distributed.

\section*{Technological function}

2-channel machining is available for the "Parallel stock removal" technology.

\section*{Leading channel/following channel}

Define the leading channel when programming workpieces that you wish to machine utilizing multiple channels. As a consequence, you define the channel-specific machine and setting data that are used to create the stock removal programs.
Turning always starts in the leading channel. The second channel, also known as the following channel, starts with the second depth infeed as soon as the leading channel has removed stock by a specified "stock removal distance" (offset). If cuts are obtained that are shorter than this offset, then machining is only carried out in the leading channel.

\section*{Offset}

If the offset is not equal to zero, then a dedicated stock removal program is generated for each channel. Automatically generated WAIT commands are used to synchronize both programs.

If the offset is equal to zero, then only one stock removal program is generated. This program is started in the leading channel. Axis coupling is used to generate the motion of the following channel.

For multi-channel lathes, the name of the generated stock removal program is extended by identifier "_C" with a subsequent two-digit channel number.

\section*{Roughing}

When roughing, typically there is an offset between the channels when machining (channel offset DCH).
- When the parameter is positive, then the leading channel machining is executed before the following channel by the channel offset.
- When the parameter is negative, then the machining sequence is interchanged. The program generated for the leading channel is in this case, started in the following channel.
- When the parameter is equal to zero, then the workpiece is machined without offset (refer to the paragraph "Finishing balance cutting").

\section*{Note \\ Constant cutting rate}

When using a constant cutting rate, ensure that the offset (DCH) is not too high.
Tools
The difference between the two cutting radii of the tools must not exceed the allowance.

\section*{Finishing}

2-channel finish cutting is only possible with Balance Cutting.
Only one machining program is generated for the leading channel, and this is executed there. The motion in the following channel is coupled to this motion.

\section*{Balance Cutting}

Balance Cutting involves symmetrical machining in both channels. In so doing, the cutting forces are evenly distributed between both sides of the workpiece. The tools are opposite one another at the same infeed depth.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications on the subject of axis coupling.

\section*{Note}

\section*{Tools}

The tool radii or cutting plate widths of the plunging tools must be the same size.

\section*{General conditions}
- For internal machining operations ensure that collisions do not occur between the tools and/or tool carriers. The control cannot ensure this, as it has no information about the mechanical dimensions.
- Before machining for the first time, position the tools so that they are in approximately in the same position.
- The tools must have the same cutting-edge position and cutting direction in both channels.
- The following parameters and settings must be the same in both channels:
- the machining plane (G17, G18, G19),
- the technology (stock removal),
- the machining (roughing, finishing),
- the dimension unit (metric, inch).

\subsection*{10.2.13.1 Job list}

One example for a ShopTurn and one for a G Code job list are described in the following.
The clock symbol in the icon of the program blocks indicates that internal WAIT commands are used to synchronize the channels involved. The cycles perform the synchronization.

\section*{ShopTurn machining schedule}

1) Machining program in the leading channel
2) Machining program in the following channel
3) Contour and machining steps in the leading channel have a bracket symbol
4) 2-channel steps; implicit WAIT marks are identified using a clock symbol.
5) The contour is only defined in the leading channel; this is the reason that there is no bracket here.

Figure 10-3 View of a 2-channel stock removal program in ShopTurn

\section*{Program view in the G code}

1) Machining program in the leading channel
2) Machining program in the following channel
3) 2-channel stock removal cycles, which contain implicit WAIT marks, are identified by a preceding clock symbol.

Figure 10-4 View of a 2-channel stock removal program in G code

\subsection*{10.2.13.2 Stock removal}

\section*{Calling up the cycle}

\section*{Precondition}
- "programSYNC" option

\section*{Procedure}
1. The part program or ShopTurn program to be processed has been created and you are in the editor.
2. Press the "Contour turning" softkey.

Stock remoual
3. Press the "Stock removal" softkey.

Only those parameters relevant for stock removal with 2 synchronized channels of a workpiece are subsequently described. All other parameters are described in Chapter "Contour turning".

\section*{Parameters}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
Machining \\
0
\end{tabular} & \begin{tabular}{l}
- \(\nabla 2\) CHAN \\
2-channel roughing \\
- \(\quad\) VD 2 CHAN \\
2-channel finishing
\end{tabular} & \\
\hline Channel 0 & Defines whether the currently set channel is the leading or following channel. All calculations are carried out in the leading channel. & \\
\hline \begin{tabular}{l}
Number \\
0
\end{tabular} & \begin{tabular}{l}
Selecting the number of the partner channel if there are more than 2 channels available. \\
- If, for the channel parameter, "Leading channel" is selected, then the channel number of the following channel must be parameterized here. \\
- If, for the channel parameter, "Following channel" is selected, then the channel number of the leading channel must be parameterized here.
\end{tabular} & \\
\hline \[
\begin{aligned}
& \mathrm{DCH} \\
& \mathrm{O}
\end{aligned}
\] & \begin{tabular}{l}
Channel offset \\
- For positive parameters, then the leading channel machining is executed before the following channel by the channel offset. \\
- For a negative parameter, when executing the program, the leading or following channel property is interchanged. \\
- If the parameter is equal to zero, then balance cutting (axis coupling) is used.
\end{tabular} & mm (inch) \\
\hline
\end{tabular}

\subsection*{10.2.14 Synchronizing a counterspindle}

For multi-channel machines, the counter-spindle steps must be synchronized across all channels.

You program handling the counter-spindle in one channel. This channel controls the motion of the counter-spindle and adapts the zero offset of the channel.

In the synchronization step, the other channels park their tools in order to avoid collisions. The zero offset is also accepted.
\begin{tabular}{|c|c|c|c|}
\hline \multicolumn{3}{|l|}{NC/WKS/GEGENSP_2KAN/GEGENSP_2KAN.JOB} & \\
\hline \% & \multicolumn{2}{|l|}{Mehrkanaldaten} & \% \\
\hline CHAK1 GEGEKSP_2KAK_1 & & CHAN2 GEGENSP_2KAH_2 & \\
\hline P H10 Program header & \(\rightarrow\) & P H10 Program header & \\
\hline Atw H20 Stock remoual & I=SCHRL & (-) H11 uaitm( \(1,1,2)\) II & \\
\hline ( ) H21 waitm (1, 1, 2) 川 & & L-d H38 Counterspindle (2) & Complete \\
\hline L-d H30 Counterspindle (1) & Synchror & Flo F 48 Cutoff & T=EIHSTI \\
\hline () Whith ( \(1,1,2)\) II & & 发e H58 Drilling centric & T=BOHRE \\
\hline ENO End of program & & (-) uaitm (1, 1, 2) & \\
\hline & & END End of program & \\
\hline
\end{tabular}

1 Synchronization step
2 Counter-spindle step
The following counter-spindle steps, which implicitly contain WAIT marks, are identified using a symbol:
- Pulling (take zero point =yes)
- Machining side
- Complete transfer
- Synchronizing
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline Function & You can select one of the following functions: & \\
U & - Synchronizing & \\
& - Complete transfer & \\
& - Gripping \\
& - Withdrawing & \\
\hline Workpiece machining & - Machining side & \\
& - Counter-spindle in main spindle & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Synchronization function & Synchronizes with the counter-spindle in the other channel. & \\
\hline Coordinate system & \begin{tabular}{l}
- MCS \\
The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. \\
- Workpiece coordinate system The park position is specified in the workpiece coordinate system.
\end{tabular} & \\
\hline XP & Park position of tool in X direction (abs) & mm \\
\hline ZP & Park position of tool in Z direction (abs) & mm \\
\hline Complete transfer function & Gripping & \\
\hline \begin{tabular}{l}
Coordinate system \\
U
\end{tabular} & \begin{tabular}{l}
- MCS \\
The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. \\
- WCS \\
The park position is specified in the workpiece coordinate system.
\end{tabular} & \\
\hline XP & Park position of tool in X direction (abs) & mm \\
\hline ZP & Park position of tool in Z direction (abs) & mm \\
\hline Flush chuck \(\cup\) & \begin{tabular}{l}
Flush counter-spindle chuck \\
- Yes \\
- No
\end{tabular} & \\
\hline DIR U & \begin{tabular}{l}
Direction of rotation \\
- \(\quad\) Q Spindle rotates clockwise \\
- \(\quad \varsigma \quad\) Spindle rotates counter-clockwise \\
- \(\quad\) Spindle does not rotate
\end{tabular} & \\
\hline S & Spindle speed - (only when the spindle rotates) & \(\mathrm{rev} / \mathrm{min}\) \\
\hline <1 & Angular offset & Degrees \\
\hline Z1 & Transfer position (abs.) & \\
\hline ZR U & \begin{tabular}{l}
Position, feedrate reduction (abs or inc) \\
Position from which a reduced feedrate is used.
\end{tabular} & \\
\hline FR & Reduced feedrate & \(\mathrm{mm} / \mathrm{rev}\) \\
\hline Fixed stop & \begin{tabular}{l}
Travel to fixed stop \\
- Yes \\
The counter-spindle stops at a defined distance away from transfer position Z1 and then traverses with a defined feedrate up to the fixed stop. \\
- No \\
The counter-spindle traverses to the transfer position Z 1 .
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Complete transfer function & Withdrawing & \\
\hline Withdraw blank & \begin{tabular}{l}
Withdraw complete blank: \\
- Yes \\
- No
\end{tabular} & \\
\hline F & Feed (only when "yes" for "withdraw blank") & \(\mathrm{mm} / \mathrm{min}\) \\
\hline Cutting-off cycle & \begin{tabular}{l}
Cutting-off cycle in the following block \\
- Yes \\
- No
\end{tabular} & \\
\hline Complete transfer function & Rear side (for main spindle in counter-spindle) & \\
\hline Work offset U & \begin{tabular}{l}
Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z , must be saved: \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57
\end{tabular} & \\
\hline Write to the work offset & \begin{tabular}{l}
- Yes \\
The \(Z\) value of the work offset can be directly written to the input screen form. \\
- No \\
The actual \(Z\) value of the work offset is used.
\end{tabular} & \\
\hline ZV - only for work offset write "yes" & \begin{tabular}{l}
- Offset Z = 0 (abs) \\
- Workpiece zero is offset in \(Z\) direction (inc, the sign is also evaluated)
\end{tabular} & mm \\
\hline Z4W & Supplementary axis of the counter-spindle machining position (abs); MCS & mm \\
\hline Complete transfer function & Front side (for counter-spindle in main spindle) & \\
\hline Work offset U & \begin{tabular}{l}
Work offset in which the coordinate system, which was shifted according to ZP and by ZV as well as mirrored in Z, must be saved: \\
Basic reference \\
G54 \\
G55 \\
G56 \\
G57
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Work offset write to & \begin{tabular}{l}
- Yes \\
The \(Z\) value of the work offset can be directly written to the input screen form. \\
- No \\
The actual \(Z\) value of the work offset is used.
\end{tabular} & \\
\hline ZV - only for work offset write "yes" & \begin{tabular}{l}
- Z value of the work offset (abs) \\
- Workpiece zero offset in the \(Z\) direction (inc); the sign is also evaluated.
\end{tabular} & mm \\
\hline Z4P & Park position of the counter-spindle supplementary axis (abs); MCS & mm \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Function, gripping & Teaching in the park position and angular offset is possible & \\
\hline Gripping a blank U & \begin{tabular}{l}
- With counter-spindle \\
The blank is gripped with the counter-spindle \\
- With main spindle \\
The blank is gripped with the main spindle
\end{tabular} & \\
\hline Also take zero point U & \begin{tabular}{l}
- Yes \\
- No
\end{tabular} & \\
\hline \begin{tabular}{l}
Work offset \(U\) \\
- for take zero point "yes"
\end{tabular} & \begin{tabular}{l}
Work offset in which the coordinate system offset by Z 1 must be saved. Basic reference \\
G54 \\
G55 \\
G56 \\
G57 \\
...
\end{tabular} & \\
\hline Coordinate system U & \begin{tabular}{l}
- MCS \\
The park position is specified in the machine coordinate system. Teaching in the park position and angular offset is only possible in the machine coordinate system. \\
- WCS \\
The park position is specified in the workpiece coordinate system.
\end{tabular} & \\
\hline XP & Park position of tool in X direction (abs) & mm \\
\hline ZP & Park position of tool in Z direction (abs) & mm \\
\hline Flush chuck \(U\) & \begin{tabular}{l}
Flush counter-spindle chuck \\
- Yes \\
- No
\end{tabular} & \\
\hline DIR U & \begin{tabular}{l}
Direction of rotation \\
- \(\quad 2 \quad\) Spindle rotates clockwise \\
- \(\quad \varsigma \quad\) Spindle rotates counter-clockwise \\
- \(\quad \otimes \quad\) Spindle does not rotate
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline S & Spindle speed - (only when the spindles rotate) & rev/min \\
\hline\(\alpha 1\) & Angular offset & Degrees \\
\hline Z1 & Transfer position (abs.) & \begin{tabular}{l} 
Position, feedrate reduction (abs or inc) \\
Position from which a reduced feedrate is used.
\end{tabular} \\
\hline ZR U & Reduced feedrate & \\
\hline FR & \begin{tabular}{l} 
Travel to fixed stop \\
Pixed \\
stop
\end{tabular} & \begin{tabular}{l} 
The counter-spindle stops at a defined distance away from transfer position \\
Z1 and then traverses with a defined feedrate up to the fixed stop. \\
No
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Function, withdrawing & & \\
\hline Withdraw blank & \begin{tabular}{l}
- From main spindle \\
- From counter-spindle
\end{tabular} & \\
\hline Also take zero pointU & \begin{tabular}{l}
Also take zero point \\
- Yes \\
- No
\end{tabular} & \\
\hline \begin{tabular}{l}
Work offset U \\
- for take zero point "yes"
\end{tabular} & \begin{tabular}{l}
Work offset in which the coordinate system offset by \(\mathrm{Z1}\) must be saved. \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57
\end{tabular} & \\
\hline Z1 & Amount by which the workpiece is withdrawn from the main spindle (inc) & \\
\hline F & Feedrate & mm/min \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameters & Description & Unit \\
\hline Machining side function & & \\
\hline Spindle selection & \(\bullet\) Main spindle & \\
\(U\) & Machining along the main spindle & \\
& \begin{tabular}{l} 
Counter-spindle \\
Machining along the counter-spindle
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline Parameters & Description & Unit \\
\hline Work offset U & \begin{tabular}{l}
Work offset in which the coordinate system, which was shifted according to ZW and by ZV as well as mirrored in Z , must be saved: \\
- Basic reference \\
- G54 \\
- G55 \\
- G56 \\
- G57
\end{tabular} & \\
\hline Work offset write to U & \begin{tabular}{l}
- Yes \\
The \(Z\) value of the work offset can be directly written to the input screen form. \\
- No \\
The actual \(Z\) value of the work offset is used.
\end{tabular} & \\
\hline ZV - only for work offset write "yes" & \begin{tabular}{l}
- Offset Z=0 \\
- Workpiece zero is offset in \(Z\) direction (inc, the sign is also evaluated)
\end{tabular} & \\
\hline Park counter-spindle & \begin{tabular}{l}
Traverse counter-spindle to park position - for "main spindle" selection \\
- Yes \\
- No
\end{tabular} & \\
\hline Z4P - for park counterspindle "yes" & Park position of the counter-spindle (abs); MCS & mm \\
\hline Z4W & Machining position of the counter-spindle (abs); MCS & mm \\
\hline
\end{tabular}

\section*{Collision avoidance (only 840D sl)}

\subsection*{11.1 Activating collision avoidance}

With the aid of collision avoidance, you can avoid collisions and therefore major damage during the machining of a workpiece or when creating programs.


\section*{Software option}

You require the "Collision avoidance (machine, working area)" software option in order to use this function.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

Collision avoidance is based on a machine model. The kinematics of the machine are described as a kinematic chain. For machine parts to be protected, protection areas are attached to these chains. The geometry of the protection areas is defined using protection area elements. The control then knows how they move in the machine coordinate system depending on the position of the machine axes. You then subsequently define the collision pairs, i.e. two protection areas which are monitored with respect to one another.

The "Collision avoidance" function regularly calculates the clearance from these protection areas. When two protection areas approach one another, and a specific safety clearance is reached, an alarm is displayed and before the corresponding traversing block, the program is stopped and/or the traversing motion is stopped.

\section*{References}

More detailed explanations on the collision avoidance can be found in the following reference:

SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual;
Function Manual, Special Functions (FB3):
- Section: "Kinematic chain (K7)"
- Section: "Geometric machine modeling (K8)"
- Section: "Collision avoidance (K9)"
- Section: "NC/PLC interface signals (Z3)" > "Collision avoidance (K9)"

\section*{Precondition}
- Collision avoidance is setup and an active machine model is available.
- The setting "Collision avoidance" has been selected for the AUTO operating mode or for the JOG and MDA operating modes.

\section*{Procedure}
\[
{ }^{\prime}
\]

Machine

\({ }_{6}\) Simult.
\(=\) record.
Further views

Machine space
2. Press the <AUTO> key.
3. Press the "Sim. rec." softkey.
4. Press the "Other views" and "Machine area" softkeys.

\subsection*{11.2 Set collision avoidance}

Using "Settings", you have the option of separately activating or deactivating the collision monitoring for the Machine operating area (operating modes, AUTO, JOG and MDI) separately for the machine and tools.

Using machine data, you define from which protection level the collision monitoring for the machine or the tool can be activated or deactivated in the operating modes JOG/MDI or AUTO.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.
\begin{tabular}{|l|l|}
\hline Setting & Effect \\
\hline \begin{tabular}{l} 
JOG/MDI operating mode \\
Collision avoidance
\end{tabular} & \begin{tabular}{l} 
They switch the collision monitoring for the operating modes JOG/MDI \\
completely on or off.
\end{tabular} \\
\hline \begin{tabular}{l} 
AUTO mode \\
Collision avoidance
\end{tabular} & \begin{tabular}{l} 
They switch the collision monitoring for the operating mode AUTO \\
completely on or off.
\end{tabular} \\
\hline \begin{tabular}{l} 
JOG/MDI \\
Machine
\end{tabular} & \begin{tabular}{l} 
If the collision monitoring for the JOG/MDI operating modes is activat- \\
ed, then as a minimum, the \\
machine protection areas are monitored. \\
The parameter cannot be changed.
\end{tabular} \\
\hline \begin{tabular}{l} 
AUTO \\
Machine
\end{tabular} & \begin{tabular}{l} 
If the collision monitoring for the AUTO operating mode is activated, \\
then as a minimum, the \\
machine protection areas are monitored. \\
The parameter cannot be changed.
\end{tabular} \\
\hline \begin{tabular}{l} 
JOG/MDI \\
Tools
\end{tabular} & \begin{tabular}{l} 
They switch the collision monitoring of the tool protection areas for the \\
operating modes JOG/MDI on or off.
\end{tabular} \\
\hline AUTO & \begin{tabular}{l} 
They switch the collision monitoring of the tool protection areas for the \\
operating mode AUTO on or off.
\end{tabular} \\
Tools
\end{tabular}

\section*{Procedure}

1. Select the "Machine" operating area.

\section*{Machine}
2. Select the "JOG", "MDI" or "AUTO" mode.

3. Press the menu forward key and the "Settings" softkey.

Settings
```

Collision auoidance

```

4. Press the "Collision avoidance" softkey. The "Collision Avoidance" window opens.
5. In the "Collision avoidance" line for the required operating modes (e.g. for JOG/MDI), select the entry "On" to activate the collision avoidance or "Off" to deactivate collision avoidance.
6. Deactivate the "Tools" checkbox if you only want to monitor the machine protection areas.

\section*{Tool management}

\subsection*{12.1 Lists for the tool management}

All tools and also all magazine locations that have been created or configured in the NC are displayed in the lists in the Tool area.
All lists display the same tools in the same order. When switching between the lists, the cursor remains on the same tool in the same screen segment.

The lists have different parameters and softkey assignments. Switching between lists is a specific change from one topic to the next.
- Tool list

All parameters and functions required to create and set up tools are displayed.
- Tool wear

All parameters and functions that are required during operation, e.g. wear and monitoring functions, are listed here.
- Magazine

You will find the magazine and magazine location-related parameters and functions for the tools / magazine locations here.
- Tool data OEM

This list can be freely defined by the OEM.
Grinding-specific tool data is provided here if you are working with grinding tools.

\section*{Sorting the lists}

You can change the sorting within the lists according to:
- The magazine
- The name (tool identifier, alphabetic)
- The tool type
- The T number (tool identifier, numerical)
- The D number

\section*{Filtering the lists}

You can filter the lists according to the following criteria:
- Only display the first cutting edge
- Only tools that are ready to use
- Only tools that have reached the pre-alarm limit
- Only locked tools
- Only tools with active code

\section*{Search functions}

You have the option of searching through the lists according to the following objects:
- Tool
- Magazine location
- Empty location

\subsection*{12.2 Magazine management}

Depending on the configuration, the tool lists support a magazine management.

\section*{Magazine management functions}
- Press the "Magazine" horizontal softkey to obtain a list that displays tools with magazinerelated data.
- The Magazine / Magazine location column is displayed in the lists.
- In the default setting, the lists are displayed sorted according to magazine location.
- The magazine selected via the cursor is displayed in the title line of each list.
- The "Magazine selection" vertical softkey is displayed in the tool list.
- You can load and unload tools to and from a magazine via the tool list.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{12.3 Tool types}

A number of tool types are available when you create a new tool．The tool type determines which geometry data is required and how it will be computed．

\section*{Tool types}
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－fauorites} \\
\hline Type & Identifier & Tool position \\
\hline 580 & Roughing tool & 《回回回 \\
\hline 518 & Finishing tool & （ \(\square^{4}\) \\
\hline 528 & Plunge cutter & 〈医可耑を \\
\hline 548 & Threading tool & 4 \(\triangle\) P A \\
\hline 558 & Button tool & \(1 \odot \bigcirc \odot \bigcirc\) \\
\hline 568 & Rotary drill & 田禹回田 \\
\hline 588 & 3 turning probe & \(\| \mathrm{c}=8 \mathrm{P}=\) \\
\hline 738 & Stop & 郘品或 \\
\hline 128 & End mill & \(\cdots \geqslant\) \\
\hline 148 & Facing tool & 的空 50 \\
\hline 158 & Side mill & 縣 \(8=0\) \\
\hline 288 & Twist drill & \(\mathbb{\sim} \boldsymbol{\sim}\) \\
\hline 248 & Tap &  \\
\hline & Multitool & 気 \\
\hline
\end{tabular}

Figure 12－1 Example of Favorites list
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－milling cutter} \\
\hline Typ & Identifier & Tool position \\
\hline 100 & Milling tool & \＄\(\|_{4}=\mathbb{\$}\) \\
\hline 110 & Cylindr．ball end & CUD \\
\hline 111 & Conical ball end & くいつ！ \\
\hline 120 & End mill &  \\
\hline 121 & End mill corner round． & ーいつ円 \\
\hline 130 & Angle head cutter &  \\
\hline 131 & Corn．round．ang．hd．cut &  \\
\hline 140 & Facing tool &  \\
\hline 145 & Thread cutter & 成 \\
\hline 150 & Side mill & \(\theta=1108\) \\
\hline 151 & Saw & \(\underline{=1}=\|\) \\
\hline 155 & Bevelled cutter & 以上こに \\
\hline 156 & Beveled cutter corner & CUD \\
\hline 157 & Tap．die－sink．cutter & cUD \\
\hline 160 & Drill\＆thread cut． &  \\
\hline
\end{tabular}

Figure 12－2 Available tools in the＂New Tool－Milling Cutter＂window
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－drill} \\
\hline Typ & Identifier & Tool position \\
\hline 200 & Twist drill & \(\leq \square \leq 0\) \\
\hline 205 & Solid drill & \(\cdots \square\) \\
\hline 210 & Boring bar & \(8^{9} 9\) \\
\hline 220 & Center drill & \(\leq b \leq n\) \\
\hline 230 & Countersink &  \\
\hline 231 & Counterbore & － \(\boldsymbol{y}_{6}\) \\
\hline 240 & Tap & 非醝啡 \\
\hline 241 & Fine tap &  \\
\hline 242 & Tap，Whitworth & 非醝非 \\
\hline 250 & Reamer &  \\
\hline
\end{tabular}

Figure 12－3 Available tools in the＂New Tool－Drill＂window
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－fauorites} \\
\hline Type & Identifier & Tool position \\
\hline 488 & Surf．grinding wheel & 〈口】丁口｜ \\
\hline 418 & Facing wheel &  \\
\hline 498 & Dresser & －\％首虽 \\
\hline
\end{tabular}

Figure 12－4 Tools listed in the＂New Tool－Grinding Tools＂window
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－turning tools} \\
\hline Type & Identifier & Tool position \\
\hline 588 & Roughing tool & 〈回口回》 \\
\hline 518 & Finishing tool &  \\
\hline 528 & Plunge cutter &  \\
\hline 538 & Cutting tool &  \\
\hline 548 & Threading tool &  \\
\hline 550 & Button tool & \○®○○ \\
\hline 568 & Rotary drill & 田禹回罒 \\
\hline 588 & 3D turning probe & \(\| \mathrm{c}=18\) \\
\hline 585 & Calibrating tool & \(40 \% 00^{\circ}\) \\
\hline
\end{tabular}

Figure 12－5 Available tools in the＂New Tool－Turning Tools＂window
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－special tools} \\
\hline Type & Identifier & Tool position \\
\hline 780 & Slotting saw &  \\
\hline & 30 probe & \(c=0 \Rightarrow\) ？ \\
\hline & Edge finder &  \\
\hline & Mono probe &  \\
\hline 713 & L probe & \(8 L_{0} 8^{9} 9\) \\
\hline 714 & Star probe &  \\
\hline & Calibrating tool & ᄃ \(\|=\) П \\
\hline & Stop & 戌号吅它 \\
\hline & Mandrel & －\(\square_{\text {－}}\) \\
\hline 732 & Steady rest &  \\
\hline
\end{tabular}

Figure 12－6 Available tools in the＂New Tool－Special Tools＂window

\section*{See also}

Changing the cutting edge position or tool type（Page 717）

\subsection*{12.4 Tool dimensioning}

This section provides an overview of the dimensioning of tools.

\section*{Tool types}


Figure 12-7 Finishing tool (Type 510)


Figure 12-8 Angle descriptions


Figure 12-9 Plunge cutter (Type 520)


Figure 12-10 Milling cutter (Type 120)


Figure 12-11 Drill (Type 200)


Figure 12-12 Threading tool (Type 540)


Figure 12-13 Button tool (Type 550)


Figure 12-14 Stop (Type 730)


Figure 12-15 Rotary drill (Type 560)


Figure 12-16 Tap (Type 240)


Figure 12-17 3D probe

Machine manufacturer
The tool length is measured to the center of the ball or to the ball circumference.
Please refer to the machine manufacturer's specifications.

\section*{Note}

A 3D probe must be calibrated before use.

\subsection*{12.5 Tool list}

All parameters and functions that are required to create and set up the tools are displayed in the tool list.

Each tool is uniquely identified by the tool identifier and the replacement tool number.
For the tool display, i.e. when displaying the cutting edge positions, the machine coordinate system is taken into account.

\section*{Tool parameters}
\begin{tabular}{|c|c|}
\hline Column heading & Meaning \\
\hline Location & \multirow[t]{2}{*}{\begin{tabular}{l}
Magazine/location number \\
- The magazine location numbers \\
The magazine number is specified first, followed by the location number in the magazine. \\
If there is only one magazine, only the location number is displayed. \\
- Load position in the load magazine \\
The following icons can also be displayed for other magazine types (e.g. for a chain): \\
- Spindle location as an icon \\
- Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons.
\end{tabular}} \\
\hline * If activated in magazine selection & \\
\hline \multirow[t]{3}{*}{Type} & Tool type \\
\hline & Specific tool offset data is displayed depending on the tool type (represented as an icon). \\
\hline & The icon identifies the position of the tool; this was selected when the tool was created. \\
\hline \(\bigcup_{\text {SELECT }}\) & You have the option of changing the tool position or the tool type using the <SELECT> key. \\
\hline Tool name & The tool is identified by the name and the replacement tool number. You can enter the name as text or number. \\
\hline & Note: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: | \# ". \\
\hline ST & Replacement tool number (for replacement tool strategy). \\
\hline D & Cutting edge number \\
\hline Length X , length Z & \begin{tabular}{l}
Tool length \\
Geometry data length \(X\) and length \(Z\)
\end{tabular} \\
\hline Radius & Tool radius \\
\hline \(\varnothing\) & Tool diameter \\
\hline
\end{tabular}
\begin{tabular}{|c|c|}
\hline Column heading & Meaning \\
\hline \begin{tabular}{l}
Width/ \\
Tip width/ \\
Tip angle / \\
Pitch \\
Drilling radius
\end{tabular} & \begin{tabular}{l}
Cutting edge for Type 150 - side milling cutter and Type 151 - saw \\
Tip width for Type 520 - plunge cutter and Type 530 - cut-off tool \\
Tip angle for Type 200 - twist drill and Type 220 - centering tool and \\
Type 230 - countersink \\
Pitch for Type 240 - tap \\
Drilling radius for Type 560 - rotary drill. Holder angle and cutting tip angle are fixed.
\end{tabular} \\
\hline  & \begin{tabular}{l}
Cutting edge graphic \\
The cutting edge graphic shows the positioning defined by the holder angle, cut direction and cutting tip angle. \\
Holder angle for Type 500 - rougher and Type 510 - finisher. \\
The reference direction for the holder angle specifies the cut direction. In addition to the holder angle, the cutting tip angle is also specified.
\end{tabular} \\
\hline N & Number of teeth for Type 110 - ball end mill for cylindrical die-sinking cutter, Type 111 - ball end mill for tapered die-sinking cutter, Type 120 end mill, Type 121 - end mill with corner rounding, Type 130 - angle head cutter, Type 140 - facing tool, Type 150 - side mill, Type 155 - bevel cutter, Type 156 - bevel cutter with corner rounding, and Type 157 - tapered die-sinking cutter. \\
\hline Tip length & \begin{tabular}{l}
Tip length of a cutting tool or grooving cutter \\
The tip length is required for displaying the tools during the simulation of the program processing.
\end{tabular} \\
\hline \# & \begin{tabular}{l}
Direction of spindle rotation \\
The direction of the spindle's rotation is relative to the tool spindle for powered tools (drilling and milling machines) and to the main or counterspindle for turning tools. \\
If you are using a drilling or milling machine for "Drilling centric" or "Thread centric", the specified direction of rotation is relative to the cutting direction of the tool. The main spindle then rotates to match the tool.
\(\square\) Spindle is not switched on
\(\square\) CW spindle rotation
CCW spindle rotation
\end{tabular} \\
\hline 工 & \begin{tabular}{l}
Coolant 1 and 2 (e.g. internal and external cooling) can be switched on and off. \\
The coolant infeed at the machine does not necessarily have to be set-up.
\end{tabular} \\
\hline M1-M4 & Other tool-specific functions such as additional coolant infeed, monitoring functions for speed, tool breakage, etc. \\
\hline
\end{tabular}

\section*{Further parameters}

If you have set up unique cutting edge numbers, these are displayed in the first column.
\begin{tabular}{|l|l|}
\hline Column heading & Meaning \\
\hline D no. & Unique cutting edge number \\
\hline SN & Cutting edge number \\
\hline SC & Setup offsets \\
U & Display of the existing setup offsets \\
\hline
\end{tabular}

You use the configuration file to specify the selection of parameters in the list.


\section*{Software option}

In order to be able to manage the parameter spindle direction of rotation, coolant and tool-specific functions (M1-M4), you require the "ShopMill/ShopTurn" option.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{References}

Information on the configuration and setting up of the tool list can be found in the following references:

SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

\section*{Icons in the tool list}
\begin{tabular}{|l|l|l|}
\hline \begin{tabular}{l} 
Icon/ \\
Marking
\end{tabular} & & Meaning \\
\hline Tool type & R & \\
\hline Red "X" & & The tool is disabled. \\
\hline \begin{tabular}{l} 
Yellow triangle pointing down- \\
ward
\end{tabular} & \(\nabla\) & The prewarning limit has been reached. \\
\hline Yellow triangle pointing upward & \(\triangle\) & \begin{tabular}{l} 
The tool is in a special state. \\
Place the cursor on the marked tool. A tool tip will \\
provide a brief description.
\end{tabular} \\
\hline Green frame & \(\square\) & The tool is preselected. \\
\hline Magazine/location number & \multicolumn{3}{|l|}{} \\
\hline Green double arrow & \multicolumn{4}{|l|}{} \\
\hline Gray double arrow & \begin{tabular}{l} 
The magazine location is positioned at the change \\
position.
\end{tabular} \\
\hline Red "X" & \begin{tabular}{l} 
The magazine location is positioned at the loading \\
position.
\end{tabular} \\
\hline
\end{tabular}

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Procedure}


Parameter

1. Select the "Parameter" operating area.
2. Press the "Tool list" softkey.

The "Tool List" window opens.

\section*{See also}

Displaying tool details (Page 707)
Changing the cutting edge position or tool type (Page 717)

\subsection*{12.5.1 Additional data}

The following tool types require geometry data that is not included in the tool list display.

\section*{Tool types with additional geometry data}
\begin{tabular}{|c|c|}
\hline Tool type & Additional parameters \\
\hline 111 Conical ballhead cutter & Corner radius \\
\hline 121 End mill with corner rounding & Corner radius \\
\hline 130 Angle head cutter & \begin{tabular}{l}
Geometry length (length \(X\), length \(Y\), length \(Z\) ) \\
Wear length ( \(\Delta\) length \(X, \Delta\) length \(Y, \Delta\) length \(Z\) ) \\
Adapter length (length \(X\), length \(Y\), length \(Z\) ) \\
V (direction vector 1-6) \\
Vector \(X\), vector \(Y\), vector \(Z\)
\end{tabular} \\
\hline 131 Angle head cutter with corner rounding & \begin{tabular}{l}
Geometry length (length \(X\), length \(Y\), length \(Z\) ) Corner radius \\
Wear length ( \(\Delta\) length \(X, \Delta\) length \(Y, \Delta\) length \(Z\) ) \\
Adapter length (length \(X\), length \(Y\), length \(Z\) ) \\
V (direction vector 1-6) \\
Vector \(X\), vector \(Y\), vector \(Z\)
\end{tabular} \\
\hline 140 Face milling & External radius Tool angle \\
\hline 155 Bevel cutter & Taper angle \\
\hline 156 Bevel cutter with corner rounding & Corner radius Taper angle \\
\hline 157 Conical die-milling cutter & Taper angle \\
\hline 700 slotting saw & \begin{tabular}{l}
Geometry length (length \(X\), length \(Y\), length \(Z\) ) \\
Wear length ( \(\Delta\) length \(X, \Delta\) length \(Y, \Delta\) length \(Z\) ) \\
Adapter length (length \(X\), length \(Y\), length \(Z\) ) \\
Geometry (slot width, projection) \\
Wear (slot width, projection)
\end{tabular} \\
\hline
\end{tabular}

You can use the configuration file to specify the data to be displayed for specific tool types in the "Additional Data" window.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}

\section*{Further data}
1. The tool list is opened.
2. In the list, select an appropriate tool, e.g. an angle head cutter.
3. Press the "Additional data" softkey.

The "Additional Data - ..." window opens.
The "Additional data" softkey is only active if a tool for which the "Additional Data" window is configured is selected.

\subsection*{12.5.2 Creating a new tool}

When creating a new tool, the "New tool - favorites" window offers you a number of selected tool types, known as "favorites".
If you do not find the desired tool type in the favorites list, then select the milling, drilling, turning or special tool via the corresponding softkeys.

\section*{Note}

\section*{Grinding tools}

Additional grinding tools are available depending on the particular machine configuration.

\section*{Procedure}
1. The tool list is opened.
2. Place the cursor in the tool list at the position where the new tool should be stored.

For this, you can select an empty magazine location or the NC tool memory outside of the magazine.

You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.

New
tool

\section*{Fauorites}

The "New tool - favorites" window opens
- OR -
```

Cutters
188-199

```

\section*{Spec.tool}

788-988


If you want to create a tool that is not in the "Favorites" list, press the softkey "Cutters 100-199", "Drill 200-299", "Grinders 400-499" "Turntools 500-599" or "Spec.tool 700-900".
The "New tool - milling cutter", "New tool - drill", "New tool - grinding tools", "New tool - turning tools" or "New tool - special tools" window opens.
4. Select the tool by placing the cursor on the corresponding tool type and on the icon of the desired cutting edge position.
5. If more than four cutting edge positions are available, select the desired cutting edge position with the aid of the <cursor left> and <cursor right> keys.
6. Press the "OK" softkey.

