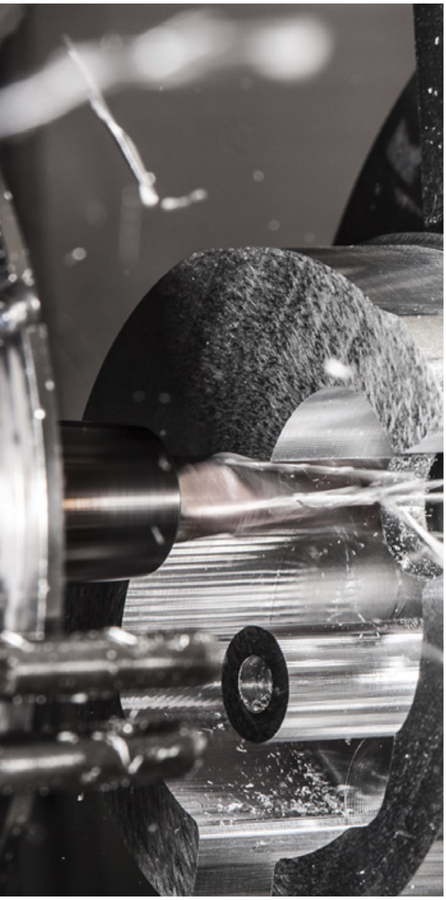
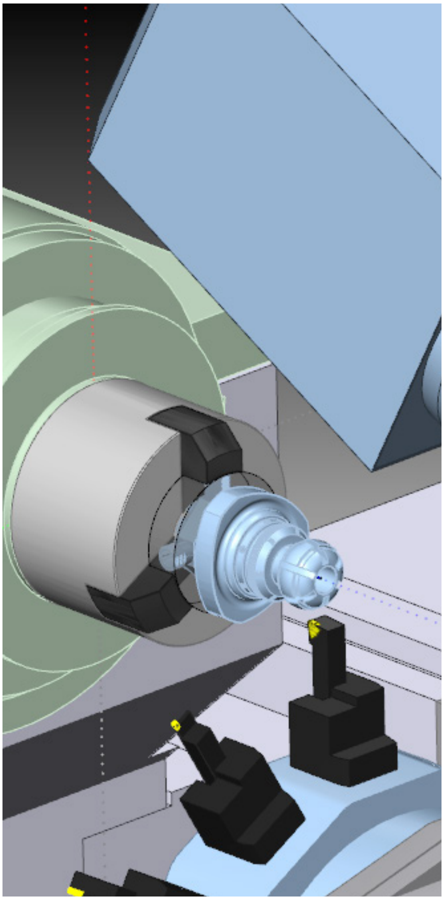
SolidCAM + SolidWorks

The complete integrated Manufacturing Solution

SolidCAM

iMachining – The Revolution in CAM!





SolidCAM Training Course: Turning & Mill-Turn

iMachining 2D & 3D | 2.5D Milling | HSS | HSM | Indexial Multi-Sided | Simultaneous 5-Axis | Turning & Mill-Turn | Solid Probe

www.solidcam.com

Turning & Mill-Turn Training Course

Contents

1. Introduction

1.1 About this course 3
1.2 Turning Module Overview 5
1.3 Mill-Turn Module Overview 7
1.4 Basic Concepts 8
1.5 Process Overview 8

2. CAM-Part Definition

Exercise #1: CAM-Part Definition in Turning 11
Exercise #2: CAM-Part Definition in Mill-Turn 25

3. Turning Operations

3.1 Basic Turning 38
Exercise #3: Turning Operations on Turning CNC-Machine 41
Exercise #4: Button Lock Machining 84
Exercise #5: Guided Ejector Bushing Machining 86
Exercise #6: Guide Pillar Machining 88
Exercise #7: Bearing Bush Machining 90
3.2 Advanced Turning: Rest Material 92
Exercise #8: Wheel Machining 93
3.3 Advanced Turning: Partial machining 105
Exercise #9: Long Shaft Machining 106
3.4 Turning on Mill-Turn CNC Machines 117
Exercise #10: Turning Operation on Mill-Turn CNC-Machine 120

4. Milling on Mill-Turn CNC-Machines

4.1 Facial Milling 144

Exercise #11: Facial Milling 145

4.2 Simultaneous 4-Axis Milling 159

Exercise #12: Simultaneous 4-Axis Milling 160

Exercise #13: Stopper Machining 164

Exercise #14: Bushing Machining 166

Exercise #15: Shaft Machining 167

4.3 Indexial Milling on Mill-Turn CNC-machines 168

Exercise #16: Indexial Milling on 4-Axis CNC-Machines 170

Exercise #17: Joint Part Machining 189

Exercise #18: Slotted Nut Machining 190

Exercise #19: Connector Part Machining 192

4.4 Using MCO on Mill-Turn CNC-machines 194

Exercise #20: Machining with Back Spindle 196

Exercise #21: Indexial Milling on 4-Axis CNC-Machines 203

Exercise #22: Indexial Milling on 5-Axis CNC-Machines 216

5. Simultaneous 5-Axis Milling on Mill-Turn CNC-Machines

Exercise #23: Turbine Blade Machining 237

Introduction 1

1.1 About this course

The goal of this course is to teach you how to use SolidCAM to machine various parts using Turning and Mill-Turn CNC-machines. This tutorial covers the basic concepts of Turning and Mill-Turn machining and is a supplement to the system documentation and online help. Once you have developed a good foundation in basic skills, you can refer to the online help for information on the less frequently used options.

Prerequisites

Students attending this course are expected to have basic knowledge of the SolidCAM software. The **SolidCAM 2.5D Milling Training Course** is recommended but not necessary to be studied before this course for better understanding of Milling with the SolidCAM software. For the last chapter of this course book, the Sim. 5-Axis Milling knowledge provided by the **SolidCAM Simultaneous 5-Axis User Guide** is required.

Course design

This course is designed around a task-based approach to training. The guided exercises will teach you the necessary commands and options to complete a machining task. The theoretical explanations are embedded into these exercises to give an overview of the SolidCAM Mill-Turn capabilities.

Using this book

This tutorial is intended to be used in a classroom environment under the guidance of an experienced instructor. It is also intended to be a self-study tutorial.

Exercises

The **Training Materials** archive supplied together with this book contains copies of the various files that are used throughout this course. The **Exercises** folder contains the files that are required for doing guided and laboratory exercises. The **Built Parts** folder inside the **Exercises** contains completed manufacturing projects for each exercise. Copy the **Exercises** folder to your hard drive. The SolidWorks files used for the exercises were prepared with SolidWorks 2013.

The **Machine files** folder contains a number of pre-processors (CNC-controller configuration file) used through the exercises of this book. Copy the content of this folder into your **..\SolidCAM2013\Gpptool** folder.

The **CNC-machine** folder contains the CNC-machines definition files used for the Simultaneous 5-axis milling exercises. Copy the contents of the **CNC-machine** folder into the machine definition folder on your hard drive (the default location is **C:\Users\Public\Documents\SolidCAM\SolidCAM2013\Tables\MachSim\xml**).

Windows® 7

The screenshots in this book were made using **SolidCAM 2013** integrated with SolidWorks 2013 running on Windows® 7. If you are running on a different version of Windows, you may notice differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

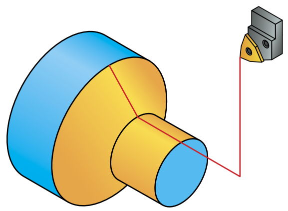
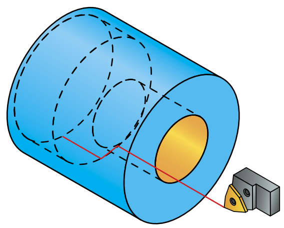
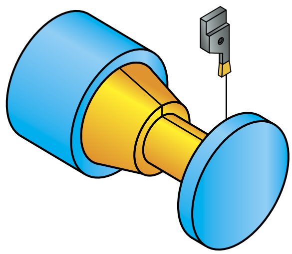
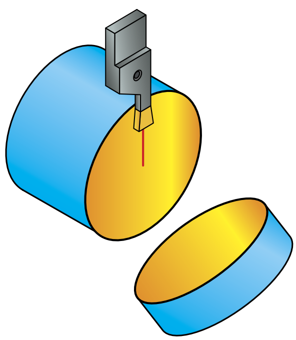
Conventions used in this book

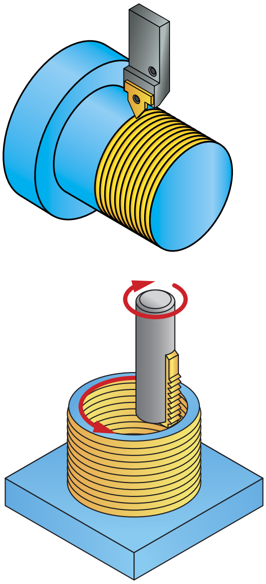
This book uses the following typographic conventions:

Bold Sans Serif	This style is used to emphasize SolidCAM options, commands or basic concepts. For example, click the **Change to opposite** button.
10. Define CoordSys Position	The mouse icon and numbered sans serif bold text indicates the beginning of the exercise action. The action explanation is as follows.
Explanation	This style combined with the lamp icon is used for the SolidCAM functionality explanations embedded into the guided exercises. The lamp icon is also used to emphasize notes.

1.2 Turning Module Overview

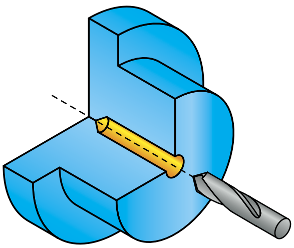
The SolidCAM Turning module enables you to prepare the tool path for the following operations:

 	**Turning** SolidCAM enables you to prepare the tool path for all types of external and internal turning operations: long external, long internal, face front, and face back.
 	**Grooving** SolidCAM enables you to prepare the tool path for all types of external and internal grooving and parting.



Threading

SolidCAM enables you to prepare the tool path for all types of external and internal threading.



Drilling

SolidCAM enables you to perform all drilling cycles to machine the holes coincident with the revolution axis of the part.

1.3 Mill-Turn Module Overview

The Mill-Turn module is intended for programming of 3-, 4- and 5-axis multi-task machines that combine several capabilities into one machine and especially for programming of 5-axis milling CNC-machines with turning capabilities (e.g. DMU FD-series of DMG). This functionality provides you with a number of significant advantages:

- The Mill-Turn module provides you with full functionality of the Coordinate System definition, identical to that of SolidCAM Milling.
- You can use the same coordinate system for milling as well as for turning without additional definition.
- You can define a Stock model to be used in SolidCAM Milling as well as in SolidCAM Turning operations.
- The Mill-Turn module enables you to perform all types of Milling and Turning operations using the same post-processor.
- The Mill-Turn module enables multi-turret and multispindle programming, with turret synchronization and full machine simulation.
- Using the Mill-Turn module, you do not need to learn new rules; you just work in regular milling environment and can add turning operations as needed.

1.4 Basic Concepts

Every manufacturing project in SolidCAM contains the following data:

- **CAM-Part**

  The CAM-Part defines the general data of the workpiece. This includes the model name, the coordinate system position, tool options, CNC-controller, etc.

- **Geometry**

  By selecting Edges, Curves, Surfaces or Solids, define **what** and **where** you are going to machine. This geometry is associated with the native SolidWorks model.

- **Operation**

  An Operation is a single machining step in SolidCAM. Technology, Tool parameters and Strategies are defined in the operation. In short, operation means **how** you want to machine.

1.5 Process Overview

Three major stages of the SolidCAM Manufacturing Project creation process are:

CAM-Part definition

This stage includes the definition of the global parameters of the Manufacturing Project (CAM-Part). You have to define Coordinate Systems that describe the positioning of the part on the CNC-machine. Optionally, you can define the Stock model that will be used for milling operations and the Target model that has to be obtained after the machining.

Machine Setup definition

When the part is to contain Turning operations, the clamping fixture has to be defined in order to supply SolidCAM with the information about fixing the part on the CNC-machine and the part position relative to the machine.

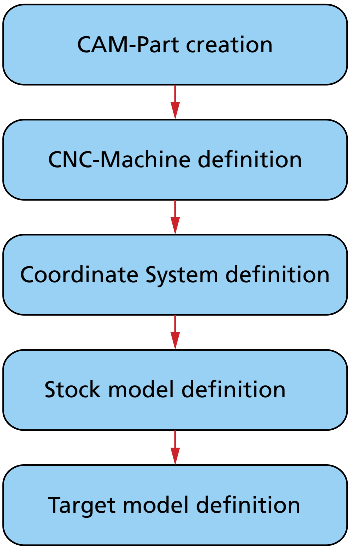
Operations definition

SolidCAM enables you to define turning and milling operations. During the operation definition, you have to select the Geometry, choose the tool from the Part Tool Table (or define a new one), define a machining strategy and a number of technological parameters.

CAM-Part Definition

2

The CAM-Part definition process consists of the following stages:



- **CAM-Part creation**. At this stage, you have to define the CAM-Part type, name and location. SolidCAM defines the necessary system files and a folder to allocate the place to store SolidCAM data.

- **CNC-Machine definition**. It is necessary to choose the CNC-controller. The controller type influences the Coordinate System definition and the Geometry definition.

- **Coordinate System definition**. You have to define the Coordinate System, which is the origin for all machining operations of the CAM-Part. You can create multiple CoordSys positions and in each machining step select which CoordSys you want to use for the operation.

- **Stock model definition**. It is necessary to define a boundary of the stock that is used for the CAM-Part machining.

- **Target model definition**. SolidCAM enables you to define the model of the part in its final stage after the machining.

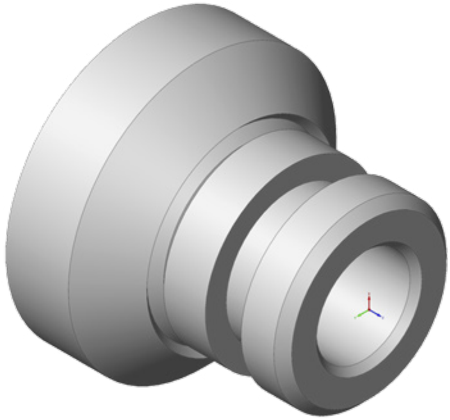
The following exercises describe the full process of the CAM-Part definition. It is recommended to go through the stages in order to understand how the CAM-Part features are built. For this purpose, you have to turn off the automatic CAM-Part definition.

Before you start, select **SolidCAM Settings** command from the **SolidCAM** menu. In the left pane, select **Automatic CAM-Part definition**. In the right pane, click the **Turning** tab and clear the following check boxes: **Create machine setup**, **Definition of Stock**, and **Definition of Target**.

These settings can be turned back on at any time.

Exercise #1: CAM-Part Definition in Turning

This exercise illustrates the process of the CAM-Part definition in SolidCAM. In this exercise, you have to create the CAM-Part for the model displayed on the illustration and define the Coordinate System, the Machine Setup, the Stock and Target model, which are necessary for the part machining. The CAM-Part will be used in the exercises further on.



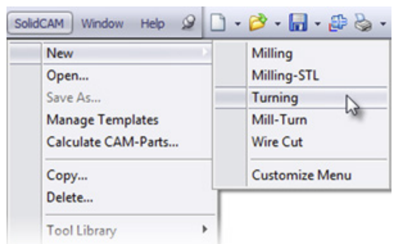
1. Load the SolidWorks model

Load the **Exercise1.sldprt** model located in the **Exercises** folder.

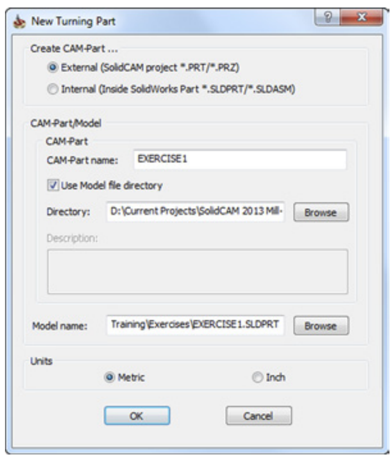
This model contains a number of features forming the solid body and several sketches used for the CAM-Part definition.

2. Start SolidCAM

To activate SolidCAM, select **SolidCAM** in the main menu of SolidWorks and choose **Turning** from the **New** submenu or click the **Turning** button on the **SolidCAM New** toolbar.



SolidCAM is started, and the **New Turning Part** dialog box is displayed.



New Turning Part dialog box

SolidCAM enables you to create a new CAM-Part using one of the following options:

- **External mode**

  In this mode, the project you create is saved in SolidCAM format (*.prt, *.prz).

- **Internal mode**

  In this mode, the project you create is saved inside SolidWorks part (*.sldprt, *.sldasm).

When you create a new CAM-Part in the External mode, you have to enter a name for the CAM-Part and for the model that contains the CAM-Part geometry.

- **Directory**

  Specify the location of the CAM-Part. The default directory is the SolidCAM user directory (defined in the **SolidCAM Settings**). You can enter the path or use the **Browse** button to define the location.

  The **Use Model file directory** option enables you to automatically create CAM-Parts in the same folder where the original CAD model is located.

- **CAM-Part name**

  Enter a name for the CAM-Part. You can give any name to identify your machining project. By default, SolidCAM uses the name of the design model.

- **Model name**

  This field shows the name and the location of the SolidWorks design model that you are using for the CAM-Part definition. The name is, by default, the name of the active SolidWorks document. With the **Browse** button, you can choose any other SolidWorks document to define the CAM-Part. In this case, the chosen SolidWorks document is loaded into SolidWorks.

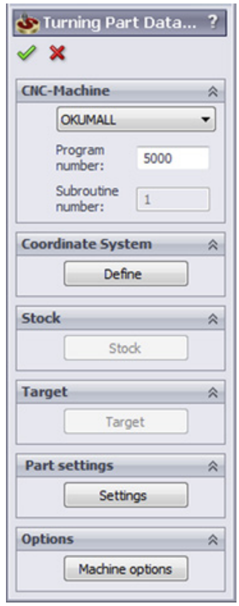
- **Units**

  This section enables you to define the measurement units to be used in the current CAM-Part.

Every time the CAM-Part is opened, SolidCAM automatically checks the correspondence of the dates of the CAM-Part and the original SolidWorks design model. When the date of the original SolidWorks model is later than the date of the CAM-Part creation, this means that the SolidWorks original model has been updated. You can then replace the SolidWorks design model on which the CAM-Part is based with the updated SolidWorks design model.

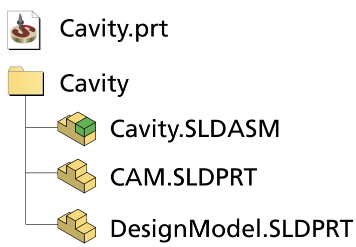
3. Confirm the CAM-Part creation

Choose the **External** mode of CAM-Part creation. After the **Directory**, **CAM-Part name** and **Model name** have been defined, click **OK** to confirm the CAM-Part creation. The CAM-Part is defined, and its structure is created. The **Turning Part Data** dialog box is displayed.



The structure of the CAM-Part

The CAM-Part includes a number of data files represented on the illustration that displays the data included in the CAM-Part named **Turning**.



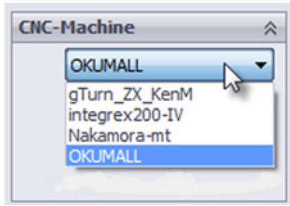
The **Turning.prt** file is located in the SolidCAM **User** directory. The **Turning** subdirectory contains all the data generated for the CAM-Part.

SolidCAM copies the original SolidWorks model to the **Turning** subdirectory and creates a SolidWorks assembly that has the same name as the CAM-Part (**Turning.sldasm**). There are two components in this assembly:

- **DesignModel.sldprt** – a copy of the SolidWorks model file.
- **CAM.sldprt** – a file that contains SolidCAM Coordinate System data and geometry data.

The SolidCAM CAM-Part uses the assembly environment of SolidWorks. This enables you to create auxiliary geometries (e.g. sketches) without making changes in the original design model. You can also insert some additional components into the assembly file such as stock model, CNC-machine table, clamping and other tooling elements.

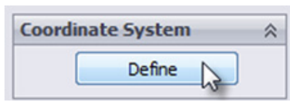
4. Choose CNC-Machine



Define the **CNC-machine controller**. Click the arrow in the **CNC-Machine** area to display the list of post-processors installed on your system.

In this exercise, use a CNC-machine with the **Okuma** CNC-controller. Choose the **OKUMALL** CNC-Machine from the list.

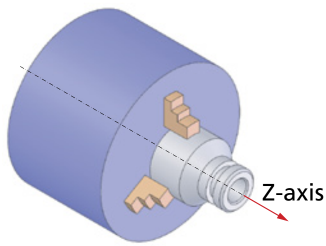
5. Define the Coordinate System



Click **Define** in the **CoordSys** area of the **Turning Part Data** dialog box to define the Machine Coordinate System.

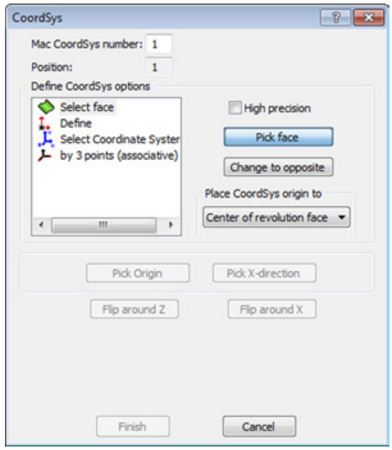
The **Machine Coordinate System** defines the origin for all machining operations on the CAM-Part. It corresponds with the built-in controller functions. It can be used for various clamping positions in various operations on the CAM-Part.

Usually Turning CNC-machines have only one machine Coordinate System; its Z-axis is the rotation axis of the spindle. The Machine Coordinate System enables you to perform all turning operations.



The **CoordSys** dialog box enables you to define the location of the Coordinate System and the orientation of the axes.

You can define the position of the Coordinate System origin and the axes orientation by selecting model faces, vertices, edges or SolidWorks Coordinate Systems. The geometry for the machining can also be defined directly on the solid model.



SolidCAM offers the following methods of **CoordSys** definition:

- **Select Face**

  This method enables you to define a new CoordSys by selecting a face. The face can be planar or cylindrical/conical. For planar faces, SolidCAM defines CoordSys with the Z-axis normal to the face. For cylindrical or conical faces, the Z-axis of the CoordSys is coincident with the axis of revolution of the specified cylindrical/conical surface.

- **Define**

  This method enables you to define the CoordSys by picking points. You have to define the origin and the directions of the X- and Y-axes.

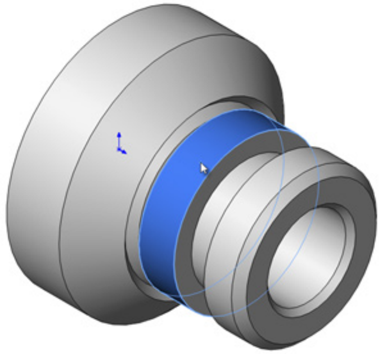
- **Select Coordinate System**

  This method enables you to choose the SolidWorks Coordinate System defined in the design model file as the CoordSys. The CoordSys origin and the orientation of the axes is the same as in the original SolidWorks Coordinate System.

- **By 3 points (associative)**

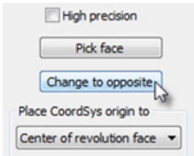
  This method enables you to define the CoordSys Origin and axes by selecting any three points.

With the **Select Face** mode chosen, click on the model face as shown. Make sure that the **Center of revolution face** option is chosen. With this option, the origin is placed automatically on the axis of revolution face.



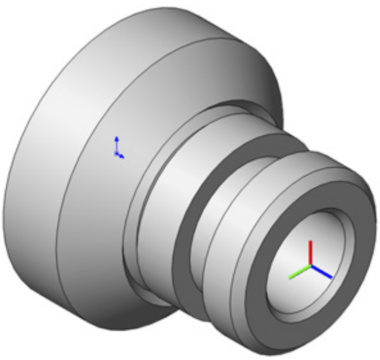
The Z-axis of the CoordSys is coincident with the axis of revolution. Note that the CoordSys origin is automatically defined on the model back face and the Z-axis is directed backwards.

Click **Change to opposite**.



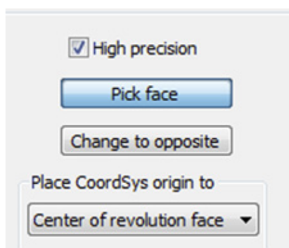
This button enables you to change the Z-axis direction to the opposite along the revolution axis.

Now the **CoordSys origin** is located on the front face of the model and the Z-axis is directed forward along the revolution axis.



High precision

When the **High precision** check box is selected, the Coordinate System is defined using the faceted model, which results in more precise definition, but may take more time to generate. When this check box is not selected, the Coordinate System is defined using CAD tools without facetting.

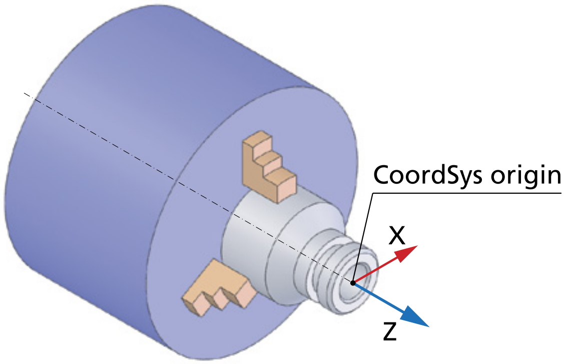


Confirm the selection by clicking **Finish**. The Coordinate System is defined.

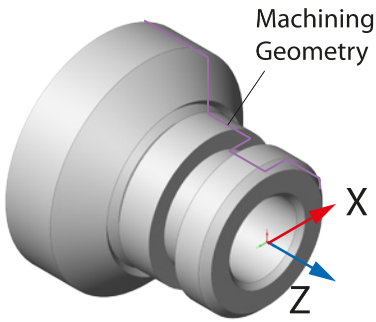
The **Turning Part Data** dialog box is displayed.

Coordinate Systems for Turning

This **Machine CoordSys** is used for the turning operations. The turning tool movements are located in the ZX-plane.



All the machining geometries are defined in the ZX-plane of the defined Coordinate System.



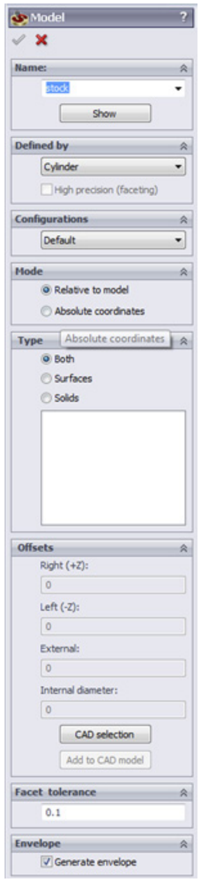
6. Define the Stock model

For each Turning project, it is necessary to define the boundaries of the stock material (**Stock**) used for the CAM-Part.

Click the **Stock** button. The **Model** dialog box is displayed.

This dialog box enables you to define the Stock model of the CAM-Part to be machined.

The following methods of Stock definition are available:



- **Cylinder**

  This option enables you to define the Stock boundary as a cylinder surrounding the selected solid model. You can define the cylindrical stock by specifying **Offsets** of the cylinder faces from the selected solid body or coordinates of its boundaries relative to the CAM-Part Coordinate System.

  The **Add to CAD model** button enables you to add a 3D sketch of the cylinder stock to the **CAM** component of the part assembly.

- **Box**

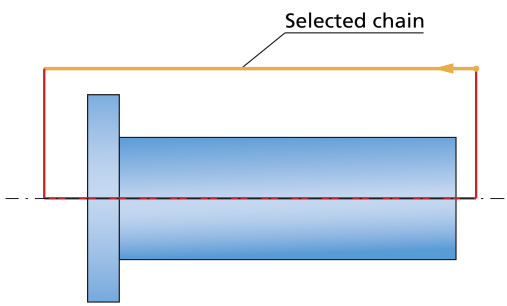
  This option enables you to define the Stock boundary as a box surrounding the selected solid model. When you click on the solid body, SolidCAM generates a 3D box around it. This box defines the geometry of the Stock.

- **Revolved boundary around Z**

  This option enables you to define the Stock boundary as a wireframe geometry chain using one of the model sketches.

  When the chain is selected, perpendiculars are dropped from its end points to the axis of rotation to define the material boundary.

The **Revolved boundary around Z** option enables you to define only one chain, either opened or closed. When more than one chain is defined, the error message is displayed.

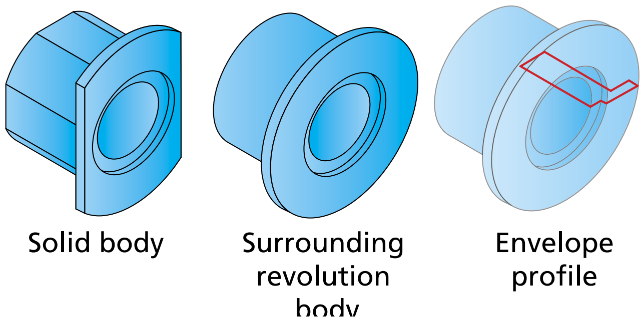


- **3D Model**

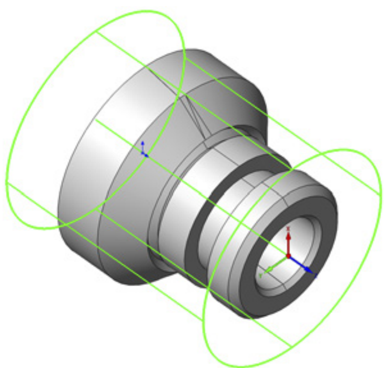
  This method enables you to define the Stock boundary by selecting the 3D Model of the stock. SolidCAM automatically generates a sketch that contains the **envelope** of the selected solid body. The Stock boundary is defined automatically on this sketch.

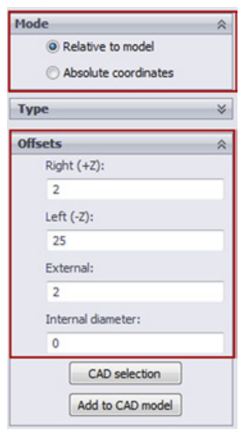
Envelope

Consider the revolution body surrounding the specified solid bodies. The section of this revolution body by the ZX-plane of the Coordinate System is the envelope. The envelope line is the profile of the part that has to be turned in order to obtain the model geometry.



Choose the **Cylinder** mode from the **Defined by** list and click on the solid body to select it.



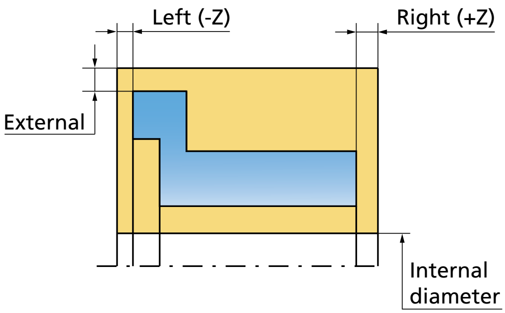


In the **Mode** section, choose **Relative to model**.

In the **Offsets** dialog section, define the following offsets:

- Set **+Z** to **2** and **-Z** to **25** to define the front and back offsets from the model
- Set the **External** offset to **2**
- Set the **Internal diameter** offset to **0**

SolidCAM defines a cylinder.



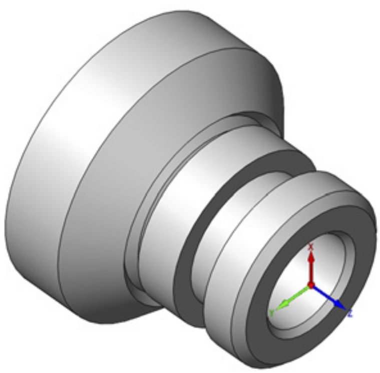
The **Mode** section enables you to define the offsets **Relative to model** or define the Stock boundaries in the **Absolute coordinates**.

When the **Internal diameter** value is different from **0**, SolidCAM defines a tube.

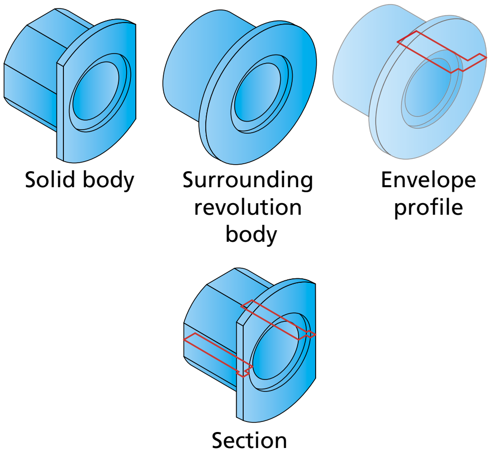
Click  to confirm the **Stock** selection.

7. Define the Target model

In the **Turning Part Data** dialog box, click the **Target** button. The **Model** dialog box is displayed. This dialog box enables you to define a 3D Model for the Target.



During the Target model definition, SolidCAM creates an **Envelope** sketch in the **CAM** component of the CAM-Part assembly. This sketch contains the geometry automatically generated by the **Envelope** function of SolidCAM. This function creates the envelope line of the specified solid bodies. Consider the revolution body surrounding the solid body. The section of this revolution body by the ZX-plane of the Turning Machine CoordSys is the envelope. This envelope is a profile of the part that has to be turned in order to create the model geometry.



The **Envelope** function takes into account all external model faces as well as the internal faces. The geometry created by the **Envelope** function can be used for the Geometry definition in SolidCAM operations.

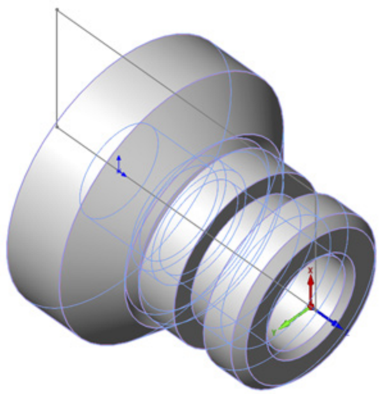
The **Mirrored Envelope** option generates the envelope sketch mirrored about the Z-axis in the ZX-plane. This option is applicable for machines with lower turret.

In addition to the Envelope, SolidCAM enables you to generate a sketch containing a **Section** of the Target model by the ZX-plane. The Section sketch is created in the **CAM** component of the SolidCAM Part Assembly.

Make sure that in the **Type** section **Both** is selected to consider both surfaces and solids for the Target model.

Click on the solid body. The wireframe model is displayed.

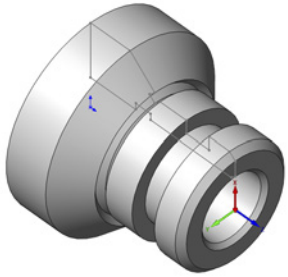
Set the **Facet tolerance** to **0.01**.



This parameter defines the accuracy of the triangulation of the stock model, target model and fixtures. The triangulated models are used later in the tool path simulation. The tighter is the tolerance, the better is the performance of the simulation.

Confirm the dialog box with .

In the process of the Target model definition, SolidCAM creates the **Envelope** sketch in the **CAM** component of the CAM-Part assembly. The **Envelope** sketch is used later for the machining geometry definition.



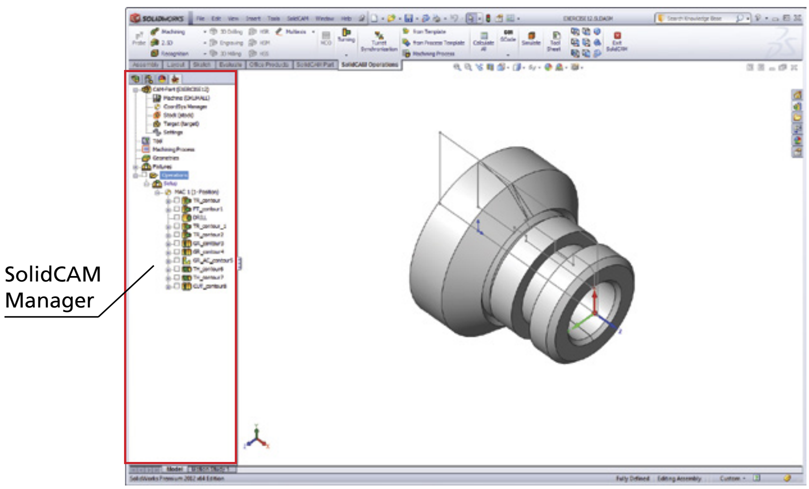
8. Save the CAM-Part data

Click in the **Turning Part Data** dialog box.

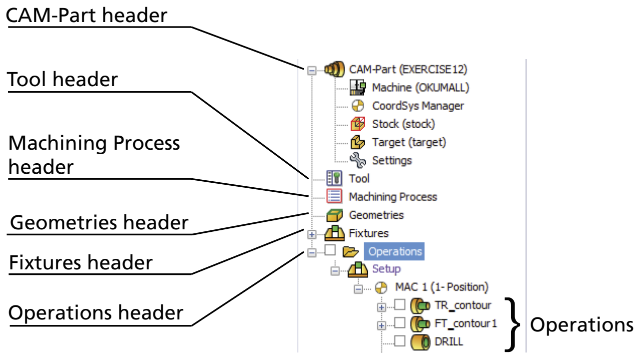
The defined CAM-Part is saved. The dialog box is closed and the **SolidCAM Manager** tree is displayed.

SolidCAM Manager

The **SolidCAM Manager** tree is the main interface feature of SolidCAM that displays complete information about the CAM-Part.



The **SolidCAM Manager** tree contains the following elements:



- **CAM-Part header**

  This header displays the name of the current CAM-Part. By right-clicking on it, you can display the menu to manage your CAM-Parts.

  The **CoordSys Manager** subheader is located under the **CAM-Part** header. Double-click this subheader to display the **CoordSys Manager** dialog box that enables you to manage your Coordinate Systems.

  The **Machine** subheader is located under the **CAM-Part** header. Double-click this subheader to open the **Machine ID Editor** dialog box that enables you to preview the machine parameters.

  The **Target** subheader is located under the **CAM-Part** header. Double-click this subheader to load the **Target model** dialog box that enables you to edit the definition of the Target model.

  The **Settings** subheader is also located under the **CAM-Part** header. Double-click this subheader to load the **Part Settings** dialog box that enables you to edit the settings defined for the current CAM-Part.

- **Tool header**

  This header displays the name of the current Tool Library. Double-click this header to display the **Part Tool Table**, which is the list of tools available to use in the current CAM-Part.

- **Machining Process header**

  This header displays the name of the current Machining Process table.

- **Geometries header**

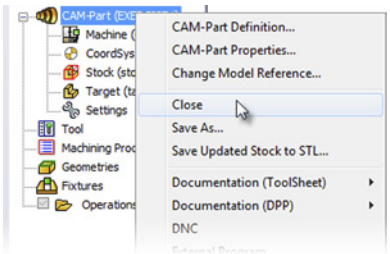
  This header displays all SolidCAM geometries that are not used in the operations.

