SIEMENS

SINUMERIK

SINUMERIK
Analyze MyWorkpiece /Toolpath

Operating Manual

Version 2.8

06/2023
A5E48913345B AH

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 ® 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.

Table of contents

Introduction 1

1.1 About SINUMERIK

From simple, standardized CNC machines to premium modular machine designs – the SINUMERIK CNCs offer the right solution for all machine concepts. Whether for individual parts or mass production, simple or complex workpieces – SINUMERIK is the highly dynamic automation solution, integrated for all areas of production. From prototype construction and tool design to mold making, all the way to large-scale series production.

Visit our website for more information SINUMERIK (https://www.siemens.com/sinumerik).

1.2 About this documentation

Target group

This documentation is intended for the user of Analyze MyWorkpiece /Toolpath.

Benefits

The Operating Manual helps users familiarize themselves with the control elements and commands. It enables users to work with Analyze MyWorkpiece /Toolpath.

Standard scope

This documentation only describes the functionality of the standard version. This may differ from the scope of the functionality of the system that is actually supplied. Please refer to the ordering documentation only for the functionality of the supplied drive system.

It may be possible to execute other functions in the system which are not described in this documentation. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

For reasons of clarity, this documentation cannot include all of the detailed information on all product types. Further, this documentation cannot take into consideration every conceivable type of installation, operation and service/maintenance.

The machine manufacturer must document any additions or modifications they make to the product themselves.

Websites of third-party companies

This document may contain hyperlinks to third-party websites. Siemens is not responsible for and shall not be liable for these websites and their content. Siemens has no control over the information which appears on these websites and is not responsible for the content and information provided there. The user bears the risk for their use.

1.3 Documentation on the internet

1.3.1 Documentation overview SINUMERIK 840D sl

You will find extensive documentation on the functions of SINUMERIK 840D sl from version 4.8 SP4 at 840D sl documentation overview (https://support.industry.siemens.com/cs/ww/en/view/109766213).

You can display documents or download them in PDF and HTML5 format.

The documentation is divided into the following categories:

- User: Operating
- User: Programming
- Manufacturer/Service: Functions
- Manufacturer/Service: Hardware
- Manufacturer/Service: Configuration/Setup
- Manufacturer/Service: Safety Integrated
- Manufacturer/Service: SINUMERIK Integrate/MindApp
- Information and training
- Manufacturer/Service: SINAMICS

1.3.2 Documentation overview SINUMERIK operator components

Comprehensive documentation about the SINUMERIK operator components is provided in the Documentation overview SINUMERIK operator components (https://support.industry.siemens.com/cs/document/109783841/technische-dokumentation-zu-sinumerik-bedienkomponenten?dti=0&lc=en-WW).

You can display documents or download them in PDF and HTML5 format.

The documentation is divided into the following categories:

- Operator Panels
- Machine control panels
- Machine Pushbutton Panel
- Handheld Unit/Mini handheld devices
- Further operator components

An overview of the most important documents, entries and links to SINUMERIK is provided at SINUMERIK Overview - Topic Page (https://support.industry.siemens.com/cs/document/109766201/sinumerik-an-overview-of-the-most-important-documents-and-links?dti=0&lc=en-WW).

1.4 Feedback on the technical documentation
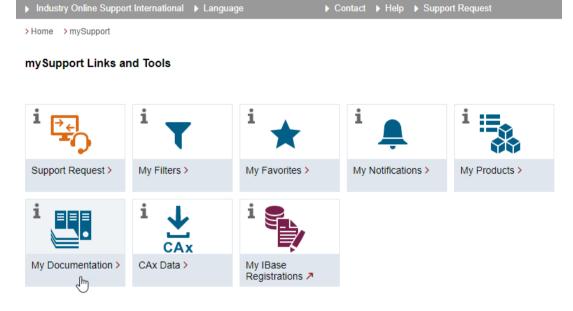
If you have any questions, suggestions or corrections regarding the technical documentation which is published in the Siemens Industry Online Support, use the link "Send feedback" link which appears at the end of the entry.

1.5 mySupport documentation

With the "mySupport documentation" web-based system you can compile your own individual documentation based on Siemens content, and adapt it for your own machine documentation.

To start the application, click on the "My Documentation" tile on the mySupport homepage (https://support.industry.siemens.com/cs/ww/en/my):

The configured manual can be exported in RTF, PDF or XML format.

Note

Siemens content that supports the mySupport documentation application can be identified by the presence of the "Configure" link.

1.6 Service and Support

Product support

You can find more information about products on the internet:

Product support (https://support.industry.siemens.com/cs/ww/en/)

The following is provided at this address:

- Up-to-date product information (product announcements)
- FAQs (frequently asked questions)
- Manuals
- Downloads
- Newsletters with the latest information about your products
- Global forum for information and best practice sharing between users and specialists
- Local contact persons via our Contacts at Siemens database (→ "Contact")
- Information about field services, repairs, spare parts, and much more (→ "Field Service")

Technical support

Country-specific telephone numbers for technical support are provided on the internet at address (https://support.industry.siemens.com/cs/ww/en/sc/4868) in the "Contact" area.

If you have any technical questions, please use the online form in the "Support Request" area.

Training

You can find information on SITRAIN at the following address (https://www.siemens.com/sitrain).
SITRAIN offers training courses for automation and drives products, systems and solutions from Siemens.

Siemens support on the go

With the award-winning "Siemens Industry Online Support" app, you can access more than 300,000 documents for Siemens Industry products – any time and from anywhere. The app can support you in areas including:

- Resolving problems when implementing a project

- Troubleshooting when faults develop

- Expanding a system or planning a new system

Furthermore, you have access to the Technical Forum and other articles from our experts:

- FAQs

- Application examples

- Manuals

- Certificates

- Product announcements and much more

The "Siemens Industry Online Support" app is available for Apple iOS and Android.

Data matrix code on the nameplate

The data matrix code on the nameplate contains the specific device data. This code can be read with a smartphone and technical information about the device displayed via the "Industry Online Support" mobile app.

1.7 Important product information

Using OpenSSL

This product can contain the following software:

- Software developed by the OpenSSL project for use in the OpenSSL toolkit

- Cryptographic software created by Eric Young.

- Software developed by Eric Young

You can find more information on the internet:

- OpenSSL (https://www.openssl.org)

- Cryptsoft (https://www.cryptsoft.com)

Compliance with the General Data Protection Regulation

Siemens observes standard data protection principles, in particular the data minimization rules (privacy by design).

For this product, this means:

The product does not process or store any personal data, only technical function data (e.g. time stamps). If the user links this data with other data (e.g. shift plans) or if he/she stores person-related data on the same data medium (e.g. hard disk), thus personalizing this data, he/she must ensure compliance with the applicable data protection stipulations.

Safety instructions

<div align="right" style="font-size:2em">2</div>

2.1 Fundamental safety instructions

2.1.1 General safety instructions

> ⚠ **WARNING**
>
> **Danger to life if the safety instructions and residual risks are not observed**
>
> If the safety instructions and residual risks in the associated hardware documentation are not observed, 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**
>
> **Malfunctions of the machine as a result of incorrect or changed parameter settings**
>
> As a result of incorrect or changed parameterization, machines can malfunction, which in turn can lead to injuries or death.
> - Protect the parameterization against unauthorized access.
> - Handle possible malfunctions by taking suitable measures, e.g. emergency stop or emergency off.

2.1.2 Warranty and liability for application examples

Application examples are not binding and do not claim to be complete regarding configuration, equipment or any eventuality which may arise. Application examples do not represent specific customer solutions, but are only intended to provide support for typical tasks.

As the user you yourself are responsible for ensuring that the products described are operated correctly. Application examples do not relieve you of your responsibility for safe handling when using, installing, operating and maintaining the equipment.

2.1.3 Security information

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, systems, machines and networks.

In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement – and continuously maintain – a holistic, state-of-the-art industrial security concept. Siemens' products and solutions constitute one element of such a concept.

Customers are responsible for preventing unauthorized access to their plants, systems, machines and networks. Such systems, machines and components should only be connected to an enterprise network or the internet if and to the extent such a connection is necessary and only when appropriate security measures (e.g. firewalls and/or network segmentation) are in place.

For additional information on industrial security measures that may be implemented, please visit
https://www.siemens.com/industrialsecurity.

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends that product updates are applied as soon as they are available and that the latest product versions are used. Use of product versions that are no longer supported, and failure to apply the latest updates may increase customer's exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed under
https://www.siemens.com/cert.

Further information is provided on the Internet:

Industrial Security Configuration Manual (https://support.industry.siemens.com/cs/ww/en/view/108862708)

⚠ **WARNING**

Unsafe operating states resulting from software manipulation

Software manipulations, e.g. viruses, Trojans, or worms, can cause unsafe operating states in your system that may lead to death, serious injury, and property damage.

- Keep the software up to date.
- Incorporate the automation and drive components into a holistic, state-of-the-art industrial security concept for the installation or machine.
- Make sure that you include all installed products into the holistic industrial security concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.
- On completion of commissioning, check all security-related settings.

2.2 Observe data security regulations

In addition to the recommendations for the Security information (Page 13), further points relating to data security and data protection must be observed.

Passing on data

When passing on recorded data to third parties or transferring data to other devices, observe the following:

- Use encrypted e-mails to transfer data.

- When forwarding files via e-mail, encrypt the data again by yourself (password protection).

Restrict the data access

Restrict the access to recorded data:

- Only grant authorized users access to servers or computers.

- Ensure safe handling of passwords, logins and access rights.

- Save data on encrypted data storage media only (hard disk encryption).

- Only use encrypted USB flash drives.

- Sign the recorded data.

Overview

<div style="text-align: right; font-size: 3em;">3</div>

3.1 Analyze MyWorkpiece /Toolpath (AMWT)

Overview

Analyze MyWorkpiece /Toolpath (AMWT) is a software for the visualization and analysis of NC programs. Application areas include turning and milling machines in the industry sectors mold making or aeronautics that demand ultraprecise surface machining.

Whereby, the AMWT allows contour accuracy, surface quality and machining speed of a workpiece to be optimized before production.

If problems occur during the production of a workpiece, the AMWT helps to rectify faults by analyzing the program and the surface.

3.2 Licensing with FlexLM

FlexLM License Manager allows you to manage licenses for Analyze MyWorkpiece /Toolpath. Depending on whether you have purchased multiple licenses and want to use them on the network or only want to use one license locally, FlexLM allows you to manage the licenses:

- Use the license server
  Configure the server to manage floating licenses.

- License file (node-locked)
  Copy the license file to your computer/server.

Licensing model

Two different licenses are available for Analyze MyWorkpiece /Toolpath:

- Professional (pro) - with this license, all functions of Analyze MyWorkpiece /Toolpath are available without restriction.

- Standard - with this license, only selected functions are available. Functions that are not available are grayed out.

Feature	Standard	Professional
4D point cloud	✔	✔
STL visualization	✔	✔
Transformation	✔	✔
Tool visualization	✔	✔
G code processing (4D coloring, filtering, etc.)	✔	✔
Manual measurement	-	✔
Data import	G code + DES_POS (EDGE)	✔

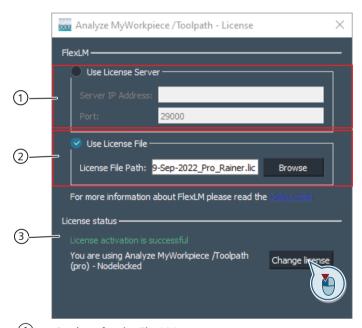| Feature	Standard	Professional
Time series view	2 graphs	✔
Surface reconstruction (virtual milling)	-	✔
Number of visual scenes	1	Unlimited
Number of models	2 per scene	Unlimited
Python scripting interface	-	✔
Teamcenter interface	-	✔
Vector visualization	Curvature + orientation	✔

Calling the license dialog

The following options are available for licensing:

- FlexLM; allows licenses to be stored in the network (floating licenses) or also locally.

Opening the license dialog

Run "Help > License" to open the dialog with the same name.

①   Settings for the FlexLM server
②   Path to the license file
③   License status

Configuring the license server

The license server provides the licenses you have purchased. The number of licenses determines how any users can work with Analyze MyWorkpiece /Toolpath at the same time. If a user closes the application, the license is released again (floating licenses).

1. Under "Help > License", select the type of licensing you want to use:

   – Under FlexLM, activate the "Use License Server" option to use floating licenses.

   – Under "Server IP Address", enter the IP address of the FlexLM server.

   – Enter the port number for the FlexLM service (default 29000).
   Note that this port must be enabled in the firewall of the server in order for Analyze MyWorkpiece /Toolpath to communicate with the license server.

2. Click on "Activate license" to apply the selected licensing type. The button is not active if a valid license was already assigned.
   The type of license being used is displayed to the left of the button.

3. Click on "Change license" to apply the new license.
   The already active license is still used if the new license is not valid.

Note

Managing licenses

LMTOOLS is provided for managing licenses. This software enables you to manage your licenses on the server or perform diagnostics, if necessary. This software is not part of the Analyze MyWorkpiece /Toolpath software package, but must be downloaded from the manufacturer (FLEXERA).

Using a license file (standalone license) (node-locked)

If you want to use only one license, you can store the corresponding license file locally on your computer or in a network directory:

1. Save the license file in the designated folder.

2. Activate option "Use license file".
   The path input field is activated.

3. Click the "Browse" button and navigate to the folder where the license file is located.

4. Click on the license file and then on "Select".
   The path to the license file is displayed and Analyze MyWorkpiece /Toolpath can be used.

5. Click on "Change license" to complete licensing.

3.3 Using the Shareware mode

License check

At startup, Analyze MyWorkpiece /Toolpath checks if a license is available:

- An error message is generated if no license is found.
- The "License" dialog is opened using function "Activate license", which allows you to setup a license via FlexLM.

Working without a license

A shareware version of the software is available for testing Analyze MyWorkpiece /Toolpath.

The demo version is subject to the following restrictions:

- The title of the software contains a supplement after "shareware version".
- You can only edit demo files that were supplied with the demo version. The "Sample Files" dialog only permits access to these files. You cannot open your own files.
- Apart from free file selection, most of the functions of the full version are available.
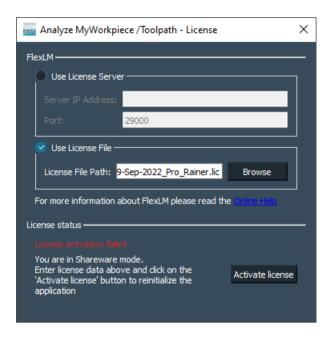
Note

Settings not active

When working without license, the settings under "Tools > Settings" are not active with the exception of the "License" tab.

FlexLM server

If Analyze MyWorkpiece /Toolpath cannot connect to the FlexLM server or if no valid FlexLM license file is found, it will start in "shareware mode" (Licensing with FlexLM (Page 17)).

3.4 Recommended hardware and software configuration

The following hardware is recommended to operate Analyze MyWorkpiece /Toolpath:

Hardware configuration

CPU: Intel Core i5 Processor @ 2.8 GHz or higher with 64-bit support (AMD not tested)
GPU: nVidia GeForce GT 710 or higher (AMD not tested)
hard disk memory: 20 GB free hard disk memory (can increase depending on the use)
Monitor resolution: 1920x1080
Internet: required for software activation, FlexLM license use and Capture connection

Operating system

Operating system: Windows 10 with the latest service pack

Security requirements

From the user side, it is not necessary to configure Analyze MyWorkpiece /Toolpath.

Analyze MyWorkpiece /Toolpath is ready for use after the compressed file has been unzipped.

You must first set up license management, or this is already available.
Account management is not used.

Reset

The application uses the AppData folder of the user for logging and to save configuration files or temporary files. The registration is used for setting data. Both can be deleted if script `reset.ps1` in the user folder is run.

Uninstalling

Uninstallation is possible if you run script `uninstall.ps1` in the user folder. This script also calls reset.

Working with the AMWT

<div style="text-align: right; font-size: 3em; font-weight: bold;">4</div>

4.1 Starting

Starting - exiting Analyze MyWorkpiece /Toolpath

To start AMWT, proceed as follows:

1. Open the directory in which Analyze MyWorkpiece /Toolpath is located.

2. Double-click on "amwt_desktop.exe". AMWT is opened with the settings last used.

3. To exit the application, in menu "File", select command "Exit" or click on button "x".

Note

Start screen

The start of the program may be slightly delayed if the connection to the license server is slow. A start screen is displayed during startup. More detailed information is available at Licensing with FlexLM (Page 17).

4.2 Using workspaces

4.2.1 Creating/loading a workspace

In AMWT, the data of the current machining is collected in a workspace. All geometric data of the model and the current processing steps are saved in the workspace. The file format is *.amwt

As soon as a file is imported, AMWT creates an implicit workspace.

Note

Not saved in a scene

The settings of the following functions are not saved in a scene:

- Surface extraction (in the case of a temporary file that has not been saved)
- Graphs
- Offset

Creating a new workspace

Proceed as follows to create a new workspace:

1. Open AMWT.

2. Perform "File > New workspace".
   The currently open workspace is closed. The unsaved changes can be rejected or saved.
   The created workspace can be saved as follows.

Loading a workspace

Proceed as follows to open an already existing workspace:

1. Open AMWT.

2. Run "File > Load Workspace".
   A prompt opens asking whether the unsaved changes to the current workspace are to be
   rejected or saved.
   The "Load Workspace" dialog opens.

3. Select a file (*.AMWT) and click "Open".

The workspace is loaded and the files are shown in the sideview and the working area.

Recent workspaces

For quick access, the recent workspaces are listed at "File > Recent Workspaces".

4.2.2   Saving a workspace

Save workspace / Save workspace as

Save the current editing status of the workspace. In addition to the changed data, for example,
geometric machinings and perspectives are also saved.

Proceed as follows to save a workspace:

1. Perform "File > Save workspace" to save the open workspace with the current settings.
   Or

2. Perform "File > Save workspace as" to save the workspace under a new name.
   The "Save workspace" dialog opens.
   If you want to change the path, select an alternative storage location for the workspace.

3. Enter a name, and click "Save".

The workspace is encrypted for saving in AMWT format. The file can only be opened with
Analyze MyWorkpiece /Toolpath .

Closing the scene/program

If you want to close a scene or Analyze MyWorkpiece /Toolpath and there are unsaved changes in scenes, a dialog will appear where you can accept or discard the changes.

4.3 Importing data

Importing a file into AMWT

Workpieces can be described in several data formats. A list of these formats can be found at Supported file formats (Page 31). These files must be created in the source programs, such as SINUMERIK Operate Trace. The AMWT itself does not provide any conversion functions.

Proceed as follows to import the data into AMWT:

1. Run "File > Import file(s)" to import data into a new scene.
   The "Import file" dialog opens.
   The supported file formats are shown in the drop-down list of the file formats.

2. Select a format filter.

3. Navigate in the file system to the desired folder.

4. Select the file and click "Open".
   For trace files, dialog "Trace File Import" is displayed. For details, see Importing trace files (Page 32).

A new scene is opened in the sideview of AMWT. The workpiece is displayed in the working area.

Alternatively, you can also import a model via the command line using the following string:

```
<application path><application name> <file path><file name>
```

Importing files with drag and drop

• Drag a file onto the icon on the desktop of your computer. If the format is compatible, AMWT starts and the file is loaded.
  - OR -

• Drag a file into the working area of the open application.

Recently imported files

The files last opened are listed in the "Last files" menu item.

1. Execute "File > Recent File(s)".

2. Select the file you want to import.

3. The file is imported and is displayed in the workspace currently being used.

4.3.1   Monitoring the import sequence

Description

For larger files, the import of data into AMWT can take some time.

The import of files can be checked with the import sequence.

1. To do this click the "Control Panel" button.

2. The import sequence is shown on the right alongside the working area.

3. Click "Cancel" to interrupt the currently running import.

Large files

Some files may be very large, so opening them may take some time. During this time, a dialog with a progress bar is shown which indicates the state of the import and, if applicable, the conversion of the data into G-code.

Error display

A warning symbol is displayed if errors occur when importing data or when opening Python scripts.

• Click on the warning symbol to open a dialog that provides support when resolving the error.

4.4   Using scenes
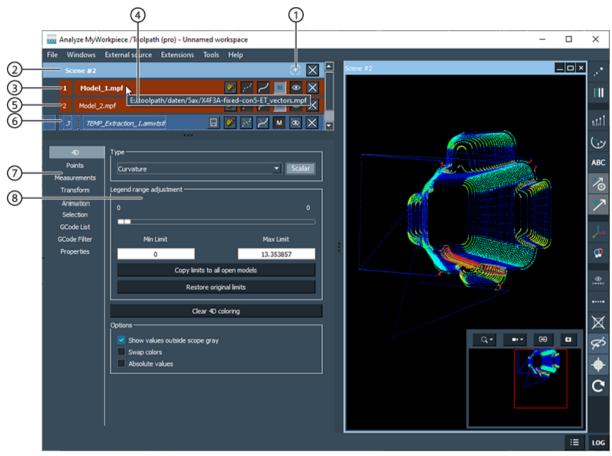
4.4.1   Scenes - overview

Scenes in AMWT

After the files have been imported into AMWT or a workspace has been opened, the workpieces are shown in the working area.

"Scenes" are created in the sideview to manage the files. The following functions can be used via scenes:

• Import further files

• Activate and deactivate display of workpieces

• Show and hide points, lines and surfaces.

• Color picker

• Display evaluation and representation functions

1	Button to add another file
2	Scene
3	Alias/path of the activated file 1; A double-click can assign an alias to the path. The following characters are prohibited: `~!@#$%^&*—+=|:;<>«»,?/{}'"[].
4	Tooltip with path of the activated file
5	The alias/path of a second, deactivated file, which can be displayed together with the first file and overlays it.
6	Temporarily generated STL file after a surface reconstruction (Surface Extractor)
7	Evaluation functions available for the selected file
8	Evaluation and display functions for the selected file

Surface elements in scenes

Button	Description
	Calls the "Import Files" dialog to load another file into the scene.
	Closes the selected file/scene
	Extends the list.

Button	Description
	Reduces the list
	Shows the dialog for color-coding points, lines and measurements.
	Shows and hides the points.
	Shows and hides the lines.
	Shows and hides the rendered surface (only for workpiece data that contains surface information)
	Shows and hides the complete workpiece.
	Saves the temporarily created STL file (Surface Extractor). Any unsaved files are discarded.
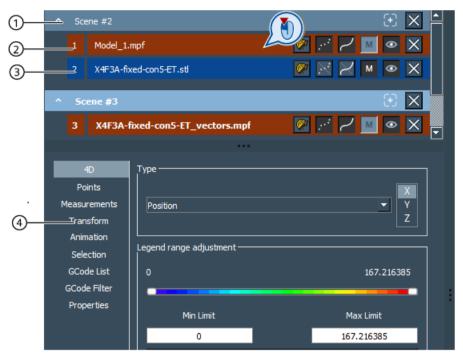
4.4.2   File selection in scenes

Evaluate files

Various functions to evaluate and represent the workpiece are available in the sideview. If several files are loaded to the scene, click on the entry under the particular scene.

The functions are integrated as widgets in the sideview.

1. Select the scene.
   The available functions are displayed in the sideview.

2. Click in the file or the model to display the functions for the particular file.

3. - Or -

4. The functions available for the selected file type are listed. The content changes with each file type

1	Name of the scene
2	MPF file
3	Mesh file
4	Evaluation and representation functions of the selected file

Figure 4-1   Scene with two loaded files (here MPF file)

4.4.3   Workpieces superimposed on one another

Loading multiple workpieces into one scene

Via a scene, you can open various data of workpieces or the data of workpieces several times in one scene. You can compare the workpieces directly by alternately showing and hiding points/ lines.

1. In a scene, click on "Add model" ("+").
   Dialog "Add model" is displayed.

2. Select a file and click "Open".
   - OR -
   Drag a file onto the working area of the active scene.
   - OR -
   Perform "File > Import file to active Scene".
   - OR -
   Execute "External source > Import Captured File(s) from Edge".
   The file is loaded into the scene.

Machining actions, such as turning or moving, are applied to all loaded workpieces.
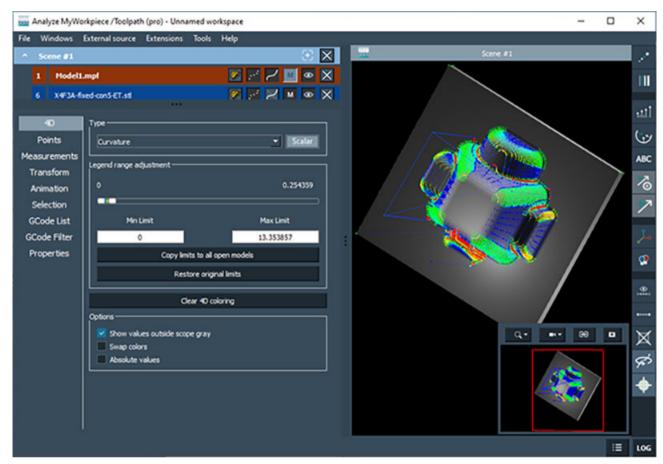
Figure 4-2   Scene with two files

- For each loaded workpiece, you can modify the display independently of each other by clicking the respective button in the file bar.

