SIEMENS

SINUMERIK

SINUMERIK ONE
System Manual Run MyVirtual Machine

System Manual

Valid for:
CNC ShopFloor Management Software
Run MyVirtual Machine V1.3.1
SINUMERIK ONE
SINUMERIK Virtual CNC SW V6.21

01/2023
A5E48105802B 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.

Siemens AG
Digital Industries
Postfach 48 48
90026 NÜRNBERG
GERMANY

A5E48105802B AH
Ⓟ 01/2023 Subject to change

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 document primarily addresses CNC programmers in job preparation as well as persons working in the area of CNC training and ongoing training. Persons in the sales domain are also addressed who use Run MyVirtual Machine for CNC or machine presentations.

Purpose

This documentation describes the fundamentals and the operation of Run MyVirtual Machine.

Benefits

This documentation enables the addressed target group to operate the system.

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 ONE

Comprehensive documentation about the functions provided in SINUMERIK ONE Version 6.13 and higher is provided in the Documentation overview SINUMERIK ONE (https://support.industry.siemens.com/cs/ww/en/view/109768483).

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
- 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).

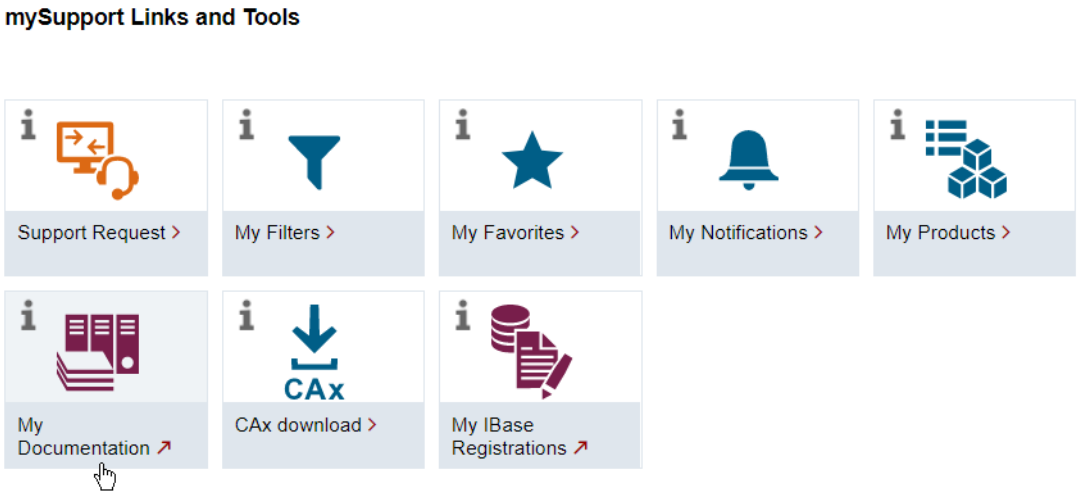
1.4 Feedback on the technical documentation

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 links and tools" (https://support.industry.siemens.com/cs/ww/en/my) portal page:



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 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)

1.8 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.

Fundamental safety instructions 2

2.1 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.

2.2 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.

2.3 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.

Overview 3

3.1 Deployed software

Software requirements

The installation of Run MyVirtual Machine and other essential software components is described in the *Run MyVirtual Machine Installation Manual*. The Installation Manual can be found on the installation medium in the folder ..\Documents\Readme\English\.

3.2 General information

Run MyVirtual Machine (Run MyVirtual Machine /Operate license)

Run MyVirtual Machine is an NC programming workstation with identical controls on the PC for machine tools using SINUMERIK ONE.

The offline programming workstation Run MyVirtual Machine simulates a machine tool controlled by SINUMERIK ONE. SINUMERIK Operate, together with a simulated machine control panel, ensures realistic operation and programming on the PC. You only require SINUMERIK CNC programming know-how.

Run MyVirtual Machine enables offline CNC programming on the PC, for example in work preparation. Exactly the same scope of CNC language commands, CNC machining cycles, and ShopMill/ShopTurn work steps is available as in the real CNC. Regardless of whether CNC programs were generated via Run MyVirtual Machine itself or via CAM systems, they can be checked for freedom from errors in the best possible way.

Run MyVirtual Machine is therefore the optimal tool to increase efficiency and process reliability in CNC programming.

Furthermore, Run MyVirtual Machine permits easy learning and professional training of CNC operation and programming without a real CNC, for example in training classrooms. For this purpose, preconfigured sample machines are available for immediate use. Using SINUMERIK Operate and the original SINUMERIK CNC kernel, all operating processes and NC programming operations can be used and CNC programs executed, without any restrictions. New functions and programming options can thus be learned, tested and demonstrated in a secure environment.

To get the highest possible match with the real CNC, you can load machine projects (*.vcp) matching the respective machine. Contact your machine manufacturer for this purpose.

Since the machine projects are always assigned to a version of the SINUMERIK Virtual CNC software, different SINUMERIK Virtual CNC software versions can be stored in Run MyVirtual Machine.

This means that machines from different manufacturers can be provided with different versions of the SINUMERIK Virtual CNC software at one workstation in the CNC work preparation.

Run MyVirtual Machine /Open

Run MyVirtual Machine /Open is an additional option to Run MyVirtual Machine /Operate. You need it to operate an external SW application, for example, your own machine room simulation.

Run MyVirtual Machine /3D

Run MyVirtual Machine /3D is an additional option to Run MyVirtual Machine /Operate. This option extends Run MyVirtual Machine to include integrated 3D machining and material removal simulation. This enables you to evaluate machine movements visually and to check for freedom from collision. Using the material removal simulation, workpiece machining can be tested in advance via simulation.

The 3D simulation is also ideal for training setup procedures and running in machines on a virtual model without exposure to any risks whatsoever.

Demarcation with respect to Create MyVirtual Machine

Using Create MyVirtual Machine, machine OEMs develop digital twins for CNC machines, commission the machines and test the machine functions in conjunction with simulation platforms. The machine project that is developed on this basis can be used in Run MyVirtual Machine, and be used as NC programming station that is identical to the control system itself.

Components of the Run MyVirtual Machine overall system

Run MyVirtual Machine (incl. SINUMERIK Virtual CNC software) consists of the following components:

- simNCK: Simulation of the NCK
- SIMATIC S7-PLCSIM Advanced: PLC simulation based on S7-1500
- HMI: SINUMERIK Operate
- MCP: Virtual machine control panel
- simDrive (drive simulation): SINAMICS S120 substitute drive component
- Integrated peripheral simulation
- Open interface for integration of external simulation tools (option)
- 3D Simulation (option)

Using Run MyVirtual Machine

4

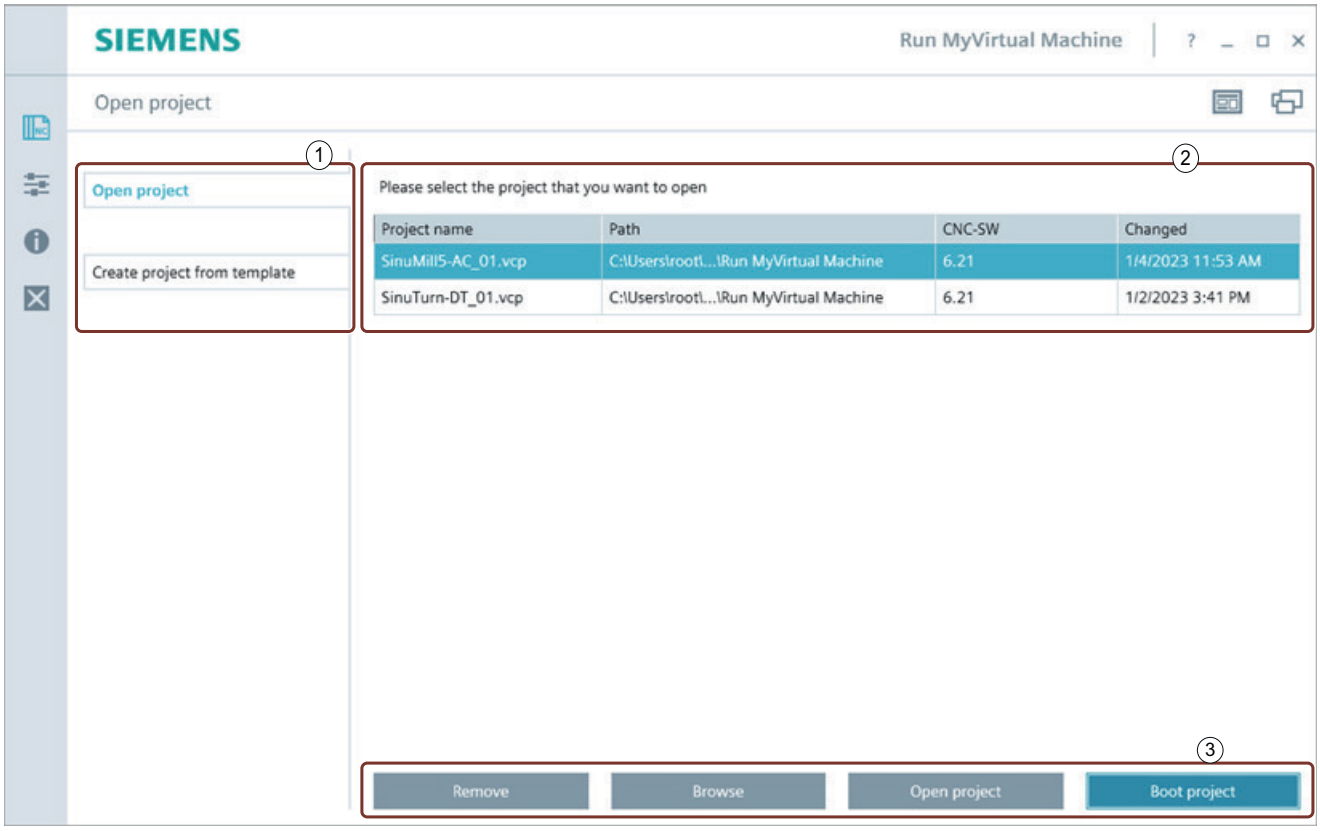
4.1 Introduction

In Run MyVirtual Machine you work with machine projects for a wide range of different types of CNC machines with SINUMERIK ONE control systems. Within the respective machine project, the virtual SINUMERIK with CNC user interface, CNC core functions, and optionally a 3D simulation of the machine room are available.

4.2 User interface

4.2.1 Project management

You manage machine projects ("*.vcp") in the project management of run MyVirtual Machine. These are either provided by machine OEMs or saved in the system as template (template/ sample machines for training purposes).



① **Open project**
Opens existing projects from the overview

Create project from template
Creates a new project based on a template.

② Overview of recently opened machine projects with storage path, CNC SW version used, and date of change.

③ **Remove**
Removes projects from the project overview. The machine project is only deleted from the overview, and remains on the data storage medium.

Browse
Searches the data storage medium for projects and enters them into the overview.

Open project
Opens a selected machine project in the overview.

Start project
Opens a selected machine project from the overview. The machine is automatically started.

4.2.2 Overview of project settings

Project settings

After opening the project, the "Project settings" are first displayed. Here, you can configure specific project settings, save the project and start.

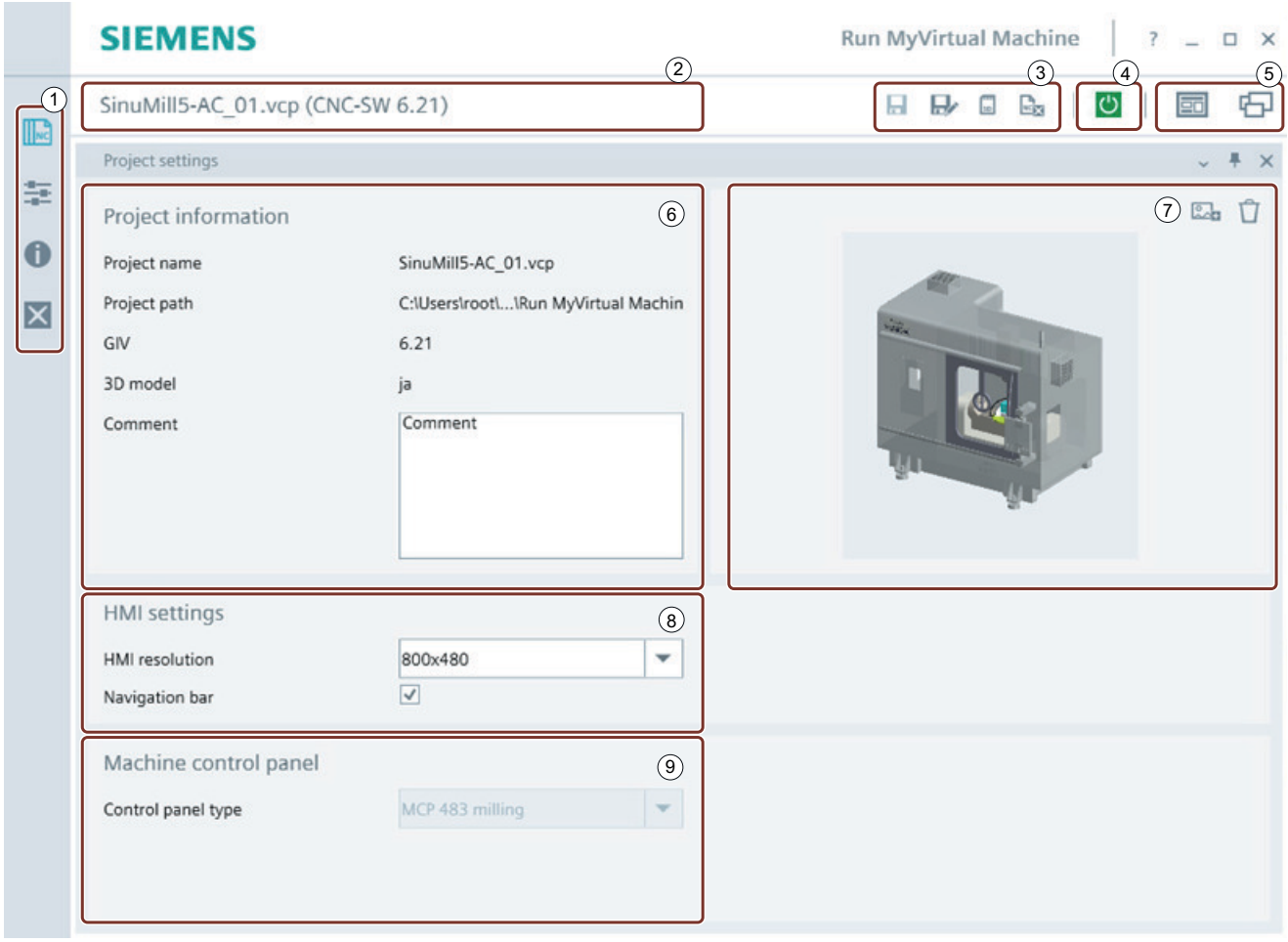


Figure 4-1 Project settings using a milling machine as example

① Basic functions

Click on the buttons to use the basic functions of Run MyVirtual Machine.

Displays of the project overview

Settings
Setting the user interface language and the communication connections

Information
Display of the version and license information

Exit
Exits Run MyVirtual Machine

② Title bar

Display of the project name and version of the CNC software.

③ Managing the opened machine project

- Save
  Saves the open machine project.
  Machine projects can be saved only if the machine simulation has been previously exited.
- Save as
  Saves the open machine project under a new name or in another folder.
  Machine projects can be saved only if the machine simulation has previously been exited.
- Memory card
  Opens Windows Explorer with the storage location of the virtual memory card.
- Close project
  Closes an open machine project. If there are unsaved changes, a note is displayed, and you can save the project before closing.

④ Start simulation

Start simulation of the machine.
The simulation control cannot be operated during the run-up phase.

⑤ Manage window arrangement and windows

Click on the button to create or select a window arrangement. You can save the arrangement of the individual windows as user-defined arrangement. You can create, save and delete user-defined window arrangements.

Click on the button to display/hide the window areas. Check/uncheck the checkbox in front of the respective window name in the displayed list.

You can remove the individual windows using the  buttons, and dock them at any other position in Run MyVirtual Machine. For example, you can display HMI SINUMERIK Operate in a separate window.

⑥ Project information

The following project information is displayed here. You define the project information when creating the project.

- Project name and project path
- CNC SW version
- Displays whether the machine project includes a 3D model (yes/no)
- You can optionally enter a comment.

⑦ Machine diagram

You can optionally assign a unique machine diagram to the project.


Click on the button to integrate a graphic file as machine diagram. You can only select valid graphic formats.


Click on the button to delete the machine diagram that was inserted.

⑧ HMI settings

You can only change the project settings if the simulation of the machine has not been started.

- HMI resolution
  Select the resolution of the HMI of SINUMERIK Operate. The HMI will be displayed in the selected resolution at the next startup.
- Navigation bar
  Select the checkbox if you want the side navigation bar to be displayed in the HMI. Using the navigation bar, you can quickly access machine areas of the HMI, e.g. program or tool list.

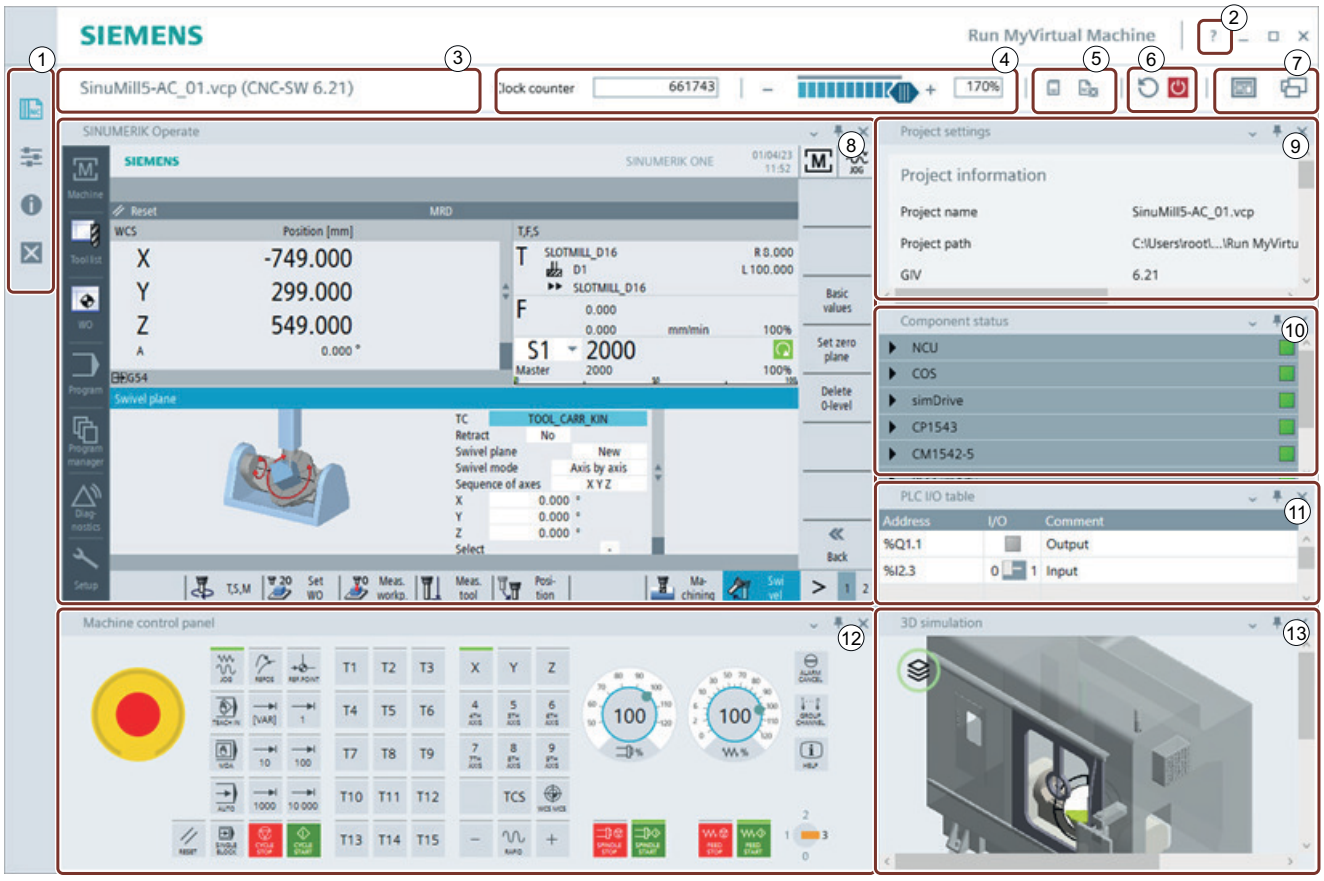
⑨ Machine control panel

The machine control panel displayed is used in the project. This cannot be changed in Run MyVirtual Machine.

- Control panel type
  Display of the machine control panel (e.g. MPC 483 for milling or MCP 483 for turning) used in the machine project.

4.2.3 SINUMERIK Operate with machine simulation

Using Run MyVirtual Machine corresponds to a real control system equipped with a SINUMERIK Operate user interface and machine control panel. The machine basic screen is displayed after the control ramp-up.



① Basic functions

Click on the buttons to use the basic functions of Run MyVirtual Machine.

- Opens the project management/project overview
- Settings
  Opens the settings to switch between languages and reset the window layout.
- Information
  Displays of the version information
- Exit
  Exits Run MyVirtual Machine

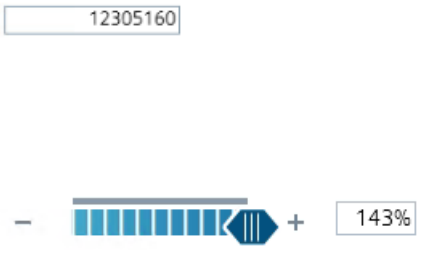
② Help

Open/close Help. The Help is displayed in a separate viewlet. You can extract the viewlet and display as a separate window.

③ Title bar

Display of the project name and version of the CNC software.

④ Simulation control



Clock cycle counter (optional)

Displaying previous servo clock cycles.

In "Settings", you can configure whether the clock cycle counter is displayed. As default, it is hidden.

Change simulation speed from standstill (pause – system stopped) up to the maximum speed.

Using the simulation control, you configure how much CPU performance should be used for Run MyVirtual Machine when in the simulation mode. For normal operation, keep the slider at the maximum position (+).

- In the left-hand position (-) 0 %, the simulation is in the pause status. In this state, the "frozen" machining process can be observed. Changes to values and operating actions are not possible or lead to inconsistent states in the machine project.
- In the center position 100 %, the simulation speed approximates the speed/clock cycle of a real machine and runs approximately in real-time.
- In the right (+) position, the simulation runs with maximum speed. The percentage display specifies how much faster the system operates when compared with real-time. For example, the value 800 % corresponds to approximately the 8-fold real-time speed. The maximum simulation speed is limited by several factors, including the computer power.

Note

Program runtimes and simulation control

The simulation control does not influence determining the program runtimes of NC programs. Independent of the position of the slider, after running through the simulation or after executing NC programs, identical runtimes are obtained. Just like at the real machine, only changes to the override influence the determined program runtime.

⑤ Managing the machine project

- Memory card
  Opens Windows Explorer with the storage location of the virtual memory card.
- Close project
  Closes an open machine project. If there are unsaved changes, a note is displayed, and you can save the project before closing.

⑥ Stop/reset simulation

Exit simulation of the machine.

Reset
Initiate NCK/PLC warm restart

⑦ Manage window arrangement and windows

Click on the button to create or select a window arrangement. You can save the arrangement of the individual windows as user-defined arrangement. You can create, save and delete user-defined window arrangements.

Click on the button to display/hide the window areas. Check/uncheck the checkbox in front of the respective window name in the displayed list.

You can detach, dock or close an individual window using the  buttons.

To release the window so that it can be moved and shown as an autonomous window, click on the **"v**" button.

- Anchor a window at another position in the basic window
  Drag the window to the required window icon using the mouse. The window is then automatically docked.
- Window on a second monitor, e.g. 3D simulation
  Drag the window to the required monitor using the mouse.

Using a docking operation, the separated windows are embedded at the original location in the basic window. Save the new window arrangement as user-defined window arrangement.

⑧ HMI SINUMERIK Operate

The SINUMERIK HMI Operate viewlet contains the SINUMERIK Operate commissioning and operating software.

⑨ Project settings

Displays the project settings. You can only change the settings if the simulation of the machine is stopped.

⑩ Displays the component status with time measurement, clock cycle counter and cycle time

This display provides only information whether the individual component runs correctly.

The time measurement is displayed in the component status display. For non-clocked components, the booting time and the shutdown time are shown. After the first ramp up, the shutdown time is still zero.
The clock cycle counters and the cycle times of the clocked components are also shown in the component status display. The minimum, the maximum and the average cycle time are shown from top to bottom.

The status of the individual components is indicated using a colored rectangle:

- Already started component (green)
- Component starting at that instant in time (gray/green)
- Component shutting down at that instant in time (green/gray)
- Not started component (gray)
- Faulty component (red)

Note

The components are monitored during the ramp-up phase and the cyclical operation of the machine. If no communication can be established between the components (e.g. faulty components are no longer available or incorrectly configured), the attempt to establish communication is terminated after a configured interval (timeout). Faulty components remain in the status red (not available).

⑪ PLC I/O table

You read and write the PLC inputs and outputs with the integrated I/O simulation. Outputs with status LEDs and inputs with toggle switches are configured in the table rows of the expandable PLC I/O table. As a machine operator, you do not usually use a PLC I/O table.

Note

In a few exceptional cases, it may be necessary for signals from the integrated adaptation control to be displayed or influenced. Contact your machine manufacturer for this.

⑫ Virtual machine control panel

- EMERGENCY STOP
  The status of the EMERGENCY STOP (pressed) is indicated by a pictogram below the red button. It may be necessary that you acknowledge the initiation of the EMERGENCY STOP, for example, using the reset button on the machine control panel. When required, contact the person who created the machine project.

- Feedrate and spindle override

- Alarm, Channel, Help keys
- User-assignable function keys
- Keyswitch (0-3)

⑬ 3D simulation (option)

During execution of an NC program in AUTOMATIC mode, 3D simulation with collision monitoring enables you to check the machining process so that any program errors can be detected.

4.3 Application settings

4.3.1 Switching over the program language user interface

You can switch over the user interface program language (Run/Create MyVirtual Machine) in all installed languages.

Note

Languages and HMI SINUMERIK Operate

Switching over the language in Run/Create MyVirtual Machine does not change the language set in HMI SINUMERIK Operate. You change the language in SINUMERIK Operate in exactly the same way as at the real machine in area "commissioning" -or switch between installed languages using key combination "STRG+L".

This is how you switch between program languages

1. Click the "Settings" button .
2. In the window that is displayed, select tab "General settings".
3. Select the new program language.
4. Click "Apply".
   The language is only switched over after Run/Create MyVirtual Machine is restarted.

4.3.2 Communication settings

4.3.2.1 Configuring communication settings (PG/PC interface)

Description

You must configure the communication settings (PG/PC interface) for communications between the TIA Portal or HMI and PLC.

If you select an interface parameter assignment, then assign this to the access point. This means that you establish the connection between the access point, the interface parameter assignment and the interface itself.

Note

Establishing communication with SIMATIC STEP 7 Professional on a second computer

The network adapter configured at the **SINUMERIK CP1543** access point is relevant to the communication between SIMATIC STEP 7 Professional and the MyVirtual Machine products. If communication takes place across computer boundaries (e.g. SIMATIC STEP 7 Professional on another computer in the network), the network adapter with which the computer is connected to the network must be configured at the **SINUMERIK CP1543** access point. The same network adapter must be configured for **SINUMERIK CP1543** and **S7ONLINE (STEP 7)**.

Options for configuring communications

You have two options when configuring the PG/PC interface

- Configuring in Create/Run My Virtual Machine under "Settings" in tab "Communication settings (Page 27)"
  → For the compact and simple setup of the PG/PC interface
- Configuring using application "Siemens communication settings (Page 29)"
  → More complex operation with access to additional communication parameters

Both variants configure the same communication parameters and visualize identical settings.

Checking the communication settings

The communication settings are checked when starting a machine project. If a valid setup is not available, and communication is not taking place an error message is displayed and the NCU remains in the stop state.

Restart the machine project after configuring the communication settings. The new communication settings are active.

4.3.2.2 Configuring access points at the CP and S7Online in MyVirtual Machine

Configure the communication settings for the access points in tab "Settings" in Create/Run MyVirtual Machine.

Precondition

- Create/Run MyVirtual Machine and SINUMERIK Virtual CNC software are installed on the computer.
- Create/Run MyVirtual Machine is open

This is how you configure communication settings

1. Click the "Settings" button .
2. Select tab "General settings" in the window that opens.
3. In selection list "Access point 1 - S7Online / CP1543", select the interface name of the hardware that is used (first network adapter). The access points for S7Online and CP1543 are configured at the same time.
   The selection of possible interfaces is logically prefiltered.

   Note

   You can display information about a specific function using the button.

   (Corresponds to communication setting "Configuring access points at the CP and S7Online (Page 29)")
4. Optional, if you are using a second access point:
   In selection list "Access point 2 - CP1543_2", select the interface name of the hardware that is used (second network adapter).
   The selection of possible interfaces is logically prefiltered. If you do not use the second access point, the selection remains empty or select the "empty" entry
   (Corresponds to communication setting "Configuring the second access point at the CP (optional) (Page 31)")

   Note

   Only different interfaces can be selected for access point 1 and access point 2.

5. Activate "Support TIA device search".
   When the switch is activated, the device search for a locally installed TIA Portal is supported. For the network adapter set in access point 1, function "DCP" in S7Online is deactivated. DCP queries are responded to via the CP1543.
   (Corresponds to communication setting deactivate DCP in Section "Configuring access points at the CP and S7Online (Page 29)")
6. Click "Apply".
   The "Apply" button is only active if the communication settings have been correctly configured.
7. If a project was open, restart the machine project after configuring the communication settings. The new communication settings are active.

Result

You have now configured the access points for the application. Create/Run MyVirtual Machine can be used.

More information

If SIMATIC STEP7 Professional and Create/Run MyVirtual Machine are installed on different computers / virtual machines, you must still make settings for cross-computer TCP communication. Observe the Preparing cross-computer communication via Ethernet section.

4.3.2.3 Configuring access points in the communication settings

Configuring access points at the CP and S7Online

You configure the PG/PC settings using the "Siemens communication settings" application.

Precondition

Create/Run MyVirtual Machine and SINUMERIK Virtual CNC software are installed on the computer.

This is how you configure communication settings

1. Enter "Control Panel" in the search field of the Windows taskbar and open the displayed application.
2. Select "All control elements".
3. Click "Communication settings". The "Siemens communication settings" window opens.
4. Click "Access points" in the navigation. The available access points are displayed in the right-hand area.
5. Click the arrow in front of "S7ONLINE" to display the settings.

6. In the "Used interface parameter assignment" field, select the entry "<network adapter>.TCPIP.1". <network adapter> is a placeholder for the interface name of your deployed hardware.

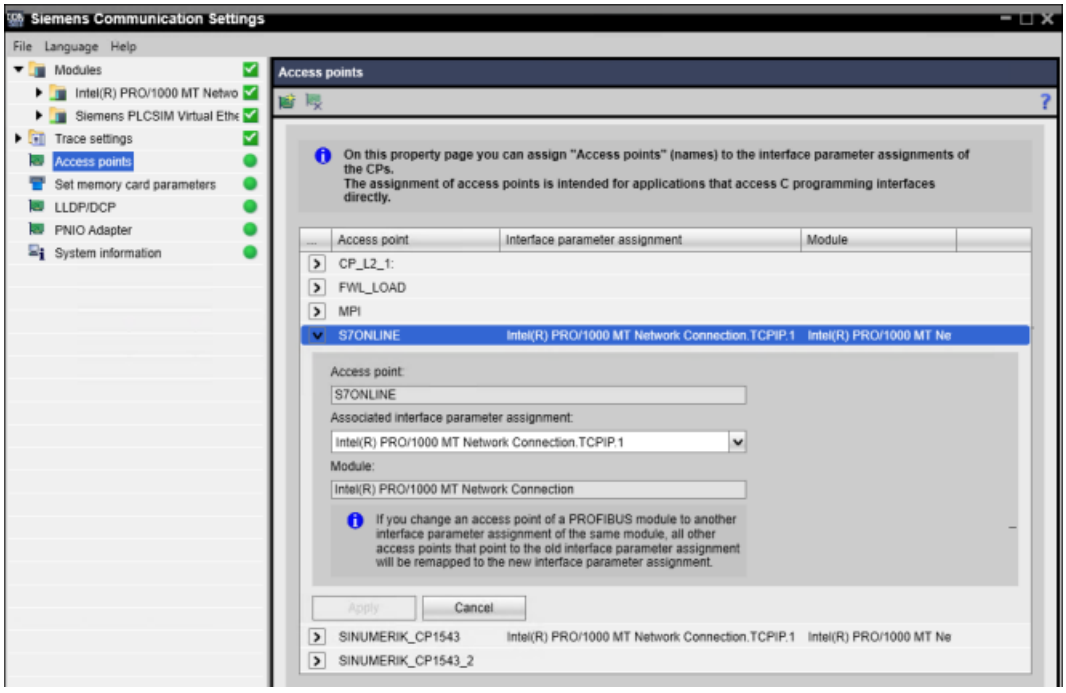


Figure 4-2 Setting the access point

7. Click "Apply".
8. Click the arrow in front of "SINUMERIK_CP1543" to display the settings.
9. In the "Assigned interface parameter assignment" field, select the entry "<network adapter>.TCPIP.1". <network adapter> is a placeholder for the interface name as used previously.
10. Click "Apply".
11. Click on "LLDP/DCP" in the navigation. The available protocols are displayed in the area on the right.

Note

To ensure communication mechanisms/device search between Create/Run MyVirtual Machine and TIA Portal, you need to deactivate the DCP function on the computer with Create/Run MyVirtual Machine (S7DOS). If the DCP function is disabled, then the "Livesearch" function is also functional when searching for the target system for Create/Run MyVirtual Machine. For the network adapter set in access point 1, function "DCP" in S7Online is deactivated. DCP queries are responded to via the CP1543.

If DCP is not deactivated, the connection to an incompatible device may be detected and reported in TIA Portal.

12. Uncheck the DCP box for the network adapter "<network adapter>" (same adapter as in the section above under point 9).

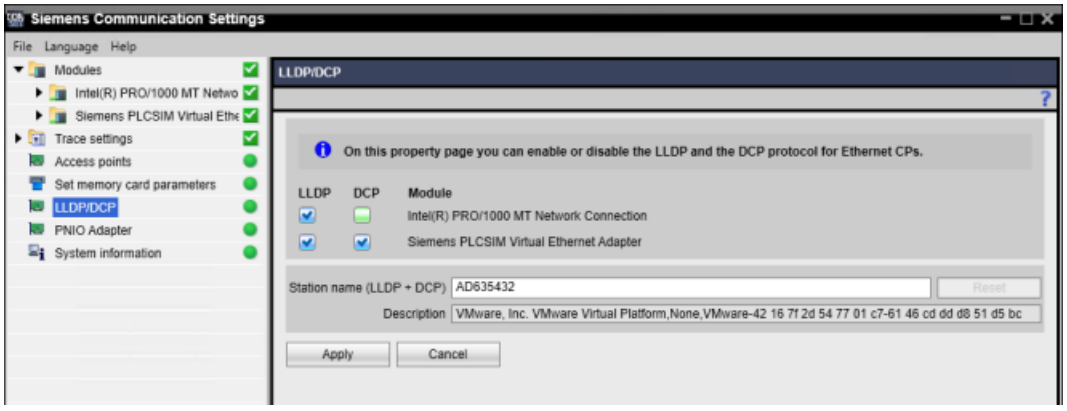


Figure 4-3 Disabling the DCP function

13. Click "Apply".
14. Select "File > Exit" in the menu to close the window.
15. Restart the computer to apply the changes.

Result

You have now configured the access points for the application. Create/Run MyVirtual Machine can be used.

More information

If SIMATIC STEP7 Professional and Create/Run MyVirtual Machine are installed on different computers / virtual machines, you must still configure the settings for cross-computer TCP communication. Observe the Preparing cross-computer communication via Ethernet section.