The tool is added to the tool list with a predefined name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

\section*{Multiple load points}

If you have configured several loading points for a magazine, then the "Select loading point" window appears when a tool is created directly in an empty magazine location or when the "Load" softkey is pressed.

Select the required load point and confirm with the "OK" softkey.

\section*{Additional data}

If configured accordingly, the "New tool" window opens after the required tool has been selected and confirmed with "OK".
You can define the following data in this window:
- Names
- Tool location type
- Size of tool

\section*{References:}

For a description of configuration options, refer to the
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\subsection*{12.5.3 Measuring the tool}

You can measure the tool offset data for the individual tools directly from the tool list.

\section*{Note}

Tool measurement is only possible with an active tool.

\section*{Procedure}

\section*{Tool \\ list} measure

Meas. tool


X
```

Z

```
```

Set
length

```
1. The tool list is opened.
2. Select the tool that you want to measure in the tool list and press the "Measure tool" softkey.

You jump to the "JOG" operating area and the tool to be measured is entered in the "T" field in the "Length Manual" screen.
3. Select the cutting edge number D and the replacement tool number ST.
4. Press the " X " or " \(Z\) " softkey, depending on which tool length you want to measure.
5. Traverse the tool towards the workpiece in the direction that is to be measured and scratch it.
6. Enter the position of the workpiece edge in XO or ZO .

If no value is entered for XO or ZO , the value is taken from the actual value display.
7. Press the "Set length" softkey.

The tool length is calculated automatically and entered in the tool list.

\subsection*{12.5.4 Managing several cutting edges}

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. The number of possible cutting edges depends on the control configuration.
Tool cutting edges that are not required can be deleted.

\section*{Procedure}

\section*{Edges}

New cutting edge

Delete cutting edge
1. The tool list is opened.
2. Position the cursor on the tool for which you would like to store more cutting edges.
3. Press the "Cutting edges" softkey in the "Tool list".
1. The tool list is opened.
4. Press the "New cutting edge" softkey. A new data set is stored in the list.
The cutting edge number is incremented by 1 and the offset data is assigned the values of the cutting edge on which the cursor is positioned.
5. Enter the offset data for the second cutting edge.
6. Repeat this process if you wish to create more tool edge offset data.
7. Position the cursor on the cutting edge that you want to delete and press the "Delete cutting edge" softkey.
The data set is deleted from the list. The first tool cutting edge cannot be deleted.

\subsection*{12.5.5 Delete tool}

Tools that are no longer in use can be deleted from the tool list for a clearer overview.

\section*{Procedure}
1. The tool list is opened.
2. Place the cursor on the tool that you would like to delete.

Delete tool
3. Press the "Delete tool" softkey.

A safety prompt is displayed.
4. Press the "OK" softkey if you really want to delete the tool.

Use this softkey to delete the tool.
If the tool is in a magazine location, it is unloaded and then deleted.

Multiple load points - tool in magazine location
If you have configured several loading points for a magazine, then the "Loading Point Selection" window appears after pressing the "Delete tool" softkey.

Select the required load point and press the "OK" softkey to unload and delete the tool.

\subsection*{12.5.6 Loading and unloading tools}

You can load and unload tools to and from a magazine via the tool list. When a tool is loaded, it is taken to a magazine location. When it is unloaded, it is removed from the magazine and stored in the tool list.
When you are loading a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.
You can unload tools from the magazine that you are not using at present. HMI then automatically saves the tool data in the tool list in the NC memory outside the magazine.

Should you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

\section*{Procedure}

\section*{Load \\ Load}
4. Press the "OK" softkey to load the tool into the suggested location.
- OR -

Enter the location number you require and press the "OK" softkey.
- OR -

Press the "Spindle" softkey.

The tool is loaded into the specified magazine location or spindle.

\section*{Several magazines}

If you have configured several magazines, the "Load to ..." window appears after pressing the "Load" softkey.

If you do not want to use the suggested empty location, then enter your desired magazine and magazine location. Confirm your selection with "OK".

\section*{Multiple load points}

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Load" softkey.
Select the required loading point and confirm with "OK".

\section*{Unloading tools}

Unload
1. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" softkey.
2. Select the required load point in the "Load Point Selection" window.
3. Confirm your selection with "OK".
- OR -

Undo your selection with "Cancel".

\subsection*{12.5.7 Selecting a magazine}

You can directly select the buffer memory, the magazine, or the NC memory.

\section*{Procedure}

Magazine selection
1. The tool list is opened.
2. Press the "Magazine selection" softkey.

If there is only one magazine, you will move from one area to the next (i.e. from the buffer memory to the magazine, from the magazine to the NC memory, and from the NC memory back to the buffer memory) each time you press the softkey. The cursor is positioned at the beginning of the magazine each time.
- OR -

If there is more than one magazine, the "Magazine Selection" window opens. Position the cursor on the desired magazine in this window and press the "Go to ..." softkey.
The cursor jumps directly to the beginning of the specified magazine.

\section*{Hiding magazines}


Deactivate the checkbox next to the magazines that you do not want to appear in the magazine list.

The magazine selection behavior with multiple magazines can be configured in different ways.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{References}

For a description of configuration options, refer to the
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{12．5．8 Code carrier connection（only 840D sl）}

\section*{12．5．8．1 Overview}

You have the option of configuring a code carrier connection．
This means that the following functions are available in SINUMERIK Operate：
－Creating a new tool from code carrier
－Unloading tools on code carrier


\section*{Software option}

In order to use the functions，you require the option＂Tool Ident Connection＂．

\section*{References}

Further details on tool management with code carrier and the configuration of the user interface in SINUMERIK Operate can be found in the following reference：
－Function Manual SINUMERIK Integrate for Production AMB，AMC AMM／E
－SINUMERIK Operate（IM9）／SINUMERIK 840D sl Commissioning Manual
With a code carrier connection，in the list of favorites，there is also a tool available．
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－fauorites} \\
\hline Type & Identifier & Tool position \\
\hline & Tool from code carrier & 婁 \\
\hline & Roughing tool & 《回回回〉 \\
\hline & Finishing tool &  \\
\hline & Plunge cutter & 〈或可耑 \\
\hline 548 & Threading tool &  \\
\hline 558 & Button tool &  \\
\hline & Rotary drill & 回可回回 \\
\hline & 3D turning probe & \(\| \mathrm{c}=8 \mathrm{P}\) \\
\hline 738 & Stop & 马『『吅 \\
\hline & End mill & 且 \\
\hline & Facing tool & ¢ \(\mathrm{d}_{8}\) \\
\hline 158 & Side mill & \(\cdots{ }_{1}\) \\
\hline 288 & Twist drill & \(\mathbb{N} \boldsymbol{N} \boldsymbol{z}\) \\
\hline 248 & Tap & 䧳非㖘 \\
\hline
\end{tabular}

Figure 12－18 New tool from code carrier in the list of favorites

\section*{Creating a new tool from code carrier}

Tool list
1. The tool list is opened.
2. Place the cursor in the tool list at the position where the new tool should be created.
For this, you can select an empty magazine location or the NC tool memory outside of the magazine.

You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.
New
3. Press the "New tool" softkey.

The "New Tool - Favorites" window opens.
4. Position the cursor on the entry "Tool from code carrier" and press the "OK" softkey.
The tool data is read from the code carrier, and is displayed in the "New tool" window with the tool type, tool name and possibly with certain parameters.
5. Press the "OK" softkey.

The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

\section*{Unloading tool on code carrier}
1. The tool list is opened.
```

Unload

```

On
code carr.
2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "On code carrier" softkeys.
The tool is unloaded and the data of the tool are then written to the code carrier.

According to the appropriate setting, after being read out on the code carrier, the unloaded tool on the code carrier is deleted from the NC memory.

\section*{Delete tool on code carrier}

\section*{14. Tool list}
1. The tool list is opened.
2. Position the cursor on the tool on code carrier that you want to delete.

Delete 3. Press the "Delete tool" and "On code carrier" softkeys.
tool The tool is unloaded and the data of the tool are written to the code
On
code carr. carrier. The tool is then deleted from the NC memory.

The deletion of the tool can be set differently, i.e. the "On code carrier" softkey is not available.

\section*{References}

A description of the configuration options can be found in the following reference:
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

\section*{12．5．9 Managing a tool in a file}

If the＂Permit tool in／out file＂option is activated in the tool list settings，then an additional entry is available in the list of favorites．
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－fauorites} \\
\hline Type & Identifier & Tool position \\
\hline & Tool from file & 四 \\
\hline 580 & Roughing tool & 〈田回回 \\
\hline 518 & Finishing tool & 4 \(\square^{4}\) \\
\hline 528 & Plunge cutter & 〈或可耑〉 \\
\hline 548 & Threading tool & 4 \(\triangle\)＊A M \\
\hline 558 & Button tool & 4○○○○＇ \\
\hline 568 & Rotary drill & 回可四回 \\
\hline 588 & 3D turning probe & \(\| \mathrm{c}=\mathrm{P}=\) \\
\hline 738 & Stop & 号口品或 \\
\hline 128 & End mill & \(\Delta{ }^{-1}\) \\
\hline 148 & Facing tool &  \\
\hline 158 & Side mill & \(\cdots \mathrm{l}=10\) \\
\hline 288 & Twist drill & \(\mathbb{\sim} \boldsymbol{N} \boldsymbol{z}\) \\
\hline 248 & Tap & 翡镃非铱 \\
\hline
\end{tabular}

Figure 12－19 New tool from file in the list of favorites

\section*{Creating a new tool from file}

1．The tool list is opened．

2．Place the cursor in the tool list at the position where the new tool should be created．

For this，you can select an empty magazine location or the NC tool memory outside of the magazine．
You may also position the cursor on an existing tool in the area of the NC tool memory．Data from the displayed tool will not be overwritten．

\section*{Fauorites}


3．Press the＂New tool＂softkey．

The＂New Tool－Favorites＂window opens．

4．Position the cursor on the entry＂Tool from file＂and press the＂OK＂soft－ key．
The＂Load tool data＂window is opened．
5．Navigate to the required file and press the＂OK＂softkey．
The tool data is read from the file，and is displayed in the＂New tool from file＂window with the tool type，tool name and possibly with certain pa－ rameters．
6. Press the "OK" softkey.

The tool is added to the tool list with the specified name. If the cursor is located on an empty magazine location in the tool list, then the tool is loaded to this magazine location.

The tool creation sequence can be defined differently.

\section*{Unloading a tool in a file}

\section*{Unload}

In file

1. The tool list is opened.
2. Place the cursor on the tool that you would like to unload from the magazine and press the "Unload" and "In file" softkeys.
3. Navigate to the required directory and press the "OK" softkey.
4. Enter the required file name in the "Name" field and press the "OK" softkey.
The field is pre-assigned with the tool names.
The tool is unloaded and the data of the tool are written to the file.
According to the corresponding setting, after being read out, the unloaded tool is deleted from the NC memory.

\section*{Deleting a tool in a file}
1. The tool list is opened.
2. Position the cursor on the tool that you wish to delete.

Delete tool

In
file

3. Navigate to the required directory and press the "OK" softkey.

3. Press the "Delete tool" and "In file" softkeys.

\subsection*{12.6 Tool wear}

All parameters and functions that are required during operation are contained in the tool wear list.

Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. The control then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent level of accuracy during workpiece machining.

\section*{Monitoring types}

You can automatically monitor the tools' working times via the workpiece count, tool life or wear.

\section*{Note}

\section*{Combination of monitoring types}

You have the option to monitor a tool by a type or any combination of monitoring types can be switched on.

In addition, you can disable tools when you no longer wish to use them.

\section*{Tool parameters}
\begin{tabular}{|c|c|}
\hline Column heading & Meaning \\
\hline Location & \begin{tabular}{l}
Magazine/location number \\
- The magazine location numbers \\
The magazine number is specified first, followed by the location number in the magazine. \\
If there is only one magazine, only the location number is displayed.
\end{tabular} \\
\hline BS & \begin{tabular}{l}
- Load position in the load magazine \\
The following icons can also be displayed for other magazine types (e.g. for a chain):
\end{tabular} \\
\hline \# & - Spindle location as an icon \\
\hline * If activated in magazine selection & - Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons \\
\hline \multirow[t]{3}{*}{Type} & Tool type \\
\hline & Depending on the tool type (represented by an icon), certain tool offset data is enabled. \\
\hline & The icon identifies the position of the tool; this was selected when the tool was created. \\
\hline \(\bigcup_{\text {SELECT }}\) & You have the option of changing the tool position or the tool type using the <Select> key. \\
\hline
\end{tabular}
\begin{tabular}{|l|l|}
\hline Column heading & Meaning \\
\hline Tool name & \begin{tabular}{l} 
The tool is identified by the name and the sister tool number. You can \\
enter the name as text or number. \\
Note: The maximum length of tool names is 31 ASCII characters. The \\
number of characters is reduced for Asian characters or Unicode charac- \\
ters. The following special characters are not permitted: |\# ".
\end{tabular} \\
\hline ST & Sister tool number (for replacement tool strategy). \\
\hline D & Cutting edge number \\
\hline\(\Delta\) length X, \(\Delta\) length Z & Wear for length X or wear for length Z \\
\hline\(\Delta\) Radius & Radius wear
\end{tabular}\(|\)\begin{tabular}{l} 
Selection of tool monitoring \\
- by tool life (T) \\
- by count (C) \\
- by wear (W) \\
The wear monitoring is configured via a machine data item. \\
Please refer to the machine manufacturer's instructions.
\end{tabular}.

\section*{Further parameters}

If you have set up unique cutting edge numbers, these are displayed in the first column.
\begin{tabular}{|l|l|}
\hline Column heading & Meaning \\
\hline D no. & Unique cutting edge number \\
\hline SN & Cutting edge number \\
\hline SC & Setup offsets \\
\(\mathbf{U}\) & Display of the existing setup offsets \\
\hline
\end{tabular}

\section*{Icons in the tool wear list}
\begin{tabular}{|c|c|c|}
\hline \begin{tabular}{l}
Icon/ \\
Marking
\end{tabular} & & Meaning \\
\hline \multicolumn{3}{|l|}{Tool type} \\
\hline Red "X" & X & The tool is disabled. \\
\hline Yellow triangle pointing downward & V & The prewarning limit has been reached. \\
\hline Yellow triangle pointing upward & \(\triangle\) & \begin{tabular}{l}
The tool is in a special state. \\
Place the cursor on the marked tool. A tooltip provides a short description.
\end{tabular} \\
\hline Green border & \(\square\) & The tool is preselected. \\
\hline \multicolumn{3}{|l|}{Magazine/location number} \\
\hline Green double arrow & \(\stackrel{\square}{4}\) & The magazine location is positioned at the change position. \\
\hline Gray double arrow (configurable) & \(\stackrel{\square}{4}\) & The magazine location is positioned at the loading position. \\
\hline Red "X" & X & The magazine location is disabled. \\
\hline
\end{tabular}

\section*{Procedure}


Parameter

Tool wear
1. Select the "Parameter" operating area.
2. Press the "Tool wear" softkey.

\section*{See also}

Displaying tool details (Page 707)
Changing the cutting edge position or tool type (Page 717)

\subsection*{12.6.1 Reactivate tool}

You can replace disabled tools or make them ready for use again.

\section*{Preconditions}

In order to be able to reactivate a tool, the monitoring function must be activated and a setpoint must be stored.

\section*{Procedure}

Reacti-

\author{
vate
}
1. The tool wear list is opened.
2. Position the cursor on the disabled tool which you would like to reuse.
3. Press the "Reactivate" softkey.

The value entered as the setpoint is entered as the new tool life or workpiece count.
The disabling of the tool is cancelled.

\section*{Reactivating and positioning}

When the "Reactivate with positioning" function is configured, the selected tool's magazine location will also be positioned at a loading point. You can exchange the tool.

\section*{Reactivation of all monitoring types}

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{References}

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Multiple load points}

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Load" softkey.

Select the required load point and confirm with the "OK" softkey.

\subsection*{12.7 Tool data OEM}

You have the option of configuring the list according to your requirements.
Depending on the machine configuration, grinding-specific parameters are displayed in the list with OEM tool data.

\section*{Grinding tool-specific parameters}
\begin{tabular}{|l|l|}
\hline Column heading & Meaning \\
\hline Min. radius & Limit value for the radius of the grinding wheel to monitor the geometry. \\
\hline Actual radius & \begin{tabular}{l} 
Displays the sum of the geometry value, wear value and, if set, the base \\
dimension.
\end{tabular} \\
\hline Min. width & Limit value for the width of the grinding wheel to monitor the geometry. \\
\hline Actual width & \begin{tabular}{l} 
The width of the grinding wheel measured, for example, after the dress- \\
ing.
\end{tabular} \\
\hline Max. speed & Maximum speed \\
\hline Max. peripheral velocity & Maximum peripheral velocity \\
\hline Wheel angle & Angle of inclined wheel \\
\hline Monitoring & \begin{tabular}{l} 
Denitoring geometry and speed \\
and when monitoring the minimum wheel radius.
\end{tabular} \\
\hline Tool base dimension & \begin{tabular}{l} 
Number of programmed spindle (e.g. grinding wheel peripheral velocity) \\
and spindle to be monitored (e.g. wheel radius and width).
\end{tabular} \\
\hline \begin{tabular}{l} 
Spindle number \\
\(U\)
\end{tabular} & \begin{tabular}{l} 
Selects the parameter to calculate the radius \\
- Length \(X\) \\
- Length Y \\
- Length Z \\
- Radius
\end{tabular} \\
\hline Param. rad.calc. & \begin{tabular}{l} 
This parameter defines which tool parameters of tool cutting edge 2 (D2) \\
and tool cutting edge 1 (D1) have to be chained to one another. When \\
the value of a chained parameter is modified, then the other cutting edge \\
is automatically used when chaining the parameter.
\end{tabular} \\
\hline Chaining rule
\end{tabular}

\section*{References}

You can find more information on grinding tools in the following manual:
Function Manual, Extended Functions W4: Grinding-specific tool offset and monitoring functions / SINUMERIK 840D sI

Refer to the following document for more information on configuring OEM tool data:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Procedure}
1 이
Parameter
OEM Tool
1. Select the "Parameter" operating area.
2. Press the "OEM tool" softkey.
3. Position the cursor on a grinding tool.

\subsection*{12.8 Magazine}

Tools are displayed with their magazine-related data in the magazine list. Here, you can take specific actions relating to the magazines and the magazine locations.

Individual magazine locations can be location-coded or disabled for existing tools.

\section*{Tool parameters}
\begin{tabular}{|c|c|}
\hline Column heading & Meaning \\
\hline \begin{tabular}{l}
Location \\
BS
\end{tabular} & \multirow[t]{5}{*}{\begin{tabular}{l}
Magazine/location number \\
- The magazine location numbers \\
The magazine number is specified first, followed by the location number in the magazine. \\
If there is only one magazine, only the location number is displayed. \\
- Load position in the load magazine \\
The following icons can also be displayed for other magazine types (e.g. for a chain): \\
- Spindle location as an icon \\
- Locations for gripper 1 and gripper 2 (applies only when a spindle with dual gripper is used) as icons
\end{tabular}} \\
\hline & \\
\hline \# & \\
\hline ) 6 & \\
\hline * If activated in magazine selection & \\
\hline \multirow[t]{3}{*}{Type} & Tool type \\
\hline & Depending on the tool type (represented by an icon), certain tool offset data is enabled. \\
\hline & The icon identifies the position of the tool; this was selected when the tool was created. \\
\hline \(\bigcup_{\text {SELECT }}\) & You have the option of changing the tool position or the tool type using the <Select> key. \\
\hline Tool name & The tool is identified by the name and number of the replacement tool (ST). You can enter the name as text or number. \\
\hline & Note: The maximum length of tool names is 31 ASCII characters. The number of characters is reduced for Asian characters or Unicode characters. The following special characters are not permitted: |\#". \\
\hline ST & Number of the replacement tool. \\
\hline D & Cutting edge number \\
\hline G & Disabling of the magazine location. \\
\hline Mag.loc. type & Display of magazine location type. \\
\hline Tool.loc. type & Display of tool location type. \\
\hline Ü & Marking a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine. \\
\hline P & \begin{tabular}{l}
Fixed location coding. \\
The tool is permanently assigned to this magazine location.
\end{tabular} \\
\hline
\end{tabular}

\section*{Further parameters}

If you have set up unique cutting edge numbers, these are displayed in the first column.
\begin{tabular}{|l|l|}
\hline Column heading & Meaning \\
\hline D no. & Unique cutting edge number \\
\hline SN & Cutting edge number \\
\hline
\end{tabular}

\section*{Magazine list icons}
\begin{tabular}{|l|l|l|}
\hline \begin{tabular}{l} 
Icon/ \\
Marking
\end{tabular} & & Meaning \\
\hline Tool type & X & The tool is disabled. \\
\hline Red "X" & & The prewarning limit has been reached. \\
\hline \begin{tabular}{l} 
Yellow triangle pointing \\
downward
\end{tabular} & \(\nabla\) & \begin{tabular}{l} 
The tool is in a special state. \\
Place the cursor on the marked tool. A tooltip provides a \\
short description.
\end{tabular} \\
\hline \begin{tabular}{l} 
Yellow triangle pointing \\
upward
\end{tabular} & \(\triangle\) & The tool is preselected. \\
\hline Green border & \(\square\) & The magazine location is positioned at the change position. \\
\hline Magazine/location number & \multicolumn{3}{|l|}{} \\
\hline Green double arrow & \multicolumn{4}{|l|}{} \\
\hline \begin{tabular}{l} 
Gray double arrow (con- \\
figurable)
\end{tabular} & \(\leftrightharpoons\) & The magazine location is positioned at the loading position. \\
\hline Red "X" & The magazine location is disabled. \\
\hline
\end{tabular}

\section*{Procedure}
1. Select the "Parameter" operating area.
2. Press the "Magazine" softkey.

\section*{See also}

Displaying tool details (Page 707)
Changing the cutting edge position or tool type (Page 717)

Turning

\subsection*{12.8.1 Positioning a magazine}

You can position magazine locations directly on the loading point.

\section*{Procedure}
1. The magazine list is opened.
2. Place the cursor on the magazine location that you want to position onto the load point.
3. Press the "Position magazine" softkey.

The magazine location is positioned on the loading point.

\section*{Multiple load points}

If you have configured several loading points for a magazine, then the "Load Point Selection" window appears after pressing the "Position magazine" softkey.

Select the desired loading point in this window and confirm your selection with "OK" to position the magazine location at the loading point.

\subsection*{12.8.2 Relocating a tool}

Tools can be directly relocated within magazines to another magazine location, which means that you do not have to unload tools from the magazine in order to load them into a different location.

When you are relocating a tool, the application automatically suggests an empty location. You may also directly specify an empty magazine location.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}

\section*{Relocate}
1. The magazine list is opened.
2. Position the cursor on the tool that you wish to relocate to a different magazine location.
3. Press the "Relocate" softkey.

The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.
4. Press the "OK" softkey to relocate the tool to the recommended magazine location.
- OR -

Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

Press the "OK" softkey.

The tool is relocated to the specified magazine location.

\section*{Several magazines}

If you have set up several magazines, then the "...relocate from magazine... location... to..." window appears after pressing the "Relocate" softkey.

Select the desired magazine and location, and confirm your selection with "OK" to load the tool.

\subsection*{12.8.3 Unload/load/relocate all tools}

You have the option of unloading all tools from the magazine list, loading all tools to the magazine or relocating tools simultaneously.

\section*{Preconditions}

The following requirements must be satisfied so that the "Unload all", "Load all" or "Relocate all" softkey is displayed and available:
- Magazine management is set up
- There is no tool in the buffer / in the spindle


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
\begin{tabular}{|c|c|c|}
\hline \[
\begin{gathered}
\text { Maga- } \\
\text { zine }
\end{gathered}
\] & 1. & The magazine list is opened. \\
\hline \multirow[t]{2}{*}{Unload all} & \multirow[t]{2}{*}{2.} & Press the "Unload all" softkey. \\
\hline & & - OR - \\
\hline Load all & & Press the "Load all" softkey. \\
\hline & & - OR - \\
\hline
\end{tabular}

Turning

\section*{Relocate}
all Cancel

Press the "Unload all" softkey.

A prompt is displayed as to whether you really want to unload, load or relocate all tools.
3. Press the "OK" softkey to continue with unloading, loading or relocating the tools.
The tools are unloaded from the magazine, loaded into the magazine or relocated in ascending magazine location number order.
4. Press the "Cancel" softkey if you wish to cancel the unloading operation.

\section*{\(12.9 \quad\) Tool details}

\subsection*{12.9.1 Displaying tool details}

All of the selected tool parameters are listed in the "Tool Details" window.
The parameters are displayed, sorted according to the following criteria
- Tool data
- Grinding data (if grinding tools have been configured)
- Cutting edge data
- Monitoring data

\section*{Protection level}

You require access level, keyswitch 3 (protection level 4) in order to edit the parameters in the detail window.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}

\subsection*{12.9 Tool details}
\[
\begin{array}{c|ll}
\hline \begin{array}{c}
\text { Cutting } \\
\text { edge data }
\end{array} & 5 . & \begin{array}{l}
\text { Press the "Cutting edge data" softkey if you wish to display the cutting } \\
\text { edge data. }
\end{array} \\
\begin{array}{c}
\text { Monitoring } \\
\text { data }
\end{array} & 6 . & \begin{array}{l}
\text { Press the "Monitoring data" softkey if you want to display the monitoring } \\
\text { data. }
\end{array}
\end{array}
\]

\subsection*{12.9.2 Tool data}

The "Tool Details" window provides the following data on the selected tool when the "Tool data" softkey is active.
\begin{tabular}{|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Meaning} \\
\hline Magazine location & \multicolumn{2}{|l|}{The magazine number is specified first, followed by the location number in the magazine. If there is only one magazine, only the location number is displayed.} \\
\hline Tool name & \multicolumn{2}{|l|}{The tool is identified by the name and the sister tool number. You can enter the name as text or number.} \\
\hline ST & \multicolumn{2}{|l|}{Sister tool number (for sister tool strategy)} \\
\hline D quantity & \multicolumn{2}{|l|}{Number of created cutting edges} \\
\hline D & \multicolumn{2}{|l|}{Cutting edge number} \\
\hline \multirow[t]{2}{*}{Tool state} & A & Activate tool \\
\hline & F & Tool enabled \\
\hline \multirow[t]{2}{*}{X} & G & Block tool \\
\hline & M & Measure tool \\
\hline \(\nabla\) & V & Reaching the prewarning limit \\
\hline & W & Tool being changed \\
\hline & P & \begin{tabular}{l}
Tool in fixed location \\
The tool is permanently assigned to this magazine location
\end{tabular} \\
\hline & I & Tool has been in use \\
\hline \multirow[t]{2}{*}{Tool size 0} & Standard & Tool does not require an additional location in a magazine. \\
\hline & Oversize & The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine. \\
\hline & \multicolumn{2}{|l|}{Special size} \\
\hline & Left & Number of half locations to the left of the tool \\
\hline & Right & Number of half locations to the right of the tool \\
\hline Tool OEM parameters 1-6 & Freely available parameters & \\
\hline
\end{tabular}

\subsection*{12.9.3 Cutting edge data}

The "Tool Details" window provides the following data on the selected tool when the "Cutting edge data" softkey is active.
\begin{tabular}{|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Meaning} \\
\hline Magazine location & \multicolumn{2}{|l|}{\begin{tabular}{l}
The magazine number is specified first, followed by the location number in the magazine. \\
If there is only one magazine, only the location number is displayed.
\end{tabular}} \\
\hline Tool name & \multicolumn{2}{|l|}{The tool is identified by the name and the sister tool number. You can enter the name as text or number.} \\
\hline ST & \multicolumn{2}{|l|}{Sister tool number (for replacement tool strategy)} \\
\hline D quantity & \multicolumn{2}{|l|}{Number of created cutting edges} \\
\hline D & \multicolumn{2}{|l|}{Cutting edge number} \\
\hline Tool type & \multicolumn{2}{|l|}{Tool symbol with type number and current cutting edge position} \\
\hline & Length X & Length Z \\
\hline Geometry & Geometry data, length X & Geometry data, length \(Z\) \\
\hline Wear & Length X wear & Length Y wear \\
\hline & \multicolumn{2}{|l|}{Radius} \\
\hline Geometry & \multicolumn{2}{|l|}{Tool radius} \\
\hline Wear & \multicolumn{2}{|l|}{Radius wear} \\
\hline \multicolumn{3}{|l|}{For type 500 (rougher) and type 510 (finisher)} \\
\hline Cutting edge graphic & & The cutting edge graphic shows the positioning defined by the holder angle, cut direction and cutting tip angle. \\
\hline Reference direction & & The reference direction for the holder angle specifies the cut direction. \\
\hline Holder angle & \multicolumn{2}{|l|}{To determine the cutting edge positioning} \\
\hline Cutting tip angle & \multicolumn{2}{|l|}{To determine the cutting edge positioning} \\
\hline \multicolumn{3}{|l|}{Type 240 - tap} \\
\hline \multicolumn{3}{|l|}{Pitch} \\
\hline Type 200 - twist drill, & tering tool and type 230 & countersink \\
\hline
\end{tabular}

Turning
\begin{tabular}{|c|c|c|}
\hline \multicolumn{2}{|l|}{Parameter} & Meaning \\
\hline \multicolumn{2}{|l|}{Tip angle} & \\
\hline \multicolumn{3}{|l|}{Type 520 -plunge cutter, type 530 - parting tool, type 540 - threading tool} \\
\hline \multicolumn{2}{|l|}{Cutting tip length} & For displa \\
\hline \multicolumn{2}{|l|}{Cutting tip width} & Width of th \\
\hline \multicolumn{3}{|l|}{Type 110 - ball end mill for cylindrical die-sinking cutter, type 111 - ball end mill for tapered die-sinking cutter, type 120 end mill, type 121 - end mill with corner rounding, type 130 - angle head cutter, type 140 - facing tool, type 150 - side mill, type 155 - bevel cutter, type 156 - bevel cutter with corner rounding and type 157 - tapered die-sinking cutter} \\
\hline \multicolumn{2}{|l|}{N} & Number of \\
\hline \multicolumn{3}{|l|}{For driven tools (drills and milling tools)} \\
\hline \multirow[t]{3}{*}{Direction of spindle rotation \#} & \% & Spindle is \\
\hline & 2 & CW spind \\
\hline & \(\rho\) & CCW spin \\
\hline \multicolumn{2}{|l|}{Э} & Coolant 1 Please re \\
\hline \multicolumn{2}{|l|}{Cutting edge OEM parameters 1-2} & \\
\hline
\end{tabular}

\section*{Software option}

In order to be able to manage the parameters spindle direction of rotation, coolant and tool-specific functions (M1-M4), you require the "ShopMill/ShopTurn" option.

\subsection*{12.9.4 Monitoring data}

The "Tool Details" window provides the following data on the selected tool when the "Monitoring data" softkey is active.
\begin{tabular}{|l|l|}
\hline Parameter & Meaning \\
\hline Magazine location & \begin{tabular}{l} 
The magazine number is specified first, followed by the location number in the magazine. If \\
there is only one magazine, only the location number is displayed.
\end{tabular} \\
\hline Tool name & \begin{tabular}{l} 
The tool is identified by the name and the sister tool number. You can enter the name as text \\
or number.
\end{tabular} \\
\hline ST & Sister tool number (for sister tool strategy) \\
\hline D quantity & \begin{tabular}{l} 
Cutting edge number created cutting edges \\
C - count \\
W - wear \\
The wear monitoring is configured via machine data. \\
Please note the specifications of the machine manufacturer.
\end{tabular} \\
\hline \begin{tabular}{l} 
Monitoring type \\
O
\end{tabular} & Actual value \\
\hline & Actual value for tool life, count or wear \\
\hline Tool life, count and wear & Setpoint \\
\hline Tool life, count and wear & Setpoint for tool life, count or wear \\
\hline & Prewarning limit \\
\hline Tool life, count and wear & Specification of the tool life, the count or wear at which a warning is displayed. \\
\hline \begin{tabular}{l} 
Monitoring OEM parame- \\
ters 1-8
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{12.10 Sorting tool management lists}

When you are working with many tools, with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

\section*{Procedure}


Parameter


Sort

By magazine
\(B y\) type

By name
\(B y\) I number
1. Select the "Parameter" operating area.
2. Press the "Tool list", "Tool wear" or "Magazine" softkey.
3. Press the ">>" and "Sort" softkeys.

The lists are displayed sorted numerically according to magazine location.
4. Press the "Acc. to type" softkey to display the tools arranged by tool type. Identical types are sorted according to their radius.

Press the "Acc. to name" softkey to display the tool names in alphabetical order.
The replacement tool numbers are used to sort tools with the same names.
- OR -

Press the "Acc. to T number" softkey to display the tools sorted numerically.

The list is sorted according to the specified criteria.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{12.11 Filtering the tool management lists}

The filter function allows you to filter-out tools with specific properties in the tool management lists.
For instance, you have the option of displaying tools during machining that have already reached the prewarning limit in order to prepare the corresponding tools for equipping.

\section*{Filter criteria}
- Only display the first cutting edge
- Only tools that are ready to use
- Only tools with active code
- Only tools that have reached the prewarning limit
- Only locked tools
- Only tools with remaining quantity of ... to ...
- Only tools with residual tool life of ... to ...
- Only tools with unloading marking
- Only tools with loading marking

\section*{Note}

\section*{Multiple selection}

You have the option of selecting several criteria. You will receive an appropriate message if conflicting filter options are selected.
You can configure OR logic operations for the various filter criteria.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{References}

A description of the configuration options is provided in SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual
12.11 Filtering the tool management lists

\section*{Procedure}


Parameter
\begin{tabular}{|c|c|}
\hline \multicolumn{2}{|l|}{(4.) \(\begin{aligned} & \text { Tool } \\ & \text { list }\end{aligned}\)} \\
\hline & \\
\hline &  \\
\hline
\end{tabular}
...
Filter

1. Select the "Parameter" operating area.
2. Press the "Tool list", "Tool wear" or "Magazine" softkey.
3. Press the ">>" and "Filter" softkeys. The "Filter" window opens.
4. Activate the required filter criterion and press the "OK" softkey. The tools that correspond to the selection criteria are displayed in the list.
The active filter is displayed in the window header.

\subsection*{12.12 Specific search in the tool management lists}

There is a search function in all tool management lists, where you can search for the following objects:
- Tools
- You enter a tool name. You can narrow down your search by entering a replacement tool number.

You have the option of only entering a part of the name as search term.
- You enter the D number and activate if necessary, the check box "Active D number".
- Magazine locations or magazines

If only one magazine is configured, then the search is made according to the magazine location.

If several magazines are configured, then it is possible to search a specific magazine location in a specific magazine or just to search in a specific magazine.
- Empty locations

The empty location search is realized using the tool size. The tool size is defined by the number of half locations required to the right, left, top and bottom. All four directions are of significance for a box magazine. For a chain magazine, a disk-type or a turret, only the half locations to the right and left are of significance. The maximum number of half locations, which a tool can occupy is limited to 7 .
If the lists with the location type are used, then the empty location search is made using the location type and location size.
You can enter the location type as numerical value or as text depending on the particular configuration.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{References}

For a description of configuration options, refer to the
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sI
12.12 Specific search in the tool management lists

\section*{Procedure}


Parameter


Search

Tool

Magazine location

Empty location


Search
\(x\)
Cancel
1. Select the "Parameter" operating area.
2. Press the "Tool list", "Tool wear" or "Magazine" softkey.
3. Press the ">>" and "Search" softkeys.
4. Press the "Tool" softkey if you wish to search for a specific tool.
- OR -

Press the "Magazine location" softkey if you wish to search for a specific magazine location or a specific magazine.
- OR -

Press the "Empty location" softkey if you wish to search for a specific empty location.
5. Press the "OK" softkey.

The search is started.
6. Press the "Search" softkey again if the tool that was found is not the tool that was being searched for.
The search term is kept and with "OK" you start the search for the next tool that corresponds to the entry.
7. Press the "Cancel" softkey to cancel the search.