- To select the evaluation functions for the model, click on the corresponding entry in the sideview.

4.4.4   Displaying functions

Evaluating workpiece paths

Functions to display and evaluate the data are shown when the corresponding file below the scene is selected. They are organized as widgets in the sideview.

Various functions are available to display and evaluate the tool paths and surfaces.

Function	Description
4D	Provides graphic evaluation functions, such as position values, velocity curve.
Points	Lists the points in the tool path.
Measurements	Enables various measurements to be made with points.

Function	Description
Transformation	Allows you to move and rotate models.
T	Enables a virtual tool to be shown and traversed along the tool path.
Selection	Enables points to be selected according to specific criteria.
Events	The recorded signals are listed in the "Events" widget. The events are listed according to HF events and OPI (NC variables).
GCode list	Lists the part program. Available for JSON and MPF files, see Displaying and filtering GCode (Page 49).
GCode filter	Depending on the GCode, allows points to be displayed or hidden.
Properties	Lists various properties of the displayed workpiece.

4.5 Base data

4.5.1 Supported file formats

File types

The possible source formats must be created with the appropriate programs.

You can open the following data types in AMWT:

- NC or part programs:
    - *.mpf: Part programs (ASCII file).
    - *.cnc
    - *.spf
- JSON
    - *.json
    - *.zip (json container)
- Mesh files:
    - *.stl: Standard Tesselation language, the file format for describing the surface of 3D bodies (ASCII file). The format is supported by popular CAD programs, such as NX. The AMWT can import not only ASCII files, but also binary STL files.
- CSV:
    - *.csv: comma separated values; trace files (also as XML)
- XML
    - *.xml; XML file of a trace.

Large files require more time for the import.

4.5.2 Importing trace files

Opening a trace file

When you import a trace file, the "Trace File Import" dialog is displayed automatically. In this dialog, you specify the parameters for the import of the trace data. Whereby, the signals recorded in the trace serve as basis.

The following parameters are displayed for files that are not data type "JSON":

Column	Description	
Signal	Allows a signal to either be selected or deselected	
Type	Type of axis	
	Position	Position value
	Velocity	Velocity value
	Acceleration	Acceleration value
	Orientation	Tool orientation
	Torque	Torque value
Assigned signal	Axes are selected under size X, Y, Z, A, B, C, SP, scalar	
Signal source type	Displays the signal source type, position value, velocity...	
Description	The description of the signal.	

For JSON files, the following parameters are shown:

Column	Description	
Signal	Allows a signal to either be selected or deselected	
Symbol name	Type of axis	
	Position	Position value
	Velocity	Velocity value
	Acceleration	Acceleration value
	Orientation	Tool orientation
	Torque	Torque value
Assigned signal	Axes are selected under size X, Y, Z, A, B, C, SP, scalar	
Name	Displays the name of the signal as it was provided from the JSON file, e.g. signal name...	
Address	Displays the address of the signal. Signals are characterized by the name and the address	
Signal source type	Displays the signal source type (X1,....)	

Selecting signals

- In the "Signal" column, select the signals that are to be considered for importing. The default is that all signals are selected.

User-defined

To condition the signals for AMWT, you must specify the axes and types:

1. At "Axis", select the axis for which the signal was recorded for each signal.

2. At "Type", select whether the signal has recorded a position, a velocity or an acceleration.

Assigning automatically

- Click "Automatic selection" to assign the signals automatically to the axes and types.

Symbol names for custom signals

- Assign unique, expressive symbol names for your custom signals. The following characters are prohibited: `~!@#$%^&*—+=|:;<>«»,?/{}'"[].

Import of 2D models

If, when importing, you only specify 2 components (i.e. X/Y), then the data set can already be imported, assuming that the third component is always 0.
X-Z --> Y is always 0.
Y-Z --> X is always 0, and so on

Tool length compensation

- Activate this option "Compensate tool length" when importing a trace file.

This allows the different lengths of tools used for machining the workpiece to be taken into account during surface reconstruction. The tools used must be created in the tool identification database for this purpose. If tool length compensation has already been performed, this option is not required.

Generating graph data

Graph data can be generated to simplify analysis. The generated signal graphs are then shown in the graph display for evaluation, see also Signal evaluation (Page 111).

Triggering an import

- Click "Import" to import the signals.

A progress bar shows the progress. After the import, the workpiece is displayed in AMWT.

4.5.3 Part programs (NC programs)

Part programs (*.mpf)

The individual movements of the tool are specified as blocks in part programs. Different information is available depending on the part program.

Whereby, each program step corresponds to a point represented in the AMWT. Supplementary information, such as the feedrate, is also contained in the part program.

3-axis programs

In 3-axis programs, the positions in the X, Y and Z direction are programmed for the tool movement. Consequently, only such information that can be derived is available in the AMWT.

For example, important information is:

- Representation of the workpiece as points and lines
- Path curvature

5-axis programs

For 5-axis programs, in addition to the X, Y and Z positions, the spatial orientation of the tool is also specified. This allows not only the block information, but also the spatial orientation vectors of the tool to be calculated.

Whereby, the axes must be defined as A, B and C axes.

For example, important information for 3-axis programs is:

- Representation of the workpiece as points and lines
- Path curvature

4.6 Surface elements

Menu bar

The main menu provides various functions for editing and representation.

Toolbar

A large number of functions are available in the toolbar.

Button	Function	Larger	Smaller
	Adapts the point size in 5 stages.	Larger	Smaller
	Adapts the line thickness in 5 stages.	Larger	Smaller
	Adapts the length of the orientation vectors in 11 stages. After reaching size 10, there is a jump back to 0 and vice versa.	Larger	Smaller
	Adapts the length of the curvature vectors in 11 stages. After reaching size 10, there is a jump back to 0 and vice versa.	Larger	Smaller

Button	Function		
ABC	Adapts the length of the ABC vectors in 11 stages (only for 5-axis CNC models). After reaching size 10, there is a jump back to 0 and vice versa.	Larger	Smaller
	Enables selection of a custom vector for display based on the name.		
	Adjusts the length of the custom vectors in 11 levels (only if custom vectors are present). After reaching size 10, there is a jump back to 0 and vice versa.	Larger	Smaller
	Shows the coordinate system		
	Increases the brightness		
	Shows and hides the non-selected points. This function is not available if signal graphs are used.		
	Selects all points between the start and end points of a tool path. If several points are selected, then the points the furthest apart from one another are used.		
	Deletes the point selection.		
	Hides all points during a rotation. The option is set in advance.		
	Shows and hides the center of rotation. The center is defined by double-clicking.		

Sideview

The sideview is used to manage, display and select the imported files. The files are listed in scenes.

4.7 Program settings

Settings

Select "Tools > Settings" and open the dialog of the same name.

In this dialog, you can specify the setting for Analyze MyWorkpiece /Toolpath by clicking on the respective tab in the foreground.

General

Specify the general settings in the "General" tab:

NC parser

the number of points to be used for polynomial interpolation by the NC parser.

This is the number of points calculated during a polynomial interpolation. The more points are entered, the higher the number of interpolation points and the longer interpolation takes.

Click "Apply" to update the polynomial interpolation with the new settings.

Sequence of the rotation of an ABC vector

ABC vectors depict the orientation of the ABC axes in space. For a rotation, you can set in advance the order in which the model is rotated about the respective axis:

- XY'Z"

- ZY'X"

Synchronization

Two methods are available to synchronize models:

- Based on center of mass

- Based on position (X, Y, Z)

Click "Apply" to update the synchronization of the scene with the new settings.

More detailed information is available at Synchronizing models (Page 68).

Logging

Flash animation of the LOG button

You use this setting to specify how fast the LOG button should flash when a message is received.

Show log notification

You use this setting to specify how long a message should be shown in the bottom right-hand corner.

Zoom

You define the zoom settings in the "Zoom" tab.

Zoom settings for the mouse wheel

- Inverting the zoom setting with the mouse wheel

- Invert the zoom direction for smooth zooming

Use scaled zooming

1. Activate this option to use the actual zoom value for zooming speeds.

2. Deactivate the option to work with a fixed zoom speed.
   In the default setting, the option is not active.

Shortcuts

In the "Shortcuts" tab, you can define which keyboard commands are to be used for the listed functions.

1. Click the shortcut button.

2. Execute the desired keyboard combination on the keyboard.

Appearance

You can assign different colors to the different model types for display in the print preview.

1. Click the color button of the model type.
   The color selection dialog opens.

2. Click on the desired color and accept the selection with "OK".

STL viewer

To view STL files, you can also configure an external viewer of your choice. The STL files generated using Python scripts are then displayed using this viewer.

1. Under "Selected STL Viewer" enter a path to the *.exe file of the viewer.

2. To use the configured viewer, activate option "Use external viewer for STL visualization".

3. Also observe the notes regarding the properties required for the configured viewer.

Accepting settings

1. Click on the "Apply" button to save the settings and keep the dialog open.
   - Or -

2. Click "OK" to accept the settings and close the dialog.
   - Or -

3. Click "Default" to restore the default settings.
   - Or -

4. Click "Cancel" to reject the settings.

Switching over the language

Under "Tools > Language" you can change the UI language. There is a choice of:

- English

- German

4.8 Licenses and help

Calling the help function

Under "Help > Help" call up the operating instructions in the HTML5 format.

The operating instructions contains detailed information about Analyze MyWorkpiece / Toolpath.

Licensing

The following options are available for Analyze MyWorkpiece/Toolpath licensing; for more information, see Licensing with FlexLM (Page 17).

• FlexLM; allows licenses to be stored in the network (floating licenses) or also locally.

Displaying a workpiece

5

5.1 Displaying workpieces

Tabs and frames

The loaded data is displayed in the working area. Depending on your specific situation, the working area can be extended with tabs and frames:

- Tabbed view; you can toggle between the various tabs by clicking them.

- Tile; tiles allow the workpieces to be displayed with different settings next to each other.

- Sideview; the currently loaded workpieces are listed in the sideview. The icons allow you to assign user-defined colors, hide or show points, lines and surfaces, or close the file.

If you reload the data for the analysis, you can display it, for example, in another perspective.

The currently active view in a tab or frame is indicated on the selected tab.

Using tabs

The loaded data is displayed in the working area.

Proceed as follows to display the data in tabs:

1. Execute "Windows > Tabbed view".
   The data opened is arranged in tabs.
   The functions displayed in the project navigation and toolbars always apply to the active tab.
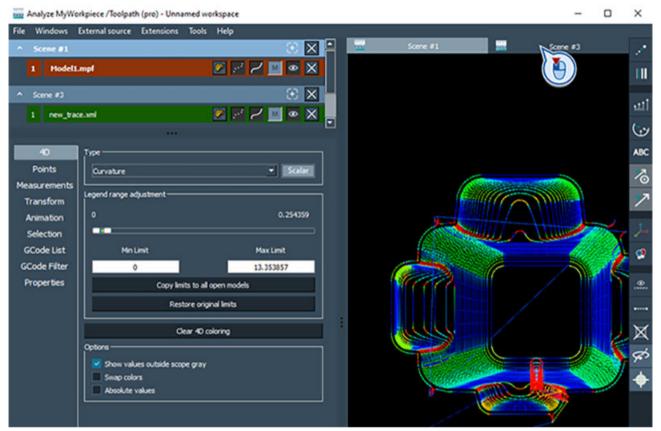

Figure 5-1   Tabs

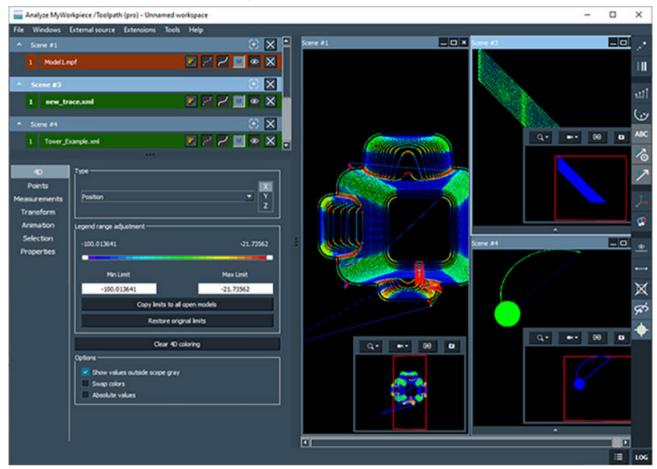2. Click on the tab to bring the tab into the foreground.

Using tiles

Tiles allow the working area to be divided vertically and horizontally. Frames can be combined with tabs.

Note

New models are loaded as default in the vertical tile mode.

---

1. Execute "File > Import File" for each file that you want to display in AMWT.

2. Execute "Windows > Tiles".
   The files are displayed in a vertical and horizontal distribution.

3. Execute "Windows > Tabbed view" and then "Windows > Tiles" again.
   The files are displayed in a vertical and horizontal distribution.

Figure 5-2   Frames

4. Click the "cross" in the tab title to close the tile, and therefore to close the scene. The tile is closed when the last tab title of a tile is closed. Alternatively, you can close the scene by clicking the "cross" in the sideview.

Size of the tiles

Depending on the requirement, you can adapt the horizontal and vertical sizes of the tile:

1. Move the mouse pointer over the boundary between two tiles.

2. When the mouse pointer becomes a double arrow, left-click.

3. Shift the mouse pointer (double arrow) with the left mouse button pressed.

Evaluation functions in the tiles

Additional evaluation functions can be used when using JSON, JSON.zip, CSV and XML files. For models that support these features, you will see an up arrow or three dots displayed in the footer that can be moved with the mouse.
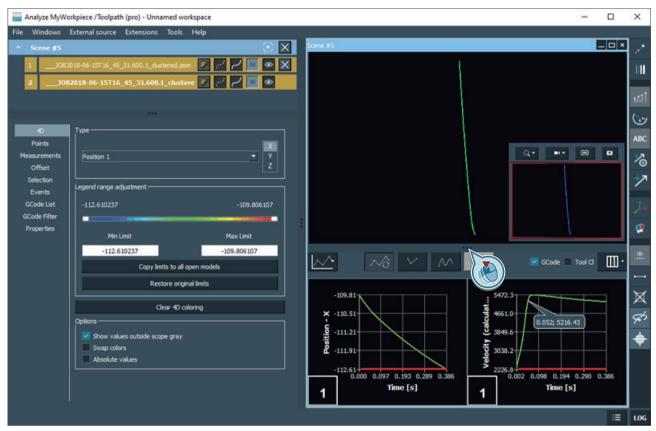

Figure 5-3   Evaluating a graph - overview

You can use the buttons to show additional graphical views of signals for evaluation. Detailed descriptions can be found under Signal evaluation (Page 111).

5.2   Displaying lines

Line representation

In the AMWT, the individual points (steps in the part program) are connected with lines.

Adapting the line thickness

The thickness of the lines can be increased incrementally in steps.

- Click the "Line thickness" button with the **left** mouse button.
  Each click makes the line thicker until the maximum line thickness is reached.
  The line thickness is adapted for all lines.

- Click the "Line thickness" button with the **right** mouse button.
  Each click makes the line thinner until the minimum line thickness is reached.

Hiding lines

The loaded files are listed in a folder structure in the sideview

1. Open the folder of the desired file.
   The shown surface elements are listed.

2. Deselect the lines to hide them.

5.3   Adapting the point display

5.3.1   Showing/hiding and selecting points

Points in the working area

Each displayed point represents a program step of the part program. To clarify the tool path, the points are connected with lines.

Selecting points

To analyze the machining of the workpieces, it is often necessary to display the properties of the individual points along the tool path.

To do this, select these points.

Selecting points with the mouse

1. Import the workpiece into the working area.

2. Select the corresponding tool path in the scene.

3. Click with the left mouse button on the points to select them.
Or:

4. Press the <Shift> key, click with the mouse pointer in the working area, and with the left-hand mouse button pressed, drag the area whose points are to be selected.
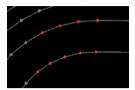The selected points are highlighted.

Figure 5-4   Selected points

Revoking the point selection

To revoke the point selection, proceed as follows:

1. Press the <Ctrl> key and click the points to be deselected.
- OR -

2. Press the <Ctrl> key, click with the mouse pointer in the working area, and drag with pressed mouse button the area whose points are to be deselected.
- OR -

3. Click the button "Deselect point" to deselect all points of the selected scene.
- OR -

4. Press <n> on your keyboard.

The selected points are somewhat larger; the size is reset when the selection is canceled.

How to customize the shortcuts is described here: Program settings (Page 35).

Hiding points

The loaded files are listed in a folder structure in the sideview

Click the "Show/hide points" button to hide the points in the view.

Displaying point information

The following information can be displayed for selected points:

• Index; shows the respective point index (number of the point)

• Coordinates; displays the coordinates of the point in the machine coordinate system.

• Velocity; shows the velocity of the tool at this point

• Curvature; shows the curvature of the tool path at this point.

• NC code; displays the NC code of the point.

1. Click the button Point information "I" to display the dialog including the available information.

2. Click the button including the information to display the relevant parameters.
   The data are displayed at selected points in the working area.

Figure 5-5   Point information

Adapting the point size

You can adapt the size of the points step-by-step in five stages.

- Click the "Point size" button with the **left** mouse button.
  Each click makes the points larger until the maximum point size is reached.
  The point size is adapted for all points.

- Click the "Point size" button with the **right** mouse button.
  Each click makes the points smaller until the minimum point size is reached.

Show selected points only

If you want to concentrate on selected points, you can hide all the non-selected points:

1. In the toolbar, click the "Show selected points only" button.

2. All non-selected points are hidden.
   If you click on the button again, all points are shown again - the previously selected points appear again.
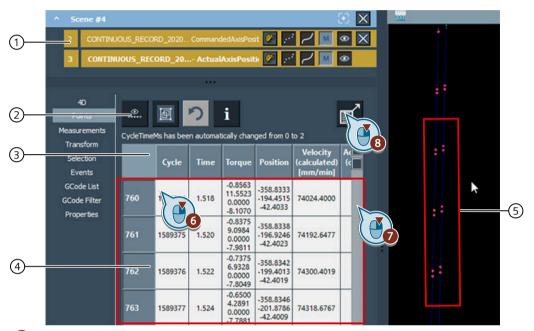
Selecting points along a tool path

Proceed as follows to select points along a tool path:

1. Select two points on a tool path.

2. In the toolbar, click the "Select all points between first and last point" button.
   All points between the two points are selected.

5.3.2 Points widget

Displaying selected points

The "Points" widget shows all points of the loaded file. If the points in the working area have been selected, the display can be restricted to the selected points.

① Selected tool path
② Buttons for showing points:
  - Show selected points only
  - Restrict display to selected points
  - Cancel restricted view
  - Show point information
③ Displayed parameters of the points
④ Selected points in the table
⑤ Selected points in the working area
⑥ Shows context menu to enter the number of the program line that is to be displayed.
⑦ Shows context menu with additional scroll functions.
⑧ Displays the point table next to the model (extend view)

Displaying information

For example, the following information is shown for the selected points:

- Numbering of the point
- X, Y, Z coordinate
- Velocity

- Orientation vector

- Curvature

- Curvature vector

- NC code

- Tool ID

The available information depends on the type of imported data.

Scroll functions

A right-click can show context menus, which offer various scroll functions:
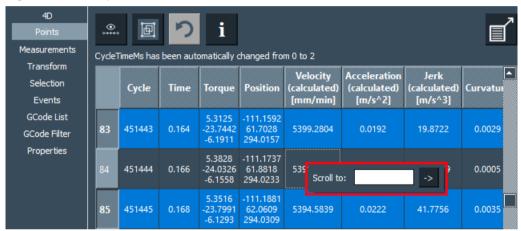
1. Right-click in the points table.

Figure 5-6   Scrolling the point widget

2. Enter the line number which you would like to display.
   - OR -

3. Right-click on the scroll bar.

4. A context menu with a range of functions is shown:

   – Scroll here; jumps to the line position where the right-click was executed.

   – Top; jumps to start

   – Bottom; jumps to end

   – Page up; moves up one page

   – Page down; moves down one page

   – Scroll up; moves up one line

   – Scroll down; moves down one line

Selecting points

Select the points either from the points table or the working area. The points table and the representation in the working area are linked.

Extend view

You can extend the view to show the points table clearly alongside the associated model. The points table is then shown alongside the model, and may cover other displayed models.

1. Click the button.

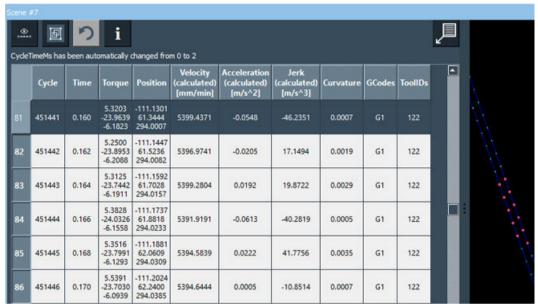2. The points table is shown alongside the model.

Figure 5-7   Extended view

3. Click the button to close the extended view.

5.3.3   Selecting points

Searching for distances between points

The AMWT provides the "Selection" widget for the analysis. The selection function allows searches for distances between points and the selection of points that meet a specific criterion. The points found are then selected.

Proceed as follows to select points at defined distances:

1. Enter an operator, such as ">", "<" or "=", to enter a distance evaluation. You can also combine "<" or ">" and "=" to represent ≤ or ≥.

2. Enter a distance [mm].

3. Click "Select" to initiate the process. The points found are then shown.

5.4   Displaying and filtering GCode

Displaying a part program in the GCode list

If the data contains a part program (GCode), the program can be displayed. The display is only possible with JSON and MPF files.

1. Select the scene.

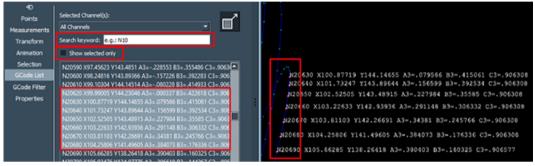2. Select the model and click on the "GCODE list " widget in the foreground.

Figure 5-8   GCode view

3. Select the lines in the GCode to select the corresponding points in the model. The points are shown in red.
- Or -

4. Mark the points in the model. The lines in the GCode appear selected.

5. Click "Extend view" to display the GCode in an extra column alongside the model view. This is only possible for JSON and MPF files.

6. Click "Select channel" to display the code of the desired channel.

7. Activate the "Show only selected" option to display only the lines of the selected points.

Note

The same NC code can apply to many different points.

Searching with keyword

You can use the keyword search to display lines of code that contain this term. The search is case-sensitive.

1. Enter a keyword in "Search keyword".

2. The lines containing the word/term are displayed immediately.

Filtering GCode

The various GCodes, which contain a part program, can be filtered under the "GCode filter" tab. In addition to GCode, points that were calculated using polynomial interpolation, can be filtered.

1. Click on "GCode filter", to display the list of GCodes contained in the part program.

2. Click on the corresponding button to show or hide the command type.
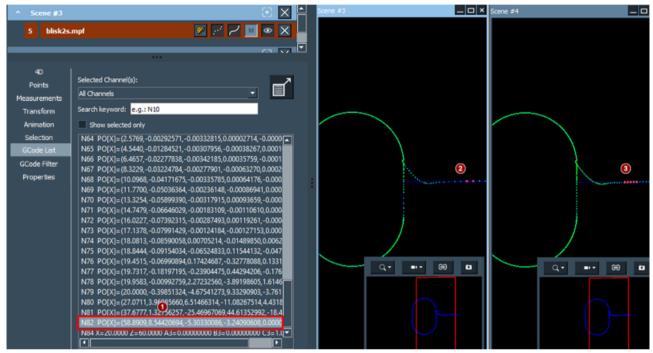

Filtering polynomials

If the part program contains lines with polynomial interpolations, then these can be filtered.

5.5 Points for the polynomial interpolation

Model with polynomial interpolation

Contains the code for the displayed polynomial interpolations model. The number of points to be used in the polynomial interpolation can be set in "Tools > Settings > General" The more points are used, the more precise the interpolation. The following graphic shows a comparison of two models with polynomial interpolation, one with 100 points and one with 20.

①   Selected line in GCode with a polynomial interpolation
②   100 points selected
③   20 points selected

Figure 5-9   Polynomial interpolation
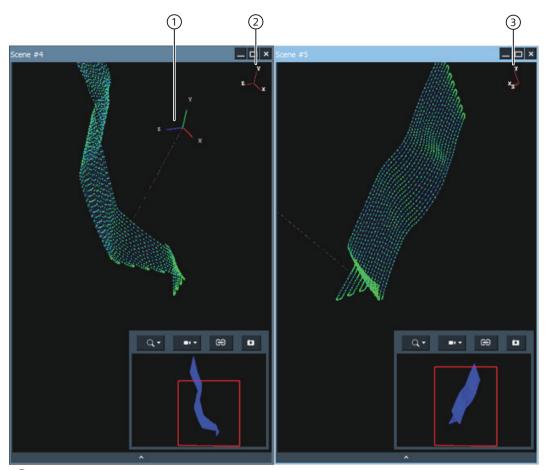
5.6 Show coordinate systems

Coordinate systems

If you have loaded a workpiece in the working area, you can display the following coordinate system:

Button	Function
	Global coordinate system (large). The coordinate system stands in the center of a virtual cuboid drawn around the borders of the model. In a rotation, the coordinate system is also moved and can go out of the field of view. The size of the coordinate system does not change when the model is enlarged or reduced.