- **Operations header**

  This header displays all SolidCAM operations defined for the current CAM-Part.

At this stage, the definition of the CAM-Part is finished. The definition of Turning operations is covered in the following exercises where this CAM-Part is used.

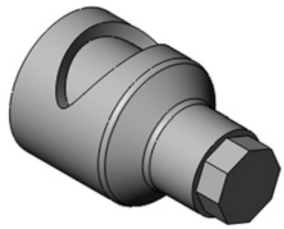
9. Close the CAM-Part



Right-click the **CAM-Part** header in **SolidCAM Manager** and choose **Close** from the menu.

The CAM-Part is closed.

Exercise #2: CAM-Part Definition in Mill-Turn



This exercise illustrates the process of the CAM-Part definition in Mill-Turn module of SolidCAM. In this exercise, you have to create the CAM-Part for the model displayed on the illustration and define the Coordinate System, the Stock model and the Target model, which are necessary for the part machining. The CAM-Part will be used in the exercises further on.

Before you start, select **SolidCAM Settings** command from the **SolidCAM** menu. In the left pane, select **Automatic CAM-Part definition**.
In the right pane, click the **Mill-Turn** tab and clear the following check boxes: **Create machine setup**, **Definition of Stock**, and **Definition of Target**.

These settings can be turned back on at any time.

1. Load the SolidWorks model

Load the **Exercise2.sldprt** model located in the **Exercises** folder.

2. Start Mill-Turn project

Click the **SolidCAM** field in the main menu of SolidWorks and choose **Mill-Turn** from the **New** submenu or click the **Mill-Turn** button on the **SolidCAM New** toolbar.

The **New Mill-Turn Part** dialog box is displayed. It is similar to the **New Turning Part** dialog box described in **Exercise #1**.

3. Confirm the CAM-Part creation

Choose the **External** mode of CAM-Part creation. After the **Directory**, **CAM-Part name** and **Model name** are defined, click **OK** to confirm the CAM-Part creation. The CAM-Part is defined, and its structure is created. The **Mill-Turn Part Data** dialog box is displayed.

4. Choose the CNC-controller

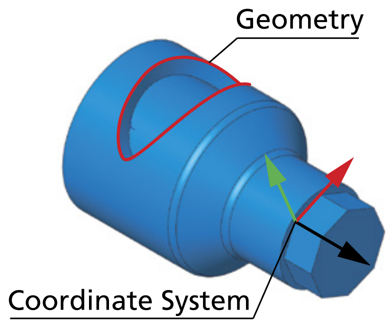
Click the arrow in the **CNC-Machine** area to display the list of post-processors installed on your system. Choose the **NTX1000** CNC-controller from the list.

5. Define the Coordinate System

Definition of the Machine Coordinate System is a mandatory step in the process of CAM-Part definition.

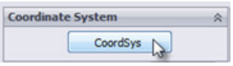
The **Machine Coordinate System** defines the origin for all machining operations on the CAM-Part. It corresponds with the built-in controller functions and can be used for various clamping positions in a variety of operations on the CAM-Part.

The Machine Coordinate System #1 (Position #1) can be used for turning operations as well as for all types of milling operations. In the Turning mode, SolidCAM uses the Turning Coordinate System created by rotating of the Machine Coordinate System #1 (Position #1) around the Z-axis.



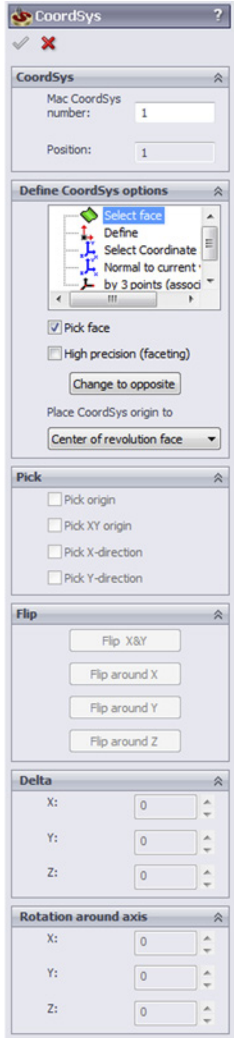
The axes orientation for the Turning Coordinate System are defined in the **CoordSys** item of the submachine in the Machine ID file of the chosen CNC-controller. At the stage of operation definition, you will need to make sure that you choose an appropriate submachine to generate the correct GCode.

Click the **Coordsys** button in the **Coordinate System** area of the **Mill-Turn Part Data** dialog box to define the Machine Coordinate System.



The **CoordSys** dialog box enables you to define the Coordinate System location and the orientation of the axes. You can define the position of the Coordinate System origin and the axes orientation by selecting model faces, vertices, edges or SolidWorks Coordinate Systems.

SolidCAM enables you to define the CoordSys using the following methods:



- **Select face**

This method enables you to define a new CoordSys by selecting a face. The face can be planar or cylindrical/conical. For planar faces, SolidCAM defines CoordSys with the Z-axis normal to the face. For cylindrical or conical faces, the Z-axis of the CoordSys is coincident with the axis of revolution of the specified cylindrical/conical surface.

When the **High precision** check box is selected, the Coordinate System is defined using the faceted model, which results in more precise definition but may take more time to generate. When this check box is not selected, the Coordinate System is defined using CAD tools without facetting.

- **Define**

This method enables you to define the CoordSys by picking points. You have to define the origin and the directions of the X- and Y-axes.

- **Select Coordinate System**

This method enables you to choose the SolidWorks Coordinate System defined in the design model file as the CoordSys. The CoordSys origin and the orientation of the axes is the same as in the original SolidWorks Coordinate System.

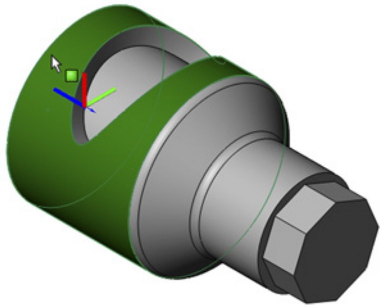
- **Normal to current view**

This option enables you to define the Coordinate System with the Z-axis normal to the model view you are facing on your screen. The CoordSys origin will lie in the origin of the SolidWorks Coordinate System, and the Z-axis will be directed normally to the chosen view of the model.

- **By 3 points (associative)**

This option enables you to define the CoordSys Origin and axes by selecting any three points.

6. Select the model face

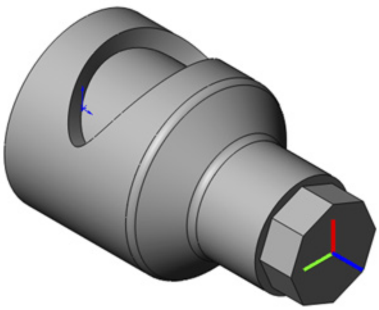


Make sure that the **Center of revolution face** option is chosen for **Place CoordSys origin to**. With this option, the origin is placed automatically on the axis of revolution face. With the **Select Face** mode chosen, select the **High precision** check box and click on the model face as shown.

Note that the CoordSys origin is automatically defined on the model back face, and the Z-axis is directed backwards. The Z-axis of the CoordSys is coincident with the axis of revolution.

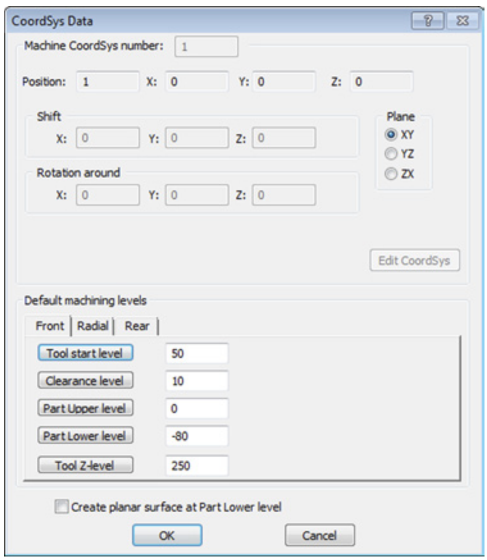
Click the **Change to opposite** button. This button enables you to reverse the Z-axis direction along the revolution axis.

Now the CoordSys origin is located on the front face of the model and the Z-axis is directed forward along the revolution axis.



Click  to confirm the selection.

The **CoordSys Data** dialog box is displayed.



7. Define the CoordSys data

The **CoordSys Data** dialog box enables you to define the Coordinate System values and machining levels such as **Tool start level**, **Clearance level**, **Part upper level**, etc.

CoordSys Data dialog box

The **Machine CoordSys number** defines the number of the CoordSys in the CNC-machine. The default value is **1**. If you use another number, the GCode file contains the G-function that prompts the machine to use the specified number stored in the controller of your machine.

The **Position** field defines the sequential number of the Coordinate System. For each Machine Coordinate System, several **Position** values can be defined for different positions; each such **Position** value is related to the Machine Coordinate System.

- **X** shows the X-value of the CoordSys.
- **Y** shows the Y-value of the CoordSys.
- **Z** shows the Z-value of the CoordSys.

The **Plane** box defines the default work plane for the operations using this CoordSys, as it is output to the GCode program. In the SolidCAM CAM module, you must always work on the XY-plane. Some CNC-machines, however, have different axes definitions and require a GCode output with rotated XY-planes.

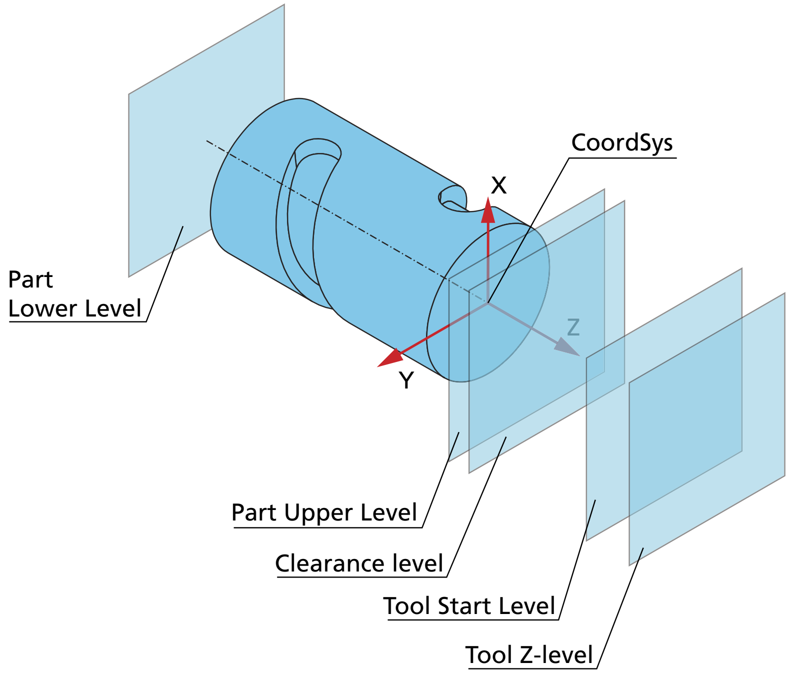
Shift is the distance from the Machine Coordinate System to the location of the Position in the coordinate system and the orientation of the Machine Coordinate System.

Rotation around is the angle of rotation around the main axes X, Y and Z.

In the Mill-Turn module, facial and radial milling is performed using the same Coordinate System. But since the part levels used for facial milling are measured along the Z-axis, whereas those used for radial milling are measured around the Z-axis, the CoordSys data must be defined separately.

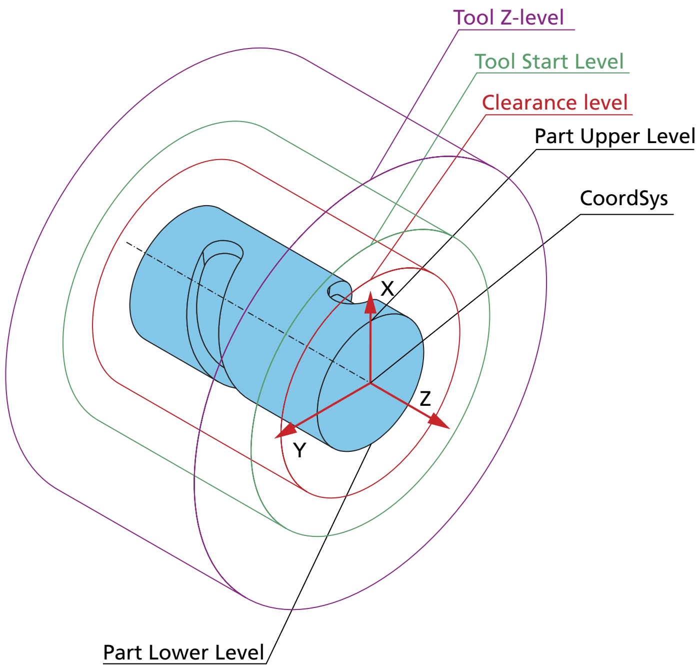
The **Front** and **Rear** tabs contain sets of facial machining levels describing the planes parallel to the XY-plane and located along the Z-axis.

The **Front** tab displays levels for milling from the positive Z-direction.



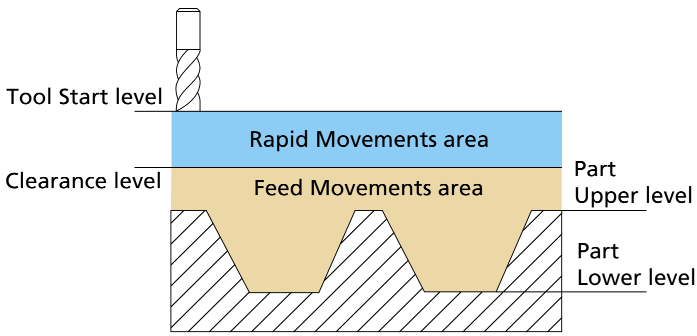
The **Rear** tab displays levels for milling from the negative Z-direction.

The **Radial** tab contains a set of machining levels describing the virtual cylinders situated around the Z-axis.



These tabs contain the following parameters:

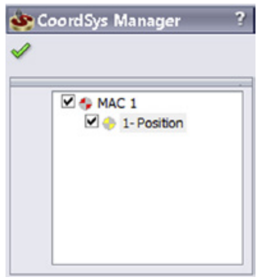
- The **Tool start level** defines the Z-level at which the tool starts working.
- The **Clearance level** is the Z-level to which the tool moves rapidly from one operation to another (in case the tool does not change).
- The **Part upper level** defines the height of the upper surface of the part to be machined.
- The **Part lower level** defines the lower surface level of the part to be machined.



- The **Tool Z-level** is the height to which the tool moves before the rotation of the 4/5 axes to avoid collision between the tool and the workpiece. This level is related to the CoordSys position and you have to check if it is not over the limit switch of the machine. It is highly recommended to send the tool to the reference point or to a point related to the reference point.

The **Create planar surface at Part Lower level** option enables you to generate a transparent planar surface at the minimal Z-level of the part so that its lower level plane is visible. This planar surface provides you the possibility to select points that do not lie on the model entities. It is suppressed by default and not visible until you unsuppress it in the **FeatureManager** Design tree.

Confirm the **CoordSys Data** dialog box with **OK**.



The **CoordSys Manager** dialog box is displayed in the **PropertyManager** area of SolidWorks. This dialog box displays one Machine Coordinate System.

Confirm the **CoordSys Manager** dialog box by clicking ✓.

The **Mill-Turn Part Data** dialog box is displayed.

8. Define the Stock model

Define the boundaries of the stock material used for the CAM-Part.

Click the **Stock** button.

The **Model** dialog box is displayed. This dialog box enables you to define the Stock model of the CAM-Part to be machined.

The following methods of Stock definition are available:

- **Cylinder**

  The Stock boundary is defined as a cylinder surrounding the selected solid model. When you click on the solid body, SolidCAM generates a cylinder around it. This cylinder defines the geometry of the Stock.

  You can define the cylindrical stock by specifying offsets of the cylinder faces from the selected solid body or coordinates of its boundaries relative to the CAM-Part Coordinate System.

- **Box**

  The Stock boundary is defined as a box surrounding the selected solid model. When you click on the solid body, SolidCAM generates a 3D box around it. This box defines the geometry of the Stock.

  You can define the box stock by specifying offsets of the box faces from the selected solid body or coordinates of its boundaries relative to the CAM-Part Coordinate System.

- **Revolved boundary around Z**

  The Stock boundary is defined as a wireframe geometry chain using one of the model sketches in the ZX-plane. When the chain is selected, perpendiculars are dropped from its end points to the axis of rotation to create the 2D geometry that will be revolved around the Z-axis to define material boundary.

- **Extruded boundary**

  The Stock boundary is defined as a closed wireframe geometry chain using one of the model sketches in the XY-plane. This chain is extruded by the Z-axis to define the material boundary.

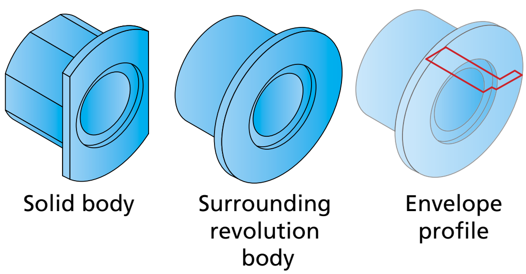
- **3D Model**

  The Stock boundary is defined by selecting a 3D model.

- **STL**

  The Stock model is defined based on a STL file that exists in your system. When you choose this mode and click the **Browse** button in the **STL file** section, the **Browse** dialog box is displayed. This dialog box enables you to choose the STL file for the stock definition.

When the **Generate envelope** check box is selected, SolidCAM generates a sketch containing the envelope of the selected solid body. The Stock boundary is defined in this sketch.

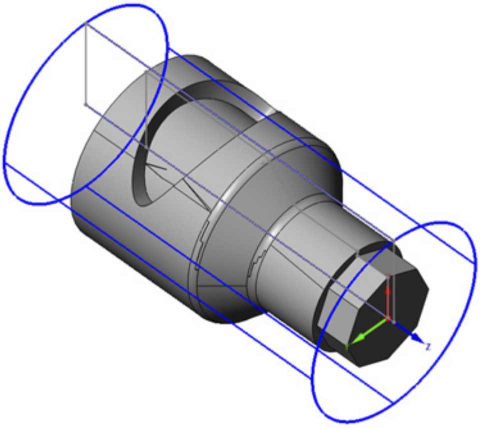


Choose the **Cylinder** mode from the **Defined by** list and click on the solid body to select it.

In the **Offsets** section, define the following offsets:

- Set **+Z** to **2** and **-Z** to **20** to define the front and back offsets from the model.
- Set the **External** offset to **2**.
- Set the **Internal diameter** offset to **0**.

SolidCAM defines a cylinder.



When the **Internal diameter** value is different from **0**, SolidCAM defines a tube.

The defined offsets are added to the cylindrical stock.

In the **Facet tolerance** section, set the value to **0.01**. This parameter defines the accuracy of triangulation of the Stock model. The triangulated model is used later in the tool path simulation. The tighter is the tolerance, the better is the performance of the simulation.

Click  to confirm the button to confirm the **Model** dialog box.

The Stock envelope sketch is added to the **CAM** component of the CAM-Part assembly and is displayed on the solid model.

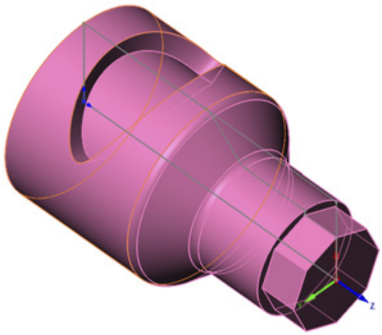
9. Define the Target model

Now you have to define the Target model. The **Target model** is the final shape of the CAM-Part after the machining. It is used for gouge checking in the **SolidVerify** simulation.

The **Target** options are similar to those used for Turning CAM-Part definition.

Click the **Target** button. The **Model** dialog box is displayed.

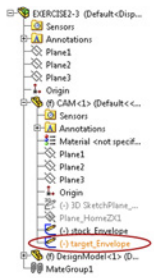
Click on the solid body. The model is highlighted.

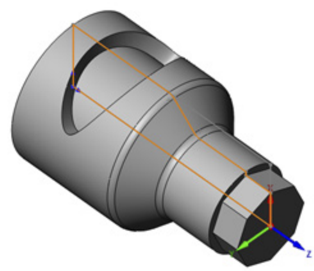


Make sure the **Envelope** option is selected in the **Generate Envelope/Section** area.

Confirm the **Target** dialog box with ✔.

The envelope sketch is added to the **CAM** component of the CAM-Part assembly and is displayed on the solid model. This sketch will be used later for the turning geometry definition.





10. Save the CAM-Part

In the **Mill-Turn Part Data** dialog box, click ✔. The dialog box is closed and the **SolidCAM Manager** tree is displayed. The defined CAM-Part is saved.

At this stage, the definition of the CAM-Part is finished. The definition of Milling and Turning operations is covered in the coming exercises using this CAM-Part.

11. Close the CAM-Part

Right-click the **CAM-Part** header in the **SolidCAM Manager** tree and choose **Close** from the menu.

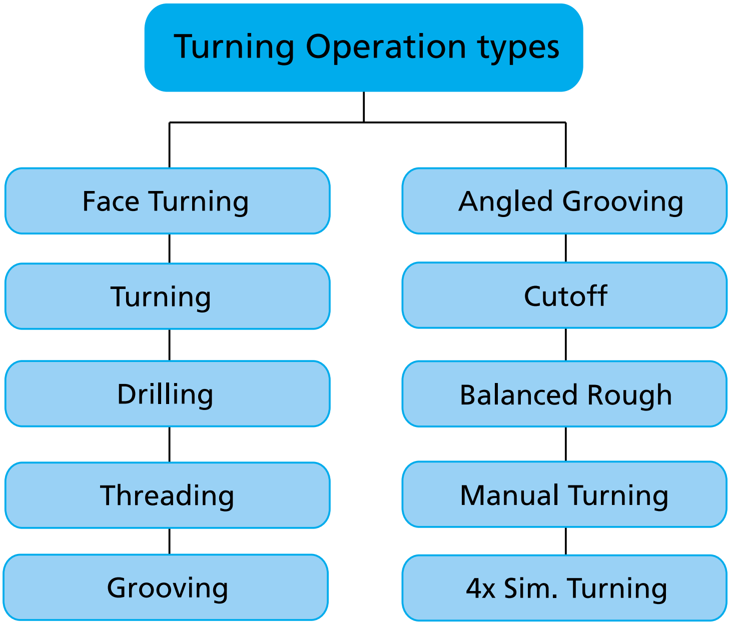
The CAM-Part is closed.

Turning Operations

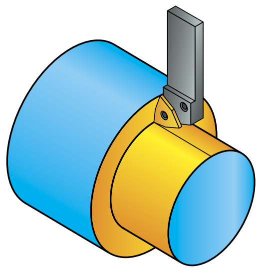
3

3.1 Basic Turning

SolidCAM enables you to perform the following types of Turning operations.

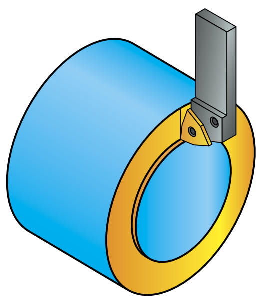


Turning Operation



This operation enables you to turn a longitudinal or facial profile. The resulting tool path can either use the turning cycles of the CNC-machine, if they exist, or it can generate all the tool movements. If the tool movements are generated by the program, then minimum tool movements length is generated taking into account the material boundary in the beginning of the particular operation. The profile geometry is adjusted automatically by the program, if needed because of the tool shape, to avoid gouging of the material.

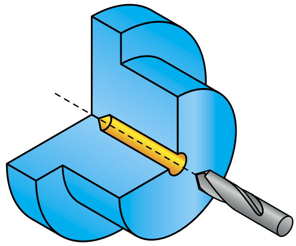
Face Turning Operation



This operation enables you to perform turning of facial profiles. The principal working direction is the X-axis direction.

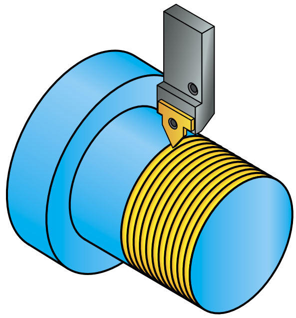
Drilling Operation

This operation enables you to perform a drilling action along the rotation axis. There is no geometry definition for this type of operation since it is enough to define the drill start and end positions.



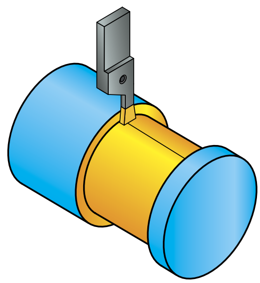
Threading Operation

This operation enables you to perform threading. The threading can be either longitudinal (internal or external) or facial. This operation can be used only if the CNC-machine has a thread cycle. SolidCAM outputs the tool path for the threading exactly with the same length as the defined geometry without any checking for material collision.



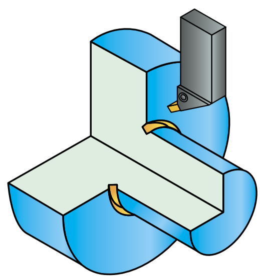
Grooving Operation

This operation enables you to perform a groove either on a longitudinal geometry (internal or external) or a facial geometry. The resulting tool path can either use a single machine cycle, generate all the tool movements (**G0**, **G1**) or generate several machine cycles.

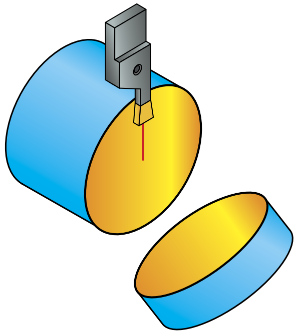


Angled Grooving Operation

This operation enables you to perform inclined grooves. The geometry defined for this operation must be inclined relative to the Z-axis of the CAM-Part Coordinate System. The **Tool angle** parameter enables you to adjust the angle of the tool cutting the material.

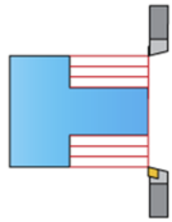


Cutoff Operation



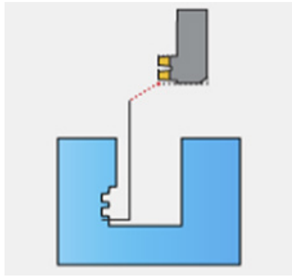
This operation enables you to perform cutoff machining. This operation is used to cut the part or to perform a groove whose width is exactly the same as the tool width. The cutting can be performed using CNC-machine cycles; chamfers and fillets can also be generated.

Balanced Rough Operation



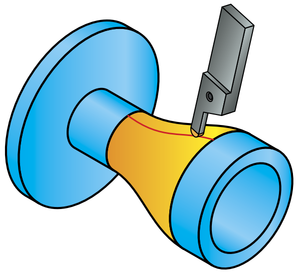
This operation enables you to work with two tools performing roughing cuts at the same time. The Master submachine and Slave submachine should include the same Table.

Manual Turning Operation



This operation enables you to perform turning according to your own geometry regardless of a stock model, target model, or envelope. The **Reverse cutting path** option enables you to machine undercuts effectively.

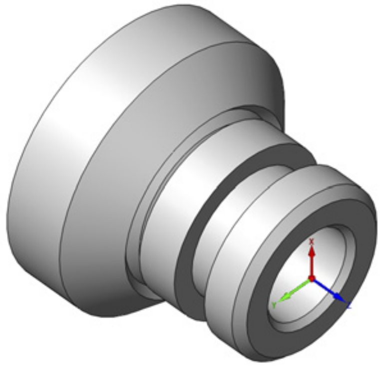
Simultaneous Turning Operation



This operation enables you to perform machining of curve-shaped tool paths using tilting capabilities of tools with round inserts. The tool tilting is defined by specifying lines that indicate the tool vector change. This operation is useful for machining of undercut areas in a single machining step.

For more detailed explanation on the Turning operations, refer to the **SolidCAM Turning Online Help**.

Exercise #3: Turning Operations on Turning CNC-Machine

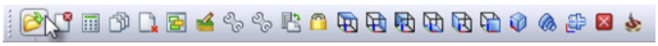


In this exercise, you have to define the machine setup and clamping fixture and to perform a number of Turning operations to conclude the machining of the CAM-Part.

The exercise uses the CAM-Part created in **Exercise #1**. First, you have to define the machine setup and clamping fixture that holds the machined workpiece on the CNC-machine table.

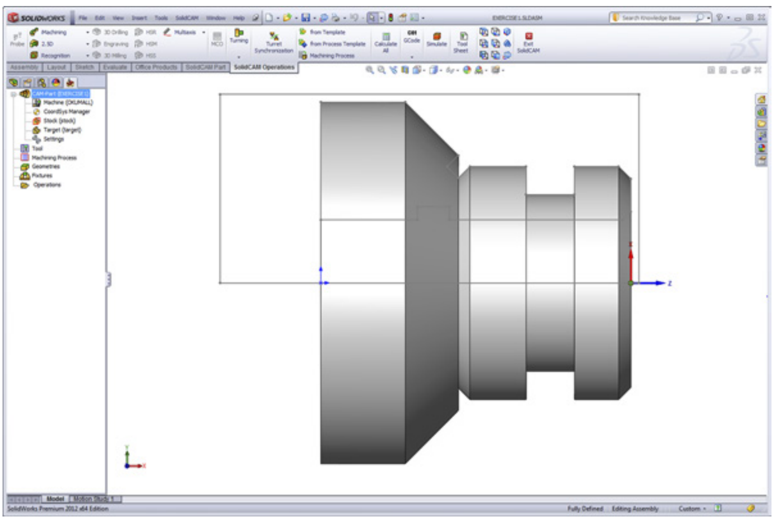
1. Load the CAM-Part

Click **SolidCAM**, **Open**, or click the **Open** button on the **SolidCAM** toolbar.



In the browser window, choose **Exercise1.prz** to load the CAM-Part that has been prepared earlier.

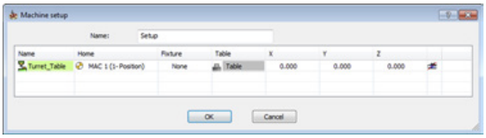
The CAM-Part is loaded.



2. Define the machine setup

Right-click the **Operations** header in **SolidCAM Manager** and choose the **Machine Setup** ( ) command.

The **Machine setup** dialog box is displayed. This dialog box enables you to define the relations between fixtures, submachines, and the Machine Coordinate System.



Machine Setup

Machine Setup definition is an optional step in definition of CAM-Parts that contain Turning operations. Using the Setup feature, you can define the fixtures and their location relative to the Coordinate Systems associated with certain submachines.

This unified representation of all setup data allows you to get a more realistic picture during simulation and check possible collisions between the cutting tools and fixtures.

This machine setup contains a submachine called **Turret_Table** with the Machine Coordinate System **MAC 1**. These parameters are defined automatically. You have to define a fixture that helps prevent a possible collision of tools during the machining.

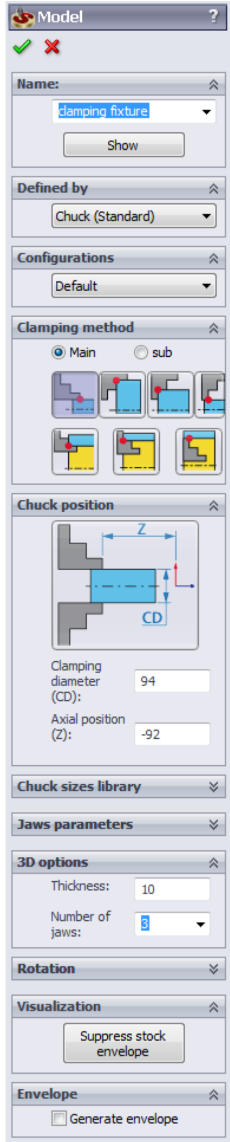
The **Fixture** column enables you to choose or define the fixture to be used with the specified submachine. To define the fixture, double-click the cell, click the arrow and choose ... .

The **Model** dialog box is displayed. This dialog box enables you to define the fixture geometry.

Make sure that the **Chuck (Standard)** option is chosen in the **Defined by** section. This option enables you to define a standard three-step chuck by specifying the clamping method, chuck position and dimensions.

Clamping method

This section enables you to define how the clamping device will be attached to the workpiece. The **Main**/**Sub** options enable you to choose the location and orientation of the current fixture.



In the **Clamping method** section, use the default **Main** option and make sure that the default clamping option is chosen.

Chuck position

The chuck positioning is defined with the **Clamping diameter** (**CD**) and **Axial position** (**Z**) parameters relative to the stock end face:

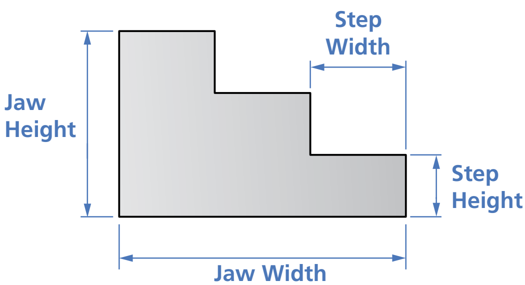
The **Clamping diameter** (**CD**) and **Axial position** (**Z**) parameters can be defined by picking on the model. When the model is picked, SolidCAM measures the X- and Z- distances from the CoordSys origin to the picked positions and displays the values in the corresponding edit boxes.

The default Chuck position is defined on the edge of the stock envelope created during the Stock model definition. The **Clamping diameter** is **94**, and the **Axial position** is **-92**.

Now you need to define the dimensions of the chuck.

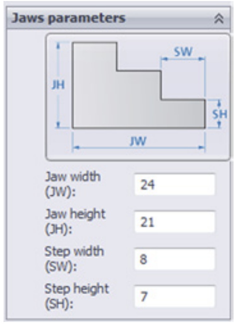
Chuck parameters

- The **Jaw width (JW)** parameter defines the overall width of a single jaw.
- The **Jaw height (JH)** parameter defines the overall height of a single jaw.
- The **Step width (SW)** parameter defines the width of the lower step.
- The **Step height (SH)** parameter defines the height of the lower step.



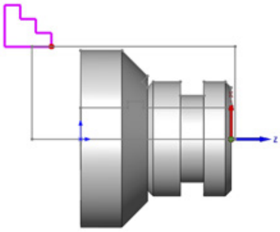
Associativity is not maintained for chucks defined by parameters.

Set the values in the **Jaws parameters** section as follows:



- Set the **Jaw width (JW)** value to **24**.
- Set the **Jaw height (JH)** value to **21**.
- Set the **Step width (SW)** value to **8**.
- Set the **Step height (SH)** value to **7**.

The clamping fixture is defined.

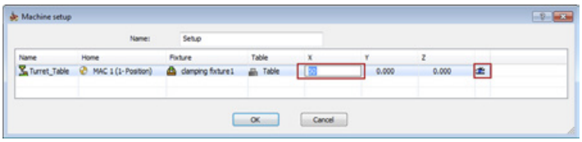


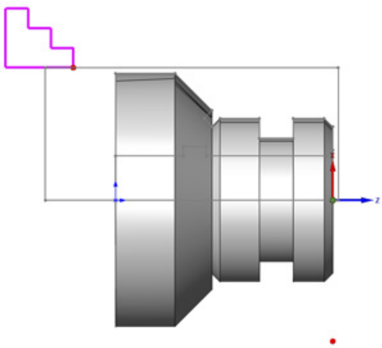
Click the button to confirm the **Model** dialog box.

The **Machine setup** dialog box is displayed again.

Now you have to define the model position relative to the submachine coordinate system.

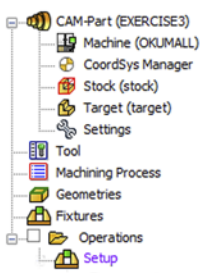
Double-click the cell in the **X** column. Set the value to **50**.





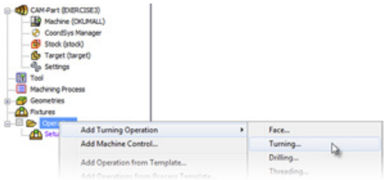
Click the **Preview** icon to see the changes. The model is located now in 50 mm above the table allowing better visualization during the machine simulation mode.

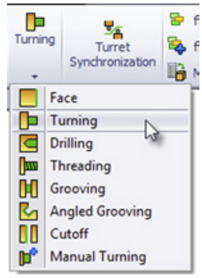
Confirm the Setup definition with **OK**. The **Setup** subheader is added to the **SolidCAM Manager** tree under the **Operations** header.



3. Add a Turning Operation

Right-click the **Operations** header in the **SolidCAM Manager** tree and choose **Turning** from the **Add Turning Operation** submenu.

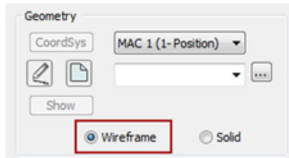




You can also select the **Turning** command from the **Turning** menu on the **SolidCAM Operations** ribbon.
The **Turning Operation** dialog box is displayed.

4. Define the Geometry

Make sure the **Wireframe** option is selected the **Geometry** section.



Turning geometry can be defined by selecting wireframe elements or by picking solid model entities such as faces, edges and vertices.

The following geometry definition options are available:

- **Wireframe**

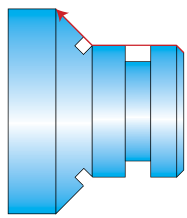
  This option enables you to define the turning geometry by wireframe geometry selection.

- **Solid**

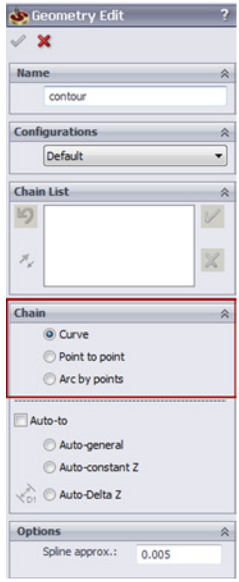
  This option enables you to define the turning geometry by selecting model entities such as faces, edges, vertices, origin and sketch points.

When model entities are picked, SolidCAM automatically defines the geometry on the envelope/section segment corresponding to the selected model elements.

You have to define the machining geometry for the external roughing operation using the **Envelope** sketch. The sketch was automatically generated in the Target model definition process described in **Exercise #1**.



Click the **New** icon ( ). The **Geometry Edit** dialog box is displayed in the SolidWorks **PropertyManager** area. This dialog box enables you to define and edit geometry chains.



SolidCAM enables you to choose the mode of the geometry selection in the **Chain** section of the **Geometry Edit** dialog box.

Chain options

You can define the geometry by selecting edges, sketch segments and points on the contour. The following options are available:

- **Curve**

  You can create a chain of existing curves and edges by selecting them one after the other.

  Associativity: SolidCAM keeps the associativity to any edge or sketch entity. Any change made to the model or sketch automatically updates the selected geometry.

- **Point to point**

  This option enables you to connect specified points; the points are connected by a straight line.