\subsection*{12.13 Changing the cutting edge position or tool type}

\section*{Procedure}

1. The tool list, the wear list, the OEM tool list or the magazine is opened.
2. Position the cursor in the column "Type" of the tool that you wish to change.
3. Press the <SELECT> key.

The "Tool types - Favorites" window opens.
4 Press the <Cursor right> or <Cursor left> key to select the new cutting edge position or the tool position for grinding tools.
- OR -

Select the desired tool type in the list of favorites or select the required tool type using the softkeys "Cutters 100-199", "Drill 200-299", "Grinders 400-499" "Turntools 500-599" or "Spec.tool 700-900".
Note: A grinding tool can only be changed into another grinding tool type.
5. Press the "OK" softkey.

The new cutting edge position / tool position or the new tool type is accepted and the corresponding icon is displayed in the "Type" column.

\subsection*{12.14 Settings for tool lists}

In the "Settings" window you have the following options to set the view in the tool lists:
- Display only one magazine in "Magazine sort"
- You can limit the display to one magazine. The magazine is displayed with the assigned buffer magazine locations and the tools not loaded.
- You may set via a configuration if you want to jump to the next magazine by clicking softkey "Magazine selection" or if the dialog "Magazine selection" is switched over to any magazine.
- Display only spindle in buffer.

In order to display only the spindle location during operation, the remaining locations of the buffer are hidden.
- Permitting tool in/out file
- When creating a new tool, the tool data can be loaded from a file.
- When deleting or unloading a tool, the tool data can be backed up to a file.
- Switch on adapter-transformed view
- Geometry lengths and the application offsets are displayed in a transformed manner in the tool list.
- In the tool wear list, the wear length and the sum offsets are displayed in a transformed manner.
no

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{References}

For further information about configuring the settings, please refer to the following references:
SINUMERIK Operate (IM9) / SINUMERIK 840D sl Commissioning Manual

\section*{Procedure}

4. Activate the corresponding check box for the desired setting.

\subsection*{12.15 Working with multitool}

Using a multitool you have the possibility of storing more than one tool at a magazine location.

The multitool itself has two or more locations to accept tools. The tools are directly mounted on the multitool. The multitool is located at a location in the magazine.

\section*{Typical applications}

One application for using multitools on lathes with revolver and counterspindle is to equip the multitool with two turning tools. One turning tool for machining on the main spindle and one turning tool for machining on the counterspindle.

An additional application is to use multitools on lathes equipped with a tool spindle. A multitool, on which several tools have been mounted, is inserted in the tool spindle at this machine. A tool change within the multitool is realized by positioning the multitool, i.e. by rotating the tool spindle.

\section*{Geometrical arrangement of the tools on the multitool}

The geometrical arrangement of the tools is defined by the clearance between the locations on the multitool.

The clearance between the locations can be defined as follows:
- Using the multitool location number or
- using the angle of the multitool location

If angle is selected here, then the value of the angle must be entered for each multitool location.

Regarding loading and unloading in a magazine, the multitool is treated as a single unit.

\subsection*{12.15.1 Tool list for multitool}

If you work with a multitool, the tool list is supplemented by the column for the multitool location number. As soon as the cursor is at a multitool in the tool list, certain column headings change.
\begin{tabular}{|l|l|}
\hline Column heading & Meaning \\
\hline Location & Magazine/location number \\
\hline MT loc. & Multitool location number \\
\hline TYPE & Symbol for multitool \\
\hline Multitool name & Name of the multitool \\
\hline
\end{tabular}


Figure 12-20 Toll list with multitool in the spindle

\section*{Procedure}

1. Select the "Parameter" operating area.
2. Press the "Tool list" softkey.

The "Tool List" window opens.

\section*{12．15．2 Create multitool}

The multitool can be selected in the list of favorites as well as in the list of special tool types．
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－fauorites} \\
\hline Type & Identifier & Tool position \\
\hline & Roughing tool & －回回回 \\
\hline & Finishing tool & 4 \(\square^{4}\) \\
\hline & Plunge cutter &  \\
\hline 548 & Threading tool & 1＊く） \\
\hline 558 & Button tool & 1○○，\(\odot \bigcirc\) \\
\hline 568 & Rotary drill & 回呵吅回 \\
\hline & 3D turning probe & \(e=0 \Rightarrow\) i \\
\hline 738 & Stop & 『嵒吅 \\
\hline 128 & End mill & \(\mathbb{E}=\mathbb{\#}=\mathbb{\$}\) \\
\hline & Facing tool & \％\({ }^{2}\) \\
\hline 158 & Side mill & \(8=11000\) \\
\hline 288 & Twist drill &  \\
\hline \multirow[t]{2}{*}{248} & Tap & 非非翡 \\
\hline & Multitool & แ時 \\
\hline
\end{tabular}

Figure 12－21 List of favorites with multitool
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|l|}{New tool－special tools} \\
\hline Type & Identifier & Tool position \\
\hline 780 & Slotting saw &  \\
\hline 710 & 30 probe & \(c=8 \Rightarrow\) ？ \\
\hline 711 & Edge finder &  \\
\hline 712 & Mono probe &  \\
\hline 713 & L probe & \(5 L_{0} 8^{9}\) \\
\hline 714 & Star probe &  \\
\hline 725 & Calibrating tool & \(\subset \|=\) П \\
\hline 738 & Stop & 『嵒曲号 \\
\hline 731 & Mandrel & －\(\square^{\text {d }}\) \\
\hline 732 & Steady rest &  \\
\hline & Multitool & W昭 \\
\hline
\end{tabular}

Figure 12－22 Selection list for special tools with multitool

\section*{Procedure}

Tool
list
1. The tool list is opened.
2. Position the cursor at the position where the tool is to be created.
For this, you can select an empty magazine location or the NC tool memory outside of the magazine.
You may also position the cursor on an existing tool in the area of the NC tool memory. Data from the displayed tool will not be overwritten.
New
tool
Spec.tool
780-980

0
5. Enter the multitool name and define the number of multitool locations.
If you wish to define the clearance of the tools based on the angle, activate the "Angle input" checkbox, and for each multitool location, enter the clearance to the reference location as angular value.
\begin{tabular}{|c|c|c|c|c|}
\hline \multicolumn{5}{|c|}{New tool} \\
\hline \multirow[t]{2}{*}{\begin{tabular}{l}
Multitool name \\
MULTITOOL3
\end{tabular}} & \multicolumn{2}{|l|}{\multirow[t]{2}{*}{\[
\]}} & \multicolumn{2}{|l|}{Multitool angle} \\
\hline & & & 1 & 0.888 \\
\hline & & & 2 & 128.888 \\
\hline & & & 3 & 230.808 \\
\hline
\end{tabular}

The multitool is created in the tool list.

\section*{Note}

The tool creation sequence can be defined differently.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{12.15.3 Equipping multitool with tools}

\section*{Precondition}

A multitool has been created in the tool list.

\section*{Procedure}

\section*{Tool list}
1. The tool list is opened.

\section*{Equip the multitool with tools}


New tool
2. Select the required multitool, position the cursor on an empty multitool location.
3. Press the "New tool" softkey.
4. Using the appropriate selection list, e.g. favorites, select the required tool.
Load multitool

2. Select the required multitool, position the cursor on an empty multitool location.

Load

0
3. Press the "Load" softkey.

The "Load with..." window opens.
4. Select the required tool.

Load the tool into the multitool
2. Position the cursor on the tool that you want to load into the multitool.

Load

Multitool

Press the "Load" and "Multitool" softkeys. The "Load to..." window opens.


0
4. Select the required multitool and the multitool location to which you wish to load the tool.

\subsection*{12.15.4 Removing a tool from multitool}

If the multitool was mechanically re-assigned (i.e. new tool were mounted), then old tools in the tool list must be removed from the multitool.

To do this, the cursor is positioned at the line where the tool is located, which is to be removed. When unloading, the tool is automatically saved in the tool list outside the magazine in the NC memory.

\section*{Procedure}
Th) Tool ..... list
Unload
Delete ..... tool
12.15.5 Delete multitool

\section*{Procedure}
1. The tool list is opened.
2. Position the cursor on the tool that you would like to unload from the multitool and press the "Unload" softkey.
- OR -

Position the cursor on the tool that you would like to remove and delete from the multitool and press the "delete tool" softkey.
1. The tool list is opened.
2. Position the cursor on the multitool that you wish to delete.
3. Press the "Delete multitool" softkey.

The multitool with all of the tools that are located in it is deleted.

\subsection*{12.15.6 Loading and unloading multitool}

\section*{Procedure}

\section*{17.7) Tool}
1. The tool list is opened.

\section*{Load a multitool into the magazine}
2. Position the cursor at the multitool that you wish to load into the magazine.
3. Press the "Load" softkey.

The "Load to" window opens.
The "... location" field is pre-assigned with the number of the first empty magazine location.

4. Press the "OK" softkey to load the multitool to the recommended empty location.
- OR -

Enter the location number you require and press the "OK" softkey.
OK
The multitool together with the tools in it is loaded to the specified magazine location.

\section*{Loading a multitool into a magazine}
2. Position the cursor on the required empty magazine location.

Load

U

3. Press the "Load" softkey.

The "Load with." window opens.
4. Select the required multitool.
5. Press the "OK" softkey.

\section*{Unloading a multitool}
2. Position the cursor on the multitool that you wish to unload from the magazine.
3. Press the "Unload" softkey.

The multitool is removed from the magazine and is saved in the NC memory at the end of the tool list.

\section*{12．15．7 Reactivating the multitool}

Multitool and tools located on the multitool can be disabled independently of one another．
If a multitool is disabled，then the tools of the multitool can no longer be changed in using a tool change．
If only one tool on a multitool has a set monitoring function and the lifetime or the unit quantity has expired，then the tool and the multitool on which the tool is located are disabled． The other tools on the multitool are not disabled．

\section*{Machine manufacturer}

Please refer to the machine manufacturer＇s specifications．

If several tools with monitoring are mounted on the multitool and the lifetime or unit quantity has expired for one tool，then only this tool is disabled．
\begin{tabular}{|c|c|c|c|c|c|c|c|c|c|c|c|}
\hline TOR 1 & \multicolumn{9}{|l|}{Tool wear} & \multicolumn{2}{|l|}{Chain＿1＿128} \\
\hline Loc． & \[
\begin{aligned}
& \text { MT } \\
& \text { L0. }
\end{aligned}
\] & Type & Tool name & ST & \(]^{\text {gth }}\) & \(\Delta\) Radius & \[
\begin{aligned}
& \mathbf{T} \\
& \mathbf{C}
\end{aligned}
\] & \begin{tabular}{l}
Tool \\
life
\end{tabular} & Set val & Prewar limit & D \\
\hline 1／185 & & 㜏 & MULTIT00L45 & & & & & & & & \(\square\) \\
\hline & 1 & 回 & SCHRUPPER＿HS & 1 & 1888 & 8.888 & T & 32.8 & 188.8 & 18.8 & \\
\hline & 2 & 区 & SCHRUPPER＿GS & 1 & 1888 & 8.888 & T & 8.0 & 188.0 & 18.0 & \(\square\) \\
\hline 1／106 & & & & & & & & & & & \\
\hline 1／187 & & & & & & & & & & & \\
\hline
\end{tabular}

\section*{Reactivating}

If a tool with expired lifetime or unit quantity that is mounted on a multitool is reactivated， then for this tool，the lifetime／unit quantity is set to the setpoint and the tool and the multitool are re－enabled（disable status is removed）．
If a multitool is reactivated，on which tools with monitoring are mounted，then the lifetime／unit quantity for all tools on the multitool are set to the setpoint no matter whether the tools are disabled or not．

\section*{Requirements}

In order to reactivate a tool，the monitoring function must be activated and a setpoint must be stored．

\section*{Procedure}

\section*{Reacti-} vate
1. Select the "Parameter" operating area.
2. Press the "Tool wear" softkey.
3. Position the cursor at the multitool that is disabled and which you would like to reactivate.
- OR -

Position the cursor on the tool that you would like to reactivate again.
4. Press the "Reactivate" softkey.

The value entered as the setpoint is entered as the new tool life or workpiece count.
The tool and the multitool are then no longer disabled (the disable is withdrawn).

\section*{Reactivating and positioning}

When the "Reactivate with positioning" function is configured, then also the magazine location at which the selected multitool is located, is positioned to the loading point. You can exchange the multitool.

\section*{Reactivation of all monitoring types}

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{References}

\subsection*{12.15.8 Relocating a multitool}

Multitools can be directly relocated within magazines to another magazine location, which means that you do not have to unload multitools with the associated tools from the magazine in order to relocate them to a different location.

When you are relocating a multitool, the system automatically recommends an empty location. You may also directly specify an empty magazine location.

\section*{Procedure}


Relocate

1. Select the "Parameter" operating area.
2. Press the "Magazine" softkey.
3. Position the cursor at the multitool that you wish to relocate to a different magazine location.
4. Press the "Relocate" softkey.

The "... relocate from location ... to location ..." window is displayed. The "Location" field is pre-assigned with the number of the first empty magazine location.
5. Press the "OK" softkey to relocate the multitool to the recommended magazine location.
- OR -

Enter the required magazine number in the "...magazine" field and the required magazine location number in "Location" field.

\section*{Note:}

Please refer to the machine manufacturer's specifications.
Press the "OK" softkey.
The multitool with the tools is relocated to the specified magazine location.

\subsection*{12.15.9 Positioning multitool}

You can position a magazine. In this case, a magazine location is positioned to the loading point.
Multitools, which are located in a spindle, can also be positioned. The multitool is rotated and therefore the multitool location involved is brought into the machining position.

\section*{Procedure}
\begin{tabular}{|c|c|c|}
\hline \multirow[t]{3}{*}{\[
\begin{aligned}
& \text { maga- Maga- } \\
& \text { Mine }
\end{aligned}
\]} & 1. & The magazine list is opened. \\
\hline & & The multitool is in the spindle. \\
\hline & 2. & Position the cursor on the multitool location that you want to bring into the machining position. \\
\hline Position multitool & 3. & Press the "Position multitool" softkey. \\
\hline
\end{tabular}

\section*{Managing programs}

\subsection*{13.1 Overview}

You can access programs at any time via the Program Manager for execution, editing, copying, or renaming.

Programs that you no longer require can be deleted to release their storage space.

\section*{NOTICE}

Execution from USB-FlashDrive
Direct execution from a USB-FlashDrive is not recommended.
There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive during operation.

Disconnecting it during operation will result in the stopping of the machining and thus to the workpiece being damaged.

\section*{Storage for programs}

Possible storage locations are:
- NC
- Local drive
- Network drives
- USB drives
- FTP drives
- V24


To display the "Local drive" softkey, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 or PC/PG).

\section*{Data exchange with other workstations}

You have the following options for exchanging programs and data with other workstations:
- USB drives (e.g. USB-FlashDrive)
- Network drives
- FTP drive

\section*{Choosing storage locations}

In the horizontal softkey bar, you can select the storage location that contains the directories and programs that you want to display. In addition to the "NC" softkey, via which the file system data can be displayed, additional softkeys can be displayed.

The "USB" softkey is only operational when an external storage medium is connected (e.g. USB-FlashDrive on the USB port of the operator panel).

\section*{Displaying documents}

You can display documents on all drives of the program manager (e.g. in the local drive or USB) and via the data tree of the system data. Various data formats are supported:
- PDF
- HTML

It is not possible to preview HTML documents.
- Various graphic formats (e.g. BMP or JPEG)
- DXF
\begin{tabular}{|l|l|}
\hline & \begin{tabular}{l} 
Software options \\
You require the "DXF reader" option in order to display DXF files. \\
\hline
\end{tabular} \\
\hline
\end{tabular}

\section*{Note}

FTP drive
It is not possible to preview documents on the FTP.

\section*{Structure of the directories}

In the overview, the symbols in the left-hand column have the following meaning:
\(\bar{\square} \quad\) Directory

直
Program

All directories have a plus sign when the program manager is called for the first time.
```

\oplus-\squarePart programs
円-Gubprograms
\square-Dorkpieces

```

Figure 13-1 Program directory in the program manager
The plus sign in front of empty directories is removed after they have been read for the first time.

The directories and programs are always listed complete with the following information:
- Name

The name name length is 24 characters.
Permissible characters include all upper-case letters (without accents), numbers, and underscores.
- Type

Directory: WPD
Program: MPF
Subprogram: SPF
Initialization programs: INI
Job lists: JOB
Tool data: TOA
Magazine assignment: TMA
Zero points: UFR
R parameters: RPA
Global user data/definitions: GUD
Setting data: SEA
Protection zones: PRO
Sag: CEC
- Size (in bytes)
- Date/time (of creation or last change)

\section*{Active programs}

Selected，i．e．active programs are identified by a green symbol．
\begin{tabular}{|c|c|c|c|c|c|}
\hline CHAN1 & Name & Type & Length & Date & Time \\
\hline \multicolumn{2}{|l|}{由－Part programs} & DIR & & 11／30／09 & 3：49：09 PM \\
\hline \multicolumn{2}{|l|}{¢－Subprograms} & DIR & & 12／02／09 & 11：24：33 AM \\
\hline \multicolumn{2}{|l|}{曰 Workpieces} & DIR & & 12／02／09 & 2：53：07 PM \\
\hline \multicolumn{2}{|l|}{守－DDREHEN1} & WPD & & 12／02／09 & 8：40：58 AM \\
\hline \multicolumn{2}{|l|}{円－－GGG} & WPD & & 12／01／09 & 12：03：39 PM \\
\hline \multicolumn{2}{|l|}{¢－JJOBSHOP＿MEHRK} & WPD & & 12／03／09 & 9：18：27 AM \\
\hline \multicolumn{2}{|l|}{円－DMEHR} & WPD & & 11／30／09 & 3：49：23 PM \\
\hline \multicolumn{2}{|l|}{¢－\(\square\) MEHRKANAL} & WPD & & 12／02／09 & 12：47：20 PM \\
\hline \multicolumn{2}{|l|}{円－\(\square\) SIM＿CHESS＿KING} & WPD & & 11／30／09 & 3：49：14 PM \\
\hline \multicolumn{2}{|l|}{円－\(\square\) SIM＿CHESS＿LADY＿26} & WPD & & 11／30／09 & 3：49：14 PM \\
\hline \multicolumn{2}{|l|}{円－\(\square\) SIM＿CHESS＿TOWER} & WPD & & 11／30／09 & 3：49：15 PM \\
\hline \multicolumn{2}{|l|}{¢－\(\square\) SIM＿ZYK＿T＿26} & WPD & & 11／30／09 & 3：49：17 PM \\
\hline \multicolumn{2}{|l|}{} & WPD & & 12／03／09 & 8：39：49 AM \\
\hline \multicolumn{2}{|l|}{\multirow[t]{2}{*}{\[
\begin{aligned}
\text { 直 UT } \\
\text { 甲 }
\end{aligned}
\]}} & MPF & 205 & 12／03／09 & 3：22：48 PM \\
\hline & & WPD & & 11／30／09 & 3：49：33 PM \\
\hline
\end{tabular}

Figure 13－2 Active program shown in green

\section*{13．1．1 NC memory}

The complete NC working memory is displayed along with all tools and the main programs and subroutines．

You can create further subdirectories here．

\section*{Proceed as follows}

NC

1．Select the＂Program manager＂operating area．

2．Press the＂NC＂softkey．

\subsection*{13.1.2 Local drive}

Workpieces, main and subprograms that are saved in the user memory of the CF card or on the local hard disk are displayed.
For archiving, you have the option of mapping the structure of the NC memory system or to create a separate archiving system.
You can create any number of subdirectories here, in which you can store any files (e.g. text files with notes).


\section*{Software options}

To display the "Local drive" softkey, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 or PC/PG).

\section*{Procedure}

1. Select the "Program manager" operating area.

On the local drive, you have the option of mapping the directory structure of the NC memory. This also simplifies the search sequence.

\section*{Creating Directories}

1. The local drive is selected.

2. Position the cursor on the main directory.

New

\section*{Directory}

4. In the "Name" entry field, enter "mpf.dir", "spf.dir" and "wks.dir" and press the "OK" softkey.
The directories "Part programs", "Subprograms" and "Workpieces" are created below the main directory.

\subsection*{13.1.3 USB drives}

USB drives enable you to exchange data. For example, you can copy to the NC and execute programs that were created externally.

\section*{NOTICE}

Interruption of operation
Direct execution from the USB FlashDrive is not recommended, because machining can be undesirably interrupted, therefore resulting in workpiece damage.

\section*{Partitioned USB-FlashDrive (only 840D sl and TCU)}

If the USB-FlashDrive has several partitions, these are displayed in a tree structure as a subtree ( \(01,02, \ldots\) ).

For EXTCALL calls, enter the partition (e.g. USB:/02/... or //ACTTCU/FRONT/02/... or //ACTTCU/FRONT,2/... or //TCU/TCU1/FRONT/02/...)

You can also configure any partition (e.g. //ACTTCU/FRONT,3).

\section*{Procedure}


\section*{Note}

The "USB" softkey can only be operated when a USB-FlashDrive is inserted in the front interface of the operator panel.

\subsection*{13.1.4 FTP drive}

The FTP drive offers you the following options - to transfer data, e.g. part programs, between your control system and an external FTP server.
You have the option of archiving any files in the FTP server by creating new directories and subdirectories.

\section*{Note}

\section*{Selecting a program / execution}

It is not possible to select a program directly on the FTP drive, and change to execution in the "Machine" operating area.

\section*{Precondition}

User name and password have been set up in the FTP server.

\section*{Procedure}
1. Select the "Program manager" operating area.
2. Press the "FTP" softkey.

When selecting the FTP drive for the first time, a login window is displayed.
3. Enter the user name and password and press the "OK" softkey to log into the FTP server.
The content of the FTP server with its folders is displayed.
4. Press the "Log out" softkey after the required data processing has been completed.
The connection to the FTP server is disconnected. In order to reselect the FTP drive, you must log on again.

\subsection*{13.2 Opening and closing the program}

To view a program in more detail or modify it, open the program in the editor.
With programs that are in the NCK memory, navigation is already possible when opening. The program blocks can only be edited when the program has been opened completely. You can follow the opening of the program in the dialog line.

With programs that are opened via local network, USB FlashDrive or network connections, navigation is only possible when the program has been opened completely. A progress message box is displayed when opening the program.

\section*{Note}

Channel changeover in the editor
When opening the program, the editor is opened for the currently selected channel. This channel is used to simulate the program.

If you change over a channel in the editor, this does not influence the editor. Only when closing the editor do you change into the other channel.

\section*{Procedure}
1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the program that you would like to edit.
3. Press the "Open" softkey.
- OR -

Press the <INPUT> key.
- OR -

Press the <Cursor right> key.
- OR -

Double-click the program.
The selected program is opened in the "Editor" operating area.
4. Now make the necessary program changes.
5. Press the "NC Select" softkey to switch to the "Machine" operating area and begin execution.

When the program is running, the softkey is deactivated.

\section*{Closing the program}


Close
- OR -


If you are at the start of the first line of the program, press the <Cursor left> key to close the program and the editor.
Press the ">>" and "Exit" softkeys to close the program and editor again.


To reopen a program you have exited with "Close", press the "Program" key.

\section*{Note}

A program does not have to be closed in order for it to be executed.

\subsection*{13.3 Executing a program}

When you select a program for execution, the control switches automatically to the "Machine" operating area.

\section*{Program selection}

Select the workpieces (WPD), main programs (MPF) or subprograms (SPF) by placing the cursor on the desired program or workpiece.

For workpieces, the workpiece directory must contain a program with the same name. This program is automatically selected for execution (e.g. when you select the workpiece SHAFT.WPD, the main program SHAFT.MPF is automatically selected).
If an INI file of the same name exists (e.g. SHAFT.INI), it will be executed once at the first part program start after selection of the part program. Any additional INI files are executed in accordance with machine data MD11280 \$MN_WPD_INI_MODE.

MD11280 \$MN_WPD_INI_MODE=0:
The INI file with the same name as the selected workpiece is executed. For example, when you select SHAFT1.MPF, the SHAFT1.INI file is executed upon <CYCLE START>.

MD11280 \$MN_WPD_INI_MODE=1:
All files of type SEA, GUD, RPA, UFR, PRO, TOA, TMA and CEC which have the same name as the selected main program are executed in the specified sequence. The main programs stored in a workpiece directory can be selected and processed by several channels.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Procedure}

Execute
1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the workpiece/program that you would like to execute.
3. Press the "Select" softkey.

The control switches automatically into the "Machine" operating area. - OR -


〈
CYCLE
START

If the selected program is already opened in the "Program" operating area,
Press the "Execute NC" softkey.
Press the <CYCLE START> key.
Execution of the workpiece is started.

\section*{Note}

\section*{Program selection from external media}

If you execute programs from an external drive (e.g. network drive), you require the "Execution from External Storage (EES)" software option.

\subsection*{13.4 Creating a directory / program / job list / program list}

\subsection*{13.4.1 Creating a new directory}

Directory structures help you to manage your program and data transparently. At all storage locations, you can create subdirectories for this purpose in a directory.

In a subdirectory, in turn, you can create programs and then create program blocks for them.

\section*{Note}

\section*{Restrictions}
- Directory names must end in .DIR or .WPD.
- The maximum name length is 28 characters including the extension.

All letters (except accented characters), numbers and underscores are permitted for name assignment. These names are automatically converted to upper-case letters. This limitation does not apply for work on USB/network drives.

\section*{Procedure}


Local drive đ USB


\section*{Directory}

1. Select the "Program manager" operating area.
2. Select the desired storage medium, i.e. a local or USB drive.
3. If you want to create a new directory in the local network, place the cursor on the topmost folder and press the "New" and "Directory" softkeys.
The "New Directory" window opens.
4. Enter the desired directory name and press the "OK" softkey.

\subsection*{13.4.2 Creating a new workpiece}

You can set up various types of files such as main programs, initialization files, tool offsets, etc. in a workpiece.

\section*{Note}

\section*{Workpiece directories}

You have the option of nesting tool directories. You must note that the length of the call line is restricted. You will be informed if the maximum number of characters is reached when entering the workpiece name.

\section*{Procedure}


The name can be a maximum of 24 characters long.
You can use any letters (except accented), digits or the underscore symbol (_).
The directory type (WPD) is set by default.
A new folder with the workpiece name will be created.
The "New G Code Program" window opens.
6. Press the "OK" softkey again if you want to create the program.

The program will open in the editor.

\subsection*{13.4.3 Creating a new G code program}

You can create G code programs and then render G code blocks for them in a directory/workpiece.

\section*{Procedure}


\section*{New}

\section*{programGUIDE}

G code
1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the folder in which you would like to store the program.
3. Press the "New" softkey.

The "New G Code Program" window opens.
4. If necessary, select a template if any are available.
5. Select the file type (MPF or SPF).

If you are in the NC memory and have selected either the "Subprograms" or "Part programs" folder, you can only create one subprogram (SPF) or one main program (MPF).
6. Enter the desired program name and press the "OK" softkey.

Program names can be a maximum of 24 characters long.
You can use all letters (with the exception of special characters, lan-guage-specific special characters, Asian or Cyrillic characters), numbers and underscores (_).

\subsection*{13.4.4 New ShopTurn program}

In the part program and workpiece directories, you can create ShopTurn programs and then subsequently generate the machining steps for them.

\section*{Procedure}
1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the folder in which you would like to store the program.
3. Press the "New" softkey.
4. Press the "ShopTurn" softkey.

The "New Step Sequence Program" window opens.
The "ShopTurn" type is specified.
5. Enter the desired program name and press the "OK" softkey.

The program name can contain up to 28 characters (name + dot +3 character extension).
You can use all letters (with the exception of special characters, lan-guage-specific special characters, Asian or Cyrillic characters), numbers and underscores (_).

\subsection*{13.4.5 Storing any new file}

In each directory or subdirectory you can create a file in any format that you specify.

\section*{Note}

File extensions
In the NC memory, the extension must have 3 characters, and DIR or WPD are not permitted.

In the NC memory, you can create the following file types under a workpiece using the "Any" softkey.
\begin{tabular}{|c|c|c|}
\hline \multicolumn{3}{|c|}{Any new program} \\
\hline \multirow[t]{4}{*}{\begin{tabular}{l}
Template \\
Type \\
Name
\end{tabular}} & J0BLIST.JOB & \(\checkmark\) \\
\hline & Job list JOB & \(\checkmark\) \\
\hline & Job list JOB & \\
\hline & \multirow[t]{2}{*}{\begin{tabular}{l}
Tool data TOA \\
TMA magazine assignment UFR zero points \\
R variable RPA \\
Definitions GUD \\
Setting data SEA \\
Protection zones PR0 \\
CEC sag compensation \\
Initialization program \(\mathbb{N} \mid\)
\end{tabular}} & \\
\hline & & \\
\hline
\end{tabular}

Turning
13.4 Creating a directory / program / job list / program list

\section*{Procedure}


New

Any
1. Select the "Program manager" operating area.
2. Select the desired storage location and position the cursor on the folder in which you would like to create the file.
3. Press the "New" and "Any" softkeys.

The "Any New Program" window appears
4. Select a file type from the "Type" selection field (for example, "Definitions GUD") and enter the name of the file to be created when you have selected a workpiece directory in the NC memory.
The file automatically has the selected file format.
- OR -

Enter a name and file format for the file to be created (e.g. My_Text.txt).
The name can be a maximum of 24 characters long.
You can use any letters (except accented), digits or the underscore symbol (_).
5. Press the "OK" softkey.

\subsection*{13.4.6 Creating a job list}

For every workpiece, you can create a job list for extended workpiece selection.
In the job list, you specify instructions for program selection in different channels.

\section*{Syntax}

The job list contains the SELECT instructions.

\section*{SELECT <program> CH=<channel number> [DISK]}

The SELECT instruction selects a program for execution in a specific NC channel. The selected program must be loaded into the working memory of the NC. The DISK parameter enables the selection of external execution (CF card, USB data carrier, network drive).
- <Program>

Absolute or relative path specification of the program to be selected.
Examples:
- I/NC/WCS.DIR/SHAFT.WPD/SHAFT1.MPF
- SHAFT2.MPF
- <Channel number>

Number of the NC channel in which the program is to be selected.
Example:
\(\mathrm{CH}=2\)
- [DISK]

Optional parameter for programs that are not in the NC memory and are to be executed "externally".

Example:
SELECT //remote/myshare/shaft3.mpf CH=1 DISK

\section*{Comment}

Comments are identified in the job list by ";" at the start of the line or by round brackets.

\section*{Template}

You can select a template from Siemens or the machine manufacturer when creating a new job list.

\section*{Executing a workpiece}

If the "Select" softkey is selected for a workpiece, the syntax of the associated job list is checked and then executed. The cursor can also be placed on the job list for selection.

\section*{Procedure}

1. Select the "Program manager" operating area.
2. Press the "NC" softkey, and in directory "Workpieces" place the cursor on the program for which you wish to create a job list.
3. Press the "New" and "Any" softkeys. The "Any New Program" window opens.
4. Select entry "Job list JOB" from the "Type" selection field and enter a name and press the "OK" softkey.

Turning

\subsection*{13.4.7 Creating a program list}

You can also enter programs in a program list that are then selected and executed from the PLC.

The program list may contain up to 100 entries
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Procedure}

1. Select the "Program manager" operating area.
2. Press the menu forward key and the "Program list" softkey. The "Prog.-list" window opens.
3. Place the cursor in the desired line (program number).

Select program

6. To remove a program from the list, place the cursor on the appropriate line and press the "Delete" softkey.
- OR -

To delete all programs from the program list, press the "Delete all" softkey.

\subsection*{13.5 Creating templates}

You can store your own templates to be used for creating part programs and workpieces. These templates provide the basic framework for further editing.

You can use them for any part programs or workpieces you have created.

\section*{Storage location for templates}

The templates used to create part programs or workpieces are stored in the following directories:
HMI Data/Templates/Manufacturer/Part programs or Workpieces
HMI Data/Templates/User/Part programs or Workpieces

\section*{Procedure}


Copy

Paste
1. Select the "Start-up" operating area.
2. Press the "System data" softkey.
3. Position the cursor on the file that you wish to store as a template and press the "Copy" softkey.
4. Select the directory in which you want to store the data - "Part programs" or "Workpieces" - and press the "Paste" softkey.

Stored templates can be selected when a part program or a workpiece is being created.

\subsection*{13.6 Searching directories and files}

You have the possibility of searching in the Program Manager for certain directories and files.

\section*{Note}

Search with place holders
The following place holders simplify the search:
- "*": replaces any character string
- "?": replaces any character

\section*{Search strategy}

The search is made in all of the selected directories and their subdirectories.
If the cursor is positioned on a file, then a search is made from the higher-level directory.

\section*{Note}

\section*{Searching in opened directories}

Open the closed directories for a successful search.

\section*{Procedure}
1. Select the "Program Manager" operating area.
2. Select the storage location in which you wish to perform the search and then press the ">>" and "Search" softkeys.
The "Find file" window appears.
3. Enter the desired search term in the "Text" field.

Note: When searching for a file, enter the complete name with extension (e.g. DRILLING.MPF).
4. When required, activate the checkbox "Observe upper and lower case".
5. Press the "OK" softkey to start the search.

OK
6. If a corresponding directory or file is found, then it is marked.
```

Continue
search
OK

- OR -
X
Cancel
OR
Press the "Cancel" softkey when you want to cancel the search.

```

\subsection*{13.7 Displaying the program in the Preview.}

You can show the content on a program in a preview before you start editing.

\section*{Procedure}


Preview window

Preview window
1. Select the "Program manager" operating area.
2. Select a storage location and place the cursor on the relevant program.
3. Press the ">>" and "Preview window" softkeys.

The "Preview: ..." window opens.
4. Press the "Preview window" softkey again to close the window.

\subsection*{13.8 Selecting several directories/programs}

You can select several files and directories for further processing. When you select a directory, all directories and files located beneath it are also selected.

\section*{Note}

Selected files
If you have selected individual files in a directory, then this selection is canceled when the directory is closed.

If the complete directory with all of the files included in it are selected, then this selection is kept when closing the directory.

\section*{Procedure}

Mark

Mark

\section*{Mark}
1. Select the "Program manager" operating area.
2. Choose the desired storage location and position the cursor on the file or directory from which you would like your selection to start.
3. Press the "Select" softkey.

The softkey is active.
4. Select the required directories/programs with the cursor keys or mouse.
5. Press the "Select" softkey again to deactivate the cursor keys.

\section*{Canceling a selection}

By reselecting an element, the existing selection is canceled.

\section*{Selecting via keys}


\section*{Selecting with the mouse}
\begin{tabular}{|l|l|}
\hline Key combination & Meaning \\
\hline Left mouse & \begin{tabular}{l} 
Click on element: The element is selected. \\
A previously existing selection is canceled.
\end{tabular} \\
\hline \begin{tabular}{l} 
Left mouse + \\
\hline \begin{tabular}{|c|} 
SHilf
\end{tabular}
\end{tabular} & \begin{tabular}{l} 
Expand selection consecutively up to the next click. \\
Pressed
\end{tabular} \\
\hline \begin{tabular}{l} 
Left mouse +
\end{tabular} & \begin{tabular}{l} 
Expand selection to individual elements by clicking. \\
An existing selection will expand to include the element you clicked. \\
\hline CTRL
\end{tabular} \\
\begin{tabular}{l} 
Pressed
\end{tabular} & \\
\hline
\end{tabular}

\subsection*{13.9 Copying and pasting a directory/program}

To create a new directory or program that is similar to an existing program, you can save time by copying the old directory or program and only changing selected programs or program blocks.

The capability of copying and pasting directories and programs can also be used to exchange data with other systems via USB/network drives (e.g. USB FlashDrive).

Copied files or directories can be pasted in a different location.

\section*{Note}

You can only paste directories on local drives and on USB or network drives.

\section*{Note}

\section*{Write protection}

If the current directory is write-protected for the user, then the function is not offered.

\section*{Note}

When you copy directories, any missing endings are added automatically.
All letters (except accented characters), numbers, and underscores are permitted for name assignment. The names are automatically converted to upper-case letters, and extra dots are converted to underscores.

\section*{Example}

If the name is not changed during the copy procedure, a copy is created automatically: MYPROGRAM.MPF is copied to MYPROGRAM_1.MPF. The next time it is copied, it is changed to MYPROGRAM__2.MPF, etc.

If the files MYPROGRAM.MPF, MYPROGRAM_1.MPF, and MYPROGRAM__3.MPF already exist in a directory, MYPROGRAM__2.MPF is created as the next copy of MYPROGRAM.MPF.