①   Coordinate system that can disappear on rotation out of the display field.

②   Fixed coordinate system that is shown at a fixed location and also rotates during rotation.

③   Coordinate system after rotation. The flexible coordinate system is no longer visible.
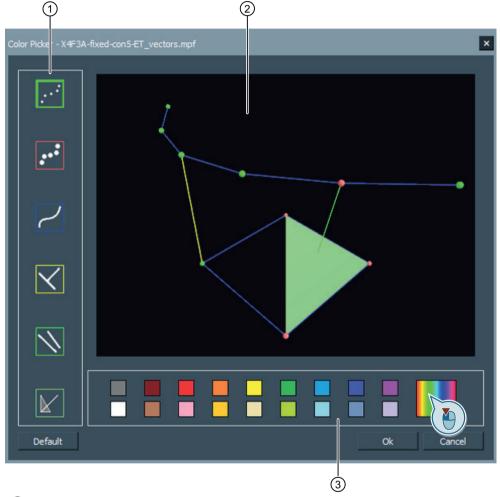
Figure 5-10   Displaying the coordinate system

For orientation, while analyzing a model, a small coordinate system can be displayed in the upper right-hand corner,

5.7 Adjust color display

Setting object colors

Using the color picker, you can assign colors to the individual objects in the working area. This allows you to color-code, for example, the lines and points of overlying workpieces in different colors for a better differentiation.

①   Element selection, buttons for selecting the element
②   Preview, shows how the elements are color-coded
③   Color palette, shows the current colors

Figure 5-11   Color picker

Elements

Use the buttons to select the elements whose colors you want to change:

Button	Description
	Allows you to change the color of the points.
	Allows you to change the color of the selected points.
	Allows you to change the color of the tool paths.
	Allows you to change the color of the result line of a distance measurement.
	Allows you to change the color of the guide lines for a measurement.
	Allows you to change the color of an area.

Change the color of elements

The colors of the workpiece elements can be changed using the color palette:

1. Click the button of the element you want to recolor.

2. Select the color in the color palette. The color palette contains various predefined colors. The preview shows how the color changes.

3. Click "OK" to apply the color.
   - OR -

4. Click on "Default" to restore the default settings.

Color palette for free colors

If the predefined colors do not match the requirements, you can use a color palette to define free colors.

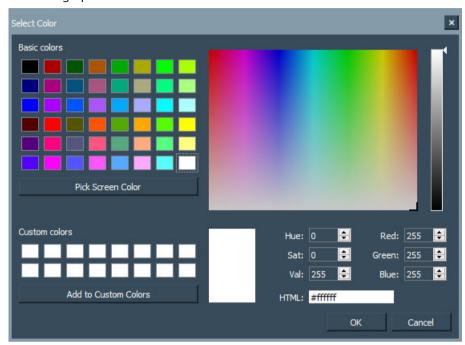1. Click the "rainbow" to call up the color palette.
   The dialog opens.

Figure 5-12   Color palette

2. Define the requested color and add it to your colors (Custom colors).

3. Click "OK" to apply the setting.

Performing transformations

<div style="text-align: right; font-size: 2em;">6</div>

6.1 View transformation

6.1.1 Moving objects

Moving the view with the mouse

This action moves the view to the object, while the object itself does not change.

Objects in the AMWT working area are moved with the mouse:

1. Right-click the object and hold the mouse button down.

2. Move the object to the desired position with the mouse. Actually, the object is not moved, but rather the view of the object is changed (the camera is moved).
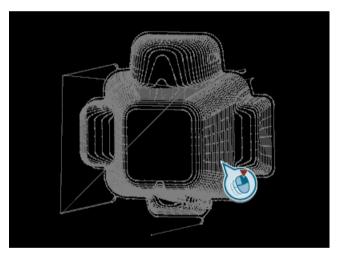
Figure 6-1   Moving the view to the object

6.1.2 Rotating objects

Turning the view of the object with the mouse

This action turns the view of the object, while the object itself does not change.

Objects in the working area are turned or rotated with the mouse.

1. Left-click the object and keep the mouse key pressed.

2. Move the mouse to turn the object in the desired view around the set pivot point. Actually, the object is not turned, but rather the view of the object is changed (the camera turns around the object).
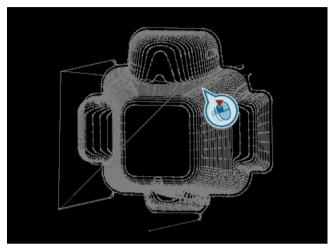

Figure 6-2   Turning objects

Hiding points

	This function is active per default. The points of each model of the corresponding scene are then hidden while the camera moves.
	Use these functions for models featuring a high number of points to perform the rotation faster and more fluently.

Defining and displaying the center of rotation
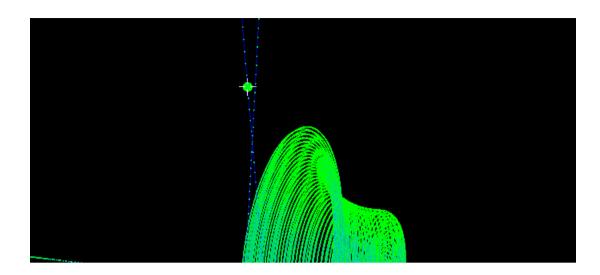
You set the center of rotation (pivot) as follows:

1. Enlarge the model until you see the individual points.

2. Double-click on the point where you want the center of rotation to be.

3. Click the "Show center of rotation" button to mark the point.

	This function shows the center of rotation as a large dot.

The center of rotation is shown highlighted,

6.1.3 Zooming objects

Enlarging or reducing objects

Enlarge objects in the working area to investigate interesting structures in large format. Zoom out to return to a clear view. Actually, the object is not enlarged or reduced, but rather the camera moves closer or further away from the object.

• The zoom direction can be inverted in the settings (Program settings (Page 35)).

Procedure

1. Click the object.

2. Coarse zoom
   Press the scroll wheel and rotate it backward (enlarge) or forward (reduce).

3. Smooth Zoom
   Keep the mouse wheel pressed, and move the mouse to zoom smoothly.

Figure 6-3   Zooming objects

Zooming via Page up - Page down

Press the <Page down> and <Page up> keys to enlarge or reduce the object gradually.

6.2 Setting a transformation

6.2.1 Widget transformation

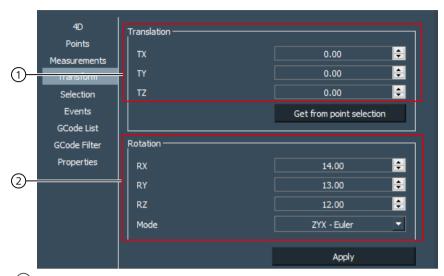**Moving and rotating models**

In the "Transformation" widget, you can move and rotate models by defined values.

If you loaded 2 models into the scene, you can align them by selecting two points.

For example, if you superimpose the trace of a machining operation over the part program, you can detect mechanical disturbances such as vibrations. To do this, the two points in the models must correspond.

Transformation widget

You can find the widget in the page view of Analyze MyWorkpiece /Toolpath.

①   Input fields for moving in the TX, TY and TZ direction

②   Input fields for rotating about the RX, RY and RZ axis

③   Apply: Updates the values in data structures after a move or rotation.

Updating (Applying) values in data structures

In the various evaluation functions, such as graphs, position values are calculated or displayed. When you move or rotate a model, the position data changes without updating the values in the data structures.

•   Click "Apply" to update the data in the different data structures (graphs, models...).

Note

Always run "Apply"

If you have moved or rotated models, always click "Apply" to keep the data structures consistent.

6.2.2   Align models

Requirement

You have loaded two models into a scene.

Shifting models (Translation)

The last loaded model can be shifted by entering the various values:

1. Enter the values for shifting in the X, Y or Z direction. The model is immediately shifted.
   - OR -

2. Click the arrows on the right edge of the input fields to change the values for shifting.
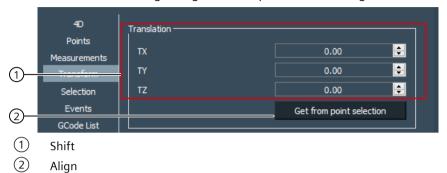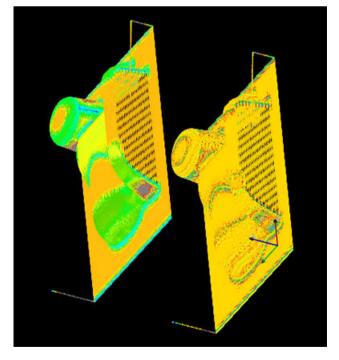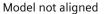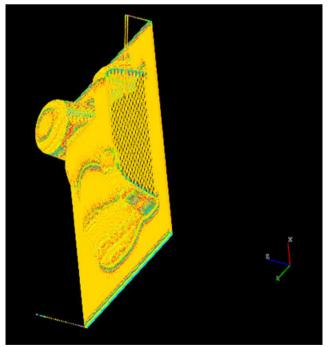

①   Shift
②   Align

Figure 6-4   Translation

Aligning models

If you want to make the two models coincide, the points in the models must correspond.

1. Select **one** point in each of the models.

2. Click the "Get from point selection" button (②).
   The two selected points and thus the two models are made to coincide.

Model not aligned   Model realigned

Updating position data in graphs

In the graphical evaluation you can display position data as graphs, for which then corresponding position data is displayed. When you shift or rotate a model, the position data changes.
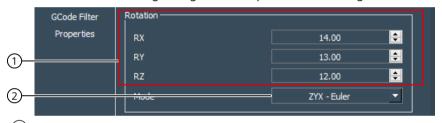
Click on "Apply" to transfer the current position data from the model to the graphical evaluation.

6.2.3 Rotating models

Rotation

The last loaded or selected model can be rotated by entering various values:

1. Enter the values for rotating in the RX, RY, or RZ direction. The model is immediately rotated.
   - OR -

2. Click the arrows on the right edge of the input fields to change the values for rotating.

① Angle values for rotation
  - RX: Rotation around the X axis
  - RY: Rotation around the Y axis
  - RZ: Rotation around the Z axis
② Display mode for the Euler angle

Figure 6-5   Rotation

Mode

Under Mode you set which Euler angle you want to use for the rotation:

- ZYX-Euler: The first rotation is about the Z axis (fixed axis), then there is a rotation about Y' (Y axis rotates along with the rotation about Z), then a rotation about X' (rotates along with the rotation about Z and Y').

- XYZ-Euler: The first rotation is about the X axis (fixed axis), then there is a rotation about Y' (Y axis rotates along with the rotation about X), then a rotation about Z' (rotates along with the rotation about X and Y').
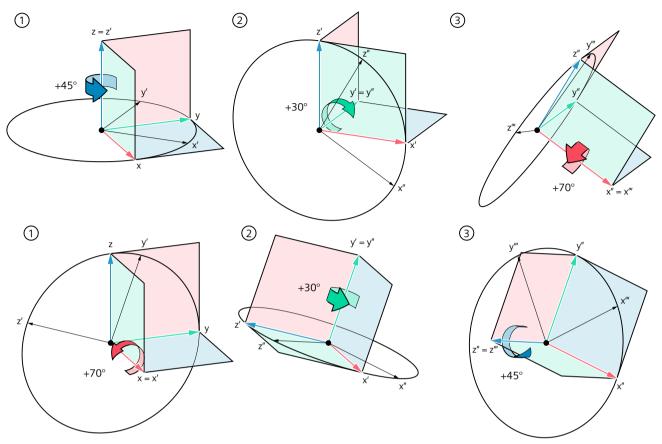
Figure 6-6   Euler angle scheme

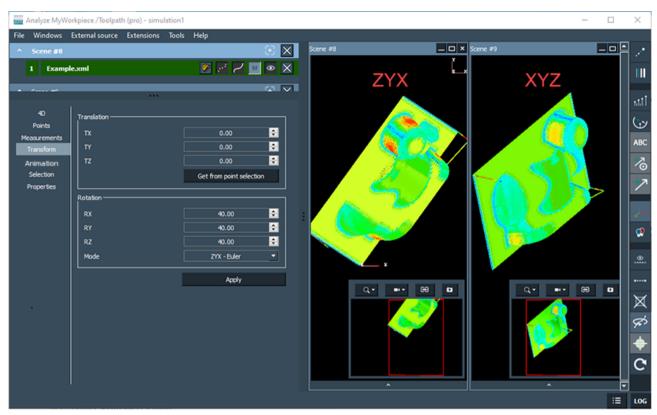The following example shows a rotation of 40° around ZYX and XYZ.

Figure 6-7   Euler angle

Using the navigation window

<div style="text-align: right; font-size: 3em;">7</div>

7.1 Navigation window

Navigation window

The navigation window is loaded into the workspace together with the model at the bottom right and is anchored.

The partial image currently displayed is shown with a red border.
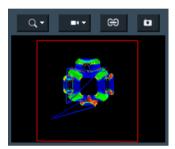

Figure 7-1   Navigation window

In the navigation window, the following points are available in the toolbar (from left to right):

- Zoom
- Set standard perspectives
- Synchronize model views
- Create and save a screenshot (report)

7.2 Set default camera view

Available perspectives

The perspective corresponds to the viewing direction of the camera aimed at the object. You set the perspective using the navigation window toolbar.

Figure 7-2   Navigation window toolbar

Several different perspectives are available in AMWT:

Button	Meaning
	From the top
	From the bottom
	From the front
	From the rear
	From the left
	From the right
	Home view (perspective when opening)

7.3 Synchronizing models

Synchronization

If you have opened your workpiece for analysis in several tabs or tiles, you can synchronize the views in the tabs or tiles. This is possible with up to five scenes. If you want to synchronize more than five scenes, the "Synchronize" button is deactivated.

- You can synchronize up to five models.

- The functions of the Navigation window are deactivated as long as the models are synchronized.
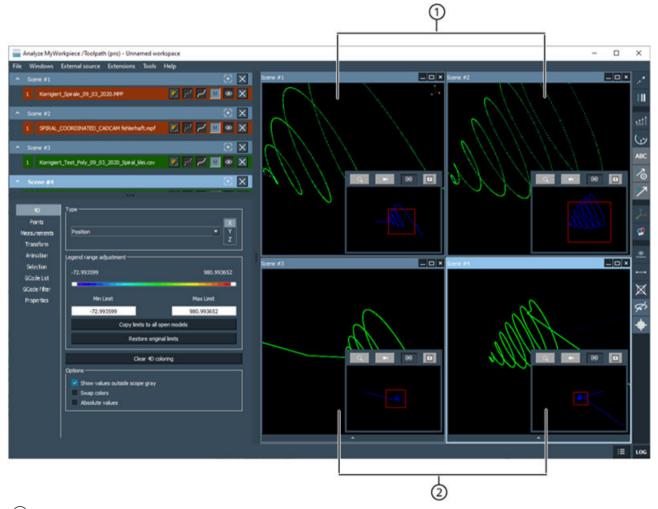
- The synchronized models can be zoomed, moved and rotated together.

- After synchronization, a tooltip is shown which displays the synchronization.

2 synchronization types are available for the function. You make the settings under "Tools > Settings" (Program settings (Page 35)).

Based on the center of gravity

Synchronization "Based on center of mass" is better for various file types; if the coordinate systems differ, then the zoom level is not synchronized. In the example below, it can be seen that the models can be synchronized in spite of the different coordinate systems.

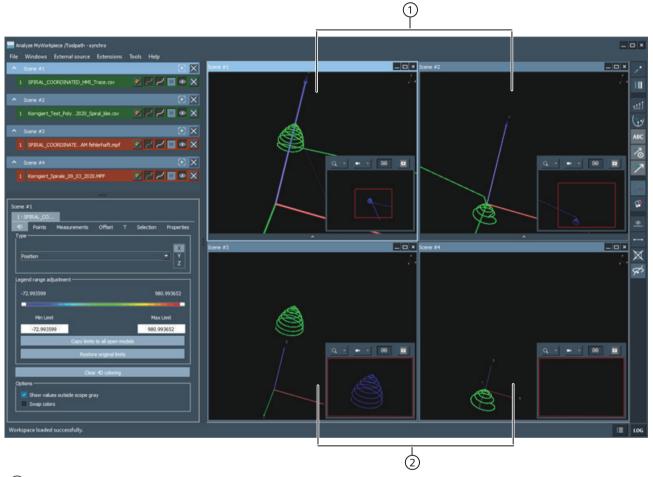- Therefore, activate option "Based on center of mass".

①   Files, type *.CSV
②   Files, type *.MPF

Based on position (X, Y, Z)

Synchronization "Based on position" has an advantage if the file type is such that the coordinate system is the same for all synchronized files, e.g. in the case of files from different machining phases. If you zoom into the examples below, then the models disappear from the camera range and are no longer visible. This is why we recommend that synchronization is realized via the center of gravity.

• Therefore, activate option "Based on position (X, Y, Z)".

①	Files, type *.CSV
②	Files, type *.MPF

Main and ancillary scenes

To synchronize scenes, a differentiation is made between the main scene and ancillary scenes. A scene is defined as main scene, where the synchronizing button was printed first. All other scenes, which are open, are classed as ancillary scenes.

The main scene defines the motion to be executed (rotating, shifting, zooming,…) of the ancillary scenes. The positions of the models before synchronization are rejected, and the models in the ancillary scenes orient themselves to the main scene.

Performing synchronization

1. To synchronize the models, call up the navigation window toolbar.

2. Click the "Synchronize" button.

3. Click the "Synchronize" button for the second display.
   The pressed button indicates that the synchronization is activated.

4. Activate the synchronization in all displays to be synchronized.
   All other buttons are deactivated during the synchronization run.
   In the example, the main scene has a red border, the ancillary scenes have a blue border, the synchronization type is "center of gravity".
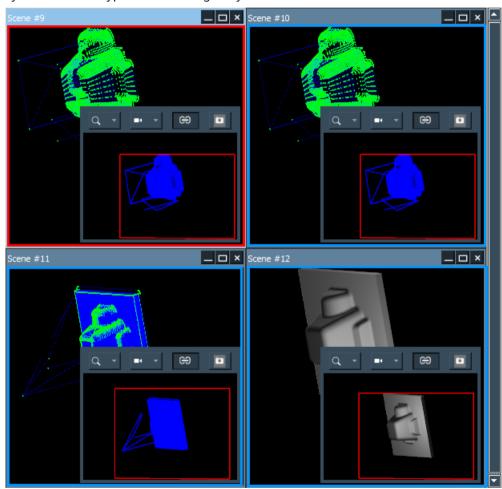
Figure 7-3   Synchronous display

The views of the selected models are now synchronized. Changes in one display are replicated concurrently in the other display.

Zooming or moving individual models

If you want to move or zoom individual models, proceed as follows:

1. Click the model to set the focus.

2. To do this, press the <Alt> key and execute the movement/zoom.

If you want to display the synchronized models again, you must deactivate and then reactivate the synchronization.

7.4 Zooming and adapting a display

Enlarging or reducing objects

The tools for enlarging or reducing objects are provided in the navigation window toolbar.

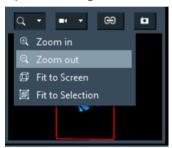1. Open the navigation window toolbar.

Figure 7-4   Enlarging and reducing

2. Perform "Zoom in" to enlarge the view.

3. Perform "Zoom out" to reduce the view.

Adapting the workpiece view

Rather than zooming manually, you can also perform two operations with fixed enlargement.

1. Perform "Fit to Screen" to adapt the workpiece to the available work area.

2. Perform "Fit to Selection" to adapt the point selection to the size of the work area.

7.5 Creating and editing screenshots

Creating screenshots

To document the analysis of a workpiece, it often suffices to show the workpiece in a specific perspective. To freeze and forward this perspective, you can capture screenshots, augment them with appropriate text or markings, and export them.

To create a screenshot, proceed as follows:

1. Click the "Save screenshot" button in the navigation window toolbar.
   AMWT creates a screenshot of the current working area.

2. Click "Save" to save the screenshot as a file.

3. Click "Copy" to copy the screenshot to the Windows clipboard.

Displaying additional information

You can display more information for documentation and the unambiguous interpretation of the displayed pictures:

- Date/time

- GCode colors

- Tool identification colors

Animating tool paths

<div style="text-align: right; font-size: 3em; font-weight: bold">8</div>

8.1　　Tool animation

Displaying tool paths

The tool paths programmed via part programs can be displayed animated in AMWT. The animations show the machining process for the workpiece.

The animations are controlled via the T widget:

1. Open AMWT.

2. Import the file and select the scene.
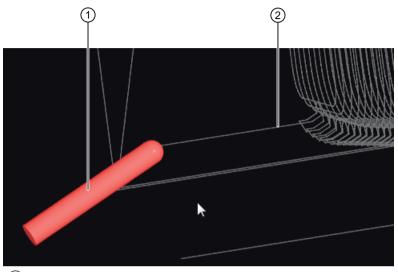
3. Click the "T" button.
   The widget is displayed.

①	File name of the tool path
②	Displaying a tool
③	The point where the tool is currently located
④	Buttons for controlling the animation
⑤	Controller for the velocity of the tool movement
⑥	Option to display the tool at selected points
⑦	Position and orientation of the tool
⑧	Properties of the tool

Virtual tool

A virtual tool is displayed automatically when you call the T widget. The position and orientation of the tool are specified. The properties of the tool can be adapted, see Properties of the tool (Page 78).

①   Virtual tool
②   Tool path
Figure 8-1   Tool display

See also

Analyzing orientation vectors (Page 124)

8.2 Controlling tool animations

Controlling animation

An animation allows the tool to be moved along these paths so that the machining can be simulated.

• To activate tool animation, click on "Display tool".

The animation can be started, stopped or interrupted via buttons:

▶	Runs the animation forward.
◀	Runs the animation backward.
▶|	Moves the tool one step forward.
|◀	Moves the tool one step backward.

▶▶|	Moves the tool to the end of the path.
|◀◀	Moves the tool to the start of the path.
■	Stops the animation.

Note

When using a NVIDIA graphic card, the tool is not displayed at the selected points.

Controlling the velocity

The velocity with which the tool moves along the path is controlled with a slider.

Figure 8-2   Controlling the velocity

8.3   Properties of the tool

Properties of the tool

You can adapt the properties of the tool for the simulation. This gives you a more realistic view of the workpiece machining. The tool is represented as a cylinder, either with a flat tip (cylindrical) or a rounded tip (spherical).

You can adapt the following properties:

- Tool diameter; the default value is 6.

- Tool length; the default value is 40.

- Depth; the value depends on the tool diameter; the value describes the penetration depth of the tool in the workpiece. A cylindrical tool does not have any depth because it is flat.

Tool tip

It is possible to choose between two tool forms:

- Cylinder; the tool is a cylinder without tip.

- Sphere; the tool is a cylinder with a rounded tip.

See also

Tool animation (Page 75)

Using vectors

<div style="text-align: right; font-size: 3em;">9</div>

9.1   Displaying orientation vectors

Using tool orientation vectors

With AMWT, you can show orientation vectors in order to analyze tool paths. The vectors are displayed at each point of the tool path. If points are selected, then the orientation vectors are shown only at the selected points.
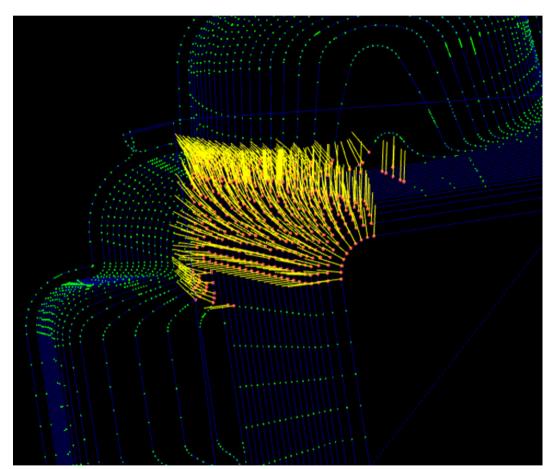

Figure 9-1   Orientation vectors at selected points

Note

Orientation vectors can be displayed only when orientations exist in the data source. Possible sources are, for example, 5-axis part programs.

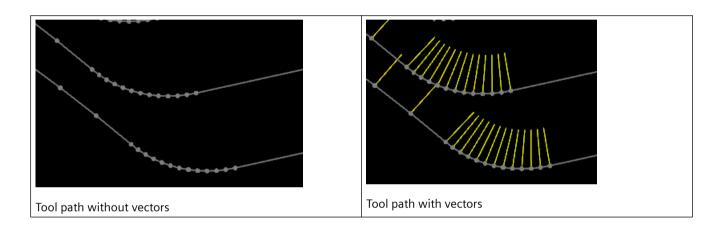To display the orientation vectors, proceed as follows:

1. Click on "Change orientation vector length".
   An orientation vector is displayed at each point.

2. Click the button with the **left** mouse button to extend the vectors until the maximum length is reached.

3. Click the button with the **right** mouse button to shorten the vectors. The orientation vectors are hidden when the length is at "0".