Configuring a second access point at the CP (optional)

The CP1543 can optionally communicate via a second network. For this purpose, you can assign another network adapter to the SINUMERIK CP1543_2 access point provided for this purpose. This can make sense if communications should be established to an external component via the second network adapter.

Without assignment, the system behaves as before and uses only the two previously known access points S7ONLINE and SINUMERIK CP1543.

Only when the second access point is assigned to SINUMERIK CP1543_2 will two interfaces actually be processed by the CP.

Precondition

- Create/Run MyVirtual Machine and SINUMERIK Virtual CNC software are installed on the computer.
- The "Siemens communication settings" are open (See, Configuring the first access point at the CP (Page 29)).

This is how you configure the second access point in the communication settings

1. Click "Access points" in the navigation. The available access points are displayed in the right-hand area.
2. Click the arrow in front of "SINUMERIK_CP1543_2" to display the settings.
3. In the "Used interface parameter assignment" field, select the entry "<network adapter>.TCPIP.1". <network adapter> is the placeholder for the interface name of the second network adapter.

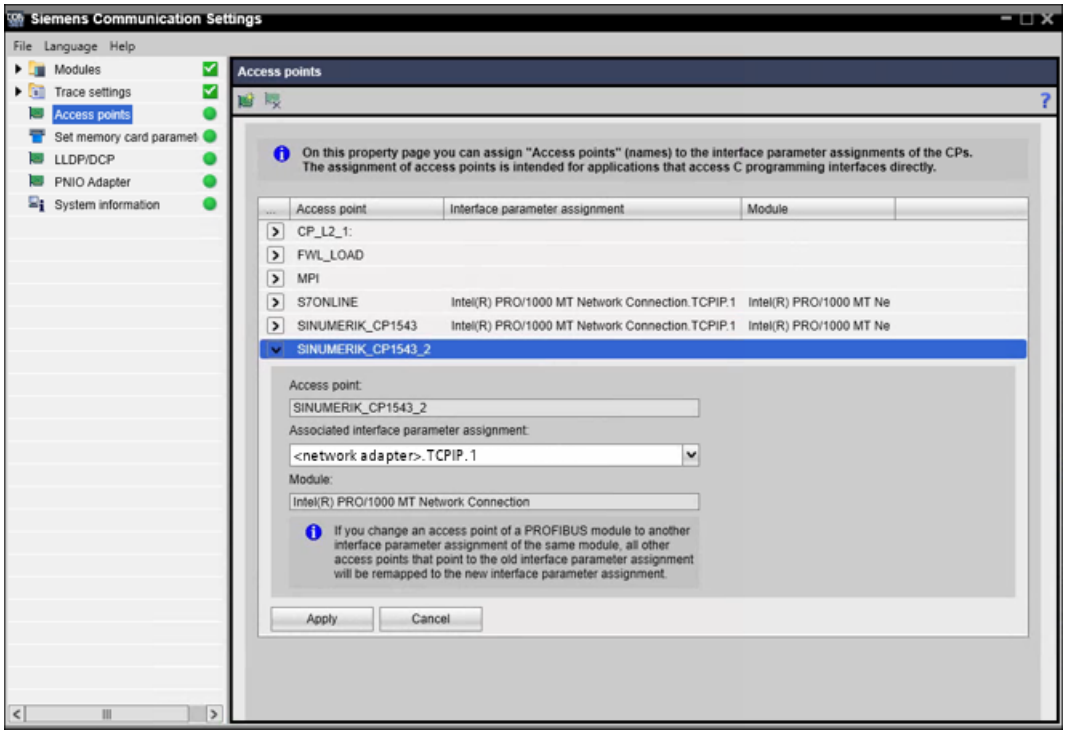


Figure 4-4 Setting the second access point CP1543_2

4. Click "Apply".
5. When using "Livesearch", deactivate the DCP functions on the second network adapter (See Configuring the first access point at the CP (Page 29)).
6. Select "File > Exit" in the menu to close the window.

4.3.2.4 Communication HMI via alternative network adapter

If the HMI is to communicate with SINUMERIK Operate not via the primary network adapter (localhost) but via the second assigned network adapter, the IP address assigned at the second network adapter must be entered in the mmc.ini configuration file of HMI Operate.

Requirement

- The Create/Run MyVirtual Machine project is open to modify mmc.ini on the virtual memory card.
- The second access point CP1543_2 (Page 31) is configured

How to enter the IP address of the second network adapter

1. In Create/Run MyVirtual Machine, click  . The Windows Explorer opens with the storage location of the virtual memory card in the directory ...\card\. (%LOCALAPPDATA% \Siemens\Automation\SINUMERIK ONE\ncu\card).
2. Navigate to the directory "..\addon\sinumerik\hmi\840evo_vc\cfg" and open the file "mmc.ini" in the editor.
3. Navigate to the line with the IP address ADDRESS1=**127.0.0.1**.
   ```
   [840EVOVC}
   ...
   ADDRESS1=127.0.0.1,Line=20,NAME=/PLC,MAX_BUB_CYCLICS=1000
   ```
4. Enter the IP address of the second network adapter instead of the IP address **127.0.0.1** of the localhost.
5. Save the file.

The HMI communicates via the alternative network adapter after starting the machine project.

General conditions

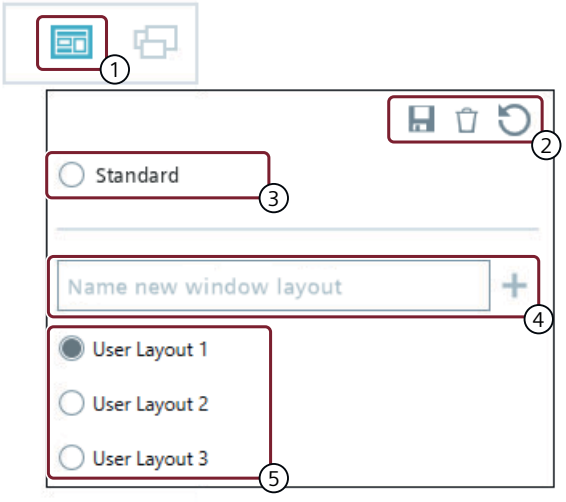
- The IP address is only stored in the changed machine project. If the HMI is to generally communicate via the second interface, you must change the IP address in each machine project.
- If the IP address of the network adapter changes, you must also add it in mmc.ini. This may be the case, for example, with dynamic IP address assignments via DHCP.
- If the changed machine project is used on another computer, the IP address must also be adjusted.
- If you want to switch between network adapters (e.g. between WLAN and LAN), then the configuration of the access points and mmc.ini may have to be adjusted.

4.3.3 Window arrangement - standard, user-defined

4.3.3.1 User-defined window arrangement

You define the arrangement of the windows and which windows are displayed via the standard window arrangement or via the user-defined window arrangements. The standard view is specified and changes cannot be saved. User-defined window arrangements can be freely configured.

Managing window arrangements



① Click on the button to display or close the menu of the window arrangements.

② Save or delete an active user-defined window arrangement or reset it to the last arrangement that was saved

③ Standard window arrangement

Changes to the standard view cannot be saved. The standard view cannot be deleted.

If you wish to display the application in the standard view, then activate "Standard".

④ Create a new user-defined window arrangement

Enter a name and click on the plus symbol "+". A new window arrangement is created, based on the current arrangement.

⑤ List of available user-defined window arrangements

In the example "User Layout 1" is active

Figure 4-5 Managing window arrangements

See also

Creating a new window arrangement (Page 35)

Reset of the window arrangement to the standard (Page 35)

4.3.3.2 Creating a new window arrangement

Using user-defined window arrangements, adapt the view of Create or run MyVirtual Machine in order to create the optimum window arrangement for the particular task. For example, if you work with 3D simulation, undock the window and shift it to a second monitor. Once you have set the window arrangement, save it as user-defined view and toggle between the different views. You can create as many user-defined window arrangements as required.

This is how you create a user-defined window arrangement

1. In the navigation bar, click on the "Define or edit window arrangement" button .
2. In the window that opens, enter a new name in the "Name of new window arrangement" field.
   The name is marked red if characters or special characters, which are not permissible, are used in the name. Only letters, numbers and spaces are permitted in the name.
3. Click on the plus symbol "+". A new user-defined window arrangement is created with the name that was entered.
4. Use the buttons in the title bar of each window to undock individual windows or to shift a window to another position. You can minimize or close windows that are not required. Appropriately adjust the window arrangement.
5. In the menu of the window arrangement, click on "Reset window arrangement" , if you wish to reset changes that have not been saved.
6. After completing the operation, click on "Save" . The current view is saved in the user-defined window arrangement.
7. You can toggle between the individual views by clicking on the window arrangements that have been created.

See also

User-defined window arrangement (Page 34)

4.3.3.3 Reset of the window arrangement to the standard

You can set a modified window arrangement back to the original view by clicking on the "Reset window arrangement" button. Selecting "Standard" corresponds to the specified view.

This is how you reset a modified window arrangement back to "Standard"

You have changed the window arrangement in the "Standard" window arrangement, and you want to reset this to the original state.

1. In the navigation bar, click on the "Define or edit window arrangement" button .
2. Check whether the "Standard" window arrangement is active.
3. In the menu of the window arrangement, click on "Reset window arrangement" .
   The view is reset to "Standard".

This is how you switch from a user-defined window arrangement to "Standard"

You want to switch from a user-defined window arrangement to the "Standard" view.

1. In the navigation bar, click on the "Define or edit window arrangement" button .
2. Activate the "Standard" window arrangement.
   The view is set to "Standard".

4.4 Information application

You can display version information about the applications that have been installed by pressing the Info button.

This is how you display the version information

1. Click the button.
2. The following information is displayed in the window that opens:
   – Version of the installed software

4.5 Project settings

4.5.1 Adapting the HMI resolution

The screen resolution of the CNC user interface (HMI SINUMERIK Operate) in the machine project can be set. This gives you the possibility to adjust the representation of the machine project as best as possible to the respective screen resolution of your simulation PC.

Requirement

You can only change the HMI resolution in the "Project settings" when the project is open. The machine simulation is in the stop state.

How to adapt the HMI resolution

1. In the opened "Project settings" viewlet, select the respective resolution from the list.

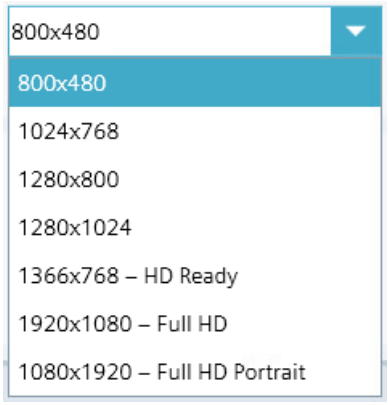


Figure 4-6 Selecting the HMI resolution

2. Start the machine project by pressing button "Start machine".
   The machine project is started with the selected resolution of the HMI.

Note

The available HMI resolutions depend on the respective version of the virtual CNC software.

Working with Run MyVirtual Machine

5

5.1 Manage machine projects

Introduction

Machine projects (*.vcp; **V**irtual **C**ommissioning **P**roject) are managed in the Run MyVirtual Machine project management. In this view, you open and delete projects. You create new projects based on the templates.

A machine project manages all required data for the machine operation. The machine project file contains NC, HMI, PLC and drive data stating the version of the CNC software used.

Machine project and CNC software version

To edit a machine project, the matching version of the CNC software must always be installed on the computer. A machine project can neither be created nor opened without the CNC software being installed. If the version of the CNC software used in the project is not installed, an error message stating the missing CNC software is displayed. In this case, first install the stated version and then edit the machine project.

Multiple CNC software versions can be installed on a single computer. The CNC software version used in the machine project is displayed in the project management in the "CNC version" column and in the title bar of the open machine project.

Starting the project management

On the desktop, double-click the "Run MyVirtual Machine" symbol - or select "Start > Siemens Automation > Run MyVirtual Machine". Run MyVirtual Machine project management is displayed.

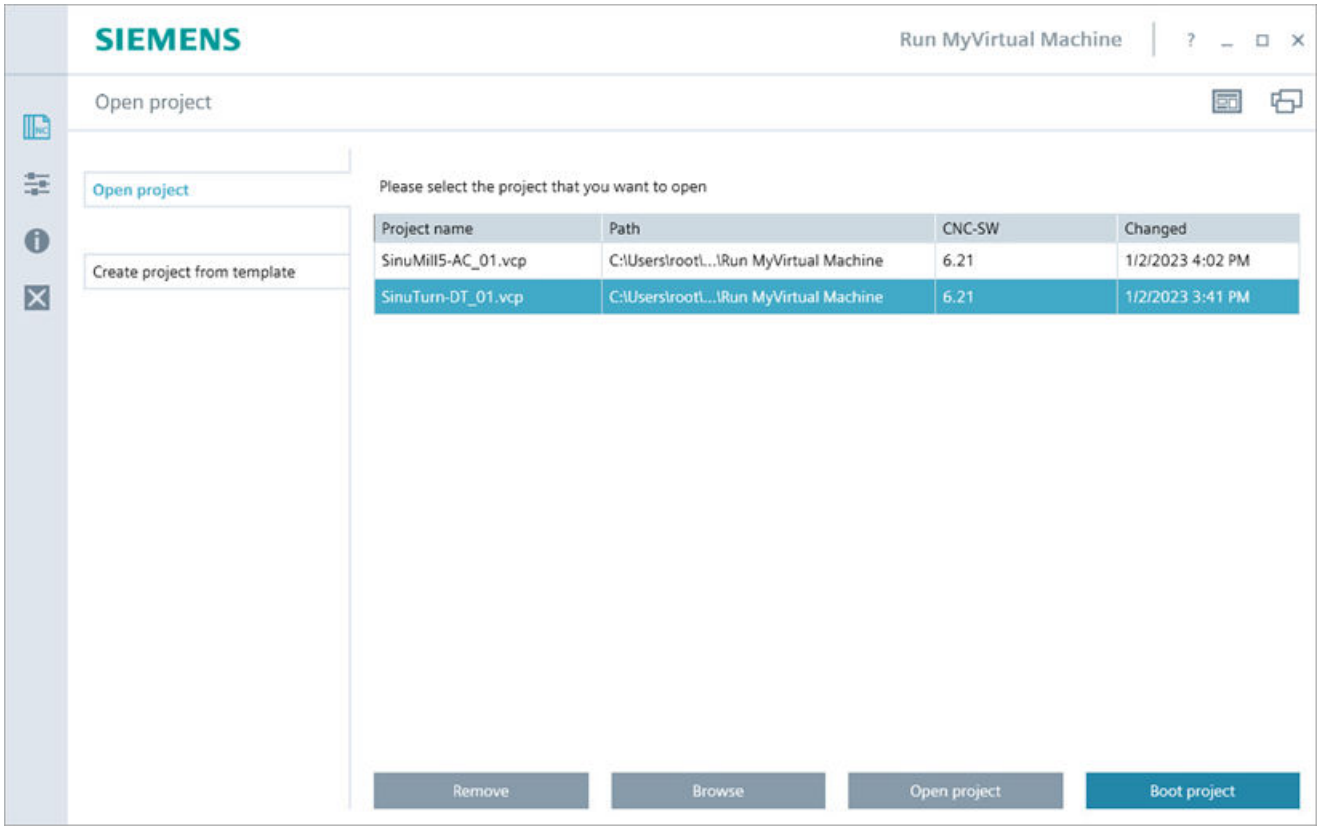


Figure 5-1 Project management of Run MyVirtual Machine - open project

5.2 Creating a machine project from a template

Description

Various template projects are available in the project management to help you create machine projects.

Note

Installing template projects

The template projects are made available via a dedicated installation package. Install the machine projects if no template projects are displayed in the project management.

More information is provided in the *Installation Manual Template Machines Create/Run My Virtual Machine*.

This is how you create a new machine project

Start Run MyVirtual Machine.

1. Click on "Create project from template".
2. Select a template project.
3. Enter a project name.
4. Select the storage path.
5. Confirm with "Create project". The project opens.

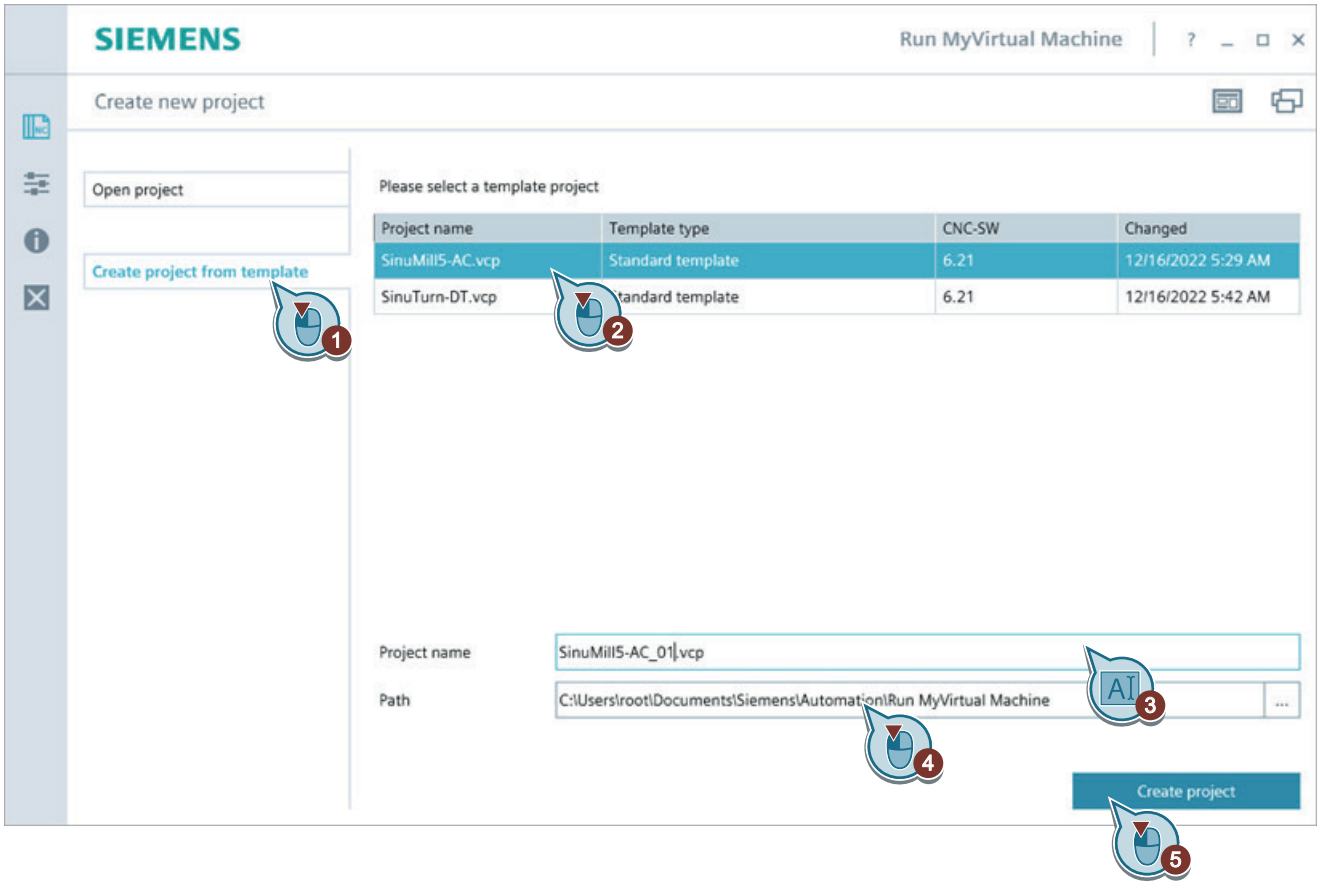


Figure 5-2 Creating a new project from a template

6. The Run MyVirtual Machine project view is displayed with "Project settings" and default values.

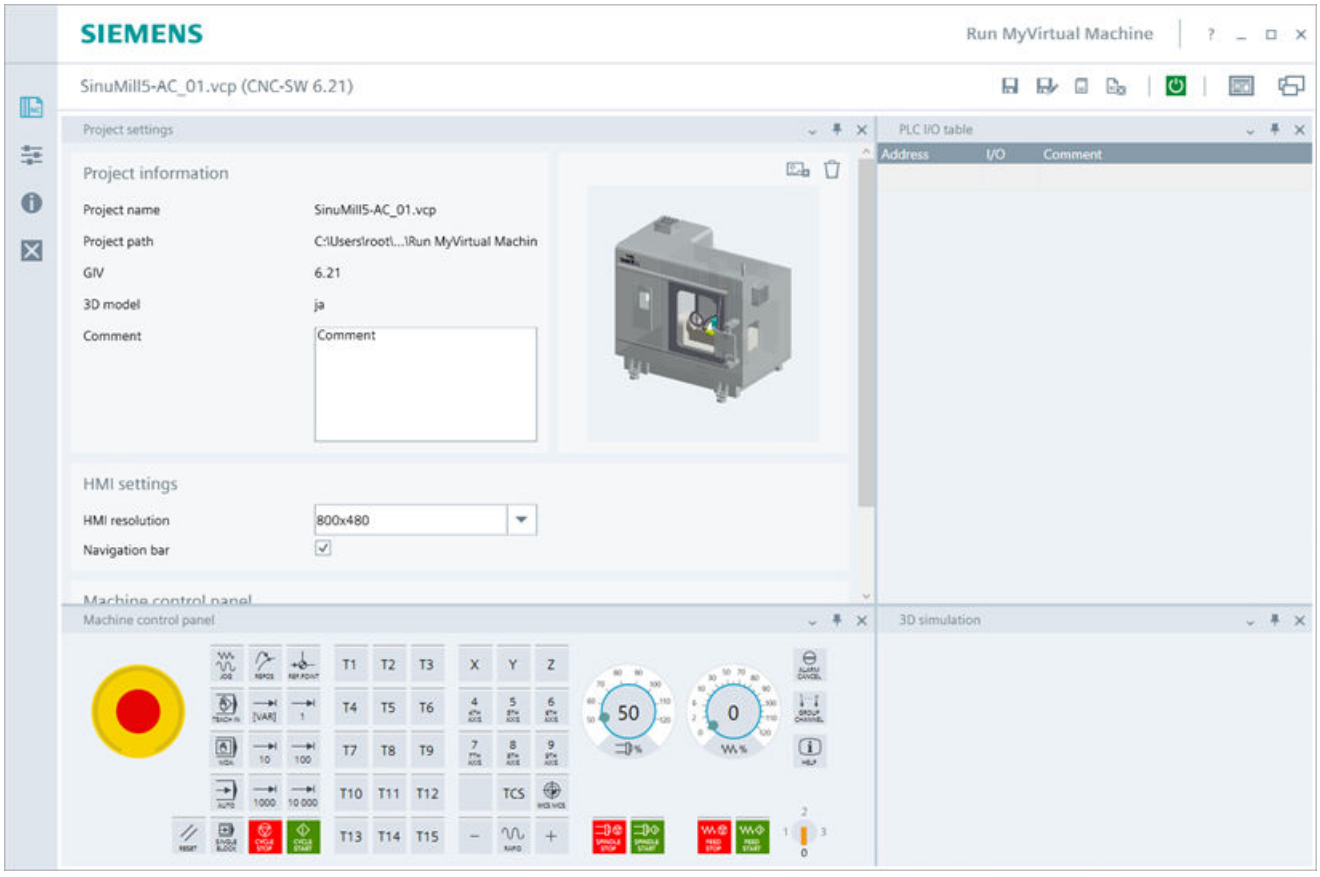


Figure 5-3 Run MyVirtual Machine with an open machine project

7. Configure the project in Section "Project settings".
   - Optionally, enter a comment and load a typical machine diagram for the project.
   - Select "HMI resolution" and display of the "Navigation bar" in the HMI.

   Then start the machine in the next step. This brings the machine into the simulation mode. The starting corresponds to the switch-on and run-up of a real machine.
8. Click the "Start machine" button in the control area. The button can be clicked only when it is no longer faded.
   The machine is started and the HMI with machine control panel opens. The machine cannot be operated during starting.
   The start of the machine is completed when the button "Exit machine" is displayed.

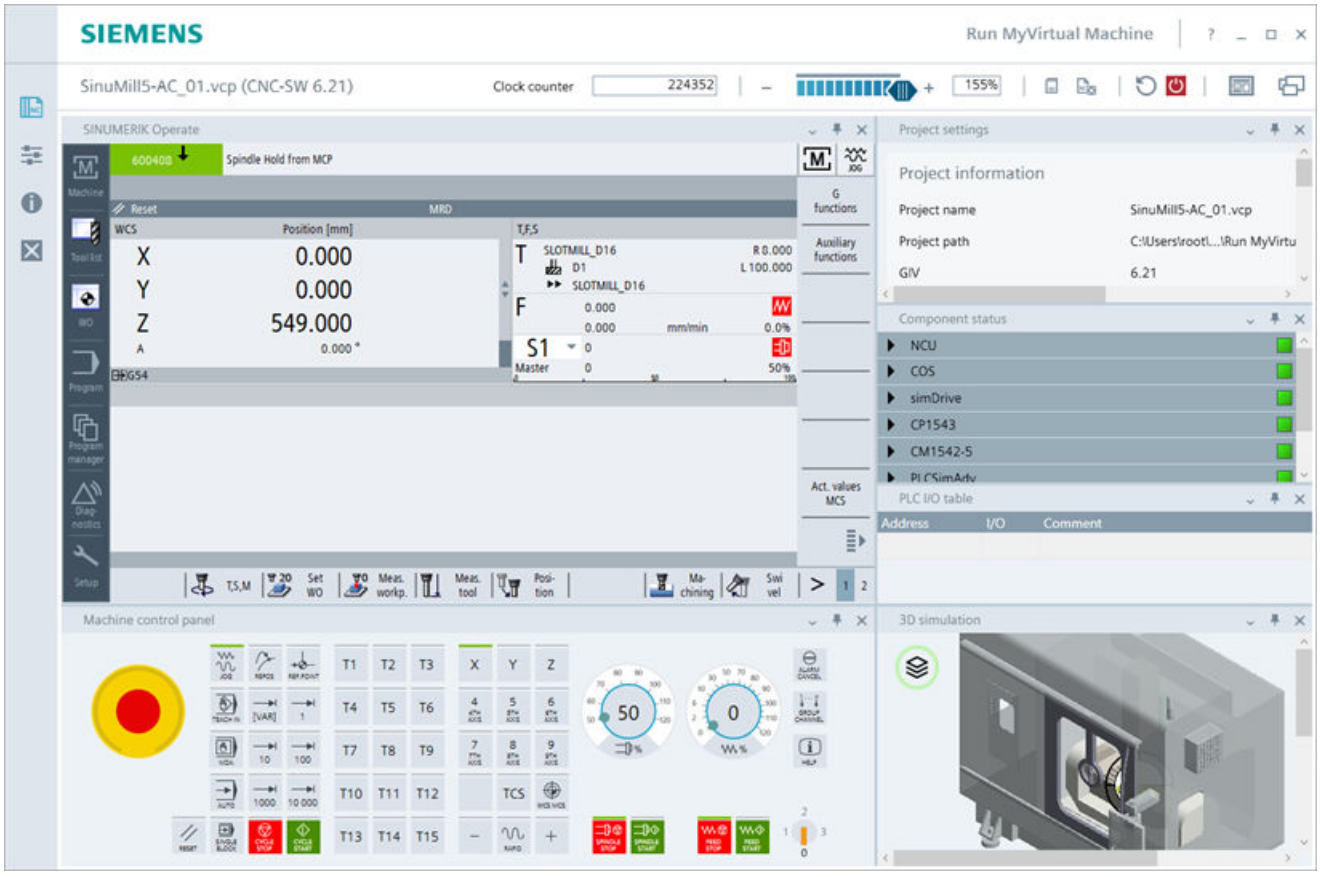


Figure 5-4 Run MyVirtual Machine: machine project newly created from the template after running up

5.3 Opening a machine project

Description

Open existing machine projects in the project in the project management.

How to open an existing machine project

Start Run MyVirtual Machine before opening an existing machine project.

1. Click "Open project".
2. Select the project if the project is on the list of recently opened projects. The CNC SW version used is also stated in the line of the project.
3. Click the "Open project" button. The selected project opens. Continue with step 8.

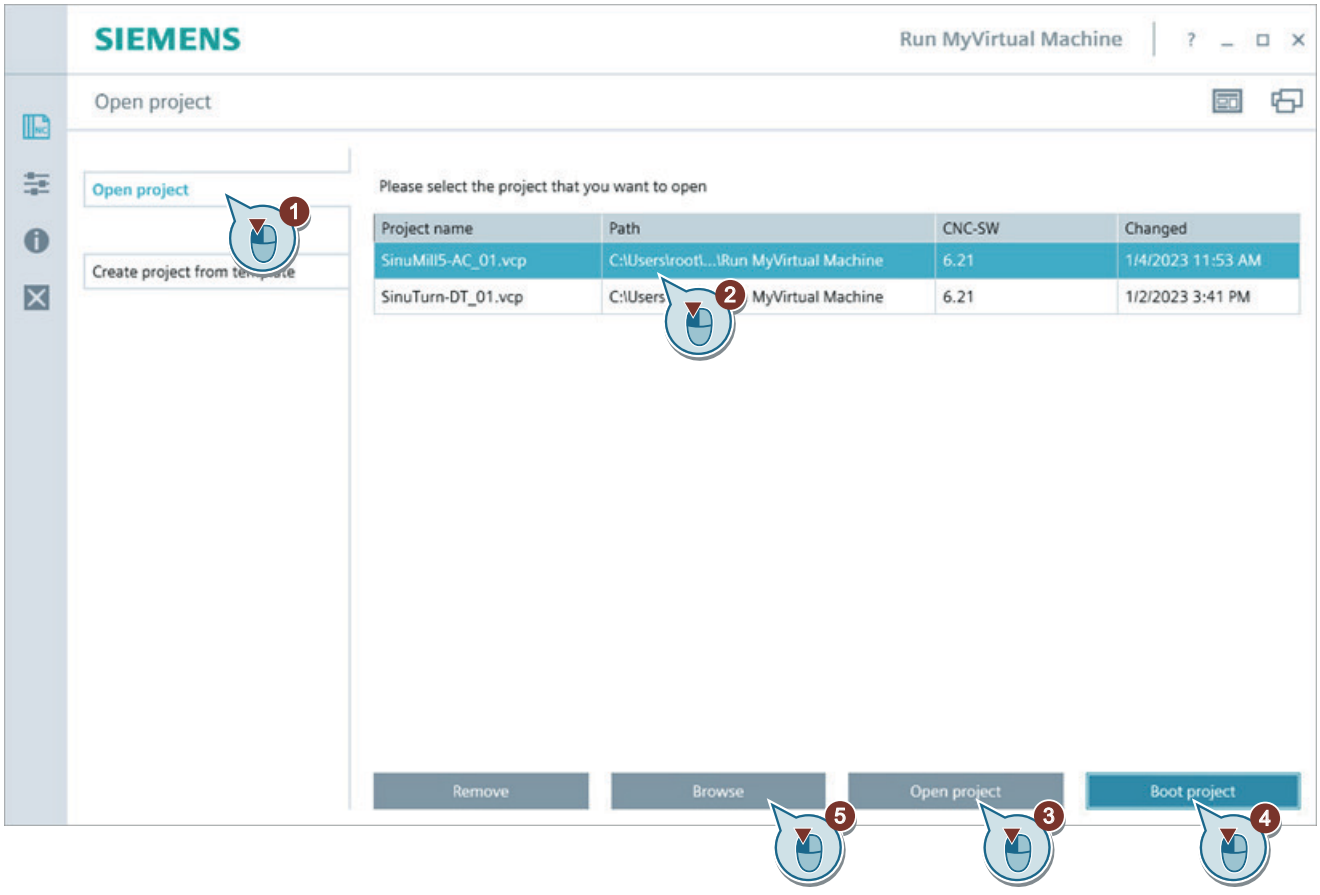


Figure 5-5 Opening a project in Run MyVirtual Machine

4. As an alternative to steps 3. and 8., you can also click on button "Start project". The project opens and the machine project is started.
5. If the project is not in the list, click the "Browse" button. The file selection dialog opens.
6. Navigate to the desired project. You have access to all local drives and network drives via the selection dialog.

7. Select the project and click "Open". The project is opened in Run MyVirtual Machine.

Note

Virtual CNC SW

Whether the Virtual CNC SW version used in the project is installed on the computer is checked when the project is opened. If the Virtual CNC SW version is not installed, a message is output stating the missing version. This Virtual CNC SW version must be installed before the project can be opened and edited.

When the project is open, the Virtual CNC SW version used in the project is displayed in the title bar.

8. Then, to start the machine project, click on button "Start machine".

The machine project is automatically added to the list of last opened projects.

5.4 Save the machine project and exit

Description

The current state of the machine project is backed up on saving. When the machine project is next opened, you continue working with the most recently saved status. Machine projects can be saved only when the simulation control has Stop status.

File size of machine projects

The machine project is saved as a compressed *.vcp file. The maximum size of the stored machine project is limited to 2 GB. The saving of larger machine projects is aborted with an error. If the machine project cannot be saved due to the file size, delete unused user data from the virtual memory card, for example.

This is how you save machine projects

1. In the control area, click the "Exit machine" button. The machine is exited.
2. Select "Save" or "Save as" from the menu bar. The save dialog is displayed
3. Save the project directly, or select a location where you want to save it – and enter a new name for the machine project.
4. Confirm with "Save". The machine project is saved with file extension "*.vcp".

Note

If you exit the program or the project without saving the current status, a message appears. You can also save the current status of the project subsequently.

5. Select "Close project" from the menu bar. The Run MyVirtual Machine project view is exited and the program management is displayed.
6. Select "Exit" in the program management. The project management and Run MyVirtual Machine are closed.

5.5 Cross-version use of machine projects

Note

Using machine projects in another version

The portability of machine projects ("*.vcp files) to next-higher versions is not guaranteed.

5.6 Project template and access rights

Access rights and keyswitch position

Machine projects that were created from the project template can generally be used with access level User and in keyswitch position 0. When simulating or when executing the program examples included, it is possible that an error message is output as write access rights are required once. These are not permissible in keyswitch position 0.

Adapting the keyswitch position

If simulation or executing the program examples is not possible, then the problem can be resolved by adapting the keyswitch position.

1. Start the machine project.
2. Set the keyswitch on the machine control panel to position 3.
3. With the new access rights it is possible to simulate and execute the program examples.
4. Save the machine project if you wish to save keyswitch position 3 in the project.

Setting the password for users

Alternatively, in SINUMERIK Operate in area "Commissioning" you can also set password "CUSTOMER" for access level 3.

5.7 Virtual memory card

Description

Just the same as on the real SINUMERIK control, the virtual SINUMERIK ONE also has a memory card (SD Card) to store data and manage programs. The virtual memory card is created temporarily in the Windows User directory when a machine project is started. Each machine project has its own virtual memory card.

When the machine project is exited, the structure and contents of the virtual memory card in the machine project are saved, and deleted from the Windows User directory. The virtual memory card is available when the machine project is next started.

When you save the machine project, the contents of the virtual memory card are stored in compressed form in the machine project. The storage capacity of the virtual memory card is limited by the maximum size of the machine project (max. 2 GB). Delete unused user data from the virtual memory card if the machine project cannot be saved due to the file size.

The structure and the handling of the virtual memory card correspond to those of the real SD Card.

Virtual memory card storage path

The virtual memory card is only displayed when the machine project is started. The virtual memory card is saved under the following path:

- C:\Users\<username>\AppData\Local\Siemens\Automation\SINUMERIK ONE\ncu\card

Exchanging data between an NC and a virtual memory card

You can exchange data, e.g. NC programs, with the NC via the virtual memory card. The following shows the directories on the virtual memory card and the corresponding directory in the program manager in SINUMERIK Operate.

Directory on virtual memory card

- C:\Users\<username>\AppData\Local\Siemens\Automation\SINUMERIK ONE\ncu\card**\user\sinumerik\data\prog**

Directory in the SINUMERIK Operate program manager

- Softkeys: "Program manager > NC Extend"

Note

To exchange data between an NC and a virtual memory card, the key switch on the machine control panel must be at position 3.