\section*{Procedure}
1. Select the "Program manager" operating area.
2. Choose the desired storage location and position the cursor on the file or directory which you would like to copy.
3. Press the "Copy" softkey.
4. Select the directory in which you want to paste your copied directory/program.

Overwrite all

No overwriting

\section*{Skip}

6. Press the "OK" or "Overwrite all" softkey if you want to overwrite existing directories/programs.
5. Press the "Paste" softkey.

If a directory/program of the same name already exists in this directory, you are are informed. You are requested to assign a new name, otherwise the directory/program is assigned a name by the system.
If the name contains illegal characters or is too long, a prompt will appear for you to enter a permissible name.
- OR -

Press the "No overwriting" softkey if you do not want to overwrite already existing directories/programs.
- OR -

Press the "Skip" softkey if the copy operation is to be continued with the next file.
- OR -

Enter another name if you want to paste the directory/program under another name and press the "OK" softkey.

\section*{Note}

\section*{Copying files in the same directory}

You cannot copy files to the same directory. You must copy the file under a new name.

\subsection*{13.10 Deleting a directory/program}

Delete programs or directories from time to time that you are no longer using to maintain a clearer overview of your data management. Back up the data beforehand, if necessary, on an external data medium (e.g. USB FlashDrive) or on a network drive.

Please note that when you delete a directory, all programs, tool data and zero point data and subdirectories that this directory contains are deleted.

\section*{Temp directory for ShopTurn}

If you want to free up space in the NCK memory, delete the contents of the "TEMP" directory. This is where ShopTurn stores the programs that are created internally for calculating solid machining operations

\section*{Procedure}

\(x\)
Cance
1. Select the "Program manager" operating area.
2. Choose the desired storage location and position the cursor on the file or directory that you would like to delete.
3. Press the ">>" and "Delete" softkeys.

A prompt appears as to whether you really want to delete the file or directory.
4. Press the "OK" softkey to delete the program/directory.
- OR -

Press the "Cancel" softkey to cancel the process.

\subsection*{13.11 Changing file and directory properties}

Information on directories and files can be displayed in the "Properties for ..." window.
Information on the creation date is displayed near the file's path and name.
You can change names.

\section*{Changing access rights}

Access rights for execution, writing, listing and reading are displayed in the "Properties" window.
- Execute: Is used for the selection for execution
- Write: Controls the changing and deletion of a file or a directory

For NC files, you have the option to set the access rights from keyswitch 0 to the current access level, to be set separately for each file.

If an access level is higher than the current access level, it cannot be changed.
For external files (e.g. on a local drive), the access rights are displayed to you only if settings have been executed for these files by the machine manufacturer. They cannot be changed via the "Properties" window.

\section*{Settings for the access rights to directories and files}

Via a configuration file and MD 51050, access rights of the directories and file types of the NC and user memory (local drive) can be changed and pre-assigned.

\section*{References}

A detailed description of the configuration can be found in the following documentation: Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Procedure}
1. Select the program manager.
2. Choose the desired storage location and position the cursor on the file or directory whose properties you want to display or change.
3. Press the ">>" and "Properties" softkeys. The "Properties from ..." window appears.
4. Enter any necessary changes.

Note: You can save changes via the user interface in the NC memory.
5. Press the "OK" softkey to save the changes.

\subsection*{13.12 Set up drives}

\subsection*{13.12.1 Overview}

Up to 21 connections to so-called logical drives (data carriers) can be configured. These drives can be accessed in the "Program manager" and "Startup" operating areas.

The following logical drives can be set up:
- USB interface
- Network drives
- CompactFlash card
- CompactFlash card of the NCU, only for SINUMERIK Operate in the NCU (for 840D sI)
- Local hard disk of the PCU, only for SINUMERIK Operate on the PCU (for 840D sl)


Software option - for 840D sl
In order to use the CompactFlash card as data carrier, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU or PC).


\section*{Software option - for 828D}

You will need the "Manage network drives" option to manage additional drives via Ethernet.

\section*{Note}

The USB interfaces of the NCU are not available for SINUMERIK Operate and therefore cannot be configured.
(for 840D sl)

\subsection*{13.12.2 Setting up drives}

The "Set Up Drives" window is available in the "Start-up" operating area for configuring the softkeys in the Program Manager.

\section*{Note}

\section*{Reserved softkeys}

Softkeys 4, 7 and 16 are not available to be freely configured.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

File
The created configuration data is stored in the "logdrive.ini" file. This file is located in the /user/sinumerik/hmi/cfg directory.

\section*{General information}
\begin{tabular}{|c|c|c|}
\hline \multicolumn{2}{|l|}{Entry} & Meaning \\
\hline \multicolumn{3}{|l|}{Drives 1-24} \\
\hline \multirow[t]{11}{*}{Type} & No drive & No drive defined. \\
\hline & NC program memory & Access to the NC memory. \\
\hline & USB local & Access to the USB interface of the active operator unit. \\
\hline & USB global & All of the TCUs in the plant network can access the USB memory medium. \\
\hline & NW Windows & Network drive in Windows systems. \\
\hline & NW Linux & Network drive in Linux systems. \\
\hline & Local drive & \begin{tabular}{l}
Local drive. \\
Hard disk or user memory on the CompactFlash card.
\end{tabular} \\
\hline & FTP & \begin{tabular}{l}
Access to an external FTP server. \\
The drive cannot be used as global part program memory.
\end{tabular} \\
\hline & User cycles & Access to the user cycle directory of the CompactFlash card. \\
\hline & Manufacturer cycles & Access to the manufacturer cycle directory of the CompactFlash card. \\
\hline & Drive Windows & Access to a local PCU/PC directory. \\
\hline
\end{tabular}

\section*{Specifications for USB}
\begin{tabular}{|l|l|l|}
\hline \multicolumn{2}{|l|}{ Entry } & \\
\hline \multirow{4}{*}{ Device } & Description \\
\hline \multirow{4}{*}{ Connection } & Front & \begin{tabular}{l} 
Names of the TCU to which the USB storage \\
medium is connected, e.g. tcu1. The NCU \\
must already know the TCU name.
\end{tabular} \\
\cline { 2 - 3 } & X203/X204 & \begin{tabular}{l} 
USB interface that is located at the front of \\
the operator panel.
\end{tabular} \\
\cline { 2 - 3 } & X204 & \begin{tabular}{l} 
USB interface X203/X204 that is located at \\
the rear of the operator panel.
\end{tabular} \\
\cline { 2 - 3 } & X212/X213 & \begin{tabular}{l} 
For SIMATIC Thin Client the USB interface is \\
X204.
\end{tabular} \\
\cline { 2 - 4 } & X20 & TCU20.2/20.3 \\
\hline Symbolic & & OP 08T
\end{tabular}\(|\)\begin{tabular}{l|l|}
\hline PCU & Symbolic drive name. \\
\hline Additional parameters under Details & \\
\hline Partition & \begin{tabular}{l} 
Partition number on the USB storage medi- \\
um, e.g. 1 or all. \\
If a USB hub is being used, then specify the \\
USB port of the hub.
\end{tabular} \\
\hline USB path & \\
\hline
\end{tabular}

\section*{Specifications for local drives}
\begin{tabular}{|l|l|l|}
\hline Entry & & Description \\
\hline Symbolic & \begin{tabular}{l} 
Symbolic drive name. \\
Assignment of the names under Details
\end{tabular} \\
\hline \multicolumn{3}{|l|}{ Additional parameters under Details } \\
\hline \multirow{3}{*}{ Use drive as: } & LOCAL_DRIVE & \begin{tabular}{l} 
The activation of the checkbox assigns the \\
symbolic name to the drive. \\
If an assignment exists already for the drive, \\
no change can be made. \\
All checkboxes are active as preassignment.
\end{tabular} \\
\cline { 2 - 3 } & CF_CARD & SYS_DRIVE
\end{tabular}

\section*{Specifications for network drives}
\begin{tabular}{|l|l|l|}
\hline \multicolumn{2}{|l|}{ Entry } & \\
\hline Description \\
\hline Computer name & \begin{tabular}{l} 
Only for network drives in \\
Windows systems.
\end{tabular} & \begin{tabular}{l} 
Logical name of the server or the IP address. \\
released
\end{tabular} \\
\hline Path & & \begin{tabular}{l} 
Start directory. \\
The path is specified relative to the released \\
directory.
\end{tabular} \\
\hline \begin{tabular}{l} 
User name \\
Password
\end{tabular} & \begin{tabular}{l} 
Enter the user name and the corresponding \\
password for which the directory is enabled \\
on the server. \\
The password is displayed in encoded form \\
as string of "*" characters and is stored in the \\
"logdrive.ini" file.
\end{tabular} \\
\hline Symbolic & & \begin{tabular}{l} 
Symbolic drive name. \\
Maximum 12 characters can be entered \\
(letters, numbers, underscore).
\end{tabular} \\
The names NC, GDIR and FTP are reserved. \\
It is also used to label the softkey if a softkey \\
text is not specified.
\end{tabular}

\section*{Specifications for FTP}
\begin{tabular}{|l|l|l|}
\hline Entry & & Description \\
\hline Computer name & & \begin{tabular}{l} 
Logical name of the FTP server or the IP \\
address.
\end{tabular} \\
\hline Path & \begin{tabular}{l} 
Start directory on the FTP server. \\
The path is specified relative to the home \\
directory.
\end{tabular} \\
\hline \begin{tabular}{l} 
User name \\
Password
\end{tabular} & \begin{tabular}{l} 
User names and the associated password for \\
login to the FTP server. \\
The password is displayed in encoded form \\
as string of "*" characters and is stored in the \\
"logdrive.ini" file.
\end{tabular} \\
\hline Additional parameters under Details & \multicolumn{2}{|l|}{\begin{tabular}{l} 
Port
\end{tabular}} \\
\hline Disconnect & \begin{tabular}{l} 
Interface for the FTP connection. The default \\
port is 21.
\end{tabular} \\
\hline & \begin{tabular}{l} 
After a disconnect timeout, the FTP connec- \\
tion is disconnected. The timeout can be \\
between 1 and 150 s. 10 s is the default \\
setting.
\end{tabular} \\
\hline
\end{tabular}

\section*{Additional specifications when using the "EES" software option}

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.
\begin{tabular}{|l|l|l|}
\hline Entry & \begin{tabular}{l} 
Only for "Drive Windows \\
(PCU)" type
\end{tabular} & \begin{tabular}{l} 
Description \\
\hline Enable drive \\
name is required. \\
The checkbox must be activated if the local \\
drive serves as global part program memory.
\end{tabular} \\
\hline \begin{tabular}{l} 
Global part program \\
memory
\end{tabular} & \begin{tabular}{l} 
Only for local drives, \\
network drives and global \\
USB drives
\end{tabular} & \begin{tabular}{l} 
The checkbox defines as to whether all sys- \\
tem participants have access to the config- \\
ured logical drive. The participants can \\
directly execute part programs from the drive. \\
Only one drive can be selected as global part \\
program memory (GDIR). If another drive has \\
already been defined as GDIR and the \\
checkbox is activated, the original setting is \\
deleted.
\end{tabular} \\
\hline \begin{tabular}{l} 
Use this drive for EES \\
program execution
\end{tabular} & Only for USB drives & \begin{tabular}{l} 
Allows a local USB storage medium to be \\
used to execute programs using EES.
\end{tabular} \\
\hline Additional parameters under Details for USB drives, local drives and local directories
\end{tabular}\(|\)\begin{tabular}{l} 
User name and the associated password for \\
release of the configured drive. \\
Windows user name \\
Windows password
\end{tabular}

\section*{Specifications for the configured softkey}
\begin{tabular}{|l|l|l|}
\hline Entry & & \begin{tabular}{l} 
Description \\
\hline Access level \\
Softkey text
\end{tabular} \begin{tabular}{l} 
Assign access rights to the connections: \\
From access level 7 (keyswitch position 0) to \\
access level 1 (manufacturer). \\
The particular assigned access level applies \\
to all operating areas.
\end{tabular} \\
\hline \begin{tabular}{l} 
Two lines are available as labeling text for \\
the softkey. \%n is accepted as a line separa- \\
tor. \\
If the first line is too long, then a line break is \\
automatically inserted. \\
If a space is present, it is taken as a line \\
separator. \\
For language-dependent softkey texts, the \\
text ID is entered, which is used to search in \\
the text file. \\
If nothing is specified in the entry field, then \\
the symbolic drive name is used as softkey \\
text.
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|c|}
\hline \multicolumn{2}{|l|}{Entry} & Description \\
\hline \multirow[t]{4}{*}{Softkey icon} & No icon & No icon is displayed on the softkey. \\
\hline & sk_usb_front.png & \multirow[t]{3}{*}{File names of the icon displayed on the softkey.} \\
\hline & sk_local_drive.png & \\
\hline & sk_network_drive_ftp.png & \\
\hline Text file & slpmdialog & \multirow[t]{2}{*}{File for softkey dependent on the language. If nothing is specified in the input fields, the text appears on the softkey as was specified in the input field "Softkey text".} \\
\hline Text context & SIPmDialog & \\
\hline
\end{tabular}

\section*{Procedure}


Setup

Log. drives

>> Level

\section*{Change}

Details
3. Select the softkey that you want to configure.
1. Select the "Start-up" operating area.
2. Press the "HMI" and "Log. drive" softkeys.

The "Set Up Drives" window opens.
4. To configure softkeys 9 to 16 or softkeys 17 to 24 , click the ">> level" softkey.
5. To allow entry fields to be edited, press the "Change" softkey
6. Select the data for the corresponding drive or enter the necessary data
7. Press the "Details" softkey if you want to enter additional parameters. Press the "Details" softkey to return to the "Set Up Drives" window.
8. Press the "OK" softkey.

The entries are checked.

A window with the appropriate message opens if the data is incomplete or incorrect. Acknowledge the message with softkey "OK".

If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.
9. Restart the control in order to activate the configuration and to obtain the softkeys in the "Program Manager" operating area.

\section*{Entering the default settings for drive release}

\section*{Note}

This function is available only on Windows systems when the "Execution from External Storage (EES)" software option is activated.


Setup

HMI

Log. drives

Global settings
\(x\)
Cancel
1. Select the "Start-up" operating area.
2. Press the "HMI" and "Log. drive" softkeys. The "Set Up Drives" window opens.
3. Press the "Glob. settings" softkey.
4. Enter the user name and the associated password for the configured drives to be released.
5. Press the "OK" softkey.

The specifications are transferred as default setting for the Windows release.

If you press the "Cancel" softkey, then all of the data that has not been activated is rejected.

\subsection*{13.13 Viewing PDF documents}

You have the option of displaying HTML documents, as well as PDFs, on all drives of the program manager via the data tree of the system data.

\section*{Note}

A preview of the documents is only possible for PDFs.

\section*{Procedure}


Open

\section*{Zoom +}

Zoom -

\section*{Search}

\section*{Uiew}

Zoom page width

Zoom page length
1. In the "Program manager" operating area, select the desired storage medium.

\section*{- OR -}

Select the desired storage location in the "Commissioning" operating area in the data tree of the "System data".
2. Position the cursor on the PDF or the HTML file that you want to display, and press the "Open" softkey.
The selected file is displayed on the screen.
The storage path of the document is displayed in the status bar. The current page as well as the total number of pages of the document are displayed.
3. Press the "Zoom +" or "Zoom -" softkey to enlarge of reduce the size of the display.
4. Press the "Search" softkey if you want to search for specific texts in the PDF.
5. Press the "View" softkey to change the representation of the PDF. A new vertical softkey bar appears.
6. Press the "Zoom page width" softkey to display the document in full width on the screen.
- OR -

Press the "Zoom page height" softkey to display the document with full height on the screen.
- OR -

Press the "Rotate left" softkey to rotate the document through 90 degrees to the left.
- OR -

Turning

< Back

Close

Press the "Rotate right" softkey to rotate the document through 90 degrees to the right.
7. Press the "Back" softkey to return to the previous window.
8. Press the "Close" softkey to exit the PDF display.

\subsection*{13.14 EXTCALL}

The EXTCALL command can be used to access files on a local drive, USB data carriers or network drives from a part program.

The programmer can set the source directory with the setting data SD \$SC42700
EXT_PROG_PATH and then specify the file name of the subprogram to be loaded with the EXTCALL command.

\section*{Supplementary conditions}

The following supplementary conditions must be taken into account with EXTCALL calls:
- You can only call files with the MPF or SPF extension via EXTCALL from a network drive.
- The files and paths must comply with the NCK naming conventions (max. 25 characters for the name, 3 characters for the identifier).
- A program is found on a network drive with the EXTCALL command if
- with SD \$SC42700 EXT_PROG_PATH the search path refers to the network drive or a directory contained on the network drive. The program must be stored directly on that level, no subdirectories are searched.
- without SD \$SC42700 the correct location of the program is specified in the EXTCALL call itself by means of a fully qualified path that can also point to a subdirectory of the network drive.
- For programs that were generated on external storage media (Windows system) observe upper- and lower-case syntax.

\section*{Note}

\section*{Maximum path length for EXTCALL}

The path length must not exceed 112 characters. The path comprises the contents of the setting data (SD \$SC42700) and the path data for EXTCALL call from the part program.

\section*{Examples of EXTCALL calls}

The setting data can be used to perform a targeted search for the program.
- Call of USB drive on TCU (USB storage device on interface X203), if SD42700 is empty: e.g. EXTCALL "//TCU/TCU1 /X203 ,1/TEST.SPF"
- OR -

Call of USB drive on TCU (USB storage device on interface X203), if SD42700 "//TCU/TCU1 /X203 ,1" contains: EXTCALL "TEST.SPF"
- Call of the USB front connection (USB-FlashDrive), if SD \$SC 42700 is empty: e.g. EXTCALL "//ACTTCU/FRONT,1/TEST.SPF"
- OR -

Call of USB front connection (USB-FlashDrive), if SD42700 "//ACTTCU/FRONT,1" contains: EXTCALL "TEST.SPF"
- Call of network drive, if SD42700 is empty: e.g. EXTCALL "//computer name/enabled drive/TEST.SPF"
- OR -

Call of the network drive, if SD \$SC42700 "//Computer name/enabled drive" contains: EXTCALL "TEST.SPF"
- Use of the HMI user memory (local drive):
- On the local drive, you have created the directories part programs (mpf.dir), subprograms (spf.dir) and workpieces (wks.dir) with the respective workpiece directories (.wpd):

SD42700 is empty: EXTCALL "TEST.SPF"
The same search sequence is used on the CompactFlash card as in the NCK part program memory.
- On the local drive, you have created your own directory (e.g. my.dir):

Specification of the complete path: e.g. EXTCALL
"/card/user/sinumerik/data/prog/my.dir/TEST.SPF"
A search is performed for the specified file.

\section*{Note}

Abbreviations for local drive, CompactFlash card and USB front connection
As abbreviation for the local drive, the CompactFlash card and the USB front connection you can use the abbreviation LOCAL_DRIVE:, CF_CARD: and USB: (e.g. EXTCALL "LOCAL_DRIVE:/spf.dir/TEST.SPF").

Alternatively, you can also use the abbreviations CF_Card and LOCAL_DRIVE.

\section*{Software options}

To display the "Local drive" softkey, you require the "Additional HMI user memory on CF card of the NCU" option (not for SINUMERIK Operate on PCU50 / PC).

\section*{NOTICE \\ Possible interruption when executing from USB FlashDrive \\ Direct execution from a USB-FlashDrive is not recommended. \\ There is no protection against contact problems, falling out, breakage through knocking or unintentional removal of the USB-FlashDrive during operation. \\ Disconnecting it during operation will result in immediate stopping of the machining and, thus, to the workpiece being damaged.}

\section*{Machine manufacturer}

Processing EXTCALL calls can be enabled and disabled.
Please refer to the machine manufacturer's specifications.

\subsection*{13.15 Execution from External Storage (EES)}

\subsection*{13.15.1 Overview}

The "Execution from external storage" function allows you to directly execute any size of part program from an external drive (e.g. local drive or network drive).


\section*{Software option}

You require the "Execution from External Storage (EES)" software option in order to use this function.


Machine manufacturer
Please refer to the machine manufacturer's instructions.

You have the option of processing the G code programs saved on the configured external drives as usual in the editor.

When executing the G code programs, as usual you obtain an actual block display of the program blocks presently being executed; you also have the option of directly processing the programs here in the reset state. In addition to the actual block display, you can also access the basis block display. You can also make corrections using the "Program correction" function.

\subsection*{13.16 Backing up data}

\subsection*{13.16.1 Generating an archive in the Program Manager}

You have the option of archiving individual files from the NC memory and the local drive.

\section*{Archive formats}

You have the option of saving your archive in the binary and punched tape format.

\section*{Save target}

The archive folder of the system data in the "Startup" operating area as well as USB and network drives are available as save target.

\section*{Procedure}


\section*{Mark}


Archive

> Generate
> archive

> Search

OK
1. Select the "Program Manager" operating area.
2. Select the storage location for the file/files to be archived.
3. In the directories, select the required file from which you want to create an archive.
- OR -

If you want to back up several files or directories, press the "Select" softkey and, using the cursor keys or the mouse, select the required directories or files.
4. Press the ">>" and "Archive" softkeys.
5. Press the "Generate archive" softkey.

The "Generate Archive: Select archiving" window opens.
6. Position the cursor to the required storage location, press the "Search" softkey, enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory.
Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
- OR -

Select the required storage location, press the "New directory" softkey, enter the required name in the "New directory" window and press the "OK" softkey to create a directory.
7. Press "OK".

The "Generate Archive: Name" window opens.
9. Select the format (e.g. archive ARC (binary format) for 840 sl or archive ARD for 828D), enter the desired name and press the "OK" softkey.
A message informs you if archiving was successful.

\subsection*{13.16.2 Generating an archive via the system data}

If you only want to backup specific data, then you can select the desired files directly from the data tree and generate an archive.

\section*{Archive formats}

You have the option of saving your archive in the binary and punched tape format.
You can display the contents of the selected files (XML, ini, hsp, syf files, programs) in a preview.

You can display information about the file, such as path, name, date of creation and change, in a Properties window.

\section*{Precondition}

The access rights depend on the relevant areas and range from protection level 7 (key switch position 0) to protection level 2 (password: Service).

\section*{Storage locations}
- CompactFlash card under /user/sinumerik/data/archive, or /oem/sinumerik/data/archive
- All configured logical drives (USB, network drives)


\section*{Software option}

In order to save archives on the CompactFlash card in the user area you require the option "Additional HMI user memory on CF card of NCU".

\section*{NOTICE}

Possible data loss when using USB flash drives
USB-FlashDrives are not suitable as persistent memory media.

\section*{Procedure}

\section*{Mark}


Preview window

\section*{Properties}

OK

\section*{Search}


Archive

Generate archive
1. Select the "Startup" operating area.
2. Press the "System data" softkey. The data tree opens.
3. In the data tree, select the required files from which you want to generate an archive.
- OR -

If you want to back up several files or directories, press the "Select" softkey and, using the cursor keys or the mouse, select the required directories or files.
4. If you press the ">>" softkey, further softkeys are displayed on the vertical bar.
5. Press the "Preview window" softkey.

The contents of the selected file are displayed in a small window. Press the "Preview window" softkey again to close the window.
6. Press the "Properties" softkey.

Information about the selected file is displayed in a small window. Press the "OK" softkey to close the window.
7. Press the "Search" softkey.

Enter the required search term in the search dialog and press the "OK" softkey if you wish to search for a specific directory or subdirectory.

Note: The place holders "*" (replaces any character string) and "?" (replaces any character) make it easier for you to perform a search.
8. Press the "Archive" and "Generate archive" softkeys.

The "Generate Archive: Select Storage Location" window opens.
The "Archive" folder with the subfolders "User" and "Manufacturer" as well as the storage media (e.g. USB) are displayed.
```

New directory

```


OK

OK
10. Enter the required name and press the "OK" softkey.

The directory is created below the selected folder.
11. Press the "OK" softkey.

The "Generate Archive: Name" window opens.
12. Select the format (e.g. archive ARC (binary format) for 840D sl and/or

Archive ARC for 828D), enter the desired name and press the "OK" softkey to archive the file(s).
A message informs you if archiving was successful.
13. Press the "OK" softkey to confirm the message and end the archiving operation.
An archive file in the .ARC (840D sl) or .ARD (828D) format type is created in the selected directory.
9. Select the required location for archiving and press the "New directory" softkey to create a suitable subdirectory.
The "New Directory" window opens.


\subsection*{13.16.3 Reading in an archive in the Program Manager}

In the "Program Manager" operating area, you have the option of reading in archives from
the archive folder of the system data as well as from configured USB and network drives.

\section*{Software option}

In order to read-in user archives in the "Program Manager" operating area, you require the option "Additional HMI user memory on CF Card of NCU" (not for 840D sl / SINUMERIK Operate on PCU50 / PC).

\section*{Procedure}

1. Select the "Program Manager" operating area.

Program manager

\section*{Archive}

Read-in archive
2. Press the "Archive" and "Read in archive" softkeys. The "Read in archive: Select archive" window opens.
3. Select the archive storage location and position the cursor on the required archive.
Note: When the option is not set, the folder for user archives is only displayed if the folder contains at least one archive.
- OR -

\section*{Search}


Overwrite
all
Press the "OK" or "Overwrite all" softkey to overwrite existing files.
- OR -

Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.
- OR -

Press the "Skip" softkey if the read-in operation is to be continued with the next file.

The "Read In Archive" window opens and a progress message box appears for the read-in process.

You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.
5. Press the "Cancel" softkey to cancel the read-in process.

\section*{See also}

Searching directories and files (Page 750)

\subsection*{13.16.4 Read in archive from system data}

If you want to read in a specific archive, you can select this directly from the data tree.

\section*{Procedure}

3. In the data tree below the "Archive" directory, in the "User" folder, select the file that you wish to read in.


\section*{Overwrite}
all

No overwriting

Skip
4. Press the "Read in" softkey.
5. Press the "OK" or "Overwrite all" softkey to overwrite existing files.
- OR -

Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.
- OR -

Press the "Skip" softkey if the read-in operation is to be continued with the next file.

The "Read In Archive" window opens and a progress message box appears for the read-in process.
You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.
6. Press the "Cancel" softkey to cancel the read-in process.

\subsection*{13.17 Setup data}

\subsection*{13.17.1 Backing up setup data}

Apart from programs, you can also save tool data and zero point settings.
You can use this option, for example, to back up tools and zero point data for a specific machining step program. If you want to execute this program at a later point in time, you will then have quick access to the relevant settings.

Even tool data that you have measured on an external tool setting station can be copied easily into the tool management system using this option.

\section*{Backing-up job lists}

If you wish to backup a job list, which contains ShopTurn and G code programs, you obtain dedicated selection boxes to backup the tool data and zero points.

\section*{Note}

\section*{Backing up setup data from part programs}

Setup data from part programs can only be backed up if they have been saved in the "Workpieces" directory.

For part programs, which are located in the "Part programs" directory, "Save setup data" is not listed.

\section*{Backing up data}
\begin{tabular}{|c|c|}
\hline Data & \\
\hline Tool data & \begin{tabular}{l}
- No \\
- All used in the program (only for ShopTurn program and job list only with ShopTurn programs) \\
- Complete tool list
\end{tabular} \\
\hline \begin{tabular}{l}
Tool data for ShopTurn programs \\
-- only available for job list with ShopTurn and G code programs
\end{tabular} & \begin{tabular}{l}
- No \\
- All used in the program \\
- Complete tool list
\end{tabular} \\
\hline \begin{tabular}{l}
Tool data for G code programs \\
-- only available for job list with ShopTurn and G code programs
\end{tabular} & \begin{tabular}{l}
- No \\
- Complete tool list
\end{tabular} \\
\hline Magazine assignment & \begin{tabular}{l}
- Yes \\
- No
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|c|c|}
\hline Data & \\
\hline Zero points & \begin{tabular}{l}
- No \\
The selection box "Basis zero point" is hidden \\
- All used in the program (only for ShopTurn program and job list only with ShopTurn programs) \\
- All
\end{tabular} \\
\hline \begin{tabular}{l}
Zero points for ShopTurn programs \\
-- only available for job list with ShopTurn and G code programs
\end{tabular} & \begin{tabular}{l}
- No \\
The selection box "Basis zero point" is hidden \\
- All used in the program \\
- Complete tool list
\end{tabular} \\
\hline \begin{tabular}{l}
Zero points for G code programs \\
-- only available for job list with ShopTurn and G code programs
\end{tabular} & \begin{tabular}{l}
- No \\
The selection box "Basis zero point" is hidden \\
- All
\end{tabular} \\
\hline Basic zero points & \begin{tabular}{l}
- No \\
- Yes
\end{tabular} \\
\hline Directory & The directory is displayed, in which the selected program is located. \\
\hline File name & Here you have the option of changing the suggested file names. \\
\hline
\end{tabular}

\section*{Note}

\section*{Magazine assignment}

You can only read out the magazine assignments if your system provides support for loading and unloading tool data to and from the magazine.

\section*{Procedure}


Program manager

\section*{NC NC}
...
Local
drive


Archive

Save setup data
1. Select the "Program Manager" operating area.
2. Position the cursor on the program whose tool and zero point data you wish to back up.
3. Press the ">>" and "Archive" softkeys.
4. Press the "Setup data" softkey.

The "Backup setup data" window opens.
5. Select the data you want to back up.
6. When required, change the specified name of the originally selected program here in the "File name" field.
7. Press the "OK" softkey.

The setup data will be set up in the same directory in which the selected program is stored.
The file is automatically saved as INI file.

\section*{Note}

\section*{Program selection}

If a main program as well as an INI file with the same name are in a directory, when selecting the main program, initially, the INI file is automatically started. In this way, unwanted tool data can be changed.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{13.17.2 Reading-in set-up data}

When reading-in, you can select which of the backed-up data you wish to read-in:
- Tool data
- Magazine assignment
- Zero points
- Basic zero point

\section*{Tool data}

Depending on which data you have selected, the system behaves as follows:
- Complete tool list

First, all tool management data are deleted and then the saved data are imported.
- All tool data used in the program

If at least one of the tools to be read in already exists in the tool management system, you can choose between the following options.

\section*{Overwrite} all

Select the "Replace all" softkey to import all tool data. Any existing tools will now be overwritten without a warning prompt.
- OR -

No owerwriting

Skip
Press the "Do not overwrite" softkey if existing tools must not be overwritten.
Already existing tools are skipped, without you receiving any queries.
- OR -

Press the "Skip" softkey if already existing tools are not to be overwritten.

For an already existing tool, you receive a query.

\section*{Selecting loading point}

For a magazine, if more than one loading point was set-up, using the "Select loading point" softkey, you have the option of opening a window in which you can assign a loading point to a magazine.

\section*{Procedure}


Program manager
1. Select the "Program Manager" operating area.
2. Position the cursor on the file with the backed-up tool and zero point data (*. INI ) that you wish to re-import.
3. Press the <Cursor right> key

\section*{- OR -}

Double-click the file.
The "Read-in setup data" window opens.
4. Select the data (e.g. magazine assignment) that you wish to read-in.
5. Press the "OK" softkey.

\subsection*{13.18 RS-232-C}

\subsection*{13.18.1 Reading-in and reading-out archives via a serial interface}

\section*{Availability of the V24 serial interface}

You have the option of reading-out and reading-in archives in the "Program manager" operating area as well as in the "Startup" operating area via the V24 serial interface.
- SINUMERIK Operate on the NCU:

The softkeys for the V24 interface are available as soon as an option module is connected and the slot is occupied.
- SINUMERIK Operate on the PCU:

The softkeys for the V24 interface are always available.

\section*{Reading-out archives}

The files to be sent (directories or individual files) are zipped in an archive (*.arc). If you send an archive (*.arc), this is sent directly without being additionally zipped. If you have selected an archive (*.arc) together with an additional file (e.g. directory), then these are zipped into a new archive and are then sent.

\section*{Reading-in archives}

Only archives can be read-in via the V24 interface. These are transferred and then subsequently unzipped.

\section*{Note}

\section*{Start-up archive}

If you read in a start-up archive via the V24 interface, then this is immediately activated.

\section*{Externally processing the punched tape format}

If you wish to externally process an archive, then generate this in the punched tape format.

\section*{Procedure}

1. Select the "Program manager" operating area, and press the "NC" or "Local drive" softkey.
- OR -

Select the "Startup" operating area and press the "System data" softkey.
2. Select the directories or the files that you wish to send via V24.

3. Press the ">>" and "Archive" softkeys.

\section*{Archive}
4. Press the "V24 send" softkey.
- OR -

Reading in an archive
Receive
RS-232-C

Press the "V24 receive" softkey if you wish to read-in files via V24.

\subsection*{13.18.2 Setting V24 in the program manager}
\begin{tabular}{|c|c|}
\hline V24 setting & Meaning \\
\hline Protocol & \begin{tabular}{l}
The following protocols are supported for transfer via the V24 interface: \\
- RTS/CTS (default setting) \\
- Xon/Xoff
\end{tabular} \\
\hline Transfer & \begin{tabular}{l}
It is also possible to use a secure protocol for data transfer (ZMODEM protocol). \\
- Normal (default setting) \\
- secure \\
For the selected interface, secure data transfer is set in conjunction with handshake RTS/CTS.
\end{tabular} \\
\hline Baud rate & \begin{tabular}{l}
Transfer rate: Transfer rates of up to 115 kbaud can be selected. The baud rate that can be used depends on the connected device, the cable length and the general electrical conditions. \\
- 110 \\
- .... \\
- 19200 (default) \\
- ... \\
- 115200
\end{tabular} \\
\hline Archive format & \begin{tabular}{l}
- Punched tape format (default setting) \\
- Binary format (PC format)
\end{tabular} \\
\hline V24 settings (d & \\
\hline Interface & - COM1 \\
\hline Parity & \begin{tabular}{l}
Parity bits are used for error detection: The parity bits are added to the coded characters to make the number of positions set to "1" an uneven number (uneven parity) or to an even number (even parity). \\
- None (default setting) \\
- Odd \\
- Even
\end{tabular} \\
\hline Stop bits & \begin{tabular}{l}
Number of stop bits for asynchronous data transfer. \\
- 1 (default) \\
- 2
\end{tabular} \\
\hline Data bits & \begin{tabular}{l}
Number of data bits for asynchronous data transfer. \\
- 5 bits \\
- ... \\
- - 8 bits (default setting)
\end{tabular} \\
\hline XON (hex) & Only for punched tape format \\
\hline XOFF (hex) & Only for punched tape format \\
\hline
\end{tabular}
\begin{tabular}{|l|l|}
\hline V24 setting & Meaning \\
\hline End of data transfer (hex) & \begin{tabular}{l} 
Only for punched tape format \\
Stop with end of data transfer character \\
The default setting for the end of data transfer character is (HEX) 1A
\end{tabular} \\
\hline Time monitoring (sec) & \begin{tabular}{l} 
Time monitoring \\
For data transfer problems or at the end of data transfer (without end \\
of data transfer character) data transfer is interrupted after the speci- \\
fied number of seconds. \\
The time monitoring is controlled by a time generator (clock) that is \\
started with the first character and is reset with each transferred char- \\
acter. \\
The time monitoring can be set (seconds).
\end{tabular} \\
\hline
\end{tabular}

\section*{Procedure}

2. Press the "NC" or "Local drive" softkey.

3. Press the ">>" and "Archive" softkeys.

Archive

RS-232-C settings

\section*{Details}
1. Select the "Program Manager" operating area.
div
4. Select the "V24 settings" softkey. The "Interface: V24" window is opened.
5. The interface settings are displayed.
6. Press the "Details" softkey if you wish to view and process additional settings for the interface.

\section*{Alarm, error and system messages}

\subsection*{14.1 Displaying alarms}

If faulty conditions are recognized in the operation of the machine, then an alarm will be generated and, if necessary, the machining will be interrupted.
The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

\section*{\(\triangle\) caution}

Dangers for persons and machines
Please check the situation in the plant on the basis of the description of the active alarm(s). Eliminate the cause/s of the alarm/s and acknowledge it/them as instructed.

Failure to observe this warning will place your machine, workpiece, stored settings and possibly even your own safety at risk.

\section*{Alarm overview}

You can display all upcoming alarms and acknowledge them.
The alarm overview contains the following information:
- Date and time
- Cancel criterion
specifies the key or softkey used to acknowledge the alarm
- Alarm number
- Alarm text

\section*{Procedure}



图
...国
1. Select the "Diagnostics" operating area.
2. Press the "Alarm list" softkey.