  Associativity: SolidCAM does not keep the associativity to any selected point. SolidCAM saves the X-, Y- and Z-coordinates of the selected points. Any change made to the model or sketch does not update the selected geometry.

- **Arc by points**

  This option enables you to create a chain segment on an arc up to a specific point on the arc.

  Associativity: SolidCAM does not keep the associativity to any selected arcs by points. SolidCAM saves the X-, Y- and Z-coordinates of the selected points. Any change made to the model or sketch does not update the selected geometry.

Auto Select options

SolidCAM automatically determines the chain entities and closes the chain contour. The **Auto select** mode offers the following options:

- **Auto-to**

  The chain is selected by specifying the start curve, the direction of the chain and the element up to which the chain is created. SolidCAM enables you to choose any model edge, vertex or sketch entity to determine the chain end.

  When the end item is chosen, SolidCAM determines the chain according to the chosen selection mode (**Auto-general**, **Auto-constant Z** or **Auto-Delta Z**). The chain selection is terminated when the selected end item is reached.

  If the chosen end item cannot be reached by the chain flow, the chain definition is terminated when the start chain segment is reached. The chain is automatically closed.

- **Auto-general**

  SolidCAM highlights all entities that are connected to the last chain entity. You have to select the entity along which you want the chain to continue.

- **Auto-constant Z**

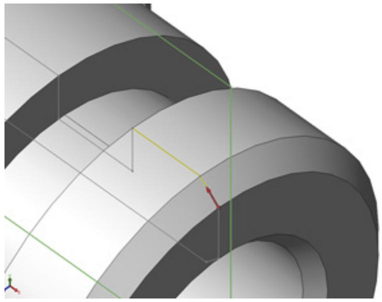
  This option identifies only the entities on the same XY-plane with the previously selected chain entity. You are prompted to identify the next chain element when two entities on the same Z-level are connected to the chain. The system tolerance for this option can be set in the **SolidCAM Settings**.

- **Auto-Delta Z**

  When you select this option, you are required to enter a positive and negative Z-deviation into the **Delta-Z** dialog box. Only entities inside this range are identified as the next possible entity of the chain.

Make sure that the default **Curve** mode is chosen.

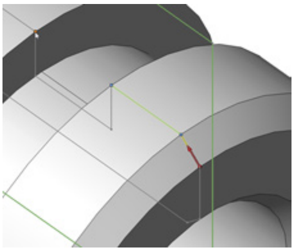
Select the sketch segments as shown.



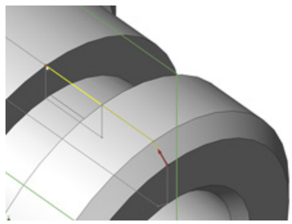
The order of the geometry selection is important, since it defines the direction of machining. Operations in SolidCAM use the direction of the chain geometry to calculate the tool path. The arrow at the start point of the chain indicates the direction of the chain.

In the **Chain** section of the **Geometry Edit** dialog box, choose the **Point to point** option. This option enables you to connect the specified points with a straight line.

Click on the sketch point as shown.

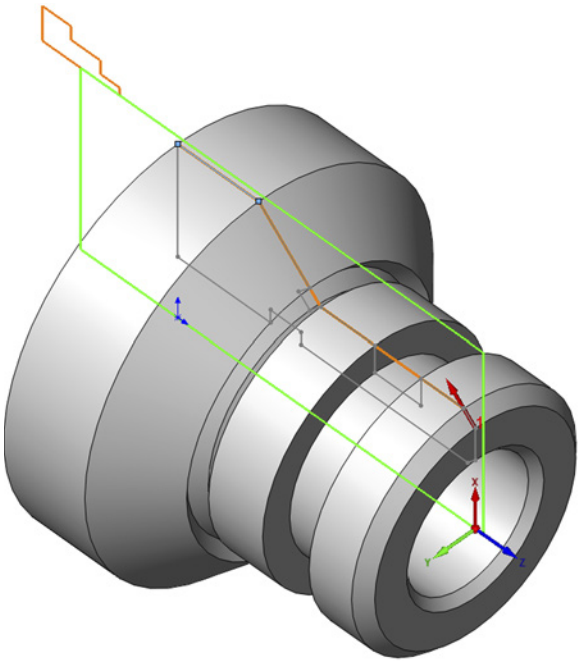


The linear geometry segment is defined.



Switch back to the **Curve** mode and pick the rest of the sketch entities.

The geometry chain is defined for the external roughing operation. Confirm the chain definition with the **Accept chain** button.



Reverse

This button enables you to reverse the chain direction.

Undo step

This button enables you to undo the last selection of a chain element.

Reject chain

This button cancels the single chain selection.

The chain icon is displayed in the **Chain List** section.

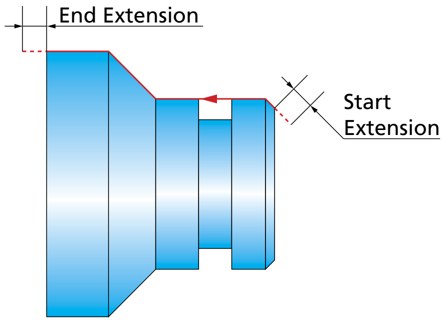
Close the **Geometry Edit** dialog box with and return to the **Turning Operation** dialog box.

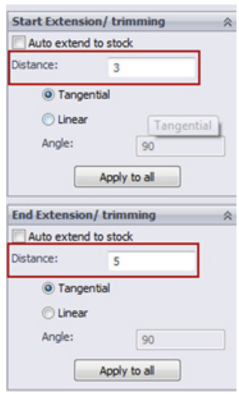
In the **Edit Geometry** section, click the **Modify Geometry** button.

The **Modify Geometry** dialog box is displayed. It enables you to modify geometries defined for SolidCAM operations: extend/trim and assign offsets, and also choose the geometry chains to be used in the operation (in case of multiple chain geometry).

The **Start Extension/trimming** and **End Extension/trimming** sections enable you to define the length of the extension/trimming applied to the start/end of the geometry chain.

The **Distance** option enables you to define the extension/trimming distance: when a positive value is defined, the chain is extended from the start/end point with straight lines of the specified length; when a negative value is defined, the chain is trimmed from the start/end point up to the specified distance measured along the geometry chains.





In the **Start Extension/trimming** section, clear the **Auto extend to stock** option and set **3** as the **Distance** value to extend the geometry outside the material. In the **End Extension/trimming** section, set the **Distance** value to **5**.

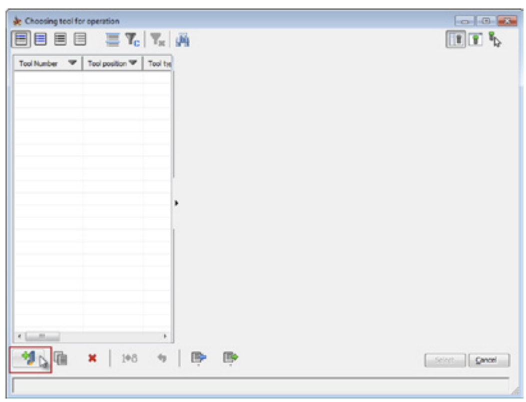
Click the **Apply to all** button to confirm the chain selection.

Close the **Geometry Edit** dialog box with . The **Turning Operation** dialog box is displayed again.

5. Define the Tool

After the geometry definition, you have to define the tool for the operation. Switch to the **Tool** page of the **Turning Operation** dialog box and click the **Select** button.

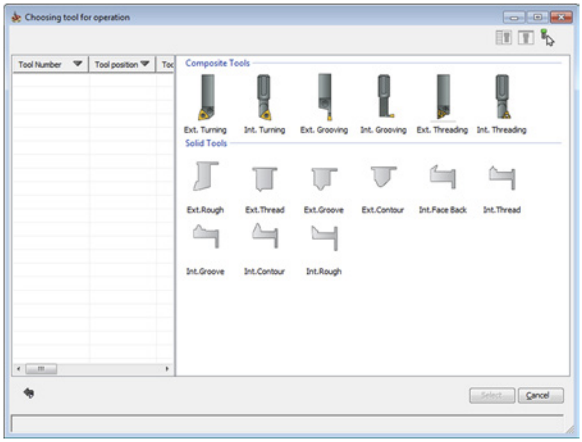
The **Part Tool Table** is displayed.



The **Part Tool Table** is a tool library that contains all the tools available for use with a specific CAM-Part. The **Part Tool Table** is stored within the CAM-Part.

Click the **Add Turning Tool** icon to start the definition of a new tool.

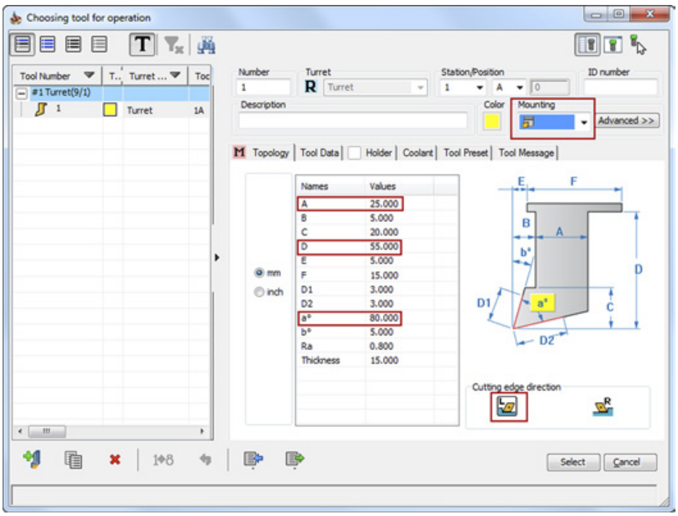
Available composite and solid tools are now displayed in the right pane of the dialog box.



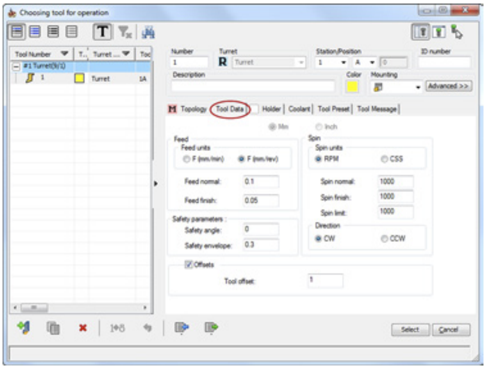
From the **Solid Tools** section, choose the **Ext. Rough** tool.

- Set the tool holder width (**A**) value to **25**.
- Set the tool height (**D**) value to **55**.
- Set the tool tip angle (**a°**) value to **80**.

Set **Cutting edge direction** to **Left** ( ) and choose the **Mounting** type as shown ( ).



Switch to the **Tool Data** tab.



This page enables you to define the general technological parameters. These parameters are associated with the current tool and applied to every operation where this tool is used.

Spin normal

This field defines the **Spin** value for Normal turning.

Spin finish

This field defines the **Spin** value for Finish turning.

Generally, the **Spin** value can be calculated using the following formula:

Spin=(1000*V)/(π*D), where **V** is the cutting speed and **D** is the diameter.

In this exercise, it is recommended to use the cutting speed of **210 m/min**.

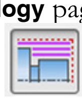
The diameter used for spin calculation is **90** mm (maximal diameter of the part).

According to the formula above, **Spin≈750**.

Choose **CSS** as **Spin Units**. Define the **Spin** parameters. Set the **Spin normal** and **Spin finish** values to **750**.

Click the **Select** button to choose the tool for the operation. At this stage, the tool is defined in the **Part Tool Table**.

6. Define the technological parameters

Switch to the **Technology** page of the **Turning Operation** dialog box. Make sure that the default **Long external**  option is chosen in the **Mode** area.

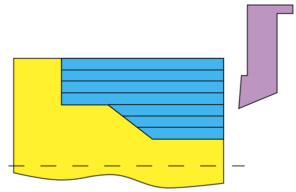
This option enables you to execute **external longitudinal turning** (the principal working direction is the Z-axis direction).

In the **Work type** area, use the default **Rough** option.

Work type

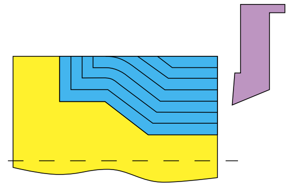
This option enables you to choose the method of the machining:

- **Rough**

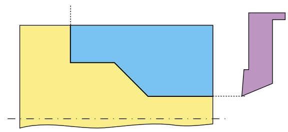
  The tool path movements are parallel to the Z-axis (longitudinal turning) or to the X-axis (facial turning). Semi-finish and finish passes are performed, if chosen, at the end of the rough stage.

- **Copy**

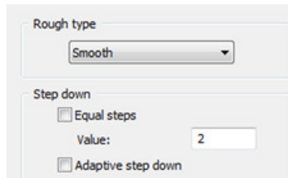
  

  The finish pass is performed, if chosen, at the end of the copy stage.

- **Finish only**

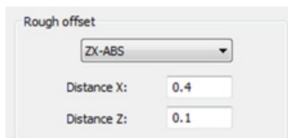
  

  This option is used for the final turning of the CAM-Part. When this option is chosen, only the semi-finish or finish pass is executed.



In the **Rough** tab, choose **Smooth** from the **Rough type** drop-down list.

Set the **Step down** value to **2**.



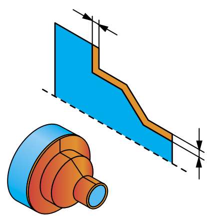
In the **Rough offset** section, choose the **ZX-ABS** option.

Set the **Distance X** value to **0.4** and the **Distance Z** value to **0.1**.

Rough offset

This drop-down list has three options:

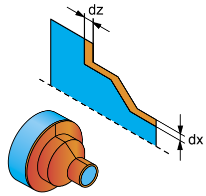
- **Distance**

  This option defines the constant offset distance from the geometry. You are prompted to enter the **Distance** value.

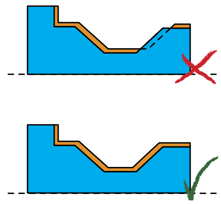
- **ZX**

  SolidCAM enables you to define different offsets in the X and Z directions. You are prompted to enter both **Distance X** and **Distance Z**.



- **ZX-ABS**

  This option is similar to the **ZX** option, except that the program chooses the sign of each vector component (**dx**, **dz**) so that the offset geometry does not intersect with the profile geometry.

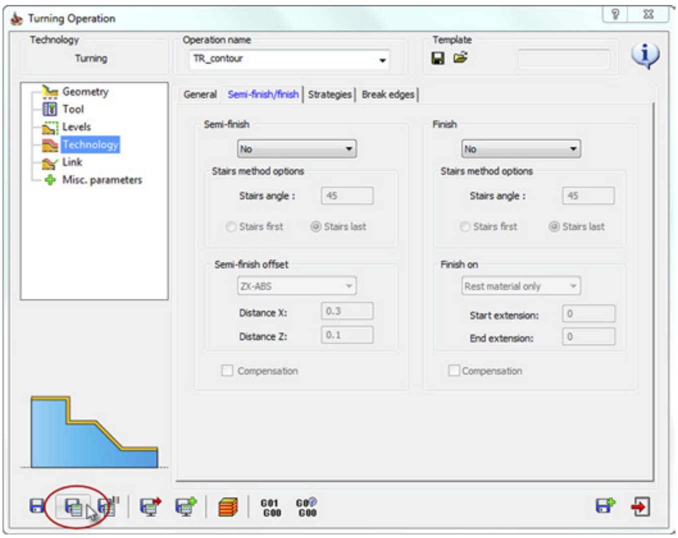


In the **Semi-finish/finish** tab, select the **No** option in the **Semi-finish** and **Finish** sections. The finishing operation will be performed later.

7. Calculate the tool path

At this stage, all of the operation parameters are defined.

Click the **Save & Calculate** icon in the **Turning Operation** dialog box to save the operation data and calculate the tool path.

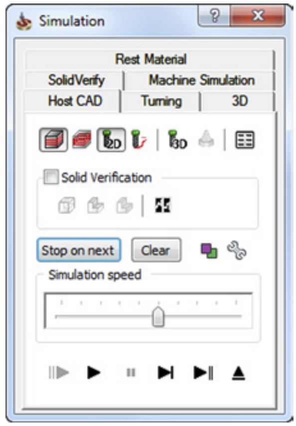


8. Simulate the tool path

Use the simulation to check and view the generated tool path after you have defined and calculated your machining operations.

If you have made mistakes in the definition of operations or used unsuitable turning strategies, the simulation helps you avoid problems that you would otherwise experience during the actual production running.

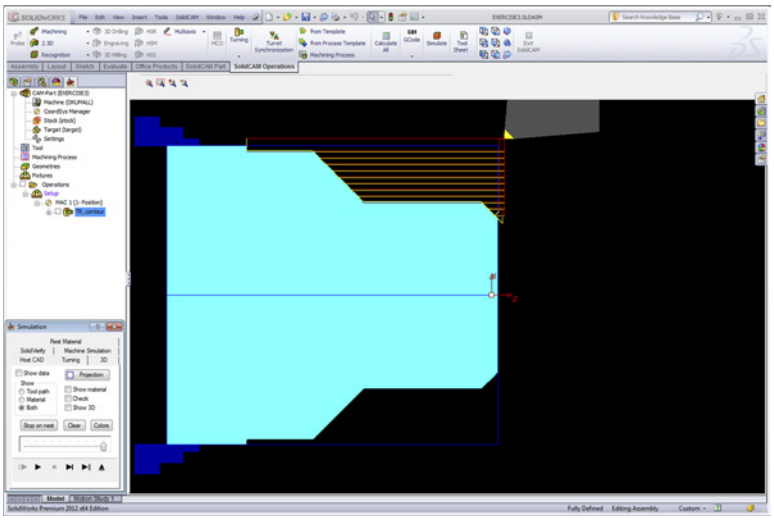
Click the **Simulate** icon in the **Turning Operation** dialog box. The **Simulation** control panel is displayed.



Switch to the **Turning** page. This simulation mode enables you to display the 2D simulation of the turning tool path.

In the **Show** section, choose the **Both** option. This option displays both the tool path and the material.

Click the **Play** button. The simulation is displayed.



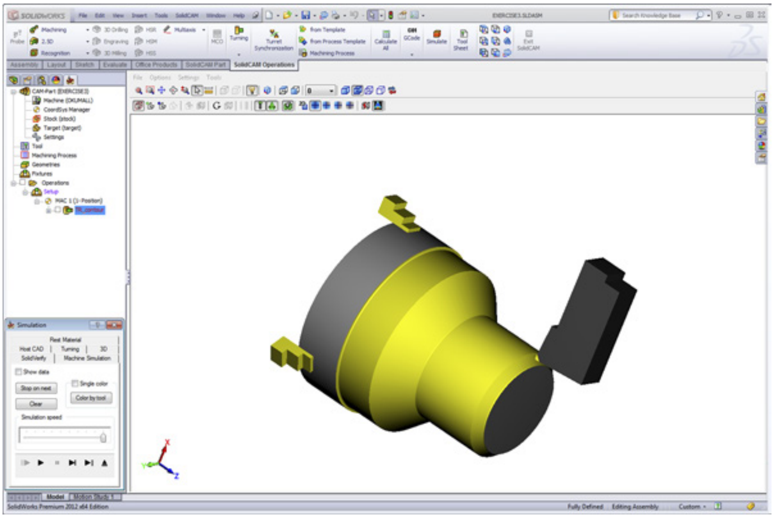
When the simulation is finished, switch to the **SolidVerify** page of the **Simulation** control panel. This simulation mode enables you to view the tool path on the 3D Model.

Rotate the model with the mouse wheel.

Click the **Play** button to start the simulation.

SolidVerify simulation mode

This mode enables the simulation of machining on the solid model. The solid stock model defined as **Stock** is used in this mode. During the machining simulation, SolidCAM subtracts the tool movements from the solid model of the stock using solid Boolean operations. The remaining machined stock is a solid model that can be dynamically zoomed or rotated.



When the simulation is finished, click the **Exit** button. The **Turning Operation** dialog box is displayed. Click the **Exit** icon to close the dialog box.

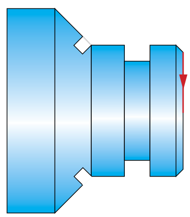
9. Add a Face Turning operation

Right-click the Turning operation defined in the previous step and choose **Face** from the **Add Turning Operation** submenu to add a new facing operation.

The **Face Turning Operation** dialog box is displayed.

10. Define the Geometry

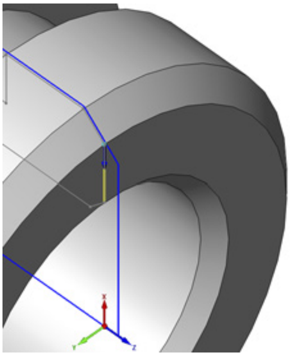
Define the geometry as shown on the illustration to machine the end face.



Click (**New**) in the **Geometry** page.

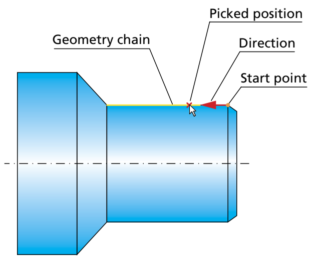
The **Geometry Edit** dialog box is displayed.

Click on the sketch segment as shown.



In this operation, the geometry must be directed to the rotation axis of the model.

Note the direction of the geometry. When you pick the first chain entity on the solid model, SolidCAM determines the start point of the picked entity closest to the picked position. The direction of the picked first chain entity is defined automatically from the start point to the picked position.



Accept the chain and get back to the **Face Turning Operation** dialog box.

Click the **Modify Geometry** button. In the **Start Extension/trimming** section, set **4** as the **Distance** value to extend the geometry outside the material. In the **End extension/trimming** section, set the **Distance** value to **20**.

Confirm the geometry modification with the button.

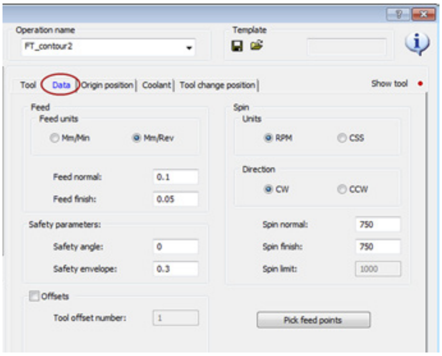
11. Define the Tool

Use the tool defined in the previous operation. Click the **Select** button in the **Tool** page to choose the tool from the **Part Tool Table**.

The **Part Tool Table** is displayed.

Choose the **Tool #1** and click **Select**. The tool is chosen for the operation.

Click the **Data** tab in the **Tool** page of the **Turning Operation** dialog box to customize the tool parameters such as **Spin** and **Feed** for the operation.



The **Spin** value can be calculated using the following formula:

Spin=(1000*V)/(π*D), where **V** is the cutting speed and **D** is the diameter.

In this exercise, it is recommended to use the cutting speed of **200 m/min**.

The diameter used for spin calculation is **58** mm (maximal diameter of the end face).

According to the formula above, **Spin≈1100**.

Set the **Spin Normal** and the **Spin Finish** to **1100**.

Set the **Feed Normal** to **0.2**.

The tool is defined for the operation.

12. Define the technological parameters

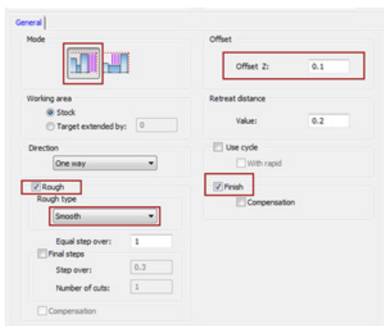
Switch to the **Technology** page of the **Face Turning Operation** dialog box.

Make sure that the **Front** option is chosen in the **Mode** area. This option enables you to machine the front end face.

SolidCAM enables you to perform roughing and finishing in a single operation.

Make sure that the **Rough** option is selected. In the **Rough type** section, select the **Smooth** option. In the **Offset** section, set **Offset Z** value to **0.1**.

Select the **Finish** option to execute a finishing pass in the direction of the geometry.



13. Save and Calculate

At this stage, all the operation parameters are defined.

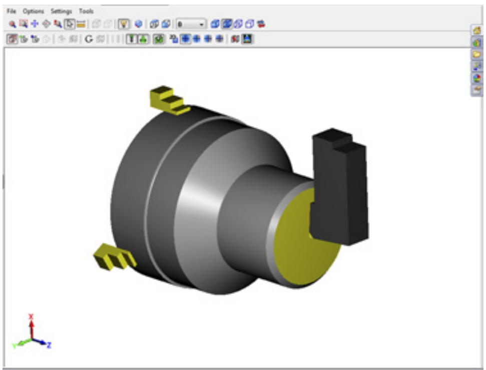
Click the **Save & Calculate** icon in the **Face Turning Operation** dialog box to save the operation data and calculate the tool path.

14. Simulate

Click the **Simulate** icon in the **Face Turning Operation** dialog box. The **Simulation** control panel is displayed.

Simulate the tool path in the **Turning** mode.

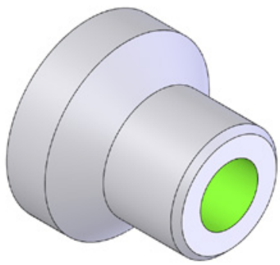
When the simulation is finished, switch to the **SolidVerify** mode and simulate the tool path again.



Close the **Simulation** control panel with the **Exit** button.

The **Face Turning Operation** dialog box is displayed. Close it with the **Exit** button.

In the following operation, you have to perform rough machining of the internal faces using a Drilling operation. The U-Drill tool is used to machine the internal hole without center drilling and to increase the machining speed by avoiding the pecking. Using the U-Drill, you can also avoid the preliminary drilling and operate the tool with the largest diameter.



15. Add a Drilling operation

Right-click the Face Turning operation and choose **Drilling** from the **Add Turning Operation** submenu to add a new Drilling operation.

The **Drilling Operation** dialog box is displayed.

16. Define the Tool

Click the **Select** button in the **Tool** page to start the tool definition.

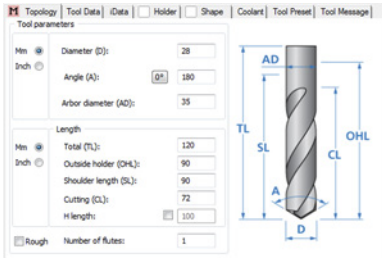
The **Part Tool Table** is displayed.

Click  to start the definition of a new tool.

Select a standard drill for this operation.

Set the following parameters of the U-Drill tool that will be used for the operation:

- Set the **D** parameter to **28**
- Set the **A** parameter to **180**
- Set the **AD** parameter to **35**
- Set the **OHL** parameter to **90**
- Set the **SL** parameter to **90**
- Set the **CL** parameter to **72**



Switch to the **Tool Data** page to define the **Spin** and **Feed** parameters of drilling.

The Cutting speed recommended for the drilling operation is **80 m/min**. According to the following formula,

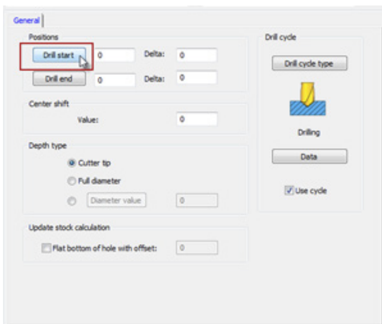
Spin=(1000*V)/(π*D), where **V** is the cutting speed and **D** is the diameter (**28** mm),

Spin≈900.

Set the **Spin rate** and **Spin finish** values to **900**.

Set the **Feed XY** value to **0.08** mm/tooth.

Confirm the tool definition with the **Select** button. The **Drilling Operation** dialog box is displayed.

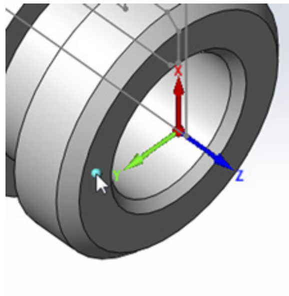


17. Define the Drill start position

The geometry for drilling is defined by two points: start and end positions. Switch to the **Technology** page of the **Drilling Operation** dialog box. Click the **Drill start** button in the **Positions** area.

The **Pick Start point** dialog box is displayed.

Click on the front face of the part as shown.



The coordinates of the selected point are displayed in the **Pick Start point** dialog box. Confirm this dialog box with the button.

The **Drilling Operation** dialog box is displayed again.

18. Define the Drill end position

Define the end position of the drilling.

Click the **Drill end** button in the **Positions** area.

The **Pick End point** dialog box is displayed.

Click on the back face of the part.

The coordinates of the selected point are displayed in the **Pick End point** dialog box. Confirm the dialog box with the button.

The **Drilling Operation** dialog box is displayed again.

In the **Depth type** section, select the **Full diameter** option.

19. Save and Calculate

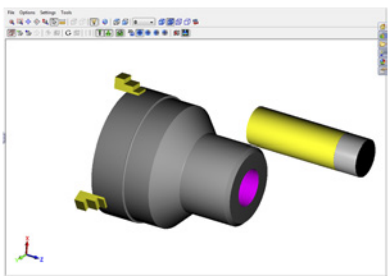
At this stage, all the parameters of the operation are defined.

Click the **Save & Calculate** icon in the **Drilling Operation** dialog box to save the operation data and calculate the tool path.

20. Simulate

Click the **Simulate** icon in the **Drilling Operation** dialog box. Simulate the tool path in the **Turning** mode.

Simulate the tool path in the **SolidVerify** mode.



Close the **Simulation** control panel with the **Exit** button. Close the **Drilling Operation** dialog box with the **Exit** icon.

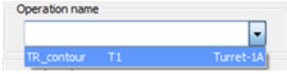
21. Add an External Finishing operation

Right-click the last defined Drilling operation in **SolidCAM Manager** and choose **Turning** from the **Add Turning Operation** submenu.

The **Turning Operation** dialog box is displayed.

22. Copy data from the existing operation

SolidCAM enables you to use an already existing operation as a template for the current operation. All the operation data is copied from the template to the current operation. This feature enables you to save the programming time.



In the **Operation name** section, choose the first operation (**TR_contour**) as a template.

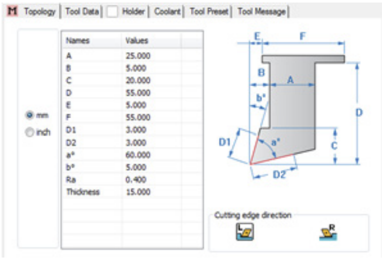
The data is copied.

In the current operation, use the same geometry that you used for the external roughing.

23. Define the Tool

Click the **Select** button in the **Tool** page. The **Choosing tool for operation** dialog box is displayed with the data of the **Tool #1** copied from the template operation. Now you have to define a new tool in the **Part Tool Table** and choose it for the operation.

Click  to start the definition of the new tool. Add a new **Ext. Rough** tool.



- Set the **A** parameter to **25**
- Set the **D** parameter to **55**
- Set the **F** parameter to **55**
- Set the **Ra** parameter to **0.4**

Choose the **Mounting** type–  .

Choose  (**Left)** in the **Cutting edge direction** area.

Click the **Select** button to confirm the tool definition and choose it for the operation.

Now you have to define the **Spin** and **Feed** values. Click the **Data** tab in the **Tool** page of the **Turning Operation** dialog box.

Generally, the **Spin** value can be calculated using the following formula:

Spin=(1000*V)/(π*D), where **V** is the cutting speed and **D** is the diameter.

In this case, the differences of diameters through the tool path do not enable you to use the common **Spin** value.

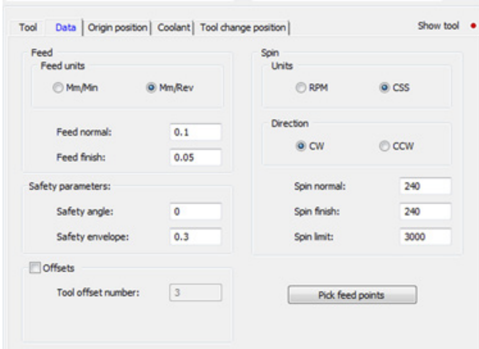
The **Constant Surface Speed** (**CSS**) option is used. This option enables you to define the cutting speed. The number of revolutions per minute is calculated automatically according to the actual diameter. Using this option, you can maintain the constant cutting speed along the tool path. In the areas of the smallest diameter, the number of revolutions is greater and vice versa.

Under **Spin**, choose the **CSS** option in the **Spin units** area.

Set the cutting speed value of **240** for the **Spin normal** and the **Spin finish**.

Set the **Spin limit** value to **3000**. With this value, SolidCAM limits the number of revolutions per minute.

Click **Select** to choose the tool for the operation.



24. Define the technological parameters

Switch to the **Technology** page of the **Turning Operation** dialog box.

Under the **General** tab, in the **Work type** area of the **Technology** page, choose the **Finish only** option.

This option is used for the finish turning of the CAM-Part. When this option is chosen, only the semi-finish or finish pass is executed.

In the **Semi-finish/finish** tab, choose the **ISO-Turning Method** option in **Finish** area. This option enables you to perform the finishing pass in the direction of the geometry.

25. Save and Calculate

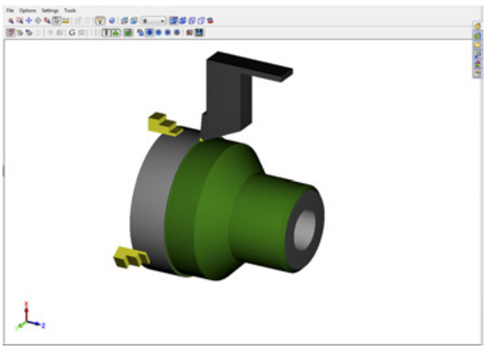
Click the **Save & Calculate** icon in the **Turning Operation** dialog box to save the operation data and calculate the tool path.

26. Simulate

Click the **Simulate** icon in the **Turning Operation** dialog box. Simulate the tool path in the **Turning** mode.

Simulate the tool path in the **SolidVerify** mode.

Close the **Simulation** control panel with the **Exit** button. Close the **Turning Operation** dialog box with the **Exit** icon.

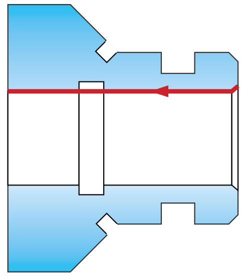


27. Add an Internal Turning operation

Right-click the last defined Turning operation in **SolidCAM Manager** and choose **Turning** from the **Add Turning Operation** submenu.

The **Turning Operation** dialog box is displayed.

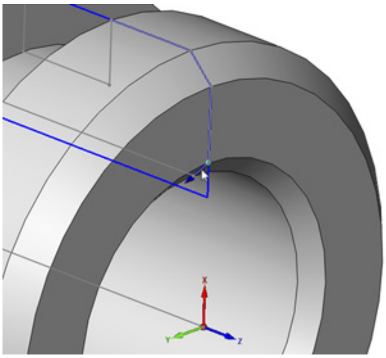
28. Define the Geometry



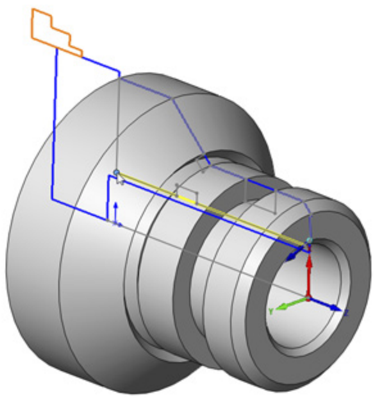
The following geometry has to be defined in order to perform the internal turning.

Click in the **Geometry** page. The **Geometry Edit** dialog box is displayed.

Select the chamfer segment as shown.



Note the direction of the geometry. The geometry defined for this operation must be directed in the negative direction of the Z-axis.



In the **Chain** section of the **Geometry Edit** dialog box, switch to the **Point to point** mode.

Click on the end point of the geometry as shown.

The geometry chain is selected.

In the **Modify Geometry** section, set **Distance** to **3** in **Start Extension/trimming** and to **1** in **End Extension/ trimming**. Confirm your selection by with .

The **Turning Operation** dialog box is displayed.

29. Define the Tool

Click the **Select** button in the **Tool** page to start the definition of a new tool for the operation. The **Part Tool Table** is displayed.

Click to define a new tool.

From the **Solid Tools** section, choose the **Int.Rough** type to define the tool for the internal finish.

Set the following parameters:

- Set the width of the tool holder (**A**) to **15**
- Set the height of the tool (**D**) to **100**
- Set the lengths of the cutting edges: **D1** to **4** and **D2** to **5**
- Set the width of the turret (**F**) to **30**
- Set the **b°** parameter to **27**
- Set the tool nose radius (**Ra**) to **0.4**

Confirm the tool definition with the **Select** button.

Click the **Data** tab in the **Tool** page of the **Turning Operation** dialog box to define the **Spin** data for the operation.

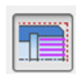
The Cutting speed recommended for the Turning operation is **180 m/min**. According to the following formula,

Spin=(1000*V)/(π*D), where **V** is the cutting speed and **D** is the diameter (**31.5** mm),

Spin≈1800.

Set the **Spin normal** and **Spin finish** to **1800**.

30. Define the technological parameters

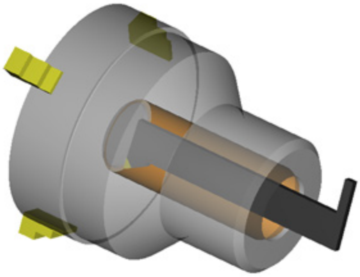
Switch to the **Technology** page. Make sure the **Long internal**  option is chosen in the **Mode** area. The combination of these options enables you to perform the internal radial turning.
In the **Work type** area, choose the **Finish only** option. This option is used for finish turning of the CAM-Part. When this option is chosen, only a semi-finish or a finish pass is executed.

Choose the **ISO-Turning Method** option for **Finish** in the **Semi-finish/finish** tab.
At this stage, all relevant technological parameters are defined.

31. Save and Calculate

Click the **Save & Calculate** icon in the **Turning Operation** dialog box to save the operation data and calculate the tool path.

32. Simulate



Click the **Simulate** icon in the **Turning Operation** dialog box. The **Simulation** control panel is displayed. Simulate the tool path in the **Turning** mode.

Simulate the tool path in the **SolidVerify** mode.

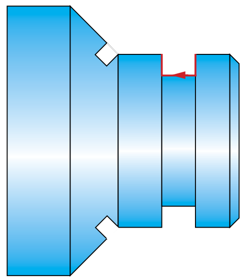
Close the **Simulation** control panel with the **Exit** button. Close the **Turning Operation** dialog box with the **Exit** icon.

33. Add an External Grooving operation

Right-click the last defined Turning operation in **SolidCAM Manager** and choose **Grooving** from the **Add Turning Operation** submenu.

The **Grooving Operation** dialog box is displayed.

34. Define the Geometry

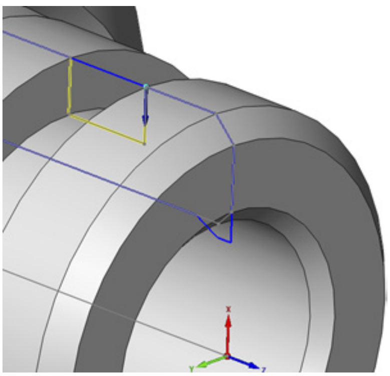


The following geometry has to be defined in order to perform the machining of the external groove.