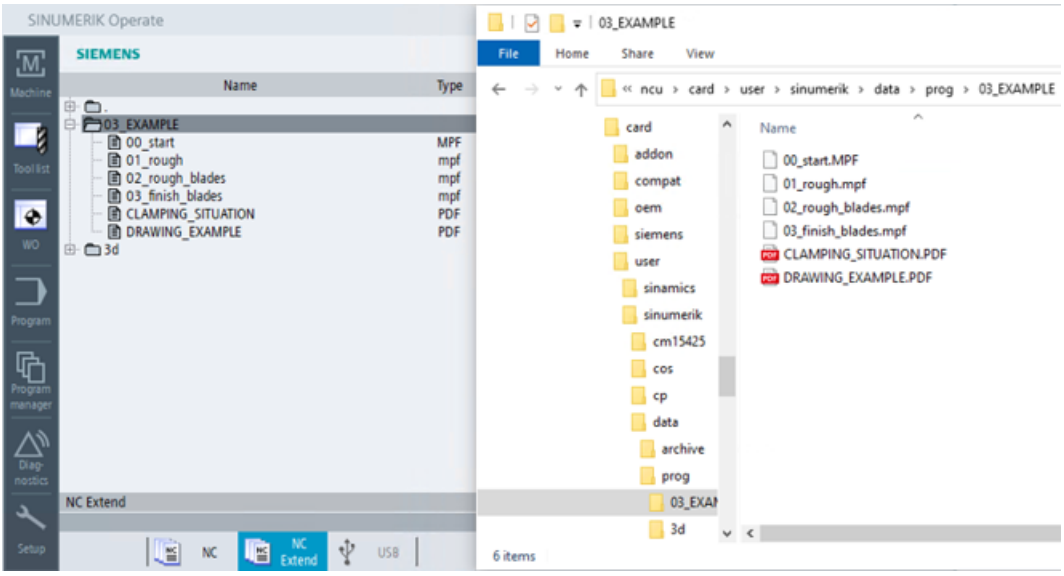


Figure 5-6 Same directory opened in the program manager and Windows Explorer on the virtual memory card

3D simulation (option) 6

6.1 Product information

This product includes or uses libraries of the following software:

- Open CASCADE Technology (https://dev.opencascade.org/)

6.2 Using 3D simulation

Description

Run MyVirtual Machine visualizes the machining process and machine movements using 3D simulation. You can simulate the processing of NC programs in AUTOMATIC mode, for example, or manual traversing movements and tool changes in JOG mode.

Note

License

The "Run MyVirtual Machine /3D" license is required to use 3D simulation.

NOTICE
Saving 3D simulation data in the machine project All changes in the 3D simulation (e.g. storage folder, library, setups) are only finally saved after saving the machine project. Before closing the machine project, you should save the current state so that no changes are lost.

Range of applications for 3D simulation

Currently, 3D simulation offers collision monitoring for CNC machine components in specific kinematic arrangements, as well as stock removal simulation when drilling, milling and turning. Contact your machine OEM regarding the availability of specific machine applications ("*.vcp" projects) or template machines.

Overview of 3D simulation

The 3D simulation is divided into functional areas. To switch between the areas, click on the corresponding tab.

Figure 6-1 3D simulation, operating areas and functions using the example of a milling machine

You can perform the following tasks in the areas:

- ① Machine
  In the "Machine" tab, you can configure the simulation components and visualize the machining process.
  - ⑤ Simulate the machining process and machine movements
  - ⑥ Define tools
  - ⑦ Select and activate the setup (clamping)
  - ⑧ Save the current setup as library elements (current machining state of the workpiece and setup)
  - ⑨ Activate collision detection
  - ⑩ Show/hide machine menu, reset view/workpiece
  - ⑪ Show/hide layer menu
- ② Library
  In the "Library" tab you can manage the components required for the simulation.
  - Tool components/holders
  - Setup (clamping)
  - Blanks
  - Protection area (workholder)
- ③ Collisions
  In the "Collisions" tab, the detected collisions are logged during the execution of an NC program or during the manual procedure.
- ④ Settings
  - Language switchover
  - Tool holder diameter
  - Exporting/importing archives

See also

Settings (Page 64)

Automatic mode with 3D simulation (Page 133)

Components in the library (Page 68)

Tools (Page 90)

Collisions (Page 128)

Setup (clamping) (Page 117)

More information

More information with example data and documentation with detailed examples can be downloaded from the Siemens Service Portal (https://support.industry.siemens.com/cs/ww/en/view/109815969).

6.3 Functional scope and fundamentals

6.3.1 Function scope of the 3D simulation

Machine processes and machine motion are visualized in the 3D simulation:

Functional scope

- Spatial simulation of machine movements on the basis of a stored kinematic machine model (geometries and kinematic description)
- Simulation of drilling, turning and milling operations
  - Simulation of stock removal for single and multi-channel machining processes on the basis of the active setup and tool data
  - Several tools being used
  - Resetting of the workpiece to the original geometry of the blank
  - Monitoring of collision (can be switched on/off) between collision groups specified in the machine model
  - Display of a log of detected collisions and deletion of the log entries
- Turning with driven tools in conjunction with the Y axis
- Simulation of measuring processes with switching probe
- Support of inch and metric systems of units
- Display of tools from the SINUMERIK tool management based on their geometric values
- Support of various types of tools
- Linking tool holders (either as "*.stl" and "*.stp" geometries or basic parameterizable geometry elements (primitive model)) with tools from the SINUMERIK tool management
- Creating, deleting and editing protection areas (workholders) ("*.stl" and "*.stp" geometries) in a library
- Creating, deleting, editing and managing blanks ("*.stl" or "*.stp" geometries or from basic geometry shapes) in a library
- Parametric changing of the spatial position and orientation of library components
- Creating, deleting and editing setups (clampings) from the library component protection area and blank
- Exporting and importing library components and setups
- Activating one or several selected setups for consideration in the spatial simulation
- Saving and exporting the in-process workpiece geometry

Figure 6-2 3D simulation in Create MyVirtual Machine

6.3.2 3D simulation restrictions

The following functions are explicitly not supported for the 3D simulation:

- Simulation of grinding operations
- Consideration of tools deviating from the tool types listed in the document (deviating tools are shown as type 120 end mill)
- Using file formats other than "*.stl" and "*.stp" to import 3D geometries
- Parallel kinematics
- All channels are considered as measuring during a simulated measurement operation
- Turning operations are restricted to the tools defined in the document

6.3.3 Constraints 3D simulation

Simulation of stock removal and collision detection have the following properties.

Properties stock removal

Stock removal with the tool cutting edge is simulated if the following requirements are met.

- The machine is not in rapid traverse (≠ G0).
- The tool spindle is rotating (milling machine or rotating tools on turning machine).
- The workpiece spindle is rotating (turning machine).
- Stock removal is independent of collision detection (active/inactive).
- Stock removal is independent of feedrate/speed of rotation.
- Stock removal is simulated when using machining cycles in JOG mode.
- Stock removal is also simulated if the above conditions are met and other components collide (e.g. machine with workholder).
- During the manual process in JOG with rotating spindle, no stock removal is simulated.

Properties collision detection

Collision detection has the following properties.

- Collisions are displayed and stored in the log.
- Collision is displayed in color.
  - Orange
    The minimum clearance between colliding parts within the safety clearance is met. The default value for the safety clearance is 3 mm. This can be configured in the machine model (machine manufacturer).
  - Red
    Collision of the components has been detected.
- If the conditions stated above in section "Properties stock removal" are not met, the contact of the tool cutting edge with the workpiece is detected as a collision.
- Collisions in rapid traverse (G0) are always detected irrespective of whether the spindle is rotating or stopped.
- Collision is detected while the spindle is stopped and during traversing movement G1, G2 and G3.
- A collision is detected for collision pairs that are configured in the machine model (machine manufacturer).
- There is a special mode for "measuring". Measuring processes are not detected as collisions. More information can be found in the Measuring processes in 3D simulation (Page 132) section.

Simulation properties

As a result of the motion modeling, simulation has the following properties.

- To model motion (axis motion) the 3D simulation directly uses the setpoints from the NC, and not the actual motion executed by the drive system. Functions, which deactivate the setpoints at the drives, e.g. when using simulation axes or for the program test (PRT) therefore have no impact on the 3D simulation. Although the drives are not assigned any setpoints, axis motion is executed in the 3D simulation.

6.3.4 Simulation of measurement procedures and machine data

Probe

Measurement performed with switching probes is supported in all operating modes and can be carried out with active collision avoidance.

The following tool types (probes) are supported:

- 710-714
- 725
- 730

Measuring cycles in the AUTO/MDI mode

During a measuring cycle in the AUTO/MDI mode, the NC sends signal MEAS1 or MEAS2, depending on the type of measuring cycle, i.e. tool or workpiece measuring cycles. These signals are interpreted by considering the bit values of MD52740 $MCS_MEA_FUNCTION_MASK. The bits in $MCS_MEA_FUNCTION_MASK define the significance of the MEAS1 and MEAS2 probe signals.

- Bit 0 = 0: MEAS1 is connected with a workpiece probe
- Bit 0 = 1: MEAS2 is connected with a workpiece probe
- Bit 16 = 0: MEAS1 is connected with a tool probe
- Bit 16 = 1: MEAS2 is connected with a tool probe

If bit 0 = bit 16, the measuring cycle type is determined based on the tool type in the spindle. If the tool in the spindle is a probe (see Section Probe above), then the MEAS signal corresponds to a workpiece measuring cycle. Otherwise, the MEAS signal corresponds to a tool measuring cycle.

Measuring process and collisions

Collision components treat measuring cycles as a special case, so that a collision alarm is not initiated for certain collision types.

These types of collisions include:

- In the workpiece measuring cycle
  Collisions between the probe/tool shank and other objects, e.g. machine parts, workpieces or fixtures.
- In the tool measuring cycle
  Collisions between the tool shank/unclamped length and the tool measuring system.

If there is still a collision between the tool shank and other objects, when the MEAS signal is deactivated, collisions between the tool shank and other objects will continue to be ignored until the tool leaves the collision state. Only afterwards is general collision avoidance reactivated.

The tool measuring system must be defined in the machine model. A generally valid specification has currently still not been defined. This is the reason that tool measuring systems will only be supported in a future version.

The NC (NCK version ≥ 4.93 or ≥ 6.13) displays alarm 26352 if a measuring cycle is active in the special collision type cases listed above. This warning is deleted as soon as the measuring cycle is exited.

Note

Currently, only workpiece measuring cycles are supported. The use of tool measuring cycles will be implemented in future versions.

Measuring cycles in JOG

A manual measuring cycle can be executed in JOG. However, the machine must be in no-load operation (idling) to activate the manual measuring cycle via API. During the manual measuring cycle, the collision avoidance component does not initiate any collision alarm for collisions between the tool shank/unclamped length and another part of the machine, the workpiece or fixtures.

The NC (NCK version ≥ 4.93 or ≥ 6.13) displays alarm 26352 if a measuring cycle is active in the special cases of the collision component listed above. This warning is deleted as soon as the manual measuring cycle is deactivated.

A manual measuring cycle can either be deactivated via the API if the machine is in no-load operation (idling mode) or by switching into the AUTO operating mode.

Restrictions for Create MyVirtual Machine

If machine data MD13210 $MN_MEAS_TYPE is = 1, measuring in Create MyVirtual Machine does not function.

6.3.5 Cutting off the workpiece and transferring to a counterspindle

Machining and transferring workpieces between the main spindle and counterspindle are supported in the 3D simulation.

Transferring the workpiece

Transferring workpieces between two clamping operations, e.g. transferring a premachined workpiece from the main spindle to the counterspindle for turning machines is supported in the 3D simulation.

Workpiece transfer sequence

1. In the settings, activate function "Activate/deactivate workpiece transfer".
2. Activate a setup with a blank on the main spindle.
3. Activate a setup without a blank on the counterspindle.
4. The workpiece transfer procedure is started if the limitation frame of the setup in the counterspindle intersects with the limitation frame of the setup in the main spindle.
5. The transfer is realized as soon as the distance between the Mount Stations increases.

Cutting off the workpiece

3D simulation functionally supports cutting off the workpiece and subsequently transferring it.

Cutting off sequence

1. In the settings, activate function "Activate/deactivate workpiece cutting off".
2. Activate a setup with a blank on the main spindle.
3. Activate a setup without a blank on the counterspindle.
4. The workpiece transfer procedure is started if the limitation frame of the setup in the counterspindle intersects with the limitation frame of the setup in the main spindle.
5. When the counterspindle grips the blank and moves in the opposite direction, the blank is transferred and moves together with the counterspindle to the initial position.
6. If the blank is cut before or after the transfer, some of the blank remains in the main spindle and the part that has been cut off is clamped in the counterspindle.

Notes and restrictions

- Even if a collision pair is defined between the setups of the main spindle and counterspindle, collisions between the workpiece to be transferred and the clamping fixture to grip the workpiece should be ignored if a workpiece transfer is involved.
- If both spindles have workpieces, a transfer is not made and a collision is documented if a collision group was defined.
- A workpiece falling when clamping or when cutting off is not simulated.

6.3.6 Traversing positioning axes (POS, POSA)

Positioning axes are traversed independently of the path axes at a separate, axis-specific feedrate. With the POS/POSA commands, the positioning axes are traversed and the sequence of motions coordinated at the same time.

Behavior of commands POS/POSA in 3D simulation

The following applies when programming commands POS/POSA:

- Material removal and collision avoidance/detection based on G code Group 1 and the spindle state/status of the axis that is positioned.

6.3.7 Tips

6.3.7.1 3D Simulation is not started/displayed

Error case

The 3D Simulation is not displayed or is terminated immediately after starting the machine project.

Requirement

First check that the following requirements are met.

- The system requirements are fulfilled.
- The option "Create MyVirtual Machine /3D" or "Run MyVirtual Machine /3D" is licensed.
- The machine project contains a 3D machine model.

Possible cause graphics card driver

If the installed graphics card driver is not compatible with the graphics hardware or offers insufficient OpenGL support, the 3D Simulation cannot be started.

Remedy

If possible and available, install an up-to-date graphics card driver with OpenGL support.

Possible cause remote connection

If you use Create MyVirtual Machine or Run MyVirtual Machine with the 3D option activated via remote access through Microsoft "Remote Desktop Connection" (RDP) from a second computer, there may be display errors or a black 3D window due to the lack of OpenGL graphics drivers for this RDP transmission protocol. This depends on the graphics hardware used or the installed graphics drivers on the remote computer on which Create MyVirtual Machine or Run MyVirtual Machine is running.

Remedy

A possible remedy is to install a software OpenGL renderer, such as the "Mesa 3D" freeware on the remote computer running Create MyVirtual Machine or Run MyVirtual Machine.

6.4 Operation and setting

6.4.1 Data archiving, export/import, file format

Storage folder, data exchange 3D Simulation

The 3D Simulation uses a storage folder on the virtual memory card. As default, this folder is used, for example, to store the exported data or to search for the data to be imported (e.g. library). Geometry data can only be exchanged with the library using this folder, e.g. geometries of tool holders and workpieces.

You can find the folder on the virtual memory card in your user directory.

- "C:\Users\<username>\AppData\Local\Siemens\Automation\SINUMERIK ONE\ncu\card\user\sinumerik\3d\"

Note

New storage folder

As of version SINUMERIK Virtual CNC software V6.15, the 3D Simulation uses the new path specified above.

If the term "storage folder" is used in the following description, the specification refers to this path.

NOTICE
Saving data in the storage folder in the machine project The data in the storage folder is only finally saved in the machine project when you save the machine project. All changes since the last time the machine project was saved are no longer available after exiting the machine project without saving.

Geometry data, geometry files, 3D geometries

The 3D simulation can use the geometry data of tool holders, workpieces, protection areas and setups in the following formats:

- STL file ("*.stl")
- STEP file ("*.step", "*.STEP", "*.stp, "*.STP")

When geometry data, geometry files and/or 3D geometries are discussed in the documentation, then it always involves files in these formats.

6.4.2 General information on operation

Tips and tricks when working with 3D simulation

When working with the 3D simulation, you are supported by helpful functions. The special features are listed below.

- The 3D simulation is operated with mouse and keyboard. The mouse should have a scroll wheel.
- Generally, when entering values, only valid values are permitted. Values cannot be saved or messages are displayed.
- If an error is detected, a red error message is displayed in the upper right area. You can close it by clicking it or the "X" in the error message.

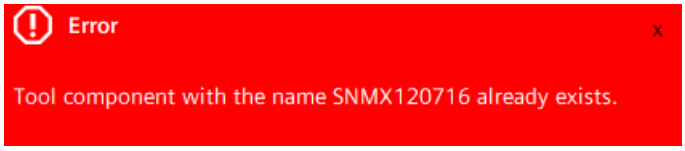


Figure 6-3 Example of an error message

- Correctly performed actions, e.g. saving, are indicated by a green message in the upper right area.
- Notes or warnings, e.g. an active component cannot be edited, are displayed as a yellow message in the upper right area.

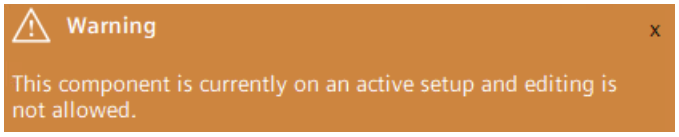


Figure 6-4 Example of warning/information

- More information, help screens and videos for the entry fields or functions are displayed when you click on the i button.
- The view of all 3D images, e.g. for library components or in the 3D simulation, can be zoomed and moved to see details better.
  To do this, click in the 3D image and change the view.
  - Zooming the view: rotate mouse wheel up/down
  - Moving along the axes: left mouse button and move mouse
  - Rotating the view: right mouse button and move mouse
  - Operation via the keyboard (Page 61)

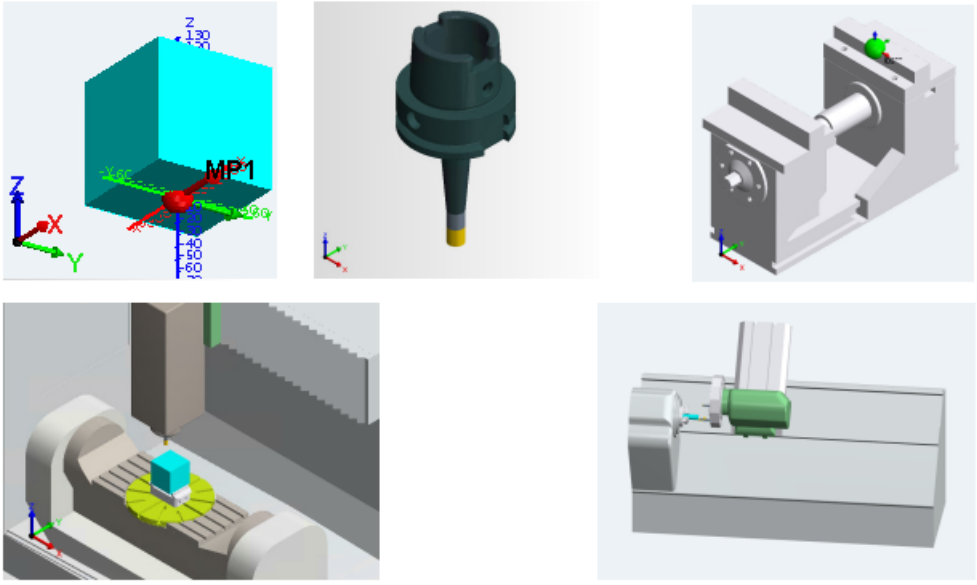


Figure 6-5 Examples of 3D images that can be zoomed, rotated and moved

6.4.3 Operating 3D simulation using the keyboard

The view of all 3D images, e.g. for library components or in the 3D simulation, can be zoomed and moved to see details better. You can also use these functions via the keyboard.

Functions of the keys and key combinations

The most important functions of the keys are listed in the function overview.

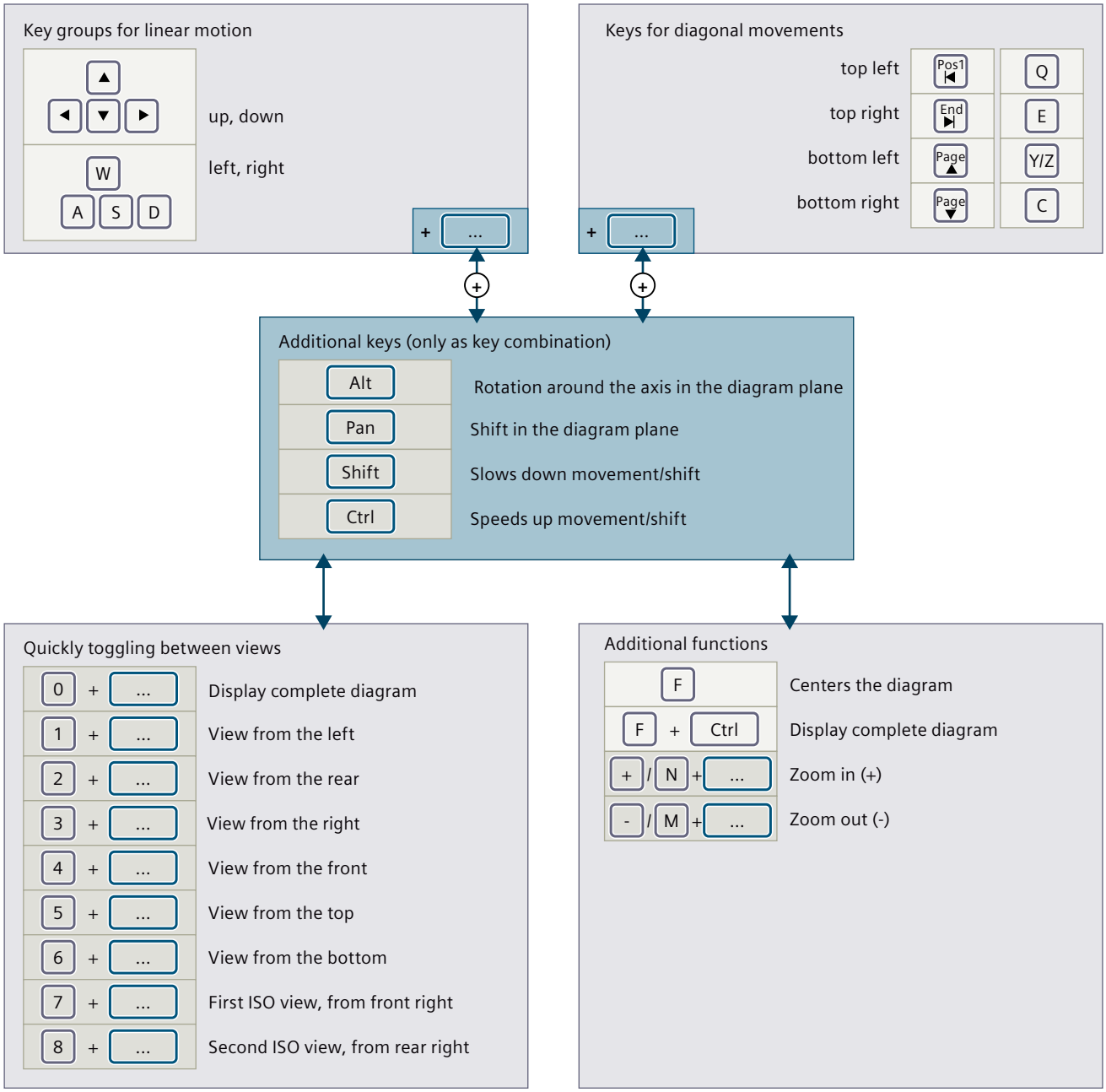


Figure 6-6 Functions of keys in 3D simulation

6.4.4 Using layers

The 3D simulation used layers to hide certain layers during the simulation. Depending on the machine model, you can show or hide different layers in the 3D simulation. By default, all layers are displayed.

Examples of layers:

- Symbol of the active work offset
- Machine housing
- Tool table
- Setup (clamping operation)

Displaying and using layers

You define the visible layers using the layer menu.

1. For example, in the Machine view, click the Layer button to display the menu.
2. Click the button of the layer you want to hide. The button will be displayed in red.

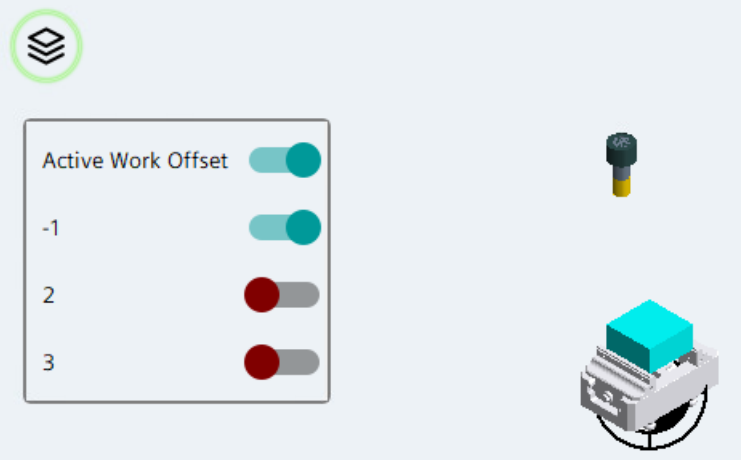


Figure 6-7 Showing and hiding a layer: machine housing and table hidden

3. Click the Layer button to hide the menu. The layer setting is saved centrally and used again at the next start.

Layer settings

The display of layers is used in the "Machine", "Setup" and "Collisions" views. The layer settings are centrally managed and the same settings are used in all views.

6.4.5 Settings

6.4.5.1 Settings

In the "Settings" tab you select the interface language and define a standard diameter for the tool holder and export/import archives.

Changing the tool holder diameter

For 3D simulation, you define a default value for the tool holder diameter. The default value (diameter 50 mm) is only used if the protection function "Fast Protection" is defined for the active tool in the simulation.

How to change the tool holder diameter

1. Enter a new value for the "Holder diameter for fast protection" in the "Settings" tab.
2. Click "Save".
   Changing the holder diameter only becomes active when "fast protection" is redefined. Existing definitions of "fast protection" are not adapted in the tool manager.
3. Switch back to the "Machine" tab.

Changing the user interface language

The 3D simulation is available in several user interface languages.

How to change the user interface language

1. Select the new user interface language in the "Settings" tab under "Language".
   The user interface language is switched to the selected language.
2. Switch back to the "Machine" tab.

Exporting/importing archives

In archives you can save settings and data of the 3D simulation or exchange them between two machine projects.

More information on archives can be found in the Exporting/importing archives (Page 65) section.

Settings for workpiece transfer/cutting off

Activate the settings to allow cutting off functions with subsequent workpiece transfer, e.g. into the counterspindle in the 3D simulation.

- Activating/deactivating workpiece transfer
- Activating/deactivating cutting off workpiece

Additional information on this function is provided in Section Cutting off workpiece and transferring to the counterspindle.

CAD settings when using "*.stp" files

You enter the tolerances here for the converting process from STEP to STL. When converting from STEP to STL, the 3D model is approximated using triangles. Using these tolerances, you configure permissible tolerances for deviation to the model.

Tolerance	Description
Chord tolerance	The chord tolerance specifies the maximum distance between a chord and the real curved surface in mm or inches. The value of the chord tolerance lies between 0.001 mm (0.00003 inches) and 0.1 mm (0.00039 inches). The default value is 0.08 mm (0.00315 inches).
Angular tolerance	The angular tolerance specifies the maximum angle between adjacent chords in degrees. The angular tolerance lies between 0 and 90 degrees. The default value is 18 degrees.

This is how you change the tolerances

1. In tab "Settings", under CAD settings, enter the values for chord and angular tolerance.
2. Click "Save". The tolerances become effective.
3. Switch back to the "Machine" tab.

6.4.5.2 Exporting/importing archives

For data backup or transfer between machine projects, you can export and import "*.zip" archives in the 3D simulation. You export/import the archive in the "Settings" tab.

Data in the archive

The archive contains the following data.

- All library components (e.g. blanks, tool components)
- Tool data (tools with defined protected areas)
- Machine model
- Settings (holder diameter, language)

Note

You can only export/import archives when 3D simulation is not active.

Exporting an archive

How to export the 3D simulation data to an archive.

1. Click on the "Export" button.
   The "Export archive" window with the selection of the data to be exported and any already existing archives is displayed.
2. Select the export data. By default, all data is exported.

   Note

   Tool data can only be exported together with the library data.

3. Enter a name for the "*.zip file" or keep the suggested name.
   The length of the name is limited to 40 characters.
4. Click "Export".
   The data are saved as "*.zip" file in the selected folder. Once the export is done, a message about the successful export is displayed.
5. Check the displayed error message if the export is canceled.

Importing an archive

You can import archives in the exchange folder into the 3D simulation.

Requirement

- No setup (clamping) is active in the 3D simulation.
- The "*.zip file" and the data it contains must conform to the import format.
- If the zero point is not present in the machine from an imported setup (clamping connected to a work offset), the connection is made to the default mount station.

How to import an archive

1. Copy the "*.zip" archive to the storage folder.
2. Click on the "Import" button.
   The "Import archive" window with the selection of existing archives is displayed in the selected folder.
3. Select the archive file.
4. Click on "Import" and confirm the message with "Yes".

   Note

   Before the import, all current data (library, tool data, machine model, settings) are deleted. Changes since the last archiving are lost in this case.

   The selected archive is imported. Once the import is done, the import dialog is closed and a message about the successful import is displayed.
   If an error occurs during an import step, all changes are undone through a rollback mechanism.
5. Save the machine project before closing it to save the imported data.

6.4.6 Switching between different unit systems ("metric"/"inch")

Unit system in 3D simulation

The 3D simulation works with the system of units that has been set in SINUMERIK Operate. You can set the effective system of units directly in SINUMERIK Operate or toggle between "metric" and "inch".

Note

Switching over the system of units

After switching over the system of units in SINUMERIK Operate, the new system of units is not yet effective in the 3D simulation. To make it effective, you have to restart the NCK. To do this, you perform a machine project reset, or you end the machine project and restart it.

Requirements

- Machine project has started
- No NC program is being processed
- 3D simulation is not active

How to switch over the system of units

1. In SINUMERIK Operate, click on the extended horizontal softkey bar 2 in the JOG or AUTOMATIC operating mode.
2. Click the "Settings" softkey in the horizontal softkey bar.
3. In the vertical softkey bar, click the "Toggle inch" or "Toggle Metric" softkey depending on which system of units is active.
   SINUMERIK Operate is switched to the new system of units.
4. Click the "Reset" button to restart the machine project.
   Following reset, the unit system switchover also becomes effective in the 3D simulation. All the tool data and other geometric data are displayed and processed in the new system of units.
5. Save the machine project to ensure that the unit system switchover is permanently saved in the project.

6.4.7 Requirements for 3D simulation

Requirements

Before you run a 3D simulation with collision detection, the following requirements must be met.

- The machine model is integrated into the machine project ("*.vcp") (by the machine manufacturer) and the project is saved
- Tool component, setup, blank, protection area are defined in the library
- The protection function and protective areas (workholders) of the tools used are defined
- Setup, matching the NC program, is activated
- Collision detection is activated

You can find more detailed information in the following sections.

See also

Components in the library (Page 68)

Tools (Page 90)

Setup (clamping) (Page 117)

Collisions (Page 128)

Simulating machining operations and workflows (Page 132)

6.5 Components in the library

6.5.1 Library

In the library you manage the components tool component, setup, blank, and protection area for the 3D simulation.

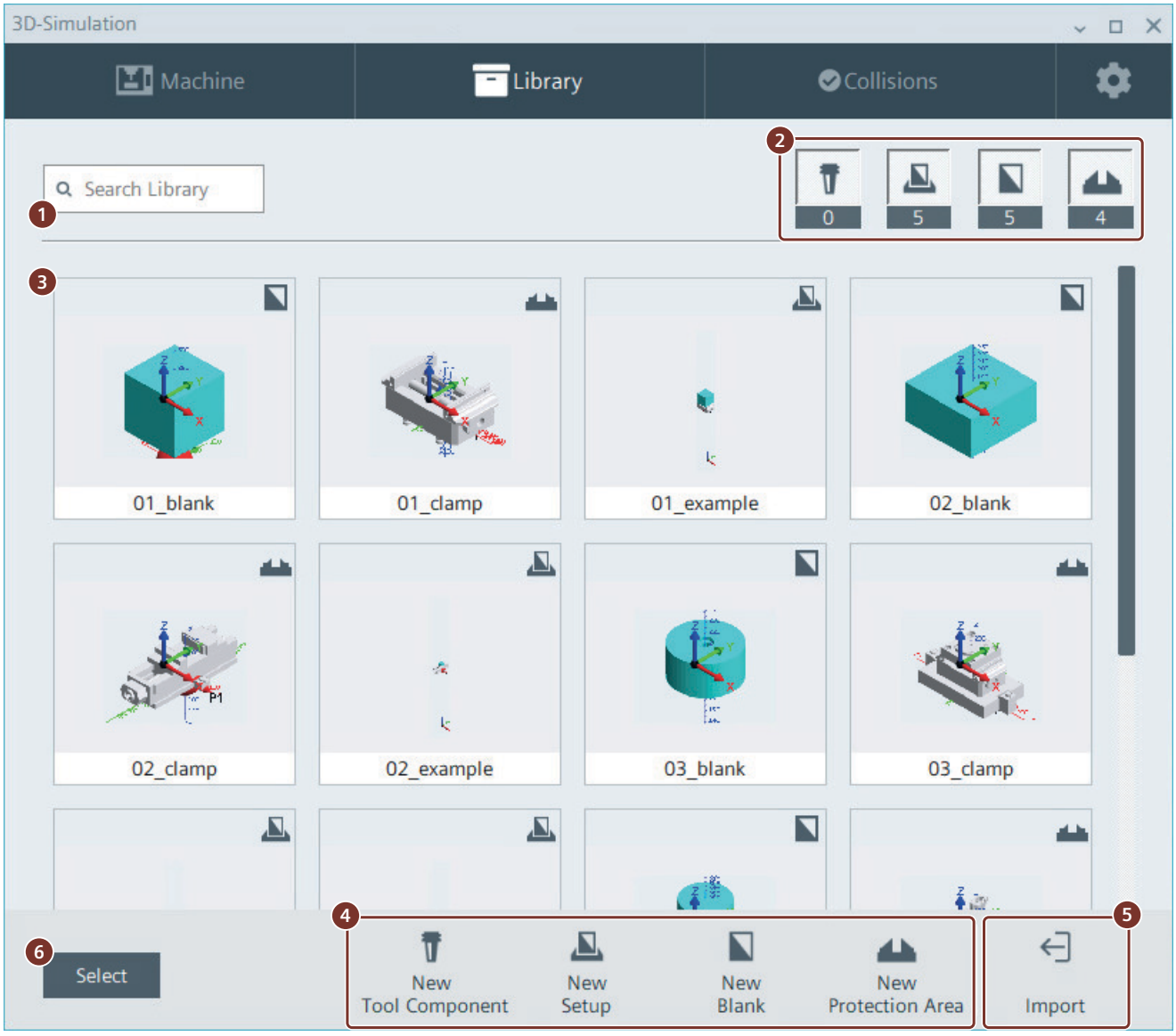
Overview of library

You create, edit and manage components in the symbolic view of the library.

Note

Active components are blocked

Components that are currently active in the 3D simulation cannot be edited or changed.



① Here you enter the term to be found in the library and press the ENTER key. The hits are displayed in the overview.
Click on the "X" in the search field to delete the search term.

② Here the configured components of the library are displayed with icon and number.
The buttons also serve as a filter option for the components. If you click on the button, it is greyed out and the components of this type are not shown in the overview. Click on the button again to show the components.

③ Here the overview of all configured components is displayed with preview image, name and symbolic icon. To edit, click on the component.

④ By clicking the buttons you create new components of the selected type.

⑤ Use Import to import library elements as "*.zip" file into the library. The imported components are extracted and displayed in the library. With this function you can import exported components from other machine projects.
The "*.zip" files in the storage folder and/or in the selected directory are available for import.

⑥ Click "Select" to select one or more components from the library. You can, for example, delete the selected components or export them as "*.zip". The exported component is saved as a file in the storage folder and/or in the selected directory.

Figure 6-8 Managing 3D simulation components in the library

6.5.2 Component types and properties

You configure and manage the following components and their parameters in the library.

Component types

In the following table the components are shown with icon and description.