The "Alarms" window appears.
All pending alarms are displayed.
The "Hide SI alarms" softkey is displayed if safety alarms are pending.
3. Press the "Hide SI alarms" softkey if you do not wish to display SI alarms.
4. Position the cursor on an alarm.
5. Press the key that is specified as acknowledgement symbol to delete the alarm.
- OR -

Press the "Delete HMI alarm" softkey to cancel an HMI alarm.
- OR -

Press the "Acknowledge alarm" softkey to delete a PLC alarm of the SQ type (alarm number as of 800000).
The softkeys are activated when the cursor is on the corresponding alarm.

\section*{Acknowledgement symbols}
\begin{tabular}{|c|c|}
\hline Symbol & Meaning \\
\hline I & Turn the unit off and back on (main switch), or press NCK POWER ON. \\
\hline / & Press the <RESET> key. \\
\hline \[
8
\] & \begin{tabular}{l}
Press the <ALARM CANCEL> key. \\
- OR - \\
Press the "Acknowl. HMI alarm" softkey.
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|l|l|}
\hline Symbol & Meaning \\
\hline PLC & Press the key provided by the manufacturer. \\
\hline
\end{tabular}

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{14.2 Displaying an alarm log}

A list of all the alarms and messages that have occurred so far are listed in the "Alarm Log" window.

Up to 500 administered, incoming and outgoing events are displayed in chronological order.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}


\section*{Display \\ new}

Store
\(\log\)
1. Select the "Diagnostics" operating area.
2. Press the "Alarm log" softkey.

The "Alarm Log" window opens.
All of the coming and going events - that have occurred since the HMI was started - are listed.
3. Press the "Display new" softkey to update the list of displayed alarms/messages.
4. Press the "Save Log" softkey.

The log that is currently displayed is stored as text file alarmlog.txt in the system data in directory card/user/sinumerik/hmi/log/alarm_log.

\subsection*{14.3 Displaying messages}

PLC and part program messages may be issued during machining.
These message will not interrupt the program execution. Messages provide information with regard to a certain behavior of the cycles and with regard to the progress of machining and are usually kept beyond a machining step or until the end of the cycle.

\section*{Overview of messages}

You can display all issued messages.
The message overview contains the following information:
- Date
- Message number is only displayed for PLC messages
- Message text

\section*{Procedure}
1. Select the "Diagnostics" operating area.
2. Press the "Messages" softkey.

The "Messages" window appears.

\subsection*{14.4 Sorting, alarms, faults and messages}

If a large number of alarms, messages or alarm logs are displayed, you have the option of sorting these in an ascending or descending order according to the following criteria:
- Date (alarm list, messages, alarm log)
- Number (alarm list, messages)

As a consequence, for every extensive lists, you can obtain the required information faster.

\section*{Procedure}


\section*{Alarm} protoc.

\section*{Date}

Decreasing

Number

Ascending
3. Press the "Sort" softkey.

The list of entries is sorted in ascending order according to date, i.e. the
1. Select the "Diagnostics" operating area.
2. Press the softkey "Alarm list", "Messages" or "Alarm log" to display the requested messages and alarms. most recent information is at the end of the list.
4. Press the softkey "Descending" to sort the list in a descending order. The most recent event is shown at the beginning of the list.
5. Press the "Number" softkey if you wish to sort the alarm list or the list with messages according to numbers.
6. Press the "Ascending" softkey if you wish to display the list in an ascending order.

\subsection*{14.5 Creating screenshots}

You can create screenshots of the current user interface.
Each screenshot is saved as a file and stored in the following folder:
/user/sinumerik/hmi/log/screenshot

\section*{Procedure}

Ctrl + P Press the <Ctrl+P> key combination.
A screenshot of the current user interface is created in .png format.
The file names assigned by the system are in ascending order from "SCR_SAVE_0001.png" to "SCR_SAVE_9999.png". You can create up to 9,999 screenshots.

\section*{Copy file}


System data

\section*{Copy}

\section*{Cut}

\section*{Paste}
2. Press the "System data" softkey.
3. Open the folder specified above, and select the required screenshots.
1. Select the "Startup" operating area.
4. Press the "Copy" softkey.
- OR -

Press the "Cut" softkey.
5. Open the required archive directory, e.g. on a USB flash drive and press the "Paste" softkey.

\section*{Note}

You can also copy the screenshots using "WinSCP" to a Windows PC (for 840D sl).

\section*{Note}

If you wish to view the screenshots, then you can open the files in SINUMERIK Operate. On a Windows PC, you can open the data using a graphic program, e.g. "Office Picture Manager".

\subsection*{14.6 PLC and NC variables}

\subsection*{14.6.1 Displaying and editing PLC and NC variables}

The "NC/PLC Variables" window allows NC system variables and PLC variables to be monitored and changed.

You receive the following list in which you can enter the desired NC/PLC variables in order to display the actual values.
- Variable

Address for NC/PLC variable.
Incorrect variables have a red background and are displayed with a \# character in the value column.
- Comment

Any comment on the variable.
The columns can be displayed and hidden.
- Format

Specify the format in which the variable is to be displayed.
The format can be specified (e.g. floating point).
- Value

Displays the actual value of the NC/PLC variables.
\begin{tabular}{|l|l|}
\hline PLC variables & \\
\hline Inputs & Input bit (Ex), input byte (EBx), input word (EWx), input double word (EDx) \\
\hline Outputs & \begin{tabular}{l} 
Output bit (Ax), output byte (ABx), output word (AWx), output double word \\
(ADx)
\end{tabular} \\
\hline Bit memory & \begin{tabular}{l} 
Memory bit (Mx), memory byte (MBx), memory word (MWx), memory dou- \\
ble word (MDx)
\end{tabular} \\
\hline Times & Time (Tx) \\
\hline Counters & Counter (Cx) \\
\hline Data & \begin{tabular}{l} 
Data block (DBx): Data bit (DBXx), data byte (DBBx), data word (DBWx), \\
data double word (DBDx)
\end{tabular} \\
\hline
\end{tabular}
\begin{tabular}{|l|l|}
\hline Formats & \\
\hline B & Binary \\
\hline H & Hexadecimal \\
\hline D & Decimal without sign \\
\hline +/-D & Decimal with sign \\
\hline F & Floating point (for double words) \\
\hline A & ASCII character \\
\hline
\end{tabular}

\section*{Notation for variables}
- PLC variables

EB2
A1.2
DB2.DBW2
- NC variables
- NC system variables - notation \$AA_IM[1]
- User variables/GUDs - notation

GUD/MyVariable[1,3]
- OPI - notation
/CHANNEL/PARAMETER/R[u1,2]

\section*{Note}

\section*{NC system variables and PLC variables}
- System variables can be dependent on the channel. When the channel is switched over, the values from the corresponding channel are displayed.
- For user variables (GUDs) it is not necessary to make a specification according to global or channel-specific GUDs. The indices of GUD arrays are, just like NC variables in the system variable syntax, 0 -based; this means that the first element starts with the index 0 .
- Using the tooltip, for NC system variables, you can display the OPI notation (with the exception of GUDs).

\section*{Changing PLC variables}

Changes can only be made to the PLC variables with the appropriate password.
\begin{tabular}{|l|}
\hline Incorrect parameterization \\
Changes in the states of NC/PLC variables have a major impact on the machine. Incorrect \\
configuration of the parameters can endanger human life and cause damage to the \\
machine.
\end{tabular}

\section*{Changing and deleting values}


Details


Display comments

\section*{Display comments}

Change

Insert variable

Filter/ search

\section*{Delete} all

11. Press the "OK" softkey to confirm the changes or the deletion.
- OR -

Press the "Cancel" softkey to cancel the changes.

\section*{Note}

\section*{"Filter/Search" when inserting variables}

The start value for "Filter/Search" of variables differs.
For example, to insert the variable \$R[0], set "Filter/Search":
- The start value is 0 , if you filter according to "System variables".
- The start value is 1 , if you filter according to "All (no filter)". In this case, all signals are displayed and shown in the OPI notation.

\section*{Changing operands}

Depending on the type of operand, you can increment or decrement the address by 1 place at a time using the "Operand + " and "Operand -" softkeys.

\section*{Note}

\section*{Axis names as index}

For axis names, the softkeys "Operand +" and "Operand -" do not act as index, e.g. for \$AA_IM[X1].
\begin{tabular}{|c|c|}
\hline & Examples \\
\hline \multirow{4}{*}{Operand +} & DB97.DBX2.5 \\
\hline & Result: DB97.DBX2.6 \\
\hline & \$AA_IM[1] \\
\hline & Result: \$AA_IM[2] \\
\hline \multirow{4}{*}{Operand -} & MB201 \\
\hline & Result: MB200 \\
\hline & /Channel/Parameter/R[u1,3] \\
\hline & Result: /Channel/Parameter/R[u1,2] \\
\hline
\end{tabular}

\subsection*{14.6.2 Saving and loading screen forms}

You have the option of saving the configurations of the variables made in the "NC/PLC variables" window in a screen form that you reload again when required.

\section*{Editing screen forms}

If you change a screen form that has been loaded, then this is marked using with * after the screen form name.

The name of a screen form is kept in the display after switching-off.

\section*{Procedure}
1. You have entered values for the desired variables in the "NC/PLC variables" window.


Save screen


Load mask
2. Press the ">>" softkey.
3. Press the "Save screen" softkey.

The "Save screen: Select archiving" window opens.
4. Position the cursor on the template folder for variable screen forms in which your actual screen form should be saved and press the "OK" softkey.
The "Save screen: Name" window opens.
5. Enter the name for the file and press the "OK" softkey.

A message in the status line informs you that the screen form was saved in the specified folder.
If a file with the same name already exists, they you will receive a prompt.
6. Press the "Load screen" softkey.

The "Load screen" window opens and displays the sample folder for the variable screen forms.
7. Select the desired file and press the "OK" softkey.

You return to the variable view. The list of all of the predefined NC and PLC variables is displayed.

\subsection*{14.7 Version}

\subsection*{14.7.1 Displaying version data}

The following components with the associated version data are specified in the "Version data" window:
- System software
- Basic PLC program
- PLC user program
- System extensions
- OEM applications
- Hardware

Information is provided in the "Nominal version" column as to whether the versions of the components deviate from the version supplied on the CompactFlash Card.


The version displayed in the "Actual version" column matches the version of the CF card.
The version displayed in the "Actual version" column does not match the version of the CF card.

You may save the version data. Version displays saved as text files can be further processed as required or sent to the hotline in the event of an error.

\section*{Procedure}


Details
1. Select the "Diagnosis" operating area.
3. Select the component for which you would like more information.
2. Press the "Version" softkey.

The "Version Data" window appears.
Data from the available components are displayed.
4. Press the "Details" softkey, in order to receive more exact information on the components displayed.

\subsection*{14.7.2 Save information}

All the machine-specific information of the control is combined in a configuration via the user interface. You can save machine-specific information on the drives that have been set-up.

\section*{Procedure}


\section*{New} directory

```

OK
OK

```
1. Select the "Diagnostics" operating area.
2. Press the "Version" softkey.

It takes some time to call the version display. While the version data is being determined a progress message box and the appropriate text are displayed in the dialog line.
3. Press the "Save" softkey.

The "Save Version Information: Select Archive" window opens. The following storage locations are offered depending on the configuration:
- Local drive
- Network drives
- USB
- Version data (archive: Data tree in the "HMI data" directory)
4. Then press the "New directory" softkey if you wish to create your own directory.
5. Press the "OK" softkey. The directory is created.
6. Press the "OK" softkey again to confirm the storage location.

The "Save Version Information: Name" window opens. The following options are available:
- In the "Name:" text field, the file name is pre-assigned with <Machine name/no.>+<CF-card number>. "_config.xml" or "_version.txt" is automatically attached to the file names.
- In text field "Comment", you can add a comment that is stored with the configuration data.
Select the following data via a checkbox:
- Version data (.TXT): Output of pure version data in text format.
- Configuration data (.XML): Output of configuration data in XML format. The configuration file contains the data you entered under Machine identity, the license requirements, the version information and the logbook entries.
7. Press the "OK" softkey to start the data transfer.

\subsection*{14.8 Logbook}

The logbook provides you with the machine history in an electronic form.
If service is carried out on the machine, this can be electronically saved. This means that it is possible to obtain a picture about the "History" of the control and to optimize service.

\section*{Editing the logbook}

You can edit the following information:
- Editing information on the machine identity
- Machine name/No.
- Machine type
- Address data
- Make logbook entries (e.g. "filter replaced")
- Deleting logbook entries

\section*{Note}

\section*{Deleting logbook entries}

Up to the 2nd commissioning, you have the option to delete all of the entered data up to the time of the first commissioning.

\section*{Output of the logbook}

You have the possibility of exporting the logbook by generating a file using the "Save version" function in which the logbook is contained as section.

\section*{See also}

Save information (Page 797)

\subsection*{14.8.1 Displaying and editing the logbook}

\section*{Procedure}


> Logbook
1. Select the "Diagnostics" operating area.
2. Press the "Version" softkey.
3. Press the "Logbook" softkey.

The "Machine logbook" window opens.

\section*{Editing end customer data}

\section*{Change}

Clear

You have the option of changing the address data of the end customer using the "Change" softkey.
- OR -

Using the "Clean up" softkey, you can delete all logbook entries.

All entries, except the date of the first commissioning, are deleted and the softkey "Clean up" is deactivated.

\section*{Note}

\section*{Deleting logbook entries}

As soon as the 2nd commissioning has been completed, the "Clean up" softkey to delete the logbook data is no longer available.

\subsection*{14.8.2 Making a logbook entry}

Using the "New logbook entry" window to make a new entry into the logbook.
Enter your name, company and department and a brief description of the measure taken or a description of the fault.

\section*{Note}

If you wish to make line breaks in the "fault diagnostics/measure" field, use the key combination <ALT> + <INPUT>.

The date and entry number are automatically added.

\section*{Sorting the entries}

The logbook entries are displayed numbered in the "machine logbook" window.
More recent entries are always added at the top in the display.

\section*{Procedure}
1. The logbook is opened.

New entry

OK
2. Press the "New entry" softkey.

The "New logbook entry" window opens.
3. Enter the required data and press the "OK" softkey.

You return to the "Machine logbook" window and the entry is displayed below the machine identity data.

\section*{Note}

\section*{Deleting logbook entries}

Up to the end of the 2nd commissioning, you have the option to delete the logbook entries up to the time of the first commissioning using the "Clean up" softkey.

\section*{Searching for a logbook entry}

You have the option for searching for specific entries using the search function.
1. The "Machine logbook" window is opened.

Search

Continue search
2. Press the "Search..." softkey and enter the desired term in the search form. You can make a search according to date/time, company name/department or according to fault diagnostics/measure.
The cursor is positioned on the first entry that corresponds to the search term.
3. Press the "Continue search" softkey if the entry found is not the one that you are looking for.

\section*{Additional search option}

\section*{Go to start}

Go to end

Press the "Go to Beginning" softkey to start the search at the latest entry.

Press the "Go to End" softkey to start the search at the oldest entry.

\subsection*{14.9 Remote diagnostics}

\subsection*{14.9.1 Setting remote access}

You can influence the remote access to your control in the "Remote diagnostics (RCS)" window.

Here, rights for all types of remote control are set. The selected rights are defined from the PLC and using the setting at the HMI.

The HMI can restrict the rights specified from the PLC, but however, cannot extend the rights beyond the PLC rights.

If the settings made permit access from outside, then this is still dependent on a manual or automatic confirmation.

\section*{Rights for remote access}

The "Specified from PLC" field shows the access rights for remote access or remote monitoring specified from the PLC.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

In the "Selected in the HMI" selection box, you have the possibility of setting rights for remote control:
- Do not permit remote access
- Permit remote monitoring
- Permit remote control

The combination of the settings in the HMI and in the PLC show the valid status as to whether access is permitted or not. This is displayed in the "Resulting from" line.

\section*{Settings for the confirmation dialog box}

If the settings made for "Specified from the PLC" and "Selected in the HMI" permit access from outside, then this is however, still dependent on either a manual or automatic confirmation.

As soon as a remote access is permitted, at all of the active operating stations, a query dialog box is displayed for the operator at the active operating station to either confirm or reject an access.

For the case that there is no local operation, then the control behavior can be set for this particular scenario. You define how long this window is displayed and whether, after the confirmation has expired, the remote access is automatically rejected or accepted.

\section*{Display of the state}


If remote access is active, using these icons you will be informed in the status line as to whether a remote access is presently active or whether only monitoring is permitted.

\section*{Procedure}


\section*{Change}

1. Select the "Diagnostics" operating area.
2. Press the "Remote diag." softkey.

The "Remote diagnostics (RCS)" window is opened.
3. Press the "Change" softkey.

The "Selected in the HMI" is activated.
4. If you desire remote control, select the entry "Permit remote control".

In order that remote control is possible, the entry "Permit remote control" must be specified in the fields "Specified from the PLC" and "Selected in the HMI".
5. Enter new values in the group "Behavior for confirming remote access" if you wish to change the behavior for confirming remote access.
6. Press the "OK" softkey.

The settings are accepted and saved.

\section*{References}

For a description of configuration options, refer to the Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sI

\subsection*{14.9.2 Permit modem}

You can permit remote access to your control via a teleservice adapter IE connected at X127.


Machine manufacturer
Please refer to the machine manufacturer's specifications.


\section*{Software option}

You need the "MC Information System RCS Host" option to display the "Permit modem" softkey.

Procedure


\section*{Hllow} rem. access

Allow
modem
1. The "Remote diagnostics (RCS)" window is opened.
2. Press the "Permit modem" softkey.

Access to the control via modem is enabled so that a connection can be established.
3. To block access again, press the "Permit modem" softkey again.

\subsection*{14.9.3 Request remote diagnostics}

Using the "Request remote diagnostics" softkey, from your control you have the option of actively requesting remote diagnostics with your machinery construction OEM.

The access via modem must be enabled if the access is to be made via modem.

Machine manufacturer
Please refer to the machine manufacturer's specifications.

When requesting remote diagnostics, you obtain a window with the corresponding preassigned data and values of the ping service. If required, you can ask your machine manufacturer for this data.
\begin{tabular}{|l|l|}
\hline Data & Meaning \\
\hline IP address & IP address of the remote PC \\
\hline Port & Standard port that is intended for remote diagnostics \\
\hline Send duration & Duration of the request in minutes \\
\hline Send interval & Cycle in which the message is sent to the remote PC in seconds \\
\hline \begin{tabular}{l} 
Ping send \\
data
\end{tabular} & Message for the remote PC \\
\hline
\end{tabular}

Turning

\section*{Procedure}


\section*{References}

Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\subsection*{14.9.4 Exit remote diagnostics}

\section*{Procedure}
1. The "Remote diagnostics (RCS)" is opened and it is possible that remote monitoring or remote access is active.
2. Block the modem access if access via modem is to be blocked.
- OR -

In the "Remote Diagnostics (RCS)" window, reset the access rights to "Permit no remote access".

\section*{Working with Manual Machine}

\subsection*{15.1 Manual Machine}
"Manual Machine" offers a comprehensive spectrum of functions for manual mode.
You can carry out all the important machining processes without writing a program.


Software options
You require the "ShopMill/ShopTurn" option for working with "Manual Machine"

\section*{Machining}

Machining before the center of rotation is typical.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Main screen}

After power-up of the control, the basic screen appears for "Manual Machine".


Turning

\section*{Machining options}

You have the following options for machining the workpieces:
- Manual mode
- Single-cycle machining

\subsection*{15.2 Measuring the tool}

All the options of the manual and automatic measurement are available to determine the tool offset data (see also Section "Measuring the tool (Page 78)").

\section*{Procedure}
1. "Manual Machine" is active.
2. Press the "Meas. tool" softkey.
3. Select the the required measuring function via the vertical softkey bar and press the appropriate softkey.

\subsection*{15.3 Setting the zero offset}

Directly select the work offset in the "Parameter" operating area in the work offset list.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
1. "Manual Machine" is active.
2. Select the "Parameter" operating area.
2. Press the "Work offset" and "G54...599" softkeys.

The "Work Offset G54...599" window opens.
3. Position the cursor on the desired work offset and press the softkey "WO selection"

Now return to the basic screen and enter the selected work offset in the "Work offset" field.

\section*{See also}

Setting the zero offset (Page 76)

\subsection*{15.4 Set limit stop}

You can limit the traversing range of the axes.
To do this, enter the values for the respective axes. The values refer to the workpiece coordinate system. The limits can be switched on and off individually.
Activated, i.e. active set limits, are indicated by a bar next to the wind rose in the direction graphic.
When a limit is reached, an alarm appears which disappears again when the axis is moved away from the limit.

\section*{Note}

Entered and activated limit stops remain active after switching from JOG mode to MDA or AUTO mode.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
1. "Manual Machine" is active
2. Press the "Limit stops" softkey.

The "Limit Stops" window opens.
3. Enter the desired position of the limit stop for each axis.
- OR -
4. In the field next to the position specification select the entry "On" to activate the desired limit stop.
The bar is displayed next to the wind rose.
5. Press the "Back" softkey to return to the basic screen.

The active limit stops are also displayed here with bars.

\section*{Turning}

\subsection*{15.5 Simple workpiece machining}

In "Manual Machine", you machine workpieces directly without creating a program.

\section*{Functions}

The following functions are available to you for machining in manual mode:
- Axis movements
- Taper turning
- Straight (face and longitudinal turning) and circle

\section*{Note}

Tool, spindle speed and direction of spindle rotation are activated with <CYCLE START>. A change in feedrate immediately becomes active.

\subsection*{15.5.1 Traversing axes}

For preparatory actions and simple traversing movements, input the parameters directly into the "Manual Machine" input fields of the basic screen.

\section*{Tool selection}
\begin{tabular}{|c|c|c|}
\hline & 1. & "Manual Machine" is active. \\
\hline \multicolumn{3}{|l|}{Tool selection} \\
\hline \multirow[t]{2}{*}{\[
\begin{gathered}
\text { Select } \\
\text { tool }
\end{gathered}
\]} & 2. & Select the desired tool in "T". \\
\hline & 3
4 & Enter the feedrate ( F ) and the spindle speed (S). \\
\hline \multirow[t]{2}{*}{\[
\bigcup_{\text {select }}
\]} & \multirow[t]{2}{*}{4.} & Select the direction of spindle rotation (e.g. clockwise direction of rotation): \\
\hline & & 2 \\
\hline & & - OR - \\
\hline & & Set the direction of rotation via the machine control panel. \\
\hline & 5. & Press the <CYCLE START> key. \\
\hline  & & \\
\hline
\end{tabular}

The spindle starts immediately after the tool is selected.
Note:
Please refer to the machine manufacturer's specifications.

\section*{Machining}
x
6. Select the axis to be traversed on the machine control panel.
7. Press the <+> or <-> key on the machine control panel.

- OR -

Select the direction with the aid of the cross-switching lever.
The axes are moved at the set machining feedrate.
Note:
Please refer to the machine manufacturer's specifications.
The active direction is graphically displayed in the basic screen by means of the wind rose.

\subsection*{15.5.2 Taper turning}

The basic effective direction can be selected via the axis direction keys or via the crossswitching lever. In addition, a taper angle ( \(\alpha 1\) ) can also be entered.

\section*{Procedure}
1. "Manual Machine" is active.
```

Taper

```
turning
2. Press the "Taper turning" softkey.
3. Select the tool, spindle, and spindle direction and specify the machining feedrate.
4. Enter the desired angle a1.

\section*{Note}

Selecting/deselecting the taper turning and changing the angle \(\alpha 1\) is only possible in the reset state.
15.5 Simple workpiece machining
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline T & Tool name & \\
\hline D & Cutting edge number & \\
\hline ST & & \\
\hline & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline \[
\begin{aligned}
& \mathrm{s} / \mathrm{V} \\
& \mathrm{u}
\end{aligned}
\] & Spindle speed or constant cutting rate & \(\mathrm{rev} / \mathrm{min}\) \(\mathrm{m} / \mathrm{min}\) \\
\hline Spindle M function U & Spindle off: Spindle is stopped & \\
\hline & CCW rotation: Spindle rotates counterclockwise & \\
\hline & \begin{tabular}{l}
2 \\
CW rotation: Spindle rotates clockwise
\end{tabular} & \\
\hline \begin{tabular}{l}
Gear stage \\
U
\end{tabular} & Specification of the gear stage (auto, I-V) & \\
\hline a1 & Rotation of the coordinate system. & Degrees \\
\hline Other M function & \begin{tabular}{l}
Input of machine functions \\
Refer to the machine manufacturer's table for the correlation between the meaning and number of the function.
\end{tabular} & \\
\hline Machining plane U & Selection of the machining plane (G17(XY), G18 (ZX), G19 (YZ)) & \\
\hline
\end{tabular}

\subsection*{15.5.3 Straight and circular machining}

\subsection*{15.5.3.1 Straight turning}

Use this function for simple, straight machining (e.g. face or longitudinal turning).

\section*{Procedure}
1. "Manual Machine" is active.
2. Press the "Straight circle" softkey.

Straight all axes
- OR -
Straight \(\mathrm{X} \alpha\)

\section*{Straight \(2 \alpha\)}

Rapid
- OR -
- OR -
4. Specify the desired value for the feedrate \(F\).
- OR -

Press the "Rapid traverse" softkey.
The rapid traverse is displayed in field " \(F\) ".
5. Enter the target position and, if required, the angle ( \(\alpha\) ) for the axis or axes to be traversed.

Using the "Graphic view" softkey, you can toggle between the help screen and the graphic view in the screen.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \[
\begin{aligned}
& F \\
& O
\end{aligned}
\] & Feedrate & \begin{tabular}{l}
\(\mathrm{mm} / \mathrm{min}\) \\
\(\mathrm{mm} / \mathrm{rev}\)
\end{tabular} \\
\hline & All axes & \\
\hline \[
\begin{aligned}
& X U \\
& Z U \\
& Y O \\
& C U \\
& Z 2 U
\end{aligned}
\] & \begin{tabular}{l}
Target position in the \(X\) direction (abs or inc) \\
Target position in the \(Z\) direction (abs or inc) \\
Target position in the Y direction (abs or inc) \\
Target position of \(C\) axis of main spindle (abs or inc) \\
Target position of an added axis, if it exists (abs or inc)
\end{tabular} & \begin{tabular}{l}
mm \\
mm \\
mm \\
mm \\
mm
\end{tabular} \\
\hline & Straight X 人 & \\
\hline \[
\begin{aligned}
& x \\
& \alpha
\end{aligned}
\] & Target position in the \(X\) direction (abs or inc) Angle of the straight line to the X axis & \begin{tabular}{l}
mm \\
Degrees
\end{tabular} \\
\hline & Straight Z \(\alpha\) & \\
\hline Z
a & Target position in the \(Z\) direction (abs or inc) Angle of the straight line to the \(Z\) axis & \begin{tabular}{l}
mm \\
Degrees
\end{tabular} \\
\hline
\end{tabular}

\subsection*{15.5.3.2 Circular turning}

You can use this function for a simple circular machining.

\section*{Procedure}
1. "Manual Machine" is active.

2. Press the "Straight circle" softkey.
3. Press the "Circle" softkey.
4. Specify the desired value for the feedrate \(F\).
5. Select the desired circle input (e.g. "End point + radius") and the direction of rotation.
6. Enter the target position as well as the circle center point or radius.

Using the "Graphic view" softkey, you can toggle between the help screen and the graphic view in the screen.

\section*{Parameters}
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline \begin{tabular}{l}
F \\
U
\end{tabular} & Feedrate & \(\mathrm{mm} / \mathrm{min}\) \(\mathrm{mm} / \mathrm{rev}\) \\
\hline Circle input
\[
0
\] & \begin{tabular}{l}
- End point + center point \\
- End point + radius
\end{tabular} & \\
\hline Direction of rotation & \begin{tabular}{l}
' 2 Clockwise direction of rotation \\
\(S\) \\
Counter-clockwise direction of rotation
\end{tabular} & \\
\hline \[
\begin{array}{|l|}
\hline X \\
\hline
\end{array}
\] & Target position in the X direction (abs and inc) & mm \\
\hline \[
\begin{aligned}
& Z \\
& U \\
& \hline
\end{aligned}
\] & Target position in the Z direction (abs and inc) & mm \\
\hline I & Circle center point I (inc) - only if circle input via end point and center point & mm \\
\hline K & \begin{tabular}{l}
Circle center point K (inc) - only if circle input via end point and center point Note: \\
Incremental dimensions: The sign is also evaluated.
\end{tabular} & mm \\
\hline R & Radius - only if circle input via end point and radius & mm \\
\hline
\end{tabular}

\subsection*{15.6 More complex machining}

The following functions are available to you for comprehensive machining in manual mode:
- Drilling (center drilling, centering, drilling, reaming, deep-drilling, threads, positions)
- Turning (stock removal, groove, undercut, threads, tapping)
- Milling (face milling, pocket, spigot, multiple edge, groove, thread milling, engraving)
- Contour turning (contour, stock removal, grooving, plunge turning)

\section*{Note}

Deselecting technological functions
Press the "Back" softkey to return to the main screen from the list of technological functions.

\section*{General sequence}

For more complex machining processes, proceed in the following order:
- Select the desired function via the corresponding softkey.
- Select the tool and enter the desired values in the parameter screen.
- Press the "Accept" softkey to save the values.

The input screen form closes.
A line with the specified parameters is displayed on the basic screen.
- Press the <CYCLE START> key.

The selected cycle is started.
- OR -
- Press the "Cancel" softkey to return to the basic screen.

\section*{Note}

Note
You can return to the parameter screen form at any time to check and correct the inputs.
Press the <"Cursor right> key to jump back to the input screen form.

\section*{Drilling a position pattern}

You can drill a position pattern:
- First select the desired function (e.g. "Centering") via the softkey in "Drilling".
- Select the appropriate tool, enter the desired values in the parameter screen and press the "Accept" softkey to confirm the technology block.

The input screen is closed and the line with the technology data is displayed in the main screen.
- Press the "Positions" softkey and select the desired position pattern (e.g. any positions) via softkey, enter the desired values in the parameter screen form press the "Accept" softkey.
The input screen is closed and the technology and positioning blocks are displayed in brackets.

\section*{Note}

You only have the option of creating a machining operation and a position pattern one after the other.

\section*{Note}

If you enter the positions first, only the drilling cycles for driven tools are offered after creating the positioning block.

\section*{Approach and retraction}

When machining the workpiece, you traverse from the current position to the machining start point. After the machining process, the tool is returned along a direct path to the starting point.

\subsection*{15.6.1 Drilling with Manual Machine}

\section*{Functions (cycles)}

The same range of technological functions (cycles) is available as in automatic mode for drilling on the face or peripheral surface of a workpiece:


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.
\begin{tabular}{|c|c|c|c|c|}
\hline - Drill. & \(\Rightarrow\) & Drilling centric & \(\Rightarrow\) & Drilling centric \\
\hline & & & & Thread centric \\
\hline & & Centering & & \\
\hline & & Drilling Reaming & \(\Rightarrow\) & Drilling \\
\hline & & & & Reaming \\
\hline & & & & Boring \\
\hline & & Deep hole drilling & \(\Rightarrow\) & Deep hole drilling 1 \\
\hline & & & & Deep hole drilling 2 \\
\hline & & Thread & \(\Rightarrow\) & Tapping \\
\hline & & & & Drl+thrd milling \\
\hline & & Positions & \(\Rightarrow\) & \\
\hline & & & & \(\cdots \infty\) \\
\hline
\end{tabular}

\section*{\#}

\section*{:}


\section*{Parameter}

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Drilling (Page 293)").

\subsection*{15.6.2 Turning with manual machine}

\section*{Functions (cycles)}

The same range of technological functions (cycles) is available for turning as in the automatic mode:


\section*{Parameter}

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Rotate (Page 355)").

\section*{Thread cutting}

In addition to the functions that are made available by "thread-cutting" under Automatic, you can insert idle cuts during the machining process under "Manual Machine."
You can interrupt the infeed of the cutting depth during the machining process by inserting idle cuts, in order to smooth the flanks for example.

> Empty cut \(\quad\) You can insert idle cuts using the "Idle cut" softkey.

\section*{Thread re-machining}

You can re-machine existing threads, when repairing previously cut threads for example or as the result of changes that arise from re-measuring (see Section "Thread synchronizing (Page 120)").
If you want to rework some threads, input the initial plunge depth D0 (inc.). This is the depth that was reached during a previous machining.

\section*{Note}

By inputting the plunge depth, you avoid unnecessary idle cuts when reworking the threads.

\section*{B axis}

\subsection*{15.6.3 Contour turning with Manual machine}

For contour turning of simple geometric shapes, the same range of technological functions (cycles) as in automatic mode is available.

Machine manufacturer
Please refer to the machine manufacturer's specifications.


\section*{Parameters}

The parameters of the input screen forms correspond to the parameters under Automatic (see Section Contour turning (Page 390)).

\section*{Last contour}

> Last If you have created a contour since the startup of the machine, the "Last contour" softkey is available. You then have the option to reopen the input screen with the last contour you entered in "Manual machine" for editing.

\subsection*{15.6.4 Milling with Manual Machine}

The same range of technological functions (cycles) is available as in automatic mode for the milling of simple geometric shapes.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.


\section*{Thread milling}

\section*{Engrauing}

\section*{Parameters}

The parameters of the input screen forms correspond to the parameters under Automatic (see Section "Milling (Page 431)").

\subsection*{15.7 Simulation and simultaneous recording}

For more complex machining processes, you can check the result of your inputs with the aid of the simulation, without having to traverse the axes (see Section "Simulating machining (Page 195)"). The execution of the work steps is graphically displayed on the screen during this period.

\section*{Software option}


You require the option "Simultaneous recording ShopTurn (real-time simulation)" for the simultaneous recording of the work steps.

\section*{Note}

In "Manual Machine", you can simulate a work step with an already opened and filled out parameter screen form.

\section*{Setting up a blank shape}

For the graphical display, a pre-defined blank shape is used. Just the same as under simultaneous recording, in automatic and simulation you can change the blank as required (see Section "Blank display (Page 206)").

\section*{Working with a B axis (only 840D sl)}

\subsection*{16.1 Lathes with \(B\) axis}

With an additional \(B\) axis, you have the option of aligning milling machines and lathes.


The initial setting in which all tools must be measured is \(\mathrm{B}=0\).
When turning, you can align the tool for special machining operations using the \(B\) axis and \(C\) axis of the tool spindle.

When milling, you can swivel the workpiece using the \(B\) and \(C\) axis of the main or counterspindle so that you can mill and drill on inclined surfaces.
The \(B\) axis is also used for aligning tools for face and peripheral surface machining.

\section*{Alignment angles \(\beta\) and \(\gamma\)}

Alignment angles \(\beta\) and \(\gamma\) are required for turning with tool alignment.

\(\beta\) : Rotation around the Y axis (with the B axis)
Y : Rotation around the Z axis (with the tool spindle)

\section*{Turning}

Alignment angles allow you to perform a wide range of different turning operations (for example, internal and external longitudinal machining, surface machining with a main spindle and counterspindle, residual material) without changing the tool.


Longitudinal machining, outer



Longitudinal machining, inner


Face machining, counter spindle

\section*{Display of the \(B\) axis}

The \(B\) axis is displayed in the following windows:
- Axis position display in actual value window
- In the "Positioning" window to position axes in manual operation
- You can display the B axis in the zero offset lists using the "Zero offset" softkey and you can also define the offset.

\subsection*{16.2 Tool alignment for turning}

The input fields for the \(\beta\) and \(\gamma\) angles for aligning the tool are available in the tool screen and in all turning screens.
\(\beta\) angle
In the entry field " \(\beta\) " you have the possibility of selecting the main alignment of the tool:
- \(\quad \leftarrow \quad: \beta=0^{\circ}\)
- \(\quad \downarrow \quad: \beta=90^{\circ}\)
- Empty : The required angle can be freely entered

Programming when machining with a counterspindle
When machining on the counterspindle, it is programmed in exactly the same way as the main spindle.

The direction display of the arrows depends on the settings.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Y angle}

In the entry field " \(\gamma\) " you have the possibility of selecting the main alignment of the tool:
- \(0^{\circ}\)
- \(180^{\circ}\)
- or -
- Free entry field to enter the required angle


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\subsection*{16.3 Milling with a B axis}

No special entries are required for face machining and peripheral surface machining.