Click in the **Geometry** page.

The **Geometry Edit** dialog box is displayed.

Choose the **Curve** option in the **Chain** section and select the sketch segments as shown.



Confirm the chain definition with the **Accept Chain** button.

In the **Modify Geometry** section, set the **Distance** values to **1** in the **Start Extension/ trimming** and **End Extension/ trimming** areas.

Close the **Modify Geometry** dialog box with the button.

35. Define the Tool

Click the **Select** button in the **Tool** page. The **Part Tool Table** is displayed.

Click  to start a new tool definition.

From **Solid Tools**, add an **Ext. Groove** tool for the operation.

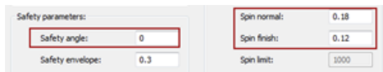
Update the following parameters of the tool:

- Set the width of the tool holder (**A**) to **25**
- Set the distance (**B**) to **-8**
- Set the height of the tool tip (**C**) to **15**
- Set the lower width of the tool tip (**G**) to **3**
- Set the lengths of the tool tip cutting edges (**D1** and **D2**) to **7**
- Set the tool tip angles **a°** and **b°** to **-1**
- Set the nose radius **Ra** to **0.2**

Select  as the **Mounting** type. Select  (**Left**) for the **Cutting edge direction**.

The tool parameters are defined. Click the **Select** button.

Click the **Data** tab in the **Tool** page of the **Grooving Operation** dialog box. These parameters allow you to define the **Feed** values.

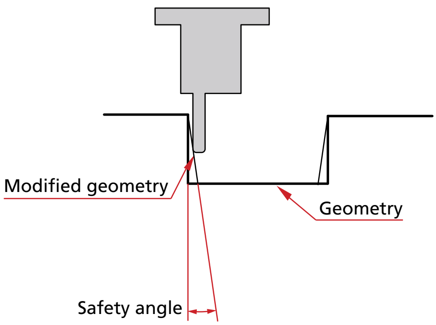


Set the **Feed normal** to **0.18** and the **Feed finish** to **0.12.**

Set the **Safety angle** value to **0**.

Safety angle

This parameter defines the angle between the material and the cutting edge of the tool. It prevents the cutting edge of the tool from coinciding with the material.



36. Define the technological parameters

Switch to the **Technology** page. Make sure that the **Long external**  option is chosen in the **Mode** area. This option enables you to perform the External grooving tool path.

SolidCAM enables you to combine roughing and finishing in one operation. Make sure that the **Rough** option is chosen in the **Work type** area.

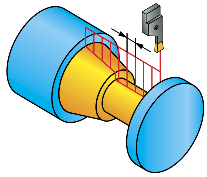
Click the **Rough** tab to define the rough grooving parameters.

In the **Rough offset** area, choose the **Distance** option. This option enables you to define a constant offset distance from the geometry. Set the **Distance** value to **0.2**.

In the **Step over** area, set the **Value** to **2.8**.

Step over

This field defines the sideways distance between each two successive groove-cut steps. This distance must be smaller than the tool width.



Switch now to **Semi-finish/Finish** tab. Make sure that the **Turn-Groove method** option is chosen in the **Finish** area.

Turn-Groove method

The finishing pass is executed. The tool movements are generated in such way that only the bottom of the tool cuts the material. This can cause the tool to move against the direction of the geometry, while the movement of the tool on the geometry is not continuous.

From the **Finish on** box, select the **Entire geometry** option.

This option performs finishing on the entire profile geometry.

Switch to the **Groove parameters** tab and set the **Finish** value to **0** in the **Delta compensation** area.

Delta compensation

For good surface finishing and better cutting conditions, the tool should be allowed to bend slightly during the side cutting. This is important so that only the corner radius of the tool will touch the material. This field defines the distance the tool retreats before going sideways.

- **Rough**

  Enter the distance the tool should retreat before moving sideways in the roughing operation.

- **Finish**

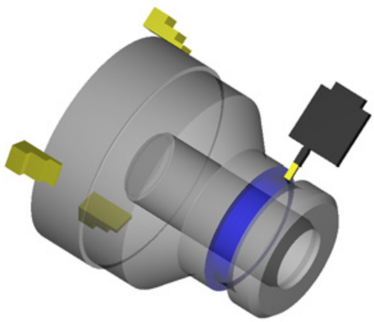
  Enter the distance the tool should retreat before moving sideways in the finishing operation.

37. Save and Calculate

Click the **Save & Calculate** icon in the **Grooving Operation** dialog box to save the operation data and calculate the tool path.

38. Simulate

Click **Simulate** in the **Grooving Operation** dialog box.
Simulate the tool path in the **Turning** mode.
Simulate the tool path in the **SolidVerify** mode.



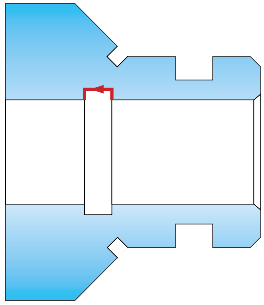
Close the **Simulation** control panel. Close the **Grooving Operation** dialog box.

39. Add an Internal Grooving operation

Right-click the last defined Grooving operation in **SolidCAM Manager** and choose **Grooving** from the **Add Turning Operation** submenu.

The **Grooving Operation** dialog box is displayed.

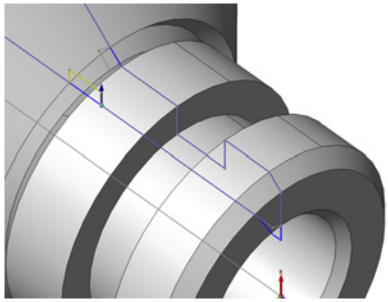
40. Define the Geometry



The following geometry has to be defined in order to perform machining of the internal groove.

Click in the **Geometry** page.

The **Geometry Edit** dialog box is displayed.



With the **Curve** option chosen, select the sketch segments as shown. Confirm the chain definition with the **Accept Chain** button.

The **Grooving Operation** dialog box is displayed. Click the **Modify Geometry** button.

Set the **Distance** values in the **Start Extension** and the **End Extension** areas to **1**. Close the **Geometry Edit** dialog box with .

41. Define the Tool

In the **Tool** page, click the **Select** button. The **Part Tool Table** is displayed.

Click to start the definition of a new tool.

From the **Solid Tools** section, choose the **Int.Groove** tool type.

The default parameters of the Internal grooving tool are displayed.

Define the following parameters:

- Set the width of the tool tip (**G**) to **3**
- Set the width of the tool tip carrier (**H**) to **3**
- Set the tool tip angles **a°** and **b°** to **-1**
- Set the nose radius (**Ra**) to **0.2**
- Set the height of the tool (**D**) to **60**

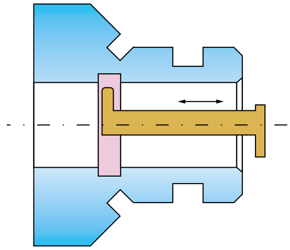
Choose the **Mounting** type according to the picture– . Set the **Cutting edge direction** to **Left** by clicking .

Click the **Tool Data** tab to define the feeds and safety angle.

- Set **Feed normal** to **0.15** and **Feed finish** to **0.1**
- Set the **Safety angle** to **0**

Confirm the dialog box with **Select**.

42. Define the technological parameters



Switch to the **Technology** page. Make sure that the **Long internal**  option is chosen in the **Mode** area. This option enables you to perform the internal radial grooving. Make sure that the **Rough** option is chosen in the **Work type** area.

Click the **Rough** tab to set the following.

Make sure that the **ZX-ABS** option is chosen in the **Rough offset** area.

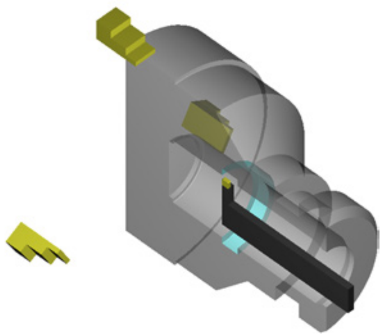
- Set the **Distance X** to **0.2**
- Set the **Distance Z** to **0.05**
- In the **Step over** area, set the **Value** to **2.8**

At this stage, the technological parameters of the operation are defined.

43. Save and Calculate

Click the **Save & Calculate** icon in the **Grooving Operation** dialog box to save the operation data and calculate the tool path.

44. Simulate



Click **Simulate** in the **Grooving Operation** dialog box.

Simulate the tool path in the **Turning** mode.

Simulate the tool path in the **SolidVerify** mode.

Click the **Half view**  button to see the section of the machined part as shown.

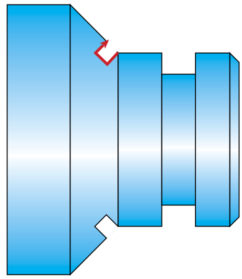
Close the **Simulation** control panel. Close the **Grooving Operation** dialog box.

45. Add an Angled Grooving operation

Right-click the last defined Grooving operation in **SolidCAM Manager** and choose **Angled Grooving** from the **Add Turning Operation** submenu.

The **Angled Grooving Operation** dialog box is displayed.

46. Define the Geometry

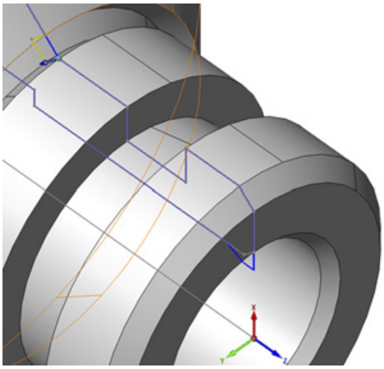


The following geometry has to be defined in order to perform the machining of the angled groove.

Click in the **Geometry** page**.**

The **Geometry Edit** dialog box is displayed.

With the **Curve** option chosen, select the sketch segments as shown.



Close the **Geometry Edit** dialog box with the button.

47. Define the Tool

In the **Tool** page, click the **Select** button. The **Part Tool Table** is displayed.

Click to start the definition of a new tool.

From the **Solid Tools** section, choose the **Ext.Groove** tool type.

The default parameters of the External grooving tool are displayed.

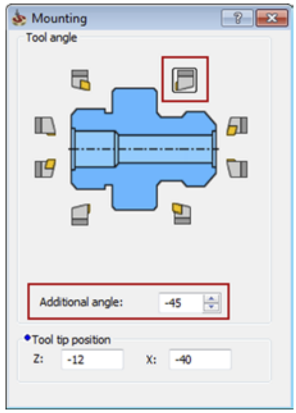
Define the following parameters:

- Set the width of the tool tip (**G**) to **2**
- Set the distance the tool tip extends beyond the tool tip carrier (**C**) to **10**
- Set the cutting edge lengths (**D1, D2**) to **4**
- Set the tool nose (**Ra**) to **0.2**
- Set the **Cutting edge direction** to **Left**

Now you have to define the tool tilting in **45°**.

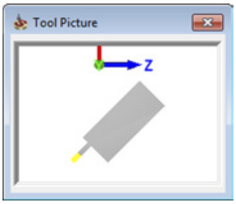
Click the **Advanced** button in top right corner to display the **Mounting** dialog box.

Click to choose orientation as shown. Set the **Additional angle** parameter to **-45°** to tilt the tool.



To make sure that the tool is mounted in the desired orientation, click the **Show tool** icon located in the top right corner.

The **Tool Picture** window is displayed showing the tool in the selected orientation.



Close the **Tool Picture** window. Click the **Select** button to confirm the tool selection.

48. Define the technological parameters

Switch to the **Technology** page. Make sure that the **Long external** option is chosen in the **Mode** area. This option enables you to perform the angled grooving.

Make sure that the **Rough** option is chosen in the **Work type** area and click the **Rough** tab to define the roughing parameters.

Make sure that **Single** is chosen in the **Step down** section. This option allows you to machine the groove in a single step down.

Set **Step over** value to **0.2**.

Switch to **Semi-finish/Finish** tab. Choose the **ISO-Turning method** in the **Finish** section.

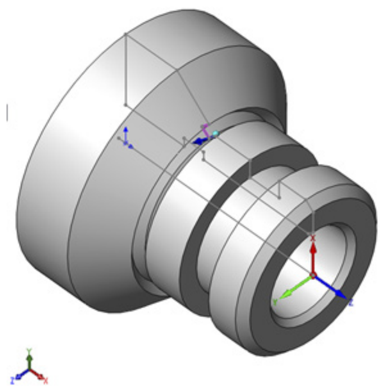
49. Define the Lead out option

Switch to the **Link** page. These options enable you to define approach and retract movements for each turning operation.

In the **Retract point** section, select the **Direct** option and set the following values :

- Set **X (Dia.)** to **60**
- Set **Z** to -**40**

These values can also be picked using the **Pick** button by clicking directly on the model.



50. Save and Calculate

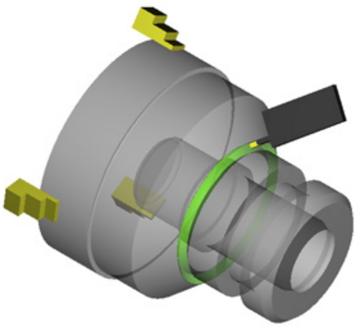
Click the **Save & Calculate** icon in the **Angled Grooving Operation** dialog box to save the operation data and calculate the tool path.

51. Simulate

Click the **Simulate** icon in the **Angled Grooving Operation** dialog box.

Simulate the tool path in the **Turning** mode.

Simulate the tool path in the **SolidVerify** mode.



Close the **Simulation** control panel. Close the **Grooving Operation** dialog box.

In the next stage, you have to perform machining of the external thread using a Threading operation. The minimal diameter of the thread is **56** mm, and the pitch is **1.5** mm.

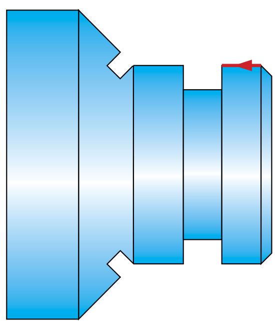
52. Add a Threading operation

Right-click the last defined Angled Grooving operation in **SolidCAM Manager** and choose **Threading** from the **Add Turning Operation** submenu.

The **Threading Operation** dialog box is displayed.

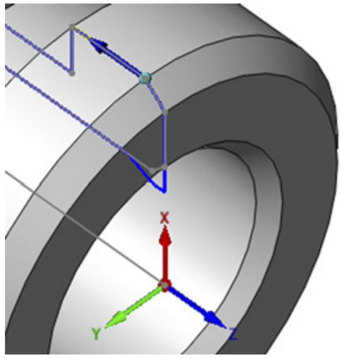
53. Define the Geometry

The following geometry has to be defined to machine the external thread.



Click  in the **Geometry** page.

The **Geometry Edit** dialog box is displayed.



With the **Curve** option chosen, select the sketch segment as shown.

Confirm the selected geometry with the button.

Notice the direction of the geometry. The geometry for this operation must be directed in the negative direction of the Z-axis.

In the **Modify Geometry** dialog box, set the **Distance** to **2** in **Start Extension/trimming** and **End Extension/trimming**. This extension enables the tool to approach the geometry from outside.

Confirm the modification with button.

54. Define the Tool

In the **Tool** page, click the **Select** button.

The **Part Tool Table** is displayed. Define a new tool for the external threading.

Click and choose the **Ext.Thread** tool from the **Solid tools** section.

- Set the tool holder width (**A**) to **25**
- Set the distance between the tool tip point and the left side of the tool holder (**B**) to **2.5**
- Set the height of the tool (**D**) to **50**
- Set the turret width (**F**) to **45**
- Set the tool nose radius (**Ra**) to **0.1**
- Set the **Mounting** type to and the **Cutting edge direction** to **Right**

Click the **Tool Data** tab.

The cutting speed chosen for the operation is **110 m/min**.

The **Spin** value can be calculated with the following formula:

Spin=(1000*V)/(π*D), where **V** is the cutting speed and **D** is the diameter.

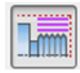
Spin=(1000*110)/(π*58)≈600 rev/min.

Set the **Spin normal** and **Spin finish** to **600**.

Click the **Select** button to confirm the tool selection.

The **Threading Operation** dialog box is displayed.

55. Define the technological parameters

Switch to the **Technology** page. Make sure that the **Long external**  option is chosen in the **Mode** area. This option enables you to perform the radial external threading

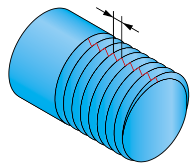
Make sure that the **Multiple** option is chosen in the **Work type** area.

SolidCAM enables you to choose between two main **Work type** options:

- **Single** – the threading is performed in a single pass (**G33**).
- **Multiple** – the threading is performed in several passes according to the machine cycle.

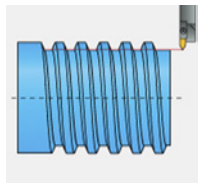
Switch to the **Data** tab. In the **Pitch unit** area, set the **Value** to **1.5**.

In the **Minor Diameter** area, set the **Value** to **56**.



Minor diameter

This option is available with **Long external** mode. It defines the minimum diameter of the thread at the first point of the geometry. The thread depth is the difference between the diameter of the first geometry point of the thread and the minimum diameter value.



Choose **Yes** under **External finish**. This option enables you to perform the finishing pass on the tips of the threads.

Choose **Yes** under **Thread finish**. The finishing pass is performed on the thread.

56. Save and Calculate

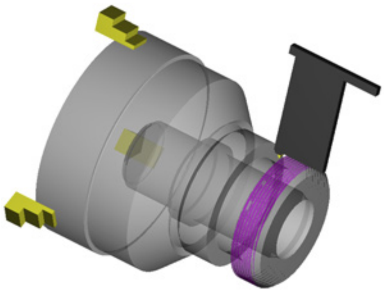
Click the **Save & Calculate** icon in the **Threading Operation** dialog box to save the operation data and calculate the tool path.

57. Simulate

Click the **Simulate** icon in the **Threading Operation** dialog box.

Simulate the tool path in the **Turning** mode.

Simulate the tool path in the **SolidVerify** mode.



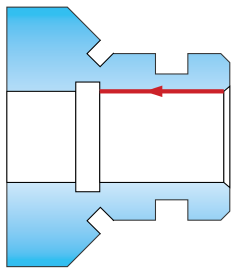
Close the **Simulation** control panel. Close the **Grooving Operation** dialog box.

At this stage, you have to perform machining of the internal thread using the Threading operation. The maximal diameter of the thread is **33.5** mm and the pitch is **1.5** mm.

58. Add an operation

Right-click the last defined Threading operation in **SolidCAM Manager** and choose **Threading** from the **Add Turning Operation** submenu.

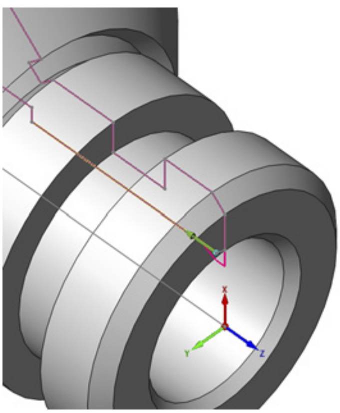
59. Define the Geometry



The following geometry has to be defined to machine the internal thread.

Click [icon] in the **Geometry** page. The **Geometry Edit** dialog box is displayed.

Select the sketch segment as shown.



Notice the direction of the geometry. The geometry for this operation must be directed in the negative direction of the Z-axis.

Confirm the selected geometry with the **Accept Chain** button.

In the **Modify Geometry** dialog box, set the **Distance** values to **2** in the **Start Extension** and **End Extension**. This extension enables the tool to approach (and retreat from) the geometry from outside.

Close the **Modify Geometry** dialog box with the button.

60. Define the Tool

In the **Tool** page, click the **Select** button to define a new tool in the **Part Tool Table** and choose it for the operation.

In the **Part Tool Table**, click .

In the **Solid Tools** section, choose the **Int.Thread** tool type.

The Internal Threading tool with the default parameters is displayed.

Set the following parameters:

- Set the width of the tool holder (**A**) to **18**
- Set the distance the tool tip extends beyond the tool tip carrier (**C**) to **4**
- Set the height of the tool (**D**) to **80**
- Set the turret width (**F**) to **32**
- Set the tool nose radius (**Ra**) to **0.1**

Set **Cutting edge direction** to left by clicking . Set the **Mounting** type to .

Click the **Tool Data**. Set the spin and feed data.

The cutting speed chosen for the operation is **80 m/min**.

The spin can be calculated with the following formula:

Spin=(1000*V)/(π*D), where **V** is the cutting speed and **D** is the maximal diameter of the thread.

Spin=(1000*80)/(π*33.5)≈750 rev/min.

Set the **Spin normal** and **Spin finish** to **750.**

Click the **Select** button to choose the tool for the operation.

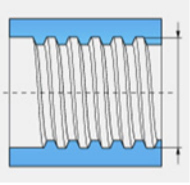
61. Define the technological parameters

Switch to the **Technology** page. Make sure that the **Long internal** option is chosen in the **Mode** area. This option enables you perform the internal radial threading.
Make sure that the **Multiple** option is chosen in the **Work type** area. Set the **Step down** to **0.1**.
Switch to the **Data** tab to set the following:

- In the **Pitch unit** area, set the **Value** to **1.5**
- In the **Major Diameter** area, set the **Value** to **33.5**

Major Diameter

This option is available with the **Long internal** mode. It defines the maximum diameter of the thread at the first point of the geometry. The thread depth is the difference between the maximal diameter value and the diameter of the first geometry point of the thread.



Choose **Yes** under **External finish**. This option enables you to perform the finish pass on the tips of the threads.

Choose **Yes** under **Thread finish**. The finish pass is performed on the thread.

At this stage, all the technological parameters of the Threading operation are defined.

62. Save and Calculate

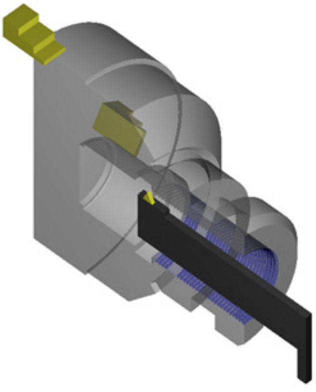
Click the **Save & Calculate** icon in the **Threading Operation** dialog box to save the operation data and calculate the tool path.

63. Simulate

Click the **Simulate** icon in the **Threading Operation** dialog box.

Simulate the tool path in the **Turning** mode.
Simulate the tool path in the **SolidVerify** mode. Use the **Half view** button to see the section of the machined part as shown.

Close the **Simulation** control panel. Close the **Threading Operation** dialog box.

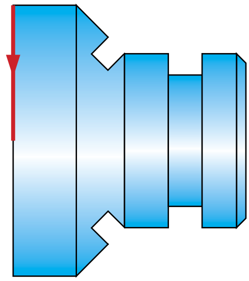


At this stage, you have to perform parting (cutting off) of the machined part using a **Cutoff** operation.

64. Add a Cutoff Operation

Right-click the last defined Threading operation and choose **Cutoff** from the **Add Turning Operation** submenu.

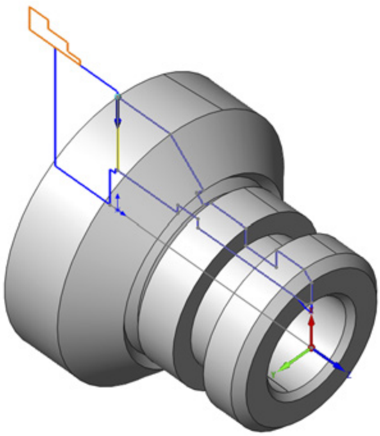
65. Define the Geometry



The following geometry has to be defined to perform the parting (cutting off).

In the **Geometry** page, click to start the geometry definition. The **Geometry Edit** dialog box is displayed.

Select the sketch segment as shown.



Note the direction of the geometry. In this operation, the geometry directed to the model rotation axis should be used.

In the **Modify Geometry** dialog box, set the **Distance** values to **2** in **Start Extension/ trimming** and **End Extension/trimming.**

66. Define the Tool

Click the **Select** button in the **Tool** page to start the definition of a new tool.

The **Part Tool table** is displayed. Click to define a new tool.

Choose the **Ext. Groove** tool from the **Solid Tools** section.

- Set the distance between the center of the radius of the tool tip and the left side of the tool holder (**B**) to **0**
- Set the height of the tool tip (**C**) to **35**
- Set the height of the tool (**D**) to **60**
- Set the position of the left side of the turret (**E**) to **0**
- Set the nose radius of the right side of the tool tip (**Ra**) to **0.2**

Set the **Cutting edge direction** to **Left** by clicking . Set the **Mounting** type to .

Click **Select** to choose the defined tool for the operation.

Click the **Data** tab in the **Tool** page of the **Cutoff Operation** dialog box. Define the spin and feed data.

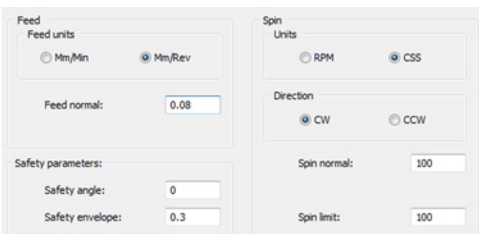
In this case, the differences of diameters through the tool path do not enable you to use the common spin value.

The **Constant Surface Speed** (**CSS**) option is used. This option enables you to define the cutting speed. The number of revolutions per minute is calculated automatically according to the actual diameter. You can maintain the constant cutting speed along the tool path. In the areas of the smallest diameter, the number of revolutions is greater and vice versa.

Switch to the **CSS** option in the **Spin Units** section.

Set the cutting speed value of **100** for **Spin normal** and **Spin limit**.

Set the **Feed normal** value to **0.08**.

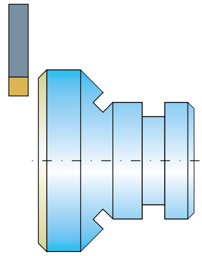


67. Define the technological parameters

Switch to the **Technology** page. This page enables you to define the parting (cutting off) technological parameters.

Make sure the **Long external**  option is chosen in the **Mode** area. Choose the **Left** option in the **Tool side** section.

In the **Corner** section, choose the **Chamfer** option. In the **Location** area, choose the **Right** option. The chamfer is machined as shown.



The **Location** options enable you to define the tool position relative to the defined geometry:

- **Left**

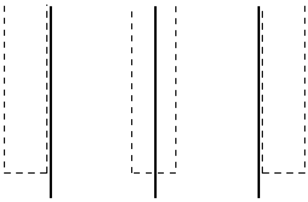
The tool cuts to the left of the geometry.

- **Right**

The tool cuts to the right of the geometry.

- **Middle**

The tool center is located on the geometry.



In the **Chamfer parameters** section, select the **Distance** option and set the **Distance X** and **Distance Z** values to **1**.

At this stage, the definition of the technological parameters of the parting operation is finished.

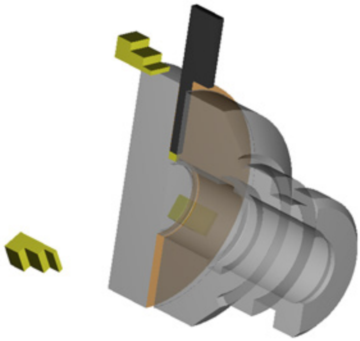
68. Save and Calculate

Click the **Save & Calculate** icon in the **Cutoff Operation** dialog box to save the operation data and calculate the tool path.

69. Simulate

Click the **Simulate** icon in the **Cutoff Operation** dialog box. Simulate the tool path in the **Turning** mode.

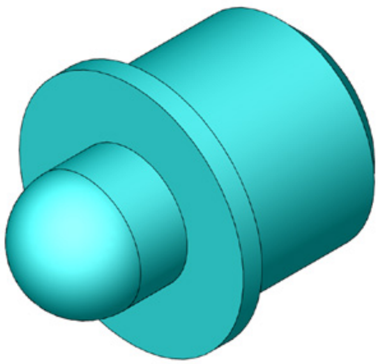
Simulate the tool path in the **SolidVerify** mode.



At this stage, the exercise is completed.

Exercise #4: Button Lock Machining

Define the CAM-Part and operations for the machining of the button lock shown below on a Turning CNC-machine.



This exercise reinforces the following skills:

- Turning CAM-Part definition
- External rough turning
- External finishing
- Parting (cutting off)

The SolidWorks model of the Button Lock (**Exercise4.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

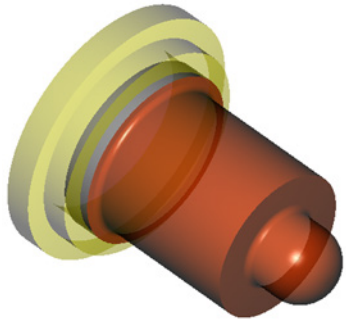
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Material boundary, the Clamp and the Target model.

The **OKUMALL** CNC-controller has to be chosen for this exercise.

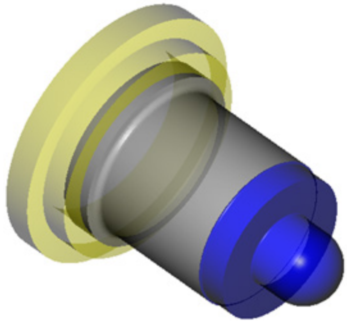
2. External roughing

Define a Turning operation to perform the rough machining of the part.



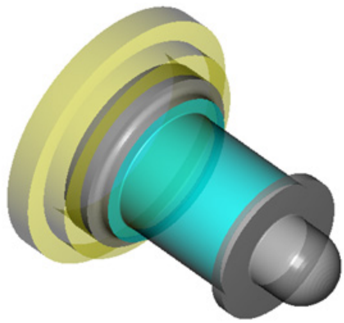
3. External finish

Perform the finishing of the front faces of the part as shown.



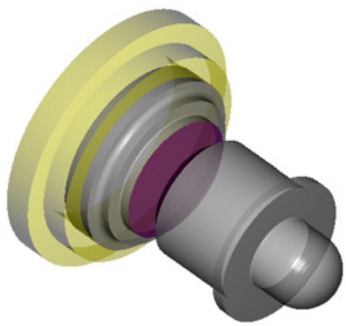
4. Back side finish

Perform the finishing of the back faces of the part.



5. Parting

Define a Cut off operation for cutting off the part and machining of the chamfer.



Exercise #5: Guided Ejector Bushing Machining

Define the CAM-Part and operations for the machining of the guided ejector bushing shown below on Turning CNC-machine.

This exercise reinforces the following skills:

- Turning CAM-Part definition
- External turning with a groove tool
- Facial turning
- Drilling
- Internal turning
- Parting (cutting off)

The SolidWorks model of the Guided Ejector Bushing (**Exercise5.sldprt**) is located in the **Exercises** folder.

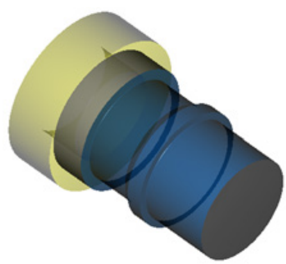
The following steps have to be implemented in order to reach the final CAM-Part:

1. Define the CAM-Part

At this stage, you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Material boundary, the Clamp and the Target model.
The **OKUMALL** CNC-controller has to be chosen for this exercise.

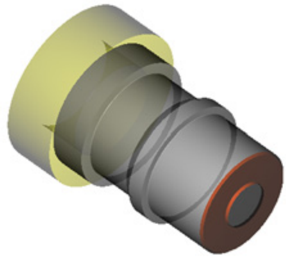
2. External turning

Define the Turning operation to perform the machining of the external radial faces. Use the grooving tool to machine the grooves located radially.



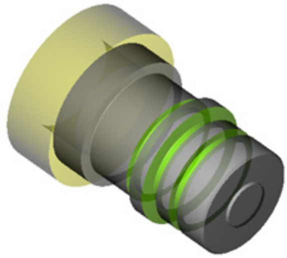
3. Facial turning

Perform the facial turning of the front face as shown. Note that it is not necessary to machine the whole face because of the next Drilling operation.



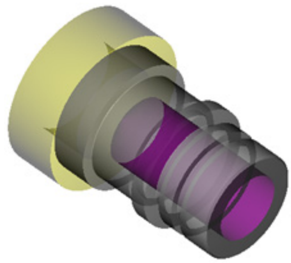
4. Grooves machining

Define a Grooving operation to machine three grooves.



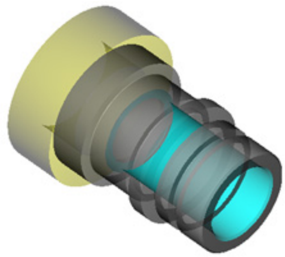
5. Drilling

Drill the preliminary hole that will be finally turned at the next stage. Use a flat drill tool (see **Exercise #3**).



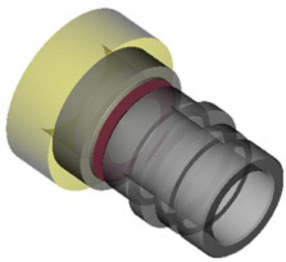
6. Internal finish

Perform the internal finish of the center hole including chamfers.



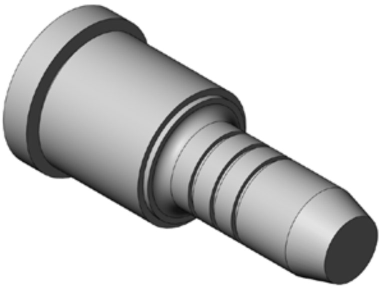
7. Parting

Define a Cut off operation for cutting off the part and machining of the chamfer.



Exercise #6: Guide Pillar Machining

Define the CAM-Part and operations for the machining of the guide pillar shown below on the Turning CNC-machine.



This exercise reinforces the following skills:

- Turning CAM-Part definition
- External rough turning
- External finish turning
- Facial turning
- Grooving
- Parting (cutting off)

The SolidWorks model of the Guide Pillar (**Exercise6.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

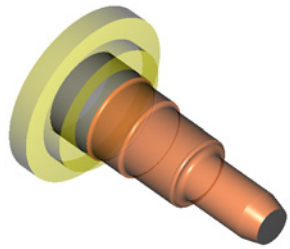
1. Define the CAM-Part

At this stage you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Material boundary, the Clamp and the Target model.

The **OKUMALL** CNC-controller has to be chosen for this exercise.

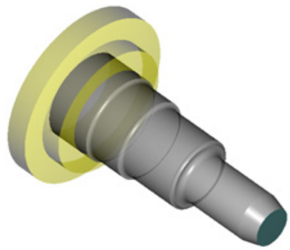
2. External rough turning

Define a Turning operation to perform the rough machining of the external radial faces.



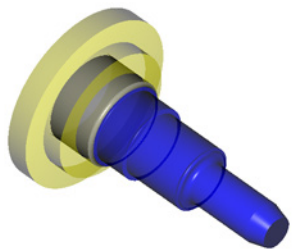
3. Facial turning

Perform the facial turning of the front face as shown.



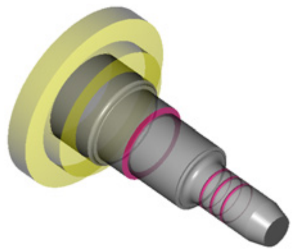
4. External finish turning

Define a Turning operation to machine the external model faces except for grooves that will be machined in one of the next stages.



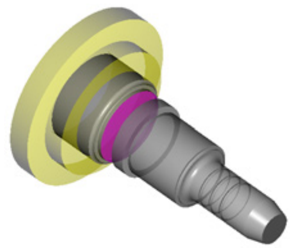
5. Grooves machining

Define a Grooving operation to machine four grooves.



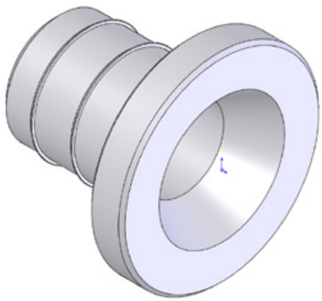
6. Parting

Define a Cut off operation for cutting off the part and machining of the chamfer.



Exercise #7: Bearing Bush Machining

Define the CAM-Part and operations for the machining of the bearing bush on the Turning CNC-machine.



This exercise reinforces the following skills:

- Turning CAM-Part definition
- External turning
- Internal turning
- Facial turning
- Drilling
- Parting (cutting off)

The SolidWorks model of the Bearing Bush (**Exercise7.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

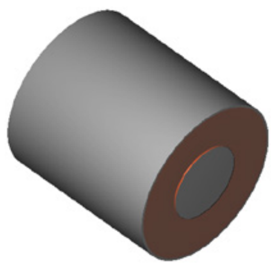
1. Define the CAM-Part

At this stage you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Material boundary, the Clamp and the Target model.

The **OKUMALL** CNC-controller has to be chosen for this exercise.

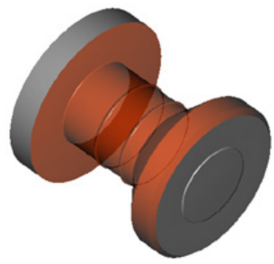
2. Facial turning

Perform the facial turning of the front face as shown.



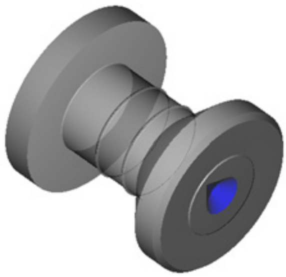
3. External faces turning

Define a Turning operation to perform the machining of the external cylindrical faces.



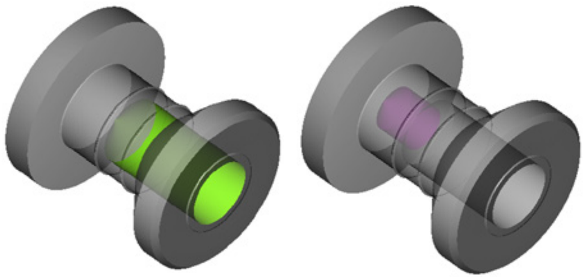
4. Center drilling

Define a Drilling operation to perform the preliminary center drilling of the center hole.



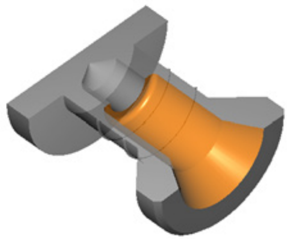
5. Drilling

Define two Drilling operations to machine the center hole segments as shown.



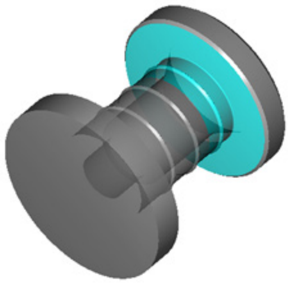
6. Internal turning

Define a Turning operation to machine the internal model faces.