Symbol	Component	Description
	Tool components	Tool adapter, tool holder For example, holders for tools with shanks with a steep or hollow taper.
	Setup	The setup consists of a protection area and the matching blank. You can also set up several protection areas and blanks, e.g. clamping towers or machine vises, in one setup.
	Blank	Workpiece blank E.g. block, cylinder
	Protection area	A protection area locates the blank. Protection areas are, for example, machine vises, chucks or clamping jaws.

Properties and parameters of components

The following table shows the parameters of the components.

Properties/parameters	Description
Component name and ID	A component has a component name that is unique throughout the project. The name can be a maximum of 40 alphanumeric characters long. Valid characters are: • A-Z • a-z • 0-9 • Underscore (_) **Note:** The character and length restrictions apply to all names assigned within the 3D simulation.
"*.stl", "*.stp"	Components are defined as 3D geometry files in the "*.stl", "*.step", "*.STEP", "*.stp", "*.STP" (3D model) formats. You create the 3D geometry using an external program and import it when you create a new component. The 3D geometries are available for import as files in the storage folder.
Geometrical shapes	Optionally, blanks can be created using parameterizable standard geometries in the library.

6.5.3 Reference system of components

Reference systems and meaning

Depending on the type, the components have different reference/coordinate systems. The coordinate origin/zero point is the basis of a coordinate system that you define for the components. The reference system can be rotated or moved.

The reference systems of the components in the 3D geometry files (clamping equipment, blanks, tool holders) usually cannot be used for 3D simulation without modifications. Therefore, after importing, you can change the zero point and rotations of the coordinate system within the components.

Note

The files with the 3D geometries for a setup (protection area, blank) should be exported from the design system with an identical component zero point. In this case, the positions for the protection area, blank, and positioning in the machine can be used without offsets.

Table 6-1 Overview of the reference systems

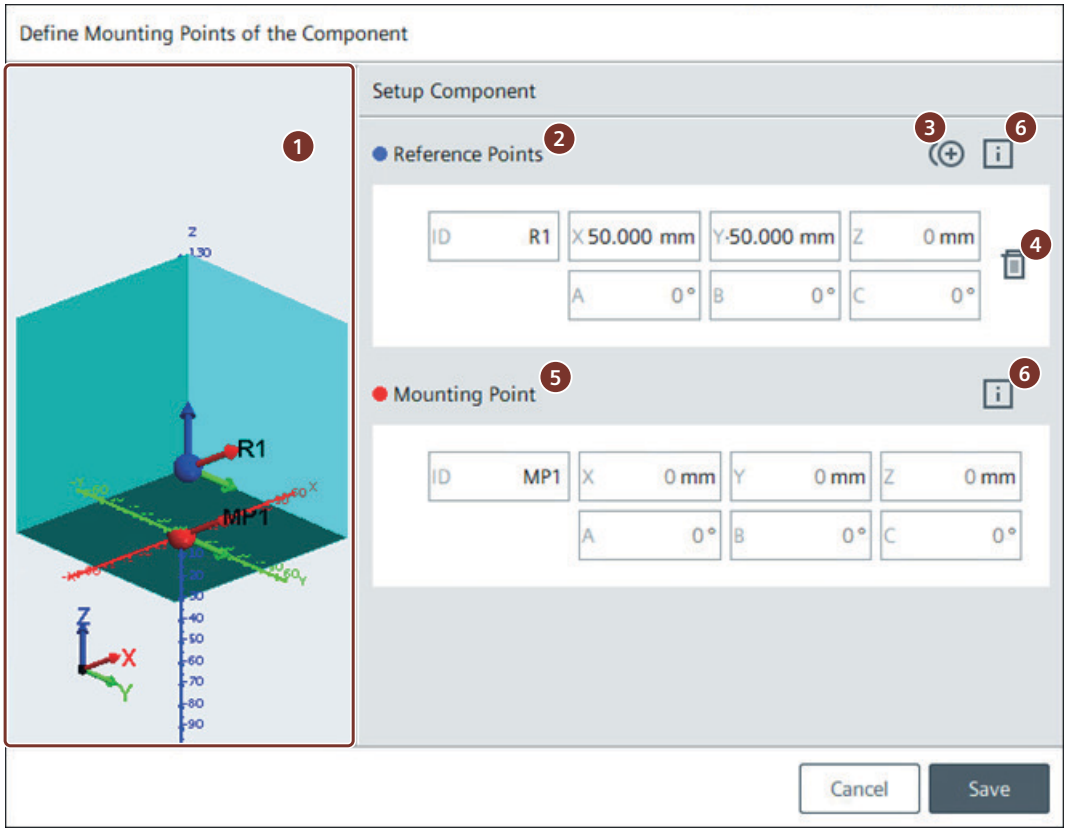
Reference system	Description
Reference Point **R**	This point is used to position the component in relation to the machine zero point. A newly created component has no reference point; only an "empty" reference point is displayed. You must first define the reference point, i.e. you must enter and save the values for the position. Blanks and clampings can have several reference points.
Mounting Point **MP** (Plug, Mounting Point)	This point is used to connect a component with other components. The Mounting Point of a component can be connected to the Mount Station of another component. Components can only have one Mounting Point.
Mount Station **MS** (Socket, Mount Station)	The Mounting Point of another component can be connected at this point. The components are aligned at the defined points. Mounting Point and Mount Station lie above one another. Using the Mounting Point and Mount Station functions, you can define a mount station on a machine vice and a mounting point on the blank. When defining the setup, the machine vice and the blank are aligned at this point.
Reference systems in the preview image	• The zero points of the reference system are displayed as a colored sphere in the preview image. • The color of the sphere in the preview image corresponds to the color code in the parameters of the reference system, e.g. the reference point is blue.

Table 6-2 Overview of the parameters of reference systems

Parameters of reference systems	Description
ID	ID of the reference system. The ID is generated automatically and should not be changed.
Coordinates X, Y, Z	The values X, Y and Z define the position of the component along the coordinate axes. Positive and negative values can be entered for the values. The value range is: [-9999.999 to +9999.999].
Rotation of axes A, B, C	The values A, B, C define the rotation of the component around the coordinate axes. Integer values between 0° and 359° can be entered.

Example of the blank component

A reference point (blue) was created in the following example at position (140, -140, 0) at the lower left-hand corner of the blank. The Mounting Point (red) lies centered at the lower side of the blank. The Mounting Point (red) lies centered at the lower side at (0, 0, 0).



① Preview image of the "*.stl/*.stp" file. You can zoom, rotate or pan the preview image.
② Coordinates of the created reference point.
③ Click "+" to create new reference points.
④ Click the "recycle bin" button to delete existing reference points.
⑤ Example of a Mounting Point, centered at the lower side of the blank
⑥ More information, help screens and videos about the input field are displayed by pressing the information button.

Figure 6-9 Example of blank with defined reference point and Mounting Point

6.5.4 Tool components

Parameter

A tool component has the following parameters.

Parameter		Description
Definition		Importing an "*.stl file" or "*.stp file" into the library with name and ID.
Reference systems		
	Reference Point	-
	Mount Station	1
	Mounting Point	1

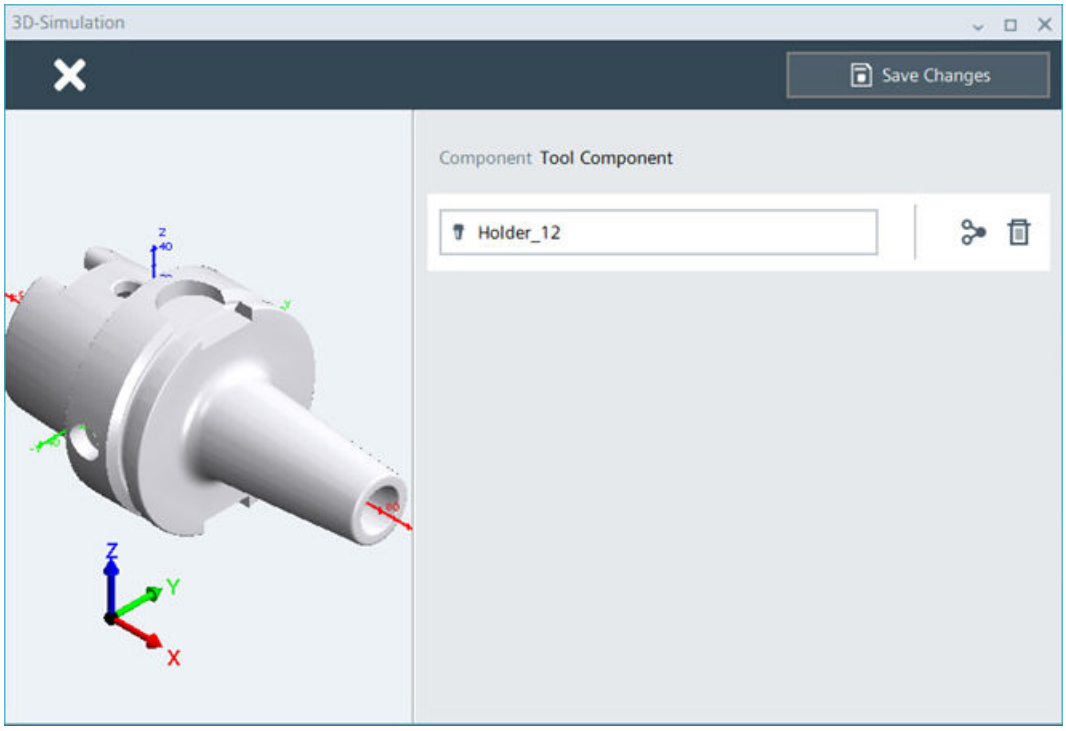


Figure 6-10 Tool component: example milling cutter holder

6.5.5 Blank

Definition

A blank has the following parameters.

Parameter		Description
Definition		Importing an "*.stl file" or "*.stp file" into the library with name and ID. Blank from "*.stl/*.stp" file (Page 79) Creating a 3D model from parameterizable, geometrical shapes, e.g. a block. Blank from a parameterizable form (Page 82)
Reference systems		
	Reference Point	n (must first be defined)
	Mount Station	-
	Mounting Point	1

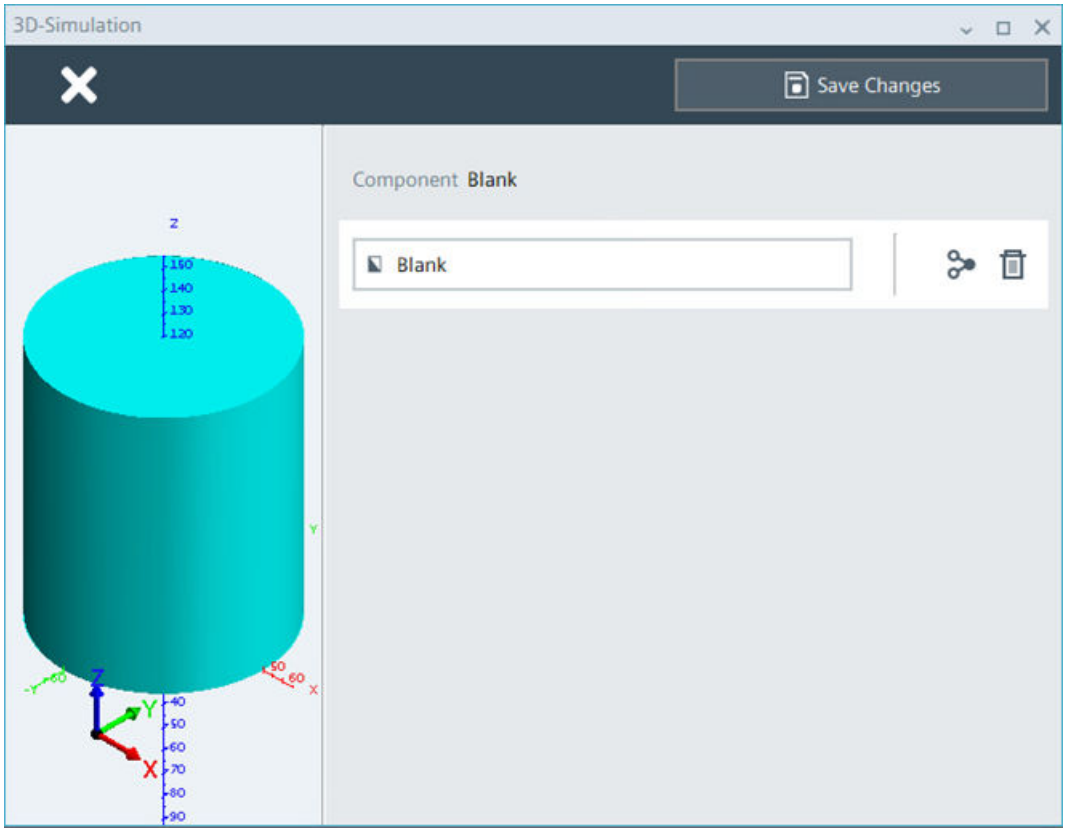


Figure 6-11 Blank imported from *.stl file and reference points defined

6.5.6 Protection area (workholder):

Parameter

A protection area has the following parameters.

Parameter		Description
Definition		Importing an "*.stl file" or "*.stp file" into the library with name and ID.
Reference systems		
	Reference Point	n (must first be defined)
	Mount Station	n (must first be defined)
	Mounting Point	1

A protection area can also consist of several components, e.g. a stationary and the movable part of a machine vise. In this case, you define several protection areas in the library and for the setup, you select the total clamping from the individual protection areas.

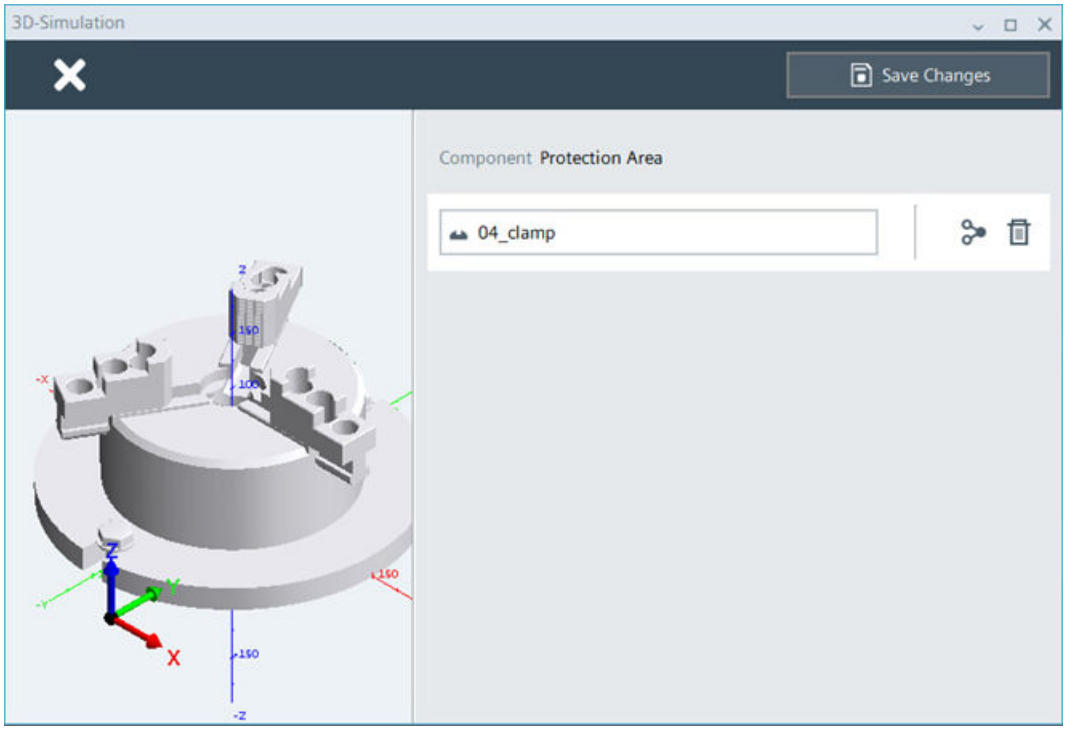


Figure 6-12 Example protection area: Machine vise

6.5.7 Setup (clamping)

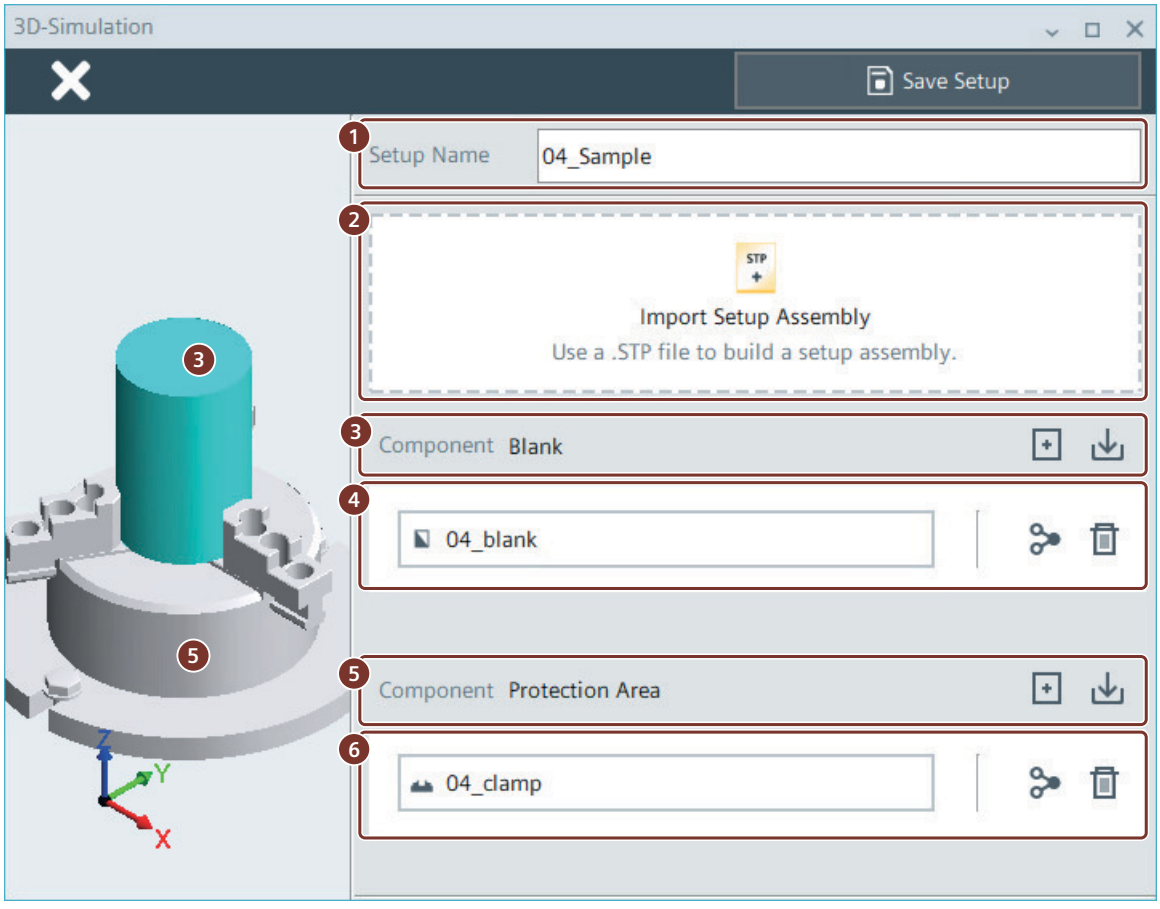
Parameter

A setup is the combination of protection area and blank and can consist of one or several blanks and protection areas, e.g. multiple clampings.

Parameter		Description
Definition		
	Blank	Import of 1 to n blanks from the library.
	Protection area	Import of 1 to n protection areas from the library. A setup can be composed of several protection areas, e.g. a stationary part and the moving part of a machine vise.

Overview of "Setup"

A setup with a blank and a machine vice (protection area) is defined in the following example. The individual setup functions are briefly explained.



① Name of the setup

② Click on button STP to import a complete setup (blank and protection area) as "*.stp" file.

③ **Blank**

Click on + to create a new blank from an "*.stl" or "*.stp" file.

Click on to import a blank from the library.

Note:

When creating a new setup, you can also create the blank from basic geometrical shapes. To do this, click on the button.

④ Name of the blank

Click on to configure the reference systems.

If the protection area has more than one Mount Station, then for the blank you must still assign the Mounting Point and the corresponding Mount Station (Page 86). If a position is not assigned, then the Mounting Point is automatically assigned to the Mount Station defined in the machine model.

Click on to delete the blank.

⑤ **Protection area**

Click on to create a new protection area from an "*.stl" or "*.stp" file.

Click on to import a protection area from the library.

⑥ Name of the protection area

Click on to configure the reference systems. For protection areas with several Mount Stations, also define the assignment for the Mounting Point of the blank.

Click on to delete the protection area.

Figure 6-13 Example of a setup with blank and protection area

6.5.8 Creating components using the example of a blank

You create components in the "Library". The creation procedure is identical for all component types except "Setup". The procedure is illustrated using a blank as an example.

Define at least one blank to use the 3D simulation. All components are necessary for realistic collision detection and simulation.

- Blank
- Protection area (optional)
- Setup (optional)
- Tool component (optional)

How to create a component in the library

1. Click the "Library" tab.
2. Click on the button "New blank". A window with an empty component is displayed.
3. Click on button "New blank from" to import a geometry.
   A dialog opens.
   More information: Optionally, you can also directly create blanks from basic geometrical shapes (Page 82) in the library.
4. Select the "*.stl / *.stp" file of the 3D model.

5. The next steps differ depending on the file format of the 3D model.
   "*.stl" file
   The 3D model is immediately imported.
   "*.stp" file
   – For multi-section 3D models, select the components that are to be imported. You can switch between the various 3D model views using the cube displayed in the foreground.
   – Click on button "Add Selected Items as Blank".

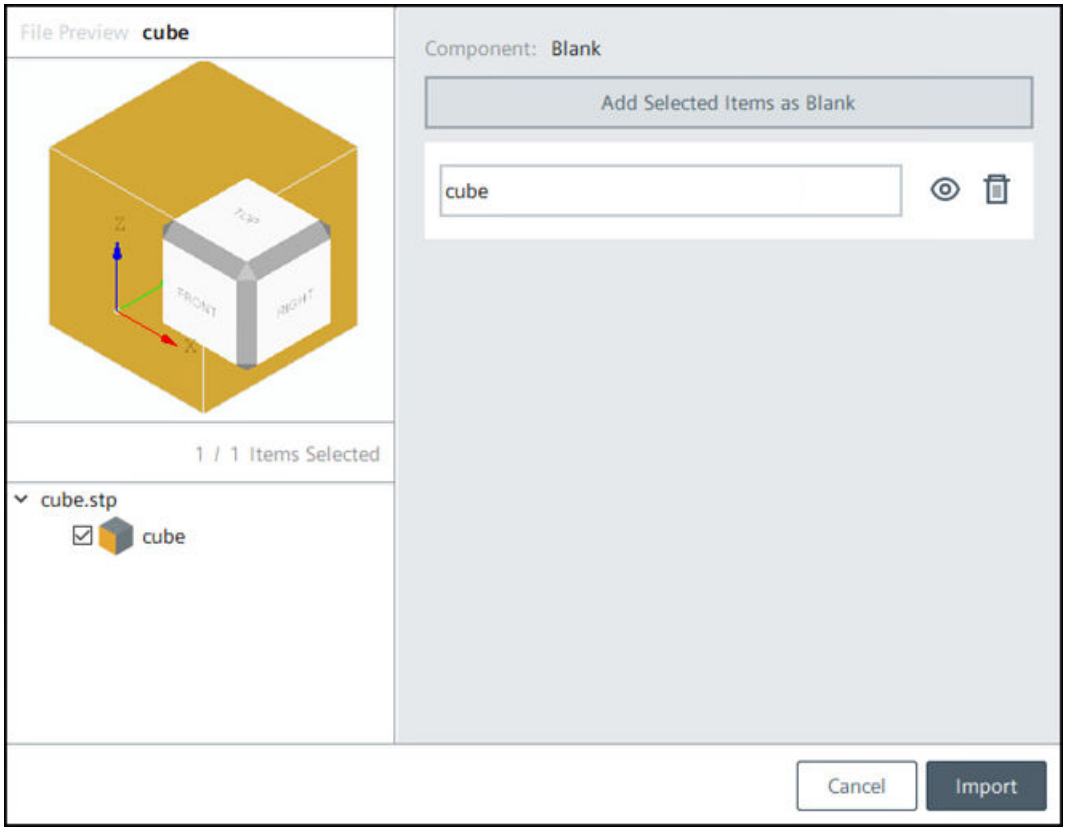


Figure 6-14 Example, blank with a component imported as "*.stp" file

   – The selection is accepted. Click on to display the selection in the 3D image for checking purposes.
   – Confirm with "Import".

   The imported 3D model is displayed.

6. Optionally, you can define the reference systems using button .
   – Click "+" to create a new reference point.
   – Enter new coordinate values for the reference system.

   In the example, a new reference point (blue) was defined at the left-hand lower corner. A Mounting Point (red) was defined centrally on the lower side.

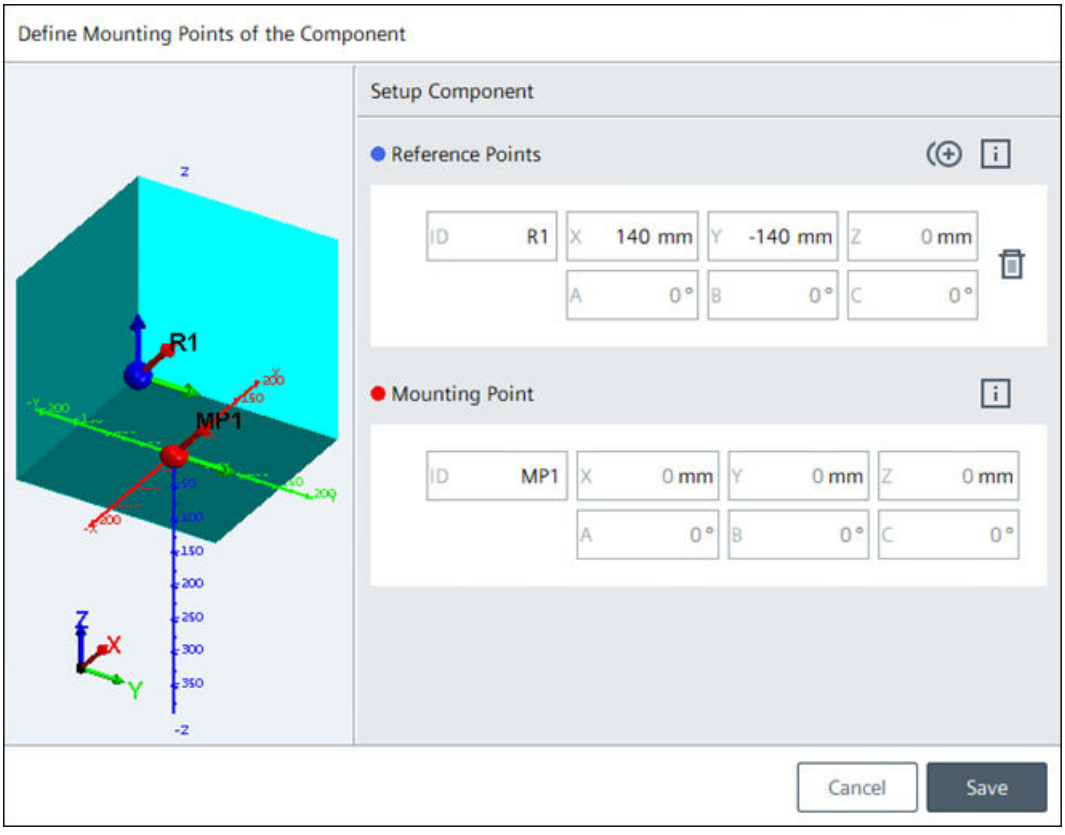


Figure 6-15 Example of blank: Reference point left-hand corner, Mounting Point, centered at the lower side.

7. Click "Save". The dialog is closed and the defined geometry for the component is displayed.

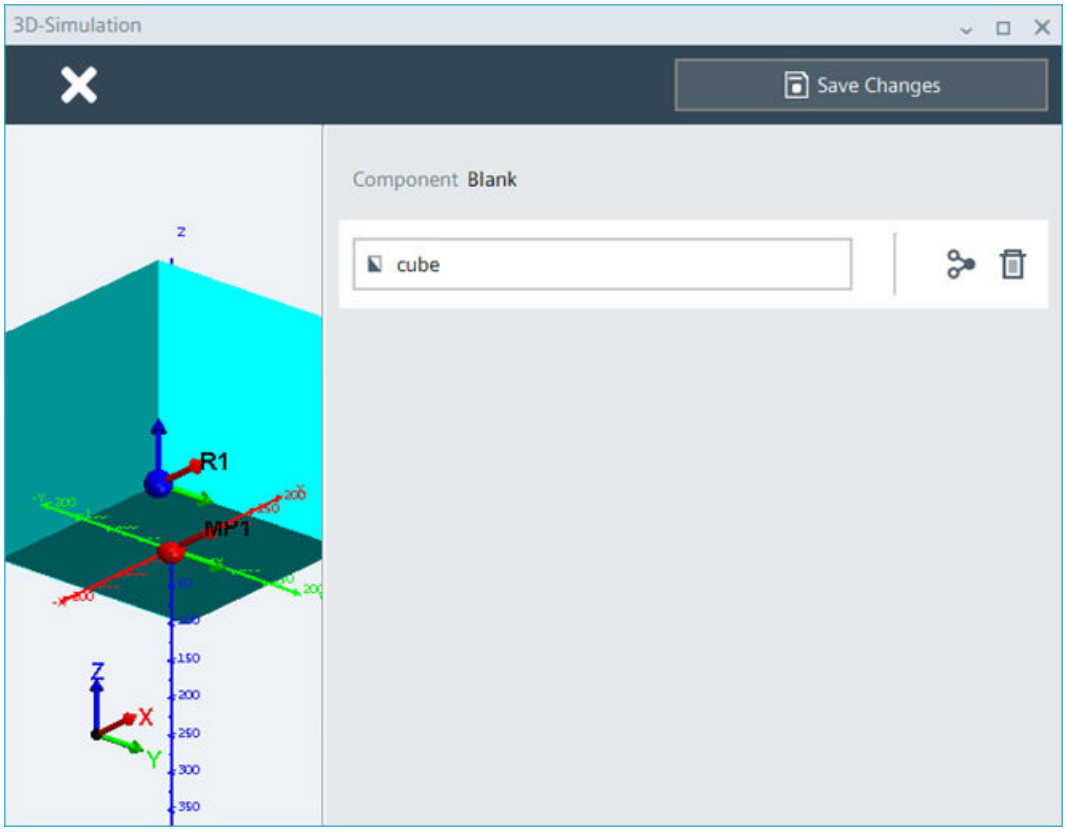


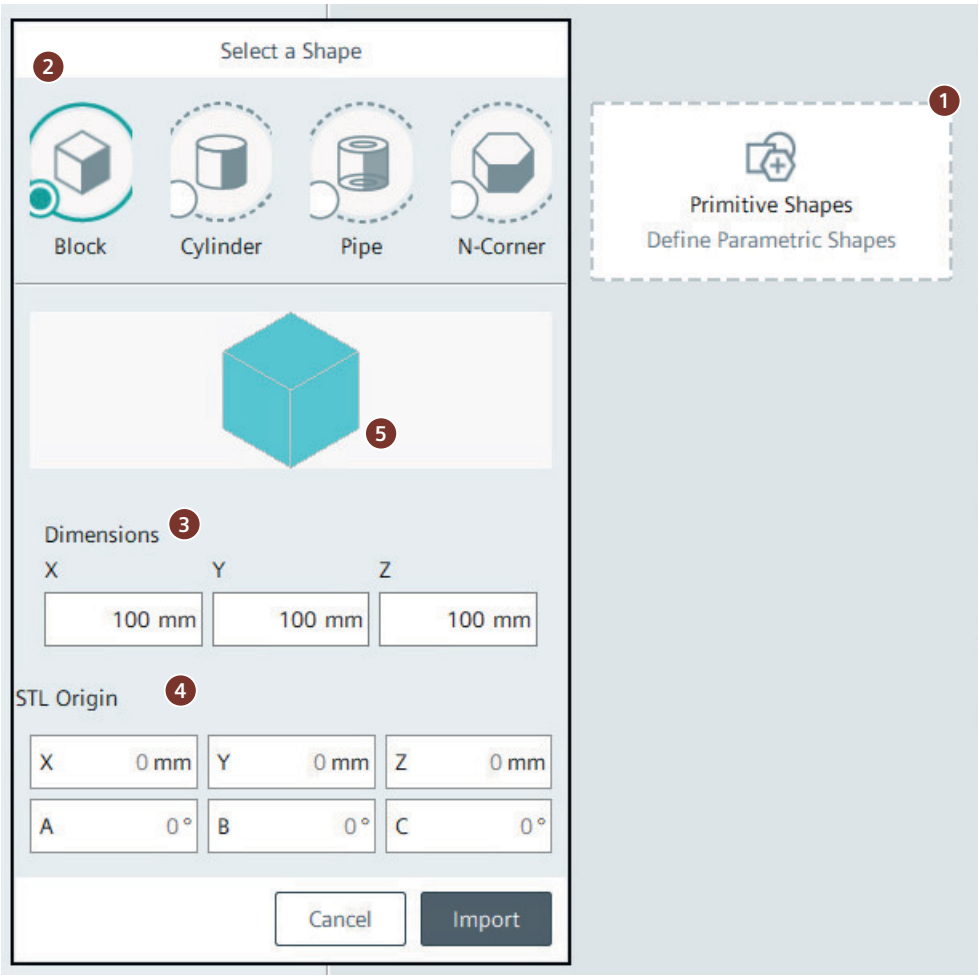
Figure 6-16 Example of blank: Components created for the library

8. Save the changes or the component. The window is closed and the new component is displayed in the library.
9. Create additional components in the library.

6.5.9 Blank from basic geometrical shapes

You create the blank in the library from specified geometrical shapes. You parameterize the standard shapes when creating the blank. "*.stl" or "*.stp" files are not necessary.

Overview of specified shapes for blanks



① When creating a new blank, click on the button to create the blank from a parameterizable shape.

② Select the basic geometrical shape of the blank. The following basic shapes are available:

- Block (dimensions in X, Y, Z)
- Cylinder (diameter and height)
- Pipe (diameter, wall thickness and height)
- N-corner (number of corners, edge length and height)

③ Depending on the selected basic shape, parameterize the dimensions of the geometric shape. The preview is shown at the top.

④ As default, the basic shape has a Mounting Point at the bottom center of the blank. You can shift the Mounting Point or rotate the reference system.

⑤ Preview of the parameterized shape

Figure 6-17 Parameterizable geometric shapes to create blanks

6.5.10 Multi-part STP component for individual components

When creating a component from the STEP file with subordinate components (e.g. complex clamping equipment), when importing the STEP file, you can select individual components and import as new component.

Requirement

A new component is created in the library.

An "*.stp" file is selected as 3D model.

This is how you work with STEP files with several components

In the example, a new protection area is created from a multi-part component.

1. Activate the checkbox in front of the component, which is to be imported as new protection area.
2. Click on button "Add Selected Items as Protection Area".