\section*{Face machining}

Milling at the face (G17) is realized on the main spindle in the \(B\) axis position \(B=0^{\circ}\). If you are machining at the face (G17) of the counterspindle, then this corresponds to the opposite setting of the \(B\) axis position \(B=180^{\circ}\).

\section*{Peripheral surface machining}

Milling on the peripheral surface is always realized in the \(B\) axis position, \(B=90^{\circ}\) (main and counterspindle).

\section*{Machining on an inclined surface}

You can define inclined surfaces in a swivel mask.
You can enter the rotation of the planes around the geometry axes ( \(\mathrm{X}, \mathrm{Y}, \mathrm{Z}\) ) of the tool coordinate system as described in the workpiece drawing. The rotation of the Work in the program is then automatically converted to a rotation for the relevant \(B\) and \(C\) axis of the machine during machining.

The swivel axes are always rotated to place the tool axis perpendicular to the machining axis for machining. During machining, the rotary axes are stationary.
The coordinate system is adapted to the surface to be machined irrespective of the required rotary axis positions.

\subsection*{16.4 Swiveling}

\section*{General sequence}
- Swivel the coordinate system into the plane to be machined via the swivel screen.
- Machining with the setting "Face B".
- If another machining type follows, swiveling is automatically deselected.

The swiveled coordinates are maintained in the reset state and after Power On. So you can still move out of an inclined hole, for example, with retraction in the \(+Z\) direction.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

Swiveling is executed one axis at a time. In the case of axis-by-axis swiveling, the coordinate system is rotated around each axis in turn, with each rotation starting from the previous rotation. The axis sequence can be freely selected.

\section*{Procedure}
1. The "Spigot Res. Mat." input window opens if you press the "Various" softkey.
2. Press the "Swivel plane" softkey.
3. Press the "Initial position" softkey if you wish to reestablish the basic setting, i.e. you wish to set the values back to 0 .
This is done, for example, to swivel the coordinate system back to its original orientation.
\begin{tabular}{|c|c|c|c|}
\hline Parameter & \multicolumn{2}{|l|}{Description} & Unit \\
\hline T & \multicolumn{2}{|l|}{Tool identifier} & \\
\hline RP & \multicolumn{2}{|l|}{Retraction plane for face B} & mm \\
\hline C0 & \multicolumn{2}{|l|}{Positioning angle for machining surface} & Degrees \\
\hline X0 & \multicolumn{2}{|l|}{Reference point for rotation} & mm \\
\hline Y0 & \multicolumn{2}{|l|}{Reference point for rotation} & mm \\
\hline Z0 & \multicolumn{2}{|l|}{Reference point for rotation} & mm \\
\hline \begin{tabular}{l}
Swivel mode \\
U
\end{tabular} & \multicolumn{2}{|l|}{\begin{tabular}{l}
- axis-by-axis: Swivel coordinate system axis-by-axis \\
- Solid angle: Swivel via solid angle \\
- Proj. angle: Swiveling via projection angle \\
- Direct: Position rotary axes directly
\end{tabular}} & \\
\hline Axis sequence 0 & \multicolumn{2}{|l|}{\begin{tabular}{l}
Sequence of the axes which are rotated around - (only for axis-by-axis swivel mode) \\
XYZ or XZY or YXZ or YZX or ZXY or ZYX
\end{tabular}} & \\
\hline X & Axis angle & \multirow[t]{3}{*}{The sequence of the axes can be interchanged as required using the Select key.} & Degrees \\
\hline Y & Axis angle & & Degrees \\
\hline Z & Axis angle & & Degrees \\
\hline X1 & \multicolumn{2}{|l|}{New zero point of rotated surface} & mm \\
\hline Y1 & \multicolumn{2}{|l|}{New zero point of rotated surface} & mm \\
\hline Z1 & \multicolumn{2}{|l|}{New zero point of rotated surface} & mm \\
\hline
\end{tabular}

\section*{Note}

Other additive transformations can be added to the offsets before (X0, Y0, Z0) or after (X1, Y1, Z1) swiveling (see Section "Work offsets").

\subsection*{16.5 Approach/retraction}

If you want to optimize approach/return for swiveling with the B axis, you can create a special cycle that ignores the automatic approach/retraction strategy.

You can insert the approach/retraction cycle between any machining step program blocks, but not within linked program blocks.

\section*{Sequence}

The starting point for the approach/retraction cycle is the safety clearance approached after the last machining operation.
If you want to perform a tool change, you can move the tool through a total of three positions (P1 to P3) to the tool change point and through a maximum of three additional positions (P4 to P6) to the next starting point.
The 1st, 3rd, 4th and 6th positions move the linear axes, while the 2 nd and 5 th positions move the rotary axes.

If no tool change is needed you can generate no more than six motion blocks.
The numbers (1-6) represent the processing sequence.

\section*{Note}

Programming additional positions
If three or six positions are not sufficient for the approach/retraction, you can call the cycle several times in succession to program further positions.

\section*{NOTICE}

\section*{Risk of collision}

Note that the tool will move from the last position programmed in the approach/retraction cycle directly to the starting point for the next machining operation.
\begin{tabular}{|c|c|c|}
\hline Parameter & Description & Unit \\
\hline F1 & Feedrate to approach the first position Alternatively, rapid traverse & mm/min \\
\hline X1 & 1. position (inc or \(\varnothing\) abs) & mm \\
\hline Z1 & 1. position (inc or \(\varnothing\) abs) & mm \\
\hline Y1 & Retraction to safety clearance & mm \\
\hline \(\beta 2\) & Beta angle for 1st swivel movement & Degrees \\
\hline Y2 & Gamma angle for 1st swivel movement & Degrees \\
\hline Tracking & The position of the tool tip is maintained during swiveling. Please refer to the machine manufacturer's instructions. & \\
\hline F3 & Feedrate to approach the third position Alternatively, rapid traverse & mm/min \\
\hline X3 & 3. position (inc or \(\varnothing\) abs) & mm \\
\hline Z3 & 3. position (inc or \(\varnothing\) abs) & mm \\
\hline Tool change & \begin{tabular}{l}
- TIChngPt: Approach the tool change point from the last programmed position and carry out a tool change \\
- Direct: Tool is not changed at the tool change position, but at the last programmed position \\
- No: Tool is not changed
\end{tabular} & \\
\hline T & Tool name (not for tool change "no") & \\
\hline D & Cutting edge number (not for tool change "no") & \\
\hline F4 & Feedrate for approach to the fourth position Alternatively, rapid traverse & mm/min \\
\hline X4 & 4. position (inc or \(\varnothing\) abs) & mm \\
\hline Z4 & 4. position (inc or \(\varnothing\) abs) & mm \\
\hline \(\beta 5\) & Beta angle for 2nd swivel movement & Degrees \\
\hline y5 & Gamma angle for 2nd swivel movement & Degrees \\
\hline Tracking & The position of the tool tip is maintained during swiveling. Please refer to the machine manufacturer's instructions. & \\
\hline F6 & Feedrate to approach the sixth position Alternatively, rapid traverse & mm/min \\
\hline X6 & 6. position (inc or \(\varnothing\) abs) & mm \\
\hline Z6 & 6. position (inc or \(\varnothing\) abs) & mm \\
\hline Y6 & Retraction to safety clearance & mm \\
\hline
\end{tabular}

Turning

\subsection*{16.6 Position pattern}

In drilling and milling operations with face B, position patterns "full circle/pitch circle" provide the following options for machining on inclined surfaces
- with swivel plane
- with C axis

\section*{Procedure}

Positions
2. Press the "Positions" and "Full circle/pitch circle" softkeys.
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline & Face B on the swivel plane & mm \\
\hline Z0 & Z coordinate of the reference point (abs) & mm \\
\hline X0 & X coordinate of the reference point (abs) & mm \\
\hline Y0 & Y coordinate of the reference point (abs) & Degrees \\
\hline a0 & \begin{tabular}{l} 
Starting angle: Angle of 1st hole with reference to X axis. \\
Positive angle: Full circle is rotated counterclockwise. \\
Negative angle: Full circle is rotated in clockwise direction.
\end{tabular} & \\
\hline a 1 & \begin{tabular}{l} 
Indexing angle: After the first hole has been drilled, all additional positions are \\
approached at this angle (only for pitch circle). \\
Positive angle: Additional positions are rotated in counterclockwise direction. \\
Negative angle: Additional positions are rotated in clockwise direction.
\end{tabular} & Degrees \\
\hline N & Radius of full circle & mm \\
\hline Positioning & Number of positions on circle & mm \\
\hline Z0 & \begin{tabular}{l} 
Straight line: Next position is approached linearly in rapid traverse. \\
Circular: Next position is approached along a circular path at the feedrate defined \\
in the machine data.
\end{tabular} & \\
\hline X0 & Face B - wit C axis & Z coordinate of the reference point (abs) \\
\hline Y0 & X coordinate of the reference point (abs) & mm \\
\hline a0 & Y coordinate of the reference point (abs) & mm \\
\hline & \begin{tabular}{l} 
Starting angle: Angle of 1st hole with reference to C axis. \\
Positive angle: Full circle is rotated counterclockwise. \\
Negative angle: Full circle is rotated in clockwise direction.
\end{tabular} & \\
\hline
\end{tabular}
\begin{tabular}{|l|l|l|}
\hline Parameter & Description & Unit \\
\hline\(\alpha 1\) & \begin{tabular}{l} 
Indexing angle: After the first hole has been drilled, all additional positions are \\
approached at this angle (only for pitch circle). \\
Positive angle: Additional positions are rotated in counterclockwise direction. \\
Negative angle: Additional positions are rotated in clockwise direction.
\end{tabular} & Degrees \\
\hline N & Number of positions on circle & \\
\hline
\end{tabular}

\subsection*{16.7 Tool selection for the manual mode}

For the preparatory actions in the manual mode, tool selection and spindle control are both performed centrally in the \(\mathrm{T}, \mathrm{S}, \mathrm{M}\) window.


Figure 16-1 TSM window for the \(B\) and \(C\) axis

\section*{Procedure}

Aligning the milling and turning tool with \(\beta\) angle:


Press the <SELECT> key and select
- \(0^{\circ}\) or
- \(90^{\circ}\) or
- Value entry box to freely enter the angle.

Aligning the turning tool with \(\gamma\) angle:


Press the <SELECT> key and select
- \(0^{\circ}\)
- \(180^{\circ}\)
- Value entry box to freely enter the angle.

\section*{See also}

Selecting a tool and spindle (Page 109)

\subsection*{16.8 Measuring a tool with the \(B\) axis}

When measuring manually, traverse the tool manually to a known reference point in order to determine the tool dimensions in the \(X\) and \(Z\) directions. The control system then calculates the tool offset data from the position of the tool carrier reference point and the reference point.

To determine the tool dimensions, the orientation, i.e. the \(\beta\) angle, must be specified. The y angle input box is also provided for turning tools.

\section*{Reference point}

The workpiece edge is used as the reference point when measuring length \(X\) and length \(Z\). The chuck of the main or counterspindle can also be used when measuring in the \(Z\) direction.
Specify the position of the workpiece edge during the measurement.

\section*{\(\beta\) angle}

In order to measure milling and turning tools, you can select the two main settings \(\beta=0^{\circ}\) and \(\beta=90^{\circ}\) and a value input field.

\section*{y angle}

In order to measure turning tools, you can select the Y angle \(0^{\circ}\) and \(180^{\circ}\).

\section*{Procedure}


1, Select "JOG" mode in the "Machine" operating area.
2. Execute the tool change and alignment in the \(T, S, M\) window before performing the measurement.
3. Press the "Meas. tool" softkey.
4. Press the " \(X\) " or " \(Z\) " softkey, depending on which tool length you want to measure.
5. Scratch the required edge using the tool.
\begin{tabular}{|c|c|c|}
\hline \multirow[t]{5}{*}{Save position} & & If you do not wish to keep the tool at the workpiece edge, then press the "Save position" softkey. \\
\hline & & The tool position is saved and the tool can be retracted from the workpiece. For instance, this can be practical if the workpiece diameter still has to be subsequently measured. \\
\hline & & If the tool can remain at the workpiece edge, then after scratching you can directly continue with step 7 . \\
\hline & \multirow[t]{2}{*}{6.} & Enter the position of the workpiece edge in X0 or ZO . \\
\hline & & If no value is entered for XO or ZO , the value is taken from the actual value display. \\
\hline \multirow[t]{2}{*}{Set length} & \multirow[t]{2}{*}{7.} & Press the "Set length" softkey. \\
\hline & & The tool length is calculated automatically and entered in the tool list. Whereby the cutting edge position and tool radius or diameter are automatically taken into consideration as well. \\
\hline
\end{tabular}

\section*{Note}

Tool measurement is only possible with an active tool.

\section*{See also}
\[
\underline{\mathrm{T}, \mathrm{~S}, \mathrm{M} \text { window (Page 109) }}
\]

Measuring a tool manually (Page 78)

\section*{Working with two tool carriers}

With SINUMERIK Operate, you can work at a lathe with two tool holders, both of which are mounted on an X axis. The tool holders may be revolvers, multifix, or a combination of both.
The main machining is performed in the negative \(X\) axis direction. As both tool holders are mounted on the same axis it is only possible to machine with one tool.
The workpiece is always located between the two tool holders. The tool lengths of all tools, i.e. of both tool holders, have the same reference point, which is usually on tool holder 1. That is why the tool lengths of the tools of the second tool holder are always longer than those of the tools on the first tool holder.

\subsection*{17.1 Programming with two tool holders}

You always program in the basic coordinate system (workpiece coordinate system of the first tool holder). You do not have to take into account in which tool holder the tool is inserted.

\section*{Tool on the second tool holder}

If a tool on the second tool holder is selected, the X and Y axes are mirrored and the main spindle and counterspindle are offset (rotated) through \(180^{\circ}\).


In the simulation, the tool is always displayed on the correct side, just the same as it is used at the machine.

The programmed \(C\) offset around \(180^{\circ}\) only affects \(C\) axes, not spindles.
It is not possible to machine a thread with tools that are distributed between both tool holders.

\section*{G code programming}

The following points must be taken into account for G code programming:
- After a tool change, tool mirroring on the second tool holder is automatically activated.
- When a TRANSMIT command is programmed, tool mirroring on the second tool holder is automatically activated.

\subsection*{17.2 Measure tool}

Selection options "Toolholder 1" and "Toolholder 2" are available for scratching when measuring a tool. This is how you set the tool holder in which the tool to be measured is located.


\section*{Teaching in a program}

\subsection*{18.1 Overview}

The "Teach in" function can be used to edit programs in the "AUTO" and "MDA" modes. You can create and modify simple traversing blocks.
You traverse the axes manually to specific positions in order to implement simple machining sequences and make them reproducible. The positions you approach are applied.

In "AUTO" teach-in mode, the selected program is "taught".
In "MDA" teach-in mode, you teach to the MDA buffer.
External programs, which may have been rendered offline, can therefore be adjusted and modified according to need.

\subsection*{18.2 General sequence}

\section*{General sequence}

Select the desired program block, press the relevant softkey "Teach position", "Rap. tra. G01", "Straight line G1" or "Circ. interm. pos. CIP", and "Circ. end pos. CIP" and traverse the axes to change the program block.
You can only overwrite a block with a block of the same type.
- OR -

Position the cursor at the desired point in the program, press the relevant softkey "Teach position", "Rap. tra. G01", "Straight line G1" or "Circ. interp. pos. CIP", and "Circ. end pos. CIP" and traverse the axes to insert a new program block.
In order for the block to be inserted, the cursor must be positioned in an empty line using the cursor key and input key.
Press the "Accept" softkey to teach-in the modified or new program block.

\section*{Note}

All defined axes are "taught in" in the first teach-in block. In all additional teach-in blocks, only axes modified by axis traversing or manual input are "taught in".
If you exit teach-in mode, this sequence begins again.

\section*{Operating mode or operating area switchover}

If you switch to another operating mode or operating area in teach-in mode, the position changes will be canceled and teach-in mode will be cleared.

\subsection*{18.3 Inserting a block}

You have the option of traversing the axes and writing the current actual values directly to a new position block.

\section*{Requirement}
"AUTO" mode: The program to be edited is selected.

\section*{Proceed as follows}
'M
Machine

\section*{(3)}

теасн IN

4. Press the "Teach prog." softkey.
5. Traverse the axes to the relevant position.

Teach position
1. Select the "Machine" operating area.
2. Press the <AUTO> or <MDA> key.
3. Press the <TEACH IN> key.
6. Press the "Teach position" softkey.

A new program block with the current actual position values will be created.

\subsection*{18.3.1 Input parameters for teach-in blocks}

\section*{Parameters for teach-in of position and teach-in of G0, G1, and circle end position CIP}
\begin{tabular}{|l|l|}
\hline Parameter & Description \\
\hline X & Approach position in X direction \\
\hline Y & Approach position in Y direction \\
\hline Z & Approach position in Z direction \\
\hline F & Feedrate ( \(\mathrm{mm} / \mathrm{r} ; \mathrm{mm} / \mathrm{min}\) ) - only for teach-in of G1 and circle end position CIP \\
U & \\
\hline
\end{tabular}

\section*{Parameters for teach-in of circle intermediate position CIP}
\begin{tabular}{|l|l|}
\hline Parameter & Description \\
\hline I & Coordinate of the circle center point in the X direction \\
\hline J & Coordinate of the circle center point in the Y direction \\
\hline K & Coordinate of the circle center point in the Z direction \\
\hline
\end{tabular}

\section*{Transition types for teach-in of position and teach-in of G0 and G1, and ASPLINE}

The following parameters are offered for the transition:
\begin{tabular}{|l|l|}
\hline Parameter & Description \\
\hline G60 & Exact stop \\
\hline G64 & Corner rounding \\
\hline G641 & Programmable corner rounding \\
\hline G642 & Axis-specific corner rounding \\
\hline G643 & Block-internal corner rounding \\
\hline G644 & Axis dynamics corner rounding \\
\hline
\end{tabular}

Motion types for teach-in of position and teach-in of G0 and G1
The following motion parameters are offered:
\begin{tabular}{|l|l|}
\hline Parameter & Description \\
\hline CP & Path-synchronous \\
\hline PTP & Point-to-point \\
\hline PTPG0 & Only G0 point-to-point \\
\hline
\end{tabular}

\section*{Transition behavior at the beginning and end of the spline curve}

The following motion parameters are offered:
\begin{tabular}{|l|l|}
\hline Parameter & Description \\
\hline Start & Automatic calculation \\
\hline BAUTO & Curvature is zero or natural \\
\hline BNAT & Tangential \\
\hline BTAN & \\
\hline \multicolumn{2}{|l|}{} \\
\hline End & Automatic calculation \\
\hline EAUTO & Curvature is zero or natural \\
\hline ENAT & Tangential \\
\hline ETAN &
\end{tabular}

\subsection*{18.4 Teach-in via Windows}

\subsection*{18.4.1 General}

The cursor must be positioned on an empty line.
The windows for pasting program blocks contain input and output fields for the actual values in the WCS. Depending on the default setting, selection fields with parameters for motion behavior and motion transition are available.

When first selected, the input fields are empty unless axes were already traversed before the window was selected.

All data from the input/output fields are transferred to the program via the "Accept" softkey.

\section*{Precondition}
"AUTO" mode: The program to be edited is selected.

\section*{Procedure}

5. Use the cursor and input keys to position the cursor at the desired point in the program.
If an empty row is not available, insert one.
Rap. tra.
G0
Circ. end
pos. CIP
7. Traverse the axes to the relevant position.
8. Press the "Accept" softkey.

A new program block will be inserted at the cursor position.
- OR -

Press the "Cancel" softkey to cancel your input.

\subsection*{18.4.2 Teach in rapid traverse G0}

You traverse the axes and teach-in a rapid traverse block with the approached positions.

\section*{Note}

Selection of axes and parameters for teach-in
You can select the axes to be included in the teach-in block in the "Settings" window.
You also specify here whether motion and transition parameters are offered for teach-in.

\subsection*{18.4.3 Teach in straight G1}

You traverse the axes and teach-in a machining block (G1) with the approached positions.

\section*{Note}

Selection of axes and parameters for teach-in
You can select the axes to be included in the teach-in block in the "Settings" window.
You also specify here whether motion and transition parameters are offered for teach-in.

\subsection*{18.4.4 Teaching in circle intermediate and circle end point CIP}

Enter the intermediate and end positions for the circle interpolation CIP. You teach-in each of these separately in a separate block. The order in which you program these two points is not specified.

\section*{Note}

Make sure that the cursor position does not change during teach-in of the two positions.

You teach-in the intermediate position in the "Circle intermediate position CIP" window.
You teach-in the end position in the "Circle end position CIP" window.
The intermediate or interpolation point is only taught-in with geometry axes. For this reason, at least 2 geometry axes must be set up for the transfer.

\section*{Note}

\section*{Selection of axes for teach in}

You can select the axes to be included in the teach-in block in the "Settings" window.

\subsection*{18.4.5 Teach-in A spline}

For Akima-spline interpolation, you enter interpolation points that are connected by a smooth curve.

Enter a starting point and specify a transition at the beginning and end.
You teach-in each interpolation point via "Teach in of position".


\section*{Software option}

You require the "Spline-Interpolation" option for A Spline interpolation.

\section*{Note}

The relevant option bit must be set to enable you to program a spline interpolation.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
:
Machine

2. Press the <AUTO> or <MDA> key.
3. Press the <TEACH IN> key.
4. Press the "Teach prog." softkey.
5. Press the ">>" and "ASPLINE" softkeys.

The "Akima-spline" window opens with the input fields.

\section*{ASPLINE}


Cancel
1. Select the "Machine" operating area.
6. Traverse the axes to the required position and if necessary, set the transition type for the starting point and end point.
7. Press the "Accept" softkey.

A new program block will be inserted at the cursor position.
- OR -

Press the "Cancel" softkey to cancel your input.

\section*{Note}

\section*{Selection of axes and parameters for teach-in}

You can select the axes to be included in the teach-in block in the "Settings" window.
You also specify here whether motion and transition parameters are offered for teach-in.

\subsection*{18.5 Editing a block}

You can only overwrite a program block with a teach-in block of the same type.
The axis values displayed in the relevant window are actual values, not the values to be overwritten in the block.

\section*{Note}

If you wish to change any variable in a block in the program block window other than the position and its parameters, then we recommend alphanumerical input.

\section*{Requirement}

The program to be processed is selected.

\section*{Procedure}


Machine

2. Press the <AUTO> or <MDA> key.

Teach position

Circ. end pos. CIP
4. Press the "Teach prog." softkey.
5. Click the program block to be edited.
3. Press the <TEACH IN> key.
6. Press the relevant softkey "Teach position, "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos. CIP", and "Circ. end pos. CIP".
The relevant windows with the input fields are displayed.
7. Traverse the axes to the desired position and press the "Accept" softkey.
The program block is taught with the modified values.
- OR -

Press the "Cancel" softkey to cancel the changes.

\subsection*{18.6 Selecting a block}

You have the option of setting the interrupt pointer to the current cursor position. The next time the program is started, processing will resume from this point.

With teach-in, you can also change program areas that have already been executed. This automatically disables program processing.

You must press reset or select a block to resume the program.

\section*{Requirement}

The program to be processed is selected.

\section*{Proceed as follows}

Machine


Block selection
2. Press the <AUTO> key.
3. Press the <TEACH IN> key.
4. Press the "Teach prog." softkey.
5. Place the cursor on the desired program block.
1. Select the "Machine" operating area.
6. Press the "Block selection" softkey.

\subsection*{18.7 Deleting a block}

You have the option of deleting a program block entirely.

\section*{Requirement}
"AUTO" mode: The program to be processed is selected.

\section*{Procedure}

\section*{' \({ }^{\prime}\)}

Machine

2. Press the <AUTO> or <MDA> key.

3. Press the <TEACH IN> key.
5. Click the program block to be deleted.

6. Press the ">>" and "Delete block" softkeys.

The program block on which the cursor is positioned is deleted.
Delete
block

\subsection*{18.8 Settings for teach-in}

In the "Settings" window, you define which axes are to be included in the teach-in block and whether motion-type and continuous-path mode parameters are to be provided.

\section*{Proceed as follows}

Machine

\section*{\(\rightarrow\)
AUTO}

图
мDA

\section*{包 \\ TEACH IN}


Settings

1. Select the "Machine" operating area.
2. Press the <AUTO> or <MDA> key.
3. Press the <TEACH IN> key.
4. Press the "Teach prog." softkey.
5. Press the ">>" and "Settings" softkeys.

The "Settings" window appears.
6. Under "Axes to be taught" and "Parameters to be taught", select the check boxes for the relevant settings and press the "Accept" softkey to confirm the settings.

\section*{19.1 \\ HT 8 overview}

The mobile SINUMERIK HT 8 handheld terminal combines the functions of an operator panel and a machine control panel. It is therefore suitable for visualization, operation, teach in, and programming at the machine.


\section*{Operation}

The 7.5 TFT color display provides touch operation.

It also has membrane keys for traversing the axes, for numeric input, for cursor control, and for machine control panel functions like start and stop.

It is equipped with an emergency stop button and two 3-position enabling buttons. You can also connect an external keyboard.

\section*{References}

For more information about connection and startup of the HT 8, see the following references:
Commissioning Manual SINUMERIK Operate (IM9) / SINUMERIK 840D sl

\section*{Customer keys}

The four customer keys are freely assignable and can be set up customer-specifically by the machine manufacturer.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Integrated machine control panel}

The HT 8 has an integrated machine control panel consisting of keys (e.g. start, stop, traversing keys, etc.), and keys reproduced as softkeys (see machine control panel menu).

See Section "Controls on the machine control panel" for a description of the individual keys.

\section*{Note}

PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

\section*{Enabling button}

The HT 8 has two enabling buttons. Thus, you can initiate enabling functions for operations that require enabling (e.g. displaying and operating of traversing keys) with either your right hand or your left hand.
Enabling buttons are available for the following key positions:
- Released (no activation)
- Enabling (center position) - enabling for channel 1 and 2 is on the same switch
- Panic (completely pushed through)

\section*{Traversing keys}

To traverse the axes of your machine using the traversing keys of the HT 8, you must select "JOG" mode or either the "Teach In" or "Ref.Point" submode. Depending on the setting, the enabling button must be activated.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Virtual keyboard}

A virtual keyboard is available for the easy entry of values.

\section*{Changing the channel}
- You are able to switch the channel by touch in the status display:
- In the Machine operating area (large status display), by touch operation of the channel display in the status display.
- In the other operating areas (no status display), by touch operation of the channel display in the screen headers (yellow field).
- The "1... n CHANNEL" softkey is available in the machine control panel menu that can be reached via the user menu key "U".

\section*{Operating area switchover}

You can display the operating area menu by touching the display symbol for the active operating area.

\section*{Handwheel}

The HT 8 is available with a hand wheel.

\section*{References}

For information about connecting the hand wheel, refer to:
Operator Components and Networking Manual; SINUMERIK 840D sl/840Di sl

\subsection*{19.2 Traversing keys}

The traversing keys are not labeled. However, you can display a label for the keys in place of the vertical softkey bar.

Labeling of the traversing keys is displayed for up to six axes on the touch panel by default.


Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Showing and hiding}

You can link the showing and hiding of the label to activation of the enabling button, for example. In this case, the traversing keys are displayed when you press the enabling button.

If you release the enabling button, the traversing keys are hidden again.


Machine manufacturer
Please refer to the machine manufacturer's specifications.


All existing vertical and horizontal softkeys are covered or hidden, i.e. other softkeys cannot be used.

\subsection*{19.3 Machine control panel menu}

Here you select keys from the machine control panel which are reproduced by the software by touch operation of the relevant softkeys.

See chapter "Controls on the machine control panel" for a description of the individual keys.

\section*{Note}

PLC interface signals that are triggered via the softkeys of the machine control panel menus are edge triggered.

\section*{Showing and hiding}

The user menu key "U" displays the CPF softkey bar (vertical softkey bar) and the user softkey bar (horizontal softkey bar).


You can expand the user softkey bar to display eight additional softkeys via the menu forward key.

You use the "Back" softkey to hide the menu bar again.

\section*{Softkeys on the machine control panel menu}
\begin{tabular}{ll} 
Available softkeys: & \\
"Machine" softkey & Select the "Machine" operating area \\
"[VAR]" softkey & Select the axis feedrate in the variable increment \\
"1...n CHANNEL" & Change the channel \\
softkey & \\
"Single Block" soft- & Switch single block execution on/off \\
key & \\
"WCS MCS" softkey Switch between WCS and MCS \\
"Back" softkey & Close the window.
\end{tabular}

\section*{Note}

The window will automatically disappear when changing regions areas with the "Menu Select" key.

\subsection*{19.4 Virtual keyboard}

The virtual keyboard is used as the input device for touch operator panels.
It opens when you double-click an operator element with input capability (editor, edit field). The virtual keyboard can be positioned anywhere on the operator interface. In addition, you can toggle between a full keyboard and a reduced keyboard that only includes the number block. Moreover, with the full keyboard, you can toggle between English key assignments and the keyboard assignment for the current language setting.

\section*{Procedure}
1. Click in the required input field in order to place the cursor there.
2. Click the input field. The virtual keyboard is displayed.
3 Enter your values via the virtual keyboard.
4. Press the <INPUT> key.
- OR -

Position the cursor on an another operator element.
The value is accepted and the virtual keyboard is closed.

\section*{Positioning of the virtual keyboard}

You can position the virtual keyboard anywhere in the window by pressing the empty bar next to the "Close window" icon with your finger or a stylus and moving it back and forth.

\section*{Special keys on the virtual keyboard}

(1) Num:

Reduces the virtual keyboard to the number block.
(2) Eng:

Toggles the keyboard assignment between the English keyboard assignment and the keyboard assignment for the current language setting.

\section*{Number block of the virtual keyboard}


Use the "Deu" or "Eng" keys to return to the full keyboard with the English keyboard assignment or the keyboard assignment of the current language setting.

\subsection*{19.5 Calibrating the touch panel}

It is necessary to calibrate the touch panel upon first connection to the controller.

\section*{Note}

Recalibration
If the operation is not exact, then redo the calibration.


\section*{Procedure}

1. Press the back key and the <MENU SELECT> key at the same time to start the TCU service screen.
2. Touch the "Calibrate TouchPanel" button.

The calibration process will be started.
3. Follow the instructions on the screen and touch the three calibration points one after the other.
The calibration process has terminated.
4. Touch the horizontal softkey "1" or the key with the number "1" to close the TCU service screen.

\section*{Ctrl-Energy}

\section*{20.1 \\ Overview}

The "Ctrl-Energy" function provides you with the following options to improve the energy utilization of your machine.

\section*{Ctrl-E Analysis: Measuring and evaluating the energy consumption}

Acquiring the actual energy consumption is the first step to achieving better energy efficiency. The energy consumption is measured and displayed at the control using the SENTRON PAC multi-function device.
Depending on the configuration and connection of the SENTRON PAC, you have the possibility of either measuring the power of the whole machine or only a specific load.

Independent of this, the power is determined directly from the drives and displayed.

\section*{Ctrl-E Profiles: Control of energy saving states of the machine}

To optimize the energy consumption, you have the option of defining energy saving profiles and saving them. For instance, your machine has a basic and a more sophisticated energysaving mode - or under certain conditions, automatically switches itself off.

These defined energy states are saved as profiles. At the user interface, you have the possibility of activating these energy-saving profiles (e.g. the so-called tea break key).

\section*{Note}

\section*{Ctrl-E Deactivating profiles}

Disable Ctrl-E profiles before a series startup in order to prevent the NCU unintentionally shutting down.


\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\subsection*{20.2 Displaying energy consumption}

The SINUMERIK CtrI-Energy entry screen provides an easy-to-interpret overview of the energy consumption of the machine. To display the values and the graphical representation, a Sentron PAC must be connected and a long-term measurement configured.

This shows a consumption display with the following bar chart:
- Current power display
- Measurement of the current energy consumption
- Comparison measurement for the energy consumption


Figure 20-1 Ctrl-Energy entry screen with display of the current energy consumption

\section*{Display in the "Machine" operating area}

The first row of the status display shows the current power status of the machine.
\begin{tabular}{|l|l|}
\hline Display & Meaning \\
\hline\(=\mathbb{M}\) & A red bar indicates that the machine is not operating productively. \\
\hline \(\bar{M}\) & \begin{tabular}{l} 
A dark-green bar in the positive direction indicates that the machine is operating \\
productively and consuming energy.
\end{tabular} \\
\hline \(\bar{M}\) & \begin{tabular}{l} 
A light-green bar in the negative direction indicates that the machine is feeding \\
energy back into the power supply system.
\end{tabular} \\
\hline
\end{tabular}

\section*{References}

Information on the configuration is provided in the following reference:
System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

\section*{Procedure}



Ctrl-
Energy

- OR -
\(+\quad\) Press the <Ctrl> + <E> keys.

E
The "SINUMERIK Ctrl-Energy" window opens.
2. Press the menu forward key and then the "Ctrl-Energy" softkey.

\subsection*{20.3 Measuring and saving the energy consumption}

For the currently selected axes, you have the option of measuring and recording the energy consumption.

\section*{Measurement of the energy consumption by part programs}

The energy consumption of part programs can be measured. Single drives are taken into account for the measurement.

They specify in which channel the start and stop of the part program should be initiated and which number of repetitions are to be measured.

\section*{Save measurements}

Save the measured consumption values so that you can subsequently compare the data.

\section*{Note}

Up to three data records are displayed. The oldest data record is automatically overwritten if there are more than three measurements.

\section*{Measurement duration}

The measurement duration is limited. When the maximum measurement duration is reached, the measurement is terminated and a corresponding message is output in the dialog line.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

\section*{Procedure}
```

CtrI-E analysis

```

Start measuremen

2. Press the "Ctrl-E analysis" softkey.

The "Ctrl-E Analysis" window opens.
3. Press the "Start measurement" softkey.

The "Setting Measurement: Select Device" window opens.
4. Select the desired device in the list, possibly activate the "Measure part
program" checkbox, enter the number of repetitions, possibly select the
4. Select the desired device in the list, possibly activate the "Measure part
program" checkbox, enter the number of repetitions, possibly select the desired channel, and press the "OK" softkey. The trace is started.
Stop
measuremen

Display meas. curves

Details

Save measuremen
1. The "SINUMERIK Ctrl-Energy" window is open.
5. Press the "Stop measurement" softkey. The measurement is terminated.
6. Press the "Graphic" softkey, to track the measurement characteristic.

The selection of the axis to be measured depends on the configuration.

\section*{References}

Information on the configuration is provided in the following reference:
System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

\subsection*{20.4 Long-term measurement of the energy consumption}

The long-term measurement of energy consumption is performed in the PLC and saved. The values from times in which the HMI is not active are also recorded.

\section*{Measured values}

The infeed and regenerative power values as well as the sum of the power are displayed for the following periods:
- Current and previous day
- Current and previous month
- Current and previous year

\section*{Precondition}

SENTRON PAC is connected.

\section*{Procedure}
```

CtrI-E analysis

```

\author{
Long-term measuremnt
}
<< Back
1. The "Ctrl-E Analysis" window is open.
2. Press the "Long-term measurement" softkey.

The "SINUMERIK Ctrl-Energy Analysis Long-term Measurement" window opens.
The results of the long-term measurement are displayed.
4. Press the "Back" softkey to terminate the long-term measurement.

\subsection*{20.5 Displaying measured curves}

You can display current or saved measurement curves graphically or as detailed tables.
\begin{tabular}{|l|l|}
\hline Display & Meaning \\
\hline Start of the measurement & \begin{tabular}{l} 
Shows the time at which the measurement was started by pressing the \\
"Start measurement" softkey.
\end{tabular} \\
\hline \begin{tabular}{l} 
Duration of the measure- \\
ment \([\mathrm{s}]\)
\end{tabular} & \begin{tabular}{l} 
Shows the measurement duration in seconds until the "Stop measure- \\
ment" softkey is pressed.
\end{tabular} \\
\hline Device & \begin{tabular}{l} 
Displays the selected measured component. \\
- Manually
\end{tabular} \\
\hline - Total, drives \\
- Total, machine
\end{tabular}\(|\)\begin{tabular}{ll} 
Supplied energy [kWh] & \begin{tabular}{l} 
Shows the supplied energy of the selected measured component in kilo- \\
watts per hour.
\end{tabular} \\
\hline \begin{tabular}{l} 
Regenerated energy \\
{\([\mathrm{kWh}]\)}
\end{tabular} & \begin{tabular}{l} 
Shows the regenerated energy of the selected measured component in \\
kilowatts per hour.
\end{tabular} \\
\hline Energy totals [kWh] & \begin{tabular}{l} 
Shows the total of all measured drive values or the total of all axes as \\
well as fixed value and Sentron PAC.
\end{tabular} \\
\hline
\end{tabular}

Display in the "Ctrl-E Analysis" window.