The vectors can be lengthened or shortened step-by-step in 11 stages. The currently set value is displayed via a tooltip. After "10", the display jumps back to "0" and the vectors are hidden. 5/10 is the default value.

Tool path without vectors

Tool path with vectors

9.2 Displaying curvature vectors

Using curvature vectors

With AMWT, you can show curvature vectors in order to analyze tool paths. The vectors are displayed at each point of the tool path. If points are selected, the curvature vectors are shown only at the selected points.
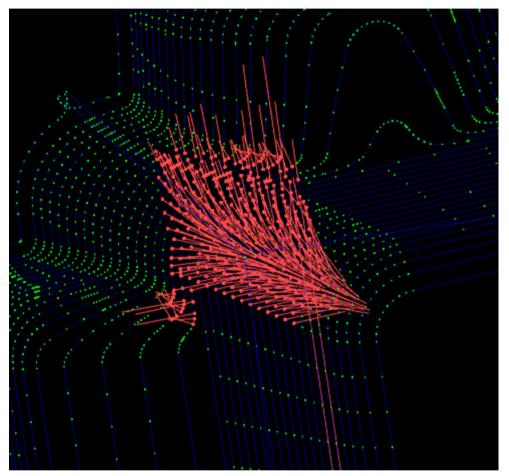

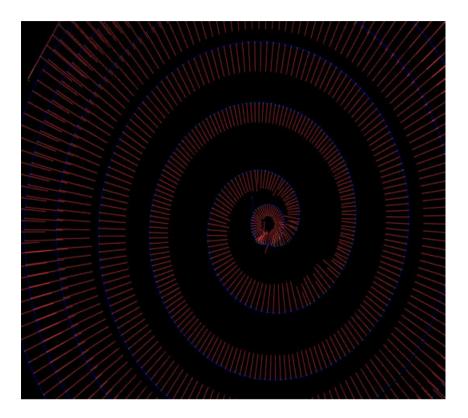Figure 9-2   Curvature vectors at selected points

To show the curvature vectors, proceed as follows:

1. Click on button "Change curvature vector length".
   A curvature vector is displayed at each point.

2. Click the button with the **left** mouse button to extend the vectors until the maximum length is reached.

3. Click the button with the **right** mouse button to shorten the vectors. The curvature vectors are hidden when the length is "0".

The vectors can be lengthened or shortened step-by-step in 11 stages. The button tooltip shows the degree of the lengthening, e.g. "0/10". After "10", the display jumps back to "0" and the vectors are hidden. "5/10" is the default value.

9.3 Displaying ABC vectors

ABC vectors

ABC vectors are only available for some models, such as the 5-axis CNC models of the file types CNC and MPF, for example. They show the orientation of the ABC axes in space. The vectors are displayed at each point of the tool path. If points are selected, the ABC vectors are shown only at the selected points.

Notifications display the following information about the part program:

• ORIWKS is **not** programmed in the part program.

• ORIMKS is programmed in the part program.

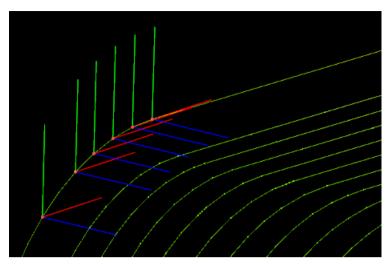Figure 9-3   ABC vectors

Proceed as follows to show the ABC vectors:

1. Click the "Change ABC vector length" button.
   An ABC vector is displayed at each point.

2. Click the button with the **left** mouse button to extend the vectors until the maximum length is reached.

3. Click the button with the **right** mouse button to shorten the vectors. The ABC vectors are hidden when the length is "0".

The vectors can be lengthened or shortened step-by-step in 11 stages. The button tooltip shows the degree of the lengthening, e.g. "0/10". After "10", the display jumps back to "0" and the vectors are hidden. 5/10 is the default value.

9.4 Displaying custom vectors

Custom vectors

For custom vectors, the import data must contain data from which vectors can be derived.

Without this output data, you cannot show custom vectors. You will be informed accordingly.

- CNC files:
  Process the feedrates F_X, F_Y, F_Z and use the result as the basis to create a custom vector, which you can visualize in AMWT.

- Trace files:
  Define custom vectors in the "Import trace file" dialog when importing the model.
  For example a 3-axis machine records the data DES_POS|1, DES_POS|2, DES_POS|3 and CTRL_DIFF|1, CTRL_DIFF|2, CTRL_DIFF|3. You see the tool with the DES_POS data points. You can then combine CTRL_DIFF|1, CTRL_DIFF|2, CTRL_DIFF|3 to create a vector <CTRL_DIFF|1, CTRL_DIFF|2, CTRL_DIFF|3 > and visualize the vectors along the data points.

If points are selected, the custom vectors are shown only at the selected points.

Trace files

The following graphic shows the torque as a custom vector.

During import, the data TORQUE|1, TORQUE|2 and TORQUE|3 are assigned to a vector, which can then be displayed.
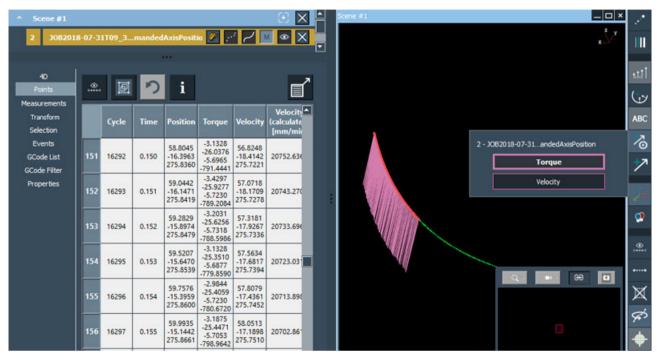

Figure 9-4   Custom vectors

Change vector length

Proceed as follows to show the custom vectors:

1. Click the "Change length of custom vector" button.
   A custom vector is displayed at each selected point. If no points have been selected, the vectors at all points are displayed.

2. Click the button with the **left** mouse button to extend the vectors until the maximum length is reached.

3. Click the button with the **right** mouse button to shorten the vectors. The custom vectors are hidden when the length is "0".

The active vectors can be lengthened or shortened step-by-step in 11 stages. The button tooltip shows the degree of the lengthening, e.g. "0/10". After "10", the display jumps back to "0" and the vectors are hidden. 5/10 is the default value.

Selecting the active custom vector

If you have defined several custom vectors for your model, you can select the respective active vector for display. The vectors are color-coded. Only one vector can be shown at a time. The calculated points of the vectors in each case are listed in the "Points" widget in a column.

1. Click the "Select active custom vector" button.
   The menu with the custom vectors is displayed.

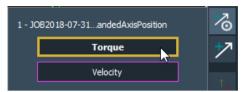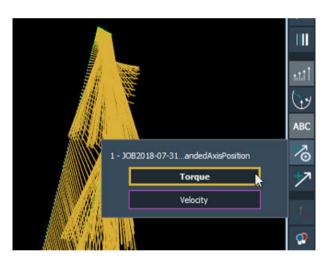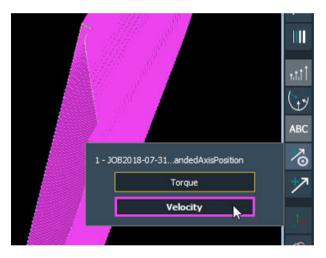2. Click the button of the custom vector to be displayed.

Figure 9-5   Select active custom vector

If you now show custom vectors, the color of the selected active custom vector is used. In the example, the vector for torque is displayed in "ocher", for velocity in "violet".

Torque

Velocity

Analyzing

<div style="text-align: right">

10

</div>

10.1 Using the fourth dimension

10.1.1 Using the fourth dimension

Overview

Workpieces are displayed and described in the three spatial dimensions. In addition to these dimensions, AMWT allows the points and the lines of the displayed workpiece to be assigned further properties using colors. In effect, the workpiece (points and lines) is represented in four dimensions. The associated properties and axes/paths can be selected, and depend on the associated base data.

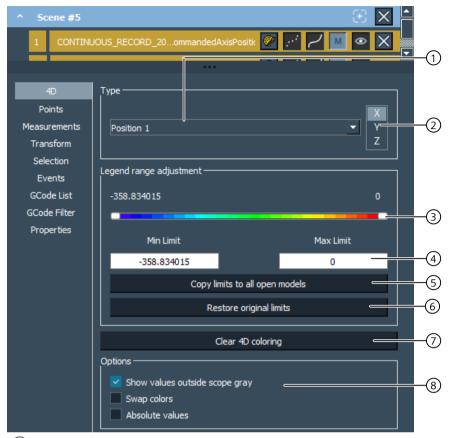The 4D evaluation is performed in the "4D" widget.

1. Select the file in the scene.
   - Or -
   Click the workpiece in the working area.
   The evaluation widget is displayed.

2. Click "4D".
   The widget with the same name is displayed.

① List of the available properties
② List of the axes
③ Slider to set the minimum/maximum value and the value range
④ Minimum/maximum values of the selected property
⑤ Copy min./max. values to other scenes
⑥ Restoration of original minimum/maximum values (previously defined values)
⑦ Restoration of original color assignment (previously defined colors)
⑧ Options for the display

Figure 10-1   4D widget

10.1.2   Variables for the 4D evaluation

Selecting a variable

Only those physical variables defined in the base data can be evaluated via the 4D widget. Not all data is offered with each data source.

The following variables can be selected:

Type	Variable	Description
Position	Position	The positions of the points in the coordinate system used in the part program
Velocity	Velocity	The velocity distribution along the tool path.
Orientation	Orientation vector	The orientation vector of the tool (the orientation of the tool in the coordinate system).
Curvature	Curvature	The curvature distribution along the tool path.
Acceleration	Acceleration	The acceleration distribution along the tool path.
Torque	Torque	Torque distribution along the tool path.
Jerk	Jerk	The jerk (change in acceleration) distribution along the tool path.
GCode		The GCodes used are displayed in color (JSON and MPF files). If part programs contain polynomial interpolations, then these are also displayed with the GCode.
Tool ID		The tool IDs are displayed in color (JSON files)
The user-defined signals from the trace import are added at the end of the table.		

The scalars can be measured or calculated. You will find a corresponding entry in the selection menu of the 4D widget.

10.1.3   Minimum/maximum values in the 4D evaluation

Selecting the type

Select from the "Type" drop-down list the type of the signal to be displayed. The content of the drop-down list depends on the source data. If, for example, no signal of the orientation vectors was recorded in the trace, you cannot select it.

The selection of the signal type also changes the unit.

Minimum values and maximum values

The minimum and maximum values are actively assigned from the "4D colors" dialog.

The values are assigned to the colors of the spectrum in values increasing from left to right. The minimum and maximum values allow the resolution to be set.

As the assignment uses the highest minimum/maximum values, you may need to edit the minimum/maximum values manually. For example, the G0 traversing velocity is used as the maximum value for velocities. This velocity, however, is significantly higher than, for example, the traversing velocity for milling, so that those areas of actual interest are not resolved.

To optimize the display, proceed as follows:

Assigning minimum/maximum values using sliders

You can change the minimum/maximum values continuously using the sliders.

1. Move the left-hand slider to adjust the minimum values.

2. Move the right-hand slider to adjust the maximum values.

3. Click on the slider and shift the complete value range to either the left or right.
   Changes to the value range and color changes are immediately accepted.

Entering minimum/maximum values

1. Enter the minimum/maximum values in the text boxes.

2. Press the <Enter> key to accept the values.

3. Click on "Restore original limits" to restore the previously set values.

4. Click "Clear 4D Coloring" to reset the coloring. The min./max. settings for coloring are not deleted.

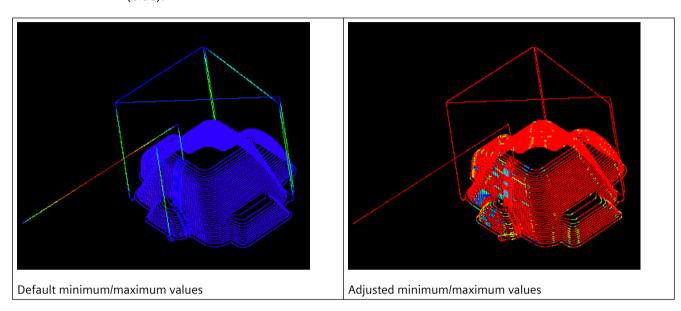5. Click on "Copy limits to all open models" to transfer selected limit values to all modules that have been opened.

Options

The following color options are available for the display:

- "Display values outside of the range in gray"; values which are outside of the min./max. values are displayed in gray.

- "Swap colors" switches the color values. The color scale is inverted (blue to red and vice versa).

- "Absolute values"; Shows the values of the selected type in absolute values. The values are color-coded.

Example

The following example shows the velocity distribution with automatically assigned minimum/maximum values (left) and manually adjusted values (right). The left-hand figure shows the areas with maximum velocity (red) during the G0 movement (red). The velocity in the workpiece is not replaced by the assigned minimum and maximum values. In the right-hand figure, the maximum value has been greatly reduced to show the areas in which the tool travels slower (blue).

| Default minimum/maximum values | Adjusted minimum/maximum values |

See also

Variables for the 4D evaluation (Page 90)

10.2 Measuring distances

10.2.1 Distance measurements

Overview

You determine the distances between points, lines and surfaces via the geometric measuring functions. This allows you to determine any irregularities in the tool path, e.g.:

- Points whose distance is too small.
- Lines whose distance varies.
- Points whose distance from a surface varies.

You can adapt the color representation of the objects, see Adjust color display (Page 53).
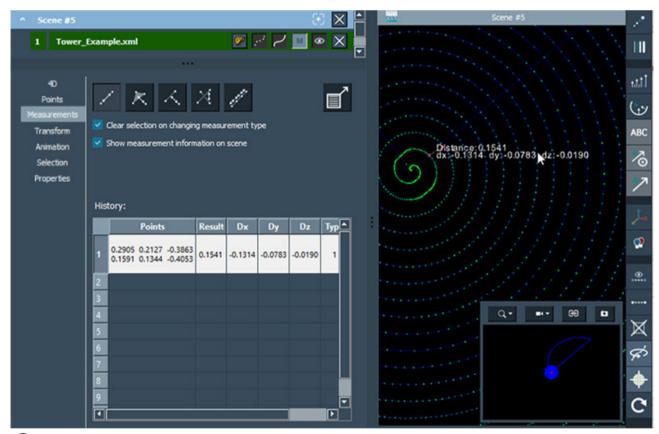
Measuring functions

The following measuring functions are available:

- Measuring the distances between points (Page 95)
- Measuring the distance between points and lines (Page 96)
- Measuring the distance of a point from a plane (Page 96)
- Measuring the distance for the points of two lines (Page 97)
- Measuring the distance between two points of a trace (Page 98)

Measurement widget

For measuring distances between points, AMWT offers the widget "Measure"

①   Buttons for the measuring functions

②   Display options for the measurements:

- Deletion of the selection upon change of the measurement method; deletes the point selection when changing the measurement method.
- Displays information on the measurements in the scene; displays the relevant information for the measurement in the scene.

③   History; the history table shows detailed information on the measurements. If you click on the lines of the measuring report table, the previous measuring calculation is performed.

Figure 10-2   Measurement widget

Call extended view

The measurement widget can be shown in the extended view. The results of the measurement can then be clearly shown.

• Click the button.

10.2.2 Measuring the distances between points

Measure the distance between two points

This function determines, for example, distances between points that are too short for the NC and so could cause performance problems.

To measure the distance between two points, proceed as follows:

1. Import the source file into the AMWT.

2. Zoom until the individual points are resolved.

3. Click the button "Measure the distance between two points".

4. Select two points.

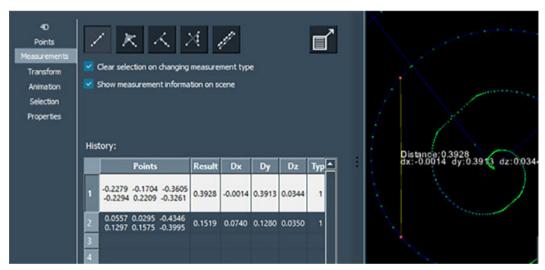5. The distance between the two points is displayed under "History" in "Results".

Figure 10-3   Measuring the point distance

10.2.3 Measuring the distance of a point from a plane

Measure the distance of a point from a plane

This function determines the distance of a point from a plane. The plane is defined by selecting three points. After selecting the fourth point, the application automatically creates a 3 or 4-point plane according to the point selection.

The original 3-point plane is extended into a 4-point plane, creating a vertical line connecting the plane to the fourth selected point.

To measure the distance, proceed as follows:

1. Import the source file into the AMWT.

2. Zoom until the individual points are resolved.

3. Click the button "Distance between a point and a plane".

4. Select the three points that define the plane.

5. Select the fourth point.

6. The plane is drawn.
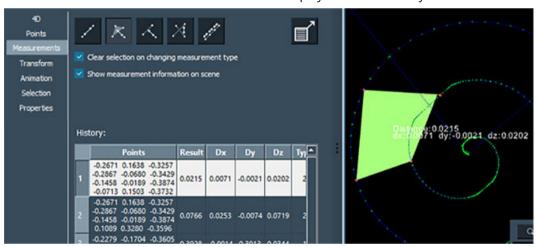   The results from the distance measurement are displayed under "History" in "Results".

Figure 10-4   Measuring the point distance to a plane

10.2.4 Measuring the distance between points and lines

Distance between line and point

This function determines the distance of a point from a line. The line is defined by selecting two points. The distance from the line is determined for the third selected point.

To measure the distance, proceed as follows:

1. Import the source file into the AMWT.

2. Zoom until the individual points are resolved.

3. Click the button "Distance between a point and a line".

4. Select the two points to define a line.

5. Select another point.
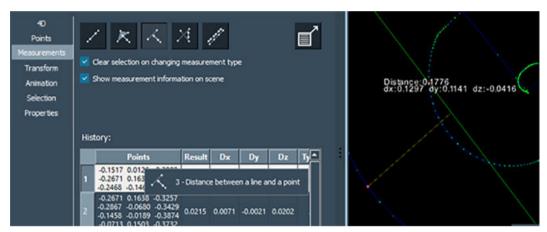   The distance between the point and the line is determined and displayed under "History" in "Results".

Figure 10-5   Measuring the distance between a point and a line

10.2.5   Measuring the distance for the points of two lines

Distance from one line to another line

The distance between two lines is measured by first defining a line by selecting two points. A second line is then defined and the perpendicular drawn from the initial line to the end point of the second line.

To measure the distance, proceed as follows:

1. Import the source file into the AMWT.

2. Zoom until the individual points are resolved.

3. Click the button "Measure the distance between two lines".

4. Select the two points that define the first line.

5. Select a third point that represents the initial point of the second line.

6. Select the fourth point.
    If necessary, the first line is lengthened until the perpendicular can be drawn to the first line.
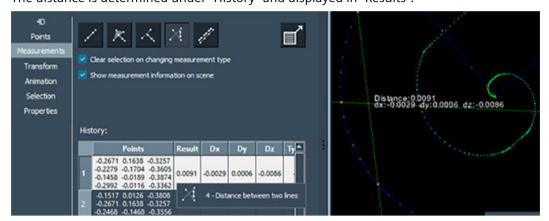   The distance is determined under "History" and displayed in "Results".

Figure 10-6   Distance between two lines

10.2.6   Measuring the distance between two points of a trace

Measure the distance between two points on the tool path

This function measures the distance between two points on the tool path

To measure the distance, proceed as follows:

1. Import the source file into the AMWT.

2. Zoom until the individual points are resolved.

3. Click the button "Measure the distance between two points on the tool path".

4. Select two points on a path that you wish to determine the distance between.

The distance between the two points is calculated by the software travelling along the tool path. If you select another point, the distance is recalculated accordingly.

Figure 10-7   Measure the point distance on the tool path

Unlike the "Measure the distances between points" function, the shortest path between two points is determined.
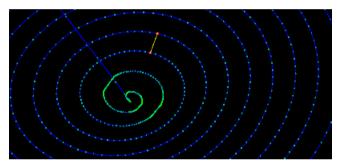

Figure 10-8   Measure the shortest distance between two points

10.3   Toolpath Extension Wizard

10.3.1   Toolpath Extension Wizard - Overview

Overview

With the analysis, signal characteristics can be mathematically analyzed using scripts. To do this, AMWT provides various predefined scripts, which can be used:

- clustering_kmeans.pyd: K-Means-Clustering algorithm - this algorithm groups data; it does this by attempting to separate samples into n groups with the same variance. This script works on several signals, which results in a model with 4D coloring.

- clustering_kmeans_plotly.py: K-Means-Clustering algorithm - for several signals (>2), supplies an HTML representation as result.

- curvature_anomaly: Curvature anomaly detector algorithm - this algorithm is an automatic anomaly detector and utilizes three different methods:
  Euler, entropy, and velocity difference
  This script works with position signals, e.g. (X,Y,Z), which result in a 4D color signal.

- signal_differences_data.py: Analyzing signal differences; using this script, two vectors (i.e. 3 signals) are compared from two trace files. A result chart (image) is generated and displayed in a web browser.

- signals_length_data.pyd: This script calculates the euclidean distance from three signals in a trace file. It is useful, for example, if you wish to analyze 'CONTROL DIFF' signals.

- signals_length_plotly.py: This script calculates the euclidean distance from three signals in a trace file. It is useful, for example, if you wish to analyze 'CONTROL DIFF' signals. A result chart (image) is generated.

- signal_differences_plotly.pyd: This script compares two vectors (i.e. three signals) in two trace files. A result chart (image) is generated and displayed in a web browser.

- surface_reconstruction_AMWSurf.pyd: Surface reconstruction from imported models. The tool database is automatically generated and assigned according to activetoolIDs. Adapt your tool data in menu "Extensions/Tool ID Database".

Result display

The results are displayed in charts, HTML plots or with a rendered surface.

Developing Python scripts

You can also develop your own Python scripts to evaluate signals. A development environment is available for this purpose.
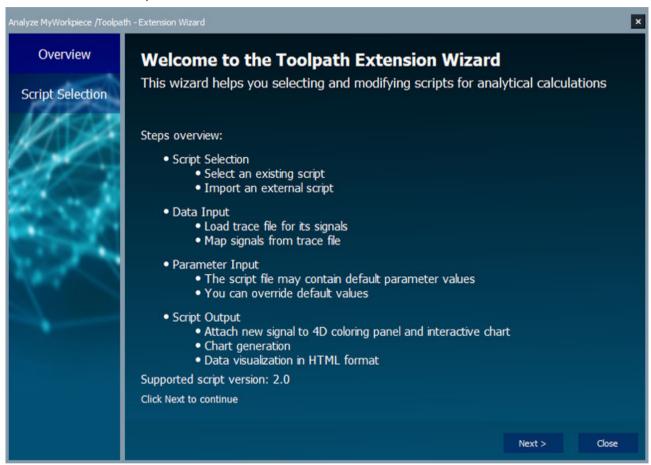
Detailed information is provided under Analyze MyWorkpiece /Toolpath - Python API Developer Guide (Page 177).

10.3.2   Running through the Toolpath Extension Wizard

Calling the wizard

You call the Toolpath Extension Wizard as follows:

- In the main menu, execute "Extension > Toolpath Extension Wizard". The wizard start page opens.

On the start page, you can see the individual steps to perform the analysis:

1. Selecting a script

2. Enter the data and assign

3. Enter parameters and assign

4. Parameterize script output

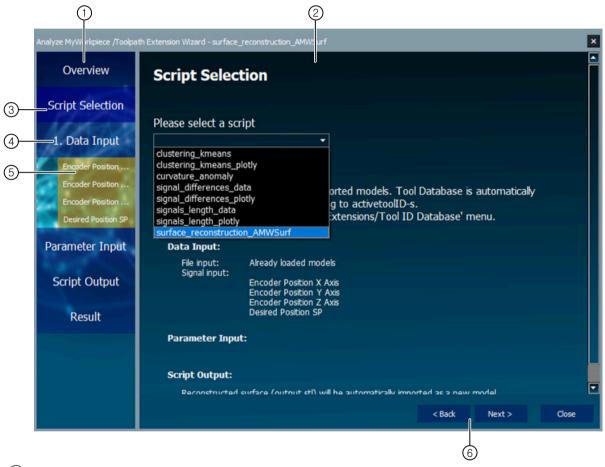The individual steps must be executed one after the other. It is not possible to skip over a step.

Errors while entering data are displayed, along with a possible countermeasure.

Note

You can also navigate through the wizard using the Enter key and the Backspace key.

Wizard structure

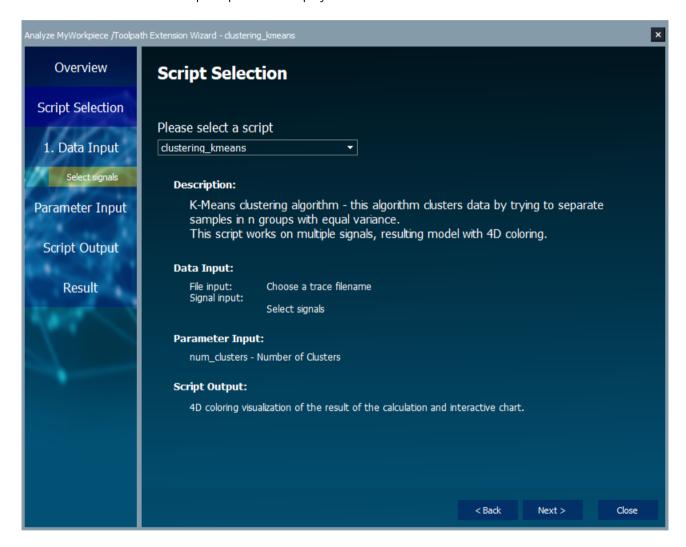The wizard is subdivided into the areas Print preview and Main view.