3. Enter a name for the new component.

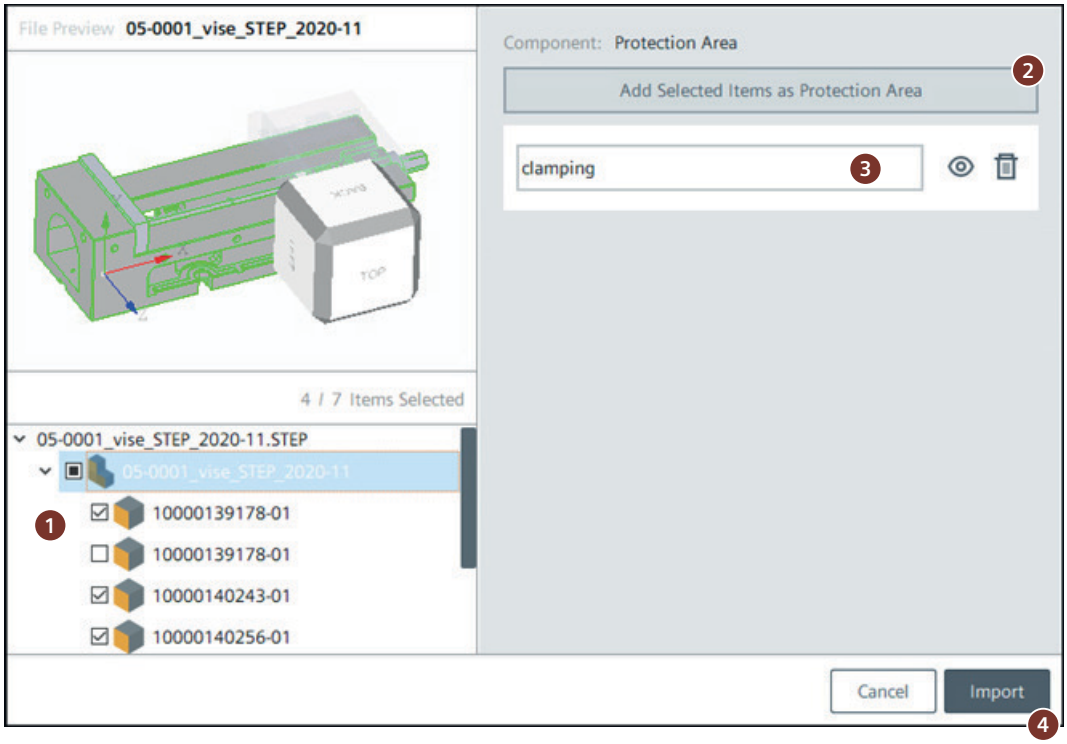


Figure 6-18 Individual components of a multi-part STEP model selected for import

4. Confirm with "Import". The imported selection is displayed as 3D model.

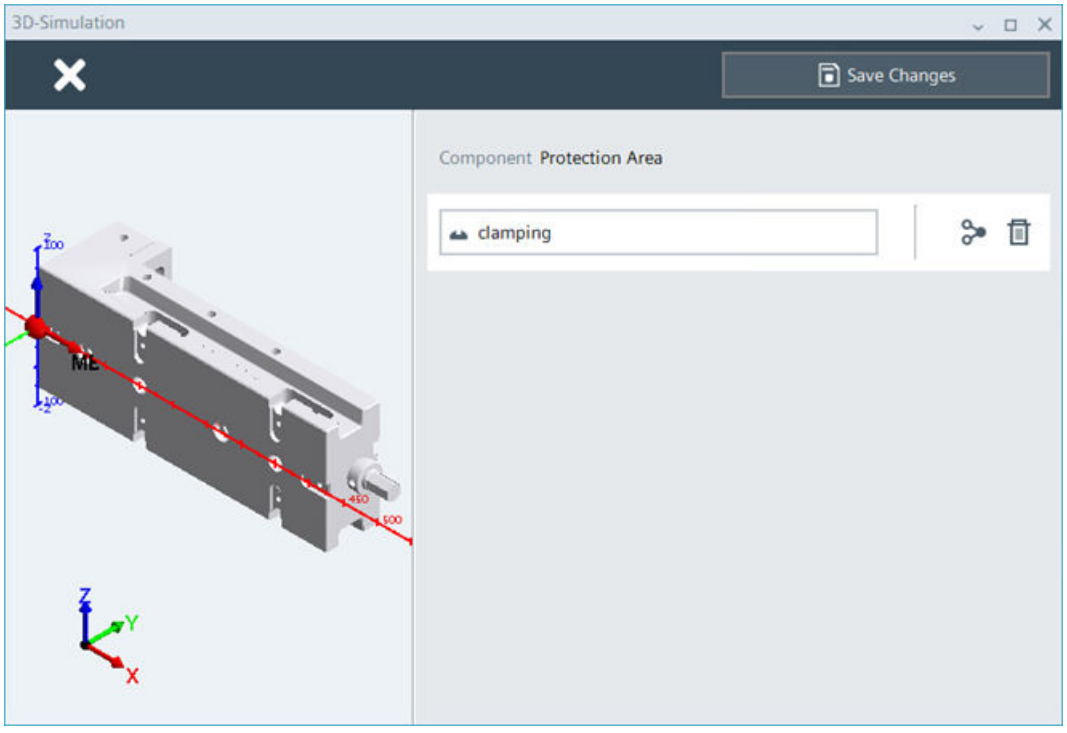


Figure 6-19 Protection area created from a STEP model

6.5.11 Assigning mount stations for setups

If, in a setup, a protection area is configured with more than two mount stations, for the blank, the mounting points must be assigned to the mount station. For example, this is the case if you are using a clamping tower with several clamping ranges.

Requirement

A setup with blank and protection area is open.

This is how you assign the mount station to the blank at the protection area

1. Open the dialog of the reference points at the blank. In addition to the mounting point, a button is displayed if the protection area has several mount stations.

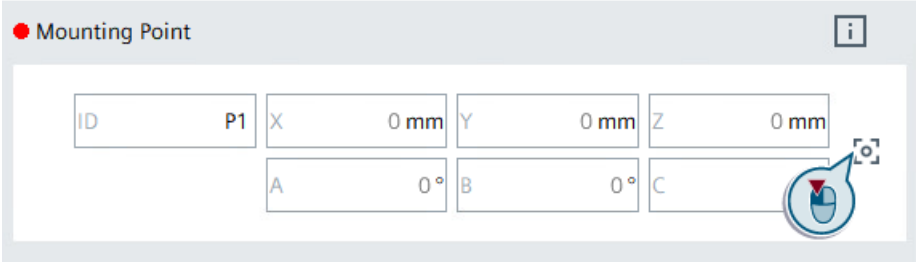


Figure 6-20 Reference points of the blank: Display of the Mounting Point

2. Click on the button. The dialog to assign the mount station at the protection area is displayed.
3. Click on the mount station that should be assigned to the mounting point of the blank.

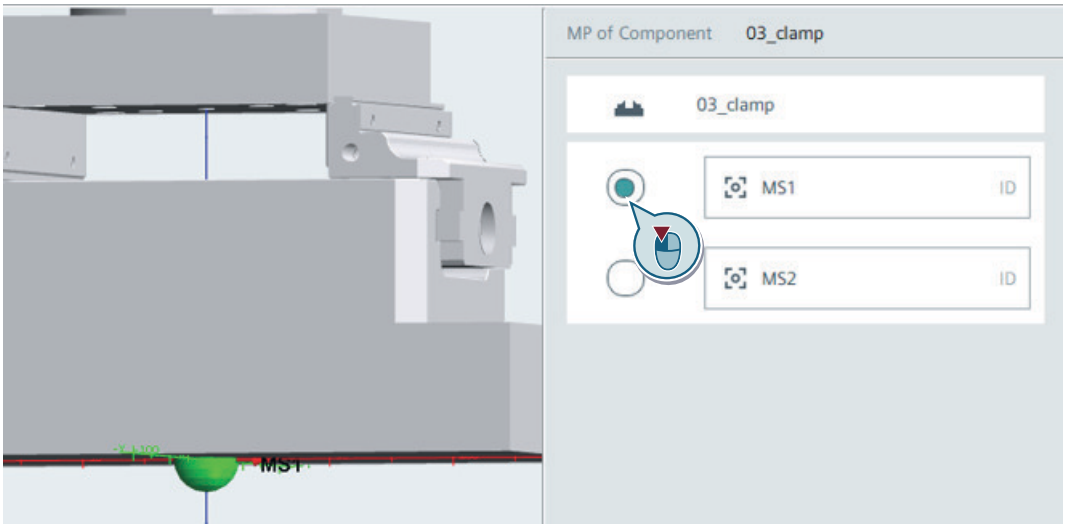


Figure 6-21 The mounting point of the blank is assigned mount station MS1

4. Save the dialog and switch to the setup. The points are linked with one another.
5. Repeat the operation if you wish to link additional points.

6.5.12 Multi-part STP component for setups

When creating a setup from a STEP file with several components (e.g. completed clamping towers with blanks), when importing the STEP file, you can select individual components and define as protection area or blank.

Requirement

A new setup has been created, and the STEP file has been imported with the protection area and the blanks.

This is how you create a setup from a STEP file

In the example, a setup is created from a clamping tower with three blanks.

1. Activate the checkbox in front of the component, which is to be imported as new protection area.
2. Click on button "Add Selected Items as Protection Area".
3. Enter a name for the new protection area.
4. Activate the checkbox in front of the component, which is to be imported as new blank.
5. Click on button "Add Selected Items as Blank".
6. Enter a name for the new blank.
7. Repeat the operation if wish to add additional blanks.

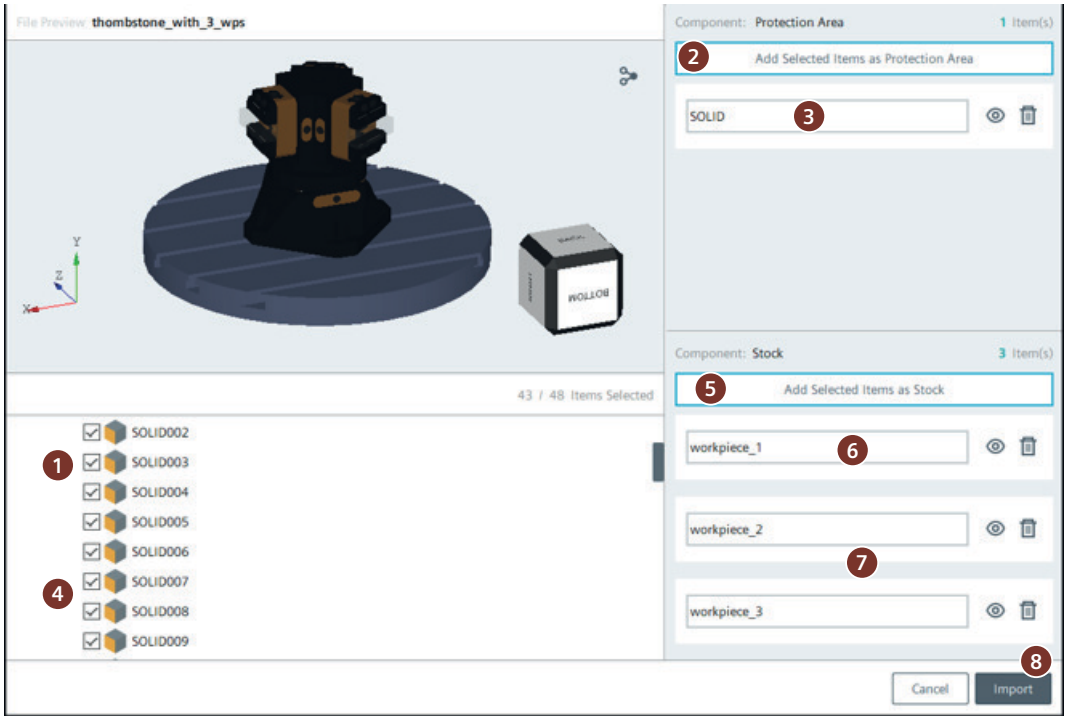


Figure 6-22 Importing a clamping tower with blanks as new setup

8. Confirm with "Import". The new setup is imported.

6.5.13 Set mounting point for configurations in the 3D model

When creating a setup from a STEP file, you can manually set the mounting point in the 3D model with the mouse pointer when importing.

Requirement

A new setup has been created, and the STEP file has been imported with the protection area and the blanks.

This is how you define the mounting point in the 3D model

In the example, the mounting point is set in the 3D model for a clamping tower.

1. Click on the button in the 3D model . The coordinate table is displayed.
2. Select a surface in the 3D model. The vertex of the surface is marked with a blue cross.
3. Double-click on the point. The new mounting point is displayed as red sphere and the coordinates are displayed.
4. You can still shift or rotate the mounting point using the coordinate table.
5. Select the components for blanks and protection area and assign the setup.

6. Confirm with "Import".

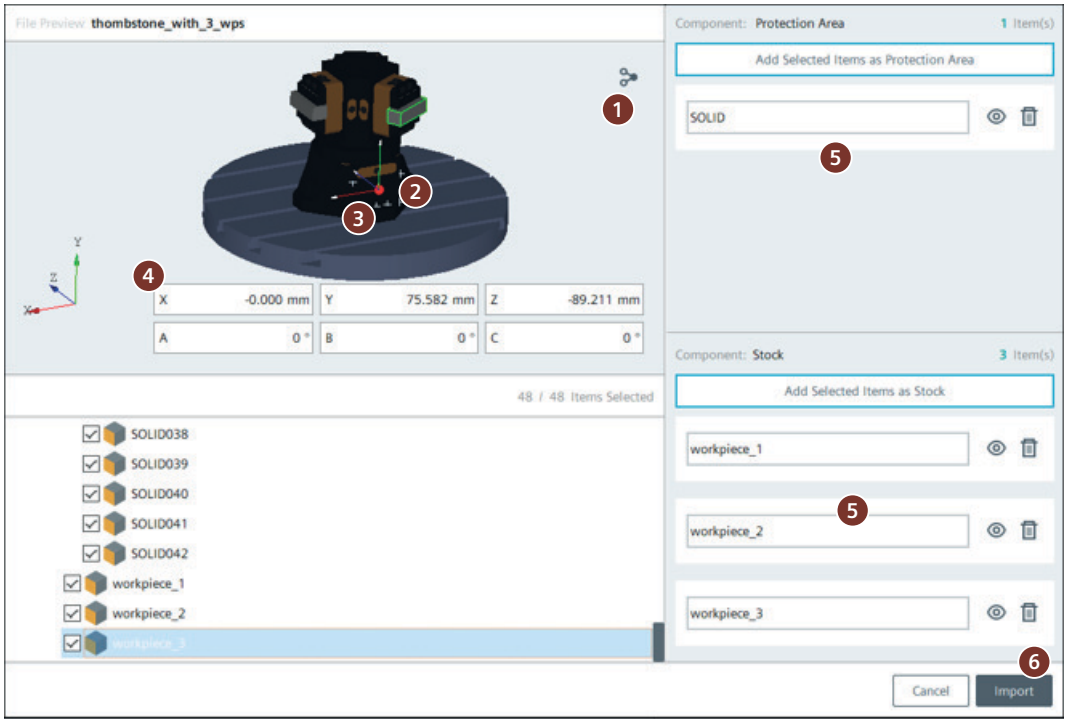


Figure 6-23 Setting the mounting point in the 3D model

7. The mounting point is used and displayed in the setup.

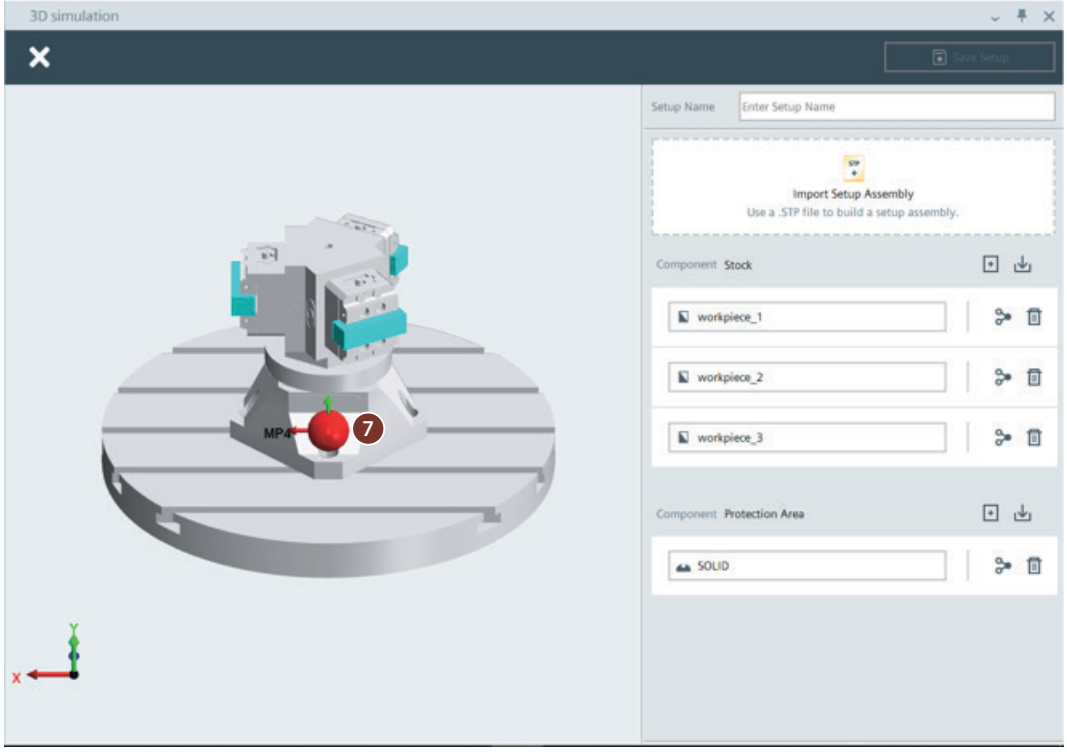


Figure 6-24 Mounting point in the setup

6.5.14 Exporting/importing library components

You can export/import library elements as a "*.zip" file, for example to exchange data with other projects. The files are exchanged via the storage folder or via a freely selectable directory.

Exporting components

1. Click on the "Select" button in the library.
2. Select the component you want to export in the library. Multiple selection is possible.
3. Click on the "Export" button.
4. Enter a name in the dialog that appears and confirm with "Export".
   The exported "*.zip" file is saved in the storage folder or in the selected directory.

Importing components

1. Copy the "*.zip" file to be imported into the storage folder or into a directory of your choice.
2. Click on "Import" in the library.
3. Select the "*.zip" file in the dialog that appears and confirm with "Import".
   The components are extracted and created in the library. If components with the same name exist, the import is canceled.

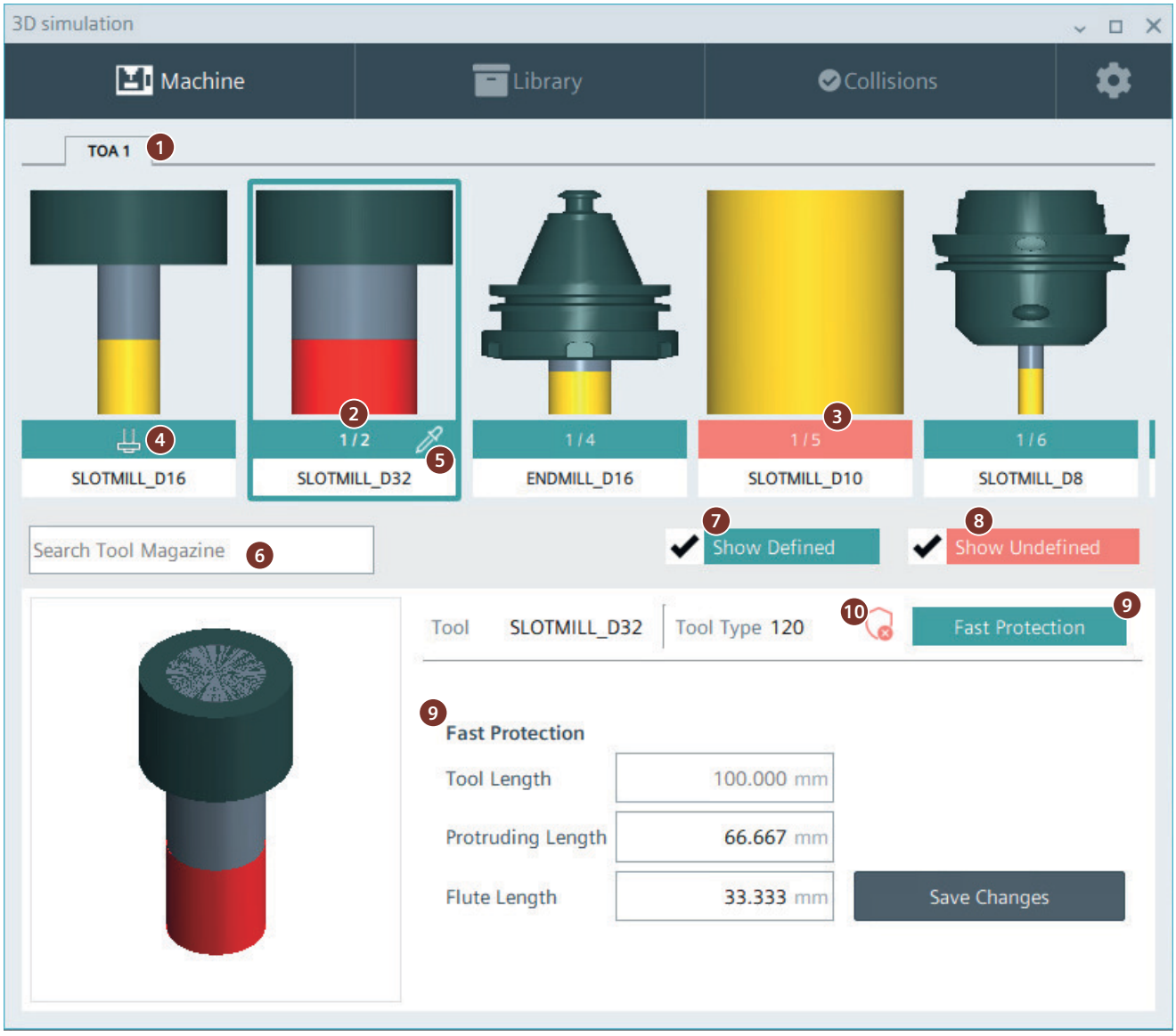
6.6 Tools

6.6.1 Tool manager

Managing tools for 3D simulation

The tool manager is displayed when you click on the tool icon  on the start page of the 3D simulation. All tools are displayed with the magazine/location number, the tool name, and with a preview image in a horizontal list. You can move the list to the right or left to display additional tools by clicking on the tool list with the left-hand mouse key pressed.

The active tool in the spindle is marked with a spindle symbol. The tool manager is synchronized with the tool list in SINUMERIK Operate and contains the tools defined there.



① TOA area for multi-channel machines

For multi-channel machines, each channel has a TOA area with its own tool list. These are shown as tab in the tool manager, e.g. TOA 1 and TOA 2. The corresponding tools are included in each TOA tab. Only one tab is displayed for 1-channel machines.

② Specification of the magazine number/location number of the tool. If no location has been assigned for the tool in the magazine, then only the magazine is displayed, but not a location number, e.g. 1/-.

The selected tool is displayed with a frame around it. If the protection (tool holder) has already been defined for the tool, the frame and the bar below are green.

③ Tools where protection has not been defined or not completely defined are shown with a red bar.

Tools where protection has not been defined are all tools without configured tool holder.

④ The active tool in the spindle is marked with a spindle symbol.

⑤ You can define a color for every tool. For stock removal simulation in the 3D view, the machined surface is displayed in the tool color. Click on the pipette icon to assign a color to a tool.

⑥ Searching for tools in the tool list
Here, enter the search term in the tool manager. At the same time as the entries are made, hits are displayed in the horizontal tool list.

⑦ Activate the checkbox to display the tools **with** defined protection.

⑧ Activate the checkbox to display the tools **without** defined protection.

⑨ A fast protection is defined in the example.

⑩ Deletes the current protection variant
Click on the red button to delete the current protection variant. You can define a new protection after deleting the protection variant.

Figure 6-25 Example: Tool manager for a 1-channel machine

Meaning of the protection

In addition to the geometry data from the SINUMERIK Operate tool list, you also define the protection variant. The protection describes the tool holder geometry (workholder) and other geometry data of the cutting edge or the tool. Thus a realistic simulation with collision detection is achieved. You set the additional data before starting the simulation. Changes made during the simulation only take effect when the simulation is restarted. Tools with a defined protection are marked using a green bar.

6.6.2 Tool manager and tool list in SINUMERIK Operate

The tools in the Manager correspond to the tool list in SINUMERIK Operate. If data of the SINUMERIK Operate tool list changes, it is also updated in the 3D simulation. If protection variants have already been defined for the tools, they are transferred to the changed tool or deleted according to certain rules.

Changed tools and protection variant

- If you change a tool parameter in SINUMERIK Operate while you are defining a protection for this tool in the 3D simulation, the window for defining tools is closed automatically.
- If you change the tool type in SINUMERIK Operate for a tool that has already been defined with protection in the 3D simulation, the following rules apply.
  The defined protection remains in place:
  – The new tool type is also a milling tool or even an unsupported tool type.

  The defined protection is deleted:
  – The old tool type and the new tool type are a turning tool.
  – The tool type is changed between milling, turning or drilling.
  – The tool length is changed to a value of the < unclamping length.

6.6.3 Overview of tool types

Only a selection of the SINUMERIK Operate tool types is supported in the 3D simulation. Supported tools are read in with all parameters from the SINUMERIK Operate tool list. Depending on the tool type, different protection variants are defined.

Note

Constraints relating to tools

- For tools that are not supported, the end mill (type 120) replacement tool is used for defining the protection variants and for simulation purposes.
- For tools with several cutting edges (flutes), D1 is used as criteria as to whether the tool is a milling or turning tool.
- Internally, drill bits have the same logic as milling tools

Overview of the supported tool types

For further information on the tool types and the associated parameters, refer to the online help of SINUMERIK Operate.

Technology	Tool type number	Tool type name/description
Milling tools		
	110	Ball nose end mill
	111	Ball end mill tapered
	120	End mill
	121	Tapered ball nose end mill
	155	Bevel cutter
	156	Bevel cutter with corner rounding
	---	Tool types that are not listed are still not supported. These are interpreted as end mills (type 120).
Drill bits		
	200	Twist drill
	220	Centering tool
	--	Tool types that are not listed are still not supported. These are interpreted as end mills (type 120).
Turning tools		
	500	Roughing tool
	510	Finishing tool
	520	Plunge cutter
	530	Cutting tool
	540	Threading tool
	550	Button tool
	560	Rotary drill
	---	Tool types that are not listed are still not supported. These are interpreted as end mills (type 120).
Special tools Probes		

Technology	Tool type number	Tool type name/description
	710	3D probe
	711	Edge finder
	712	Mono probe
	713	L probe
	714	Star probe
	725	Calibrating tool
	730	Stop
	---	Tool types that are not listed are still not supported. These are interpreted as end mills (type 120).
Options (from CNC-SW V6.15 SP2)		
	---	Angle head

6.6.4 Tools in the angle head

Angle heads are tool components comprising a stationary part (that does not rotate when the spindle rotates) and a rotating part. Normally, the axis of rotation of the angle head is not aligned to the axis of rotation of the spindle. As a result of the angle head, the rotation of the spindle is "deflected" into another direction e.g. 90° or 45°. This means that the tool cutting edge has an orientation direction other than the standard orientation.

Configuring the angle head

To configure the angle head, in addition to the tool length offset (length 1 to length 3 from $TC_DP3 to $TC_DP5), the adapter length offset (length 1 to length 3 from $TC_DP21 to $TC_DP23) is also taken into consideration. The adapter length defines the origin of the deflected axis of rotation.

The axis to be rotated is derived based on the values of the system variables for tool orientation $TC_DPV (if possible, based on the value of MD18114 $MN_MM_ENABLE_TOOL_ORIENTATION) or the values of orientation vector $TC_DPV3 to $TC_DPV5.

MD number	Identifier	Values	Description
MD18114	$MN_MM_ENABLE_TOOL_ORIENTATION	2 or 3	Necessary, if the tool alignment is to be defined using a vector via the tool data. If the vector definition is deactivated, although the HMI always displays the fields to enter data, these data are not saved and are not transferred to the 3D simulation.

The tool tip is located at the point of intersection obtained from the tool and adapter offset, and is aligned corresponding to the tool orientation.

Tool components can be configured so that these belong to the stationary part of the angle head. The placing for tool components remains unchanged, only the higher-level kinematic element (either rotates or does not rotate with the spindle) is adapted, based on this setup.

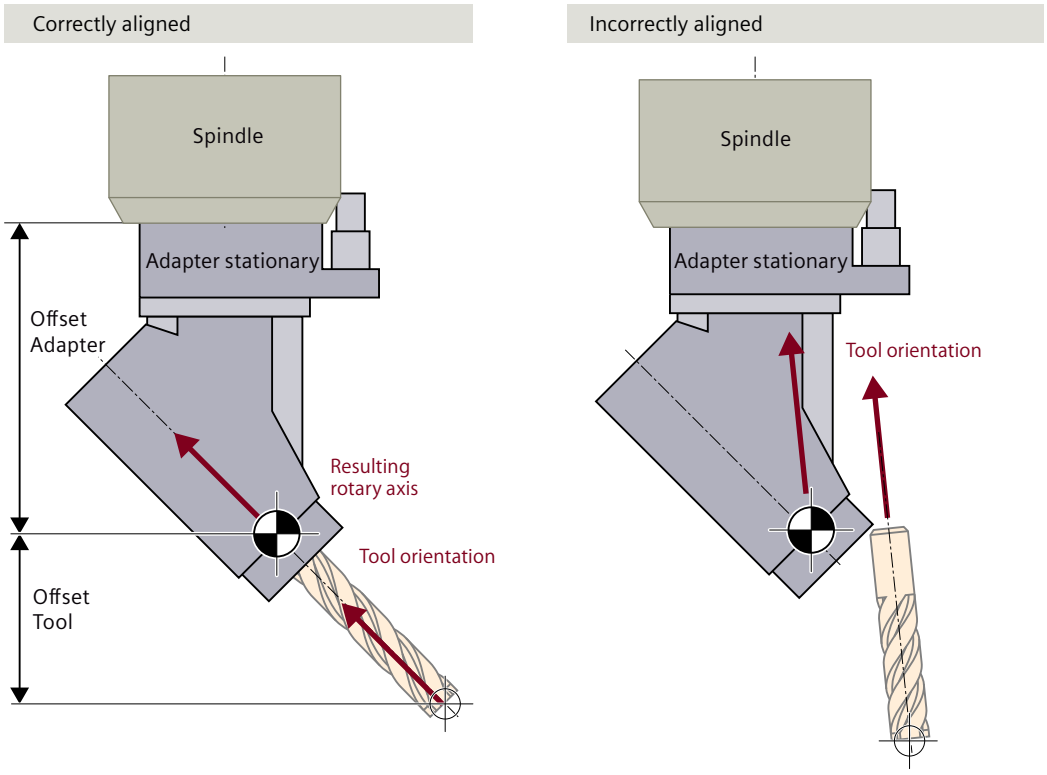


Figure 6-26 Angle head at 45° with specification of the relevant key variables

Note

Aligning the tool and axis of rotation

A tool cutting edge is not displayed in the 3D simulation if the tool is not aligned to the axis of rotation.

6.6.5 Protection variants for milling tools/drill bits

Protection variants

For the protection variants for milling tools/drill bits, choose between the following options:

- Fast protection
- Detailed protection

You can define the protection variant for the selected tool by clicking on the appropriate button with red border.

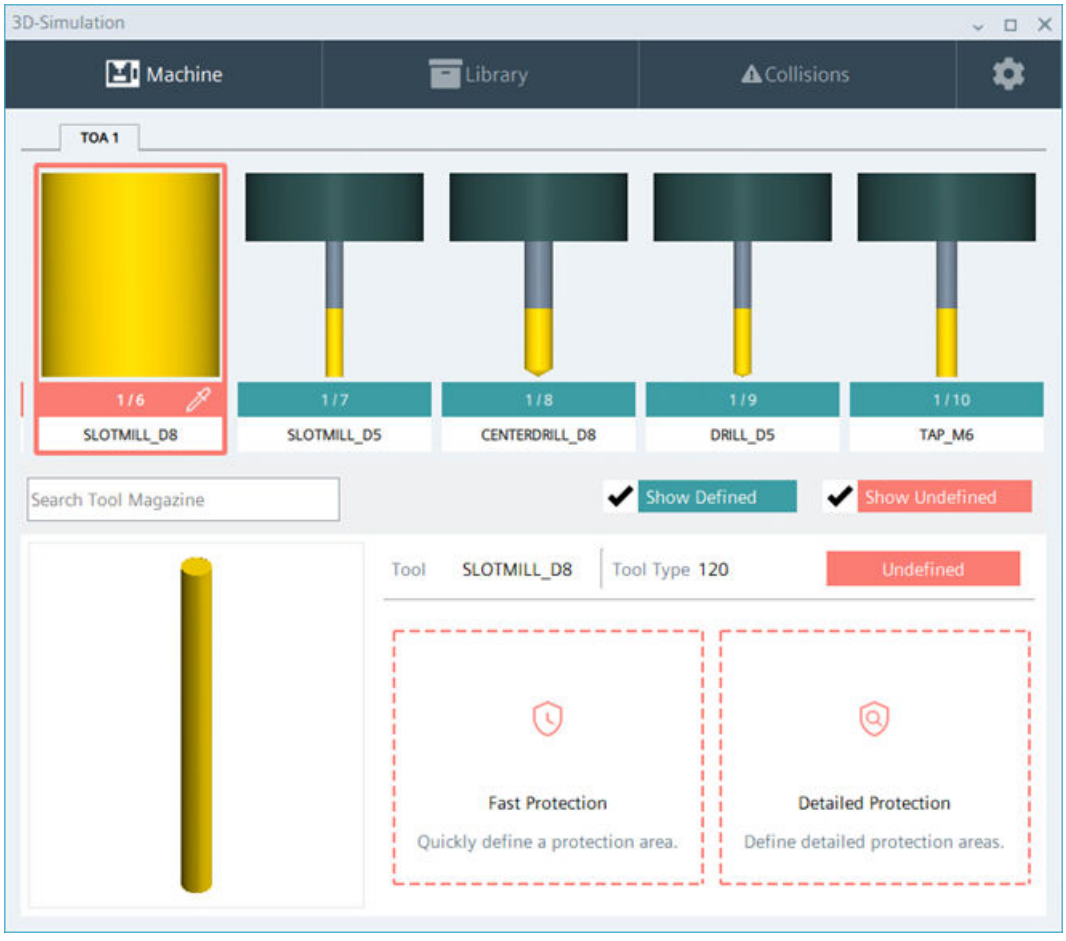


Figure 6-27 Selecting the protection variant for a tool

Fast protection

For fast protection, you define the following additional tool and holder parameters:

- Holder as standard cylindrical shape
  For the holder diameter, the value entered for "Holder diameter for fast protection" in the "Settings" tab is used.
- Tool length (taken from SINUMERIK Operate tool list)
- Flute length of the tool (colored area of the tool in the diagram)
- Protruding length (unclamping length/collar length) of the tool when used in a holder
  The protruding length must always be shorter than the tool length stored in SINUMERIK Operate.

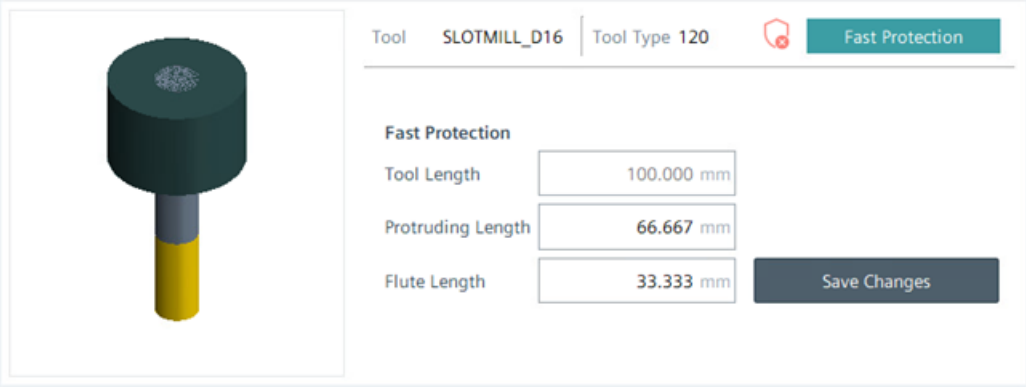


Figure 6-28 Fast protection for a tool defined using the example of an end mill

Fast Protection allows you to define the flute length and holder for a tool without importing an *.stl file of the tool holder. This option defines a basic holder without geometric details.

Detailed protection

For detailed protection, define the toolholder and tool parameters using geometry data. The following options are available to you for this

- Complete assembly comprising tool holder and tool geometry (including inserts) as "*.stp" file ①
- Tool holder geometry as "*.stl", "*.stp" file ②
- Tool holder geometry from the library ③
- Flute length of the tool (colored area in the diagram) ④
- Optional: Tool geometry including flute length as "*.stl" or "*.stp" file ⑤
  Tool data (diameter, length) are **not** taken from the SINUMERIK Operate tool list (NC), but from the imported geometry data. The switch must be brought into the "Off" position to import the geometry data.