\section*{Procedure}
\begin{tabular}{c|} 
CtrI-E \\
analysis \\
\hline Display \\
meas. curves \\
\hline Last \\
measuremen \\
\hline Last \\
measuremen
\end{tabular}

Details
1. The "Ctrl-E Analysis" window is open and you have already performed and saved measurements.
2. Press the "Graphic" and "Saved measurements" softkeys. The measurement curves are displayed in the "Ctrl-E Analysis" window.
3. Press the "Saved measurements" softkey again if you only want to see the actual measurement.
4. Press the "Details" softkey to display the exact measurement data and consumption values of the last three saved measurements and possibly a current measurement.

\subsection*{20.6 Using the energy-saving profile}

In the "Ctrl-E Profile" window, you can display all of the defined energy-saving profiles. Here, directly activate the required energy-saving profile - or inhibit or release profiles.

\section*{SINUMERIK CtrI-Energy energy-saving profiles}
\begin{tabular}{|l|l|}
\hline Display & Meaning \\
\hline Energy-saving profile & All energy-saving profiles are listed. \\
\hline active in [min] & The remaining time until the defined profile is reached is displayed. \\
\hline
\end{tabular}

\section*{Note}

Disable all energy-saving profiles
For example, in order not to disturb the machine while measurements are being made, select "Disable all".

Once the pre-warning time of a profile has been reached, an alarm window that shows the remaining time is displayed. Once the energy-saving mode has been reached, then an appropriate message is displayed in the alarm line.

\section*{Predefined energy-saving profiles}
\begin{tabular}{|l|l|}
\hline Energy-saving profile & Meaning \\
\hline \begin{tabular}{l} 
Simple energy-saving mode \\
(machine standby)
\end{tabular} & \begin{tabular}{l} 
Machine units that are not required are either throttled or shut down. \\
When required, the machine is immediately ready to operate again.
\end{tabular} \\
\hline \begin{tabular}{l} 
Full energy-saving mode (NC \\
standby)
\end{tabular} & \begin{tabular}{l} 
Machine units that are not required are either throttled or shut down. \\
Wait times are incurred at the transition into the ready to operate \\
state.
\end{tabular} \\
\hline \begin{tabular}{l} 
Maximum energy-saving \\
mode (auto shut-off)
\end{tabular} & \begin{tabular}{l} 
The machine is completely switched-off. \\
Longer wait times are incurred at the transition into the ready to op- \\
erate state.
\end{tabular} \\
\hline
\end{tabular}

\section*{Machine manufacturer}

The selection and function of the displayed energy-saving profiles can differ.
Please refer to the machine manufacturer's specifications.

\section*{References}

Information on the configuration of the energy-saving profiles is provided in the following reference:

System Manual "Ctrl-Energy", SINUMERIK 840D sl / 828D

\section*{Procedure}


CtrlEnergy


E

Ctrl-E profiles

Activate immediately

Disable profile

\section*{Enable}
profile
Disable

Enable all
1. Select the "Parameter" operating area.
2. Press the menu forward key and then the "Ctrl-Energy" softkey.
- OR -

Press the <CTRL>+<E> keys.

Press the "Ctrl-E profile" softkey.
The "Ctrl-E Profile" window opens.
3. Position the cursor on the required energy-saving profile and press the "Activate immediately" softkey if you wish to directly activate this state.
4. Position the cursor on the required energy-saving profile and press the "Disable profile" softkey if you wish to disable this state.
The profile is disabled. The energy-saving profile cannot be active, i.e. it is grayed out and displayed without time details.
The labeling of the "Disable profile" softkey changes to "Enable profile".
Press the "Enable profile" softkey in order to withdraw the energysaving profile disable.
5. Press the "Disable all" softkey in order to disable all states.

All the profiles are disabled and cannot be active.
The labeling of the "Disable all" softkey changes to "Enable all".
6. Press the "Enable all" softkey to withdraw the disable for all profiles.
20.6 Using the energy-saving profile

\section*{Easy Message (828D only)}

\section*{Overview}

Easy Message enables you to be informed about certain machine states by means of SMS messages via a connected modem:
- For example, you would like to be informed about emergency stop states
- You would like to know when a batch has been completed

\section*{Control commands}
- HMI commands are used to activate or deactivate a user.

Syntax: [User ID] deactivate, [User ID] activate
- A special area is reserved in the PLC to which you can send commands in the form of PLC bytes using SMS commands.

Syntax: [User ID] PLC DataByte
The user ID is optional and required only if a corresponding ID has been specified in the user profile. The string PLC indicates that a PLC byte is to be written. It is followed by the data byte to be written in the following format: Base\#Value. Base can take the values 2, 10 and 16 and defines the number base. Then follows the separator \(\#\) and the value of the byte. Only positive values are allowed to be sent.

Examples:
2\#11101101
10\#34
16\#AF
Machine manufacturer
Please refer to the machine manufacturer's specifications.

\section*{Active users}

In order to receive an SMS message for certain events, you must be activated as user.

\section*{User logon}

As registered user, you can log on via SMS to inquire about messages.

\section*{Action log}

You can obtain precise information about incoming and outgoing messages via SMS logs.

\section*{References}

Information on the GSM modem can be found in the
PPU SINUMERIK 828D Manual

\section*{Calling the SMS Messenger}
1. Select the "Diagnostics" operating area.
2. Press the "Easy Msg." softkey.

\subsection*{21.2 Activating Easy Message}

To commission the connection to the modem for the SMS Messenger, activate the SIM card at the initial start-up.

\section*{Requirement}

The modem is connected and activated.

\section*{Machine manufacturer}

The modem is activated via the machine data 51233
\$MSN_ENABLE_GSM_MODEM.
Please refer to the machine manufacturer's specifications.

\section*{Procedure}

\section*{Activating the SIM card}

Easy
Msg.
1. Press the "Easy Msg." softkey.

The "SMS Messenger" window appears.
"Status" shows that the SIM card has not been activated with a PIN.
2. Enter the PIN, repeat the PIN and press the "OK" softkey.

OK
3. If you made an incorrect entry several times, enter the PUK code in the "PUK Input" window and press the "OK" softkey to activate the PUK code.
The "PIN input" window is opened and you can enter the PIN number as usual.

Activating a new SIM card
\begin{tabular}{|c|c|c|}
\hline 区 Easy & 1. & \begin{tabular}{l}
Press the "Easy Msg." softkey. \\
The "SMS Messenger" window appears.
\end{tabular} \\
\hline & & "Status" shows that the connection to the modem has been activated. \\
\hline Settings & 2. & Press the "Settings" softkey. \\
\hline Delete PIN & 3. & \begin{tabular}{l}
Press the "Delete PIN" softkey to delete the stored PIN. \\
Enter the new PIN in the "PIN Input" window at the next power up.
\end{tabular} \\
\hline
\end{tabular}

Turning

\subsection*{21.3 Creating/editing a user profile}

User identification
\begin{tabular}{|l|l|}
\hline Display & Meaning \\
\hline User name & Name of the user to be created or logged on. \\
\hline Telephone number & \begin{tabular}{l} 
Telephone number of the user to which the messages are to be sent. \\
The telephone number must include the country code in order that \\
control commands can identify the sender (e.g. +491729999999)
\end{tabular} \\
\hline User ID & \begin{tabular}{l} 
The user ID has 5 digits (e.g. 12345) \\
- It is used to activate and deactivate the user via SMS. (e.g. \\
"12345 activate")
\end{tabular} \\
& \begin{tabular}{l} 
The ID is used to additionally verify the incoming and outgoing \\
messages and to activate the control commands.
\end{tabular} \\
\hline
\end{tabular}

\section*{Events that can be selected}

You must set-up the events for which you receive notification.

\section*{Note}

\section*{Selecting alarms}

You have the option of selecting tool management type or measuring cycles alarms. This means that you obtain notification by SMS as soon as alarms are output, without having to know the number ranges.

\section*{Requirement}

The modem is connected.

\section*{Procedure}

\section*{Creating a new user}
\begin{tabular}{|c|cl}
\begin{tabular}{c} 
User \\
profiles
\end{tabular} & 1. & \begin{tabular}{l} 
Press the "User profiles" softkey. \\
The "User Profiles" window appears.
\end{tabular} \\
& 2. & \begin{tabular}{l} 
Press the "New" softkey.
\end{tabular} \\
New & &
\end{tabular}
3. Enter the name and telephone number of the user.
4. If required, enter the ID number of the user.
5. In the area "send SMS for the following events" area, activate the appropriate checkbox and when required, enter the desired value (e.g. the unit quantity, which when it is reached, a notification should be sent).
- OR -

Press the "Default" softkey.
Standard
The appropriate window is opened and displays the default values.
Send
test SMS
6. Press the "Send test message" softkey.

An SMS message with predefined text is sent to the specified telephone number.

\section*{Editing user data and events}

Edit

Standard
1. Select the user whose data you want to edit and press the "Edit" softkey.
The input fields can be edited.
2. Enter new data and activate the desired settings.
- OR -

Press the "Default" softkey to accept the default values.

\subsection*{21.4 Setting-up events}

In the "Send SMS for the following events" area, select the events using the check box, which when they occur, an SMS is sent to the user.
- Programmed messages from the part program (MSG)

In the part program, program an MSG command via which you receive an SMS.
Example: MSG ("SMS: An SMS from a part program")
- Select the following events using the <SELECT> key
- The workpiece counter reaches the following value

An SMS is sent if the workpiece counter reaches the set value.
- The following program progress is reached (percent)

An SMS is sent if, when executing a part program, the set progress is reached.
- Actual NC program reaches runtime (minutes)

An SMS is sent after the set runtime has been reached when executing an NC program.
- Tool usage time reaches the following value (minutes)

An SMS is sent if the usage time of the tool reaches the set time when executing a part program (derived from \$AC_CUTTING_TIME).
- Messages/alarms from the Tool Manager

An SMS is sent if messages or alarms are output to the Tool Manager.
- Measuring cycle messages for tools

An SMS is sent if measuring cycle messages are output that involve tools.
- Measuring cycle messages for workpieces

An SMS is sent if measuring cycle messages are output that involve workpieces.
- Sinumerik messages/alarms (error when executing)

An SMS is sent if NCK alarms or messages are output that cause the machine to come to a standstill.
- Machine faults

An SMS is sent if PLC alarms or messages are output that cause the machine to come to a standstill (i.e. PLC alarms with Emergency Off response).
- Maintenance intervals

An SMS is sent if the service planner registers pending maintenance work.
- Additional alarm numbers:

Here, specify additional alarms where you should be notified if they occur.
You can enter individual alarms, several alarms or alarm number ranges.
Examples:
1234,400
1000-2000
100,200-300

\section*{Requirement}
- The user profile window is opened.
- You selected the event "Measuring cycle messages for tools", "Measuring cycle messages for workpieces", "Sinumerik messages/alarms (errors when executing)", "Machine faults" or "Maintenance intervals".

\section*{Editing events}

\section*{Details}

Delete

\section*{New}


\section*{Save}

Standard
1. Activate the required check box and press the "Details" softkey.

The appropriate window opens (e.g. "Measuring cycle messages for workpieces") and shows a list of the defined alarm numbers.
2. Select the corresponding entry and press the "Delete" softkey to remove the alarm number from the list.
- OR -

Then press the "New" softkey if you wish to create a new entry.
The "Create new entry" window opens.
Enter the data and press the "OK" softkey to add the entry to the list.

Press the "Save" softkey to save the settings for the result.
3. Press the "Standard" softkey to return to the standard settings for the events.

Turning

\subsection*{21.5 Logging an active user on and off}

Only active users receive an SMS message for the specified events.
You can activate users, already created for Easy Message, with certain control commands via the user interface or via SMS.

\section*{Easu Msg. \\ active}

The connection has been established to the modem.

\section*{Procedure}
\begin{tabular}{|c}
\begin{tabular}{c} 
User \\
profiles
\end{tabular} \\
\hline \begin{tabular}{c} 
User \\
active
\end{tabular} \\
\hline \\
\hline \begin{tabular}{c} 
User \\
active
\end{tabular} \\
\hline
\end{tabular}
1. Press the "User profiles" softkey.
2. Select the desired user in the User name field and press the "User active" softkey

Note
Repeat step 2 to activate further users.
- OR -

Send an SMS with the User ID and the "activate" text to the control (e.g. "12345 activate")
If the telephone number and the user ID match the stored data, the user profile is activated.
You receive a message of success or failure per SMS.
3. Press the "User active" softkey to log off an activated user.

\section*{- OR -}

Send an SMS with the "deactivate" text (e.g. "12345 deactivate") to log off from the Messenger.
An SMS message is not sent to the deactivated user for the events specified in the user profile.

\subsection*{21.6 Displaying SMS logs}

The SMS data traffic is recorded in the "SMS Log" window. In this way, it is possible to see the chronological sequence of activates when a fault occurs.
\begin{tabular}{|l|l|}
\hline Symbols & Description \\
\hline \multicolumn{3}{|l|}{} & Incoming SMS message for the Messenger. \\
\hline 9 & \begin{tabular}{l} 
Message that has reached the Messenger, but which has not been \\
processed (e.g. incorrect user ID or unknown account).
\end{tabular} \\
\hline \hdashline & SMS message sent to a user. \\
\hline Message that has not reached the user because of an error. \\
\hline
\end{tabular}

Requirement

\section*{Easy Msg.} active

The connection has been established to the modem.

\section*{Procedure}
1. Press the "SMS log" softkey.

The "SMS Log" window appears.
All the messages that have been sent or received by the Messenger are listed.
Outgoing

\section*{Note}

Press the "Incoming" or "Outgoing" softkey to restrict the list.

\subsection*{21.7 Making settings for Easy Message}

You can change the following Messenger configuration in the "Settings" window:
- Name of the controller that is part of an SMS message
- Number of sent messages
- The SMS counter provides information on all sent messages.
- Limit the number of sent messages in order to receive an overview of the costs through SMS messages, for example.

\section*{Setting the SMS counter to zero}
\begin{tabular}{c|l}
\begin{tabular}{c} 
Reset SMS \\
counter
\end{tabular} & \begin{tabular}{l} 
When a set limit is reached, no further SMS messages are sent. \\
Press the "Reset SMS counter" softkey to reset the counter to zero.
\end{tabular}.
\end{tabular}

\section*{Requirement}

\section*{Easy Msg. active}

The connection has been established to the modem.

\section*{Procedure}

\section*{Settings}
1. Press the "Settings" softkey.
2. Enter an arbitrary name for the controller in the "Machine name" field.
3. If you want to limit the number of sent SMS messages then select the "Specify limit for SMS counter" entry and enter the desired number.
When the maximum number of messages is reached, you obtain a corresponding error message.

\section*{Note}

Check the SMS log to see the exact time when the limit was reached.
- OR -
3. Press the "Default" softkey.

If you have freely selected a machine name, this is replaced by a default name (e.g. 828D).

\section*{Easy Extend (828D only)}

\section*{22.1 \\ Overview}

Easy Extend enables machines to be retrofitted with additional units, which are controlled by the PLC or that require additional NC axes (such as bar loaders, swiveling tables or milling heads), at a later point in time. These additional devices are easily commissioned, activated, deactivated or tested with Easy Extend.

The communication between the operator component and the PLC is performed via a PLC user program. The sequences to be executed for the installation, activation, deactivation and testing of a device are stored in a statement script.

Available devices and device states are displayed in a list. The view of the available devices can be controlled for users according to their access rights.

The subsequent chapters are selected for example only and are not available in every statement list.

\section*{Machine manufacturer}

Please refer to the machine manufacturer's specifications.

Up to 64 devices can be managed.

\section*{References}

SINUMERIK 828D Turning and Milling Commissioning Manual

\subsection*{22.2 Enabling a device}

The available device options can be protected with a password.
Machine manufacturer
R Please refer to the machine manufacturer's specifications.

\section*{Procedure}


Parameter


Enable function

OK
1. Select the "Parameter" operating area.
2. Press the menu forward key and then the "Easy Extend" softkey. A list of the connected devices is displayed.
3. Press the "Enable function" softkey. The "Enabling of the Devices Option" window opens.
4. Enter the option code and press the "OK" softkey.

A tick appears in the appropriate checkbox in the "Function" column and the function is enabled.

\subsection*{22.3 Activating and deactivating a device}
\begin{tabular}{|l|l|}
\hline Status & Meaning \\
\hline 0 & Device activated \\
\hline D & System waiting for PLC checkback signal \\
\hline \(\mathbf{D}\) & Device faulty \\
\hline \(\mathbf{D}\) & Interface error in the communication module \\
\hline
\end{tabular}

Procedure
```

Easy-
Extend

```


\section*{Activate}

Deactivate
1. Easy Extend is opened.

2 You can select the desired device in the list with the <Cursor up> and <Cursor down> keys.
3. Position the cursor on the device option for which the function has been unlocked and press the "Activate" softkey.
The device is marked as activated and can now be used.
4. Select the desired activated device and press the "Deactivate" softkey to switch the device off again.

\subsection*{22.4 Initial commissioning of additional devices}

Normally, the device has already been commissioned by the machine manufacturer. If an initial commissioning has not been performed or if, for example, function tests are to be performed again (e.g. when retrofitting additional devices), this is possible at any time. The "Start-up" softkey has been declared as Manufacturer data class (M).

\section*{Procedure}


Parameter


Start-up

2. Press the menu forward key and then the "Easy Extend" softkey.
4. Press the "Comm. start-up" softkey to start the commissioning. Before starting, a complete data backup is generated which you can then use in case of an emergency.
5. Press the "Cancel" softkey if you want to abort the commissioning prematurely.
6. Press the "Restore" softkey to load the original data.
1. Select the "Parameter" operating area.
3. Press the "Start-up" softkey.

A new vertical softkey bar appears.
7. Press the "Device function test" softkey to test the machine manufacturer's intended function.

\section*{Service Planner (828D only)}

\subsection*{23.1 Performing and monitoring maintenance tasks}

With the "Service Planner", maintenance tasks have been set up that have to be performed at certain intervals (e.g. top up oil, change coolant).
A list is displayed of all the maintenance tasks that have been set up together with the time remaining until the end of the specified maintenance interval.

The current status can be seen in the status display.
Messages and alarms indicate when a task has to be performed.

\section*{Acknowledging a maintenance task}

Acknowledge the message when a maintenance task has been completed.

\section*{Note}

\section*{Protection level}

You require protection level 2 (service) to acknowledge completed maintenance tasks.

\section*{Service Planner}
\begin{tabular}{|l|l|}
\hline Display & Meaning \\
\hline Pos & Position of the maintenance task in the PLC interface. \\
\hline Maintenance task & Name of the maintenance task. \\
\hline Interval \([\mathrm{h}]\) & Maximum time until next servicing in hours. \\
\hline Remaining time \([\mathrm{h}]\) & Time until the interval expires in hours. \\
\hline Status & \multicolumn{3}{|l|}{\begin{tabular}{l} 
Display of the current status of a maintenance task. \\
The maintenance task has been started. \\
The maintenance task is completed. \\
The maintenance task is deactivated.
\end{tabular}} \\
\hline
\end{tabular}
23.1 Performing and monitoring maintenance tasks

\section*{Procedure}

D) Mainten

Maintenance
completed
1. Select the "Diagnostics" operating area.
2. Press the menu forward key and then the "Service planner" softkey. The window with the list of all the maintenance tasks that have been set up appears.
3. Perform the maintenance task when the maintenance interval has nearly expired or when prompted to do so by alarms or a warning.
4. After you have performed a pending maintenance task and the task is signaled as "Completed", position the cursor at the appropriate task and press the "Servicing performed" softkey.
A message is displayed confirming the acknowledgment, and the maintenance interval is restarted.

\section*{Note}

You can perform the maintenance tasks before the interval expires. The maintenance interval is restarted.

\subsection*{23.2 Set maintenance tasks}

You can make the following changes in the list of maintenance tasks in the configuration mode:
- Set up a maximum of 32 maintenance tasks with interval, initial warning and number of warnings to be acknowledged
- Change the interval, time of the initial warning and the number of warnings to be output
- Delete a maintenance task
- Reset the times of the maintenance tasks

\section*{Acknowledging a maintenance task}

You can acknowledge the maintenance tasks with the "Servicing performed" softkey.

\section*{Note}

\section*{Protection level}

You require protection level 1 (manufacturer) to set up and edit maintenance tasks.

\section*{Service Planner}
\begin{tabular}{|l|l|}
\hline Display & Meaning \\
\hline Pos & Position of the maintenance task in the PLC interface. \\
\hline Maintenance task & Name of the maintenance task. \\
\hline Interval \([\mathrm{h}]\) & Maximum time until next servicing in hours. \\
\hline 1. warning \([\mathrm{h}]\) & Time in hours at which an initial warning is displayed. \\
\hline Number of warnings & \begin{tabular}{l} 
Number of warnings that can be acknowledged by the operator before \\
an alarm message is output for the last time.
\end{tabular} \\
\hline Remaining time \([\mathrm{h}]\) & \begin{tabular}{l} 
Time until the interval expires in hours. \\
The remaining time cannot be edited.
\end{tabular} \\
\hline Status & \begin{tabular}{l} 
Display of the current status of a maintenance task. \\
The maintenance task has been started. \\
The maintenance task is completed. \\
The maintenance task is deactivated, i.e. the time has been \\
stopped.
\end{tabular} \\
\hline & The status cannot be edited. \\
\hline
\end{tabular}

\section*{Procedure}


Mainten

New
task
OK

Change task

Reset all

\author{
Delete task
}
1. Select the "Diagnostics" operating area.
2. Press the menu forward key and then the "Service planner" softkey. The window opens and displays a list of all the tasks that have been set up.
The values cannot be edited.
3. Press the "New maintenance task" softkey to set up a new maintenance task.
A message informs you that a new maintenance task will be set up at the next free position. Enter the required data in the columns and press the "OK" softkey.
- OR -

Position the cursor on the desired maintenance task and press the "Change task" softkey to change the associated times.
All columns except Remaining time and Status can be edited.
- OR -

Press the "Reset all" softkey to reset all the times.
- OR -

Position the cursor on the desired maintenance task and press the "Delete task" softkey to remove the maintenance task from the list.

\section*{Edit PLC user program (828D only)}

\section*{24.1 \\ Introduction}

A PLC user program consists to a large degree of logical operations to implement safety functions and to support process sequences. These logical operations include the linking of various contacts and relays. These logic operations are displayed in a ladder diagram.

You can edit these ladder diagrams using the following tools:
- Ladder add-on tool

Using the ladder add-on tool, you can track down program errors or troubleshoot faults and directly make small corrections or modifications.
- Ladder editor

You require the Ladder Editor to use all of the operations supported by the particular PLC type.

Software option
You require the "SINUMERIK 828 Ladder Editor" option to edit PLC user programs.

\subsection*{24.2 Ladder add-on tool}

The failure of a single input, output or relay normally results in the failure of the complete system.

Using the Ladder add-on tool, you can perform a PLC diagnosis in order to find fault causes or program errors. You can directly make small corrections or modifications.

\section*{Generating an INT_100/INT_101 block}

If an INT_100 or INT_101 block is missing, it can be added via the vertical softkey bar. If these INT blocks exist in a project, they can be deleted via the vertical softkey bar. You also have the opportunity to change the networks of interrupt routines on the control as well as to save and load these changes.

\section*{Editing the interrupt routines}

You can edit the following interrupt programs:
- INT_100 - interrupt program (is executed before the main program)
- INT_101 - interrupt program (is executed after the main program)

\section*{Marshalling data}

Using the Ladder add-on tool, you can "re-wire" inputs (via INT_100) or outputs (via INT_101) for service purposes.

\section*{Note}

\section*{Saving the PLC project when changing the operating area}

If you have created INT_100/INT_101 blocks or inserted, removed or edited networks in an INT block, you must save the project before you change from the PLC area into another operating area. Transfer the project into the PLC using the "Download to CPU" softkey. If this is not done, all of the changes will be lost and must be re-entered.

Please take the appropriate program note into consideration.

\section*{References}

The editing of the INT_100 and INT_101 interrupt programs can be enabled or disabled.
Related information is contained in the Base Functions function manual, Chapter P4: PLC for SINUMERIK 828D

\section*{See also}

Inserting and editing networks (Page 899)

\subsection*{24.3 Structure of the user interface}


Figure 24-1 Screen structure

Table 24-1 Key to screen layout
\begin{tabular}{|c|c|c|}
\hline Screen element & Display & Meaning \\
\hline 1 & \multicolumn{2}{|l|}{Application area} \\
\hline 2 & \multicolumn{2}{|l|}{Supported PLC program language} \\
\hline & * & Program change exists \\
\hline 3 & \multicolumn{2}{|l|}{\begin{tabular}{l}
Name of the active program block \\
Representation: Symbolic name (absolute name)
\end{tabular}} \\
\hline 4 & Program
Run Abs & \\
\hline & Run & Program is running \\
\hline & Stop & Program is stopped \\
\hline & \multicolumn{2}{|l|}{Status of the application area} \\
\hline & Sym & Symbolic representation \\
\hline & Abs & Absolute representation \\
\hline 5 & \multicolumn{2}{|l|}{\[
\hat{\rightharpoonup} \mid 0
\]} \\
\hline 6 & \multicolumn{2}{|l|}{\begin{tabular}{l}
Focus \\
Performs the tasks of the cursor
\end{tabular}} \\
\hline 7 & \multicolumn{2}{|l|}{Information line Displays information, e.g. for searching} \\
\hline
\end{tabular}

\subsection*{24.4 Control options}

In addition to the softkeys and the navigation keys, there are further shortcuts in this area.

\section*{Shortcuts}

The cursor keys move the focus over the PLC user program. When the window borders are reached, scrolling is performed automatically.

\begin{tabular}{|c|c|c|}
\hline \multicolumn{2}{|l|}{Shortcuts} & Action \\
\hline \multicolumn{2}{|l|}{-or-} & \\
\hline CTRL & \[
\nabla
\] & \\
\hline CTRL &  & Open the next program block in the same window \\
\hline CTRL & ( & Open the previous program block in the same window \\
\hline \[
\bigcup_{\text {select }}
\] & & \begin{tabular}{l}
The function of the Select key depends on the position of the input focus. \\
- Table line: Displays the complete text line \\
- Network title: Displays the network comment \\
- Command: Displays all operands
\end{tabular} \\
\hline \[
\overrightarrow{\text { INPUT }}
\] & & If the input focus is positioned on a command, all operands including the comments are displayed. \\
\hline
\end{tabular}

\subsection*{24.5 Displaying PLC properties}

The following PLC properties can be displayed in the "SIMATIC LAD" window:
- Operating state
- Name of the PLC project
- PLC system version
- Cycle time
- Machining time of the PLC user program

\section*{Procedure}


\section*{PLC \\ PLC}

\section*{PLC - CPU}

Reset pro. time
1. Select the "Start-up" operating area.
2. Press the "PLC" softkey.

The ladder diagram display opens and displays the PLC information.
3. Press the "Reset mach.time" softkey. The data of the machining time is reset.

\subsection*{24.6 Displaying information on the program blocks}

You can display all the logic and graphic information of a program block.

\section*{Display program block}

In the "Program block" list, select the program block that you want to display.

\section*{Logic information}

The following logic information is displayed in a ladder diagram (LAD):
- Networks with program parts and current paths
- Electrical current flow through a number of logical operations

\section*{Further information}
- Properties

Name of the block, author, number of the subprogram, data class, date it was generated, date of the last change and comment.
- Local variable

Name of the variable, variable type, data type and comment.

\section*{Access protection}

Protected
When a program block is protected by a password, then the "Protection" softkey can be used to enable display in the ladder diagram format.

\section*{Displaying the program status}

\section*{Program stat.}

\section*{Program} stat.
1. Press the "Program stat." softkey to hide the program status in the status display.
2. Press the "Program stat." softkey to display the program status display in the status display again.

\section*{Display progress status}

If your PLC has the function "Progress status" available, the status values are displayed at the time of execution of the operations. The status of the local data memory and the accumulators is also displayed.
The "Progress status" display is also controlled using the "Program stat." softkey

\section*{Colors used to display the progress status or program status}

In progress status, different colors are used to display information.
\begin{tabular}{ll} 
Display & Color \\
Signal flow of power rail, when status active & Blue \\
Signal flow in the networks & Blue \\
\begin{tabular}{l} 
All operations that are active and that are executed without \\
error (corresponds to signal flow)
\end{tabular} & Blue \\
\begin{tabular}{l} 
Status of the Boolean operations (corresponds to signal \\
flow)
\end{tabular} & Blue \\
Times and counters active & Green \\
Error during execution & Red \\
No signal flow & Gray \\
No network executed & Gray \\
STOP mode & Gray
\end{tabular}

\section*{Note}

\section*{Colored display in program status}

In the display of the program status only the color of the signal flow is relevant.

\section*{Enlarging/reducing the display of the ladder diagram}

Zoom +

Zoom -

\section*{PLC - GPU}

Window 1 \(0 B 1\) ...

\section*{Program} block

Properties
1. Press the "Zoom +" softkey to enlarge the section of the ladder diagram.
After enlarging, the "Zoom -" softkey is available.
2. Press the "Zoom -" softkey to reduce the section of the ladder diagram again.

\section*{Procedure}
1. Ladder add-on tool is opened.
2. Press the "Window 1" or "Window 2" softkey.
3. Press the "Program block" softkey.

The "Program block" list is displayed.
4. Press the "Properties" softkey if you wish to display additional information.
- OR -

Press the "Local variables" softkey if you wish to display data of a variable.

\subsection*{24.7 Displaying and editing NC/PLC variables}

The "NC/PLC Variables" window enables the monitoring and modification of NC system variables and PLC variables.

You receive the following list in which you enter the desired NC and PLC variables in order to display the actual values.
- Variable

Address for NC/PLC variable.
Faulty variables have a red background and \# is displayed in the Value column.
- Comment

Arbitrary comment on the variables.
The column can be shown or hidden.
- Format

Specification of the format in which the variable is to be displayed.
The format can be permanently defined (e.g. floating-point).
- Value

Display of the current value of the NC/PLC variable.

\section*{Procedure}

\section*{PLC - GPU}

NC/PLC status
1. Ladder add-on tool is opened.
2. Press the "NC/PLC variables" softkey.

The "NC/PLC Variables" window appears.

\subsection*{24.8 Loading modified PLC user program}

Download the project data into the PLC if some changes have been made to the project data and a new PLC user program is available.
When the project data is loaded, the data classes are saved and loaded to the PLC.

\section*{Requirement}

Check whether the PLC is in Stop mode.

\section*{Note}

\section*{PLC in RUN mode}

If the PLC is in RUN mode, a corresponding message is displayed and the "Load in Stop" and "Load in Run" softkeys appear.
With "Load in Stop", the PLC is set to Stop mode and the project is stored and loaded to the CPU.

With "Load in Run", the loading operation is continued and the PLC project loaded to the PLC. Only those data classes that have really been changed are loaded, i.e. generally INDIVIDUAL data classes.

\section*{Procedure}

\section*{PLC - GPU}

\section*{PLC STOP}

Loading in CPU

PLC START
1. Ladder add-on tool is opened.

You have changed project data.
2. Press the "PLC Stop" softkey if the PLC is in the run mode.
3. Press the "Load to CPU" softkey to start the loading operation. All data classes are loaded.
4. When the PLC project has been loaded, press the "PLC Start" softkey to switch the PLC to Run mode.

\subsection*{24.9 Displaying local variable table}

You have the option of displaying the local variable table of an INT block.
The following information is listed in the tables.
\begin{tabular}{ll} 
Name & Freely assign. \\
Variable type & Selection:
\end{tabular}
- IN
- IN_OUT
- OUT
- TEMP

Data type Selection:
- BOOL
- BYTE
- WORD
- INT
- DWORD
- DINT
- REAL

Comment Freely assign.

\section*{Procedure}

Window 1 1. The ladder diagram display (LAD) is opened.

\section*{Window 2}

SBRO
2. Press the "Program block" softkey.

Local variables
3. Press the "Local variables" softkey.

The "Local Variables" window appears and lists the created variables.

\subsection*{24.10 Creating a new block}

Create INT blocks to make changes in the PLC user program.
\(\left.\begin{array}{ll}\text { Name } & \text { INT_100, INT_101 } \\
\text { The number from the selection field "Number of subprogram" is taken } \\
\text { for the name of the INT block. }\end{array}\right\}\)\begin{tabular}{ll} 
A maximum of 48 characters is permitted. \\
Author & \\
\begin{tabular}{ll} 
Number of subpro- \\
gram
\end{tabular} & 100,101 \\
Data class & Individual \\
Comment & A maximum of 100 lines and 4096 characters are permitted.
\end{tabular}

\section*{Note}

Access protection
You have the option of protecting blocks that have been newly created against being accessed.

\section*{Procedure}

\section*{Window 1}

081

\section*{Window 2 \\ SBRO}

Program block

Add
2. Press the "Program block" softkey to open the list of program blocks.
3. Press the "Add" softkey.

The "Properties" window appears.
4. Enter author, number of the INT block and, if required, a comment. The data class of the block is specified.
5. Press the "OK" softkey to transfer the block to the list.

\subsection*{24.11 Editing block properties subsequently}

You can edit the title, author and comments of an INT block.

\section*{Note}

You cannot edit the block name, subprogram number and data class assignment.

\section*{Procedure}

Window 1
\(0 B 1\)
Window 2 SBRO

\section*{Program} block

Properties
1. The ladder diagram display is opened.
2. Select the relevant block and press the "Program block" softkey.
3. Press the "Properties" softkey.

The "Properties" window appears.

\subsection*{24.12 Inserting and editing networks}

You can create a new network and then insert operations (bit operation, assignment, etc.) at the selected cursor position.

Only empty networks can be edited. Networks, that already include statements, can only be deleted.

A simple, single line can be edited for each network. You can create a maximum of 3 columns per network.
\begin{tabular}{|c|c|c|}
\hline Column & Operation & \\
\hline Column 1 & \begin{tabular}{l}
- NO contact \\
- NC contact
\end{tabular} &  \\
\hline Column 2 (optional) & \begin{tabular}{l}
NOT \\
Rising edge \\
Falling edge \\
Assign \\
Set \\
Reset
\end{tabular} & \[
\begin{aligned}
& \hline-|\mathrm{NOT}|- \\
& -|\mathrm{P}|- \\
& -|\mathrm{N}|- \\
& -(\mathrm{O} \\
& -(\mathrm{S}) \\
& -(\mathrm{R}) \\
& \hline
\end{aligned}
\] \\
\hline \begin{tabular}{l}
Column 3 \\
(only possible if no assign, set or reset operations were specified in the second column)
\end{tabular} & \begin{tabular}{l}
Assign \\
Set \\
Reset
\end{tabular} & \[
\begin{aligned}
& \hline-() \\
& -(\mathrm{S}) \\
& -(\mathrm{R}) \\
& \hline
\end{aligned}
\] \\
\hline
\end{tabular}

\section*{Note}

Logical AND (serial contact) and logical OR (parallel contact) are not possible.

The bit combinations comprise one or several logical operations and the assignment to an output / bit memory.
If the cursor is moved further to the left with the arrow key, the type of assignment or a logic operation can be selected. A further logic operation cannot be placed to the right of an assignment. A network must always be terminated with an assignment.

\section*{References}

For further information about PLC programming, please refer to:
Function Manual Basic Functions, PLC for SINUMERIK 828D (P4)

\section*{Procedure}
1. An INT100 or INT101 routine has been selected.


Delete operation

Delete network
2. Press the "Edit" softkey.
3. Position the cursor on a network.
4. Press the "Insert network" softkey.
- OR -

Press the <INSERT> key.

If the cursor is positioned on "Network x", a new, empty network is inserted behind this network.
5. Position the cursor on the desired element below the network title and press the "Insert operation" softkey.
The "Insert Operation" window appears.
6. Select the desired bit operation (NC contact or NO contact) or assignment and press the "OK" softkey.
7. Press the "Insert operand" softkey.
8. Enter the logic operation or the command and press the <INPUT> key to complete the entry.
9. Position the cursor on the operation that you want to delete and press the "Delete operation" softkey.
- OR -

Position the cursor on the title of the network that you want to delete and press the "Delete network" softkey.
- OR -

Press the <DEL> key.