①	Print preview with the steps to be executed
②	Main view with information about the various steps and entering data
③	Script selection
④	Buttons (blue) to show the file selection
⑤	Buttons (orange) for additional data input, e.g. signal selection or signal configuration.
⑥	Navigation buttons

10.3.3 Selecting a script

Analytics scripts

You can find the available scripts under script selection in the drop-down list.

1. Open the wizard.

2. In the drop-down list, select the script to be evaluated.
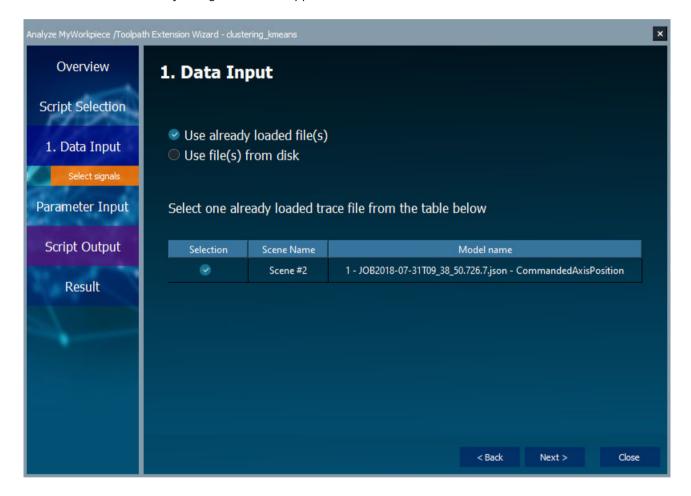   The steps required are displayed in the main view.

10.3.4 Data input

Entering data and parameters

Section "Data input" can be subdivided into two parts. The data (model data) must first be loaded and then the signals selected.
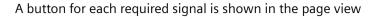
Data is required to run a Python script. This data can be selected by selecting the model already being used in the application or files, and loaded from the hard disk.
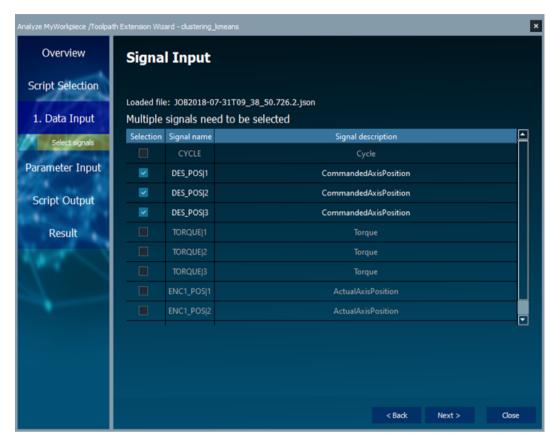

10.3.5 Selecting signals

Signal selection

The available signals are listed in the signal selection. If the Python script contains additional signal requirements, then the list is automatically filtered, and the labeling indicates the expected signal selection.
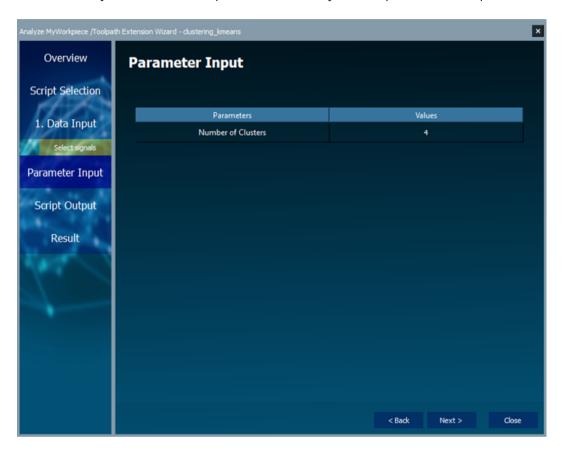
A button for each required signal is shown in the page view

1. Click on the button to show the data input elements.

2. Select the signal that you must configure. The script specifies the input.

3. Continue with the next step.

10.3.6 Entering parameters

Parameter input

You can enter your own values as parameter for the Python script in this data input window.
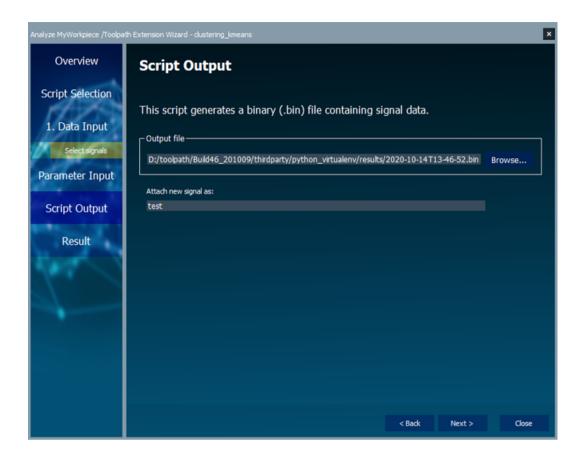

10.3.7 Script output

Output of result files

The Python script generates a *.bin, an *.HTML, or an *.STL file. This binary file contains a new signal and an interactive chart as data, which are available for the generated signal. The file is automatically loaded if the selected data file is not available after running the script. The new signal is displayed in 4D coloring.

1. Adapt the path under "Output file".

2. Under "Attach new signal as", enter a name for the signal file.

As a graph in the signal display

The new signal can be displayed as a graph together with the other model signals.
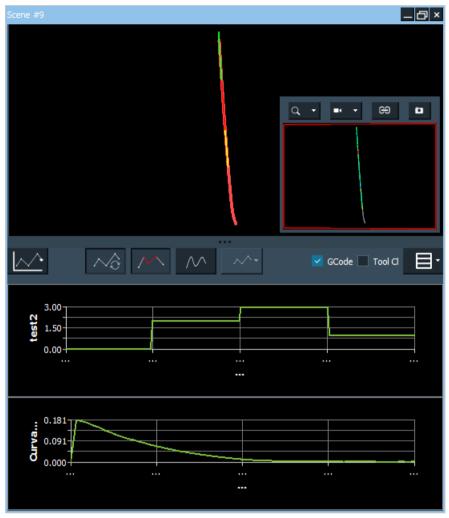

Figure 10-9   Display signal

As a signal in the 4D evaluation

The signal is displayed in the 4D evaluation after successful script execution.
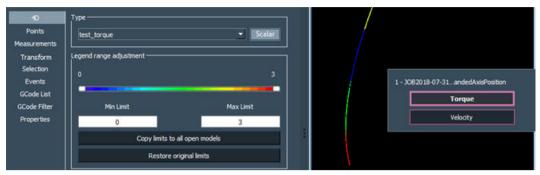

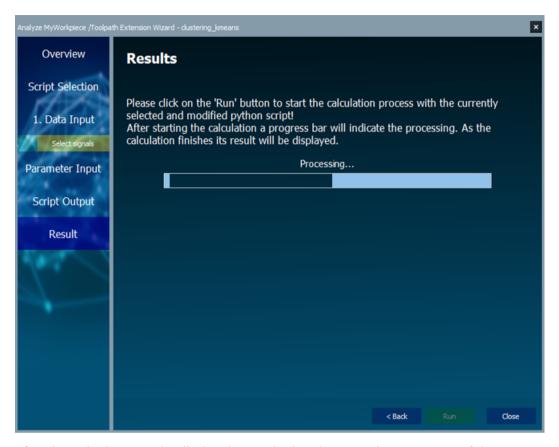Figure 10-10   Signal in 4D

10.3.8   Starting the script

Start to run the script

Start to run the script after you have entered the data.

- Click on the "Run" button.
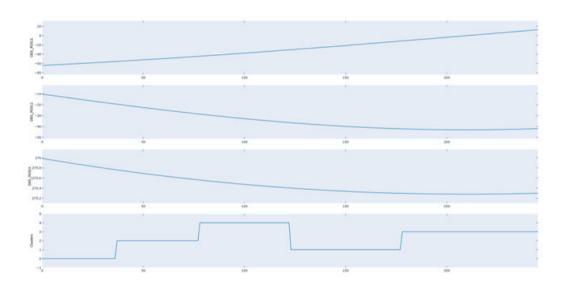 The progress of the Python script is displayed.

After the script has run, the display shows whether the execution was successful.

HTML data visualization

The script generates an HTML file, which is automatically opened in the standard browser after the script has been run successfully.

10.4 Signal evaluation

Displaying signal graphs

If the models are based on JSON, CSV or XML files, then the associated signals can be displayed for evaluation. To do this, an upward arrow which serves as a splitter is displayed in the footer of the working area. Up to five signals can be displayed.

1. Click on the splitter to show the display.
   - Or -

2. Place the mouse pointer on the splitter (upward arrow or 3 dots).
   The mouse pointer changes its appearance.

3. Move the splitter with pressed mouse button.
   The graph display becomes visible.
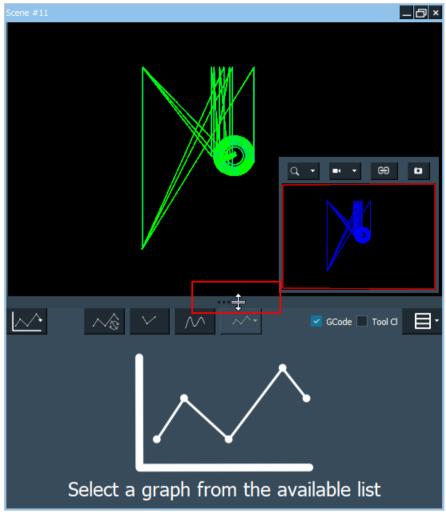
Figure 10-11   Displaying the graph

The following functions are available via the buttons:

Button	Function
	Shows a menu for selecting the model and the signals.
	Applies the zoom to all the displayed graphs.
	The points shown in the signal display are shown as "selected" in the display of the model.
	The points selected in the model are also displayed in the signal graph.
	Allows threshold values to be defined.
G-code series	Displays the G-code used in the footer of the displayed signal.

Button	Function
Tool change	Shows in the graph where a tool change took place, including the number of the new tool.
	Allows the display of multiple signals to be subdivided into tiles.

10.4.1   Selecting signals

Selecting signal below the model

The graphs can be displayed for every JSON, CSV or XML model. If multiple files are open in a scene, then they are shown in the signal display. The corresponding number is displayed in the lower left-hand corner for each model.
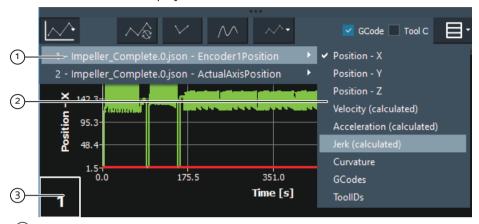
Correspondingly, only one entry is shown for one file:

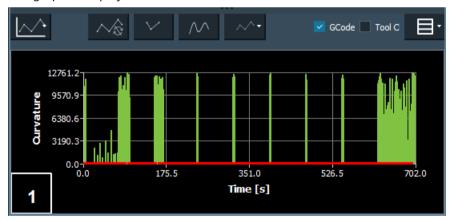1. Click the button to call the selection menu for the loaded files and signals.

The selection menu is displayed.

① Selected model
② Signal selection list
③ Number of the model in the scene

Figure 10-12   Signal selection

2. Open the submenu for signal section, and select the desired signal.
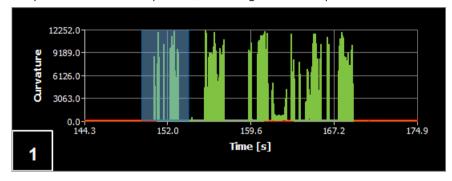   The graph is displayed.

You can display up to five signals.

10.4.2 Zooming graphs
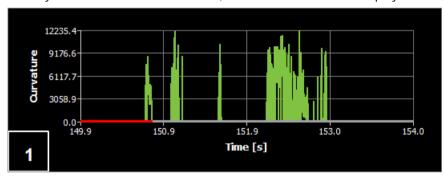
Zooming a graph area

If you would like to see an area of the signal curve in detail, you can zoom the graph.

1. Left-click in the graph.

2. Keep the mouse button pressed, and drag the mouse pointer in the desired direction.

When you release the mouse button, the selected section is displayed.

Zooming via the shortcut menu

Alternatively, you can zoom via the shortcut menu:

1. Right-click in the graph.

2. Select "Zoom in" from the shortcut menu to zoom in the graph by one stage.
   - Or -

3. Select "Zoom out" from the shortcut menu to zoom out the graph by one stage.

Restore view

You can restore the original view:

1. Right-click in the graph.

2. Select "Restore original view" from the shortcut menu.
   The graph is displayed in its original size.

Synchronizing the zoom in all graphs

If you opened multiple signals, you can synchronize the representation of the graphs. The zoom in/zoom out is transferred to the other graphs by a zoom action.

- Click the button to activate the function:

If synchronization is deactivated, then the Y is adapted so that all visible points of the graph are displayed.
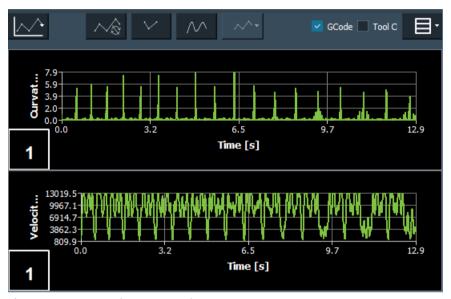
Figure 10-13   Graph not zoomed

Figure 10-14   The upper graph is zoomed, the lower graph as well

The points shown in the graph are shown as "selected"
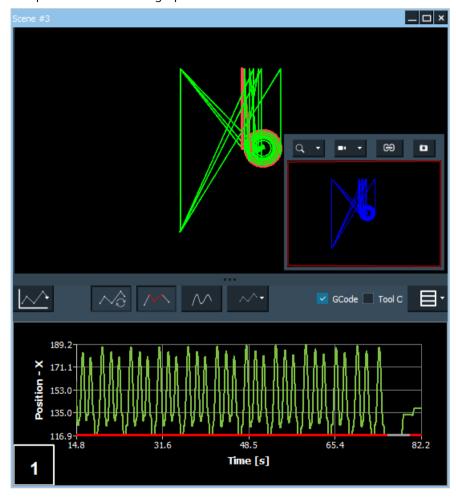
The view of the graph can also be synchronized with the display of the model. The points shown in the graph can be shown as selected in the model. This helps you to see which area of the model contains the displayed graph.

The function "Select the points shown in the graph" is preset.

1. Click the inactive button to activate the point selection. It is then no longer grayed out.

The points shown in the graphs are then shown as "selected" in the model.

Hiding points in the model that are not shown in the graph

Points that are outside the graphs are not displayed.

• Click the active button to activate the point selection. It is then no longer grayed out.

Points which are not shown in the graphs are also hidden in the model.

Figure 10-15   Hiding points in the model

Marking selected points in the graph

The point selection in the model can also be displayed on the graph shown:

*   Click the button to activate the point selection in the graph. It is then no longer grayed out.

The points selected in the model are also marked in the graph.
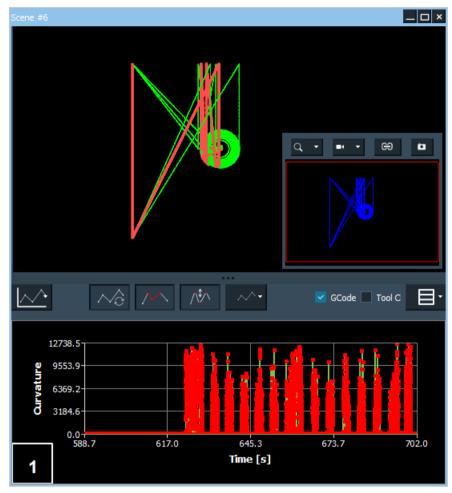
Figure 10-16   Selecting points in the model and graph

10.4.3   Setting threshold values

Areas of graphs in the signal display can be masked by threshold values. The affected parts of the graph are then no longer shown as selected.

Threshold values cannot be set for all signal types. With corresponding signals, the button then appears inactive.

Single threshold value

A single threshold value can be moved from the bottom to the top:

1. Load the signal into the display.

2. Click the "threshold value" button and select "Single threshold value" from the shortcut menu. The button displays the single threshold value.

3. The threshold value is displayed as a gray area with a line. The current threshold value appears in the display field.

4. Move the line until you have covered the area of no interest.
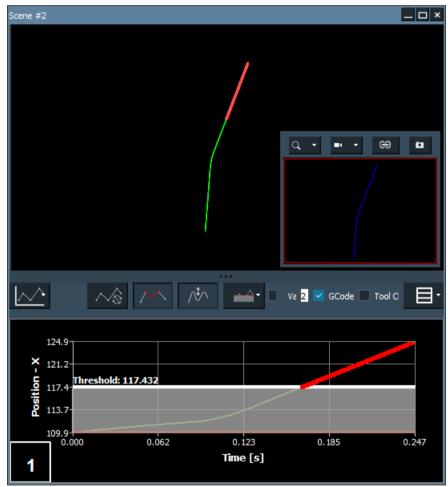
Figure 10-17   Single threshold value

Double threshold value

With a double threshold value, you can mask an area in the graph:

1. Click the "threshold value" button, and select "Double threshold value" from the shortcut menu. The button displays the double threshold value.

2. The threshold values are displayed as gray areas at the top and bottom of the display. The values are displayed in the "Upper value" and "Lower value" display fields.

3. Move the lines until you have covered the areas of no interest.
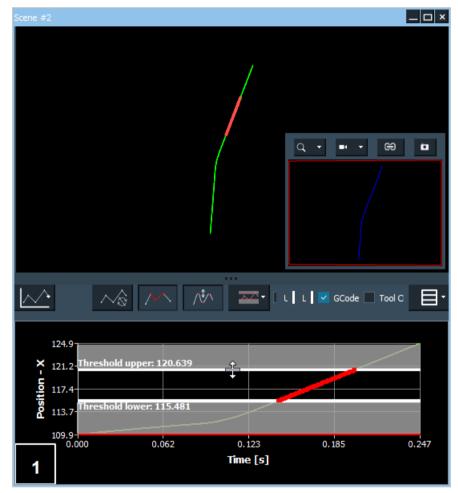
Figure 10-18   Double threshold value

Invert areas

If you want to mask the opposing areas, activate the "Inverted areas" option.

The mask is now shown inverted.

Deleting threshold values

If you want to hide the threshold values again, cancel the selection in the shortcut menu.

10.4.4 Scaling a signal display

Displaying multiple graphs
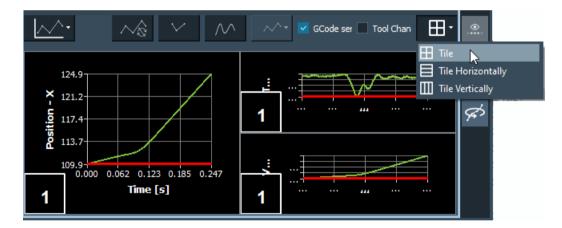
If you have loaded multiple signals into the display - up to five are possible - you can scale the display.

1. Click the button to arrange the graphs in tiles.

2. Select how you want to arrange the graphs in the displayed menu:

   – Tiles

   – Vertical only

   – Horizontal only

The representation of the graphs changes correspondingly.

Adapting colors

You can adapt the colors of the background, graphs and threshold values in each tile individually.

1. To do this, right-click to call a shortcut menu.

2. Select "Appearance" to call a submenu.

3. Select background, graph or threshold value to call a submenu for color selection.

4. Select the color in which you want to see the representation.
   The change in color is retained when the system resets to the original view.

10.5 Events widget

Structure and properties

After the data have been imported, they are displayed in Analyze MyWorkpiece/Toolpath. The recorded signals are listed in the "Events" widget.

The events are listed according to HF events and OPI (NC variables).
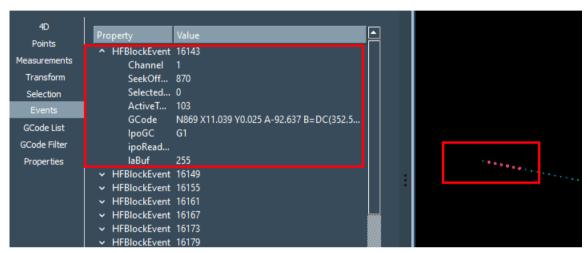
Figure 10-19   Events widget

If you click an event, the entry is opened, and the points are shown as "selected" in the model.

10.6 Property widget

Listed data

The data of the model are listed in the Properties widget. Depending on the loaded model, the data may vary:

- For MPF files: number of points, number of orientations, for example
- For XML or CSV files: channel information, for example
- For capture files: machine properties, signal properties, for example

10.7 Analyzing orientation vectors

Tool orientation for 5-axis programs

Orientation vectors can be used currently only for 5-axis programs that have defined the tool orientation via A, B and C axes. Orientation vectors can be displayed only in such programs. Such vectors describe the spatial orientation of the tool. For information on how to display orientation vectors, see Displaying orientation vectors (Page 81).

Visual assessment

Like points and curvatures, an initial visual assessment can also be made for orientation vectors. Adjusting the direction of view and adapting the display size allows any already inharmonious vectors to be detected. You can also display the tool path in an animation, see also Tool animation (Page 75).

- Are the vectors roughly parallel (provided permitted by the geometry) - ① and ②?
  If the vectors run inwards or outwards, the tool must also perform unnecessary movements.
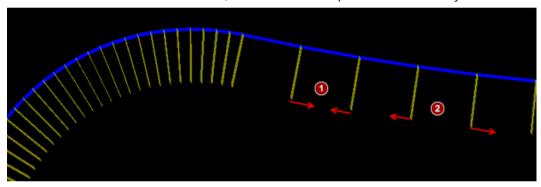
Figure 10-20   Orientation vector - Inclination

- Do the vectors lie in a plane - ③?
  If the vectors do not lie in a plane, the tool must perform unnecessary movements (③).
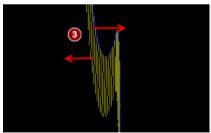
Figure 10-21   Orientation vector - Plane

10.8   Analyze points and velocities visually

Visually analyzing points and lines

Significant faults in the path and surface can be recognized in the first analysis. To do this, load the workpiece into AMWT and optimize the display until all details are well visible.

High quality data shows an overall harmonic picture:

Point distances are uniform

The points should be distributed uniformly.
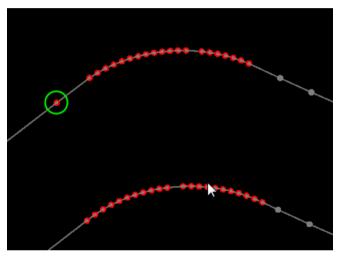
Figure 10-22   Isolated points

If areas with anomalies are apparent, further analysis steps can be performed, for example, by measuring the distances between points and lines, see also Measuring the distance between points and lines (Page 96).

You can determine the associated data in the program/trace from the information available for the point.
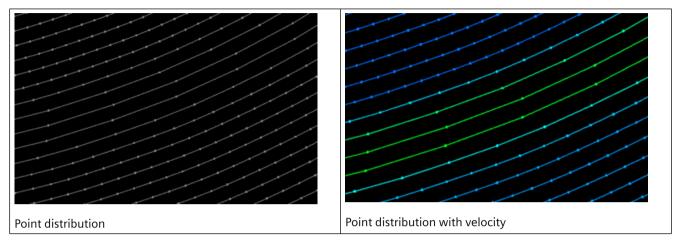
Velocity distribution is even

An even velocity distribution indicates that the tool is moving at constant velocity. The tool does not decelerate, but moves at constant velocity along the path.

If the visual check does not suffice, you can use the 4D widget for the evaluation.

The following figure shows places where points in the path are missing. The associated 4D evaluation shows a color change at these places, see also Using the fourth dimension (Page 89). To evaluate velocities, "Velocity" must be selected as the type.

Point distribution	Point distribution with velocity

Unusual velocity change (color change from green to blue) indicates that the tool is braking or accelerating.

Reconstructing/extracting a surface

<div style="text-align: right">

11

</div>
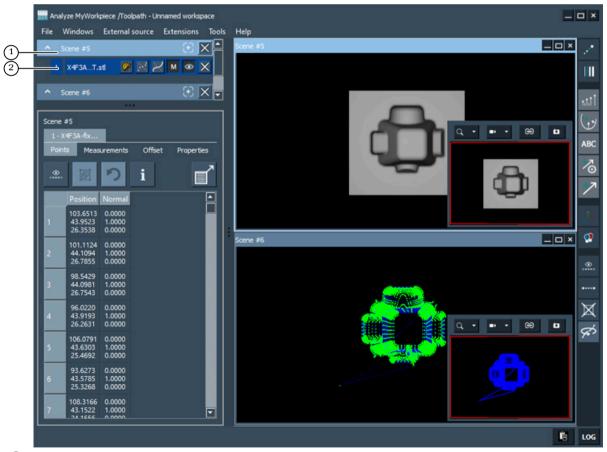
11.1 Using the mesh view

Using the mesh view

The "Mesh" function displays the surface representation contained in an *.STL file.