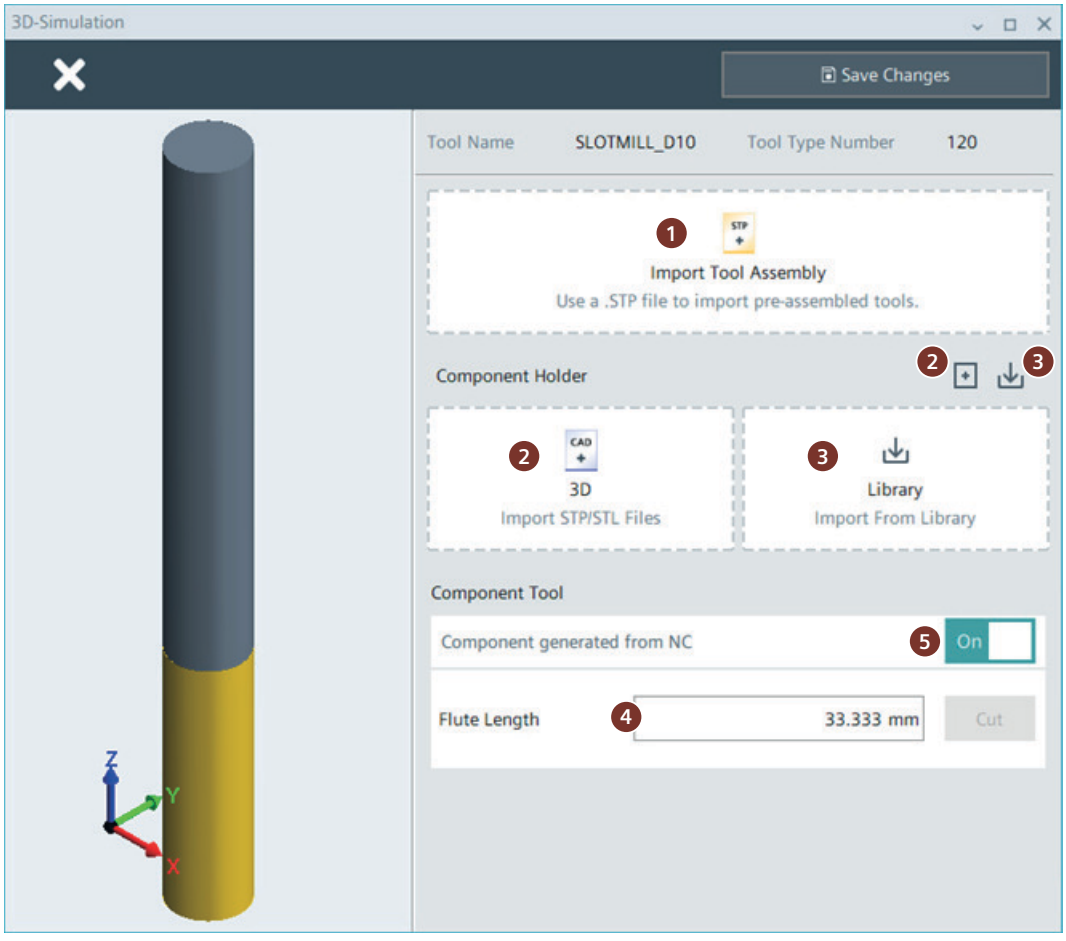


Figure 6-29 Overview of detailed protection for a tool defined using the example of an end mill

The unclamped length of the tool is defined for the holder geometry using the reference points.

The detailed protection function allows you to enter the flute length and define the tool holder geometry using an "*.stl" or "*.stp" file". The files that are stored in the storage folder are available for import.

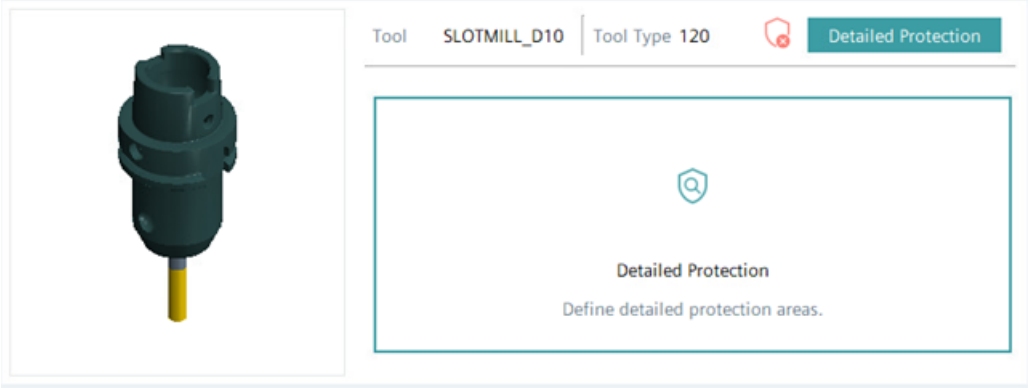


Figure 6-30 Detailed protection for a tool defined using the example of an end mill with a holder from the library

See also

Defining detailed protection for milling tools/drill bits (Page 102)

Defining fast protection for milling tools/drill bits (Page 101)

6.6.6 Protection variants for turning tools

Protection variant

For turning tools, define a detailed protection as protection variant. You can define the protection variant for the selected tool by clicking on the appropriate button with red border.

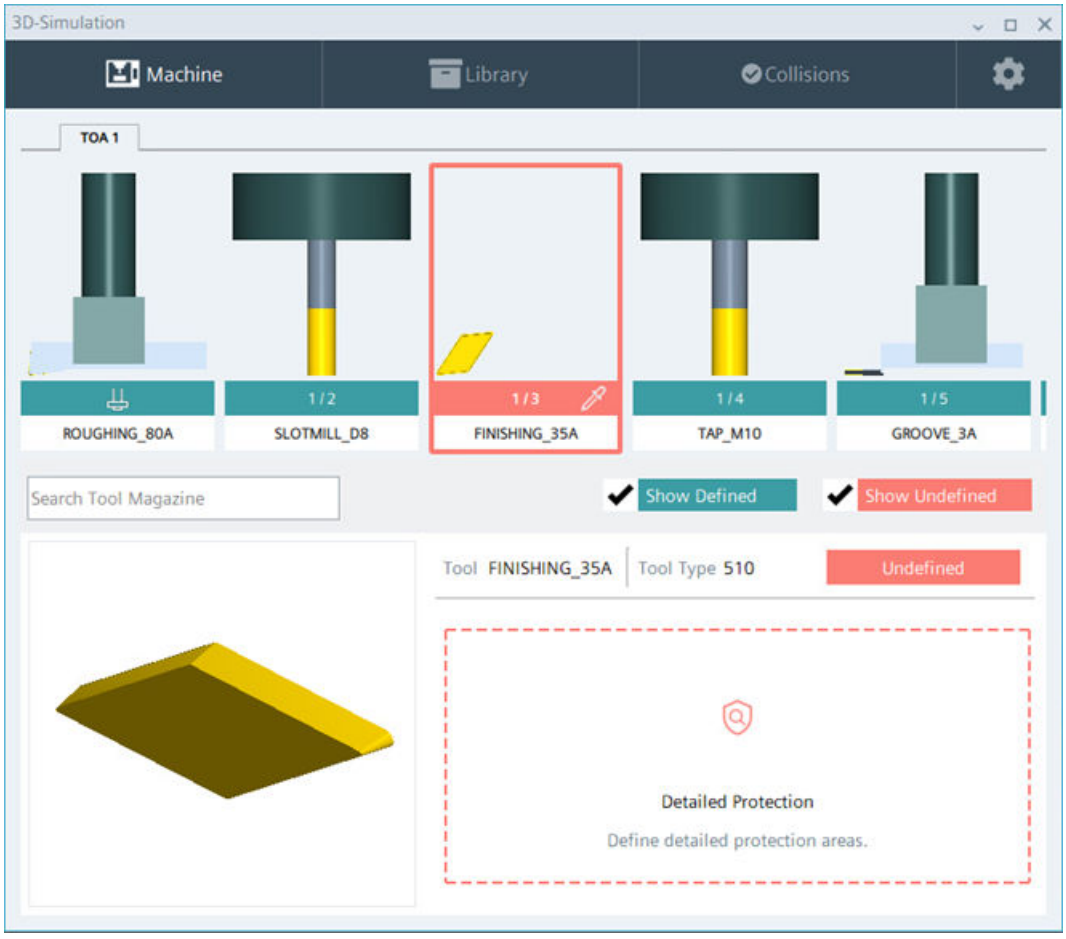


Figure 6-31 Selecting the protection variant for a turning tool

a shank, and optionally an adapter that describes the holder geometry.

Detailed protection

For detailed protection, define the tool (shank) and tool holder parameters (adapter) using geometry data. The following options are available to do this:

- Complete assembly comprising tool holder and tool geometry (including insert) as "*.stp" file ①
- Tool holder geometry as "*.stl", "*.stp" file ②
- Tool holder geometry is defined using parameters ③
- Tool holder geometry from the library ④
- Tool geometry (shank) is defined using parameters ⑤

The indexable insert data are taken as standard from the SINUMERIK Operate tool list. The insert thickness can be defined using parameters.

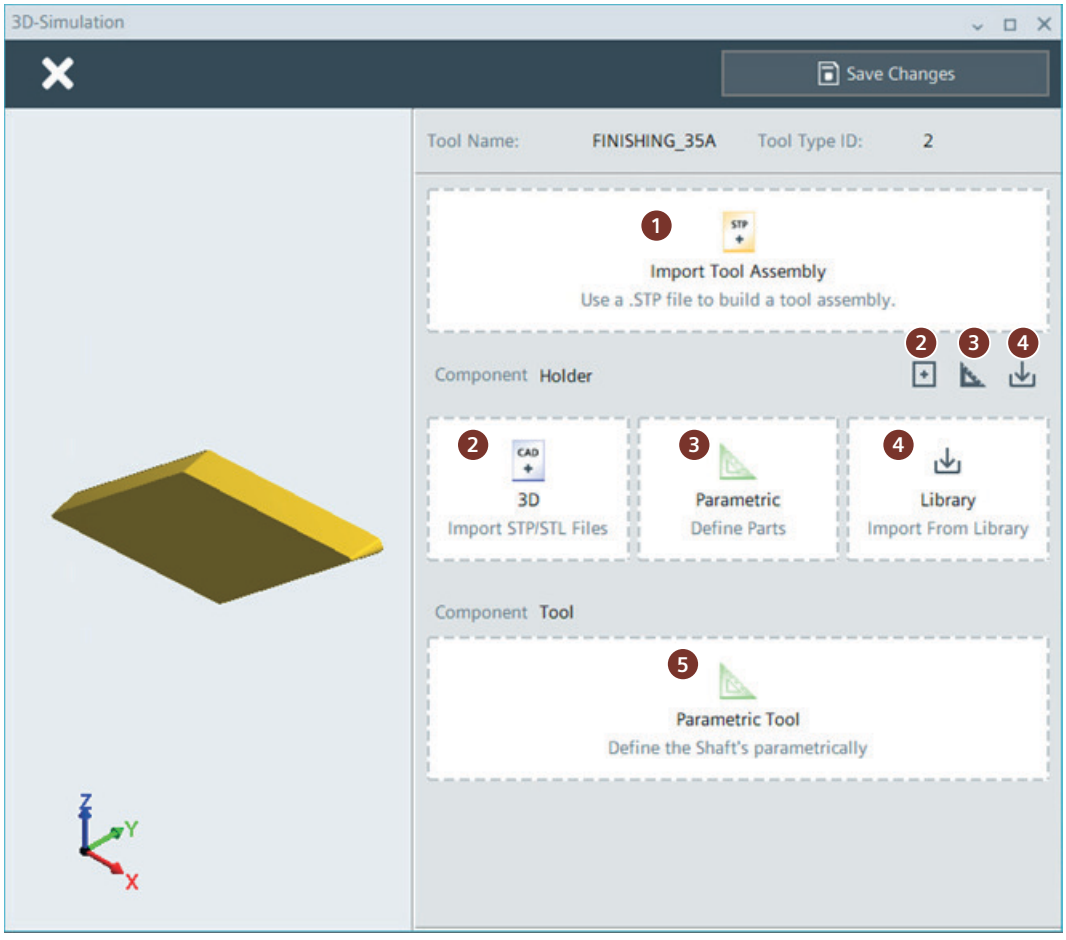


Figure 6-32 Detailed protection: Defining detailed protection for a turning tool

Example of detailed protection of holder and tool using parameters

The holder (adapter) and tool parameters (shank) are preassigned with default values. Change the parameters according to the holder and tool being used. The holder and the tool are shown in the 3D preview.

Press button "**V**" or "**Λ**" next to "Legend" to show or hide a drawing with explanations of the individual parameters.

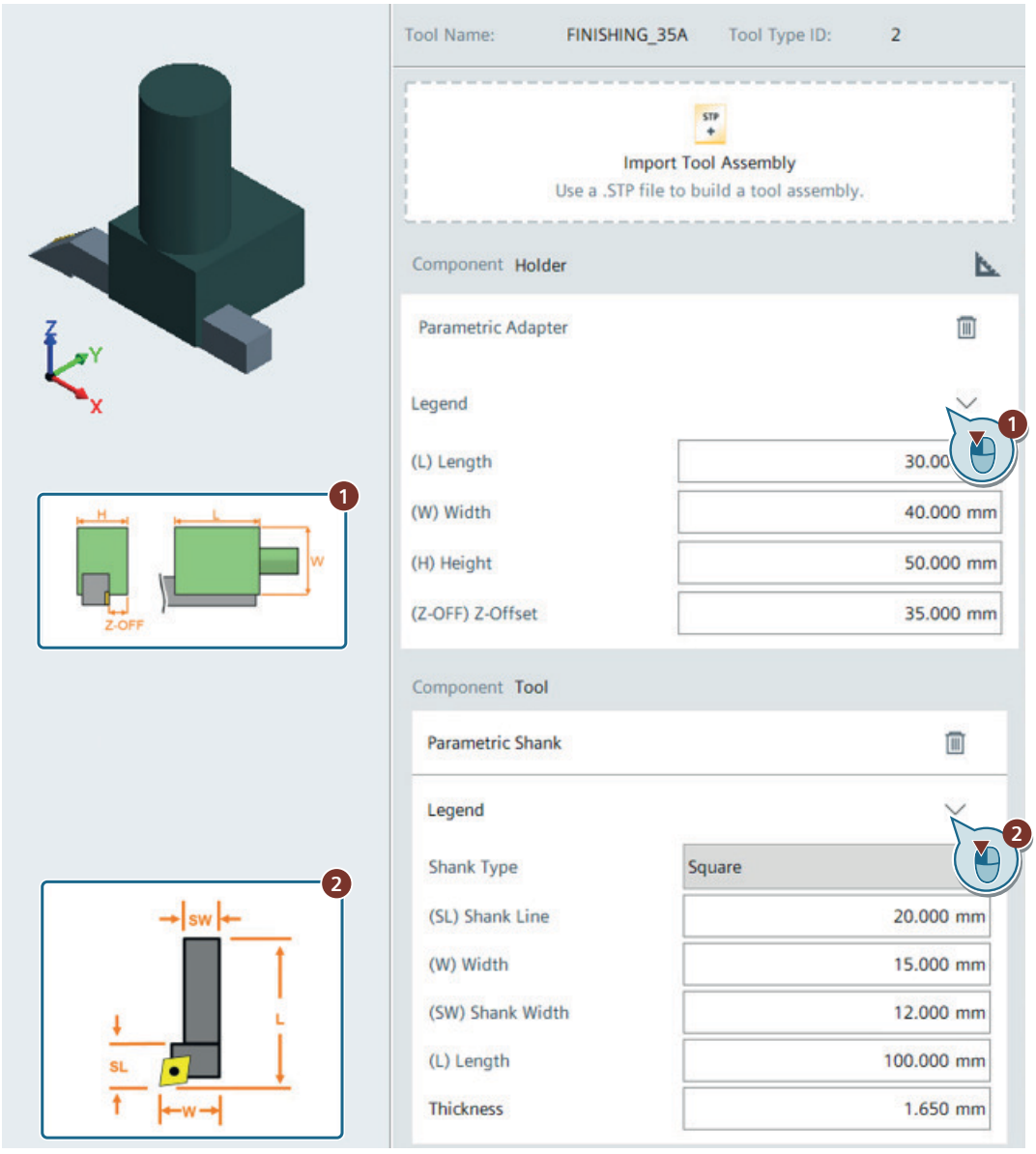


Figure 6-33 Defining a protection variant for a turning tool with display of the legend for the parameters

See also

Defining detailed protection for a turning tool (Page 104)

6.6.7 Defining fast protection for milling tools/drill bits

With "fast protection" you define additional tool parameters in the tool manager.

How to define "fast protection" for tools

1. Click on the tool for which you want to define a protection.
2. Click on the "Fast protection" button. The parameters that can be edited are displayed.
3. Enter a value in mm for the "Protruding length" and the "Flute length".
   The "Tool length" is taken from the tool list and it cannot be edited.

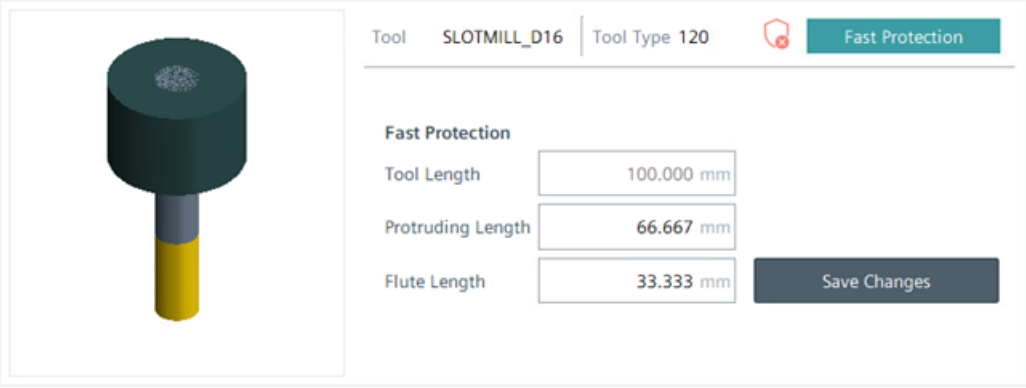


Figure 6-34 Defining fast protection for tool using the example of an end mill

4. Click on "Apply fast protection". The values are saved and a preview image with a yellow flute length is displayed.
   The tool has the status "defined".
5. Repeat the work steps if you want to define more tools.

Click on the "Machine" tab to close the tool manager.

Note

Configuring the holder diameter

For fast protection, the value entered under "Settings" is used as the holder diameter. The configured holder diameter is used for all milling tools/drill bits with "fast protection".

6.6.8 Defining detailed protection for milling tools/drill bits

With "detailed protection" you assign a tool component (tool holder) to the tool in the tool manager. Optionally, you can still import a tool geometry. To define "detailed protection" you can use various import variants of the 3D geometry, and combine as required; e.g. the geometry of the tool holder from the library and the geometry of the tool by importing a tool assembly ("*.stp).

This is how you define a detailed protection for milling tools/drill bits

A holder for an end mill is defined in the example. Optionally, you can specify the tool geometry data using an "*.stl/*.stp" file.

1. Click on the tool for which you want to define a protection function.
2. Click on "Detailed protection". The parameters that can be edited are displayed.

3. You can import the geometries in the window that opens using button "Import tool assembly" or in area "Component Holder".
   - Button
     Import a holder geometry and optionally a tool geometry from a tool assembly ("*.stp")
     In parallel to the import, a holder geometry is created in the library, which can then also be used as library component.
   - Button or .
     Imports an "*.stl/*.stp" file of the holder geometry
     In parallel to the import, a holder geometry is created in the library, which can then also be used as library component.
   - Button
     Use of already defined holder geometries from the library

   The following conditions apply to the holder geometries:
   - A tool holder can be composed of up to 5 part geometries in the specified sequence.
   - The positions of the Mounting Points and Mount Stations of the individual holder geometry define the overall geometry of the tool holder.

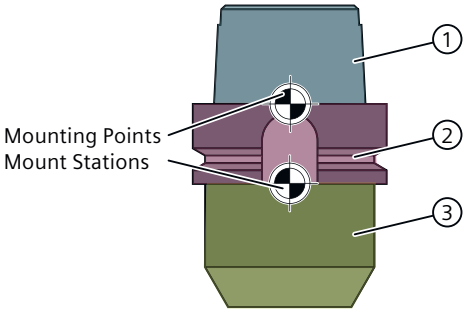


Figure 6-35 Example of tool holder, comprising three part geometries

   - You set the position of the holder with respect to the tool using the Mounting Point, e.g. you can shift the Mounting Point in the Z direction to set the unclamped length of the tool.
4. Click on if you wish to configure the reference system to align the components.
5. For each holder, define whether it should be "Rotating" or "Rigid". As default, all holders should be configured as "Rotating".

Note

Angle head

If an "Angle head" is used as holder, then the holder must be configured as "Rigid" (Tools in the angle head (Page 94)).

Figure 6-36 Angle head SINUMERIK Operate tool list

6. Keep switch "Component generated from NC" in the "On" position

Note

Not generating components from the NC

As standard, tool data are taken from the SINUMERIK Operate tool list. Optionally, you can also import the tool geometry file. Click on switch "Component generated from NC". The option is marked with "Off". In the field displayed, you can import the tool geometry data.

For collision avoidance, in this case, you can use the geometry data of the imported tool and not the data from the SINUMERIK Operate tool list, e.g. tool diameter.

7. Enter a value in mm for the "Flute length".
8. Click on "Save changes". The values are saved and the tool manager is displayed. The tool is defined.

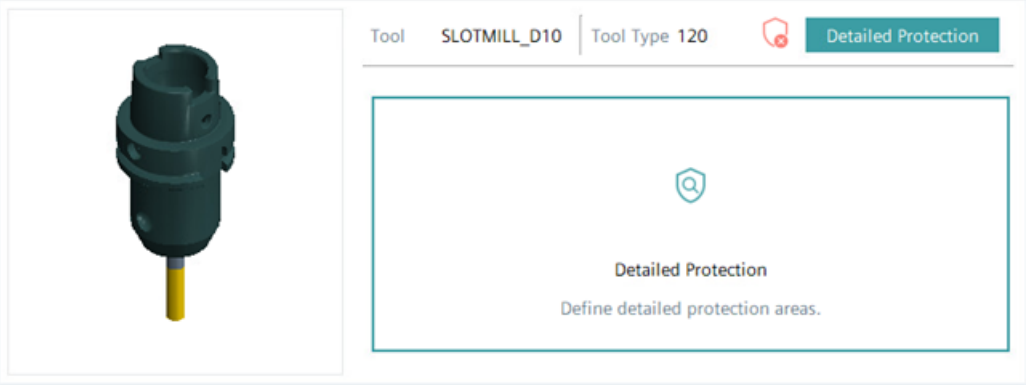


Figure 6-37 Detailed protection: Example of an end mill with imported tool holder geometry

9. Repeat the work steps if you want to define more tools.

Click on the "Machine" tab to close the tool manager.

See also

Using the holder geometry from the library (Page 109)

Importing holder geometry as "*.stl, *.stp" (Page 109)

6.6.9 Defining detailed protection for a turning tool

For "detailed protection" of turning tools, you define a tool (shank), a holder geometry (adapter) and optionally, a geometry of the cutting plate. To define "detailed protection" you can use various import variants of the 3D geometry, and combine as required; e.g. the geometry of the tool holder from the library and the geometry of the tool using parameters.

How to define a protection for turning tools

1. Click on the turning tool for which you want to define a protection function.
2. Click on "Detailed protection". The parameters that can be edited are displayed.

3. You can import the geometries in the window that opens using button "Import tool assembly" or in area "Component Holder".
   - Button
     Import a holder geometry and optionally a tool geometry from a tool assembly ("*.stp")
     In parallel to the import, a holder geometry is created in the library, which can then also be used as library component.
   - Button or .
     Imports an "*.stl/*.stp" file of the holder geometry
     In parallel to the import, a holder geometry is created in the library, which can then also be used as library component.
   - Button or .
     Defines the tool and holder geometry using parameters
   - Button
     Use of already defined holder geometries from the library

   The following conditions apply to the holder geometries:
   - A tool holder can be composed of up to 5 part geometries in the specified sequence.
   - The positions of the Mounting Points and Mount Stations of the individual holder geometry define the overall geometry of the tool holder.
   - You set the position of the holder with respect to the tool using the Mounting Point, e.g. you can shift the Mounting Point to set the tool position.
4. Click on if you wish to configure the reference system to align the components.
5. Enter a value in mm for the "Thickness" of the plate.
6. Click on "Save changes". The values are saved and the tool manager is displayed. The tool is defined.

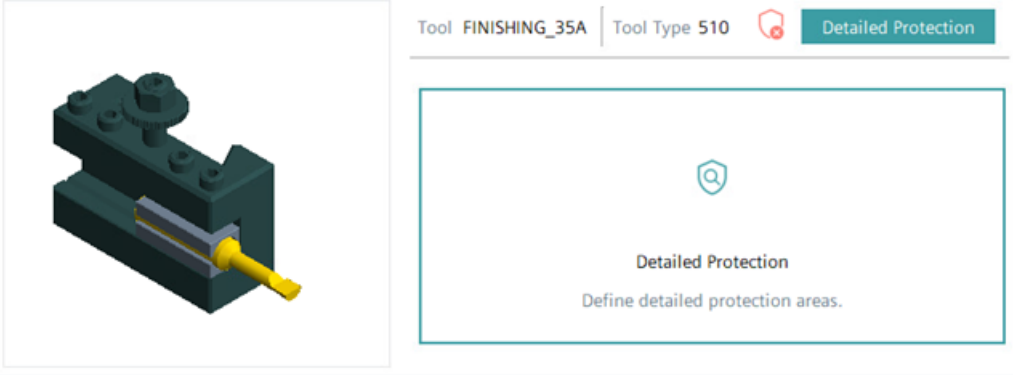


Figure 6-38 Detailed protection: Example of an end mill with imported geometries of the tool holder and the tool

Note

If you define invalid values for the holder or adapter parameters, no 3D preview is displayed for this tool. The tool then has no defined protection.

7. Repeat the work steps if you want to define more tools.

Click on the "Machine" tab to close the tool manager.

See also

Defining holder/tool geometries using parameters (only turning tool) (Page 112)

Using the holder geometry from the library (Page 109)

Importing holder geometry as "*.stl, *.stp" (Page 109)

6.6.10 Defining holder and tool geometries via the assembly

To define "detailed protection", you can import a complete assembly (holder and tool) as "*.stp" file. You can import either all or only individual elements of the assembly, e.g. only the tool or only the tool holder.

Requirement

The dialog to configure "detailed protection" for the tool opens.

This is how you define a detailed protection with an assembly

In the example, an assembly comprising a tool holder and an end mill is imported. The procedure is identical for turning tools.

1. Click on "Import tool assembly".
2. In the dialog that is displayed, select the "*.stp" file and click "Open".
   The 3D model ①, the individual elements ②, the "*.stp" file and the selection of the holder ④ or the tool ⑤ are displayed in the window.

   Note

   Mounting Point orientation (Mounting Point) ③

   You change the orientation of the tool and/or the assembly in the coordinate system by specifying the rotation in axes A, B, C of the Mounting Point. Before the import, possibly adapt the orientation to the current requirements. The orientation must be defined for every element of the assembly (holder, shank, indexable insert).

3. To import, activate the checkboxes for the individual elements that you wish to import as part geometry. You can activate 1-n elements for a part geometry.

4. Then click on "Add as ..", to import the selected individual elements as holder, tool or insert geometry into the component. For tools with indexable inserts you can separately import the tool shank and the inserts.
   - A holder can comprise a maximum of 5 holder geometries ④.
   - A tool can comprise a part geometry for the shank and a part geometry for the inserts ⑤.

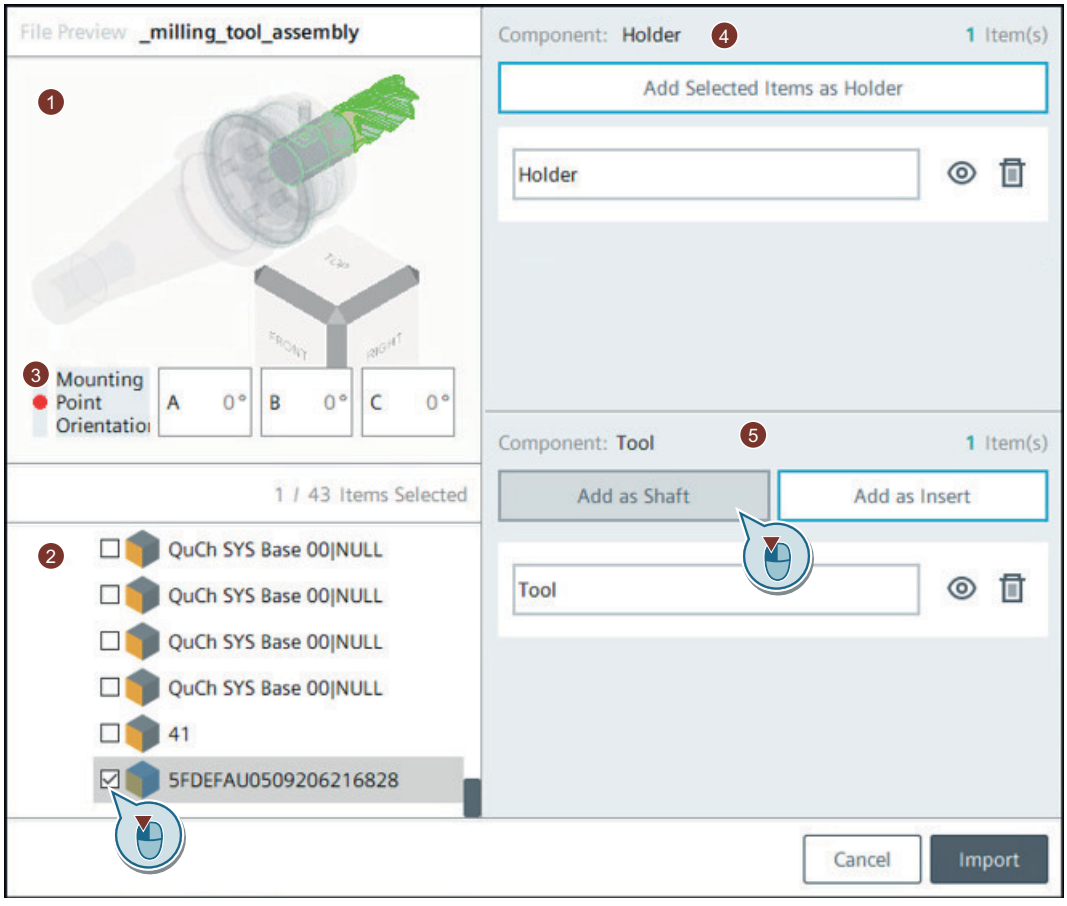


Figure 6-39 Importing an assembly comprising tool holder and tool as protection

5. Optionally, enter a name for the imported geometry.

Note

Name that is unique throughout the project

The name must be unique throughout the project. If the name is already being used for an existing tool component, then an error message is displayed when saving the tool component. In this case, enter a unique name for the component.

6. As a default, the Mounting Point③ is set at the zero point of the reference system of the STEP file. You change the orientation of the reference system by rotating axes (A, B, C).

7. Click on "Import" if you have created the required components.

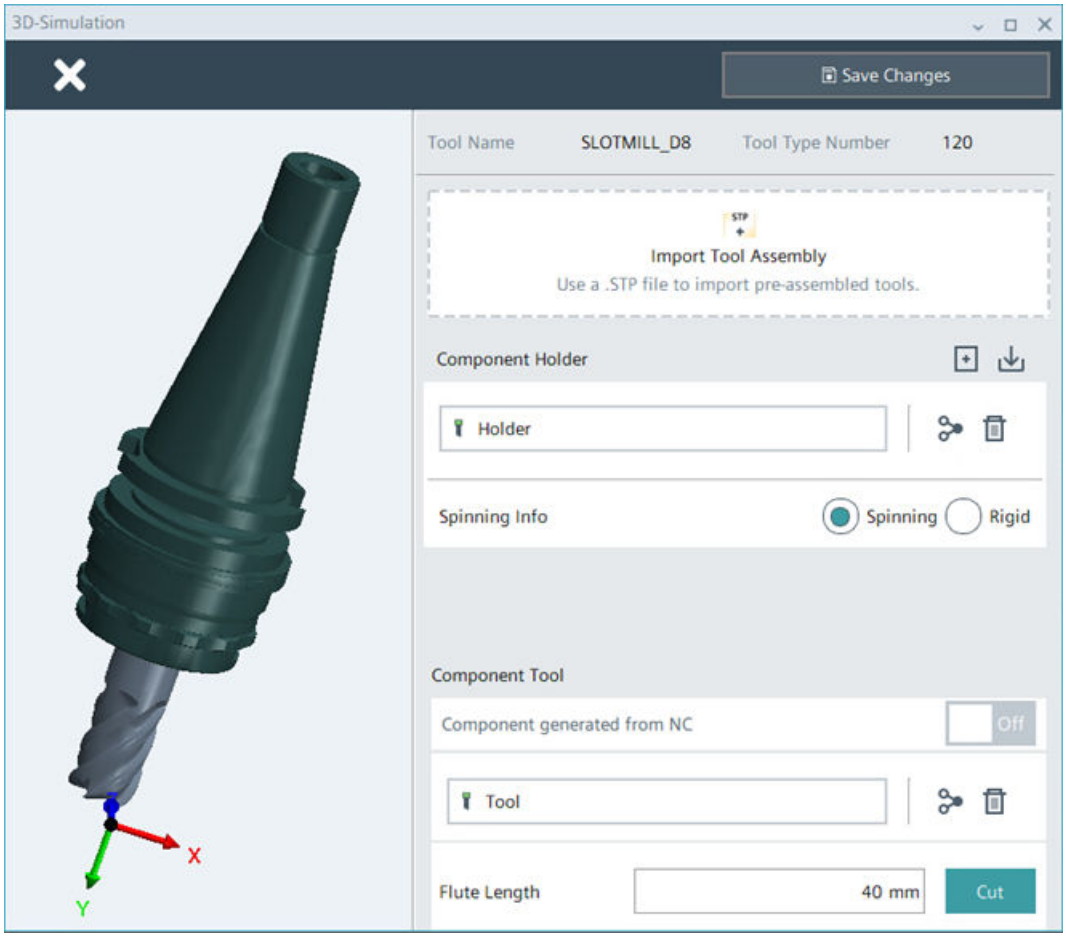


Figure 6-40 Detailed protection for tool holder and tool created from "*.stp" assembly

8. Click on if you wish to configure the reference system to align the components.
9. If you have only defined one tool component (no indexable insert in the geometry), then you can enter a flute length and set this using button Cut. The 3D geometry of the tool is horizontally cut-off at this length.

Note

Complex geometries

Depending on the complexity of the cutting edge geometry and the length of the cutting edge, it can take some time to calculate the tool geometry. The new flute length calculation must first be completed before you perform any additional actions.

10. Repeat the last steps if you want to assign additional part geometries to the holder.
11. Click on if you wish to delete existing holder geometries or the tool geometry.
12. Click on "Save changes" to complete the setup.

6.6.11 Importing holder geometry as "*.stl, *.stp"

To define tool protection, import the holder geometries, using an "*.stl" file as example.

Requirement

The dialog to configure "detailed protection" for the tool opens.

This is how you define protection of a tool using imported "*.stl/*.stp" files

In the following, the procedure is illustrated using a milling tool as an example. The procedure is identical for turning tools.

1. Click on or next to the tool component. The dialog to select the "*.stl/*.stp" file is displayed.
2. Select a file with the 3D geometry of the holder.
3. Confirm with "Open". The 3D program is displayed.
   The next step depends on the selected file type.
   - "*.stl" file
     The holder component is immediately imported.
   - "*.stp" file
     To import, activate the checkboxes for the individual elements that you wish to import as part geometry. Confirm with "Import".

   The imported holder component is displayed.
4. Optionally, you can also define the reference system to configure the alignment of the tool holder with respect to the tool.
5. For milling tool/drill bit holders, define whether it should be used "Rotating" or "Rigid". As default, all holders should be configured as "Rotating".
6. Repeat the last steps if you want to assign further holder geometries to the tool.
7. Click on the "recycle bin" button to delete existing holder geometries.
8. Confirm with "Save changes".