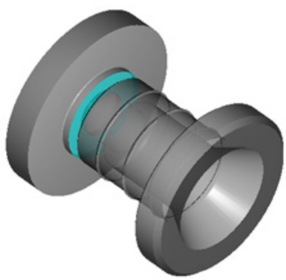
7. Back side grooving

Define a Grooving operation to machine the back faces as shown.



8. Parting

Define a Cutoff operation for cutting off the part and machining of the chamfer.

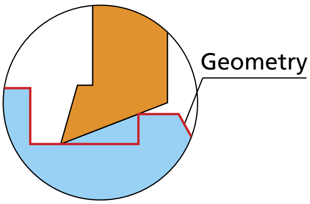


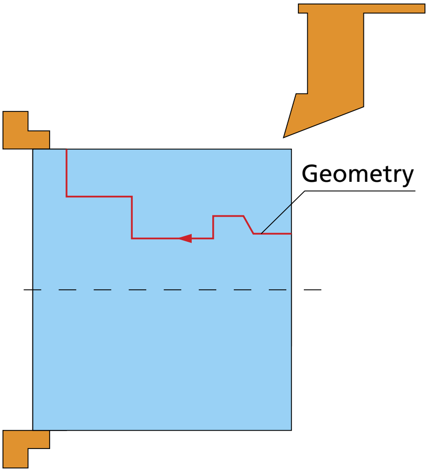
3.2 Advanced Turning: Rest Material

SolidCAM automatically recognizes the rest material areas left unmachined after the previous operations.

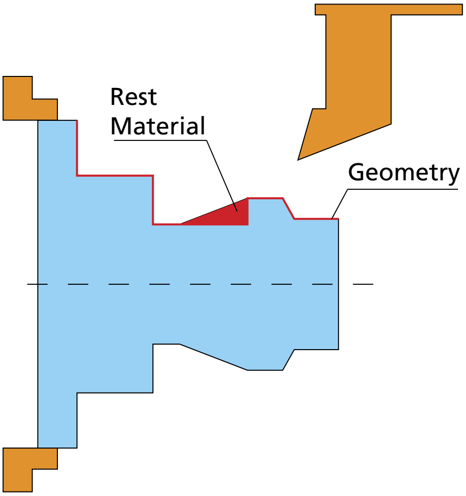
Consider the longitudinal turning of the geometry presented on the illustration with an external roughing tool.

According to its geometry, the tool cannot reach some areas of the part in the process of machining.





In SolidCAM, such unmachined areas are called **Rest Material** areas.

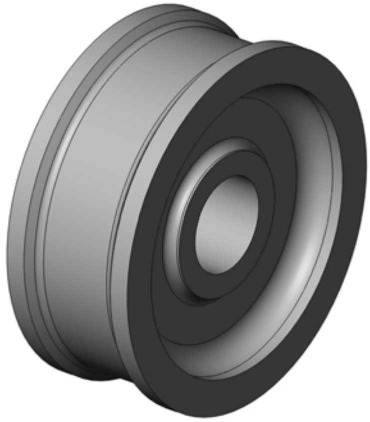


SolidCAM offers you a number of methods to determine and machine Rest Material areas.

The following exercises illustrate SolidCAM functionality of the Rest Material turning.

Exercise #8: Wheel Machining

The following exercise illustrates SolidCAM functionality of Rest Material machining during longitudinal and facial rough/finish turning operations performed on the Wheel part.



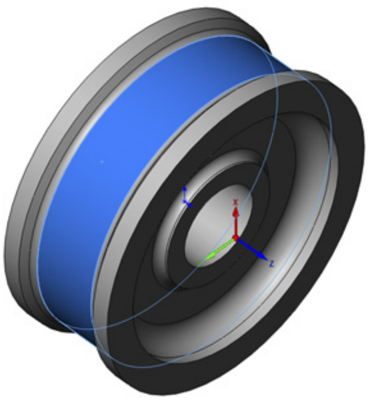
1. Load the model

Load the **Exercise8.sldprt** model located in the **Exercises** folder.

2. Define the CAM-Part

Since the CAM-Part definition process was explained earlier, the details of the CAM-Part definition process are omitted.

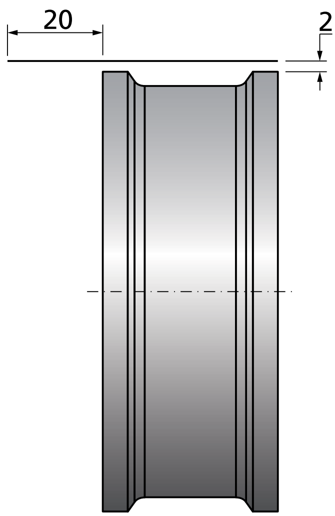
Define the **Coordinate System** in the pierce point of the revolution axis and the front face of the model.



Choose the **OKUMALL** CNC-controller.

Define the material boundary sketch on the **Plane 1** as shown. Make sure that the end points of the sketched line are coincident to the model front and back sides.

Select the sketched line for the definition of the **Stock**.



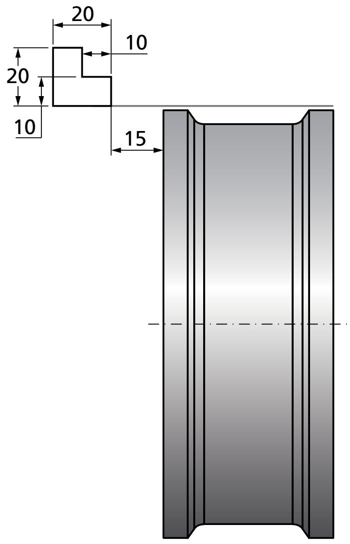
Create a new sketch for the clamp on the same plane.

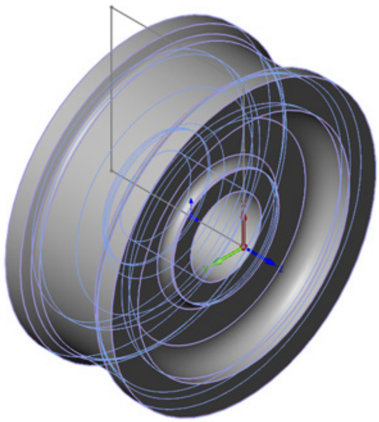
Sketch the clamp geometry as shown.

Define the SolidCAM **Clamp** geometry using this sketch.

Define the **Target model** by selecting the solid body.

When defining the Target model in the **Model** dialog box, set the **Facet tolerance** value to **0.01** to get better performance during the simulation.





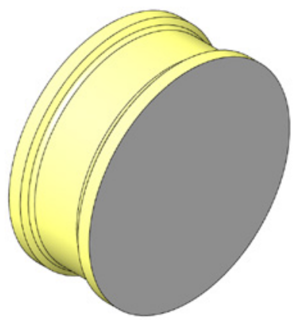
SolidCAM automatically generates the **Envelope** geometry that will be used later in the definition of operations.

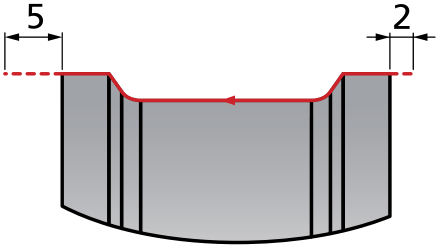
Confirm the CAM-Part definition.

3. Perform the external roughing

Define a new Turning operation to perform the rough machining of the external model faces.

Define the geometry for the operation using the **Envelope** sketch as shown below.

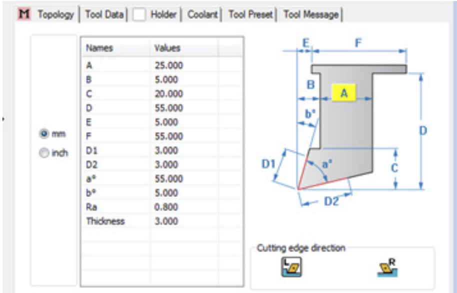




Define an **External roughing** tool for the operation.

Set the tool tip angle (**a°**) value to **55°**.

Choose the defined tool for the operation.

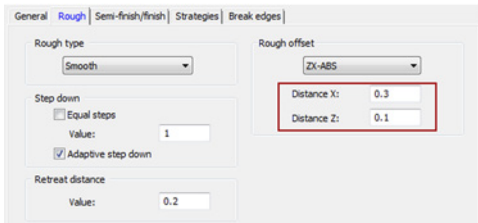


Define the parameters of External roughing. Select **Rough** from the **Work type** area in the **General** tab. Switch to the **Rough** tab to define roughing parameters:

In the **Step down** section, select the **Adaptive step down** option.

Make sure that the default **ZX-ABS** option is chosen in the **Rough offset** area.

- Set the **Distance X** value to **0.3**
- Set the **Distance Z** value to **0.1**

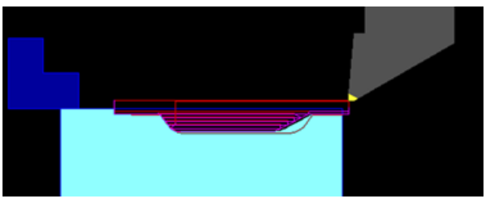


The defined offset remains unmachined till the next finishing operation.

Switch to the **Semi-finish/finish** tab. Make sure that the **No** option is selected under **Finish** and **Semi-finish**. The finish machining will be performed later in a separate operation.

Save the operation data and calculate the tool path.

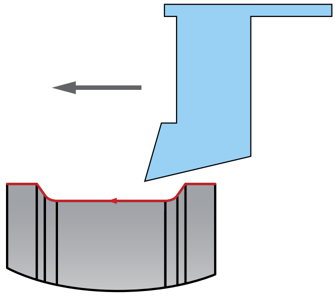
Simulate the tool path.



Some areas of the current geometry are unreachable for the tool. These areas are left unmachined.

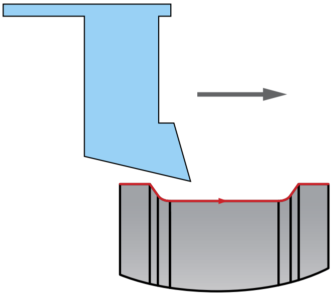
The next operation uses the **Rest material** option to perform the additional machining only in the unmachined area.

4. Perform the rest material roughing



In the previous external roughing operation, the tool with the **Left** orientation was moving in the negative Z-direction.

In the Rest material roughing operation, the tool moves in the positive direction of the Z-axis. The **Right** orientation of the tool must be chosen.

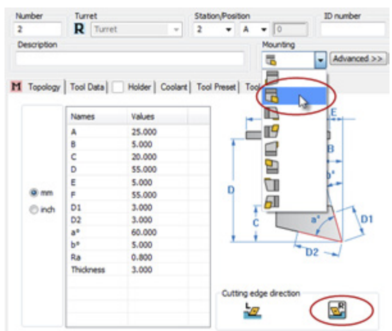


Add a new Turning operation to perform the rough machining of the **Rest material** area.

SolidCAM enables you to use an already existing operation as a template for the current operation. All the data is copied from the template to the current operation. This feature enables you to save the programming time.

In the **Operation name** section, choose the first operation (**TR_contour**) as a template.

The data is copied. This operation contains the same geometry and technological parameters as the previous Turning operation.

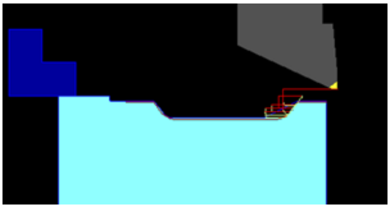


Define a new **External roughing** tool for the operation. Use the default geometric parameters of the tool.

Change **Cutting edge direction** from **Left** to **Right**. Choose the **Mounting** as shown. The tool orientation enables you to use this tool for cutting in the positive Z-direction.
Choose the defined tool for the operation.

Save the operation data and calculate the tool path.

Simulate the tool path.



During the tool path calculation, SolidCAM automatically determines the Rest material area suitable for machining with the defined tool and technological parameters. The machining is performed in this Rest material area only.

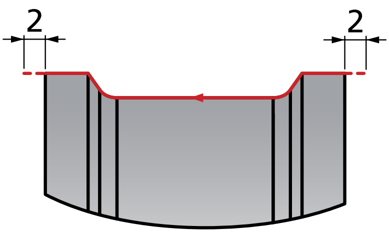
At this stage, the rough machining of the external faces is completed. At the next stage, you have to perform the finish machining.

5. Perform the external finishing

Perform the finishing of the external faces that were rough machined in the previous steps.

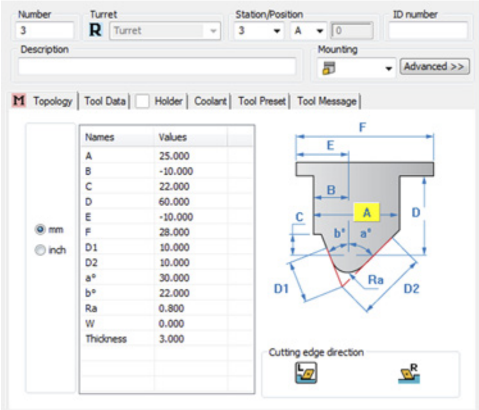
Add a new Turning operation.

Define the geometry for the operation as shown.



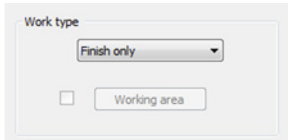
Define a new tool of the **Ext.Contour** type for the operation.

- Set the height of the tool tip (**C**) to **22**
- Set the height of the tool (**D**) to **60**
- Set the lengths of the two cutting edges of the tool tip (**D1**, **D2**) to **10**
- Set the tool tip angle (**a°**) to **30**
- Set the tool nose radius (**Ra**) to **0.8**

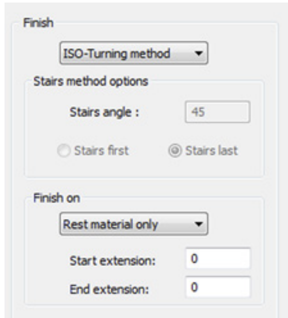


Confirm the tool definition and choose the tool for the operation.

In the **Technology** page, choose the **Finish only** option under **Work type**. This option enables you to perform the finishing of the external faces.



Switch to the **Semi-finish/finish** tab. Make sure that the **ISO-Turning method** is chosen for **Finish**. With this option, the finishing tool path is performed in the direction of the geometry.

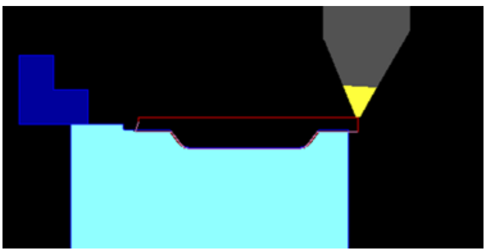


Note that the default **Rest material only** option is chosen in the **Finish on** section. This option enables you to perform finishing only in Rest material areas.

In this case, the Rest material area is the allowance left unmachined in the previous operations.

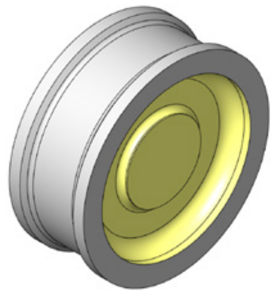
Save the operation data and calculate the tool path.

Simulate the tool path.



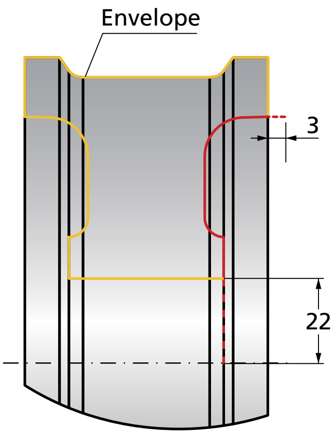
At this stage, the external faces turning is complete.

6. Perform the facial roughing



Perform the facial rough turning of the end faces of the wheel.

Add a new Turning operation and define the geometry as shown. Use the **Envelope** sketch automatically generated by SolidCAM during the Target model definition.

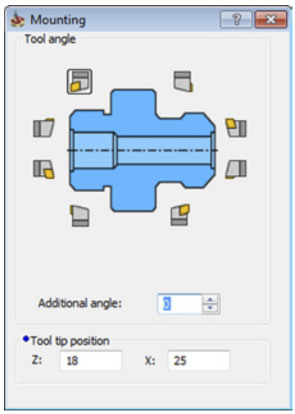


Define the tool for the operation. Add a new tool of the **Ext.Rough** type to the **Part Tool Table**. Edit the following tool parameters:

- Set the height of the tool (**D**) to **80**
- Edit the position of the left side of the turret with reference to the tool tip point (**E**). Set the value to **-15**
- Set the tool tip angle (**a°**) to **35°**
- Set the lengths of the cutting edges of the tool (**D1**, **D2**) to **6**

In the **Mounting** list, set the tool mounting to  .

Tool mounting



This parameter enables you to rotate the tool to perform the machining from different sides. Tools with the same geometry but with different rotation can be mounted on the same tool holder. These tools share the same **Tool number**, but their **Positions** are different.

Click the icon of the desired orientation to choose the major direction for the tool. SolidCAM enables you to choose different orientations for the major application direction.

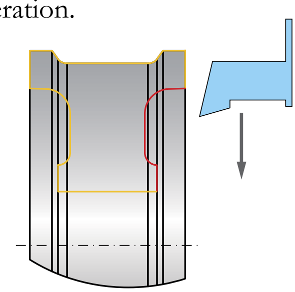
In addition to the major application direction, the **Additional angle** box enables you to perform the fine tuning of the application angle.

To display the **Mounting** dialog box, click the **Advanced** button next to the **Mounting** list.

The tool **Cutting edge direction** is automatically set to **Right**, and it cannot be changed.

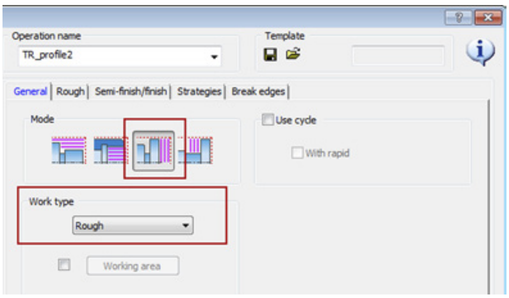
The combination of these options enables you to use the tool for the facial turning directed from outside to the revolution axis.

Confirm the tool data and choose the tool for the current operation.



In the **Technology** page, choose the **Face front**  option in the **Mode** area. With this option, SolidCAM performs the facial turning of the front faces.

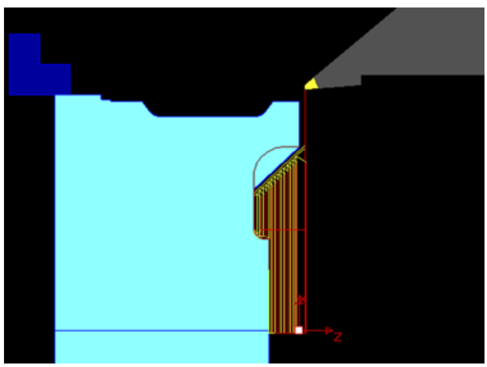
Choose the **Rough** option in the **Work type**. The finish machining is performed later in a separate operation.



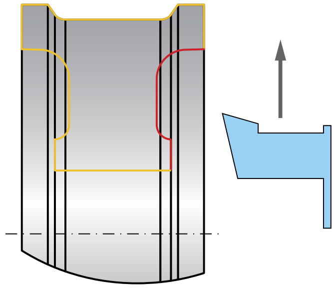
Save the operation data and calculate the tool path.

Simulate the tool path.

There is an area of the geometry that is not reachable for the tool during the machining. This area will be machined in a separate operation using the **Rest material** strategy.



7. Perform the facial Rest material machining

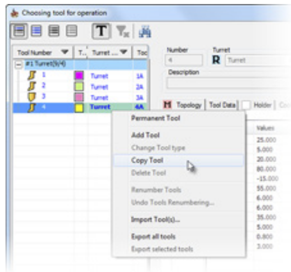


At this stage, you have to perform an additional roughing operation that removes the bulk of material in the area left unmachined after the previous operation.

In this operation, the Rest material is removed with a left-oriented tool in a number of cuts in the positive X-direction (from the revolution axis to the outside).

Add a new Turning operation. In the same manner as explained in **Step #4**, use the previous operation as a template for the current one.

In this operation, you need to use the tool with the same geometric parameters as the one used for the previous operation, except for its orientation.

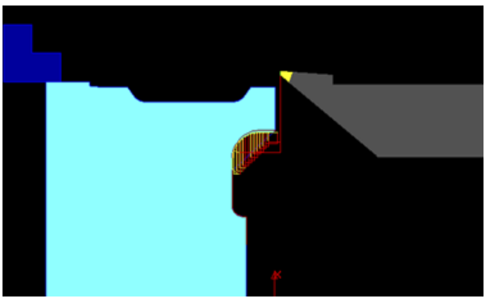


Right-click the **Tool #4** in the list and choose the **Copy Tool** command. A new tool identical to the previous one is added. Choose **Left** for **Cutting edge direction**.

Confirm the tool definition and choose the tool for the operation.

Save the operation data and calculate the tool path.

Simulate the tool path.



During the tool path calculation, the Rest material area suitable for the machining with the defined tool and technological parameters is determined automatically. The machining is performed in this Rest material area only.

At this stage, the rough machining of the front faces is completed. At the next stage, you have to perform the finish machining.

8. Perform the facial finishing

Define a new Turning operation. Use the Turning operation defined in **Step #6** as a template to copy all the data into the current operation. Use the same tool and the same geometry as in the template operation.

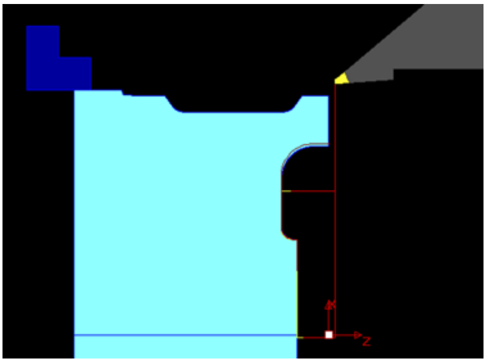
Click the **Data** tab in the **Tool** page and set the **Safety angle** to **5** in the **Safety parameters** area.

In the **Work type** list of the **Technology** page, choose the **Finish only** option. This type enables you to perform the finish machining along the geometry.

Make sure that the **ISO-Turning method** option is chosen in the **Finish** area of the **Semi-finish/finish** tab.

The **Rest material only** option is automatically chosen in the **Finish on** box. With this option, SolidCAM performs the machining only in the Rest material areas. Set the **Start extension** and **End extension** values to **1**. These values enable you to overlap the rest material area.

Save the operation data and calculate the tool path. Simulate the tool path.

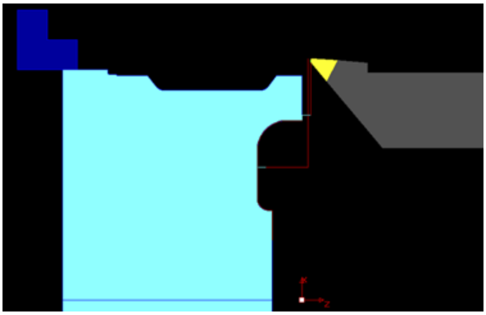


In the same manner, define a new Turning operation based on the operation defined in **Step #7**. This operation performs the facial finish machining in the Rest material area. Use an **External roughing** tool with the lengths of cutting edges (**D1**, **D2**) of **9** mm.

In the **Finish** section, choose the **ISO-Turning method** option.

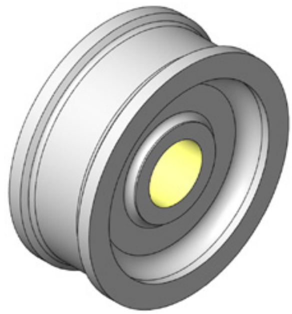
The **Rest material only** option is automatically chosen in the **Finish on** box. With this option, SolidCAM performs the machining only in the Rest material areas. Set the **Start extension** and **End extension** values to **1**. These values enable you to overlap the rest material area.

Save the operation data and calculate the tool path. Simulate the tool path.

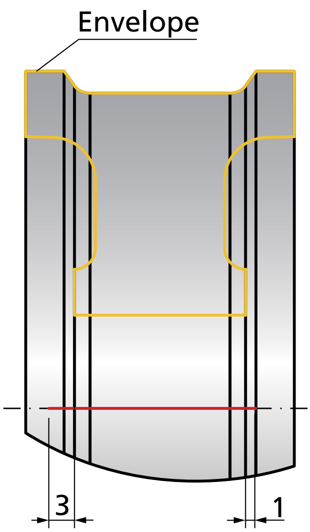


At this stage, the facial machining of the front faces is finished.

9. Hole machining

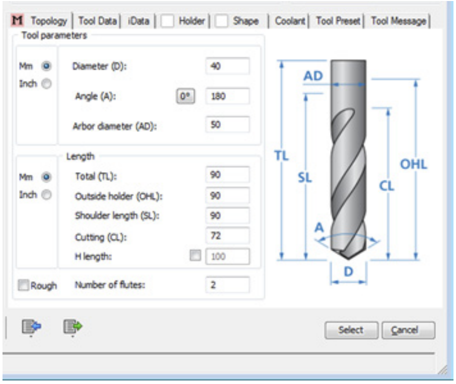


At the first stage, sketch a line as shown below to define the start and the end points for drilling.

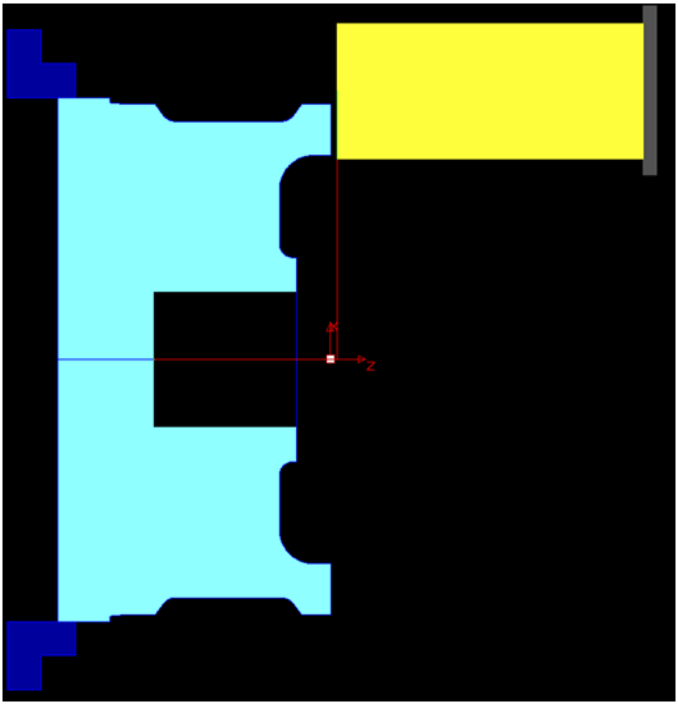


Define a new Drilling operation to machine the hole.

In the same manner as explained in **Steps #15-21** of **Exercise #3**, define a new **U-drill tool** for the operation. Use the tool of **Ø40**.



Define the **Drill start** and the **Drill end** at the end points of the line.

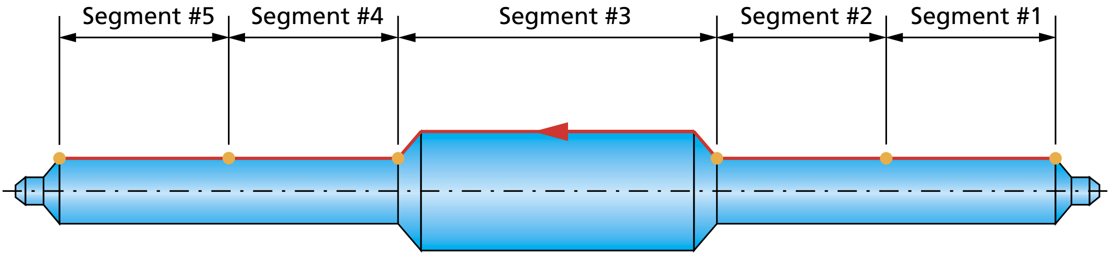


Save the operation data and calculate the tool path. Simulate the tool path.

At this stage, the exercise is completed.

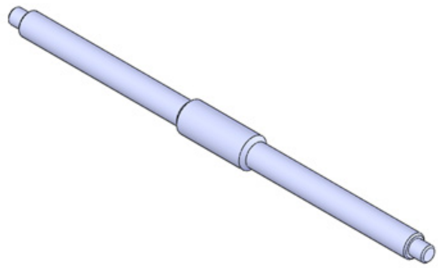
3.3 Advanced Turning: Partial machining

This functionality supports rough, semi-finish and finish machining of long turning parts. In long turning parts the machining of the whole geometry all at once might cause problems due to the deformation of the workpiece under the cutting tool. Partial machining provides the capability to machine the geometry by dividing it into several segments



Exercise #9: Long Shaft Machining

The following exercise illustrates SolidCAM functionality of Partial machining in turning operations performed on the Long Shaft part.



1. Load the part file

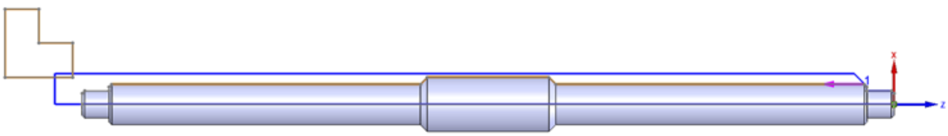
Load the **Exercise9.prz** file located in the **Exercises** folder.

This file contains the pre-machined part with two Turning operations in which the machining of the front face and the first step is performed.

2. Machine the first segment of the geometry

Add a new Turning operation in **SolidCAM Manager**.

Define the full machining geometry as shown.

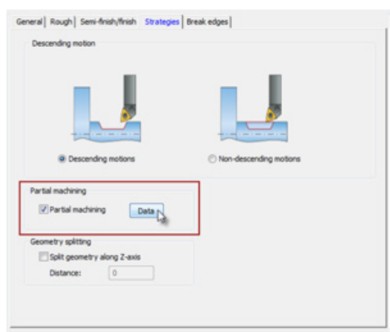


This geometry will be machined in several segments in a number of Turning operations.

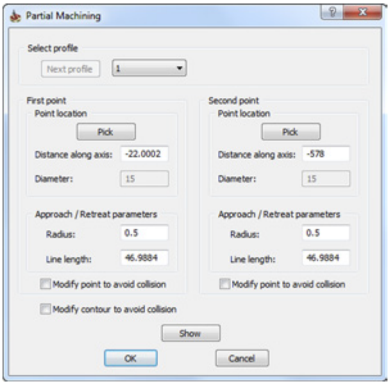
In the **Part Tool Table**, choose the **Ext.rough tool** (**Tool #2**) used in the previous operation.

In the **Technology** page, click **Strategies** tab and select the **Partial Machining** check box.

Click the **Data** button that becomes available. The **Partial Machining** dialog box is displayed, and the geometry chain is highlighted on the model.



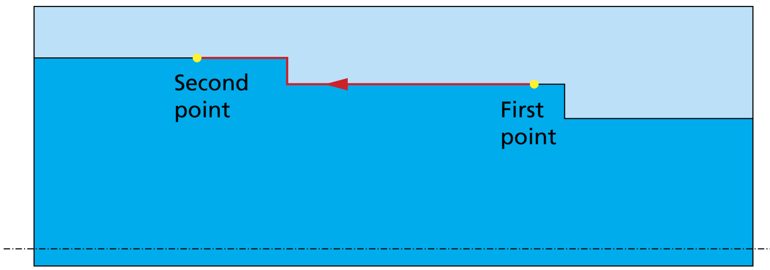
This dialog box enables you to define the geometry segment that will be machined in the operation and related parameters such as its initial and final points, distance and approach/retreat data.



First point/Second point

The **First point** and **Second point** sections enable you to define the profile segment that will be machined in the operation and the parameters of the tool approach/retreat movements.

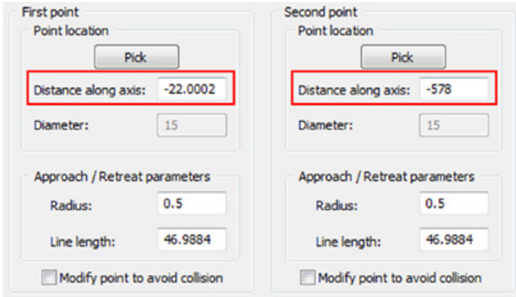
The generated partial cutting pass is performed between the first and second points; the cutting pass direction is determined according to the **Profile direction** option state.



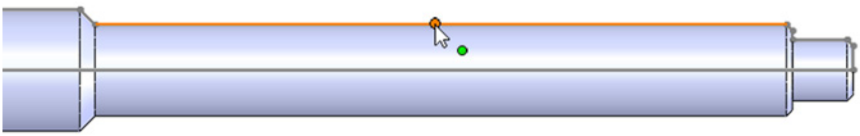
SolidCAM automatically generates the approach and retreat passes at the **First point** and **Second point** according to the machining direction. In the illustration above, the approach movement is performed at the **First point** and the retreat movement is performed at the **Second point**.

The approach/retreat pass consists of an arc tangential to the partial machining geometry at the **First/Second point**, and a line connected to the arc. The line direction is automatically chosen according to the **Tool type**, **Tool tip angle** and **Safety angle** parameters.

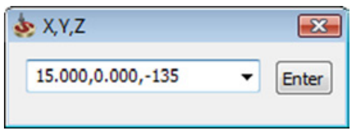
By default, the **Distance along axis** parameter located in the **First point** and **Second point** sections displays the minimal/maximal Z-coordinate of the geometry. Since the first segment to be machined starts at the beginning of the geometry, do not change the default value of this parameter for the **First point**.



Pick the final point of the segment to be machined by clicking approximately in the middle of the edge as shown.



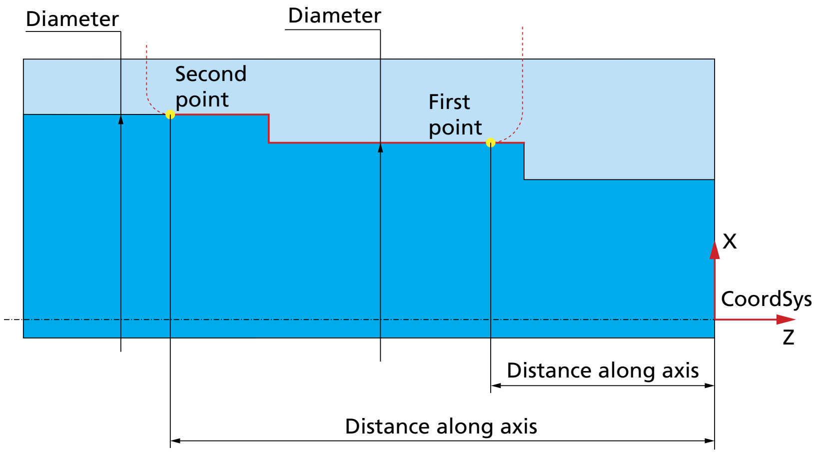
The **Distance along axis** value for this point is calculated automatically (**-135.61**) and displayed in the **X,Y,Z** dialog box. Round it down manually to **-135**.



When the point has been picked, SolidCAM calculates the **Distance along axis** and **Diameter** values for the picked point and displays them in the **X,Y,Z** dialog box.

The **Distance along axis** parameter displays the distance from the CoordSys to the **First/Second point** along the Z-axis. When the **Pick** option was used, the value is calculated automatically. It can also be used as an alternative to the **Pick** option to define the **First/Second point** locations. Even if the **Pick** option was already used, you can edit the value received from the model and thus change the **First/Second point** locations.

The **Diameter** parameter displays the value of the part diameter at the specified location and is calculated automatically according to the geometry and the **Distance along axis** value. When there are several **Diameter** values for a single **Distance along axis** value, SolidCAM chooses the maximal **Diameter** value determined.

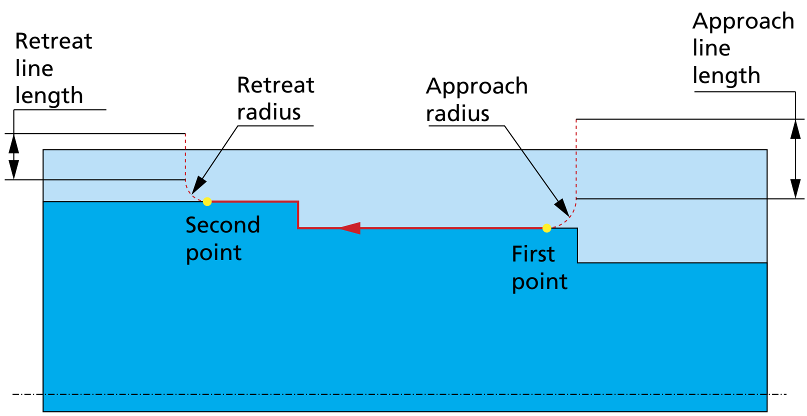


Now you need to define the approach and retreat parameters.

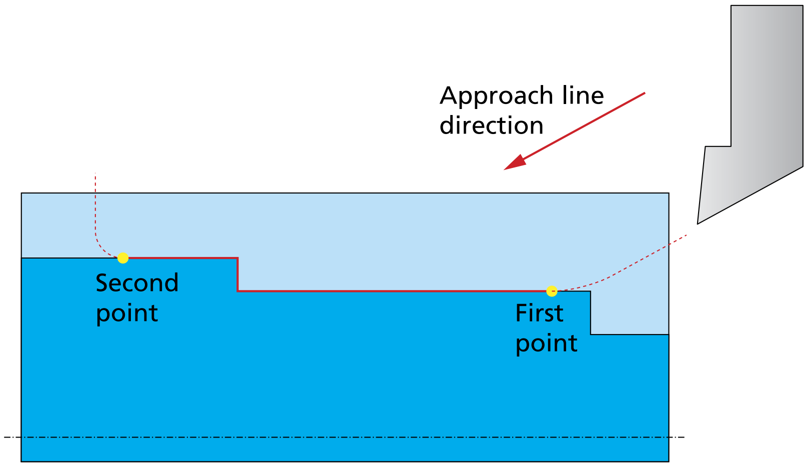
Approach/Retreat parameters

The **Radius** parameter defines the radius of the approach/retreat arc tangential to the profile at the **First/second point**.

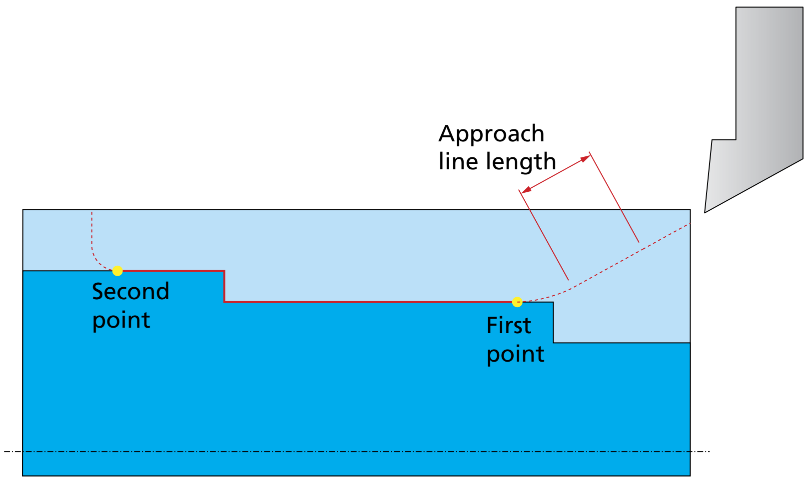
The **Line length** parameter defines the length of the straight line tangential to the arc. The direction of the approach/retreat lines, connected to the approach and retreat arcs, is chosen according to the **Tool type**.