The network, including all the logic operations and operands, or the selected operation is deleted.

\subsection*{24.13 Editing network properties}

You can edit the network properties of an INT block.
Network title and network comment
The title can have a maximum of three lines and 128 characters. The comment can have a maximum of 100 lines and 4096 characters.

\section*{Procedure}

Window 1 0B1

Window 2 SBRO


Change

1. The ladder diagram display (LAD) is opened.
2. Use the cursor keys to select the network that you want to edit.
3. Press the <SELECT> key.

The "Network title / comment" window opens and shows the title and a possibly assigned comment for the selected network.
5. Press the "Change" softkey.

The fields can be edited.
6. Enter the changes and press the "OK" softkey to transfer the data to the user program.

\subsection*{24.14 Displaying/canceling the access protection}

You can password protect your program organizational units (POUs) in the PLC 828 programming tool. This prevents other users from accessing this part of the program. This means that it is invisible to other users and is encrypted when it is downloaded.

A lock symbol is used to show a password-protected POU in the block overview and in the ladder diagram.

\section*{Procedure}

Window 1 \(0 B 1\)

Window 2 SBRO

\section*{Program}
block

Protected
1. The ladder diagram display (LAD) is opened.

2 Select the relevant program organizational unit (POU) in the overview and press the "Program block" softkey.
3. Press the "Protection" softkey. The "Protection" window appears.

\section*{Removing protection}
4. Enter the password.
- "This program block remains protected" is activated:

You have the option of editing or deleting the block. Protection is reactivated when you load the PLC user program to the PLC.
- "This program block remains protected" is deactivated:

Block protection is permanently withdrawn. After being loaded to the PLC, the PLC user program is not protected.

\section*{Setting protection}
5. Enter the required password in the first line "Please enter password" and re-enter the password in the second line.
6. Activate the checkbox "Protect all program blocks using this password", if you wish to protect all of the user program blocks.
Note:
Program blocks that are already password-protected are not influenced.
4. Press the "Accept" softkey.

\subsection*{24.15 Displaying and editing symbol tables}

You can display the symbol tables that are used to obtain an overview of the global operands available in the project - which you can then edit.

The name, address and possibly also a comment is displayed for each entry.

\section*{Procedure}

\section*{PLC - GPU}

\section*{Symbol \\ table}

Select
sym table

\section*{Open}

4. Use the cursor keys to select the desired entry and the field to be changed.

5. Enter the value to be changed.
- OR -

Press the "Attach line" softkey to insert an empty line after the selected entry.
- OR -

Press the "Delete line" softkey to remove the selected entry from the list.
- OR -

Enter a new value in the selected field.
6. Press the "OK" softkey to confirm your action.

\subsection*{24.16 Searching for operands}

You can use the search function to quickly reach points in very large programs where you would like, for example, to make changes.

\section*{Restricting the search}
- "Window 1" / "Window 2","Symbol table"

With "Go to", you can jump directly to the desired network.
- "Cross references"

With "Go to", you can jump directly to the desired line.

\section*{Requirement}

Window 1 / window 2 , the symbol tables or the list of cross references is opened.

\section*{Procedure}

\section*{Search}

Continue search
\(\times\) Cancel
1. Press the "Find" softkey.

A new vertical softkey bar appears. The "Find / Go To" window opens at the same time.
2. Select the "Find operand" entry in the first input field if you are searching for a specific operand and enter the search term in the "Find" input field.
3. Select the search range (e.g. Find all).
4. Select the "In this program unit" or "In all program units" entry if you are in "Window 1" or "Window 2" or in the symbol table in order to restrict the search.
5. Press the "OK" softkey to start the search.

If the operand you are searching for is found, the corresponding line is highlighted.
Press the "Continue search" softkey if the operand found during the search does not correspond to the element you are looking for.
- OR -

Press the "Cancel" softkey if you want to cancel the search.

\section*{Further search options}

\section*{Goto}
start

Goto end
1. Press the "Go to start" softkey to jump to the start of the ladder diagram in window 1 or window 2 , or the list (cross references, symbol table).

\subsection*{24.17 Inserting/deleting a symbol table}

New user symbol tables can be generated and changed. Tables that are no longer used can be deleted.

\section*{Note}

\section*{Delete symbol table}

The "Delete" softkey is only available if a user symbol table has been selected.

\section*{Procedure}
```

Symbol
table

```
Select
sym table

Insert sym table


Change
 sym table

Delete
1. The symbol table is opened.
2. Press the "Select sym. table" softkey.

The "Symbol Table - Selection" window appears.
3. Position the cursor at the desired position and press the "Insert sym. table" softkey.
The "Create Symbol Table" window appears.
4 Enter a symbolic name and press the "OK" softkey.
The new user symbol table is inserted in the line after the cursor position.
- OR -

Select a symbol table and press the "Change symbol table" softkey if you wish to change the properties of the symbol table.
5. Position the cursor on the symbol table that you want to delete and press the "Delete" softkey.

\subsection*{24.18 Displaying the network symbol information table}

All of the symbolic identifiers used in the selected network are displayed in the "Network symbol information table" window.
The following information is listed:
- Names
- Absolute addresses
- Comments

The symbol information table remains empty for networks that do not contain any global symbols.

\section*{Procedure}

Window 1 \(0 B 1\) Window 2 SBRO

Symbol info
1. The ladder diagram display (LAD) is opened.
2. Select the desired network and press the "Symbol info" softkey. The "Network Symbol Information Table" window appears.

3. Use the cursor keys to move within the table.


\subsection*{24.19 Displaying and editing PLC signals}

PLC signals are displayed and can be changed here in the "PLC status list" window.
The following lists are shown
Inputs (IB)
Bit memories (MB)
Outputs (QB)
Variables (VB)
Data (DB)

\section*{Setting the address}

You can go directly to the desired PLC address to monitor the signals.
Editing
You can edit the data.

\section*{Procedure}

\section*{PLC - GPU}
\begin{tabular}{c}
\begin{tabular}{c} 
Status \\
list
\end{tabular} \\
\begin{tabular}{c} 
Set \\
address
\end{tabular} \\
\\
OK \\
\hline
\end{tabular}

\section*{Change}
OK
1. Ladder add-on tool is opened.
2. Press the "Status list" softkey.

The "Status List" window appears.
3. Press the "Set address" softkey.

The "Set Address" window appears.
4. Activate the desired address type (e.g. DB), enter the value and press the "OK" softkey.
The cursor jumps to the specified address.
5. Press the "Change" softkey.

The "RW" input field can be edited.
6. Enter the desired value and press the "OK" softkey.

\subsection*{24.20 Displaying cross references}

You can display all the operands used in the PLC user project and their use in the list of cross references.

This list indicates in which networks an input, output, bit memory etc. is used.
The list of cross references contains the following information:
- Block
- Address in the network
- Context (command ID)

\section*{Symbolic and absolute address}

You can choose between specification in absolute or symbolic address.
Elements for which there are no symbolic identifiers are automatically displayed with absolute identifiers.

\section*{Opening program blocks in the ladder diagram}

From the cross references, you have the option of going directly to the location in the program where the operand is used. The corresponding block is opened in window 1 or 2 and the cursor is set to the corresponding element.

\section*{Searching}

Using a specific search, you can go directly to the location that you wish to view in more detail:
- Search for operand
- Jump to sought line

\section*{Procedure}

\section*{PLC - GPI}

\section*{Cross}
refs.

\section*{Symbolic} address

Absolute address
1. Ladder add-on tool is opened.
2. Press the "Cross refs." softkey.

The list of cross references appears and the operands are displayed sorted according to absolute address.
3. Press the "Symbol. address" softkey. The list of operands is displayed sorted according to symbolic address.
4. Press the "Absolute address" softkey to return to the display showing the absolute addresses.

Open in window 1
Open in window 2

Search


Continue search
5. Select the desired cross reference and press the "Open in window 1" or "Open in window 2 " softkey.
The ladder diagram is opened, and the selected operand is marked.
6. Press the "Find" softkey.

The "Find / Go To" window appears.
7. Select "Find operand" or "Go to" and enter the sought element or the desired line and select the search order (e.g. search up).
8. Press the "OK" softkey to start the search.
9. If an an element is found that corresponds to the sought element, but is not at the appropriate position, press the "Continue search" softkey to find where the search term occurs next.

\section*{Appendix}

\section*{A. 1 \\ 840D sl documentation overview}


Turning
A. 1 840D s/ documentation overview

\section*{Index}

\section*{2}

2-channel grooving, 654
2-channel plunge turning, 654
2-channel stock removal, \(\underline{654}\)

\section*{A}

Actual-value display, 45
Adapter-transformed view, 718
Additional components Initial commissioning, 880
Alarm log
display, 787
Sorting, 789
Alarms
Displaying, 785
Sorting, 789
Aligning milling tool - CYCLE800
Function, 563
Parameter, 566
Aligning turning tool
Parameter, 565
Alignment angle
B axis, 824
Any file
Creating, 745
Approach/retract cycle
Parameter, 271
Approach/retraction cycle
ShopTurn program, 270
Arbitrary positions - CYCLE802
Function, 336, 337
Parameter, 341
Archive
Generate in the program manager, 770
generating in the system data, 771
Punched tape format, 770
read in archive from system data, 774
reading-in in the Program Manager, 773
Auxiliary functions
H functions, 168
M functions, 168
Axes
Defined increment, 114
Direct positioning, 116

Referencing, 68
Repositioning, 130
Traversing, 114
Variable increment, 115

\section*{B}

B axis
Alignment angle, 824
Approach/retraction, 828
Function, 821
Measure tool, 833
Milling, 825
Position pattern, 830
Swiveling, 826
Backing up
Data - in the Program Manager, 770
Data - via the system data, 771
Setup data, 776
Base offset, 90
Basic blocks, 126
Binary format, 770
Blank input
Function, 222
Parameter, 224, 628
Simultaneous recording, 207
Block
Search, 131
Searching - interruption point, 134
Searching - search pointer, 135
Block search
Mode, 137
Multi-channel view, 652
Program interruption, 134
Search pointer, 135
Search target parameters, 136
Search target specification, 133
Using, 131
Boring - CYCLE86
Function, 303
Parameter, 305

\section*{C}

Calculator
Fits, 400

Center drilling
Function, 574
Input simple, 574
Parameter - input complete, 577
Parameter - input simple, 577
Centering - CYCLE81
Function, 294
Parameter, 296
Channel switchover, 73
Chuck dimensions, 100
Circle, 588
Manual Machine, 812
Circle position pattern-HOLES2
Parameter, 349, 352
Circle/pitch circle position pattern - HOLES2
Function, 347
Circular pocket-POCKET4
Function, 444
Input simple, 445
Parameter - input complete, 451
Parameter - input simple, 451
Circular spigot - CYCLE77
Parameter - input simple, \(\underline{465}\)
Circular spigot - CYCLE77
Function, 461
Input simple, 462
Parameter - input complete, 465
Circumferential slot - SLOT2
Function, 482
Input simple, 483
Parameter - input complete, 487
Parameter - input simple, 487
Clamping the spindle, 247
Coarse and fine offsets, 90
Code carrier connection, 691
Collision avoidance, 665
Context-sensitive online help, 64
Contour call - CYCLE62
Function, 403, 525
Parameter, 404, 526
Contour elements
Representation, 391, 514
Contour pocket residual material - CYCLE63
Function, 544
Parameters, 546
Contour spigot residual material - CYCLE63
Function, 552
Parameters, 554
Contour turning
Manual machine, 818
Overview, 390

\section*{Contours}

Programming, 390
Coordinate system
Switching over, 74
Coordinate transformation, 581
Copying
Directory, 754
Program, 754
Counter-spindle, 100
Counterspindle, fixed
Parameter, 609
Counter-spindle, traversable
Parameter, 604
Creating
Any file, 745
Directory, 742
G code program, 744
Job list, 746
Multi-channel program, \(\underline{618}\)
Multitool, 722
NC directory on the local drive, 735
Program block, 150
Program list, 748
Workpiece, 743
Ctrl-Energy
Energy analysis, 858
Energy-saving profiles, 864
Functions, 857
Measuring the energy consumption, 860
Saved measuring curves, 863
Cut-off - CYCLE92
Function, 388
Parameter, 389
Cutting edge numbers, 681
Cutting edge width, 680
Cutting tip length, 680
Cutting tip width, 680
CYCLE60 - Engraving
Function, 507
CYCLE61 - Face milling
parameters, 434
CYCLE61- Face milling
Function, 431
CYCLE62- contour call
Function, 403, 525
Parameter, 404, 526
CYCLE63-Contour pocket residual material
Function, 544
Parameters, 546
CYCLE63 - Contour spigot residual material
Function, 552
Parameters, 554

CYCLE63 - Milling contour pocket
Function, 538
Input simple, 539
Parameter, 542
Parameter - input simple, 542
CYCLE63 - Milling contour spigot
Function, 547
Input simple, 547
Parameter - input complete, 549
Parameter - input simple, 550
CYCLE64 - Predrilling contour pocket
Function - Centering, 534
Function - Predrilling, 534
Parameters - Centering, 537
Parameters - Milling, 538
CYCLE70 - engraving
Parameter, 513
CYCLE70 - thread milling
Function, 503
Parameters, 505
CYCLE72 - Path milling
Function, 526
Parameter, 531
CYCLE76 - rectangular spigot
Function, 454, 455
Input simple, 455
Parameter - input complete, 459
Parameter - input simple, 459
CYCLE77 - circular spigot
Function, 461
Input simple, 462
Parameter - input complete, \(\mathbf{4 6 5}\)
Parameter - input simple, 465
CYCLE78 - Drill thread milling
Function, 331
Parameters, 334
CYCLE79 - multi-edge
Function, 468
Input simple, 468
Parameter - input complete, 471
Parameter - input simple, 471
CYCLE8 - deep-hole drilling
Function, 312
CYCLE800-Aligning milling tool
Function, 563
Parameter, 566
CYCLE800-Setting milling tool
Function, 567
Parameter, 568
CYCLE800-Swiveling
Function, 555
Parameter, 563

CYCLE801 - frame position pattern
Function, 343
CYCLE801- grid position pattern
Function, 343
CYCLE801 - grid/frame position pattern
Function, 341
Parameter, 343, 345, 346
CYCLE802 - arbitrary positions
Function, 336, 337
Parameter, \(\mathbf{3 4 1}\)
CYCLE81 - centering
Function, 294
Parameter, 296
CYCLE82 - drilling
Function, 296
Input simple, 296
Parameter - input complete, 299
Parameter - input simple, 299
CYCLE83 - deep-hole drilling
Function, 306
Input simple, 306
Parameter - input complete, 310
Parameter - input simple, 310
CYCLE830 - deep-hole drilling
Input simple, 312
CYCLE830 - deep-hole drilling 2
Parameter - input complete, 319
Parameter - input simple, 319
CYCLE832 - High Speed Settings
Function, 568
Parameters, 571
CYCLE84 - tapping
Parameter - input simple, 329
CYCLE84 - tapping without compensating chuck
Function, 322
Input simple, 323
Parameter - input complete, 328
CYCLE840-tapping
Parameter - input simple, 329
CYCLE840 - tapping with compensating chuck
Function, 322
Input simple, 323
Parameter - input complete, 328
CYCLE85 - reaming
Function, 301
Parameter, \(\underline{302}\)
CYCLE86-boring
Function, 303
Parameter, 305
CYCLE899 - Milling open slot
Parameter - input complete, \(\underline{498}\)

CYCLE899 - open slot
Function, 490
Input simple, 490
Parameter - input simple, 498
CYCLE92 - cut-off
Function, 388
Parameter, 389
CYCLE930 - groove
Function, 358
Parameter, 360
CYCLE940 - Undercut
Function - DIN thread, 363
Function - Form E, 361
Function - Form F, 361
Function - Thread, 363
Parameter - DIN thread, 364
Parameter - Form E, 362
Parameter - Form F, 363
Parameter - Thread, \(\underline{365}\)
CYCLE951 - stock removal
Function, 355
Parameter, 358
CYCLE952-Grooving
Function, 415
Input simple, 416
Parameter - input complete, \(\underline{418}\)
Parameter - input simple, 420
CYCLE952-Grooving residual material
Function, 420
Parameters, 422
CYCLE952 - Plunge turning
Function, 423
Input simple, 424
Parameter - input complete, 426
Parameter - input simple, 428
CYCLE952 - Plunge turning residual material
Function, 429
Parameters, 430
CYCLE952 - Stock removal
Function, 404
Input simple, \(\underline{406}\)
Parameter - input complete, 409
Parameter - input simple, 412
CYCLE952 - Stock removal residual material
Function, 412
Parameters, 414
CYCLE98 - thread chain
Parameter - input simple, 387
CYCLE98 - thread chain
Function, 382
Input simple, 383
Parameter - input complete, \(\underline{385}\)

CYCLE99 - Thread turning
Parameter - Input complete - longitudinal
thread, 370
Parameter - Input simple- face thread, 382
Parameter - Input simple- longitudinal thread, 373
Parameter - Input simple- tapered thread, 377
CYCLE99 - Thread turning
Function - Face thread, 366
Function - Longitudinal thread, \(\underline{366}\)
Function - Tapered thread, 366
Input simple, 366
CYCLE99 - Thread turning
Parameter - Input complete - tapered thread, \(\underline{375}\)
CYCLE99 - Thread turning
Parameter - Input complete - face thread, 380
Cycles
Current levels, 219
Hiding cycle parameters, 231
Screen forms, 219

\section*{D}

Data block (SB2), 125
Deep-hole drilling - CYCLE83
Function, 306
Input simple, 306
Parameter - input complete, 310
Parameter - input simple, 310
Deep-hole drilling - CYCLE830
Function, 312
Input simple, 312
Deep-hole drilling 2-CYCLE830
Parameter - input complete, 319
Parameter - input simple, 319
Delete
Multitool, 725
Deleting
Directory, 756
Program, 756
Device
Activate/deactivate, (See Easy Extend)
Enabling, (See Easy Extend)
Dictionary
Importing, 59
Dimension
Absolute, 245
Incremental, 245
Directory
Copying, 754
Creating, 742
Deleting, 756
Highlight, 752

Pasting, 754
Properties, 757
Selecting, 752
Displaying
Energy consumption, 858
HTML documents, 765
PDF documents, \(7 \overline{65}\)
Program level, 128
DRF (handwheel offset), 139
Drill thread milling - CYCLE78
Function, 331
Parameters, 334
Drilling
Manual Machine, 814
Drilling - CYCLE82
Function, 296
Input simple, 296
Parameter - input complete, 299
Parameter - input simple, 299
Drilling radius, 680
Drive
Logical drive, 758
Setting up, 759
DRY (dry run feed), 139
Dual editor, 152
Duplo number, (See replacement tool number)
DXF file
Cleaning, 182
close, 181
Open, 181
Reference point, 186
Tolerance, 187
DXF-Reader, 181
Editor, 181
Program manager, 181

\section*{\(E\)}

Easy Extend, 877
Activate/deactivate device, 879
Enabling a device, 878
Easy Message, 867
Commissioning, 869
Settings, 876
User log on/off, 874
Editor
Calling, 144
Settings, 153
EES
Creating drives, 759

Elongated hole - LONGHOLE
Function, 501
Parameter, 502
Enabling button, 850
Energy consumption
Displaying, 858
Measuring, 860
Energy-saving profiles, 864
Engraving - CYCLE60
Function, 507
Parameter, 513
Equipping tools
Multitool, 724
Executing Program, 740
EXTCALL call, 767

\section*{F}

Face milling - CYCLE61
Function, 431
Parameters, 434
Feed data
Actual value window, 48
Fit calculator Master dimension programming, \(\underline{400}\)
Frame position pattern - CYCLE801 Function, 343

\section*{G}

G code program
Blank input, 222
Creating, 744
G functions
Display all G groups, 166
G Functions
Displaying selected G groups, 164
GCC (G code converter), 139
Global user variables, 158
Graphic view ShopTurn program, 235
Grid position pattern - CYCLE801 Function, 343
Grid/frame position pattern - CYCLE801
Function, 341
Parameter, \(343,345,346\)
Groove - CYCLE930
Function, 358
Parameter, 360

Grooving - CYCLE952
Function, 415
Input simple, 416
Parameter - input complete, 418
Parameter - input simple, 420
Grooving residual material - CYCLE952
Function, 420
parameters, 422

\section*{H}

Handheld terminal 8, 849
Handwheel
Assigning, 103
High Speed Settings - CYCLE832
Function, 568
Parameters, 571
Highlight
Directory, 752
Program, 752
High-Speed Cutting, 166
HOLES1 - line position pattern
Function, 341
Parameter, 343, 345, 346
HOLES2 - circle position pattern
Parameter, 349, 352
HOLES2 - circle/pitch circle position pattern
Function, 347
HSC information, 166
HT 8
Enabling button, 850
Overview, 849
Touch Panel, 856
Traversing keys, 852
User menu, 853
Virtual keyboard, 854
HTML documents
Displaying, 765

\section*{I}

\section*{IME}

Chinese characters, 55
Korean characters, 6
Initial commissioning
Additional components, 880
Input simple
Center drilling, 574
Circular pocket - POCKET4, 445
Circular spigot - CYCLE77, 462
Circumferential slot - SLOT2, 483

CYCLE830 - deep-hole drilling, 312
CYCLE99 - Thread turning, 366
Deep-hole drilling - CYCLE83, 306
Drilling - CYCLE82, 296
Grooving - CYCLE952, 416
Longitudinal slot - SLOT1, 474
Milling contour pocket - CYCLE63, 539
Milling contour spigots - CYCLE63, 547
Multi-edge - CYCLE79, 468
Open slot - CYCLE899, 490
Plunge turning - CYCLE952, 424
Rectangular pocket - POCKET3, 435
Rectangular spigot - CYCLE76, 455
Stock removal - CYCLE952, \(40 \overline{6}\)
Tapping with compensating chuck -
CYCLE840, 323
Tapping without compensating chuck -
CYCLE84, 323
Thread chain - CYCLE98, 383
Interruption point
Approaching, 134

\section*{J}

Job list
Creating, 746
Multi-channel support, 625

\section*{L}

Ladder viewer, 885
Layer selection, 182
Limit stops, 807
Line position pattern - HOLES1
Function, 341
Parameter, 343, 345, 346
Loading
Multitool, 726
Logbook
Delete entries, 799
Displaying, 798
Edit the address data, 798
Entry search, 800
Making an entry, 799
Output, 797
Overview, 798
LONGHOLE - elongated hole
Function, 501
Parameter, 502
Longitudinal slot-SLOT1
Function, 473

Input simple, 474
Parameter - input complete, 479
Parameter - input simple, 479

\section*{M}

Machine control panel
Operator controls, 37
Machine model, 665
Machine-specific information
Save, 797
Machining
Canceling, 124
Starting, 123
Stopping, 123
Machining step program, 235
Magazine
Positioning, 704
Magazine list, 702
Magazine management, 670
Main spindle, 100
Maintenance tasks
Monitoring/performing, 881
Setting up, 883
Manual machine
Contour turning, 818
Zero/work offset, 806
Manual Machine, 805
Circle, 812
Drilling, 814
Manual mode, 808
Milling, 819
Set limit stop, 807
Simulating, 820
Single-cycle machining, 813
Straight line, 810
Traversing axes, 808
Turning, 816
Manual mode, 109
Circle, 812
Settings, 122
Straight line, 810
Traversing axes, 114
Manually
Retraction, 117
Master dimension
Fit calculator, 400
MDA
Deleting a program, 108
MDI
Executing a program, 107

Loading a program, 105
Saving a program, 106
Measurement
Tool, 686
Measurement log
Settings, 88
Measurement result log
Tool, 84
Workpiece zero, 87
Messages
Displaying, 788
Sorting, 789
Milling
B axis, 825
Manual Machine, 819
Milling contour pocket-CYCLE63
Function, 538
Input simple, 539
Parameter, 542
Parameter - input simple, 542
Milling contour spigots - CYCLE63
Function, 547
Input simple, 547
Parameter - input complete, 549
Parameter - input simple, 550
Milling open slot - CYCLE899
Parameter - input complete, 498
Mirroring
Parameters, 586
Mode groups, 72
Mold making
G functions, 166
Mold making view
Changing a graphic, 175
Changing the section, 176
Editing a program block, 174
Programs, 170
Searching for program blocks, 174
Starting, 173
MRD (Measuring Result Display), 139
Multi-channel data
Multi-channel support, 619
Multi-channel editor, 617
Synchronized view, 640
Time synchronous view, 644
Wait points, 644
Multi-channel program
Creating, 618
Editing a G code program, 626
Editing the ShopTurn program, \(\underline{628}\)
Multi-channel support
Changing the job list, 625

Defining multi-channel data, 619
Multi-channel view, 617
Program blocks, 636
Running-in a program, 651
Simulation, 648
Multi-channel view, 611
"Machine" operating area, 611
Block search, 652
Multi-channel support, 617
Program control, 652
Settings, 616, 639
Multi-edge - CYCLE79
Function, 468
Input simple, 468
Parameter - input complete, 471
Parameter - input simple, 471
Multitool, 720
Creating, 722
Delete, 725
Equipping tools, 724
Loading, 726
Parameters in the tool list, 721
Positioning, 730
Reactivating, 727
Relocating, 729
Removing tool, 725
Unloading, 726

\section*{N}

NC directory
creating on local drive, 735
NC/PLC variables
Changing, 793
Displaying, 791
New contour
Function - Milling, 516
Function - Turning, 392
Parameter - Turning, 394
Parameters - Milling, \(\overline{518}\)
Number of teeth, 680

\section*{0}

Offset, 583
Online help
Context-sensitive, 64
Open
DXF file, 181
Program, 738
Second program, 152

Open slot - CYCLE899
Function, 490
Input simple, 490
Parameter - input simple, 498
Operating area
Changing, 49
Operating mode
AUTO, 71
Changing, 49
JOG, 70, 109
MDI, 71
REP \(\overline{O S}, 71\)
TEACH \(\overline{\mathrm{IN}, 72}\)
Operator panel fronts, 26

\section*{P}

Parameter
Calculating, 52
Changing, 52
Counterspindle, fixed, 609
Counter-spindle, traversable, \(\underline{604}\)
Entering, 51
Pasting
Directories, 754
Program, 754
Path milling - CYCLE72
Function, 526
Parameter, 531
Path motion, 588
PDF documents
Displaying, 765
PLC diagnostics
Ladder add-on tool, 885
Plunge turning - CYCLE952
Function, 423
Input simple, 424
Parameter - input complete, 426
Parameter - input simple, 428
Plunge turning residual material - CYCLE952
Function, 429
Parameters, 430
POCKET3 - rectangular pocket
Function, 435
Input simple, 435
Parameter - input complete, 441
Parameter - input simple, 441
POCKET4 - circular pocket
Function, 444
Input simple, 445
Parameter - input complete, 451
Parameter - input simple, 451

Polar coordinates
Overview, 246
Positioning
Magazine, 704
Multitool, 730
Positions
display, 352
Hiding, 352
Predrilling contour pocket - CYCLE64
Function - Centering, 534
Function - Predrilling, 534
Parameters - Centering, 537
Parameters - Predrilling, \(\underline{538}\)
Preview
Program, 751
Program
Closing, 738
Copying, 754
Creating with cycle support, 221
Deleting, 756
Executing, 740
Highlight, 752
Open, 738
Opening a second program, 152
Pasting, 754
Preview, 751
Properties, 757
Selecting, 752
Program block
Changing, 259
Copying and inserting, 148
Current, 49, 126
Delete, 148
linked, 240
Numbering, 149
Repeat, 257
Replacing, 253
Search, 145
Selecting, 148
Structure, 240
Program blocks, 150
Multi-channel support, 636
Numbering, 148
Program control
Modes of operation, 139
Multi-channel view, \(\underline{652}\)
Program editing, 129
Program header, 250
Program level
Displaying, 128
Program list
Creating, 748

Program Manager, 731
Searching directories and files, 750
Program runtime, 177
Program settings
changing, 260
Parameters, 261
Program views G code program, 215 ShopTurn program, \(\underline{235}\)
Programmed stop 1, 139
Programmed stop 2, 139
Programming variables, \(\underline{232,268}\)
Programs
Correcting, 129
Editing, 144
Managing, 731
Renumbering blocks, 149
Replacing text, 146
Running-in, 125
Searching for a program position, 145
Selecting, 124
Teach-in, 837
Properties
Directory, 757
Program, 757
Protection levels
Softkeys, \(\underline{62}\)
PRT (no axis motion), 139

\section*{Q}

Quantity, 258

\section*{R}

R parameters, 157
Reactivating
Multitool, 727
Tool, 699
Reading in
Setup data, 779
Reaming - CYCLE85
Function, 301
Parameter, 302
Rectangular pocket - POCKET3
Function, 435
Input simple, 435
Parameter - input complete, 441
Parameter - input simple, 441
Rectangular spigot - CYCLE76
Parameter - input simple, \(\underline{459}\)

Rectangular spigot - CYCLE76
Function, 454, 455
Input simple, 455
Parameter - input complete, \(\underline{459}\)
Reference, 68
Relocating
Multitool, 729
Tool, 704
Remote access
permit, 803
Setting, 801
Remote diagnostics, 801
Exit, 804
requesting, 803
Removing tool
Multitool, 725
Repeat positions
Function, 354
Parameter, 354
Replacement tool number, 679
Repositioning, 130
Retract
Retraction, 117

\section*{Retraction}

Manually, 117
RG0 (reduced rapid traverse), 139
Rotation, 584
Running-in
Multi-channel program, 651

\section*{S}

Save
Setup data, 776
SB (single blocks), 139
SB1, 125
SB2, 125
SB3, 125
Screenshots
Copy, 790
Creating, 790
Open, 790
Search
in the Program Manager, 750
Search mode, 137
Search pointer, 135
Searching
Logbook entry, 800
Selecting
Directory, 752
Program, 752
Service Planner, 881

Setting actual values, (See setting zero offsets)
Setting milling tool - CYCLE800
Function, 567
Parameter, 568
Settings
Editor, 153
For automatic operation, 179
For manual operation, 122
Measurement log, 88
Multi-channel function, 639
Multi-channel view, 616
Teach-in, 848
Tool lists, 718
Setup data
Backing up, 776
reading in, 779
ShopTurn program
Approach/retraction cycle, 270
Circle with known center point, \(\underline{593}\)
Circle with known radius, 595
Coordinate transformation, \(\underline{581}\)
Creating, 248
Cutting edge (D), 254
Cutting rate (V), \(\underline{255}\)
Entering the quantity of workpieces, \(\underline{258}\)
Feedrate (F), 255
Machining cycle, approach/retraction, 243
Machining planes, 241
Machining type, 256
Machining with counter-spindle, 601
Mirroring, 586
Polar coordinates, 597
Program blocks, 253
Program header, 250
Program settings, 260
Radius compensation, 254
Rotation C axis, 587
Scaling, 585
Spindle speed (S), 255
Straight and circle, 588
Straight line, 591
Straight line polar, 598
Structure, 240
Tool (T), 254
Simulation
Alarm display, 214
Changing a graphic, 210
Manual Machine, 820
Multi-channel support, 648
Program control, 208
Showing and hiding the path display, 208
Views for multi-channel support, 650

Simultaneous recording, 197
Blank input, 207
Single block
Coarse (SB1), 125
Fine (SB3), 125
Skip blocks, 140
SKP (skip blocks), 139
SLOT1 - longitudinal slot
Function, 473
Input simple, 474
Parameter - input complete, \(\mathbf{4 7 9}\)
Parameter - input simple, \(47 \overline{9}\)
SLOT2 - circumferential slot
Function, 482
Input simple, 483
Parameter - input complete, 487
Parameter - input simple, 487
SMS messages, 867
Log, 875
Special characters, 26
Spindle chuck data
Parameter, 101
Store chuck dimensions, 100
Spindle data
Actual value window, 48
Spindle speed limitation, \(\underline{99}\)
Status display, 42
Stock removal
in JOG, 118
Parameters, 119
Stock removal - CYCLE951
Function, 355
Parameter, 358
Stock removal - CYCLE952
Function, 404
Input simple, 406
Parameter - input complete, 409
Parameter - input simple, 412
Stock removal residual material - CYCLE952
Function, 412
Parameters, \(\underline{414}\)
Straight line, 588
Manual Machine, 810
Submode
REF POINT, 71
Subprogram
Parameters, 573
Switching off, 67
Switching on, 67
Switching over
Channel, 73

Coordinate system, 74
Unit of measurement, 75
Swiveling - CYCLE800
Function, 555
Parameter, 563
Symbol tables, 903
Synchronized actions
Displaying status, 168
Synchronized view
Multi-channel editor, 640
Synchronizing
Thread, 120
Synchronizing a counter-spindle
Function, 659
Multi-channel support, \(6 \underline{69}\)
Parameters, 659

\section*{T}

Tailstock, 101
Tapping-CYCLE84
Parameter - input simple, \(\underline{329}\)
Tapping-CYCLE840
Parameter - input simple, 329
Tapping with compensating chuck - CYCLE840
Function, 322
Input simple, 323
Parameter - input complete, 328
Tapping without compensating chuck - CYCLE84
Function, 322
Input simple, 323
Parameter - input complete, \(\underline{328}\)
Teach-in, 837
Changing blocks, 845
Circle intermediate position CIP, 843
Continuous-path mode, 839
Deleting blocks, 847
General sequence, 837
Inserting a position, 838
Inserting blocks, 841
Motion type, 839
Parameter, 839
Rapid traverse G0, 842
Selecting a block, 846
Settings, 848
Traversing block G1, 842
Templates
Creating, 749
Storage locations, 749
Thread
Synchronizing, 120

Thread centered
Function, 578
Parameters, 580
Thread chain - CYCLE98
Function, 382
Input simple, 383
Parameter - input complete, 385
Parameter - input simple, \(\underline{387}\)
Thread milling - CYCLE70
Function, 503
Parameters, 505
Thread turning - CYCLE99
Function - Face thread, 366
Function - Longitudinal thread, 366
Function - Tapered thread, 366
Input simple, 366
Parameter - Input complete - face thread, 380
Parameter - Input complete - longitudinal thread, 370
Parameter - Input complete - tapered thread, 375
Parameter - Input simple- face thread, 382
Parameter - Input simple- longitudinal thread, 373
Parameter - Input simple- tapered thread, 377
Tip angle, 680
Tool
Change type, 717
Changing the cutting edge position, 717
Delete, 687
Details, \(\overline{707}\)
Dimensioning, 674
Loading, 688
Measure manually, 78
Measurement, 686
Measurement result log, 84
Measuring, 78
measuring using a magnifying glass, 83
Reactivating, 699
Relocating, 704
Unloading, \(\underline{688}\)
Tool data
Actual value window, 47
Backing up, 776
reading in, 779
Tool list, \(\underline{679}\)
Tool lists
Settings, 718
Tool management, 669
List filtering, 713
Sorting lists, 712
Tool parameters, \(\underline{674}\)
Tool types, \(\underline{671}\)

Touch Panel
Calibrating, 856
Transformed view, 718
Turning
Manual Machine, 816

\section*{U}

Undercut - CYCLE940
Function - DIN thread, 363
Function - Form E, 361
Function - Form F, \(\overline{361}\)
Function - Thread, 363
Parameter - DIN thread, 364
Parameter - Form E, 362
Parameter - Form F, \(\overline{363}\)
Parameter - Thread, \(\underline{365}\)
Unique cutting edge numbers
Unique, 681
Unit of measurement
Switching over, 75
Unloading
Multitool, 726
User agreement, 69
User data, 156
User variables
Activating, 162
Channel GUD, 159
Defining, 162
Global GUD, 158, 162
Local LUD, 160
Program PUD, 161
R parameters, 157
Searching, 162

\section*{V}

Variable screen forms, 795
Virtual keyboard, 854

\section*{W}

Wear list, 696
Work offset list
Manual Machine, 806
Work offsets
Calling, 256
Delete, 97
Overview, 92
Work plan
ShopTurn program, 236

Working area limitation
Defining, 98
Workpiece
Creating, 743
Workpiece counter, 177
Workpiece zero
Measurement result log, 87

\section*{Z}

Zero offsets
Active ZO, 91
Displaying details, 95
Overview, 90
Settable ZO, 94
Setting, 76
Zero point
DXF file, 186
Zero point settings
Backing up, 776
reading in, 779```