The tool protection has been configured.

See also

Defining detailed protection for milling tools/drill bits (Page 102)

6.6.12 Using the holder geometry from the library

To define the tool protection (detailed protection), use holder geometries from the library.

Requirement

The dialog to configure "detailed protection" for the tool opens.

How to define a protection of the tool with geometries from the library

In the following, the procedure is illustrated using a milling tool as an example. The procedure is identical for turning tools.

1. Click on the button next to the tool component. The library is opened.
2. Select a holder geometry and click on "Import". The tool component is assigned to the tool.

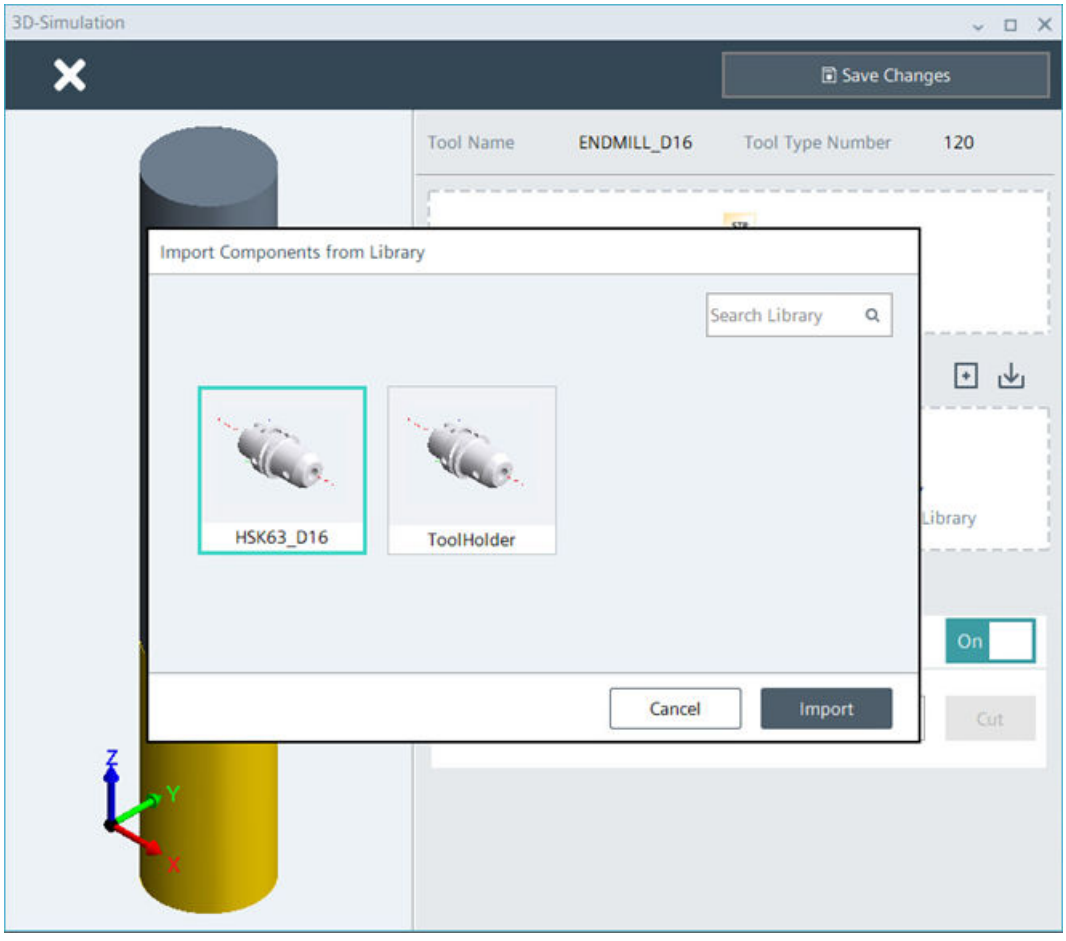


Figure 6-41 Importing a tool holder geometry from the library

3. Click on button next to the geometry if you want to check the values for "Mounting Point" and "Mount Station".

Note

The values for the reference system cannot be edited when imported from the library. If you want to change these values, you must change the tool component in the library. For example, to adapt the orientation of the tool holder to the tool, for "Mounting Point", enter a shift in fields X, Y, Z or an angle in A, B C.

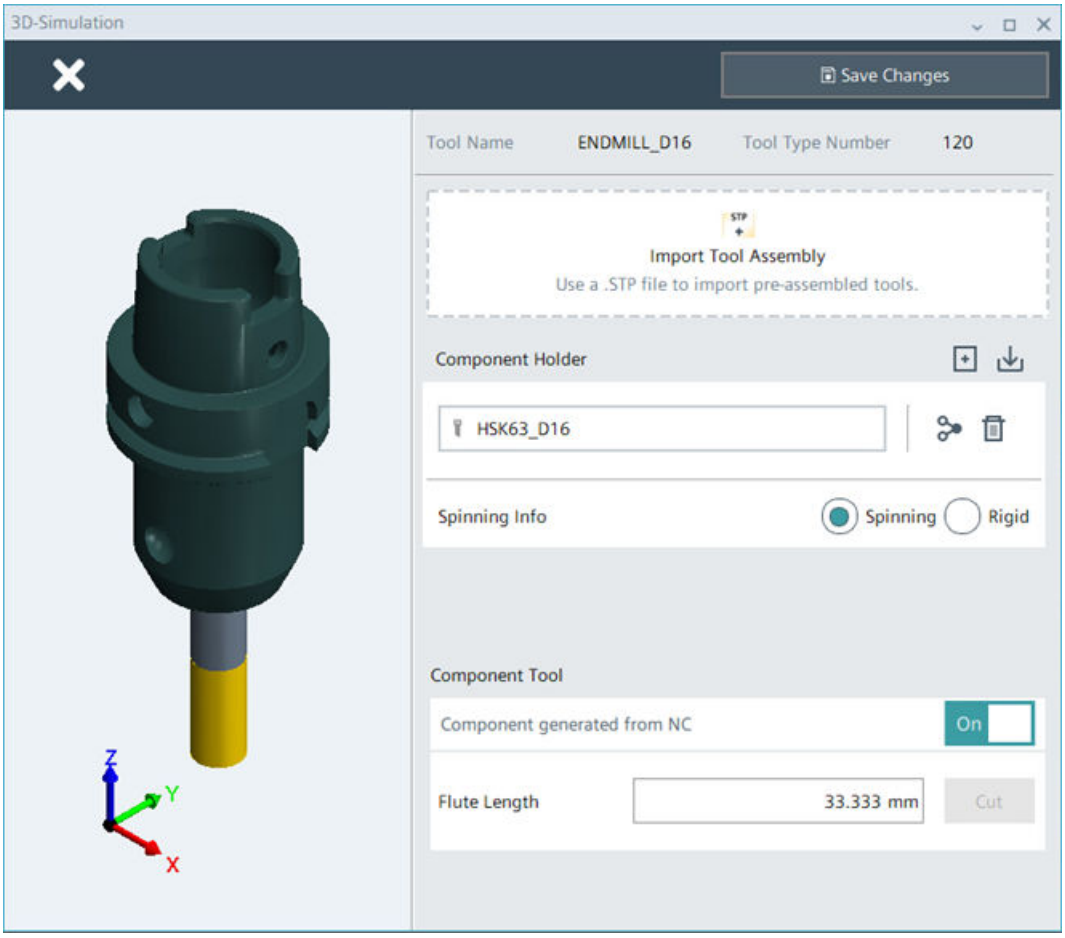


Figure 6-42 Tool holder from the library assigned to the tool

4. For milling tool/drill bit holders, define whether it should be used "Rotating" or "Rigid". As default, all holders should be configured as "Rotating".
5. Repeat the last steps if you want to assign additional part geometries to the holder.
6. Click on if you wish to delete existing holder geometries.
7. Confirm with "Save changes".

See also

Defining detailed protection for milling tools/drill bits (Page 102)

6.6.13 Defining holder/tool geometries using parameters (only turning tool)

To define the tool protection, parameterize a tool holder (adapter) and the tool shank. You define different parameters depending on the tool type. The dimensions of the cutting plate are taken from the SINUMERIK Operate tool list.

Precondition

The dialog to configure "detailed protection" for the tool opens.

This is how you define protection of a tool using parameters

In the following, the procedure is illustrated using a turning tool as an example.

Note

Sequence when defining

When defining, you should first parameterize the shank, and then the holder. This is because the parameters for the holder are automatically restricted depending on the shank type. First define the holder to be able to use the full selection of parameters.

1. In the "Component Tool" area, click on . The parameters are displayed.
2. Click the "v" button to display a detailed drawing of the shaft values.
3. Select the shank type and enter the parameter values.

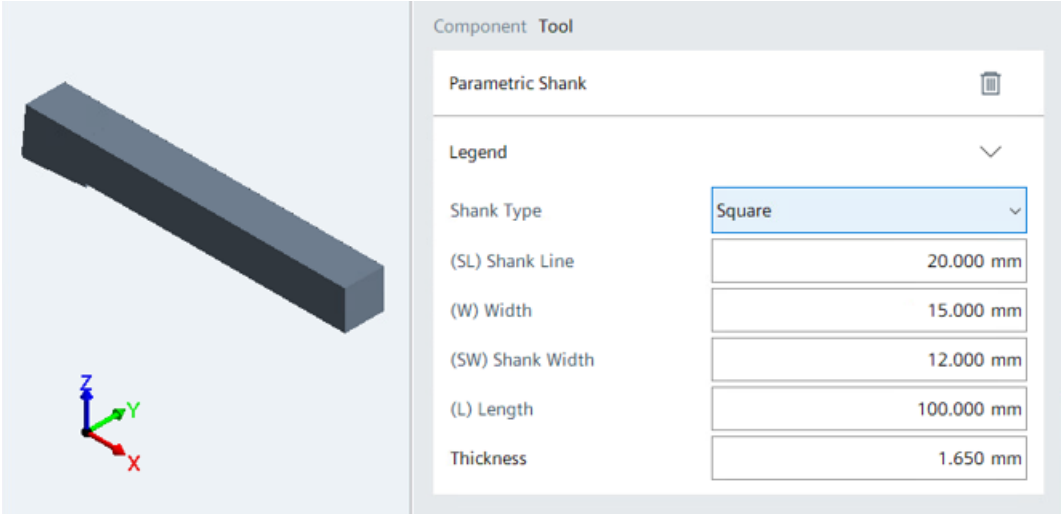


Figure 6-43 Defining the turning tool shank using parameters using a square shank as example

4. In the "Component Holder" area, click on or . The parameters are displayed.
5. Click on "v" to display a detailed drawing of the holder.

6. Enter the parameter values.
   The selection of parameters is restricted for a square shank. For a round shank, first select the "axial" or "radial" design of the holder. Other parameters are displayed dependent on this.

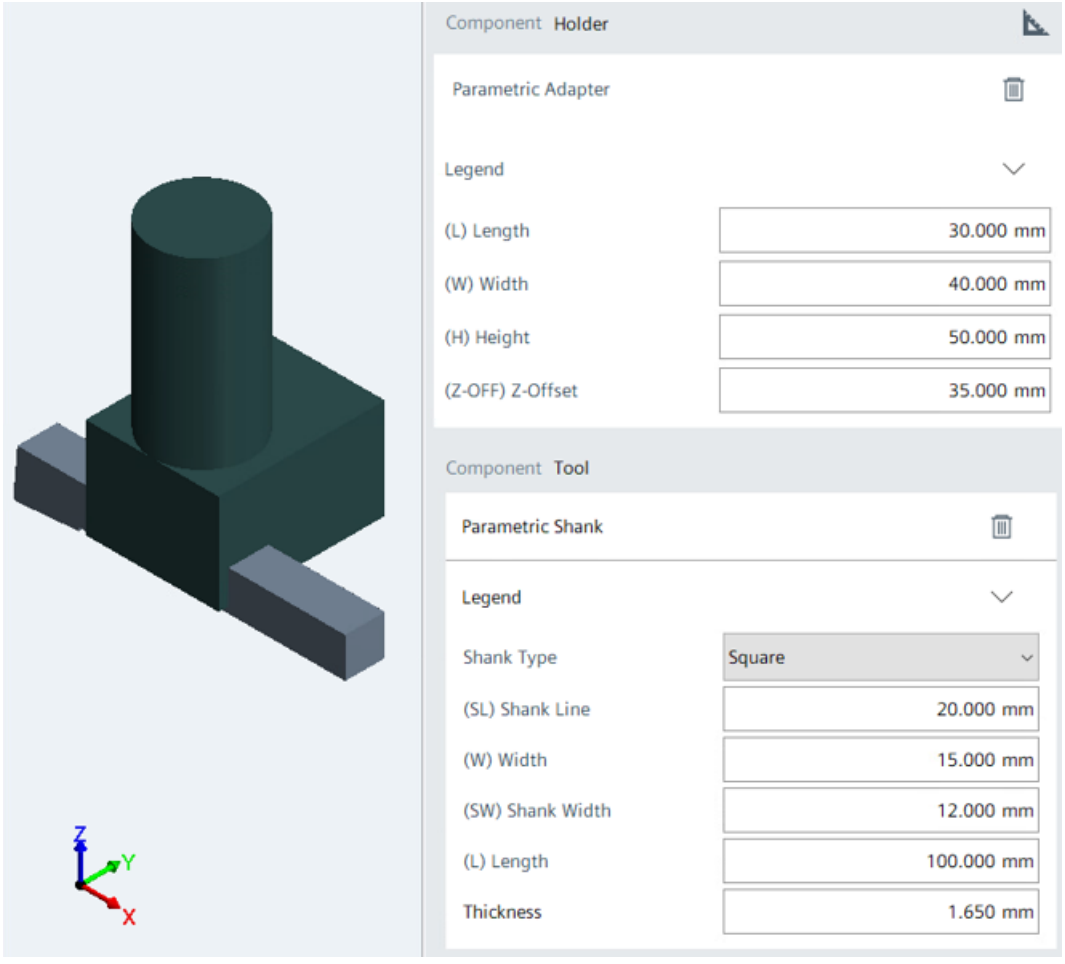


Figure 6-44 Defining a parametric adapter using a turning tool as an example

7. Confirm with "Save changes".

6.6.14 Importing a tool geometry as "*.stl./*.stp"

To define the tool geometry, import an "*.stl/*.stp" file. For the collision avoidance, tool data from the SINUMERIK Operate tool list are no longer used, but instead imported tool geometry. For milling tools/drill bits, the tool geometry comprises the tool shank and optionally, the indexable inserts.

Requirement

The dialog to configure "detailed protection" for the tool opens.

This is how you define a tool geometry using imported "*.stl/*.stp" files

In the following, the procedure is illustrated using a milling tool as an example.

1. In the tool component area, bring the switch next to "Components generated from the NC" into the "Off" position. The field to import the tool geometry is displayed.
2. Click on the button next to the tool component. The dialog to select the "*.stl/*.stp" file is displayed.
3. Select a file with the 3D geometry of the holder.
4. Confirm with "Open". The 3D program is displayed.
   The next step depends on the selected file type.
   - "*.stl" file
     The tool geometry is immediately imported.
   - "*.stp" file
     To import, activate the checkboxes for the individual elements that you wish to import as part geometry. You can define separate geometries for the shank and indexable inserts. Confirm with "Import".

   The imported tool component is displayed.

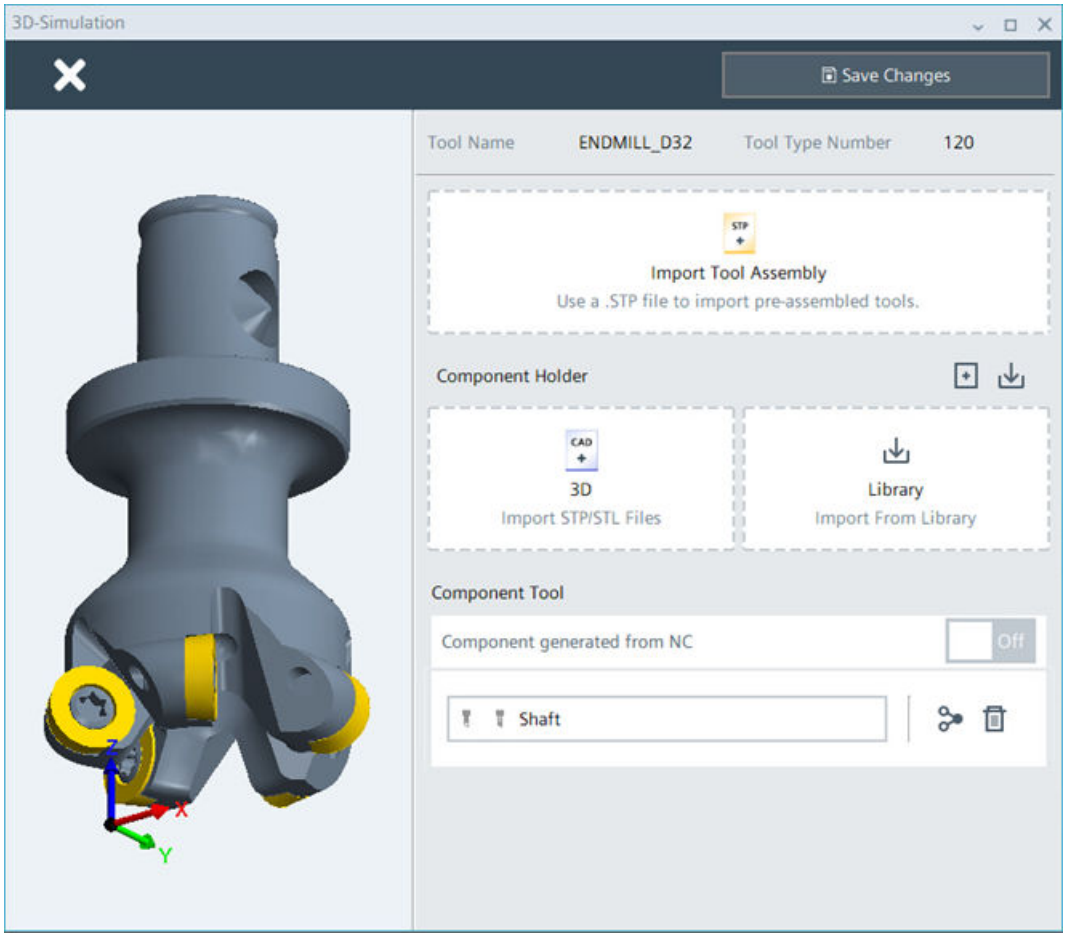


Figure 6-45 Importing the tool geometry as "*.stp" file (shank and indexable inserts)

5. Click on if you wish to configure the reference system to align the components.

6. Import a geometry for the tool holder to complete the detailed protection for the tool. The tool is displayed as "Not-defined" in the tool manager if you do not define a tool holder.
7. Confirm with "Save changes".

The tool geometry is defined.

6.6.15 Tool-specific stock removal in different colors

In the tool manager, define a specific cutting edge color for each tool. For stock removal simulation, in the 3D model, the surface machined by the tool is displayed in the defined cutting edge color. In stock removal simulation, you can immediately identify which tool is presently being used for machining or the tool that was used.

This is how you define a tool-specific cutting edge color

The cutting edge color of the tool can only be defined if the spindle and feedrate have been stopped.

1. In the tool manager, for the selected tool, click on icon to select the color. The window to select the color is displayed.
2. Select a color and confirm with "OK".
   The tool cutting edge is displayed in the tool manager with the defined color.

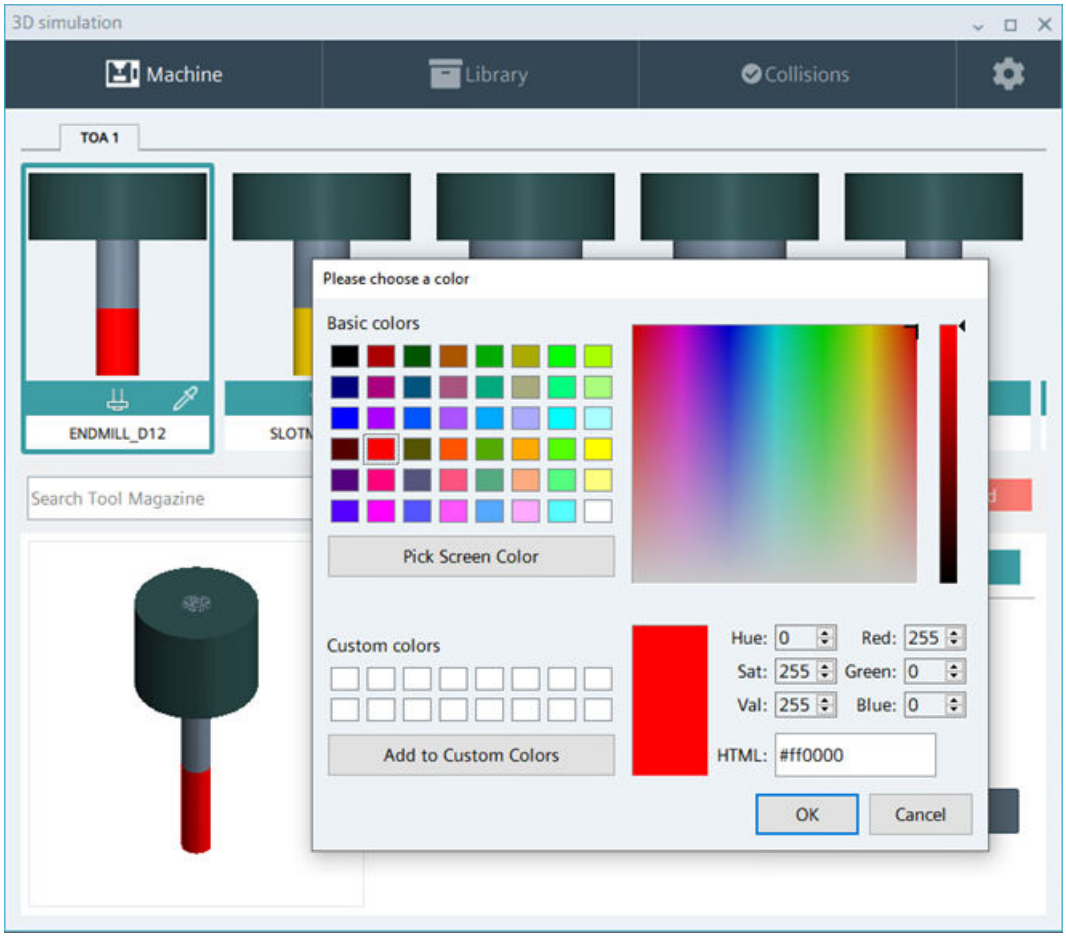


Figure 6-46 Example of an end mill: Defining a cutting edge color for a tool

3. Repeat the steps if you want to define additional cutting edge colors for tools.

In the stock removal simulation, the machined surface is displayed in the color of the tool cutting edge.

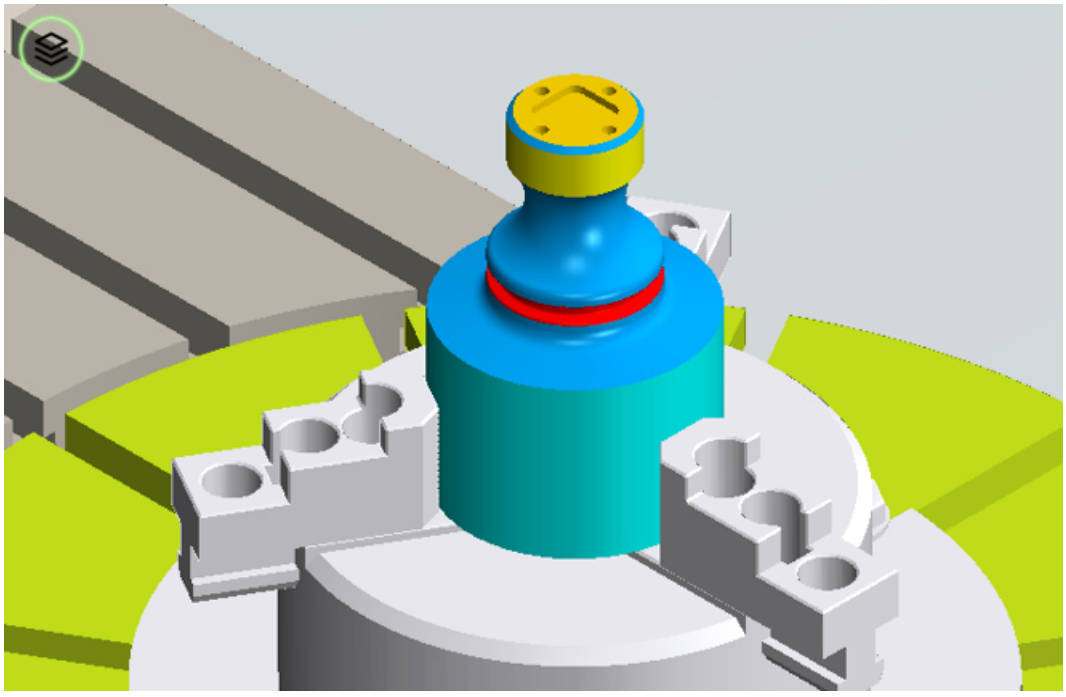


Figure 6-47 Stock removal simulation Machined surfaces are displayed in the colors of the tool cutting edges

6.7 Setup (clamping)

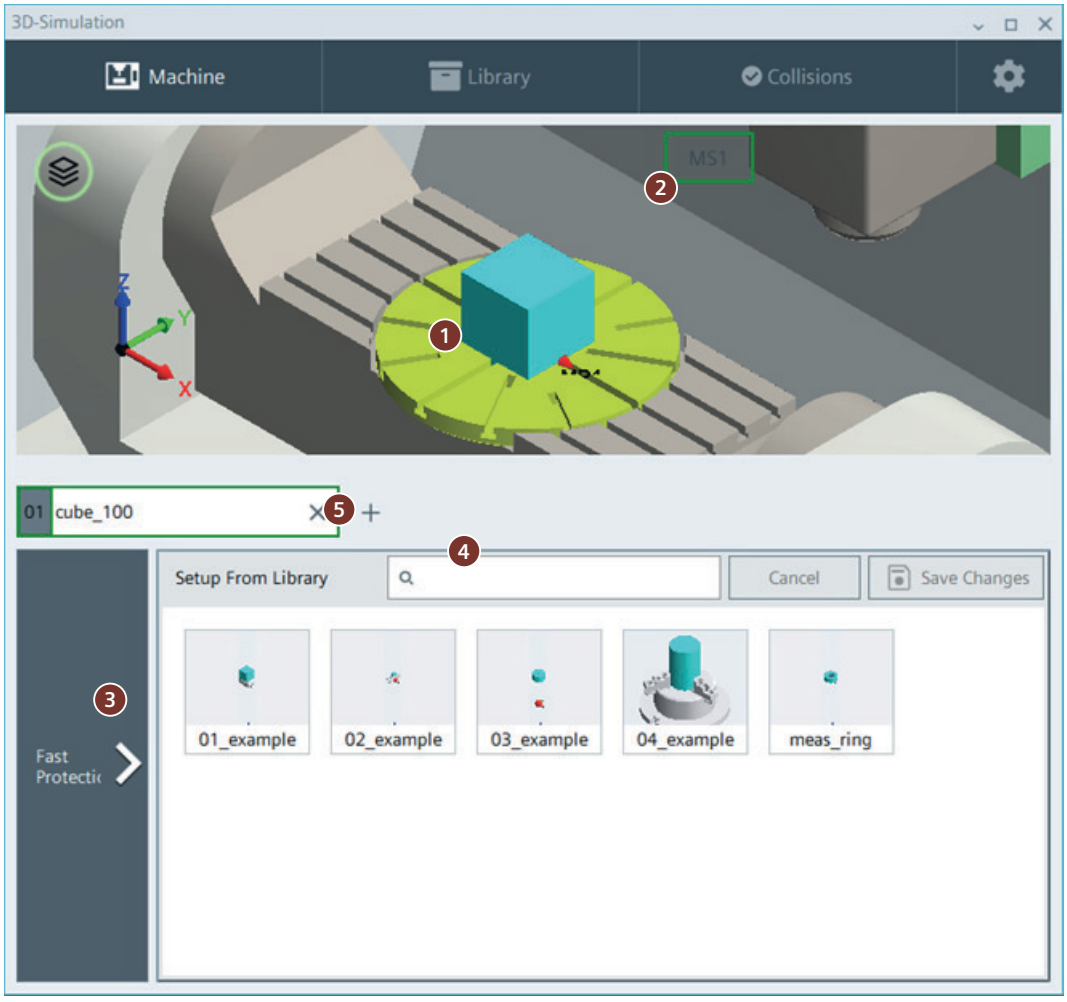
6.7.1 Setup manager (clampings)

Active setup (clamping)

For the 3D simulation, activate/deactivate the effective setup in the "Setup" tab. The window is displayed when you click the "SETUP" icon on the 3D simulation start page.

Functions of Setup Manager

You can select the possible setups using the "Fast Protection" or "Detailed Protection" button. Click on the button in each case to display and parameterize the corresponding setup.



① Preview of the machine space
② Number of Mount Stations (MS). One MS1 is available as standard.
③ Activate setup as "Fast protection"
④ Activate the setup from the library "Detailed protection"
⑤ Deactivate the setup by clicking on "X"

Figure 6-48 Setup Manager: Example, fast protection has been configured

6.7.2 Fast protection (setup)

With the "fast protection" you define a protection area around a blank geometry. No 3D geometry required as clamping equipment.

How to enable "fast protection"

The procedure for activating "Fast Protection" is shown using an example of a milling machine with one mount station MS1 .

1. Click on the "Fast protection" button. The area is displayed.
2. Click the "Select blank" button. The available blanks from the library are displayed.
3. Select a blank and confirm with "Import". The blank is imported.
4. Enter a "height" for the protection area. The effective protection area size is automatically scaled on the basis of the protection area stored in the machine model.
5. Click "Activate".
   The defined, fast protection is taken over as setup. The blank and the setup are displayed. The frame of MS1 is shown in green.

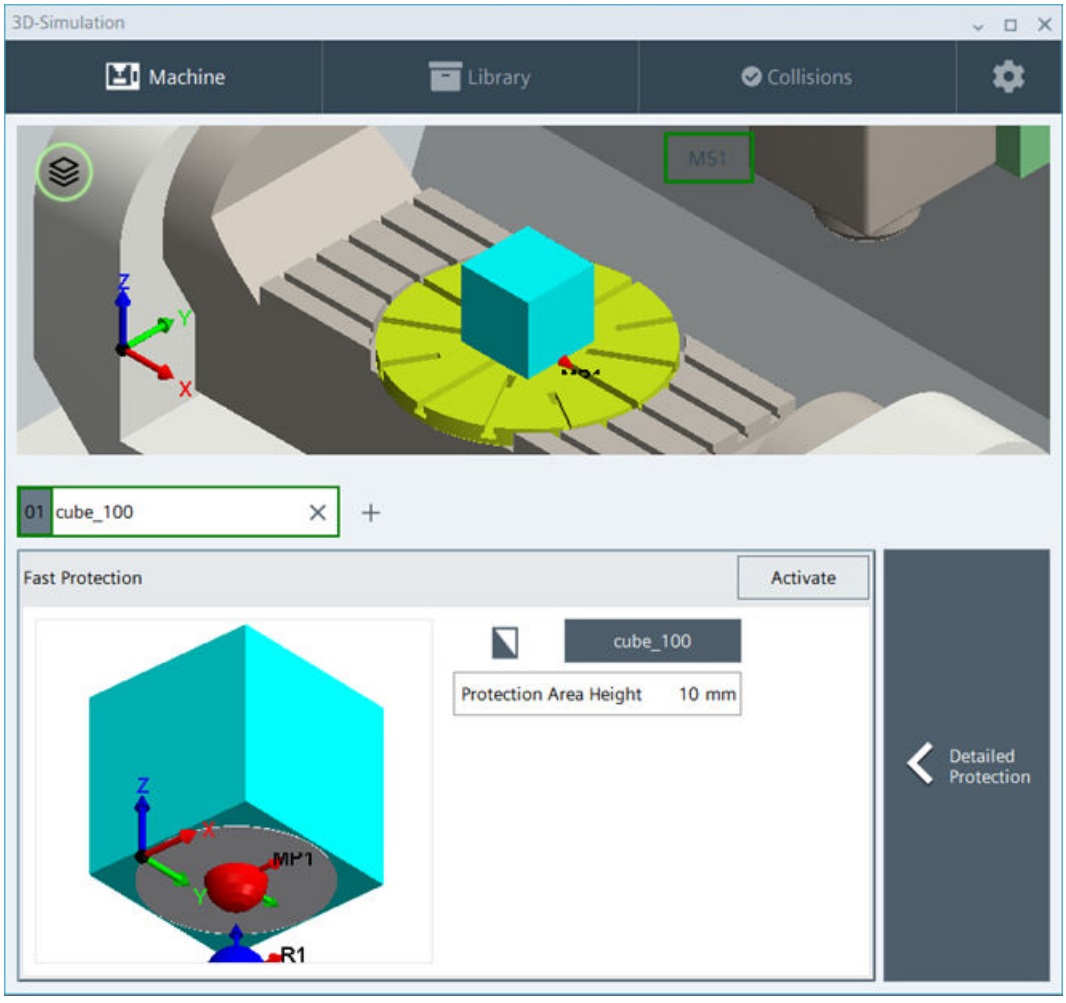


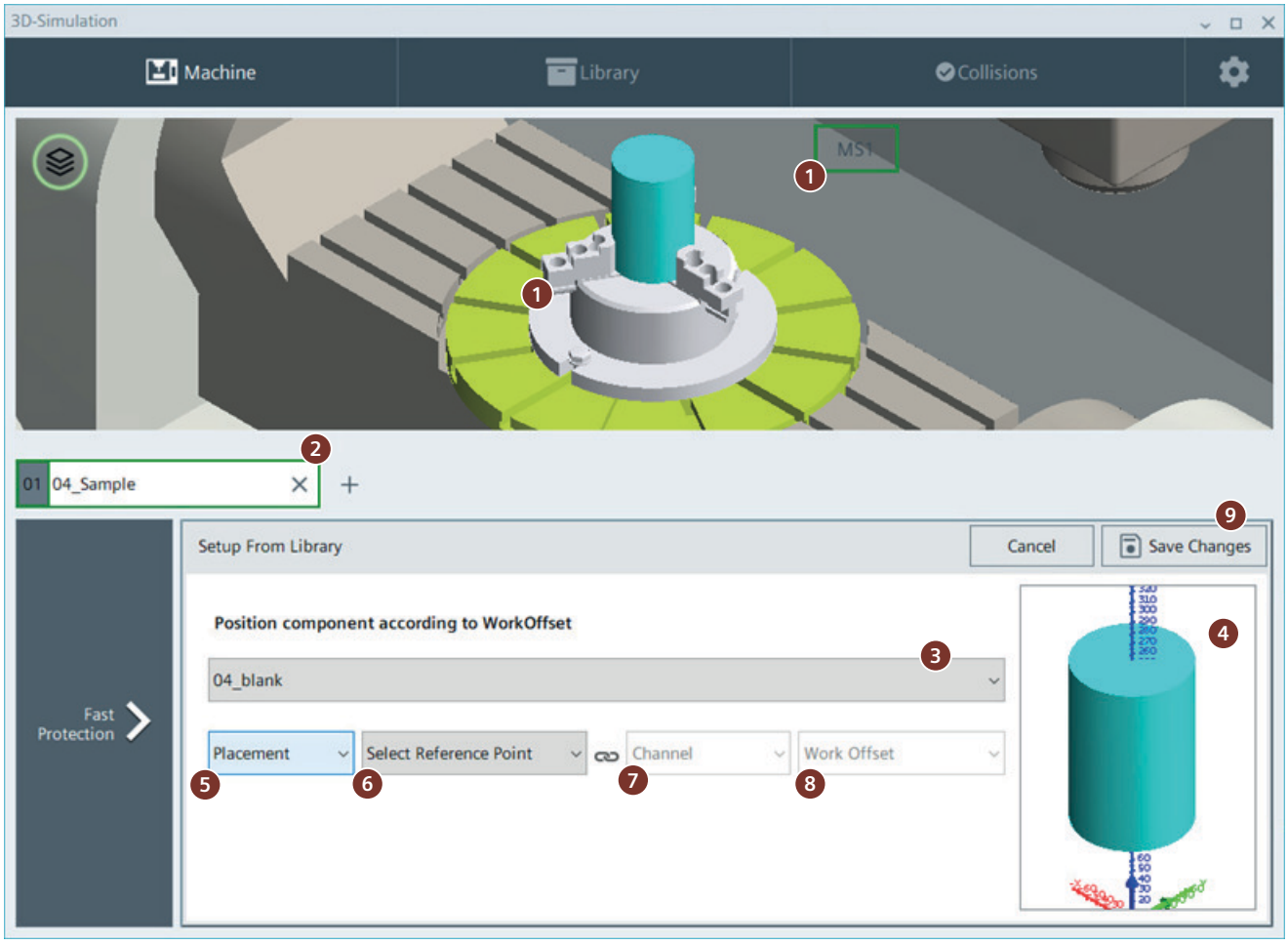
Figure 6-49 Active setup with "Fast Protection" The mounting point (red) lies centered at the lower side of the blank.