For **External Rough** tools, the approach line is determined as follows:

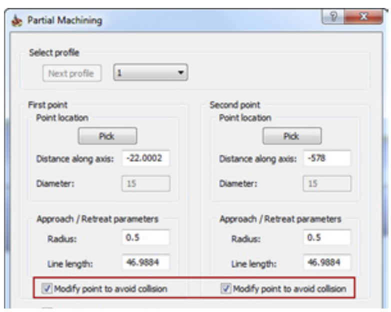


When the approach/retreat lines are located inside the Material boundary of the pre-machined stock, SolidCAM automatically extends them till the Material boundary.



Make sure that the **Radius** value for both **First point** and **Second point** is **0.5**. It must be equal to the tool nose radius (**Ra**).

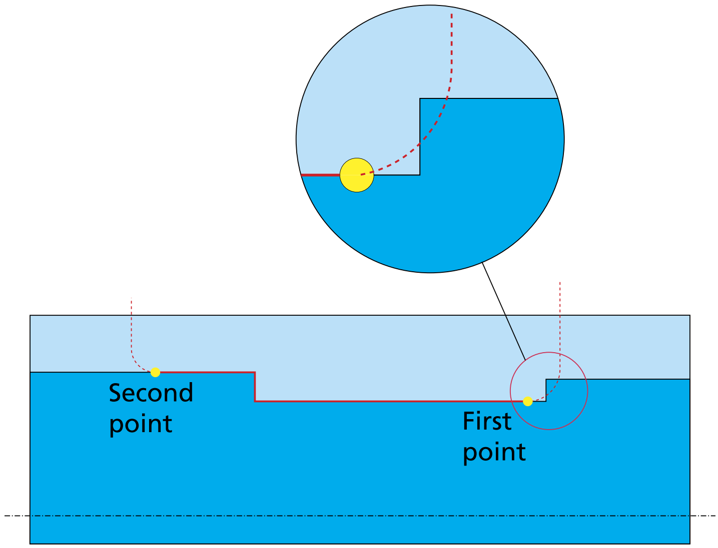
Select the **Modify point to avoid collision** check box for both **First point** and **Second point**.



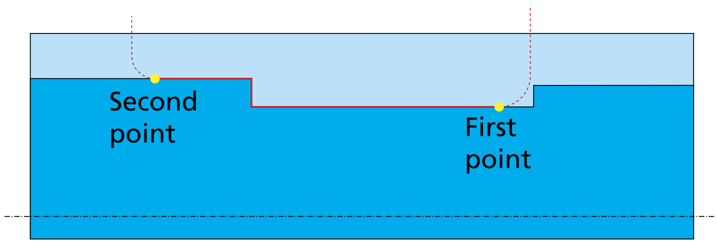
Modify point to avoid collision

When this check box is selected, automatic modification of the defined **First** and **Second points** is performed according to the geometry. The point modification enables you to avoid possible collisions during the approach and retreat movements.

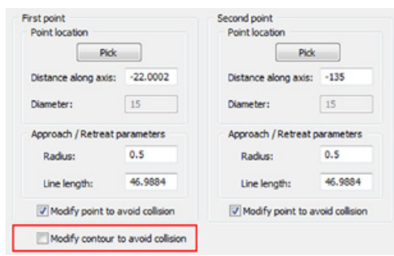
When the **First point** is not modified, the approach is performed as follows:



The possible collision during the approach movement can be avoided by the automatic point modification.



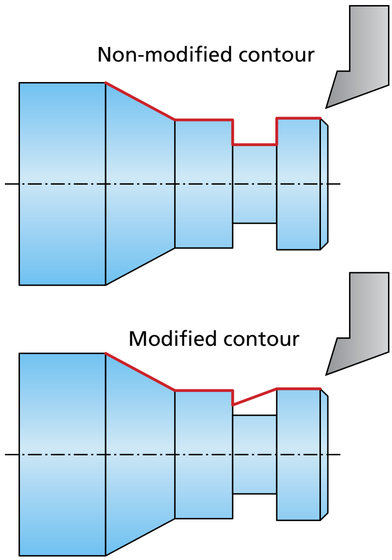
Note the **Modify contour to avoid collision** check box at the bottom of the **Partial Machining** dialog box.



Modify contour to avoid collision

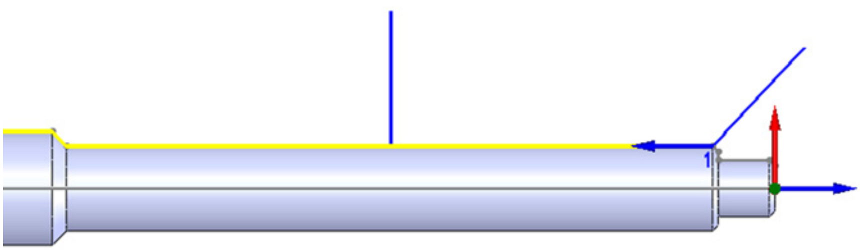
This check box enables you to perform the modification of the partial machining geometry.

When this check box is selected, the geometry modification is performed according to the tool shape. The modification enables you to avoid possible collisions between the tool and the geometry during the machining. During the geometry modification, the machining geometry is changed in areas where the possible collisions are detected. The geometry is changed taking into account the tool shape.



The **Modify contour to avoid collision** check box affects only the finish and semi-finish tool path when the **Profile** option is chosen for the **Work type**. When the **Rough** option is chosen, the geometry modification is automatically performed, irrespective of the state of the **Modify contour to avoid collision** check box.

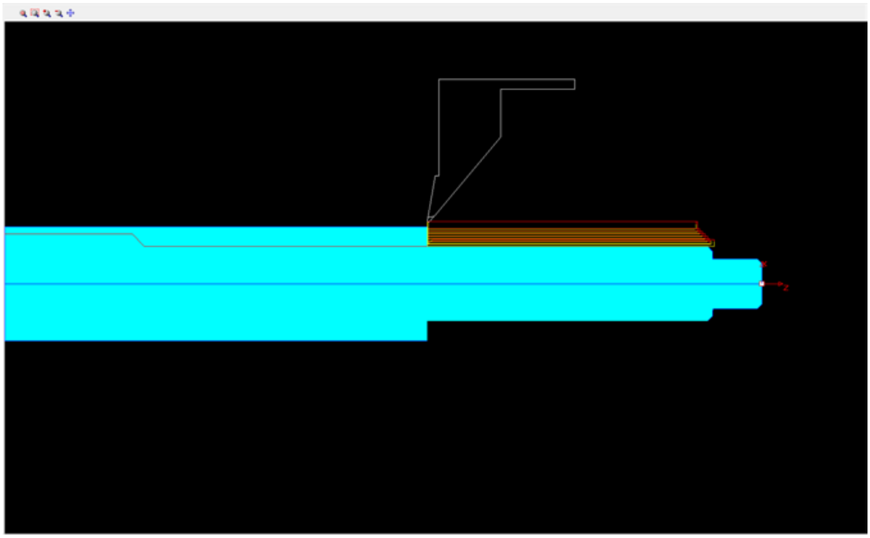
Click the **Show** button to check the machining geometry and the updated partial geometry between the **First point** and the **Second point** with the tangential arcs and connected lines.



Confirm the **Partial Machining** check box by clicking the **OK** button.

Click the **Save & Calculate** icon in the **Turning Operation** dialog box.

Simulate the tool path in the **Turning** mode.



3. Machine the second segment of the geometry

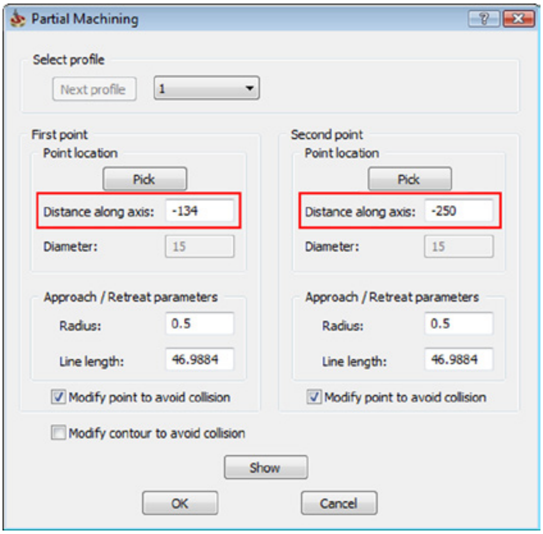
Click the **Save & Copy** icon in the **Turning Operation** dialog box. An additional Turning operation is added in **SolidCAM Manager**. This operation is a copy of the previous one and contains the same geometry and tool data. Define the Partial machining parameters to machine the next geometry segment.

Switch to the **Technology** page and click **Strategies** tab. Check the **Partial Machining** check box and click on the **Data** button. The **Partial Machining** dialog box is displayed.

Define the first point of the segment in such way that there will be an overlap between the surface machined in the previous operation and the one to be machined. In the previous operation, the length of the machined segment was 135 mm, i.e. the **Distance along axis** value for the **Second point** was **-135**. Set the **Distance along axis** value for the **First point** to **-134** to perform the 1 mm overlap in the machined surface.

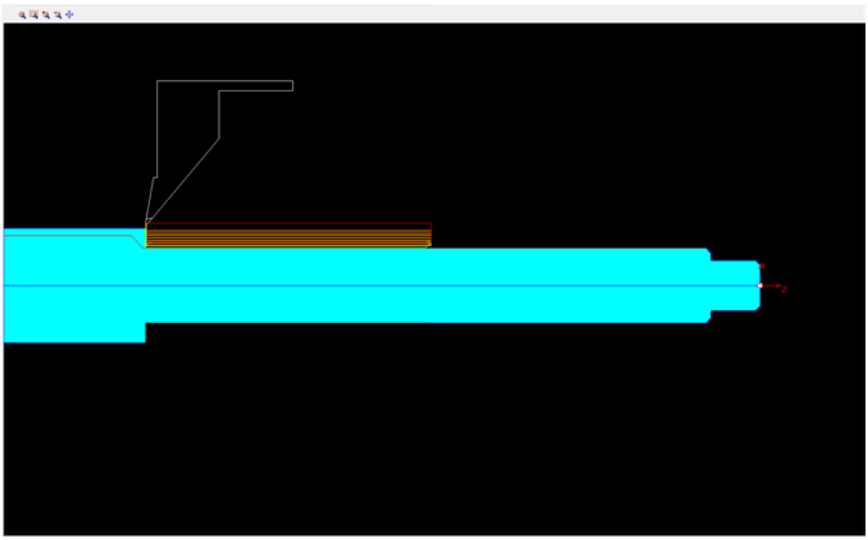
Click the **Pick** button in the **Second point** section. Pick the point as shown.

The **Distance along axis** value for the **Second point** is calculated: **-250**.



Confirm the **Partial Machining** dialog box. Click the **Save & Calculate** icon in the **Turning Operation** dialog box.

Simulate the tool path in the **Turning** mode.



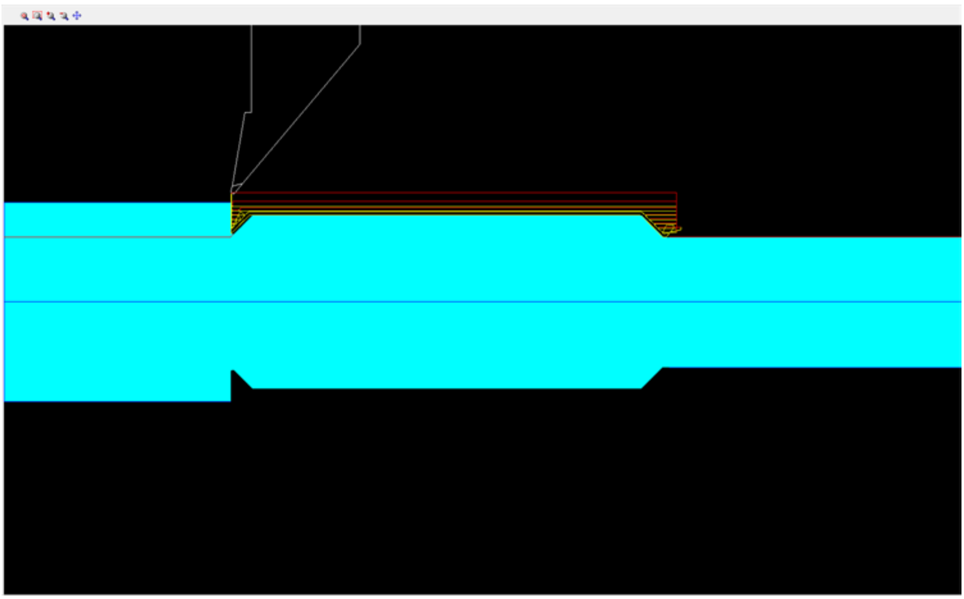
4. Machine the third segment of the geometry

In the same manner as explained in the previous step, add a new Turning operation by saving and copying the last one. Open the **Partial Machining** dialog box and set the **Distance along axis** value for the **First point** to **-249**. To define the **Second point**, click on the **Pick** button and select the point on the model as shown.

The **Distance along axis** value for the **Second point** is calculated: **-350**.

Confirm the **Partial Machining** dialog box. Click the **Save & Calculate** icon in the **Turning Operation** dialog box.

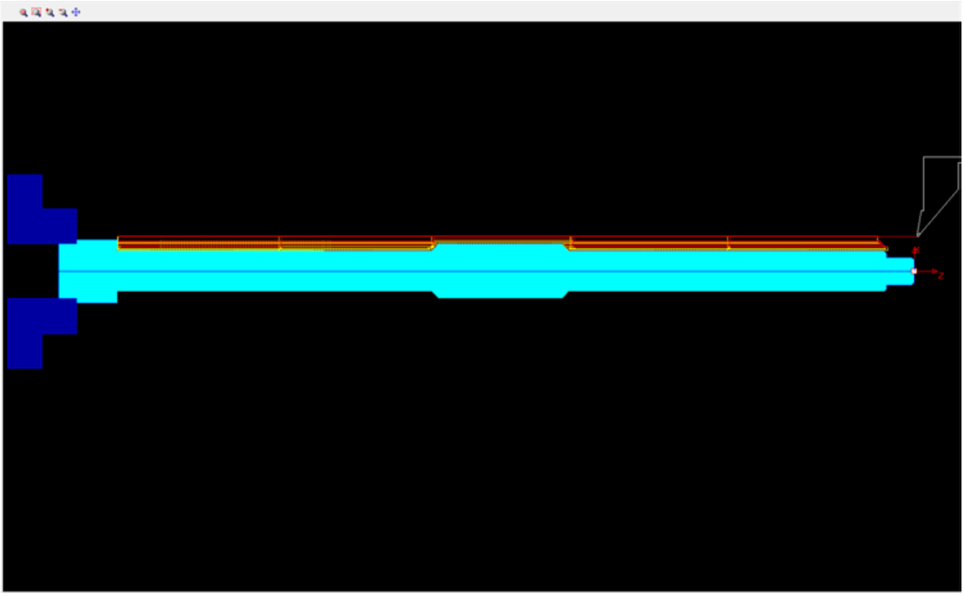
Simulate the tool path in the **Turning** mode.



5. Machine the rest of the geometry

In the same manner as explained in the previous steps, machine the rest of the geometry in two segments.

Simulate all of the Partial machining operations in the **Turning** mode.

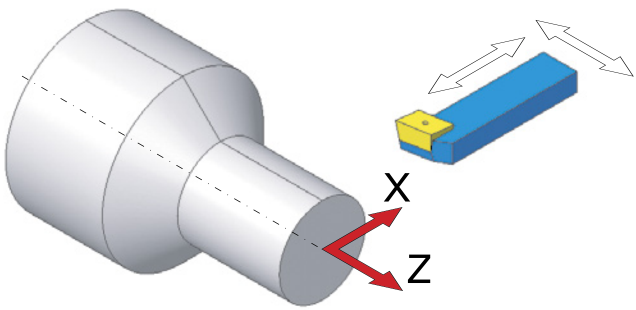


3.4 Turning on Mill-Turn CNC Machines

SolidCAM supports all types of Mill-Turn CNC-machines. The Mill-Turn CAM-Part is not strictly associated with the only type of machine on which it is intended to be manufactured.

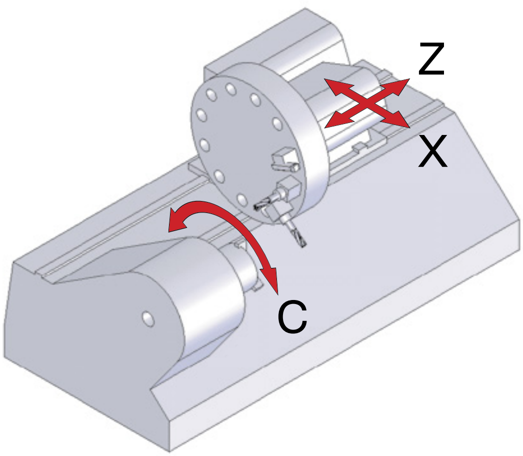
In the Mill-Turn module, the flexibility of CAM-Part definition in terms of CNC-machine usage enables you to easily change the type of machines for parts. The type of the chosen CNC-machine though applies certain restrictions on the technology: for example, if your programming is intended for a 3-axis CNC-machine that supports the XZC coordinates, the tool path is not supposed to contain movements along the Y-axis (except for the movements that can be translated into the available axes of the machine). Such functionality enables you to create a required machining technology and then to adapt it for usage on a certain CNC-machine.

All types of Mill-Turn CNC-machines enable you to perform turning operations.

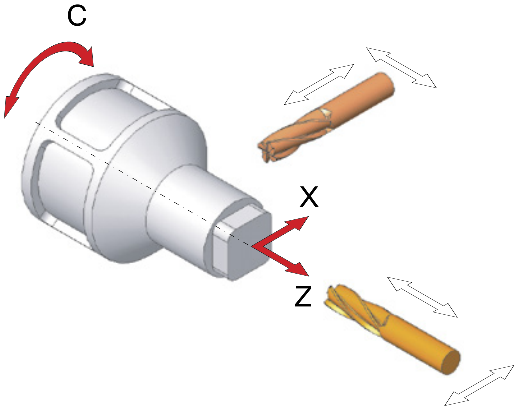


The following types of milling operations are supported according the CNC-machine type:

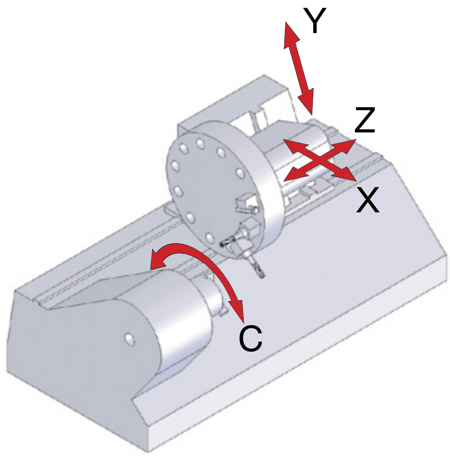
3-Axis Mill-Turn machine



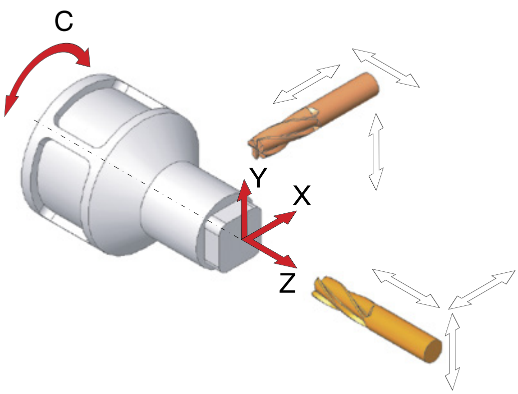
This type of machine enables you to perform facial, indexial and simultaneous milling operations using the XZC axes.



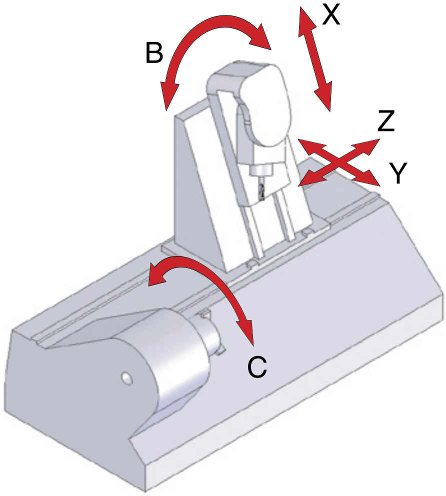
4-Axis Mill-Turn machine



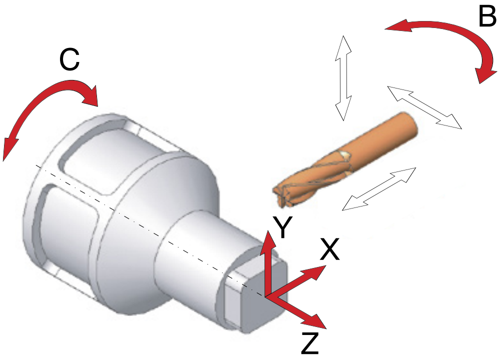
This type of machine enables you to perform facial, indexial and simultaneous milling operations using the XYZC axes.



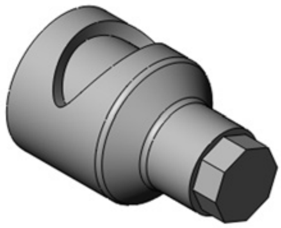
5-Axis Mill-Turn machine



This type of machine enables you to perform facial, indexial and simultaneous milling operations using the XYZCB axes.



Exercise #10: Turning Operation on Mill-Turn CNC-Machine

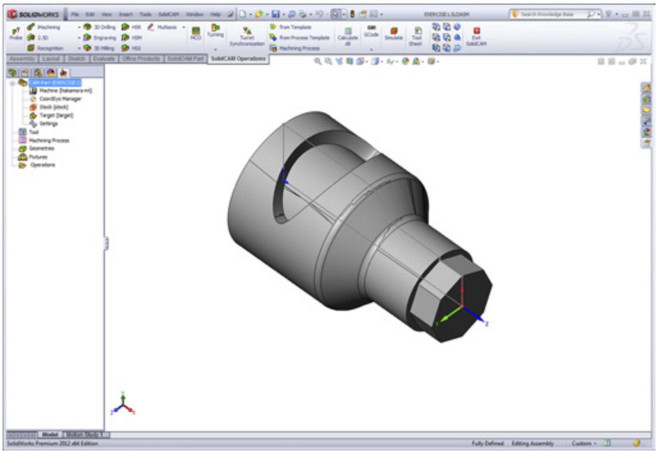


In this exercise, you have to define the machine setup and clamping fixture and to perform two Turning operations to machine the external faces of the model and prepare it for the next Milling operations.

The exercise uses the CAM-Part created in **Exercise #2**. In the process of operation definition, you have to define the machining geometry, the tool and the technological parameters.

1. Load the CAM-Part

Open the **Exercise2.prz** CAM-Part prepared earlier.



2. Define the machine setup

Right-click the **Operations** header in **SolidCAM Manager** and choose the **Machine Setup** ( ) command.

The **Machine setup** dialog box is displayed. This machine setup contains a submachine called **Spindle_Table** with the Machine Coordinate System **MAC 1**.

The **Fixture** column enables you to choose or define the fixture to be used with the specified submachine. To define the fixture, double-click the cell, click the arrow and choose ... .

The **Model** dialog box is displayed. This dialog box enables you to define the fixture geometry.

Make sure that the **Chuck (Standard)** option is chosen in the **Defined by** section.

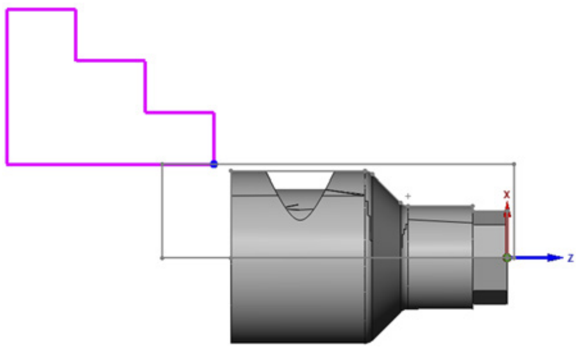
In the **Clamping method** section, use the default **Main** option and make sure that the default clamping option is chosen.

The default Chuck position is defined on the edge of the stock envelope created during the Stock model definition.

- Set the **Clamping diameter** to **54**
- Set the **Axial position** to **-85**

Set the values in the **Jaws parameters** section as follows:

- Set the **Jaw width (JW)** value to **60**
- Set the **Jaw height (JH)** value to **45**
- Set the **Step width (SW)** value to **20**
- Set the **Step height (SH)** value to **15**



The clamping fixture is defined.

Click  to confirm the **Model** dialog box.

The **Machine setup** dialog box is displayed again.

Now you have to define the model position relative to the submachine coordinate system.

Double-click the cell in the **Z** column. Set the value to **125**.

Click the **Preview** icon  to see the changes. The model is located now in 125 mm to the right allowing better visualization during the machine simulation mode.

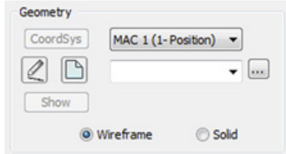
Confirm the Setup definition with **OK**. The **Setup** subheader is added to the **SolidCAM Manager** tree under the **Operations** header.

3. Add an operation

Right-click the **Operations** header in **SolidCAM Manager** and choose **Turning** from the **Add Turning Operation** submenu.

The **Turning Operation** dialog box is displayed.

4. Define the Geometry



Make sure that the **Wireframe** option is chosen on the **Geometry** page of the **Turning Operation** dialog box.

The turning geometry has to be located in the **ZX-plane** and can be defined by selecting wireframe elements or solid model entities such as faces, edges and vertices.

The following geometry definition options are available:

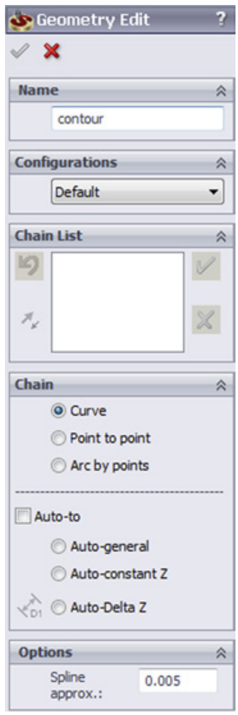
- **Wireframe**

  This option enables you to define the turning geometry by wireframe geometry selection.

- **Solid**

  This option enables you to define the turning geometry by selecting model entities such as faces, edges, vertices, origin and sketch points.

When model entities are picked, SolidCAM automatically defines the geometry on the envelope/section segment corresponding to the selected model elements.



You have to define the machining geometry for the turning operation using the **Envelope** sketch that was automatically generated during the **Target model** definition described in **Exercise #1**.

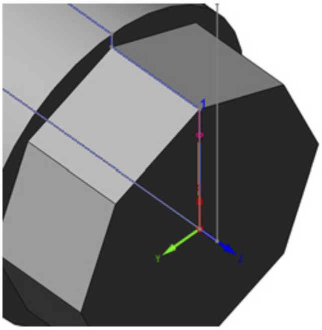
Click the **New** button.

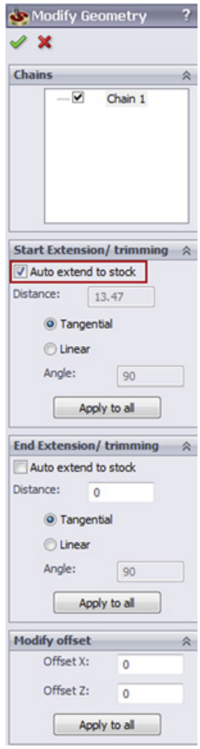
The **Geometry Edit** dialog box is displayed in the SolidWorks **PropertyManager** area. This dialog box enables you to define and edit geometry chains.

The default **Curve** option enables you to create a chain by selecting model edges or wireframe sketch elements one after the other.

In this operation, the geometry must be directed opposite to the X-axis direction.

Click on the entity of the Envelope sketch as shown. The selected entity is highlighted.



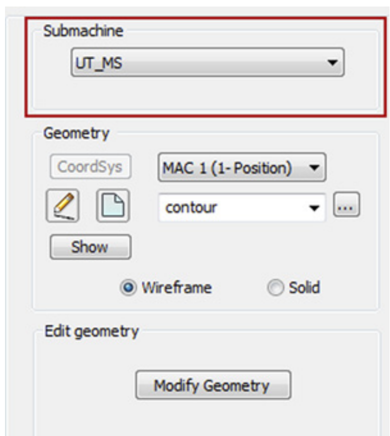


Click the **Accept Chain** button to finish the chain definition and confirm the **Geometry Edit** dialog box with the button.
In this exercise, the geometry has to be extended in order to enable the tool approach and retreat outside of the machining geometry. On the **Geometry** page of the **Turning Operation** dialog box, click the **Modify Geometry** button.
The **Modify Geometry** dialog box is displayed.
Makes sure that in the **Start Extension/trimming** section, the **Auto extend to stock** option is selected.
Close the **Modify Geometry** dialog box with the button.

5. Define the Submachine

To generate the correct GCode for Turning operations, you need to choose an appropriate submachine. Submachine is a combination of turret and table on the CNC-machine that will be used to perform this operation.

Make sure that the **UT_MS** option is chosen in the **Submachine** section on the **Geometry** page of the **Turning Operation** dialog box. The structure and properties of this submachine are available in the Machine ID file of the chosen machine.



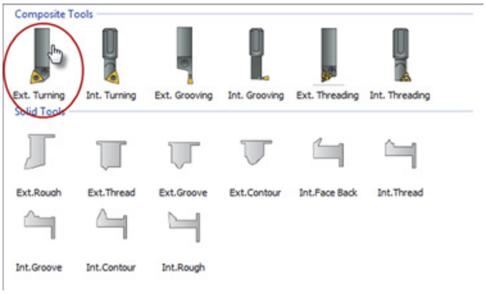
6. Define the Tool

Now you have to define the tool for the turning operation.

Click the **Select** button on the **Tool** page of the **Turning Operation** dialog box.

The **Part Tool Table** is displayed.

Click  . The **Part Tool Table** provides you with the choice of tools available for the current operation. Choose the **Ext. Turning** tool from the **Composite Tools** section.



In the **Insert** page, define the following parameters:

- Set the **Insert Shape** to **W (80°)**
- Set the **IC Diameter** to **06 (9.52mm)**
- Set **Thickness** to **T3 (3.97mm)**
- Set **Corner Radius** to **04 (0.40mm)**

In the **Shank** page, define the following parameters:

- Set the **Insert Lead Angle** to **L**
- Set the **M** value to **32**

For a more detailed explanation of the turning tools parameters, refer to the **SolidCAM Turning Online Help**.

Click the **Select** button to choose the defined tool for the current Turning operation.

The **Turning Operation** dialog box is displayed.

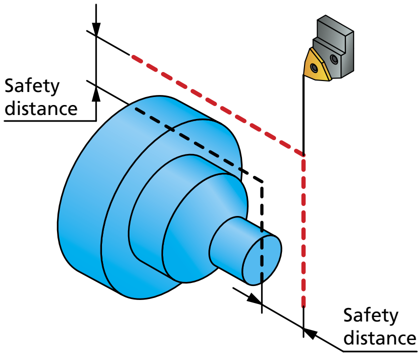
7. Define the machining levels

Switch to the **Levels** page of the **Turning Operation** dialog box. This page enables you to specify the machining levels for the operation.

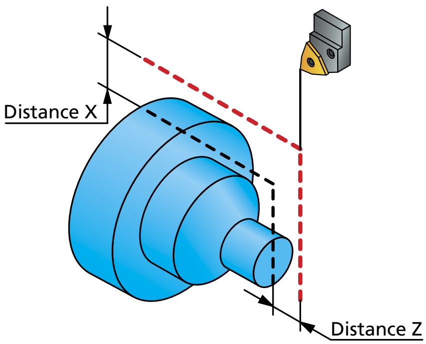
Use the default **Safety distance** value (**2**).

Safety distance

This parameter defines the distance from the workpiece, until which the tool movements are performed in the Rapid mode. Upon reaching the Safety distance, the tool switches to the working feed and performs machining of the part. In this manner, the tool path passes are extended to ensure that the tool does not drop into the material in the Rapid mode in the beginning of machining and between one step down to another.



When the **Custom safety distance** check box is selected, SolidCAM enables you to define the distance at which the tool is positioned relative to the material in the beginning and in the end of the operation. The **Distance X** and **Distance Z** parameters enable you to define different distance values in the X- and Z-directions.



8. Define the technological parameters

Switch to the **Technology** page of the **Turning Operation** dialog box.

In the **General** tab, make sure that the **Rough** option is chosen in the **Work type** area.

Make sure that the **Face front**  option is chosen in the **Mode** area. This option enables you to perform **face turning** (the principal working direction is the X-axis direction).

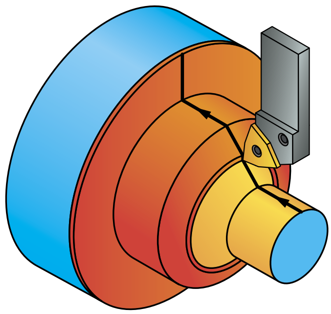
Use the default parameters of the Rough machining defined in the **Rough** tab on the **Technology** page.

To define the parameters of the finishing machining, switch to the **Semi-finish/finish** tab on the **Technology** page.

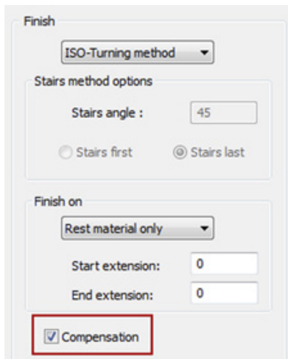
Choose the **ISO-Turning method** option from the **Finish** list.

Finish

A finishing pass is a single pass that is executed in the end of the operation. When the **ISO-Turning method** option is chosen, the finishing pass is performed as follows: the tool moves in the direction of the geometry; continuous tool path is calculated along the geometry in the specified direction. The direction of machining is maintained throughout the tool path.



In the **Finish** section, choose the **Compensation** option.



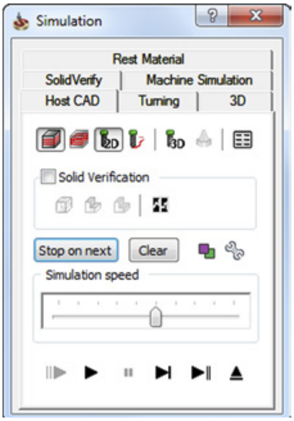
9. Calculate the operation

Click the **Save & Calculate** icon in the **Turning Operation** dialog box. The operation data is saved, and the tool path is calculated.

10. Simulate the operation in the Host CAD mode

Click the **Simulate** icon in the **Turning Operation** dialog box.

The **Simulation** control panel is displayed.

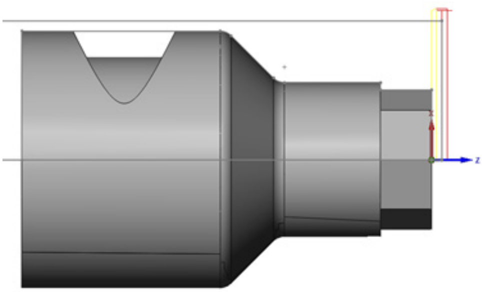


Host CAD simulation mode

This mode enables you to display the tool path directly on the model in the SolidWorks window. Since all the View options of SolidWorks are available during the simulation, you can see the tool path from different perspectives and zoom in on a certain area of the model.

Rotate the model to the side view by clicking the icon of the **CAM Views** toolbar and click the **Play** button on the **Simulation** control panel.

The tool path is simulated in the SolidWorks window.



Note that the lead in and lead out movements and work feed movements are displayed with different colors.

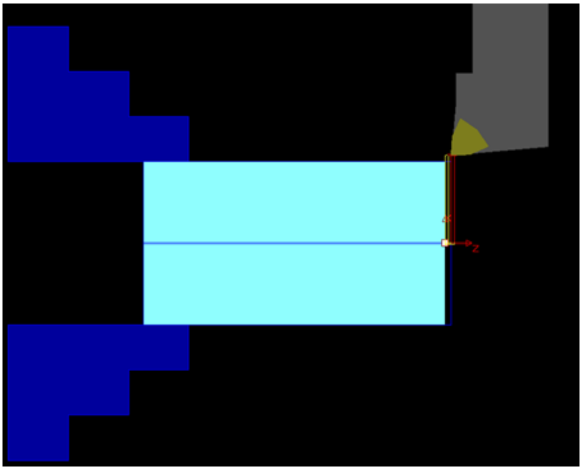
11. Simulate the operation in the Turning mode

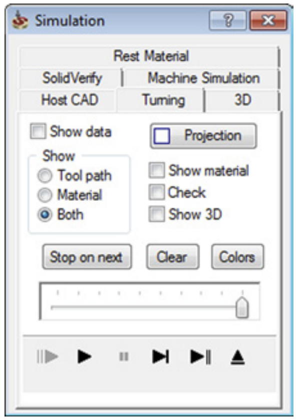
Switch to the **Turning** page on the **Simulation** control panel.

Turning simulation mode

This mode enables you to display the section view simulation of the turning tool path.

Click the **Play** ▶ button. The tool path is simulated.





12. Simulate the operation in the SolidVerify mode

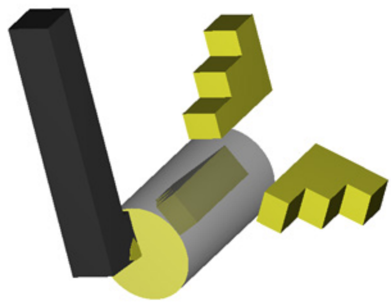
Switch to the **SolidVerify** page on the **Simulation** control panel.

SolidVerify simulation mode

This mode displays the simulation of machining on the solid model. The 3D Stock model is used in this mode with the clamping fixture displayed. During the machining simulation, SolidCAM subtracts the tool movements from the solid model of the stock using solid Boolean operations. The remaining machined stock is a solid model that can be dynamically zoomed or rotated.

The stock model is displayed in the simulation window. Rotate the model with the mouse and click the **Play** button.

The simulation is performed.



During the **SolidVerify** simulation, you can use the **View** buttons to place the simulation model properly on the screen.

The **Isometric View** button displays the isometric view of the simulation model.