To display the workpiece rendered, proceed as follows:

1. Import the file into Analyze MyWorkpiece Toolpath.

2. In the sideview, click the scene containing the file.

3. The "M" button is operated automatically if you loaded an STL file.

4. Click the button for point or line representation to show or to hide these. AMWT displays the surface representation for the workpiece.

①   Scene
②   File with rendered surface elements (M)

Figure 11-1   Mesh

11.2 Reconstructing a surface

11.2.1 Surface reconstruction

Trace files can be recorded via Analyze MyWorkpiece /Capture and transferred to Analyze MyWorkpiece /Toolpath. The data from these trace files can be used to reconstruct the surface of the workpiece. If the recording took place in the machine coordinate system (MCS), Toolpath transfers the coordinates to the workpiece coordinate system (WCS).

Note

No G0 commands

No G0 commands are considered for the reconstruction.

Requirement

- The trace data contain the XYZ coordinates (3-axis machine) or the XYZ coordinates and two rotary axes for a 5-axis machine.
- Tool length compensation was used during import to account for the lengths of the different tools.
  - Axes and spindle have been defined.
- The tools used are listed in the tool identification database.

Procedure

- Import trace file, consider tool length compensation, select axis signals and spindle
- Select the "surface_reconstruction_AMWSurf" script in the extension wizard
- Select scene (the WCS scene)
- Define signals (the WCS signals)
- Select spindle
- Set reconstruction parameters
- Set file output
- Start reconstruction

Import trace file

In order to perform a surface reconstruction, the trace file must contain the necessary data (XYZ coordinates, tool identifiers or Active Tool ID, etc.)

1. Execute "File > Import file" and select the desired file.
   The import dialog opens.

2. Activate the "Tool length compensation" option.

3. Select the signals for the X, Y and Z signals (see example above).

4. Select a spindle signal.

5. Click "Import" to import the data with the appropriate settings.
   Depending on the amount of data, the import may take a long time.
   After the import, the model is displayed in the workspace.

Importing XML and CSV files

For XML and CSV files, the signal address and description of the recorded signals are also displayed:

Changing symbolic names

Symbolic signal names are displayed in the second column. You can change these names:

- To do this, click on the pencil icon and enter a new name.
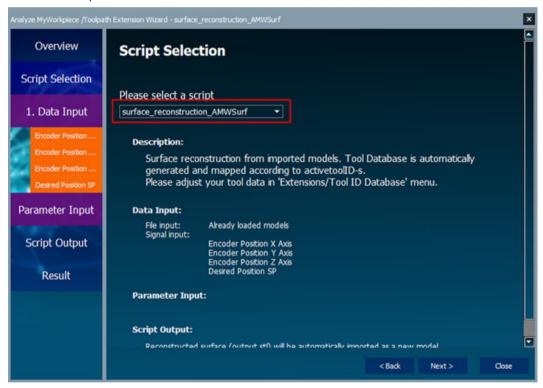
11.2.2 Running through the Toolpath Extension Wizard

Start the wizard

After you have imported the trace file, proceed as follows:
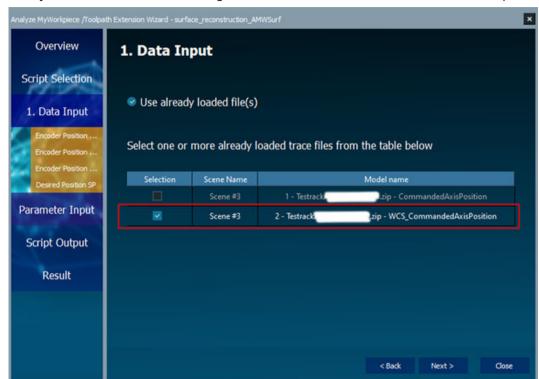
1. Execute "Extension > Toolpath Extension Wizard".
   The wizard will appear and guide you through the process step by step.

2. Select the script.

The wizard then describes the next steps.

3. Click on "Next" to call the following wizard page.

4. Here you select the scene containing the back-transformed data (WCS...) of the workpiece.

Figure 11-2   Selecting the scene
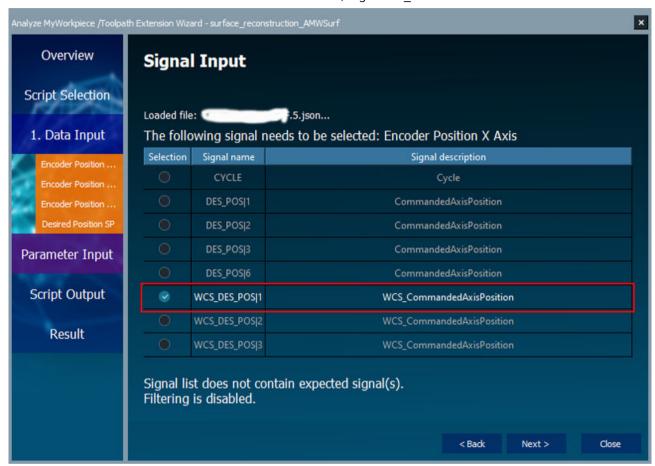
5. Click on "Next" to call the following wizard page.

11.2.3 Defining axes

Defining X, Y, and Z axes

After the back transformation, the signals of the axes are mapped with the prefix "WCS".

1. Select the axes one after the other, e.g. "WCS_CommandAxisPosition".

2. Click on "Next" in each case to call the following wizard page.

Defining the spindle

For the reconstruction it is necessary to select the spindle signal.

1. Select the signal you chose for the spindle during import.
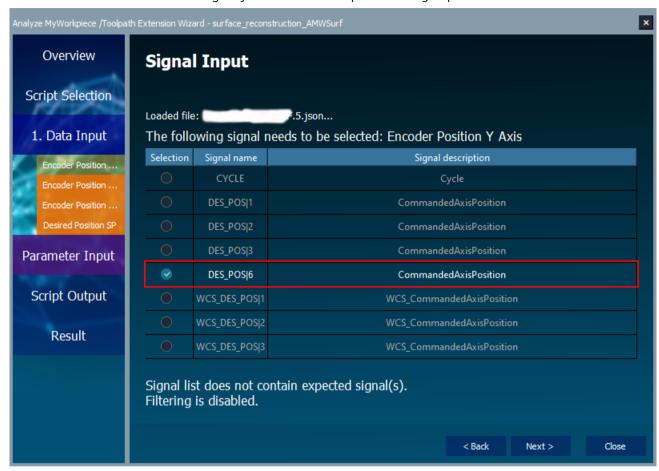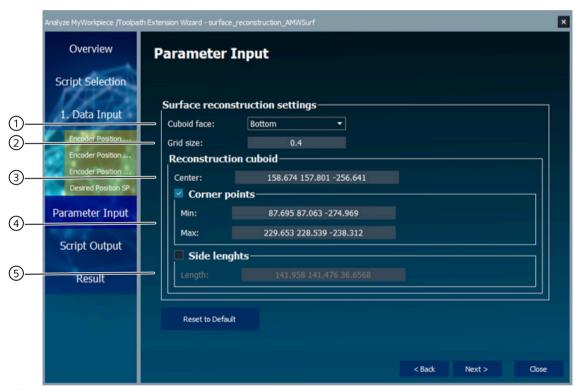
Figure 11-3   Defining the spindle

2. Click on "Next" to call the following wizard page.

11.2.4 Set reconstruction parameters

Entering parameters

With different parameters you define the volume section, the accuracy and the camera view of the reconstruction. For the reconstruction, a cuboid auxiliary body (bounding box) is defined, which defines the reconstructed volume. Only points that lie within the cuboid are used for reconstruction.

①	Camera view of the auxiliary object (bounding box)
②	Grid size
③	Center of the bounding box
④	Corner points of the bounding box
⑤	Side length of the bounding box

Cuboid area and grid size

Use the "Cuboid view" to select the perspective from which you are looking at the workpiece.

• Select the appropriate perspective in the drop-down list.

With the grid size you define the precision of the reconstruction. The smaller the value, the tighter the grid, the more accurate the reconstruction, but the greater the computation time.
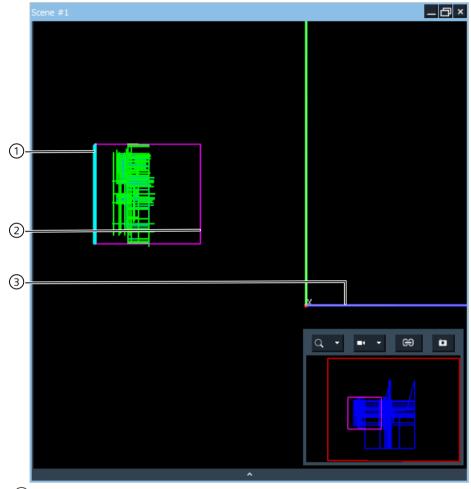
• Enter a value for the grid size. The default setting is 0.4.

Define the volume of the cuboid (bounding box)

The bounding box is calculated from the minimum or maximum TCP data in the workpiece coordinate system (WCS) after the G0 data has been removed. The center is the center of gravity of the bounding box.

The edges of the cuboid are displayed in color, with the selected cuboid view highlighted in blue. The bounding box can be rotated and zoomed together with the workpiece.

When you open the "Parameter Input" dialog, the bounding box will be displayed.

①   Minimum value in Z direction (values < 0)
②   Maximum value in Z direction
③   Z axis in the coordinate system

Figure 11-4   Example of a workpiece with bounding box (camera view from behind)

The individual points can be used to position the box in space and to change its size. Negative values are possible.

The input fields each contain the values for the X, Y and Z directions (in that order).

- Center: By changing the X, Y and Z coordinates, the center of the cuboid can be moved in space. The position of the cuboid to the workpiece can be checked in the navigation window.

  - Center: XY Z

- Corners: The corners define the side faces of the cuboid. By entering the X, Y and Z values, the corners and thus the side faces of the cuboid can be moved by entering minimum or maximum values.

  - Min.: X Y Z

  - Max.: X Y Z

  This causes an enlargement/reduction as well as a shift of the cuboid in space.

- Side length: Select the "Side lengths" option to adjust the sides of the cuboid in the X, Y, and Z directions respectively. The position of the cuboid in space remains unchanged.

  - Length: X Y Z

Restricting the point cloud

If you want to reconstruct only a certain section of the workpiece, you can select the point cloud of interest before the reconstruction and hide all other points in the workspace using the "Show selected points only" option. This makes it possible to reconstruct smaller areas with higher resolution without losing the overview.

11.2.5 Performing reconstruction

Defining the script output

When you have entered all the parameters of the surface reconstruction, you can start the reconstruction with the last two steps in the wizard:
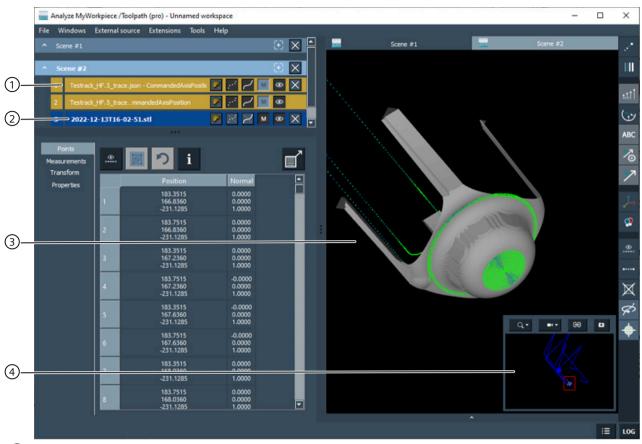
1. Click on "Browse" to specify the storage location and name of the output file.
   Alternatively, you can use the default path and file name.

2. Click on "Next" to call the "Results" dialog.

Start reconstruction

- Click on "Execute" to start the script execution.
  The progress display shows the current status of reconstruction.

As soon as the reconstruction is finished, the result parameters will be displayed. Some background processes are still running to display the reconstructed surface.

The reconstructed workpiece is then displayed in the tool area, and the associated *.STL file is displayed in the scene.

① Source file
② Result file
③ Reconstructed model
④ Navigation window with model

After finishing reconstruction, the wizard can be run backwards by pressing "Back". The reconstruction can be started again with changed parameters.

Error

If an error occurs, processing of the script is aborted. Possible causes could be:

- There is no active tool in the tool database. The error message shows the missing tool with the number. Enter the tool in the database and start the script again

- The spindle was not selected correctly.

- The correct signals were not selected for reconstruction.

11.2.6 Assigning a tool

Tool IDs for surface reconstruction

For surface reconstruction using a script, you require the data of the tools used. Tool data are managed in the Tool ID Database.

You can use the Tool ID Database Editor to assign tools to an active ID and create tools that are not available in the collection. For the definition, you require the tools and their current data that are used in the machine.

The active tool ID is then used for surface reconstruction.

Tool collection and tool assignment

In the tool collection, tools can be created with all the necessary data. In the tool assignment, they are then assigned to an active ID.

If no tool has been assigned to an active ID, an error message is displayed after the script has been processed. The message lists the IDs that are not assigned.

Adding a new tool

In order to add a new tool, proceed as follows:

1. Execute "Extension > Tool ID Database".
   The editor of the Tool ID Database is displayed.

2. Click the tool collection into the foreground.

3. Click on "Add New".
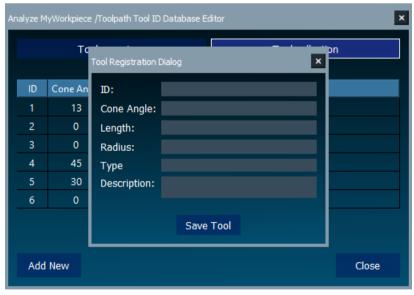
Figure 11-5   Creating a new tool

4. Enter the tool data and click on "Save Tool".

   – ID: number for identifying the tool. To avoid confusion, it is recommended to use the Active Tool ID here (see below).

   – Cone Angle: characteristic of the tool used

   – Length: characteristic of the tool used

   – Radius: characteristic of the tool used

   – Type: type number of the tool (tool list in the tool management in SINUMERIK Operate)

   – Description: text used to describe the tool in more detail

5. The tool is created using the entered data.

6. If you want to revise the data, right-click on the entry and execute "Edit" in the displayed dialog.

7. To delete the tool, right-click on the entry and execute "Remove" in the displayed dialog.

Registering a tool

To assign a tool to an active ID, proceed as follows:

1. Execute "Extension > Tool ID Database".
   The Tool ID Database Editor is displayed.

2. Click tool assignment into the foreground.

3. Click on "Add New".
   The "Tool Mapping" dialog opens.
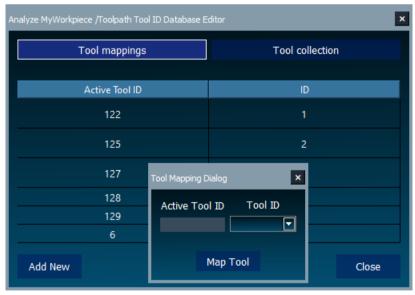
Figure 11-6   Assigning a tool

4. Enter an active tool ID.

5. Under "Tool ID", select the ID corresponding to the tool. You can assign the ID during tool registration.

6. Click on "Map Tool".

7. The tool then appears in the list of assigned tools.

Determining the Active Tool ID

You determine the Active Tool ID required for the tool identifier in the Events widget for the loaded model.

1. To do this, open the Events widget.

2. Open an HFBlockEvent.
The parameters of the event are listed. You find the value under "Active Tool".
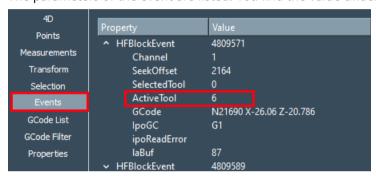
Figure 11-7   Active Tool ID

11.3   Surface extraction

Extraction

Surfaces can be extracted from point clouds using surface extraction. This can be used, for example, to check the surface quality.

The extraction is a computationally intensive process and can be parameterized in the following dialog.
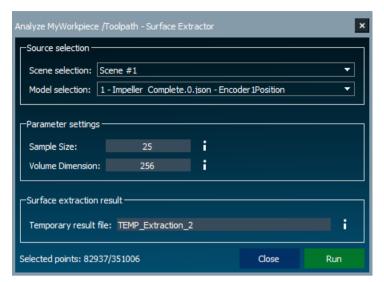
Figure 11-8   Surface extraction

Selecting points

1. Select the points in the model for which you want to perform a surface extraction.

Scene selection

1. Select the scene that contains the model.

2. Select the model with which you want to perform the surface extraction. Extraction only works for models that contain usable points/data. Accordingly, a message is displayed in the footer of the dialog.
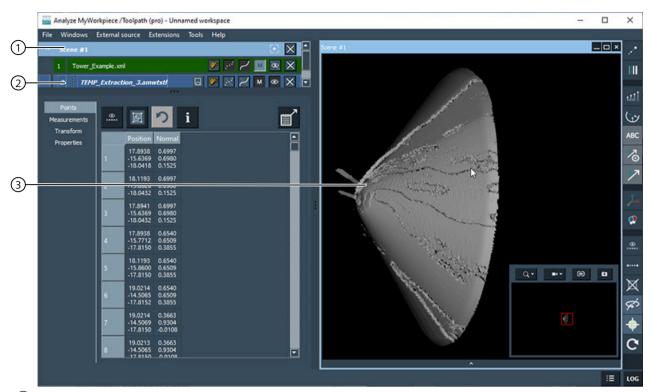
Parameter settings

1. In "Sample size", enter the number of points to be considered for the surface extraction of a point. These are the points in the immediate vicinity that are used to calculate the normals. The larger the value, the more accurate the extraction, but the more computationally intensive the execution.

2. Enter a value in "Volume dimension". Use this parameter to specify the i, j, and k dimensions to be applied to the distance function for surface extraction.

Surface extraction result

• Enter a name for the file under which the surface extraction is to be stored. The file is displayed in the scene and must be explicitly saved, otherwise the data will be lost when the scene is closed.

① Model file used for the extraction.

② Temporarily generated surface extraction that can be saved.

③ Extracted model

Figure 11-9   Extracted surface

Machine simulation

<div style="text-align: right; font-size: 3em;">12</div>

Simulating workpiece machining

The machine simulation simulates the tool movements required for the workpiece on the volumetric model. The workpiece is machined step by step.

For this purpose, the model is first imported and positioned.

Possible data types are:

- JSON
- XML
- CSV

Tools

The following tools are supported:
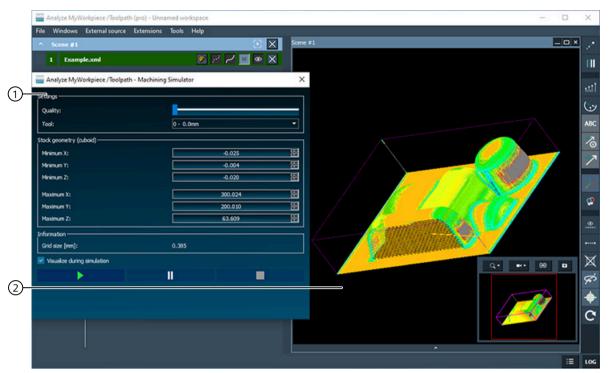
Tool type	Description
110	Ball nose end mill
120	End mill

Machine simulation

The machine simulation is controlled via the "Machine simulator" dialog:

1. To start the machine simulation, run "Extensions > Machine simulation".

2. The "Machine simulator" dialog appears.

Overview

①   Dialog for controlling the simulation
②   Display area of the model

Figure 12-1   Machine simulation - Overview

12.1 Performing a simulation

Defining simulation parameters

To perform a machine simulation, you need to set the following parameters:

① Set quality (4 levels)
② Select tool
③ X, Y, Z - minimum values for the bounding box
④ X, Y, Z - maximum values for the bounding box
⑤ Display of the grid size (voxel)
⑥ Switch off visualization of the simulation
⑦ Operate simulation

Figure 12-2   Machine simulation - Dialog

Quality of the simulated workpiece

The quality of the simulation can be adjusted via a slider, which thus determines the size of the volume units (voxels). Up to four quality levels are possible. The higher the set quality, the more complex the simulation calculation and the longer it takes to simulate the workpiece.

The voxel size is displayed under "Information".

Tools
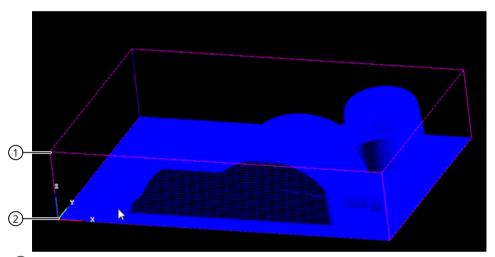
The tools are displayed in the "Tools" drop-down list:

1. Open the drop-down list.

2. Then select your tool.

Defining a bounding box

The bounding box defines the volume from which the workpiece is machined. The larger the box, the longer the simulation will take.

If only parts of the workpiece are of interest, the rest can be removed using the bounding box.

①   Boundaries of the bounding box

②   Origin of the workpiece coordinate system (0 point)

Figure 12-3   Bounding box

Changing the size of the bounding box

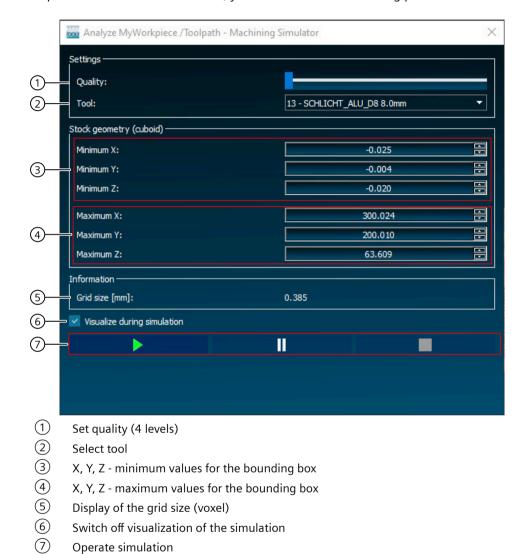1. In the "Machine simulator editor" dialog, enter the new values for the bounding box in the respective fields.
   - OR -

2. Use the arrow keys on the right edge of the input field.

   – When entering minimum values, the X, Y and Z values become smaller or negative with respect to the origin. The boundaries of the box are moved down (-Z), forward (-Y), or to the left (-X).

   – When entering maximum values, the X, Y and Z values become larger with respect to the origin. The boundaries are moved up (Z), backward (Y), or to the right (X).

Using the 4D function

If you made settings for the model in the 4D viewlet, e.g. the velocity distribution is to be displayed in color, then these settings (colors) are also displayed in the simulated model.

Starting simulation

Once all parameters have been entered, the simulation can be started.

Note

Disable the option "Visualize during simulation" to save computing time.

1. Click on the "Start" button to start the simulation.
   The progress of simulation is displayed in a progress bar and can be tracked in the working area.
   - OR -

2. Click on the "Pause" button to pause the simulation.
   - OR -

3. Click on the "Stop" button to stop the simulation.
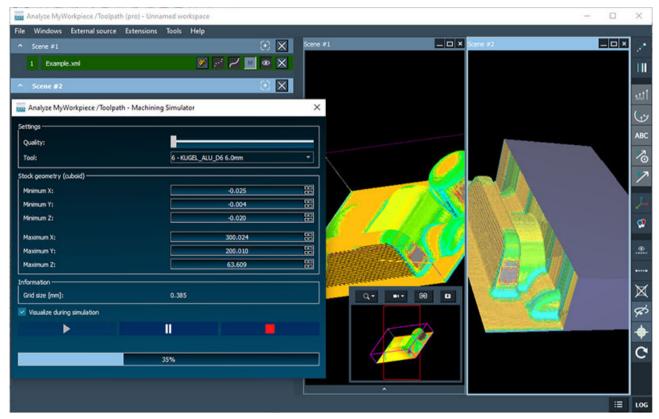
Figure 12-4   Performing a simulation

After the end of the simulation, the model can be moved, enlarged, or reduced.

Note

It is not possible to save the simulation.

Displaying log files

13

13.1 Displaying a log file
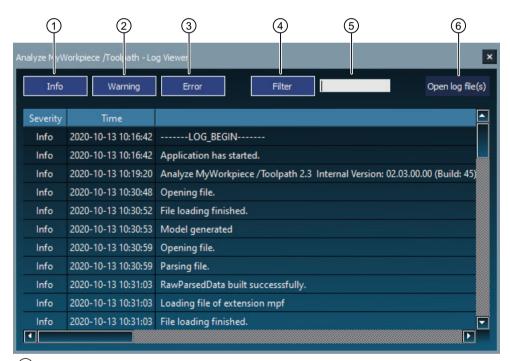
Log viewer

The log viewer displays messages relating to events in Analyze MyWorkpiece /Toolpath. The messages can be filtered and examined.

If messages are pending, the button in the footer starts flashing.

LOG Click the button to open the log viewer.

The log viewer includes a toolbar with functions for filtering and examining messages. With multiple selections, the active filters are evaluated.