6. If the position of the blank is not the required position in the machine space, then you can adjust the reference points in the library and redefine fast protection. As standard, the mounting point defined for the blank is placed at the mount station of the machine.
7. You can disable the active protection area by clicking the "X" next to the setup.

6.7.3 Detailed protection (setup from library)

To define a "detailed protection", activate a setup (clamping) from the library.

Overview of Setup Manager



① 3D preview of the setup for the selected mount station MS1

② Active setup. The "X" deactivates the active setup.

③ List of the components used in the setup. The position of each component can still be changed.

Select the component whose position you wish to change.

④ 3D preview of the selected component (blank, protection area)

⑤ **Placement**

As standard, the component is placed with the mounting point MP (Mounting Position) at the mount station (Mount Station) of the machine. Using the drop-down list, you can change the position of mounting point and therefore the position of the component.

MP Position

Enter the shift (Z, Y, X) and rotation (rZ, rY, rX) of the reference point. The new component position is immediately displayed.

⑥ **Select reference point**

You can align defined reference point R of the component to a defined work offset, e.g. G54. This means that you change the position of the component.

⑦ **Channel**

If you align a reference point to a work offset, you must first select the channel in which the work offset is active.

⑧ **Work offset**
Here, select the work offset (Gxxx), to which the component reference point should be aligned.

⑨ Click on "Save changes" if the setup should be saved. With "Cancel" the changes are discarded.

Figure 6-50 Setup Manager: Example of a setup from the library

How to enable "Detailed protection"

1. Click on "Detailed protection". The area with the setups from the library is displayed.
2. Double-click the setup from the library that you want to activate. The setup is activated and can be configured.
3. Optionally, you can change the position of components blank and protection area.
   – Changing the position of the mounting point of the component (Page 122)

   or

   – Align reference point R to a work offset (Page 123)
4. Confirm the setup with "Save changes".
   The setup from the library is now active.

Note

An active setup from the library cannot be edited, i.e. reference points cannot be changed. The setup must be deactivated before it can be edited in the library.

More information

Using several active setups (Page 124)

Using several mount stations (MS) of a machine (Page 126)

6.7.4 Changing the position of the MP (mounting point) in a setup

The position of the mounting point of components (blanks, protection areas) can be changed in the Setup Manager. For example, this is necessary if the complete clamping is to be shifted on the machine table, or the position of the blank is to be changed in the clamping equipment.

Requirement

The setup of a "Detailed Protection" is selected in the Setup Manager.

This is how you change the position of a component mounting point

1. Select the component whose mounting point is to be changed. A blank is used in the example.
2. Select entry "MP position"
3. Enter the values for the shift and the rotation. In the example, the blank is shifted to X=-20 mm.

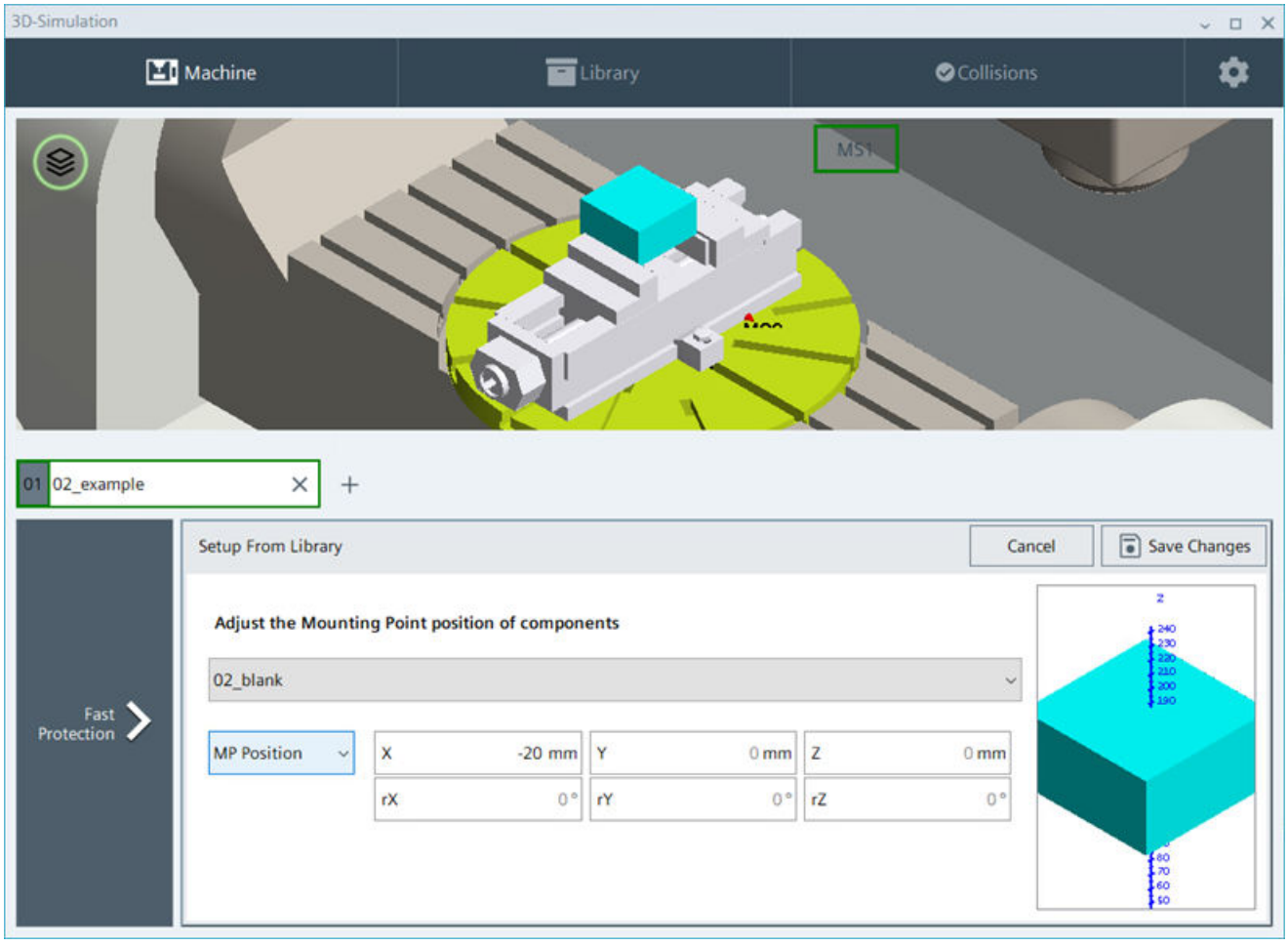


Figure 6-51 Example of blank: Position of the reference point (Mounting Point) shifted to X=-20 mm

6.7.5 Align reference point R to a work offset

Reference point R of a component (blank, protection area) can be assigned to a work offset in the Setup Manager.

Requirement

- The setup of a "Detailed Protection" is selected in the Setup Manager.
- The component has at least one reference point.

This is how you connect a reference point with a work offset

1. Select the component whose reference point is to be connected with the work offset. A blank is used in the example.
2. Select entry "Placement".
3. Select the reference point, e.g. R1.
4. Select the channel in which the work offset is active. For 1-channel machines, the channel is permanently preassigned with CHAN1.
5. Select the work offset, e.g. G54.

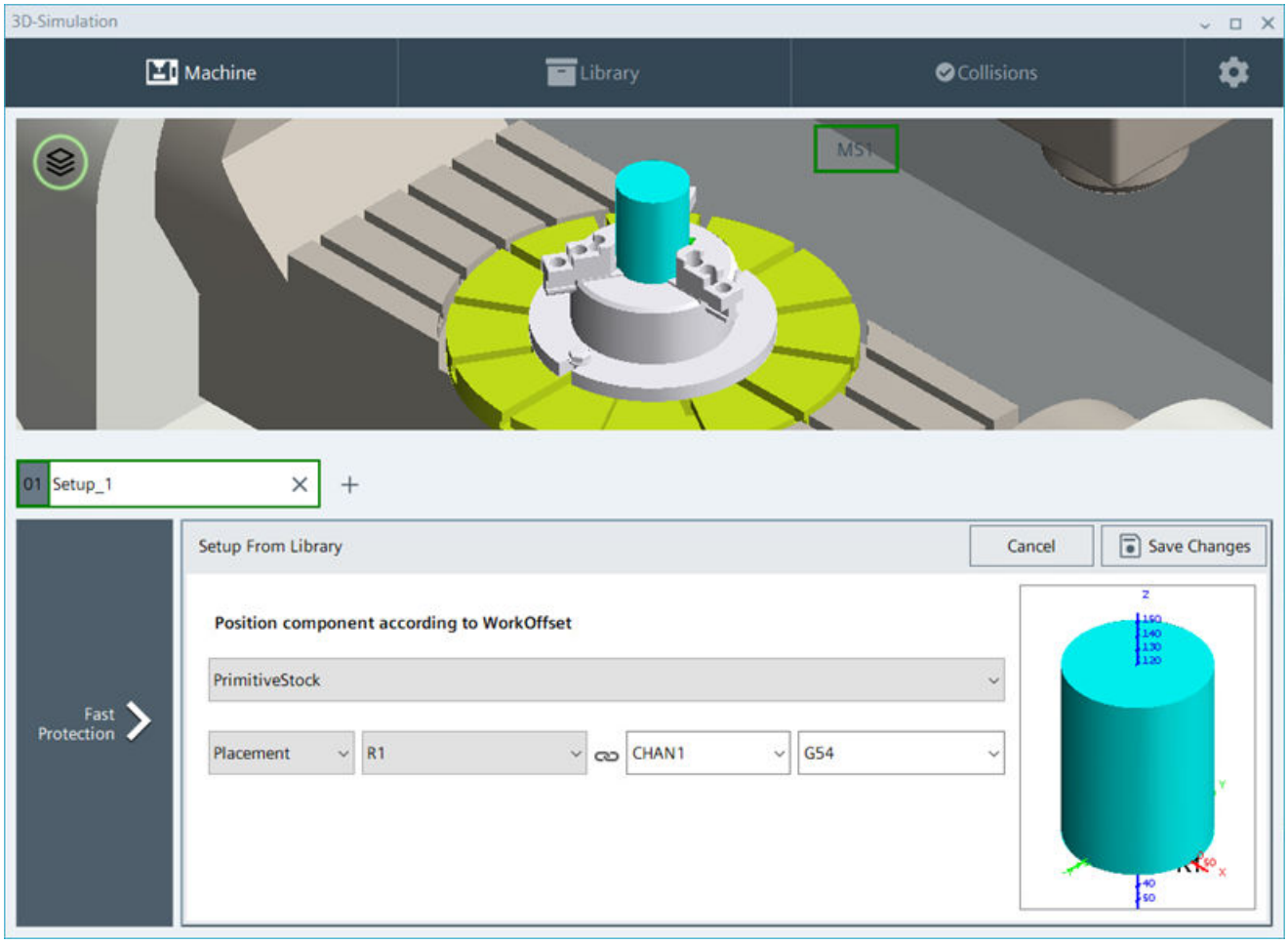


Figure 6-52 Reference point R1 of the blank is connected with work offset G54

6.7.6 Several setups for a mount station

You can define and activate several setups for a mount station in a machine. This is automatically used for a machine with only one mount station. For several mount stations, first select the mount station (e.g. MS1 or MS2) that you wish to use.

You can define the setups as "Fast Protection" or "Detailed Protection".

Requirement

A setup is already activated.

This is how you define several setups

1. Click on "+" next to the active setup.
   The area with the setups is displayed.
2. Select setup "Fast Protection" or "Detailed Protection".
3. Define the protection corresponding to the selected setup
   - Configuring fast protection
   - Configuring detailed protection

   The defined setups are displayed.
4. To edit an active setup, click on the corresponding setup. The frame of the active setup is shown in green.

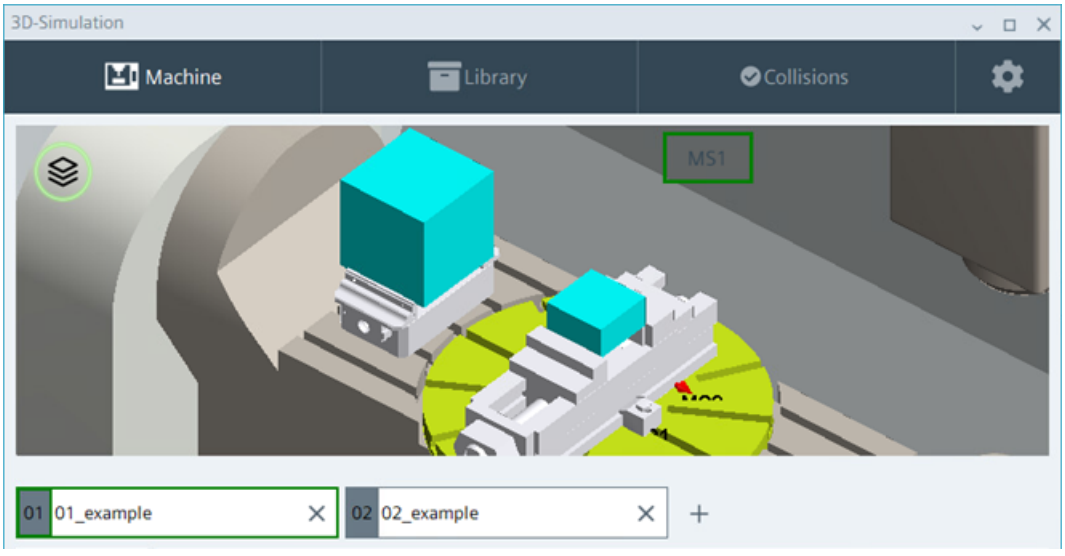


Figure 6-53 Example, milling machine with one mount station (MS1): Two setups are defined and activated

5. Repeat the steps if you want to define and activate additional setups.

6.7.7 Using several mount stations

If the machine has several mount stations (MS), you can define setups for each mount station and then switch between these. In the 3D simulation, the mount stations are displayed with MS1 to MSx. Mount stations in which setups are defined are shown with a green frame. Mount stations in which no setup has been defined are shown with a red frame. The active mount station has a thick frame.

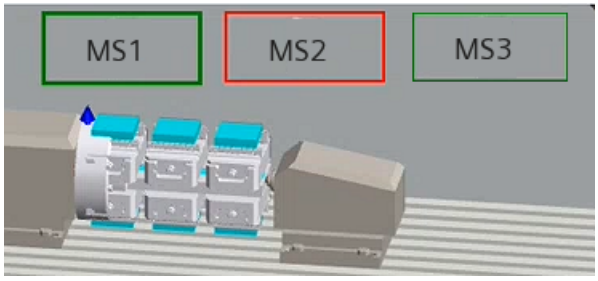


Figure 6-54 Examples of mount stations: MS1 configured and active, MS2 not configured, MS3 configured and not active

This is how you use several mount stations

1. Click on the mount station that you wish to configure.
   The mount station is shown with a thick frame and the area with the setups is displayed.
2. Select setup "Fast Protection" or "Detailed Protection" and define the protection.
   You can define several setups for a mount station
3. Save the changes.

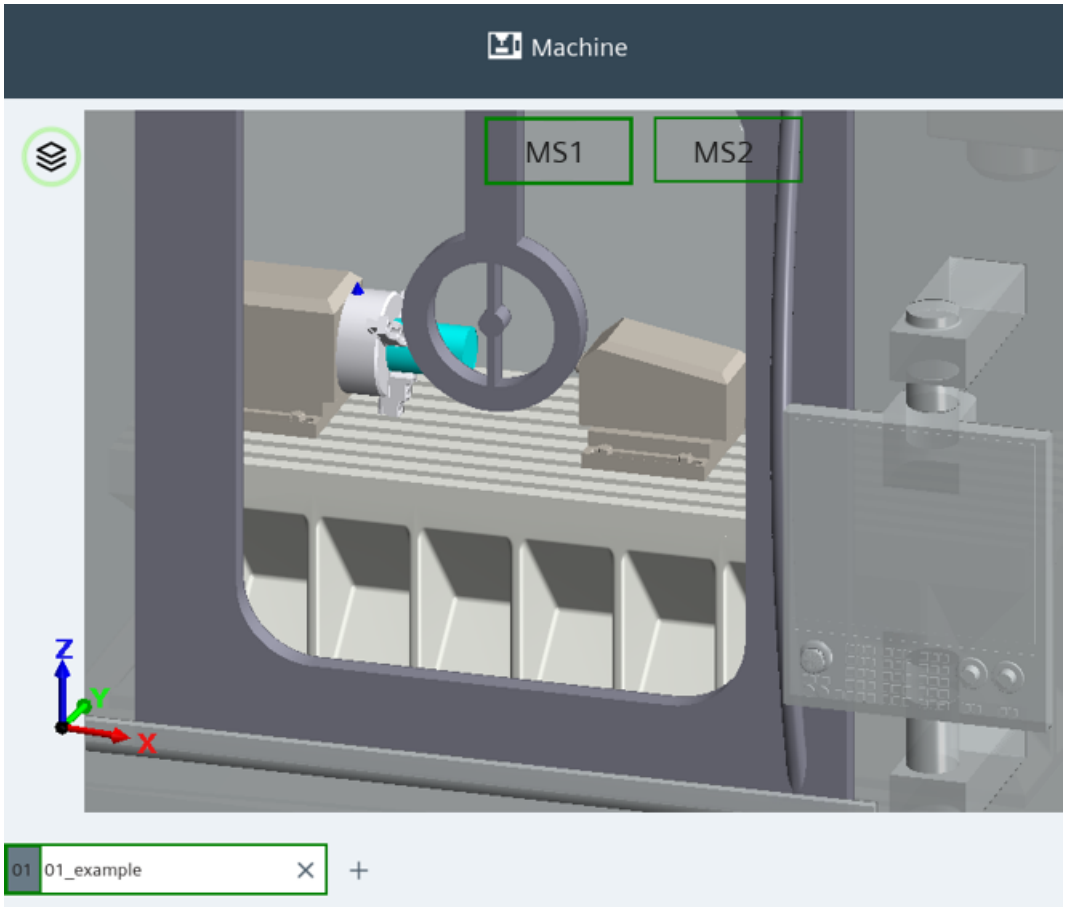


Figure 6-55 Example, mount station MS1 is active and configured

The configured and active mount station is marked with a thick, green frame.

6.8 Collisions

6.8.1 Collision detection

During 3D simulation of the machining process, collisions that occur while collision detection is active are logged and visually displayed in the 3D simulation. The collision components are marked in color.

- Orange: Safety clearance of the components violated, but no collision yet
- Red: collision of the components

Collision log

In the "Collisions" tab, all collisions that have occurred are displayed in tabular form and with a 3D image and additional information. The collisions are saved at the same time in the "collision.log" log file.

Storage path for log file:

"C:\Users\<username>\AppData\Local\Siemens\Automation\SINUMERIK ONE\ncu\card\user\sinumerik\3d\collisions\collision.log"

List of collisions

The following data is listed in the collision list or saved in the log for each collision:

- Collisions type
- Date, time
- Number of components involved

All logged collisions and the saved log can be deleted using the "Delete" button. Single collisions can be deleted line-by-line in the table.

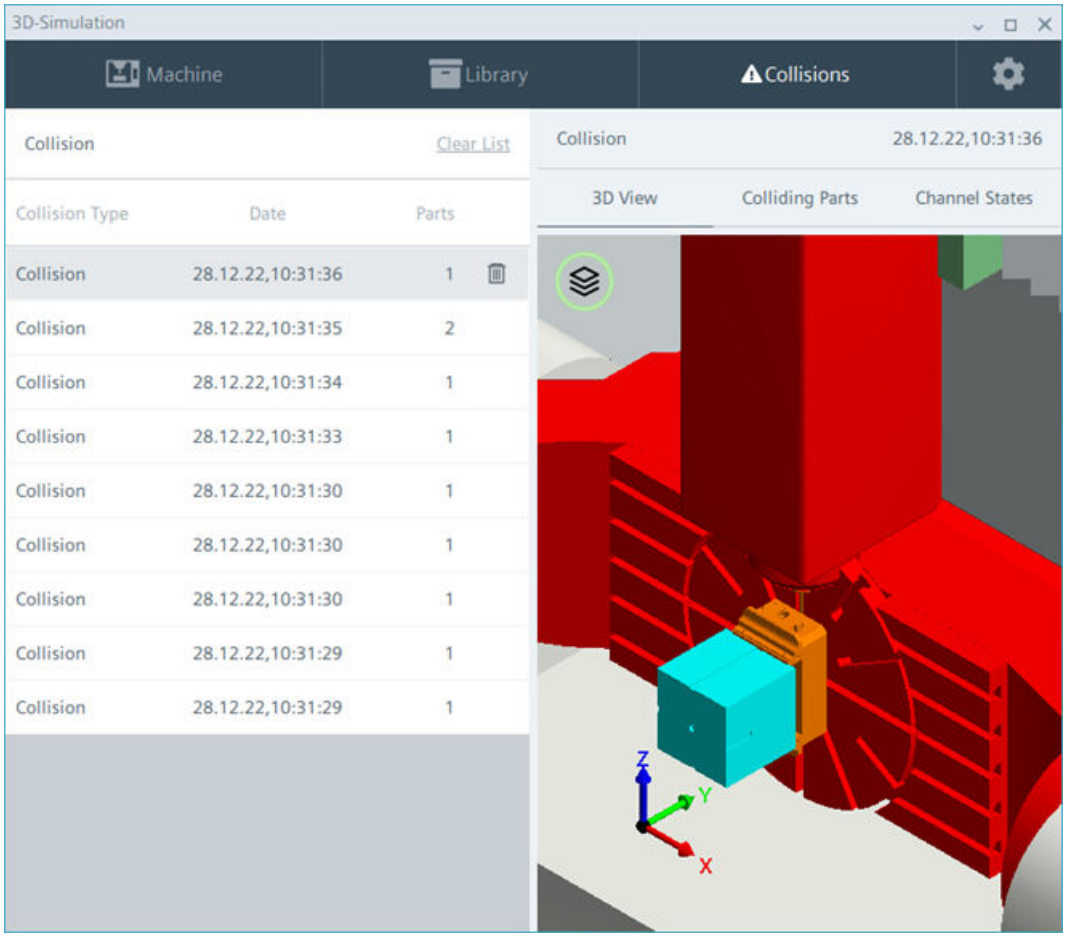
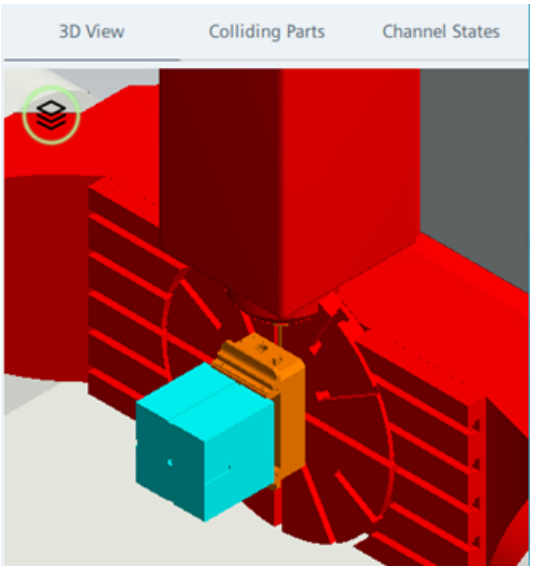
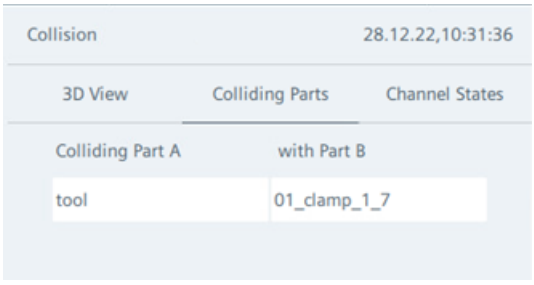
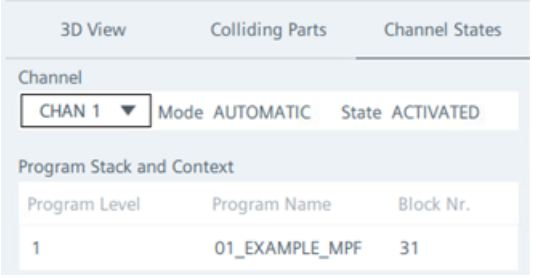


Figure 6-56 Collision screenshot, components involved are shown in color

Collision details

Details on the selected collision are displayed in the table in the tabs.

Tab	Meaning
3D image of the collision	
	The 3D image can be zoomed, rotated and moved in order to better display the details. From the layer menu, you can specifically hide layers, for example the machine housing, in order to better analyze the collision situation.
Colliding parts	
	Which components are involved in the collision, e.g. tool, protection area, machine component
Channel states	
	**Channel** For multi-channel machines, first select the channel to display details on the collision. A selection is not possible for single-channel machines. **Mode** Operating mode in which the collision occurred, e.g. JOG, AUTOMATIC. **State** State at the time of collision, e.g. ACTIVATED **Program stack and context** Stack level of the program together with the program name and the line number in which the collision occurred.

6.8.2 Activating collision detection

Before you start the 3D simulation of the machining process, activate collision detection so that any collisions that occur during machining are displayed and logged. Collision detection cannot be enabled or disabled during the machining process.

How to activate/deactivate collision detection

1. Click on the red "Deactivated - Click to activate" button in the "Machine" tab in the right-hand footer of the window.
   The button is displayed in green with the caption "Activated". Collision detection is now active.
2. If you want to disable the collision detection again, click the button again.
   The button is shown in red and detection is deactivated again.

Collisions that occur are logged on the "Collisions" tab.

6.9 Simulating machining operations and workflows

6.9.1 Measuring processes in the 3D simulation

The 3D simulation simulates and visualizes measuring processes with switching probes. The measuring process is identical to the procedure on a real machine and has the same prerequisites.

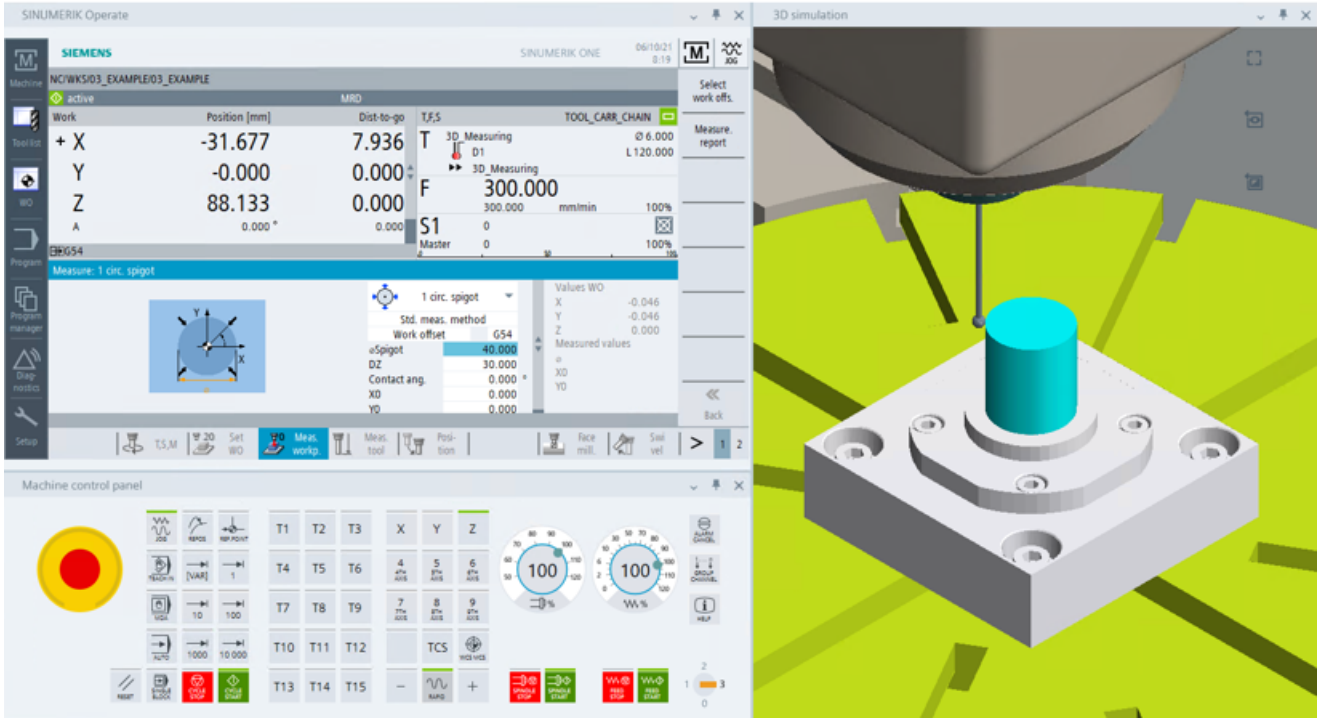


Figure 6-57 Measuring using the example of a measuring cycle in JOG mode

Note

To record the measured values correctly, only real axes may be used, i.e. the axes must be assigned to drives, just as on a real machine. Measuring with virtual axes or in simulation mode does not provide correct results.

Functions

The following functions are supported.

- Measuring the workpiece or clamping
- Use of the workpiece measuring cycles in JOG, MDI and AUTOMATIC
- Support from the first measurement channel
- Contact of the probe with surfaces (e.g. workpiece, clamping) is not evaluated as a collision, but provides the edge for measured signal acquisition.

Restrictions

- No measuring of tools, e.g. with load cells or laser measurement.
- Collisions between the probe (shaft, unclamping) and other collision elements are not permitted.
- Measurements only with probe-tool types that are listed in Section Probe (Page 93).
- Only HighActive probes are supported (no LowActive).

6.9.2 Automatic mode with 3D simulation

During execution of an NC program in AUTOMATIC mode, 3D simulation with collision detection enables you to check the processing so that any program errors can be detected.

Requirement

- Requirements for 3D simulation are fulfilled
- The NC program is loaded in AUTOMATIC mode

How to reset the view

Before starting the simulation, reset the view to the "Machine" tab.

1. Click on "Maximize or minimize the machine view". The view of the machine is maximized or minimized.
2. Click on "Reset view". The zoomed and moved view of the machine is reset to the default view.
3. Click on "Reset workpiece". The processed workpiece is reset to the blank view.

How to work with the 3D simulation

1. Check whether all of the requirements have been met.
2. Click on NC-Start in SINUMERIK Operate. The program is processed and machining of the blank is simulated.

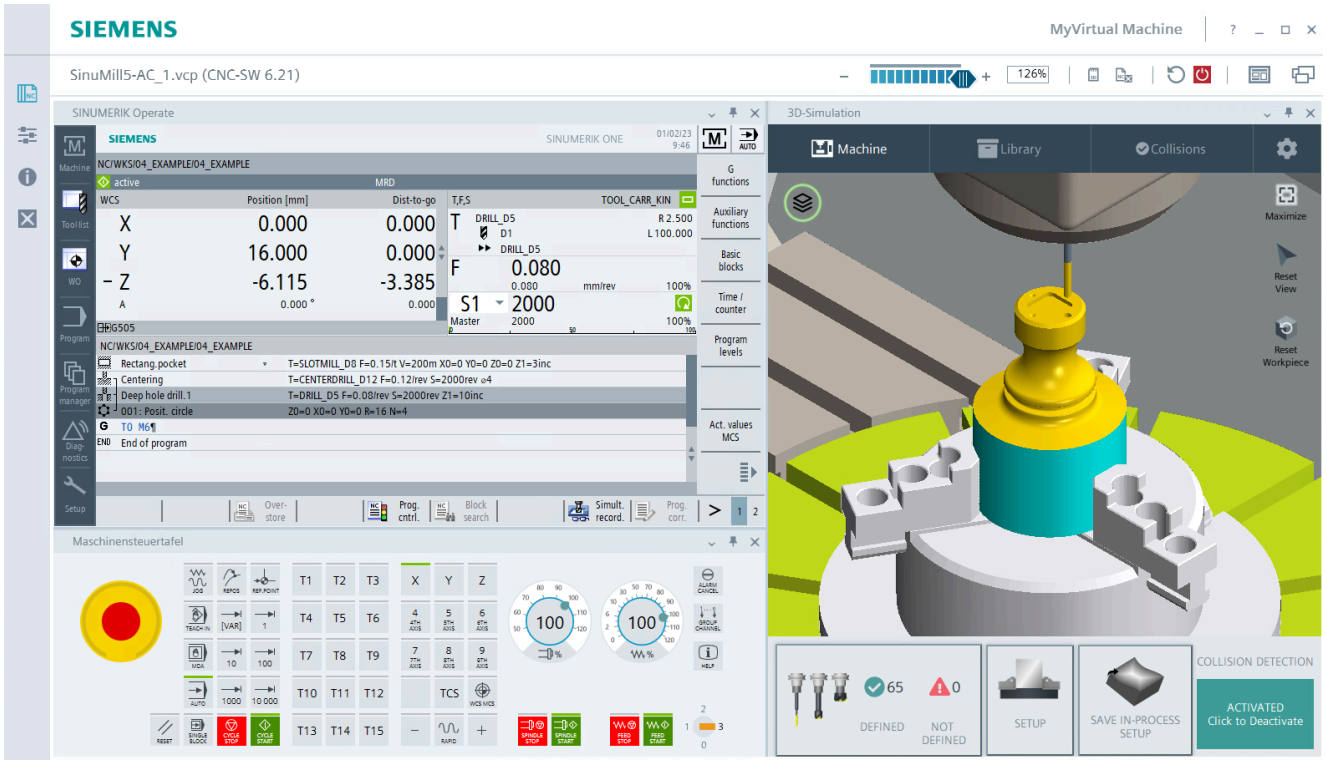


Figure 6-58 Example: simulation of machining process

The following functions are provided to assist you during the processing and simulation

- Visualization of the workpiece machining and all machine movements
- Visualization of collisions using colored highlighting
- Logging of the collisions
- Zooming and moving the view to better recognize the details

6.9.3 Saving the configuration in the project

You can save the current machining state of the workpiece and the active setup by pressing button "Save in-process setup". This may be necessary, for example, to save the current machining and use it again as a basis for further machining at a later time.

The following components are saved as a copy under an automatically generated name:

- Storage folder:
  - STL file of the machined workpiece
- Library:
  - Machined workpiece
  - Active setup

Constraints

- Only one setup and one workpiece can be saved.
- Saving is only possible during no-load operation (spindle and feedrate stop).

How to save the machining state

1. Stop the spindle and feedrate.
2. In the "Machine" view, click on "Save in-process setup".
   The workpiece and the setup are saved. A message signals the successful saving process.
3. Start the spindle and feedrate. Machining is continued.

Open interface (option) 7

7.1 Introduction

Description

The Open Interface of Run MyVirtual Machine allows external applications to control the Run MyVirtual Machine system and to communicate at runtime. Usually, the Open interface is used by machine manufacturers or simulation software providers. As a machine operator, you are provided with a complete solution by the machine manufacturer.

Possible applications of the Open Interfaces are:

- Remotely controlling Run MyVirtual Machine. An external application starts, operates and exits Run MyVirtual Machine.
- Operating Run MyVirtual Machine in an external simulation product.
- Connecting to an external peripheral simulation.

Note

License

The "Run MyVirtual Machine /Open" license is required to use the Open Interface.

Index

S

V

W