The **Top View** button displays the top view of the simulation model.

The **Front View** button displays the front view of the simulation model.

The **Move** button moves the simulation model to any point on the screen. This operation is also available by click-and-drag of the middle mouse button.

The **Rotate** button rotates the simulation model. This operation is also available by applying the combination of the **Ctrl** button and click-and-drag of the middle mouse button.

The **Zoom** button zooms in and out the image on the screen. This operation is also available by applying the combination of the **Shift** button and click-and-drag of the middle mouse button.

13. Close the simulation

Click the **Exit** button on the **Simulation** control panel.

The **Turning Operation** dialog box is displayed.

14. Close the Turning Operation dialog box

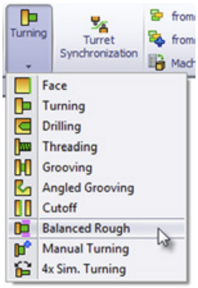
In the **Turning Operation** dialog box, click the **Exit** icon.

The dialog box is closed.

15. Add an operation

Click the last Turning operation in **SolidCAM Manager** and choose **Balanced Rough** from the **Turning** submenu on the **SolidCAM Operations** ribbon.

The **Balanced Rough operation** dialog box is displayed.



16. Define the Geometry

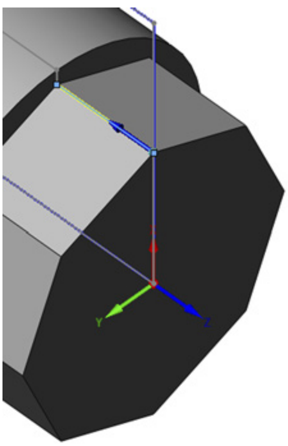
Select the submachine: choose **UT_MS** as a Master submachine and **LT_MS** as a Slave submachine. Make sure that the **Wireframe** option is chosen in the **Geometry** section.

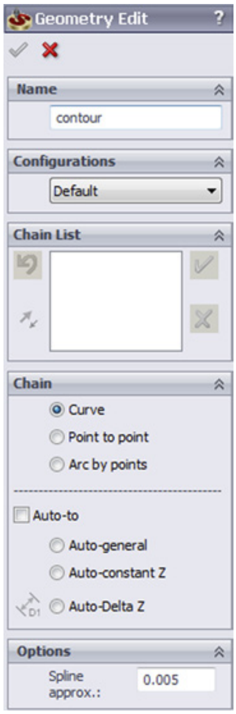
Click . The **Geometry Edit** dialog box is displayed.

The default **Curve** option enables you to create a chain by selecting model edges or wireframe sketch elements one after the other.

In this operation, the geometry must be directed opposite to the Z-axis direction.

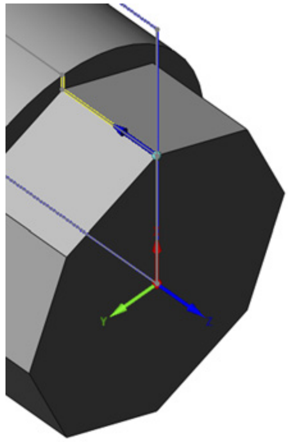
Click on the entity of the Envelope sketch as shown. The selected entity is highlighted.





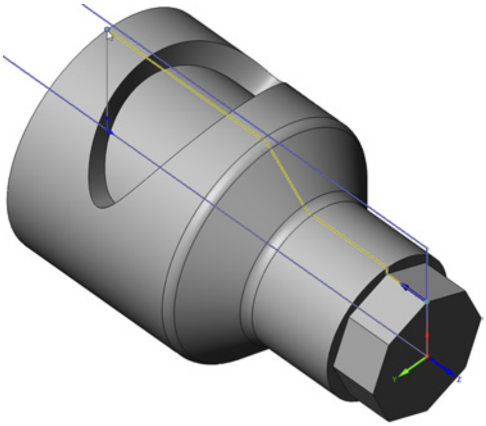
Click on the next sketch entity as shown.

In the **Geometry Edit** dialog box, select the **Auto-to** check box and choose the **Auto-general** option.

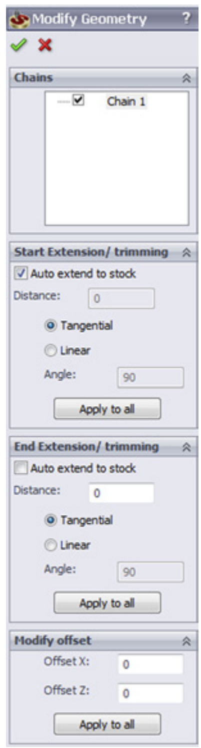


With the **Auto-to** and **Auto-general** options, define the end point of the chain. Click on the vertex as shown.

The complete selected chain is highlighted.

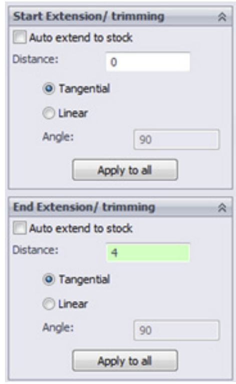


Click the **Accept Chain**  button to finish the chain definition and confirm the **Geometry Edit** dialog box with the  button.
In this exercise, the geometry has to be extended in order to enable the tool approach and retreat outside of the machining geometry. On the **Geometry** page of the **Turning Operation** dialog box, click the **Modify Geometry** button.
The **Modify Geometry** dialog box is displayed.



In the **Start Extension/trimming** section, set the **Distance** value to **0**. In the **End Extension/trimming** section, set the **Distance** value to **4**.

Close the **Modify Geometry** dialog box with the button.



17. Define the Main Tool

In the Balanced Rough operation, two tools are used simultaneously.

The **Tool** page enables you to select the tool used with the upper turret (UT). Click the **Select** button.

The **Part Tool Table** is displayed.

Use the same tool as in the previous operation. Click the **Select** button to choose it for operation.

18. Define the Slave Tool

Switch to the **Slave Tool** page. This page enables you to select the tool used with the lower turret (LT). Click the **Select** button.

The **Part Tool Table** is displayed.

Click . Choose the **Ext. Turning** tool from the **Composite Tools** section. Use the default tool parameters. Set **Mounting** to . Click the **Select** button to choose it for operation.

The **Balanced Rough operation** dialog box is displayed.

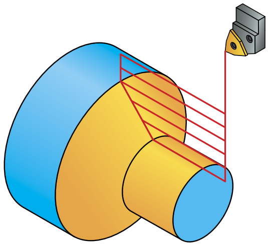
19. Define the technological parameters

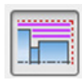
Switch to the **Technology** page of the **Balanced Rough operation** dialog box.

In the **General** tab, make sure that the **Rough** option is chosen in the **Work type** area.

Rough Work type

The tool path movements are parallel to the Z-axis (longitudinal turning) or to the X-axis (facial turning). Semi-finishing and finishing passes are performed, if chosen, at the end of the rough stage.



Make sure that the **Long external**  option is chosen in the **Mode** area. This option enables you to perform **longitudinal turning** (the principal working direction is the Z-axis direction).

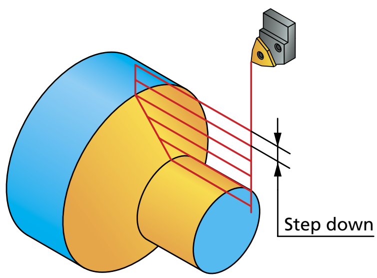
In the **Balanced mode** section, choose the **Trailing** option. With this option, one of the tools starts working before the other tool, so that each of them turn different diameters.

Set the **Distance** value to **5** to define the distance on the Z-axis between two tools at their start points.

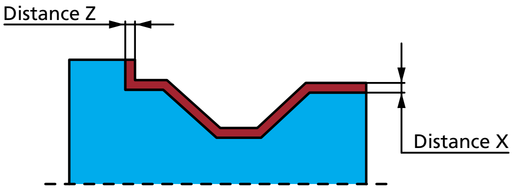
To define the parameters of the Rough machining, switch to the **Rough** tab on the **Technology** page of the **Turning Operation** dialog box.

Select the **Smooth** option in the **Rough type** section.

Define the **Step down** for roughing. This parameter enables you to define the distance between each two successive roughing passes. Use the default value of **1**.



Define the offset from the geometry that will remain after the roughing stage of the operation. In the **Rough offset** section, choose the **ZX-ABS** option. This option enables you to define different offsets from the geometry in the X- and Z-axis directions. The program chooses the sign of the delta-X and delta-Z vector components in such a way that the offset geometry does not intersect with the profile geometry.



Use the default **Distance X** value of **0.6** and **Distance Z** value of **0.2**.

20. Calculate the operation

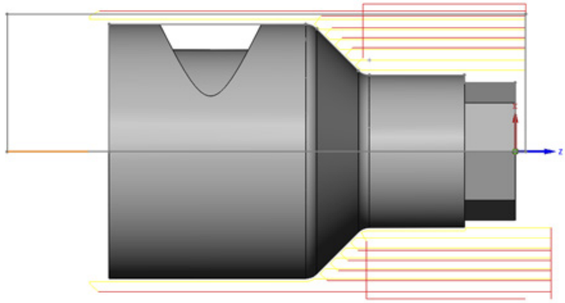
Click the **Save & Calculate** icon in the **Turning Operation** dialog box. The operation data is saved and the tool path is calculated.

21. Simulate the operation in the Host CAD mode

Click the **Simulate** icon in the **Turning Operation** dialog box.

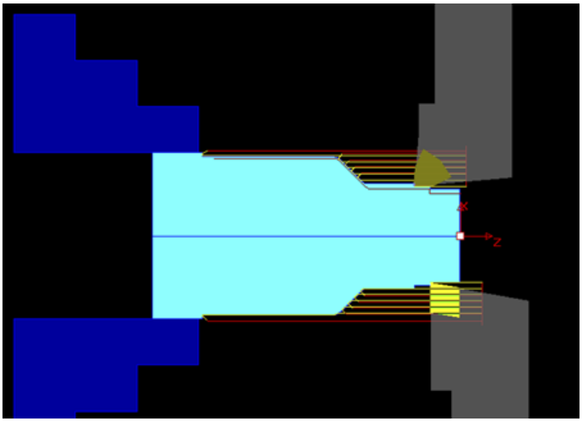
The **Simulation** control panel is displayed.

Play the simulation in the Host CAD mode.



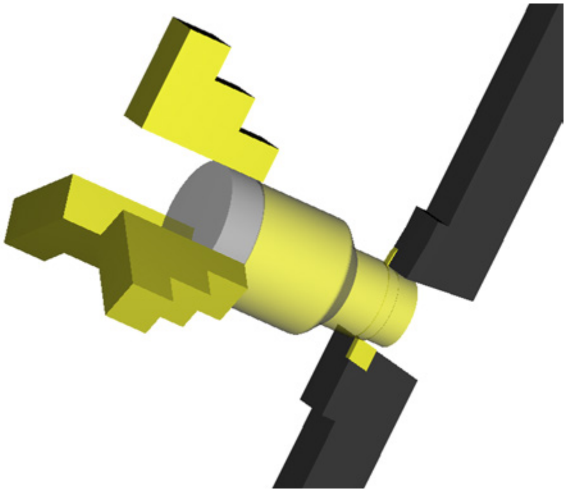
22. Simulate the operation in the Turning mode

Switch to the **Turning** page on the **Simulation** control panel. Play the simulation in the Turning mode.



23. Simulate the operation in the SolidVerify mode

Switch to the **SolidVerify** page on the **Simulation** control panel. Play the simulation in the SolidVerify mode.



24. Close the simulation

Click the **Exit** button on the **Simulation** control panel.

The **Turning Operation** dialog box is displayed.

25. Close the Turning Operation dialog box

In the **Turning Operation** dialog box click the **Exit** icon.

The dialog box is closed.

26. Close the CAM-Part

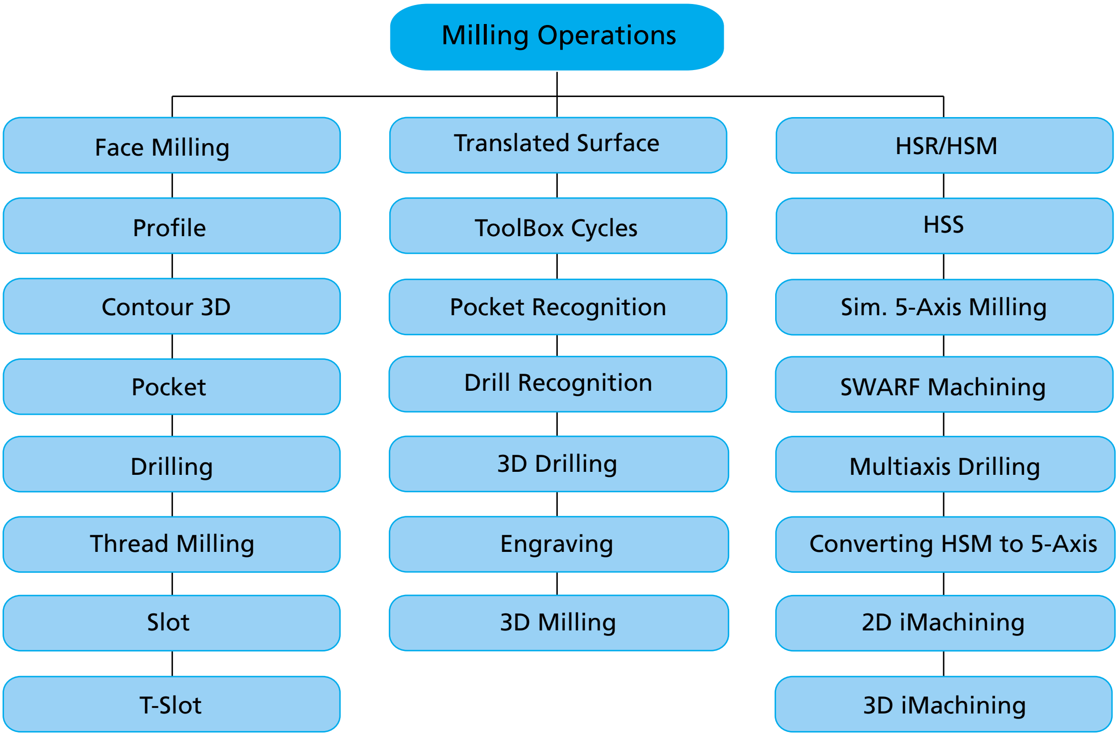
Right-click the **CAM-Part** header in **SolidCAM Manager** and choose **Close** from the menu.

The CAM-Part is closed.

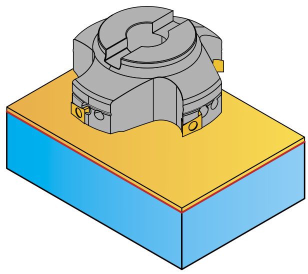
Milling on Mill-Turn CNC-Machines

4

The Mill-Turn module enables you to use all types of milling operations to generate the tool path for the driven tools.



Face Milling Operation

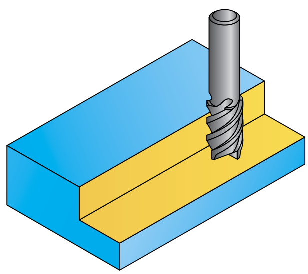


This operation enables you to machine large flat surfaces with face mill tools.

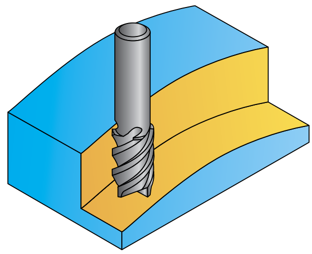
Profile Operation

You can mill on or along a contour. The profile geometry can be open or closed. In profile milling you can optionally use tool radius compensation to the right or to the left side of the geometry. SolidCAM offers two types of profiling:

- Milling a single profile to the specified constant or variable depth in one step or in several user-defined down steps.
- Concentric profiles to the specified constant or variable depth; this type of profiling generates several concentric profiles that start from the defined clear offset distance from the profile, and finish on the profile geometry, thus clearing the area around the profile to a constant depth.



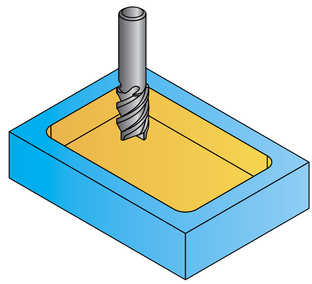
Contour 3D Operation



This operation enables you to utilize the power of the 3D Engraving technology for the 3D contour machining. In this operation, SolidCAM enables you to prevent the gouging between the tool and the 3D contour.

Pocket Operation

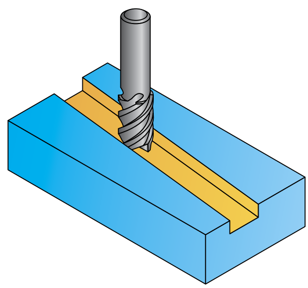
In pocket milling, you remove material from the interior of a closed geometry. SolidCAM offers two types of pocketing:



- **Pocket without islands**. When a profile geometry consists of one or more profiles and none of them are enclosed or intersect with one another, each is milled as a separate pocket without islands.
- **Pocket with islands.** When a profile geometry consists of several profiles, any profile that is enclosed or intersects with another profile is treated as an island. You can define an unlimited number of islands within a single pocket.

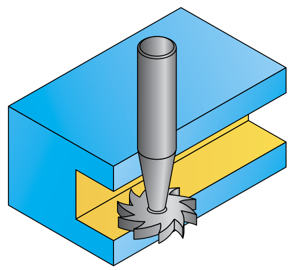
Slot Operation

This operation generates a tool path along the centerline to the right or to the left of one or more profiles. Two types of slots can be defined: the Slot with constant depth operation machines the slot in several steps until the final depth is reached. In Slot with variable depth, the depth profile is also defined by a 2D section. The slot can be pre-machined using rough and semi-finish cycles. The finish cut produces a tool path according to the specified scallop height on the floor of the slot. With available parameters for the right and left extension and the side step, you can mill a slot wider than the tool diameter.

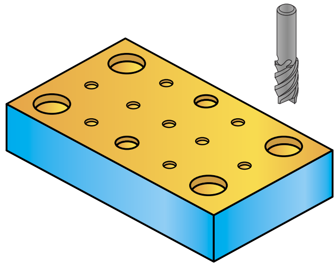


T-Slot Operation

This operation enables you to machine slots in vertical walls with a slot mill tool.

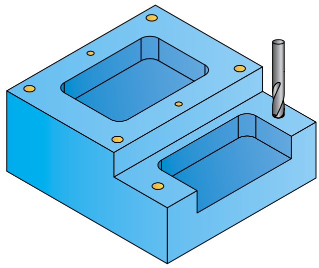


Drilling Operation



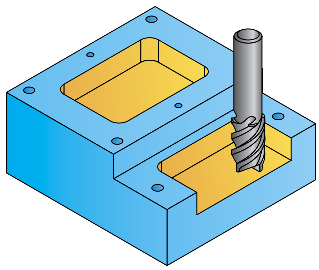
This operation enables you to perform drills and other canned drill cycles. SolidCAM supports the canned drill cycles provided by your particular CNC-machine such as threading, peck, ream, boring, etc. If your CNC-machine has no canned drill cycles of its own, they can be defined using the General Pre- and Post-processor program (GPPTool).

Drill Recognition Operation



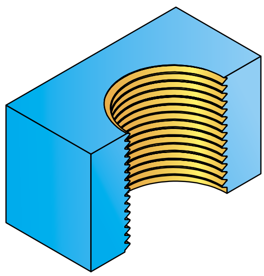
This operation performs powerful drill feature recognition and automatic Drill geometry creation using the AFRM module functionality. This operation enables you to handle separate sets of Milling levels for each drill position. The initial values of the Milling levels sets are automatically recognized from the model; they can be edited by the user.

Pocket Recognition Operation



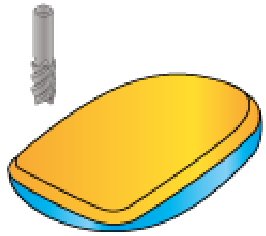
This operation enables you to recognize the pocket features on the solid model and perform the machining of these features.

Thread Milling Operation



This operation enables you to generate a helical tool path for the machining of internal and external threads with thread mills.

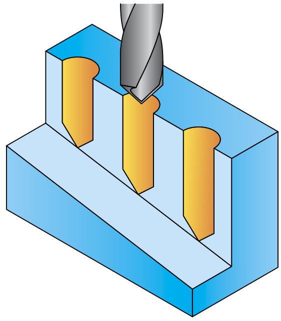
3D Milling Operation



You can perform 3-axis gouge-free machining on solid and surface models. This operation offers a wide range of roughing, semi-finishing and finishing strategies for free-form models. It can be used to manufacture molds, dies, electrodes, prototypes and other 3D Models.

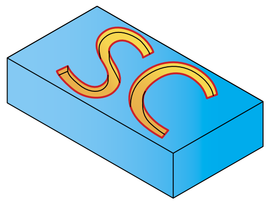
The HSM module offers unique machining and linking strategies for generating high-speed tool paths.

3D Drilling Operation

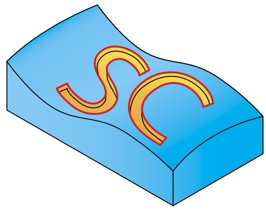


This operation enables you to perform drills and other canned drill cycles, while taking into account the solid model geometry. SolidCAM uses the holes prepared in this operation for the descent during 3D Model roughing.

Engraving Operations

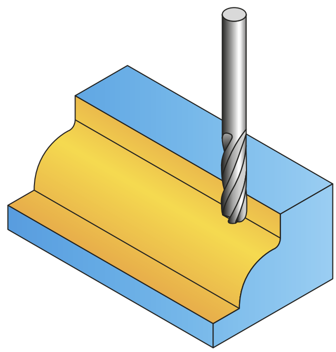


You can mill text or any other profile on a 2D or 3D geometry. The profile is projected on the surface, engraving the contour at a specified depth.

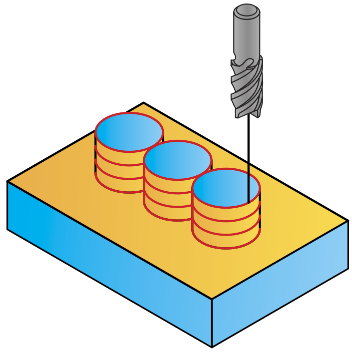


Translated Surface Operation

A translated surface is generated by moving a section along a profile geometry. Limit geometries can be projected on the translated surface. You can machine the resulting translated surface inside, outside or along the limit geometry.

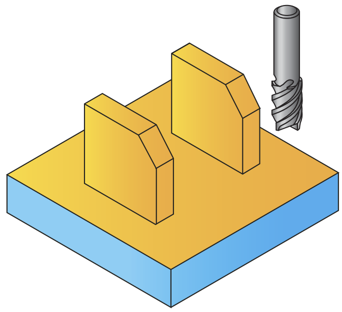


ToolBox Cycles



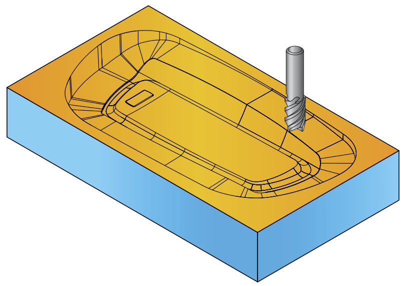
ToolBox sub-operations are a set of 2.5D Milling operations, and each one of these operations is intended for a specific machining case. For example, a number of operations cover different strategies of the counterbore machining. Each operation provides you with a specific machining strategy optimal for a particular machining case.

HSR/HSM Operation



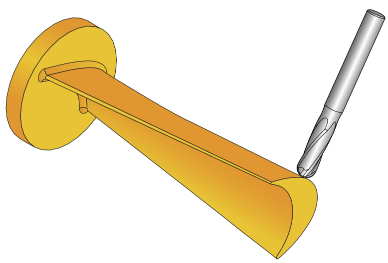
SolidCAM HSR/HSM module smooths the paths of both cutting moves and retracts wherever possible to maintain a continuous machine tool motion – an essential requirement for maintaining higher feed rates and eliminating dwelling. The retracts to high Z-levels are kept to a minimum. Angled where possible, smoothed by arcs, retracts do not go any higher than necessary, thus minimizing air cutting and reducing machining time. The result of HSM is an efficient, smooth, and gouge-free tool path.

HSS Operation



SolidCAM HSS module is a high speed surface machining module for smooth and powerful machining of localized surface areas in the part, including undercuts. It provides easy selection of the surfaces to be machined, with no need to define the boundaries. It supports both standard and shaped tools.

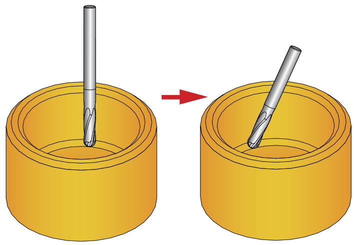
Sim. 5 Axis Operation



This operation enables you to perform simultaneous 5-axis machining. 5-axis machining strategies enable the use of SolidCAM for machining of complex geometry parts such as mold cores and cavities, aerospace parts, cutting tools, cylinder heads, turbine blades and impellers. SolidCAM enables collision checking between the tool and the machine components.

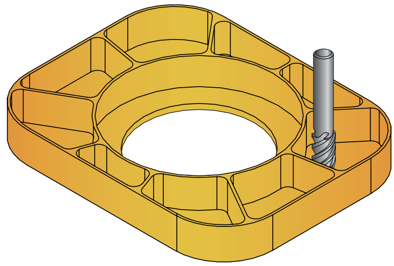
Converting HSS/HSM to Sim. 5 Axis Operation

In some machining cases, there is a need to perform machining of 3D parts using the 5-axis capabilities. For example, 3D machining of deep cavities requires the use of tools of great length, which can cause tool breakage; the same cavities can be machined using a tool of smaller length while tilting this tool to follow the same tool path.



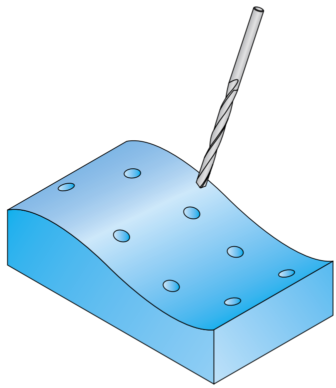
2D iMachining / 3D iMachining

SolidCAM **iMachining** is an advanced technology of high speed machining that generates optimized tool paths, efficient cutting and high material removal rates, while reducing machining time and tool wear. The **iMachining Technology Wizard** enables you to define the optimum cutting conditions for sustainable high speed machining. Using the cutting conditions produced by the Wizard, the **iMachining Tool Path Generator** calculates all the tool paths necessary to efficiently produce the target geometry from the given stock geometry.



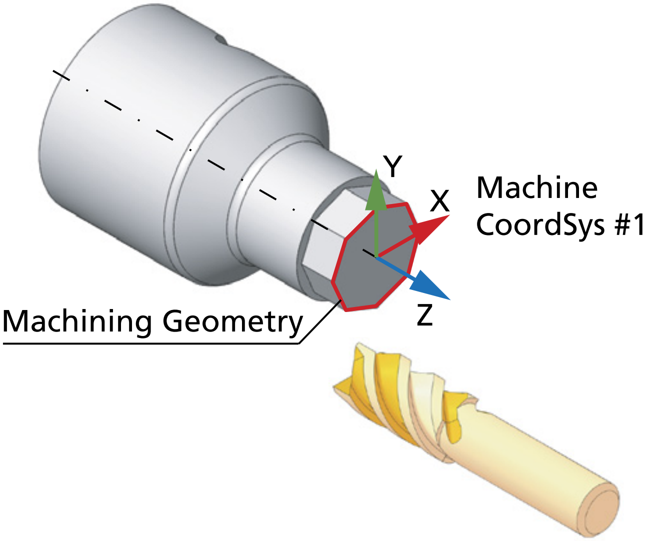
Multiaxis Drilling Operation

This operation enables you to machine a series of drills that have different orientations.

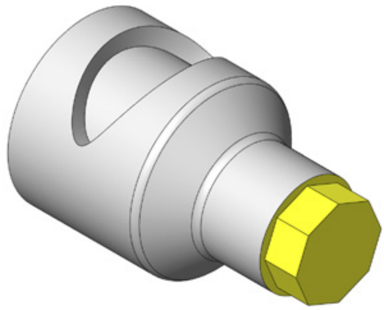


4.1 Facial Milling

SolidCAM enables you to perform facial milling on Mill-Turn CNC-machines. In facial milling, the tool axis is parallel to the revolution axis of the part. **Machine CoordSys #1** with the Z-axis collinear with the part revolution axis is used for this operation. The Geometry is defined directly on the solid model.



Exercise #11: Facial Milling



This exercise illustrates the capabilities of facial milling. The CAM-Part prepared in the previous exercises is used. During the facial milling, the octagon and the adjacent faces of the model are machined using the Profile operation.

1. Load the CAM-Part

Load the **Exercise10.prz** CAM-Part prepared in the previous exercises.

2. Add an operation

Right-click the last operation in the **SolidCAM Manager** tree and choose **Profile** from the **Add Milling Operation** submenu.

The **Profile Operation** dialog box is displayed.

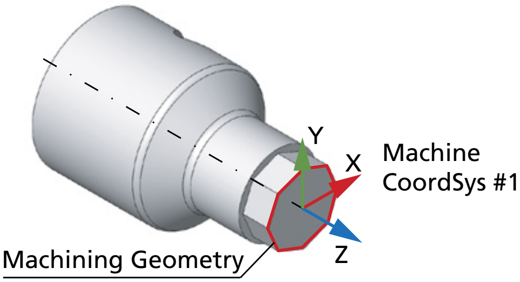
3. Define the Geometry

Click on the **Geometry** page.

The **Geometry** section enables you to choose the appropriate Coordinate System for the operation and define the machining geometry.

The **Machine CoordSys #1** is chosen by default for the operation. This Coordinate System enables you to perform the facial milling operation. The tool axis is parallel to the Z-axis of the **Machine CoordSys #1**.

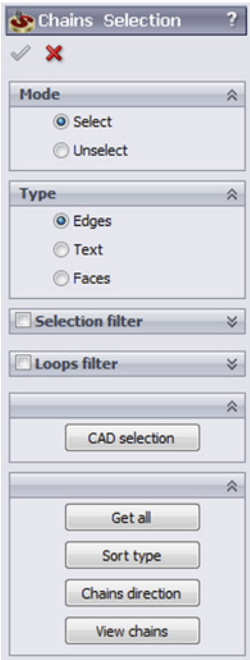
The Geometry has to be located in the plane parallel to the **XY-plane** of the **Machine CoordSys #1**.

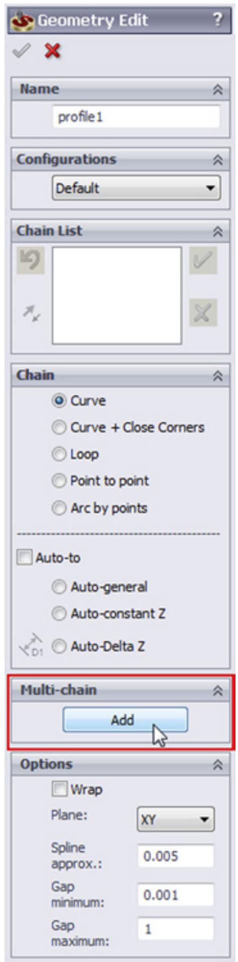


The **Geometry Edit** dialog box is displayed.

In the **Multi-Chain** section, click the **Add** button.

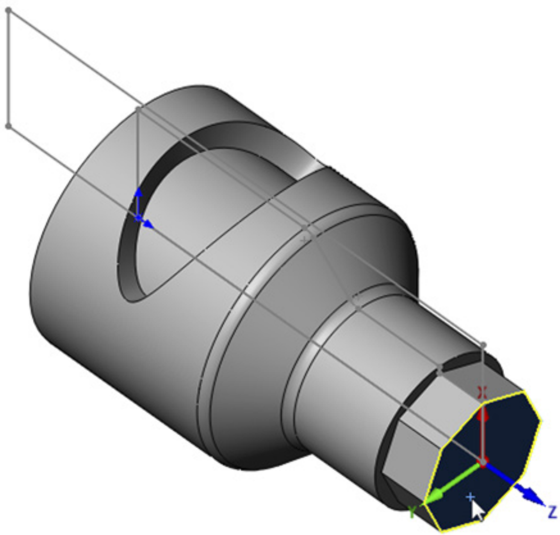
The **Chains Selection** dialog box is displayed in the SolidWorks **PropertyManager** area.





This dialog box enables you to define several chains by selecting the model elements. The chains are created automatically from the selected elements.

Click on the front face of the model octagon as shown.



The face is selected.

Click the  button to confirm your selection.

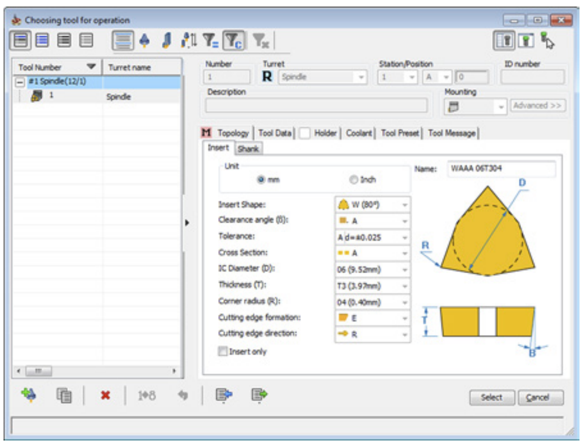
The chain is generated, and the chain icon is displayed in the **Geometry Edit** dialog box.

Confirm the chain definition by clicking the  button. The geometry is defined for the operation.

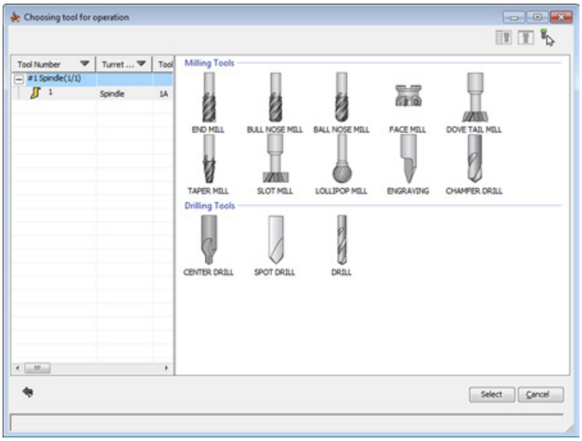
4. Define the Tool

Click the **Select** button on the **Tool** page.

The **Part Tool Table** is displayed. It shows all the tools that can be used in the CAM-Part machining. Currently, the **Part Tool Table** contains only one tool used for the Turning operations executed in the previous exercise. For this operation, you have to define a new milling tool.

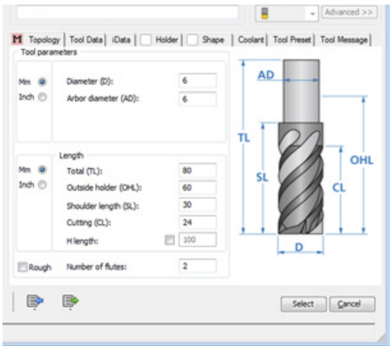


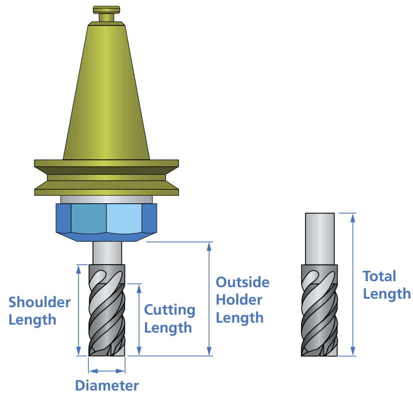
Click the **Add Milling Tool** icon (). The **Part Tool Table** provides you with choice of tools suitable for profile milling.



Choose the **End mill** tool from the **Milling Tools** section.

A new milling tool is added to the **Part Tool Table**.





5. Edit the Tool parameters

By default, SolidCAM offers you a **Ø6 End mill** tool. Edit the tool definition as follows:

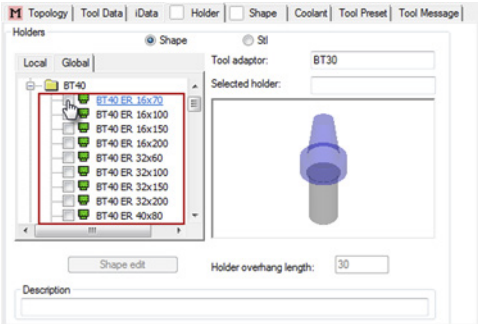
- Set the **Diameter** to **10**
- Set the **Total** length to **60**
- Set the **Outside holder** length to **50**
- Set the **Shoulder length** to **40**
- Set the **Cutting** length to **35**

Now you need to define the tool holder.
In the **Part Tool Table**, switch to the **Holder** page.

The **Global** tab displays the **Global holders table**.

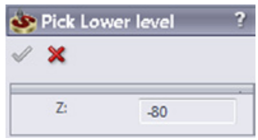
The Global holders table is supplied within SolidCAM. This table contains a number of frequently used tool holder components. The Global holders table can be modified by the user.

Expand the **BT40** header and choose the **BT40 ER16x70** tool holder from the list by selecting its check box.



Click the **Select** button in the **Part Tool Table** to confirm the tool definition.

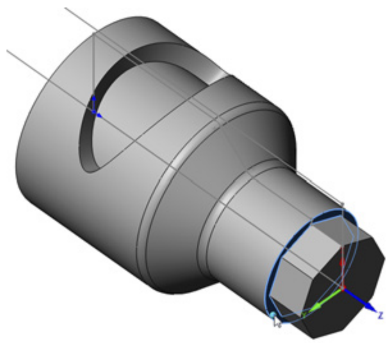
6. Define the Machining depth



Switch to the **Levels** page and click the **Profile depth** button.

The **Pick Lower level** dialog box is displayed.

SolidCAM enables you to define the Lower level directly on the model. The Profile depth is calculated automatically relative to the Upper level.



Click on the model face as shown. The picked value (**-10**) is displayed in the **Pick Lower level** dialog box.

Confirm your selection by clicking the  button. The **Profile Operation** dialog box is displayed.

The **Profile depth** value is calculated automatically as the difference between the **Upper level** and the **Lower level** values.

The **Lower level** parameter is associative to the solid model. Associativity enables SolidCAM to be synchronized with the solid model changes; SolidCAM automatically updates the CAM data when the model is modified. The **Profile depth** parameter is indirectly associative. The associativity is established for the Lower level. When the Upper level or the Lower level is synchronized, the Depth is updated. The background of the parameter box defined associatively to the solid model changes to the color chosen in **SolidCAM setting > Miscellaneous** for associative fields (by default, pink).

7. Define the Tool side

Switch to the **Technology** page of the **Profile Operation** dialog box.