①	Shows all information.
②	Shows all alarms.
③	Shows all errors
④	Activates the text box for full text search
⑤	Text box for search term The display is immediately refreshed.
⑥	Opens the log file or allows log files to open.

In the table, the messages are specified by additional data, such as time, scene, model or file.

Importing data from Analyze MyWorkpiece / Capture

14

14.1 Users and certificates

14.1.1 Creating a new user

The next steps are only necessary if one of the "amwtoolpath" users assigned to the "amwtoolpath" group is not yet present on this device.

Requirement
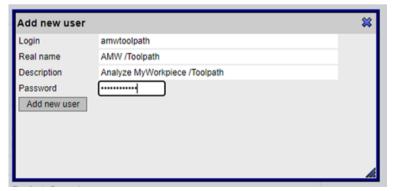
You have already installed an Analyze MyWorkpiece /Capture client certificate. A description of this can be found in the "Analyze MyWorkpiece /Capture" Operating Manual.

Adding a user

Execute the following steps to create a user:

1. Open the Miniweb administrator dialog in your web browser.
   At `https://<ip_or_dnsname_of_your_edge_box>:5443`.
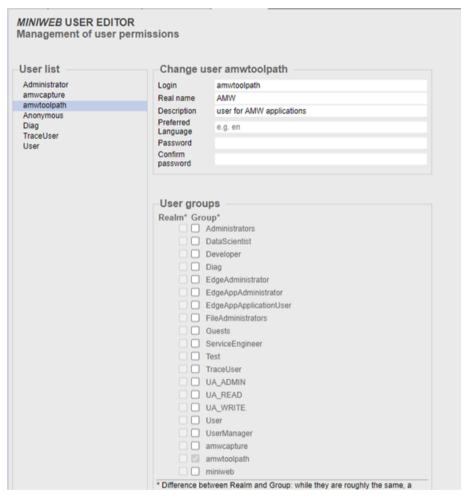
2. Log in as administrator.

3. Navigate to "Security>User Manager".
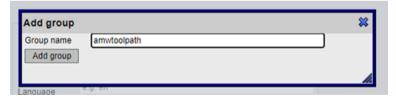
4. Click "Add new user".

5. Enter the following data:

   – Login: amwtoolpath

   – Real name: optional, i.e. "AMW /Toolpath"

   – Description: optional, i.e. "Analyze MyWorkpiece/Toolpath"

   – Enter a password.
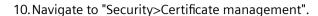
6. Click "Add new user" to create the new user.

7. Select the newly created user "amwtoolpath" from the "User list", and click the "Add group" button.
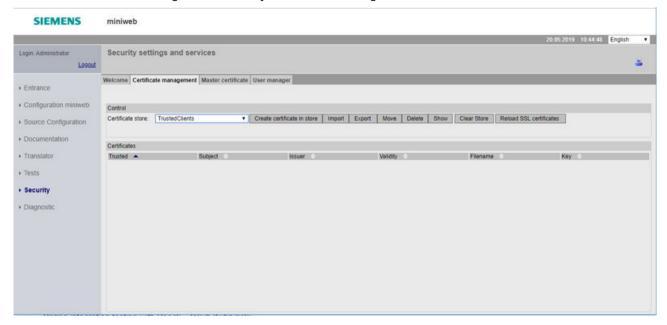
8. Enter "amwtoolpath" as the group name. The entry is case-sensitive.

9. Click "Add group".

10. Navigate to "Security>Certificate management".
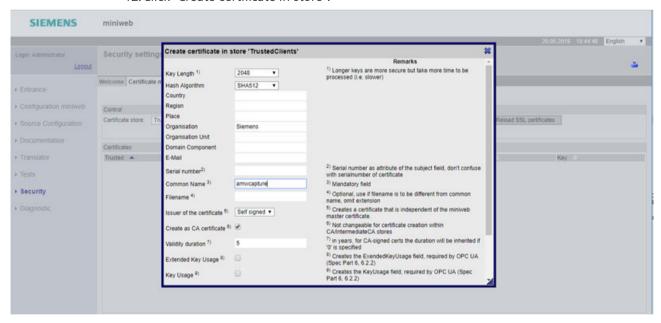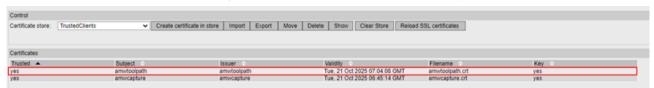
11. Select "TrustedClients" from the "Certificate store" drop-down list.

12. Click "Create certificate in store".
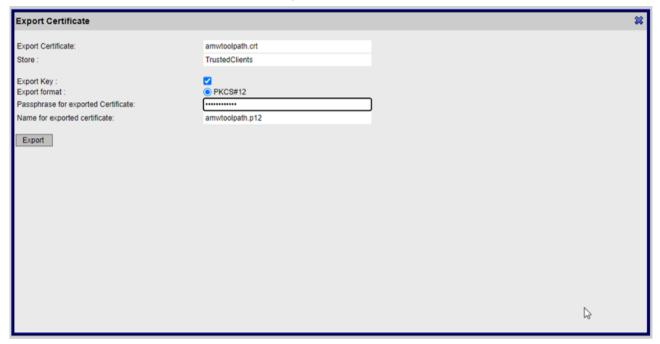
13. Enter "amwtoolpath" in the "Common name" field, scroll down and click "Create certificate in store". The certificate is created and the dialog is closed.

14. Select the newly created client certificate, and click "Export".

15. Ensure that the correct key is selected.

16. Enter a password for the export, and click the "Export" button.

14.1.2   Installing a client certificate

When installing the client certificate on your computer, the steps may differ depending on the operating system being used. The following steps have been tested with Windows 10:

Procedure

1. Right-click the downloaded certificate "amwtoolpath.p12".

2. Click "Install certificate" or "Install PFX".

3. Click "Current user > Next".

4. Click "Next".

5. Enter the password that was entered while generating the client certificate in the "Password:" field.

6. Click "Next".

7. Click "Place all certificates in the following store".

8. Select the "Personal" folder and click "OK".

9. Click "Finish".
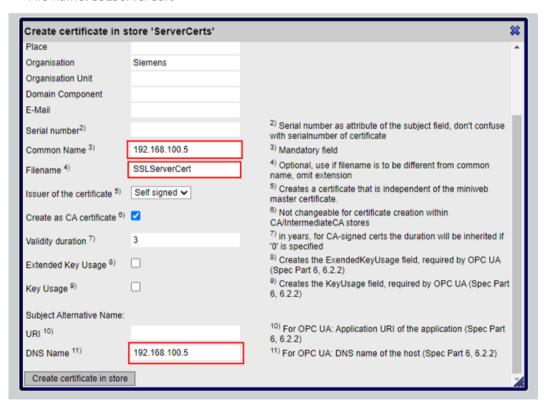
14.1.3 Creating a server certificate

The following steps may vary depending on the IT infrastructure. In this manual, it is assumed that the SINUMERIK Edge device can be accessed via a static IP address, and the server certificate is based on a self-signed certificate.

Procedure

1. Navigate to "Security>Certificate management".

2. Select "ServerCerts" from the "Certificate store" drop-down list.
   If the stated certificate shows the IP address of the SINUMERIK Edge device under "Subject" and "Issuer", continue with step 5. Otherwise, select the listed certificate, and remove it by clicking "Delete".

3. Click "Create certificate in store", and enter the following data:
   – Common name: Static IP address of this SINUMERIK Edge device
   – DNS name: Static IP address of this SINUMERIK Edge device
   – File name: SSLServerCert

4. Click "Create certificate in store".
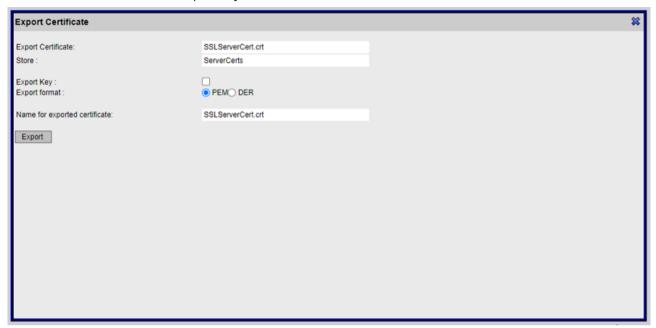
5. Select the certificate from the list, and click "Export".

6. Click "Export" again.
Note: "Export key" must not be selected.

7. Restart your SINUMERIK Edge.

14.1.4 Installing a certificate

Installing a server certificate

When installing the server certificate on your computer, the steps may differ depending on the operating system being used. The following steps have been tested with Windows 10:

1. Right-click the downloaded certificate "SSLServerCert.crt".

2. Click "Install certificate".

3. Click "Current user".

4. Click "Next".

5. Click "Place all certificates in the following store".

6. Select the "Trusted Root Certification Authorities" folder and click "OK".

7. Click "Finish".

8. Restart the program if Analyze MyWorkpiece /Toolpath was active during the installation.

Installing a client certificate

1. Install the client authentication certificate (amwtoolpath.p12) in the dialog "Manage user certificates" (type "certmgr.msc" in the Windows search) in the personal memory area of the PC.

2. Go through the steps of the installation wizard.
   A user-defined password is required for the client authentication certificate.

This is how to install the 'SIEMENS miniweb' certificate type `https://<url>:5444/capture` in Internet Explorer:

1. At the end of the address bar, click on the icon "Safety report" (padlock), and in the pop-up dialog on the button "Show certificates".

2. Click "Install certificate..." in the newly opened "Certificate" dialog, and go through the steps of the installation wizard.
   The certificate must be installed in the "Trusted Root Certification Authorities".

To install the 'SIEMENS miniweb' certificate, proceed as follows:

1. Enter https://<url>:5444/capture as address in the browser.

2. At the end of the address bar, click on the icon "Safety report" (padlock), and in the pop-up dialog on the button "Show certificates".

3. Click on "Install certificate..." in the newly opened "certificate" window and go through the steps of the installation wizard.
   The certificate must be installed in the "Trusted Root Certification Authorities".

SINUMERIK Edge can be configured such that an alias is used instead of the IP address. In this case, the alias must be defined in the file "%system32%-drivers-etc-hosts".

14.2 Establishing a connection to Edge

After recording data with Analyze MyWorkpiece /Capture, you can import them to Analyze MyWorkpiece Toolpath for evaluation.

When using the "External source > Import captured file(s) from Edge" function, it is assumed that the necessary MiniWeb HTTPS certificate is installed in the Trusted Root Certificate Authorities, and the client certificate is installed in the User Certificate Store.

Note

Files in Analyze MyWorkpiece /Capture
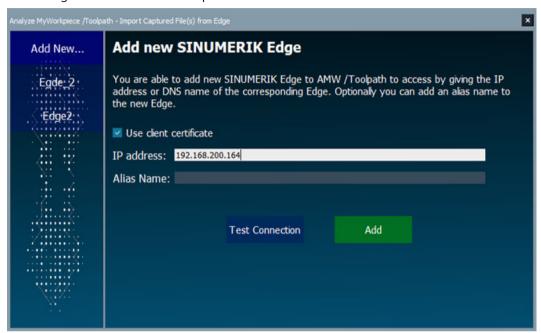
The .zip files that were downloaded remain in the %App% directories of the user and are not deleted. If you download jobs with the same names, then the files are appropriately numbered, e.g. job_(1).zip, job_(2).zip... The data can be managed in the file manager.

Establishing the connection

To start the import, a connection must first be established to a SINUMERIK Edge:

1. Execute "External source > Import captured file(s) from Edge" in Analyze MyWorkpiece / Toolpath.
   The dialog with the same name opens.

2. Click on "Use client certificate" to use the certificate you created as described in Users and certificates (Page 151).

3. Click on "Add New..." to add an Edge connection.
   Analyze MyWorkpiece /Toolpath automatically searches for the appropriate version.

4. Enter the IP address of SINUMERIK Edge.

5. Assign an alias name, under which the Edge connection is shown in the sideview. The IP address can also be assigned as an alias name.
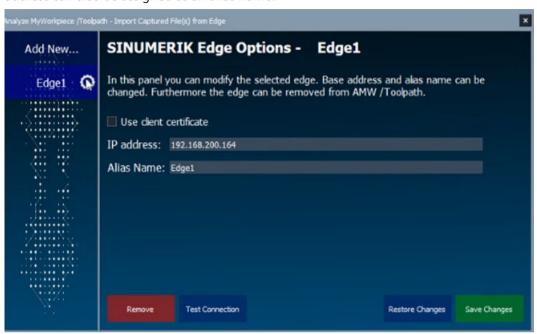
Figure 14-1   Starting the import of SINUMERIK Edge

If certificate installation or IP address aliasing fails, an error message is shown in the "External source > Import captured file(s) from Edge" window.

6. Click "Test Connection" to check the validity of the inputs and establish a test connection.

7. When you log in to Edge for the first time, you will be prompted for the username and password you assigned.

8. If you are already logged into Analyze MyWorkpiece /Capture, the username and password are already entered.

Editing connections

Once you have established a connection, the alias names of all connections are displayed in the sideview. Proceed as follows to edit a connection:

1. Click on the entry in the sideview and run "Options".

2. The following options are shown:

   – "Remove"; removes the data that has been entered.

   – "Restore Changes"; redisplays the data that has been saved.

   – "Save Changes"; saves the changes.

   – "Test Connection"; checks the connection data via a test connection.

14.3 Importing data

Loading the job to Analyze MyWorkpiece /Toolpath

As soon as a connection has been established to Edge, job data is transferred out of SINUMERIK Edge. A progress bar displays the transfer status. The list of jobs is displayed once all data has been transferred.

1. Select the required job.
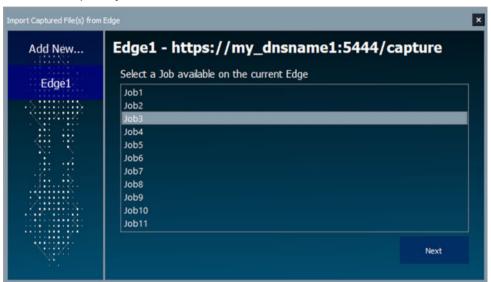
Figure 14-2   Loading the job list

2. Click on "Next" to display the content of the job.
   The jobs are displayed according to ascending recording time; the latest is at the beginning.
   The recording time is displayed at the end of the file name with date and time.
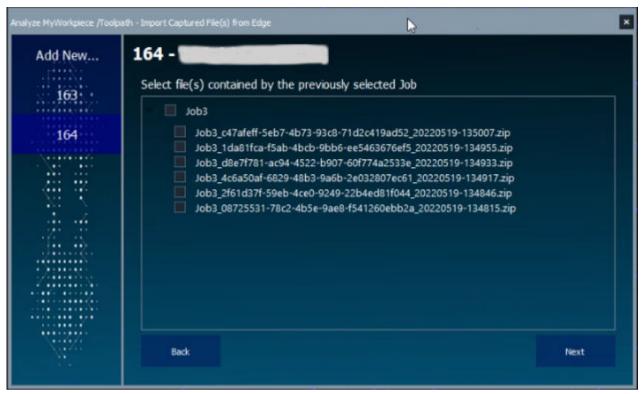
Figure 14-3   Job content

3. Select the files that you wish to import.

4. Click "Next".
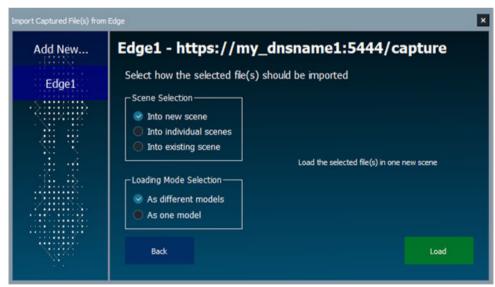
Figure 14-4   Selecting scene and module

5. Select in which scene the file should be loaded:

   – Into a new scene; AMWT creates a new scene, and imports the file into this scene

   – Into individual scenes; the files are each loaded into a scene.

   – Into an existing scene; the file is loaded into an existing scene.

6. Select how the file is to be imported into models:

   – As different models; the file is split up into different models.

   – As one model; the file is imported into one model.

7. Click "Load".
    If you have selected existing scenes, then you can select into which scene the file is loaded.
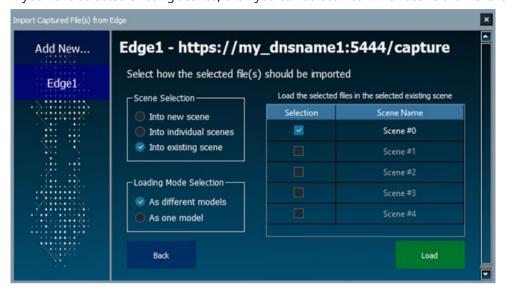
Figure 14-5   Selecting an existing scene

8. Select the scene and click on "Load".
    The data is automatically loaded and saved without a password prompt. The password is generated automatically and is decoded by Analyze MyWorkpiece /Toolpath.
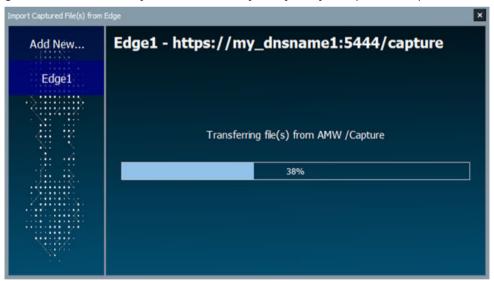
Figure 14-6   Importing a file from AMW /Capture

14.4 Managing stored data

Transferring data from the Edge

> The data transferred from the Edge to Analyze MyWorkpiece /Toolpath can only be opened there. The imported ZIP files are password-protected to prevent unauthorized access by other applications.
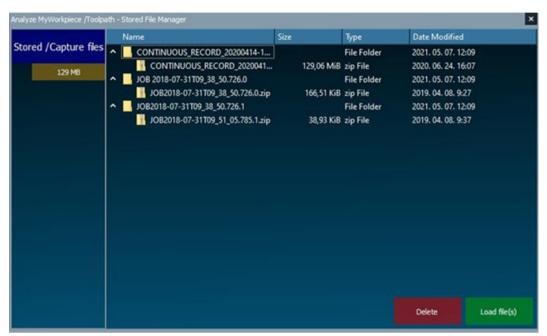
Figure 14-7   Manage stored data

You can use the dialog to load files and delete saved files.

The size of the file is displayed in the left column.

Changing the password

You change the password in the dialog of the same name.

Figure 14-8   Changing the password

Connecting Teamcenter

15

15.1 Teamcenter

By connecting to Teamcenter, you can access the data archived there. AMWT establishes a connection to Teamcenter.

Establishing a connection to Teamcenter

To connect to Teamcenter, you need the following information:

- IP address
- Port
- User name
- Password

To log on to Teamcenter via AMWT, proceed as follows:

1. Execute "External source > Teamcenter".
   The "Analyze MyWorkpiece /Toolpath - TC" dialog opens.

2. Enter the following data:
   - IP address: Enter the IP address of the Teamcenter server here.
   - Port: Enter the port number here.
   - User name: Enter your user name in Teamcenter here.
   - Password: Enter your Teamcenter password here.

3. Click "Login".
   AMWT establishes a connection to Teamcenter.

Troubleshooting

16

16.1 Problems with the graphics card

Error description

Error occurs immediately after installation (during normal startup and when starting as administrator).

Figure 16-1 Error image 1
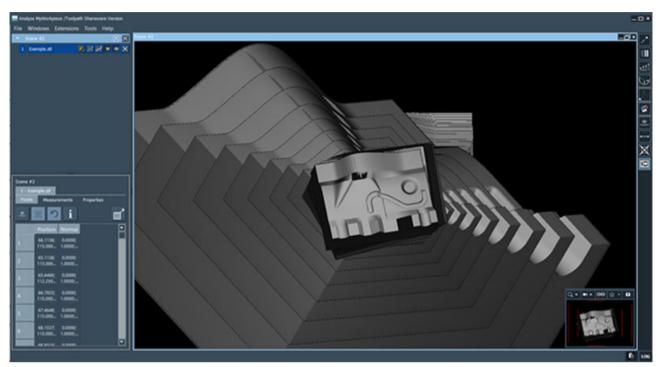
Figure 16-2   Error image 2

The error occurs when zooming and moving the object (regardless of the file type stl, xml, mpf).

You have a computer with two graphics cards, as the following example shows:
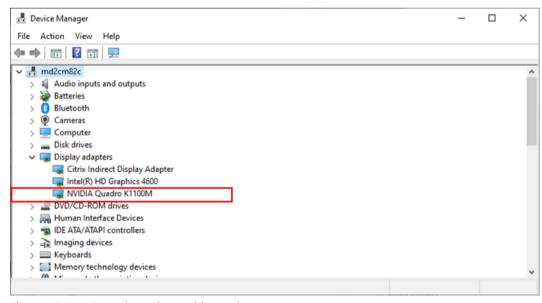
Figure 16-3   Control panel - graphics card

Troubleshooting

You can solve the problem by booting the system with the NVIDIA graphics chip:

1. Open the control panel of the graphics card by clicking on the icon of the graphics card in the taskbar, for example.
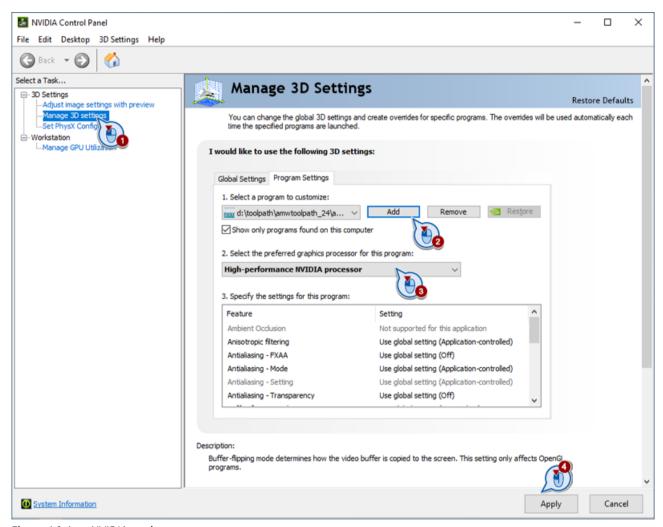
Figure 16-4   NVIDIA setting

1. Select "Manage 3D settings".

2. Under the "Program settings" tab, add Analyze MyWorkpiece /Toolpath as the program for which you want to use the 3D settings.

3. Select "High-performance NVIDIA processor" as the graphics processor.

4. Click "Apply".

Figure 16-5   Error resolved

Alternatively, you can also start Analyze MyWorkpiece /Toolpath via the shortcut menu (right-click on the exe file) and select the high-performance processor there. However, this setting is not saved.
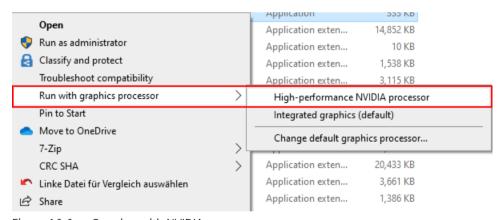
Figure 16-6   Opening with NVIDIA

16.2 Connection between AMW /Capture and AMW /Toolpath

Connection problems

If you use AMW /Toolpath with AMW /Capture, a connection-related error may be displayed. The following table contains information on how to correct the specified error.

Error message	Possible cause	Remedy
WinHttpSendRequest: 12044: A certificate is required to complete client authentication	The client authentication certificate is not imported into the user's personal certificate store.	Import the client authentication certificate into the personal certificate store.
WinHttpSendRequest: 2148074277: The certificate chain was issued by an authority that is not trusted	Although the client authentication certificate is available, it is possible that the root certificate is missing from the Trusted Root Certification Authorities store.	Import the existing root certificate in the Trusted Root Certification Authorities store. It is recommended that you regenerate the root, server and client authentication certificates if the problem persists.
WinHttpSendRequest: 2148073488: Access denied.	An invalid password was specified or the password request session was terminated by the user.	Enter the correct password.
SSL Error: WINHTTP_CALLBACK_STATUS_FLAG_INVALID_CERT SSL certificate invalid.	A secure connection with AMW /Capture cannot be established due to missing or invalid MiniWeb Server certificates.	Make sure that the generated server certificate is imported into the MiniWeb security configuration page and the root certificate is imported into the Trusted Root Certification Authorities store on the client computer. It is recommended that you regenerate the root, server and client authentication certificates if the problem persists.
error calling getServerInfo: FORBIDDEN (1)	An incorrect Edge platform version was selected.	Select the correct Edge platform version.
SSL Error: WINHTTP_CALLBACK_STATUS_FLAG_CERT_REV_FAILED failed to check revocation status. WINHTTP_CALLBACK_STATUS_FLAG_CERT_CN_INVALID SSL common name does not match.	The DNS name is used when connecting to AMW /Capture. Previously, the server certificate was configured using only the DNS name and not the IP address.	To connect to AMW /Capture, enter the IP address of the Edge device. If the DNS name must be used, the certificates must be regenerated with the correct configuration. We recommended that you check the system host file. Even with properly configured certificates, the host file must contain the DNS name - IP address assignment.
AMWT version < V2.5	Connection mechanism between Capture and AMWT has been changed	Edge devices added to AMWT in a previous AMWT version (for example, v2.4) can no longer connect due to the changes in the Toolpath Capture integration.

Analyze MyWorkpiece /Toolpath - Python API Developer Guide

A

A.1 Overview

This section describes and explains the fundamental concepts of extending the core functionality of Analyze MyWorkpiece /Toolpath based on user-defined Python scripts. This section guides you through the development techniques:

- Installation of the development environment
- Coding instructions based on examples
- Compilation and deployment

Function overview

Analyze MyWorkpiece /ToolPath runs precompiled Python scripts (*.pyd -files), which are saved in subfolder `thirdparty\python_virtualenv` of your Analyze MyWorkpiece /ToolPath installation folder. To run a script, run Extensions > Extension wizard in Analyze MyWorkpiece / ToolPath.
You can select the list of these files from the drop-down list "Script selection" of the extension wizard.

A.2 Development environment - Installation

Installation

Download the Python 3.7.x environment from the official website:

https://www.python.org/downloads/release/python-379/

Install "Windows x86-64 executable installer" in a user-defined directory: e.g. c:\projects\Python37

Open a command line in the installation directory.

Create a virtual environment with Python 3.7:

```
python -m pip install virtualenv
python -m virtualenv venv
```

Start virtualenv:

```
cd venv
```

```
.\Scripts\activate
```

Copy the following content in a file from requirements.txt in folder (`/venv`):

Source code

```
Cython==0.29.21
joblib==1.1.0
llvmlite==0.38.0
numpy==1.21.5
pandas==1.3.5
plotly==5.6.0
python-dateutil==2.8.2
pytz==2022.1
scikit-learn==1.0.2
scipy==1.7.3six==1.16.0
sklearn==0.0
tenacity==8.0.1
threadpoolctl==3.1.0
```

Run the pip installation program:

```
python -m pip install -r requirements.txt
```

A.3　　Use the Cython compiler

Analyze MyWorkpiece /Toolpath supports extension scripts, which are precompiled by Cython. Cython is an optimizing static compiler for the Python programming language, see

https://cython.org/ (https://cython.org/)

Any .py source files can be compiled as follows.

Create the cython_compile.py help file in directory venv/ with the following content:

Source code

```
# Copyright (C) Siemens AG 2019-2021. All rights reserved.
#!python
#cython: language_level=3

from setuptools import setup
from Cython.Build import cythonize
import glob
import os

def compile():
    file_list = glob.glob("*.py")
    for file in file_list:
        if file.endswith('cython_compile.py'):
            continue
        setup(
            ext_modules = cythonize(file)
        )
```

This script compiles (cythonizes) every file with py extension using the Cython compiler. Detailed information about the compilation process is provided under:

Cython user guide (https://cython.readthedocs.io/en/latest/src/userguide/source_files_and_compilation.html)

The next line is the compilation process itself. Generate the pyd file using the following command:

```
python.exe -c "from cython_compile import *;compile()" build_ext --
inplace
```

The generated *.pyd files must be copied to `<AMWT_install directory dir>\thirdparty\python_virtualenv\`.

After Analyze MyWorkpiece /ToolPath restarts, the new extension is available in the menu of the extension wizard via "Extensions > Extension wizard".

A.4   Storage location of Python extensions in the AMW /ToolPath

The Python environment and the scripts are installed in the following folder:

`<AMW Toolpath>\thirdparty\python_virtualenv\`

A.5 Python skeleton template code

Obligatory functions

This section describes the obligatory functions of a Python file.

Source code

```
import package

def version():
    # version required for handshaking
def checkParameters(parameters):
    # script specific parameter check
#
# main business logic is implemented in calculate function
#
def calculate(params):
    # on error print a message and quit with exit code 1
    if error:
        print("Error message")
        sys.exit(1)
    # on success print a message and quit with exit code 0
    print("Work done")
    sys.exit(0)

def description():
    """
    Put your description here. Note the three quotation marks.
    """
    return None

def getParamTemplate():
    """
    Put your json formatted parameter definition here as described later in this document.
Note the three quotation marks.
    """
    return None

def manual_test():
    # you may want to invoke the script outside of AMW Toolpath. This is the entry point

if __name__=="__main__":
    manual_test()
```

Every script has its own description, which is displayed by AMW /Toolpath.
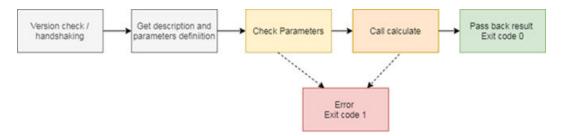
In general, "`Exit-Code 1`" refers to an error, and "`Exit-Code 0`" to normal execution. Analyze MyWorkpiece /Toolpath captures and displays everything that you print in the standard output.
The business logic must be implemented in the `calculate()` function. The requested parameters are defined using JSON data. These are subsequently described, so that each script can define its own List Manual.

You can perform a validation, as the parameters are dynamic and specific for the actual Python script.

Call sequence

The call sequence that is executed in Analyze MyWorkpiece /Toolpath is shown in the following graph.

A.6 Examples

A.6.1 Visualizing signals and calculating higher order derivations

This Python test source code shows an algorithm that processes an input signal. It can read a specified signal from a predefined trace file (xml, csv, zip, json), and calculate the first, second and third derivations (velocity, acceleration and jerk).

A.6.2 Obligatory functions

You must specify the following functions to be able to communicate with the GUI:

Function name	Description
`version()`	`version()` must return a character string with the version code of the API; current version **1.0**. The graphic ToolPath user interfaces uses scripts, where the version information matches.
`description()`	The textual description of the actual function must be specified in the comment format in triple exclamation marks (""").
`getParamTemplate()`	This function returns definitions of input parameters in the Python dictionary format.
`calculate()`	Calculates results, generates files and returns values.

A.6.3 Parameter template format

JSON format to describe input and output data. The following complex example illustrates the use.

Source code

```
{
  "inputs": [
    {
      "inputFiles": "Choose one or more trace filenames|multipleInputFiles",
      "signals": [
        "Select signals|multipleSignals",
      ]
    },
    {
      "inputFiles": "Choose a trace filename|singleInputFile",
      "signals": [
        "Select X position signal|singleSignal",
        "Select Y position signal|singleSignal",
        "Select Z position signal|singleSignal",
      ]
    }
  ],
  "params": {
      "num_clusters": "Number of Clusters||3",
      "metrics": "Metrics Type|l1#l2#euclidean#manhattan|l2"
  },
  "outputs": {
      "resultFileName": "result.bin",
      "resultImageName": "image.png"
  }
}
```

Code	Description
inputFiles	Input prompt and definition separated by |. Definition can be `multipleInputFiles` or `singleInputFile`.
signals	Array, signal input 0 or more. Every input element has an input prompt and a definition, which are separated by | Definition can be `multipleSignals` or `singleInputFile`.
params	Defines the parameters required by the script as "parameter_name": "Definition" Definition is a character string separated with "|" with the following data: "Input prompt | Data set | Default value", where "Data set" can be empty or comprise individual values separated with "#".
outputs	Defines the output generated by the script as "Definition": "File name" "Definition" can be `resultFileName`, which generates binary data, or `resultImageName`, which generates an image file that should be displayed or `autogeneratedOutputFile|. ext`, with which Analyze MyWorkpiece /Toolpath can generate a file name with the specified extension.

A.6.3.1 Special parameters

You can specify hidden parameters, which are not displayed in the parameter setting dialog, as shown in the following example:

Source code

```
"params": {
        "requiresToolDatabase|Tool data required|true#false|true",
        "exportSignalData|hidden 1||input/signals.bin",
        "useLoadedModelsOnly|hidden 2||true"
        }
```

Note

For more than one "hidden" parameter, use a unique number to make a differentiation between them.

Code	Example
requiresToolDatabase	If the setting is "true", then a tool database is transferred in the JSON format.
exportSignalData	The case of a surface reconstruction, the selected signals are exported to a temporary binary file.
useLoadedModelsOnly	With this parameter, the user can only use loaded models.

Note

Prompt is only a descriptive text; there is no validation. This means that if you use "Select X position signal" there is no validation; you can transfer any signal.

A.6.3.2   Python code - Example 1

Example 1

Source code

```python
import sys
import json
from fileFormatUtils import load_data

# returns version information
def version():
    return '1.0'

#
# checks input parameter template, in case of errors call: sys.exit(1)
# error messages print to standard output
#
def checkParameters(parameters):
    # missing 'inputs' attribute
    if "inputs" not in parameters:
        print("No inputs specified")
        sys.exit(1)

    inputs = parameters["inputs"][0]

    # missing 'inputFiles' argument
    if "inputFiles" not in inputs:
        print("No input file specified")
        sys.exit(1)

    # missing 'signals' argument
    if "signals" not in inputs:
        print("No signals specified")
        sys.exit(1)

    signals = inputs["signals"]

    # we can use only one signal in this example
    if len(signals) != 1:
        print("Only one signal must be specified")
        sys.exit(1)

    if "params" not in parameters:
        print("No 'params' in parameters.")
        sys.exit(1)

    params = parameters["params"]

    # this section guarantees that we are using all
    # the three input parameters and one output parameter required
    if "fig_size_X" not in params:
        print("Missing parameter: Figure Size X")
        sys.exit(1)

    if "fig_size_Y" not in params:
```

Source code

```
        print("Missing parameter: Figure Size Y")
        sys.exit(1)

    if "outputs" not in parameters:
        print("No 'outputs' in parameters.")
        sys.exit(1)

    if not parameters["outputs"]["resultFileName"].endswith('.html'):
        print("Output file extension must be .html")
        sys.exit(1)

    return

#
# main business logic
#
def calculate(params):
    # convert input string to JSON object
    parameters = json.loads(params)
    # check input parameters
    checkParameters(parameters)

    # obtain filename from the input JSON
    inputs = parameters["inputs"][0]
    inputFile = inputs["inputFiles"]
    signals = inputs["signals"]

    # load data automatically with a file reader
    df = load_data(inputFile, signals)

    if df is None:
        print("Unable to read input file")
        sys.exit(1)

    import scipy.signal as signal
    import matplotlib.pylab as plt
    import matplotlib
    import numpy as np

    #
    # create output signal names: velocity, acceleration and jerk
    # apply signal filtering
    #
    velocity_column_name = signals[0] + ' VELOCITY'

    # Apply a Savitzky-Golay filter to an array and differentiate
    df[velocity_column_name] = np.gradient(signal.savgol_filter(df[signals[0]], 41, 1), 1)

    acceleration_column_name = signals[0] + ' ACCELERATION'
    df[acceleration_column_name] = np.gradient(df[velocity_column_name], 1)

    jerk_column_name = signals[0] + ' JERK'
    df[jerk_column_name] = np.gradient(df[acceleration_column_name], 1)

    # create a Figure
    f, ax = plt.subplots(nrows=len(df.columns), ncols=1,
```

A.6 Examples

Source code

```
                         figsize=(parameters["params"]["fig_size_X"], parameters["params"]
["fig_size_Y"]),
                         subplotpars=matplotlib.figure.SubplotParams(hspace=0.2, wspace=3))

    # draw signals one by one
    for i in range(len(df.columns)):
        ax[i].plot(df[df.columns[i]])
        ax[i].margins(x=0, y=0.25)
        ax[i].set_ylabel(df.columns[i], fontsize=10, rotation=0)
        ax[i].yaxis.set_label_coords(-0.1, 0.5)
        ax[i].set_xticklabels([])

    # create a plotly offline plot and render results in a html file
    import plotly
    import plotly.tools as tls
    fig = tls.mpl_to_plotly(f)
    plotly.offline.plot(fig, filename = parameters["outputs"]["resultFileName"],
auto_open=False)

    # open the default browser to show result
    import os
    # open html in the default browser
    os.startfile(parameters["outputs"]["resultFileName"])
    sys.exit(0)

def description():
    """
    Calculate velocity and acceleration of a selected signal - Using Savitzky-Golay
filtering.
    """
    return None

def getParamTemplate():
    """
    {
      "inputs": [
        {
          "inputFiles": "Choose a trace filename|singleInputFile",
          "signals": [
            "Select signals|oneSignal"
          ]
        }
      ],
      "params": [
          "fig_size_X|Figure X size||20",
          "fig_size_Y|Figure Y size||10"
      ],
      "outputs": {
          "resultFileName": "|autoGeneratedOutputFile|.html"
      }
    }
    """
    return None

def manual_test():
    return calculate("""
```

Source code

```
{
 "inputs": [
   {
     "inputFiles": "x:/Testrack_Apple_HF.5.json",
     "signals": [
       "DES_POS|1"
     ]
   }
 ],
 "params": {
     "fig_size_X": "20",
     "fig_size_Y": "10"
 },
 "outputs": {
     "resultFileName": "results/out.html"
 }
}
""")
```

A.6.3.3   Python code - Example 2

Example 2

The following example calculates the vector length of three scalar signals (axes). The result is an array, which is returned to Analyze MyWorkpiece /ToolPath.

Note

The `manual_test()` function allows developers to run this script outside of Analyze MyWorkpiece /ToolPath. (the Python environment and the required components should be installed)

Source code

```python
import sys
import json
from fileFormatUtils import load_data, make_full_path

def version():
    return '2.0'

def checkParameters(parameters):
    if "inputs" not in parameters:
        print("No inputs specified")
        sys.exit(1)

    if len(parameters["inputs"]) != 1:
        print("Number of inputs must be one")
        sys.exit(1)

    for inputs in parameters["inputs"]:
        if "inputFiles" not in inputs:
            print("No input file specified")
            sys.exit(1)

        if "signals" not in inputs:
            print("No signals specified")
            sys.exit(1)

        signals = inputs["signals"]

        if len(signals) != 3:
            print("Number of signals must be 3")
            sys.exit(1)

    if "params" not in parameters:
        print("No 'params' in parameters.")
        sys.exit(1)

    if "outputs" not in parameters:
        print("No 'outputs' in parameters.")
        sys.exit(1)

    if not parameters["outputs"]["resultFileName"].endswith('.bin'):
        print("Output file extension must be .bin")
        sys.exit(1)

    return

def calculate(params):
    # convert string parameters into JSON
    parameters = json.loads(params)
    # check input syntax
    checkParameters(parameters)

    input = parameters["inputs"][0]
    inputFile = input["inputFiles"]
    signals = input["signals"]
```

Source code

```python
    # load input
    df1 = load_data(inputFile, signals)

    # check errors
    if df1 is None:
        print('Error: inputfile cannot be loaded: ', inputFile)
        sys.exit(1)

    result = df1

    # calculate euclidean distance
    result['Vector length'] = (df1[signals[0]] ** 2 + df1[signals[1]] ** 2 + df1[signals[2]]
** 2) ** 0.5
    import numpy
    # write result into a file
    # file data will be displayed in AMWT Toolpath
    result_array = numpy.array(result['Vector length'])

    # result file has absolute path? if not then add current absolute path
    resultFile = make_full_path(parameters["outputs"]["resultFileName"])
    result_array.astype("float32").tofile(resultFile)

    # normal exit without errors
    sys.exit(0)

def description():
    """
    This script calculates Euclidean distance of three signals in one input trace file.
    It is usefull when you want to analyse e.g. 'CONTROL DIFF' signals.
    """
    return None

def getParamTemplate():
    """
    {
      "inputs": [
            {
          "inputFiles": "Choose first trace filename|singleInputFile",
          "signals": [
            "Controll Diff X Axis|CTRL_DIFF|singleSignal",
            "Controll Diff Y Axis|CTRL_DIFF|singleSignal",
            "Controll Diff Z Axis|CTRL_DIFF|singleSignal"
          ]
        }
    ],
      "params": [
      ],
      "outputs": {
          "resultFileName": "Choose an output binary file|singleOutputFile|.bin"
      }
    }
    """
    return None

def manual_test():
    return calculate("""
```

Source code

```
    {
      "inputs": [
          {
          "inputFiles": "tests/Testrack_HF.5_trace.json",
          "signals": [
            "CTRL_DIFF|1",
            "CTRL_DIFF|2",
            "CTRL_DIFF|3"
          ]
        }
    ],
      "params": {
      },
      "outputs": {
          "resultFileName": "results/out.bin"
      }
    }
    """)


if __name__=="__main__":
    manual_test()
```

A.6.4   Format of parameter template V2

Overview

Using Python script parameter template format 2.0, the behavior of /Toolpath Python extension can be extended at the front end. This extended script implementation acts on object "params" of the function content `getParamTemplate()`. The structure is based on JSON and the parent-child relationship.

The 2.0 script can modify the user interface corresponding to the implementation of the Python script; further, predefined validation functions ensure that the value of the specified parameter object satisfies the necessary requirements.

Source code

```python
def getParamTemplate():
    """
    {
      "inputs": [
        {
          …
        }
      ],
      "params": [
        {
            "<Object-name>": { Name of the created object in /Toolpath
                "UI-Type": "<ui-tpye>", Type of the parameter object (see more in Supported
Parameter Types)
                "Display-Name": "<display-name>", This name is displayed on the ui next to the
corresponding ui object
                "Max-Width": "<max-width>", Maximum width of the given parameter object
                "Max-Height": "<max-height>", Maximum height of the given parameter object
                "Parameter-ID": "<parameter-id>", Id of the parameter. This id is used by the
python script when executing it
                "Default-Value": "<default-value>", Initial default value of the parameter
object
                "Values": [ List of possible parameter values.
                  "<value1>",
                  "<value2>",
                  "<value3>",
                  "<value4>"
                ],
                "Child-Objects": [ List of additional parameter objects
                ]
        }
      }
    ],
      "outputs": {
          …
      }
    }
    """
    return None
```

A.6.4.1 Parameter types

Supported parameter types

Depending on the defined UI type the specified parameter object can be displayed or hidden, can be interactive or static.

UI type "Group"

A group field can be created with the specified properties. A group field can contain additional parameter objects.

Source code

```
"Group_Object": {
        "UI-Type": "Group",
        "Display-Name": "Parameter Group",
        "Max-Width": "250",
        "Max-Height": "300",
        "Child-Objects": [
            ...
        ]
}
```

UI type "Input"

A text input field is created. The default value is required. Validation and value change functions can be added.

Source code

```
"Input_Object": {
        "UI-Type": "Input",
        "Display-Name": "Parameter Input:",
        "Max-Width": "150",
        "Parameter-ID": "param_id",
        "Default-Value": "10",
        "Function": "Function_1"
}
```

UI type "Combo Box"

A combo box is created. Default values and optional values are required. Validation and value change functions can be added.

Source code

```
"Combobox_Object" : {
        "UI-Type": "Combobox",
        "Display-Name": "Parameter combo:",
        "Max-Width": "150",
        "Parameter-ID": "param_id",
        "Default-Value": "value3",
        "Values": [
            "value1",
            "value2",
            "value3",
            "value4"
        ],
        "Function": " Function_1"
}
```

UI type "None"

Invisible parameter object. Its value cannot be changed via the user interface.

Source code

```
"Nonvisible_object": {
    "UI-Type": "None",
    "Parameter-ID": "param_id",
    "Default-Value": "value1"
}
```

A.6.4.2 Validation

Pre-defined validation functions

Predefined validation functions can be attached to parameter objects to ensure that the actual parameter value is correct. These functions can only be attached to parameter objects, type Input and Combo box, and they are executed each time the value changes.

If the specified parameter value does not comply with the necessary requirements that are defined by the attached validation function, then you cannot continue with the script configuration in the Python-extension of Analyze MyWorkpiece /Toolpath.

- Validator_IsNumeric: Checks whether the specified parameter value is numerical.

- Validator_IsFloat: Checks whether the specified parameter value is a "float" value

- Validator_IsVector3D: Checks whether the specified parameter value defines a 3D vector. Each value is separated by a blank (e.g.: 10.6 24.3 0.4)

Source code

```
"Input_with_validation": {
              "UI-Type": "Input",
              "Display-Name": "Parameter with validation:",
              "Max-Width": "150",
              "Parameter-ID": "input_validation",
              "Default-Value": "30",
              "Validation": "Validator_IsNumeric"
}
```

A.6.4.3 Parameter template - example

Program example

Source code

```python
def getParamTemplate():
    """
    {
      "inputs": [
        {
          "inputFiles": "Choose a trace filename|singleInputFile",
          "signals": [
            "Select signals|multipleSignals"
          ]
        }
      ],
      "params": [
        {
          "1_Group": {
            "UI-Type": "Group",
            "Display-Name": "Parameter-Group-1",
            "Child-Objects": [
              {
                "1_Parameter": {
                  "UI-Type": "Combobox",
                  "Display-Name": "Parameter 1:",
                  "Max-Width": "150",
                  "Parameter-ID": "1_Parameter",
                  "Default-Value": "val3",
                  "Values": [
                    "val1",
                    "val2",
                    "val3",
                    "val4"
                  ]
                },
                "2_Parameter": {
                  "UI-Type": "Combobox",
                  "Display-Name": "Parameter 2:",
                  "Max-Width": "150",
                  "Parameter-ID": "2_Parameter",
                  "Default-Value": "val3",
                  "Values": [
                    "val1",
                    "val2",
                    "val3",
                    "val4"
                  ]
                },
                "1_Sub_Group": {
                  "UI-Type": "Group",
                  "Display-Name": "Sub-Parameter-Group-1",
                  "Child-Objects": [
                    {
                      "1_Sub-Parameter": {
```

Source code

```
              "UI-Type": "Combobox",
              "Display-Name": "Sub-Parameter 1:",
              "Max-Width": "150",
              "Parameter-ID": "1_Sub-Parameter",
              "Default-Value": "val3",
              "Values": [
                "val1",
                "val2",
                "val3",
                "val4"
              ]
            },
            "2_Sub-Parameter": {
              "UI-Type": "Input",
              "Display-Name": "Sub-Parameter 2:",
              "Max-Width": "150",
              "Parameter-ID": "2_Sub-Parameter",
              "Default-Value": "val"
            },
            "3_Sub-Parameter": {
              "UI-Type": "Input",
              "Display-Name": "Sub-Parameter 3:",
              "Max-Width": "150",
              "Parameter-ID": "3_Sub-Parameter",
              "Default-Value": "val"
            }
          }
        ]
      },
      "2_Sub_Group ": {
        "UI-Type": "Group",
        "Display-Name": "Sub-Parameter-Group-2",
        "Child-Objects": [
          {
            "4_Sub-Parameter": {
              "UI-Type": "Combobox",
              "Display-Name": "Sub-Parameter 1:",
              "Max-Width": "150",
              "Parameter-ID": "4_Sub-Parameter",
              "Default-Value": "val3",
              "Values": [
                "val1",
                "val2",
                "val3",
                "val4"
              ]
            },
            "5_Sub-Parameter": {
              "UI-Type": "Input",
              "Display-Name": "Sub-Parameter 2:",
              "Max-Width": "150",
              "Parameter-ID": "5_Sub-Parameter",
              "Default-Value": "val"
            },
            "6_Sub-Parameter": {
              "UI-Type": "Input",
```
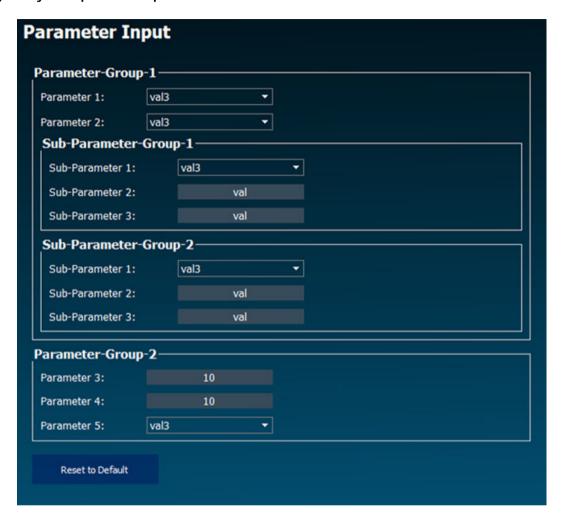
Source code

```
                        "Display-Name": "Sub-Parameter 3:",
                        "Max-Width": "150",
                        "Parameter-ID": "6_Sub-Parameter",
                        "Default-Value": "val"
                      }
                    }
                  ]
                }
              }
            ]
          },
          "2_Group ": {
            "UI-Type": "Group",
            "Display-Name": "Parameter-Group-2",
            "Child-Objects": [
              {
                "3_Parameter": {
                  "UI-Type": "Input",
                  "Display-Name": "Parameter 3:",
                  "Max-Width": "150",
                  "Parameter-ID": "3_input",
                  "Default-Value": "10"
                },
                "4_Parameter": {
                  "UI-Type": "Input",
                  "Display-Name": "Parameter 4:",
                  "Max-Width": "150",
                  "Parameter-ID": "4_input",
                  "Default-Value": "10"
                },
                "5_Parameter": {
                  "UI-Type": "Combobox",
                  "Display-Name": "Parameter 5:",
                  "Max-Width": "150",
                  "Parameter-ID": "5_input",
                  "Default-Value": "val3",
                  "Values": [
                    "val1",
                    "val2",
                    "val3",
                    "val4"
                  ]
                }
              }
            ]
          }
        }
      }
    ],
    "outputs": {

    }
  }
  """
  return None
```

Result in Analyze MyWorkpiece /Toolpath

Index