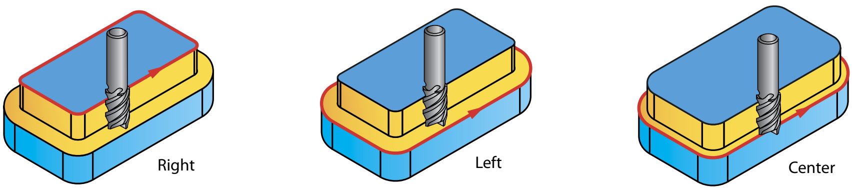
Tool Side

The **Tool side** option enables you to determine the tool position relative to the geometry.

Right–the tool cuts on the right side of the profile geometry.

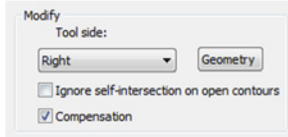
Left–the tool cuts on the left side of the profile geometry.

Center–the center of the tool moves on the profile geometry



(**No** compensation G4x can be used with this option).

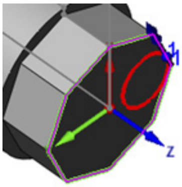
The **Geometry** button displays the **Modify Geometry** dialog box that enables you to define the modification parameters of the geometry and to choose which geometry chains are active in the operation (in case of multiple chain geometry). The chain geometry of the profile is displayed on the model with the chain direction indicated and a circle representing the tool relative to the geometry.

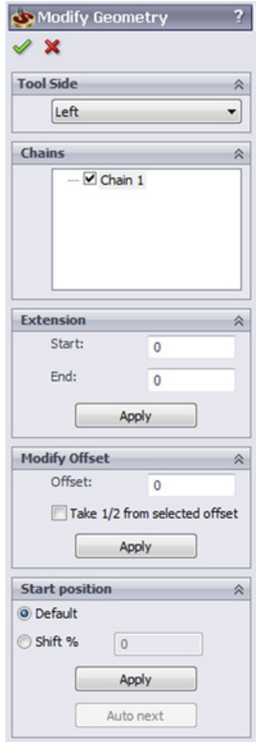


In the **Modify** section, click the **Geometry** button to check the tool location relative to the selected geometry.

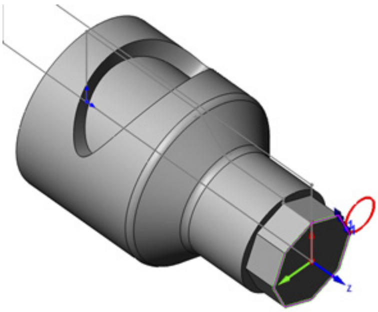
The **Modify Geometry** dialog box is displayed.

As you see, the tool side offered by default is not suitable: the tool is located inside the defined geometry. The **Tool side** has to be changed.





In the **Modify Geometry** dialog box, choose the **Left** option form the **Tool Side** list.
Now the tool is located outside the defined geometry.



Confirm the **Modify Geometry** dialog box with the  button.

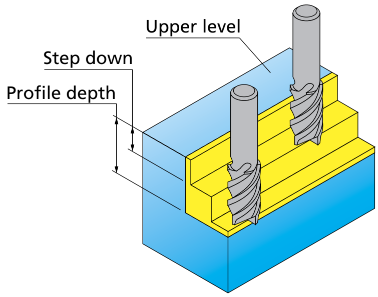
8. Define the Rough and Finish machining parameters

SolidCAM enables you to perform rough and finish machining of the profile in a single Profile operation.

Define the parameters of the Profile roughing. Select the **Rough** check box. Define the **Step down** parameter for roughing.

Step down

Profile roughing is performed in constant Z-passes. The **Step down** parameter defines the distance between each two successive Z-levels.



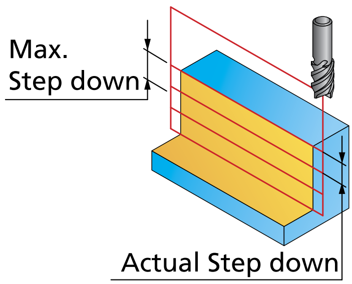
Set the **Step down** to **3**. With this value, SolidCAM performs three cuts at the following Z-levels: **-3**, **-6**, **-9**; the last cut is performed at the Z-level defined by **Profile depth**: **-10**. The distance between the Z-levels of the last cut and the preceding cut is smaller than that between the previous cuts, because the machining depth value is not divisible exactly by the **Step down** value.

In this operation, use the **Equal step down** option to keep an equal distance between all Z-levels.

Equal step down

This option enables you to perform all cuts at an equal Z-level distance one from the other. SolidCAM automatically calculates the actual step down to keep an equal distance between all passes.

When the **Equal step down** check box is selected, **Step down** is replaced by **Max step down**. This value is taken into account during the calculation of the actual step down so that it is not exceeded.

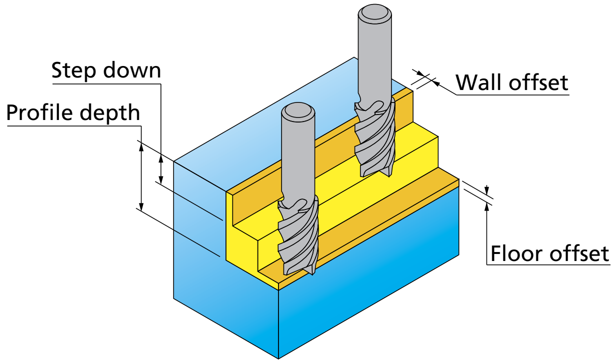


With the **Max step down** value set to **3**, the actual step down is automatically calculated as the largest possible value within the bounds of 3 so that the machining depth can be divided exactly into equal Z-levels. Four roughing passes will be performed at the followings Z-levels: **-2.5**, **-5**, **-7.5**, **-10** with the **Step down** of **2.5** mm.

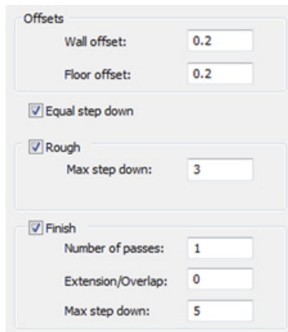
Now you need to define the wall and floor offsets that will remain after the roughing passes.

Offsets

The **Wall offset** and **Floor offset** parameters enable you to define the allowances that remain on the walls and the floor of the machined part till the profile finish machining. These allowances can be removed with the finish passes in the same Profile operation or in an additional Profile operation with another tool.



In the **Offsets** section, set the **Wall offset** and **Floor offset** values to **0.2**. The allowance of 0.2 mm is left on the walls and the floor of the steps during the profile roughing. This allowance is removed with a separate finishing cut in the end of the profile machining.



Select the **Finish** check box to perform the finishing of the profile. Set the **Max step down** for the profile finishing to **5** mm.

9. Define the Lead in and the Lead out

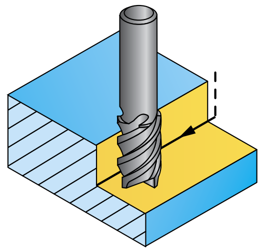
Switch to the **Link** page of the **Profile Operation** dialog box.

This page enables you to define the way the tool approaches the profile and retreats away. The lead in movement is necessary to prevent vertical entering of the tool into the material. With the lead in strategies the tool descends to the machining level outside of the material and then horizontally penetrates the material with the lead in movement. The lead out strategy enables you to perform the retract movements outside the material.

The following options are available:

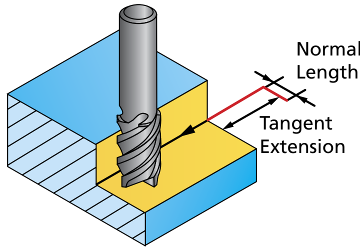
- **None**

  The tool leads in to and out from the milling level exactly adjacent to the start point of the profile.



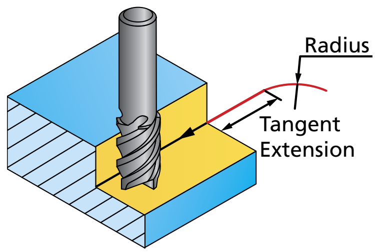
- **Normal**

  The tool leads in to and out from the profile from a point normal to the profile. The length of the normal can be set in the **Value** field.



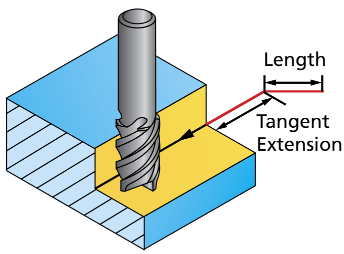
- **Arc**

  The tool leads in to and out from the profile with a tangential arc. The arc radius can be set in the **Value** field.



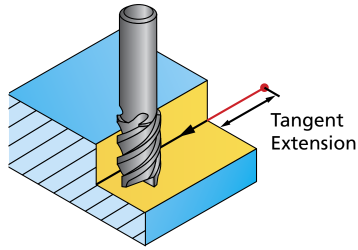
- **Tangent**

  The tool leads in/out on a line tangent to the profile. The length of the tangent can be set in the **Value** field.



- **Point**

  The tool leads in/out from a picked position. From this position, the tool moves on a straight line to the start point of the profile. When you select this option, the **Pick** button is activated so that you can select a position directly on the solid model.

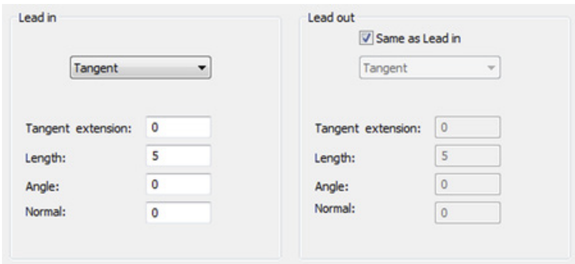


- **User-defined**

  The tool's leads in/out trajectory is specified by a user-defined wireframe chain similar to the geometry definition.

When the **Same as lead in** check box is selected, the strategy and parameters defined for **Lead in** are used for **Lead out**.

Under **Lead in**, choose the **Arc** option from the list, set the **Radius** value to **6**, **Arc angle** to **45** and the **Distance** to **0.6**. Select the **Same as Lead in** check box under **Lead out**.

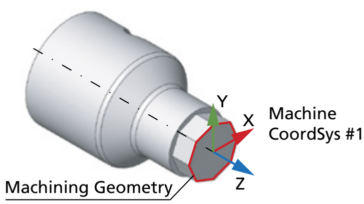


10. Define the technological parameters of facial milling

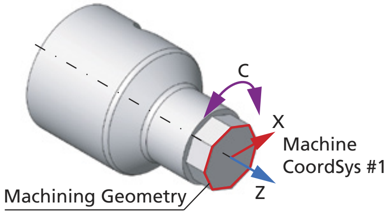
Switch to the **Motion control** page and select the **4th Axis** check box. This check box enables you to define the technological parameters of facial milling on Mill-Turn CNC-machines.

4th axis type

When the **Diameter** option is chosen, you receive the GCode output for the part machining in standard linear coordinates (X, Y, Z).



The **Face** option enables you to perform milling using the rotary axis by translating the linear movements in the XY-plane into the rotary XC-movements.



Note that the technology offered by SolidCAM is universal: there is no limitations in terms of machines on which the part can be milled. But if you use the GCode output in the XYZ coordinates (the **Diameter** option), the usability of this GCode is limited to machines with possibility of movements along three linear axes (X, Y, Z). Therefore, to enable the possibility of milling the part on 3-axis Mill-Turn CNC-machines that support only the XZC coordinates where the movements along the Y-axis is impossible, you need to choose the **Face** option.

Choose the **Face** option.

Coordinate type

This option enables you to determine whether the GCode will consist of blocks in polar/Cartesian coordinates:

- **Polar**

  When this option is chosen, the tool path is calculated in polar coordinates.

- **Cartesian**

  The tool path lines and arcs are calculated in Cartesian coordinates; the CoordSys position is zero for linear coordinates. The milling is processed using the rotary axis by translating the linear movements into the rotary-linear movements according to the plane.

In the **Coordinate type** section, choose the **Polar** option to obtain the tool path in polar coordinates.

Plane

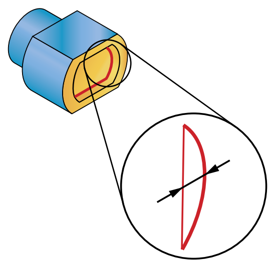
This option offers you the choice of plane in which the translated rotary-linear movements are performed.

Make sure **XC** is chosen for **Plane** to perform all movements in these coordinates.

Define the **Tolerance**.

Tolerance

The tool path lines and arcs are split into many blocks that do not exactly generate the required tool path. The tolerance value in the **Approximate arcs by lines within tolerance** option defines the maximum allowed error; the smaller is the value of this field, the greater is the number of GCode blocks generated.



Set the **Tolerance** value to **0.01**.

The definition of the basic technological parameters of the profile milling is finished.

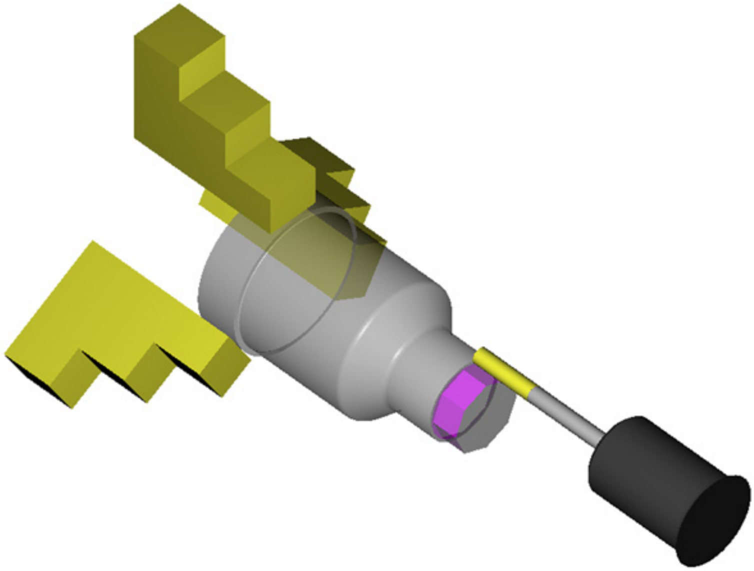
11. Calculate the Tool path

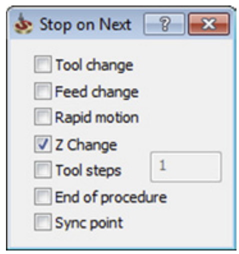
Click the **Save & Calculate** button. The operation data is saved, and the tool path is calculated.

12. Simulate the operation

Click the **Simulate** button in the **Profile Operation** dialog box.

The **Simulation** control panel is displayed. Switch to the **SolidVerify** page and start the simulation with the ▶ button.





When the simulation is finished, click the **Stop on next** button on the **Simulation** control panel. The **Stop on Next** dialog box is displayed. This dialog box enables you to define specific points where the simulation process is stopped.

Select the **Z Change** check box. This option stops the simulation at every change of the Z-coordinate of the tool.

Play the simulation again. Every time the simulation stops on Z Change, click the ▶ button to continue. The **Z Change** option enables you to see the tool path simulation in detail.

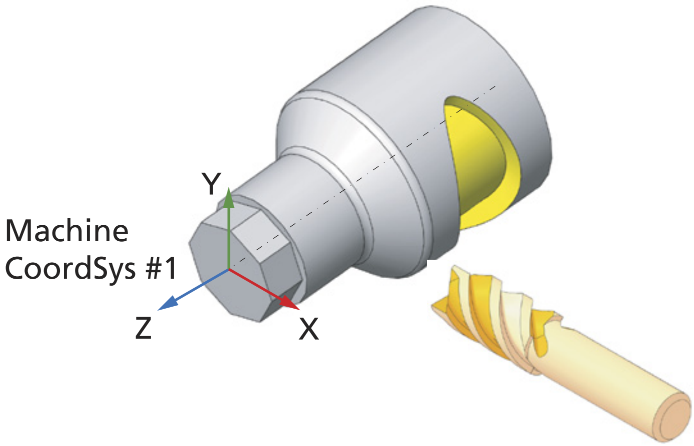
Exit the simulation.

13. Close the operation dialog box

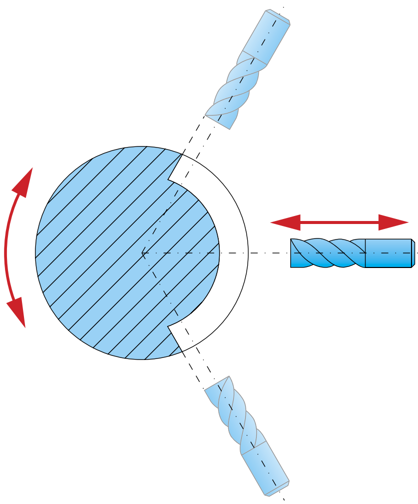
Click the **Exit** button. The **Profile Operation** dialog box is closed.

4.2 Simultaneous 4-Axis Milling

SolidCAM enables you to perform simultaneous 4-axis milling on Mill-Turn CNC-machines. **Machine Coordinate System #1 Position #1** is used for the geometry definition for simultaneous 4-axis milling.



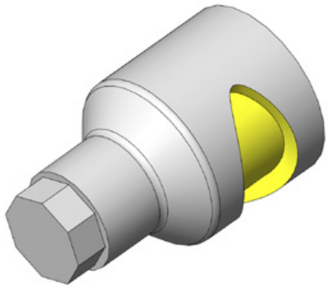
The SolidCAM simultaneous 4-axis capabilities enable you to perform machining with the tool axis always intersecting with the part revolution axis.



Exercise #12: Simultaneous 4-Axis Milling

This exercise illustrates the SolidCAM capabilities for simultaneous 4-axis milling using Mill-Turn CNC-machines. This exercise uses the pre-machined CAM-Part defined in the previous exercises.

The wrapped elliptical pocket is machined in this exercise.



1. Load the CAM-Part

Load the **Exercise11.prz** CAM-Part prepared in the previous exercise.

2. Add an operation

Right-click the last operation in **SolidCAM Manager** and choose **Pocket** from the **Add Milling Operation** submenu.

The **Pocket Operation** dialog box is displayed.

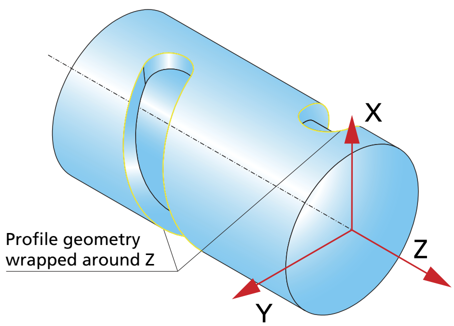
3. Define the Geometry

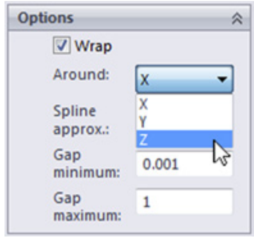
On the **Geometry** page, click to start the geometry definition.
The **Geometry Edit** dialog box is displayed.

Under **Options**, select the **Wrap** check box.

SolidCAM enables you to perform radial milling using the same Coordinate System as for facial milling.

When the **Wrap** check box is selected, you can wrap the selected profile geometry around any axis of the Coordinate System. The **Around** list offers you the choice of axis around which the geometry is wrapped.

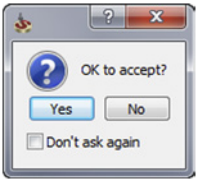


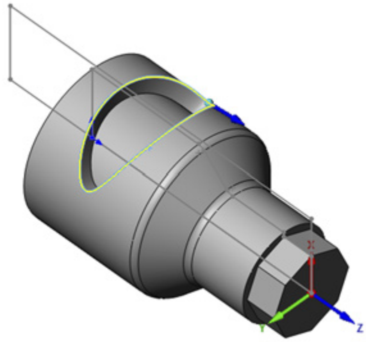


Choose **Z** from the **Around** list. The selected geometry will be wrapped around the Z-axis.

Select the model edge as shown.

The confirmation message is displayed.





Click **Yes**. The chain icon is displayed in the **Geometry Edit** dialog box.

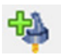
Click  to confirm the Geometry selection.

The **Pocket Operation** dialog box is displayed.

4. Define the Tool

On the **Tool** page of the **Pocket Operation** dialog box, click the **Select** button to add a new tool to the **Part Tool Table** and use it in the operation.

The **Part Tool Table** is displayed.

Click  to define a new milling tool.

Choose the tool of the **End mill** type. A new **Ø6 End mill** tool is added to the **Part Tool Table**.

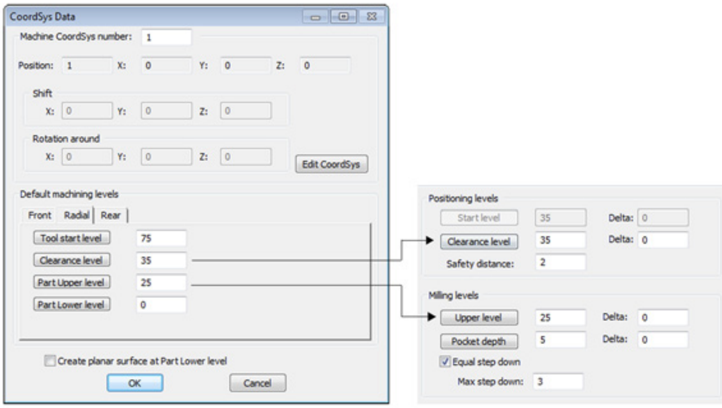
Choose the **BT40 ER16x70** tool holder from the **Local holders table** and click the **Select** button to choose the tool for the operation.

The Local holders table contains the tool holders that were already used in the current CAM-Part. When a new holder is chosen from the Global holders table, it is copied to the local table to make a further use easier.

5. Define the Pocket depth

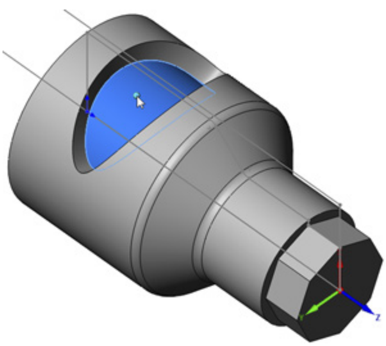
Switch to the **Levels** page of the **Pocket Operation** dialog box.

When the geometry for the operation is defined in the wrapping mode, SolidCAM substitutes the current levels with defaults received from the **Radial** set defined in the **CoordSys Data** dialog box.



Click the **Pocket depth** button to pick the **Lower Level** directly on the solid model. Click on the bottom face of the wrapped pocket as shown.

Confirm the level selection with the button. The **Pocket depth** value (**5**) is automatically calculated according to the **Upper** and **Lower levels**.



Set the **Step down** to **3** and select the **Equal step down** check box.

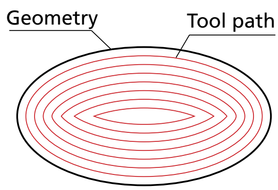
6. Define the Pocket machining strategy

Switch to the **Technology** page of the **Pocket Operation** dialog box.

SolidCAM offers the following strategies for the pocket machining:

- **Contour:** the pocket is machined in a round pattern.
- **Hatch:** the pocket is machined in a linear pattern.
- **Hatch+Finish:** the pocket is machined in a linear pattern, and the profile is cleaned up on each cutting depth.
- **Plunging pattern:** the tool moves up and down in a drilling motion, travelling inside the pocket.

Use the default **Contour** strategy. With this strategy, the tool moves at offsets parallel to the pocket contour.



7. Save & Calculate

Click the **Save & Calculate** icon to save the operation data and calculate the tool path.

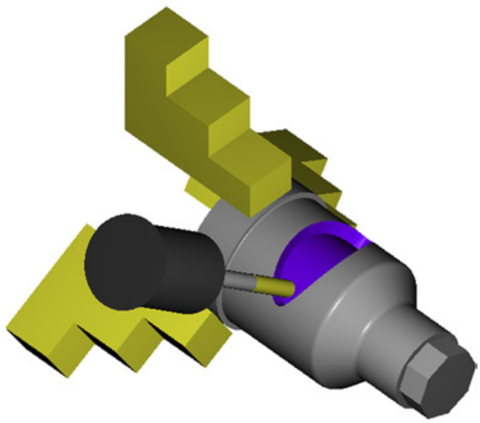
8. Simulate the operation

Click the **Simulate** button and simulate the CAM-Part in the **SolidVerify** mode.

Play the simulation with the ▶ button.

Close the **Simulation** control panel.

Close the **Pocket Operation** dialog box with the **Exit** button.

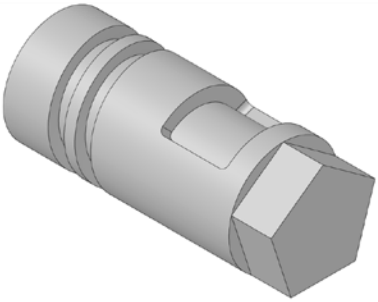


9. Close the CAM-Part

Right-click the **CAM-Part** header in **SolidCAM Manager** and choose the **Close** item from the menu.

At this stage, the exercise is finished.

Exercise #13: Stopper Machining



Define the CAM-Part and operations for the machining of the stopper presented on the illustration on a Mill-Turn CNC-machine.

This exercise reinforces the following skills:

- CAM-Part definition
- Turning
- Facial milling
- Simultaneous 4-axis machining

The SolidWorks model of the Stopper (**Exercise13.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

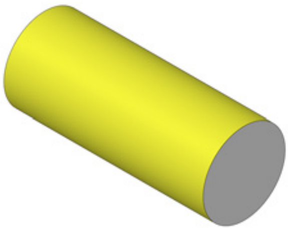
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Stock model and the Target model.
The **NTX1000** CNC-controller has to be chosen for this exercise.

2. Define the clamping fixture

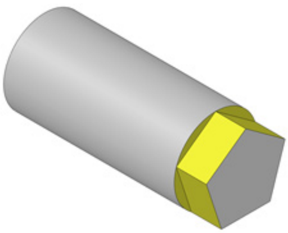
Define the fixture for clamping of the part on the CNC-machine table.

3. External turning



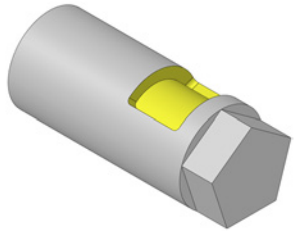
Define a Turning operation to obtain the following revolution body.

4. Facial milling of the prismatic projection



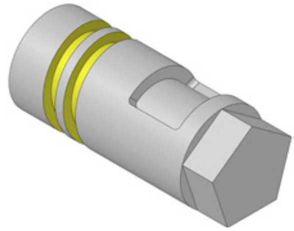
Machine the prism using the facial milling feature.

5. Pocket machining



Machine the pocket wrapped on the cylindrical surface of the stopper using the simultaneous 4-axis milling.

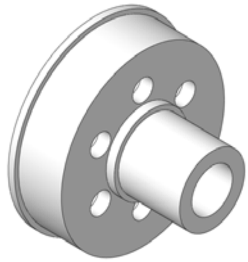
6. Helical Slot machining



Machine the helical slot wrapped on the cylindrical surface of the stopper using the simultaneous 4-axis milling.

Exercise #14: Bushing Machining

Define the CAM-Part and operations for the machining of the bushing presented on the illustration on a Mill-Turn CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Turning
- Facial milling

The SolidWorks model of the Bushing (**Exercise14.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

1. Define the CAM-Part

At this stage, you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Stock model and the Target model.

The **NTX1000** CNC-controller has to be chosen for this exercise.

2. Define the clamping fixture

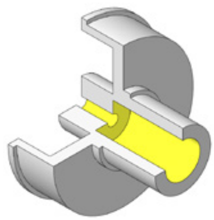
Define the fixture for clamping of the part on the CNC-machine table.

3. External turning

Define a Turning operation to obtain the following revolution body.

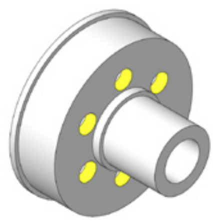
4. Machining of the inner faces

Define a number of Turning Drilling operations to obtain the following model faces.



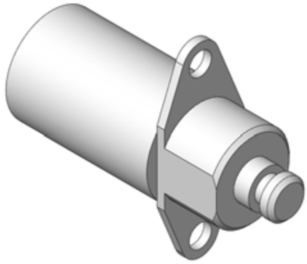
5. Facial Drilling

Machine the circular pattern of drills using the facial milling capabilities.



Exercise #15: Shaft Machining

Define the CAM-Part and operations for the machining of the shaft presented on the illustration on a Mill-Turn CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Turning
- Facial milling
- Radial milling

The SolidWorks model of the Shaft (**Exercise15.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

1. Define the CAM-Part

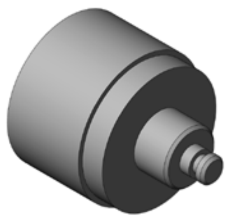
At this stage, you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Stock model and the Target model.
The **NTX1000** CNC-controller has to be chosen for this exercise.

2. Define the clamping fixture

Define the fixture for clamping of the part on the CNC-machine table.

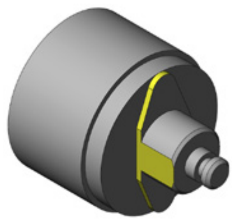
3. External turning

Define a Turning operation to obtain the following revolution body.



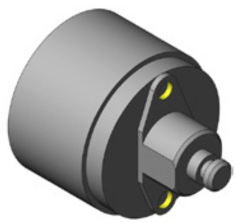
4. Facial milling

Machine the highlighted model faces using the facial milling capabilities.



5. Drilling

Machine two holes using the facial milling capabilities.



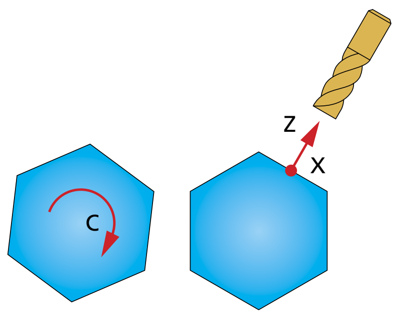
4.3 Indexial Milling on Mill-Turn CNC-machines

SolidCAM enables you to perform **indexial 4-axis milling operations** on Mill-Turn CNC-machines. The workflow of the indexial milling in the Mill-Turn module is similar to the indexial milling workflow in SolidCAM Milling. At the first stage, you have to define a specific position of the Machine Coordinate System. This Coordinate System position is used later for the definition of operations. The geometry intended to be used in the indexial milling operation has to be located in the plane parallel to the XY-plane of the Coordinate System position. During the machining, SolidCAM maintains the tool axis orientation parallel to the Z-axis of the Coordinate System position.

Generally, SolidCAM does not limit you in the machining methods used for indexial milling. All the operations and technology options are available. But, at the same time, the technology choice depends on the type and possibilities of your CNC-machine. During the operation programming, you have to take care that the generated tool path will contain only the linear/rotational movements supported by your CNC-machine. The tool path that contains the movements illegal for the current CNC-machine will cause errors during postprocessing.

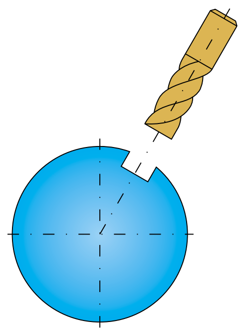
For example, the indexial milling tool path that contains movements along the Y-axis cannot be executed on a 3-axis CNC-machine (XZC axes). For machines of this type, indexial machining is performed using ZX-movements only; the positioning is performed using the C-axis (revolution axis).

In the CNC-machines of this type, the tool axis is always perpendicular to and intersecting with the spindle revolution axis. The positioning is performed by rotating the part around the revolution axis keeping the same tool orientation. Therefore, the position of the Coordinate System must be defined in such a way that the Z-axis of the position will be located in the plane parallel to the XY-plane of the Machine Coordinate System. Incorrect definition of the Coordinate System position causes illegal tool movements that result in errors during postprocessing.



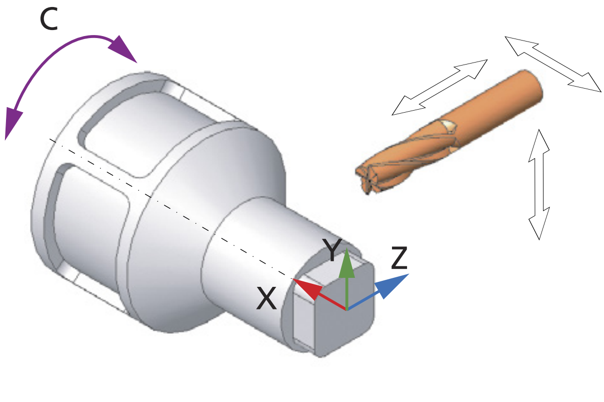
Positioning Machining

During the indexial milling operations on the machines of this type, the tool axis must always intersect with the revolution axis of the part. This applies restrictions on the range of machining methods that can be used. For example, indexial milling on the machines of this type enables you to perform machining of key grooves with the symmetry plane intersecting with the part revolution axis, because this type of machining can be performed with the ZX-movements only.



SolidCAM enables you to perform machining on 4-axis Mill-Turn CNC-machines (XYZC axes). This type of CNC-machines provides you with all functionality of 3-axis Mill-Turn CNC-machines. In addition, it provides the capability of Y-axis movements for milling.

The 4-axis Mill-Turn CNC-machines provide you with all the possibilities of indexial milling around the revolution axis (C-axis). Generally, the capabilities of these machines are similar to those of a 3-axis Milling machine equipped with the rotary axis (4-axis milling CNC-machine). The positioning in these machines is performed using the C-axis, similar to the 3-axis Mill-Turn CNC-machines; when the positioning is performed, the machining is performed using the XYZ axes.

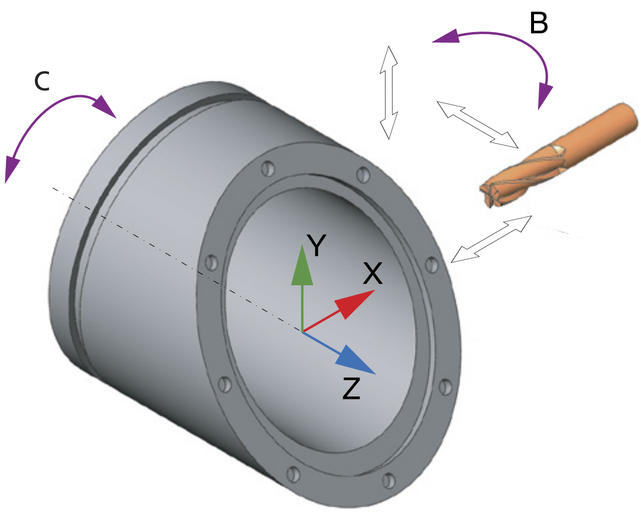


The Coordinate System position has to be defined similar to the one defined for 3-axis machines; the Z-axis of the position must be located in the plane parallel to the XY-plane of the Machine Coordinate System.

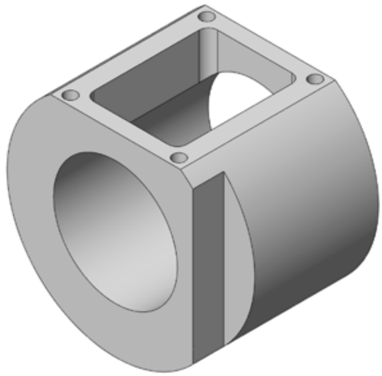
With such Coordinate System position, all 2.5D and 3D Milling functionality is available.

SolidCAM enables you to perform machining on 5-axis Mill-Turn CNC-machines (XYZCB axes). This type of CNC-machines provides you with all the functionality available on the 4-axis Mill-Turn CNC-machines. In addition, it provides the capability of indexial rotation around either Y-axis or X-axis.

The Coordinate System position for the indexial milling has to be defined according to the rotation limitations of the CNC-machine revolution axes. This means that the rotation angles between the axes of the Machine Coordinate System and Coordinate System position have to be kept within the range supported by the CNC-machine.



Exercise #16: Indexial Milling on 4-Axis CNC-Machines



This exercise covers the process of CAM-Part definition and Milling on 4-axis Mill-Turn CNC-machines (supporting the XYZC coordinates). The following model of support is used.

Since the CAM-Part definition process was explained in detail in **Exercise #2**, the details of this CAM-Part definition process are omitted.

1. Load the SolidWorks model

Load the **Exercise16.sldprt** model located in the **Exercises** folder.
This model contains a number of features forming the solid body and sketches that are used in the CAM-Part definition.

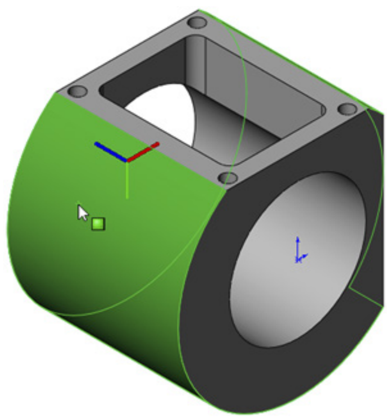
2. Start a new SolidCAM Mill-Turn part

Click **SolidCAM** in the main menu of SolidWorks and choose **Mill-Turn** from the **New** submenu. SolidCAM is started and the **New Mill-Turn Part** dialog box is displayed. Choose the **External mode** and confirm this dialog box with **OK**.
The **Mill-Turn Part Data** dialog box is displayed.

3. Define the CNC-controller

In this exercise, the CNC-machine with an upper rotary turret and two spindles (main and back) is used. Choose the **urt_ms_bs** CNC-controller from the list.

4. Define the Coordinate System



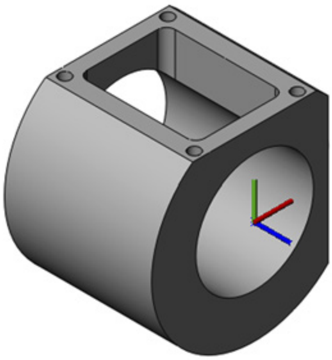
Click the **CoordSys** button to start the Coordinate System definition.

The **CoordSys** dialog box is displayed.

In the **Place CoordSys origin to** list, choose **Center of revolution face**. Select the **High precision** check box and click on the cylindrical face of the model as shown.

The Z-axis of the Machine Coordinate System is collinear to the revolution axis of the part.

Click the **Change to opposite** button in the **CoordSys** dialog box. The Machine Coordinate System is defined on the front face of the model.



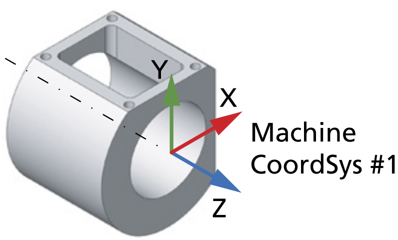
Click  to confirm the definition.

The **CoordSys Data** dialog box is displayed. Close it with **OK**.

The **CoordSys Manager** dialog box is displayed in the **SolidWorks PropertyManager** area. Confirm this dialog box with .

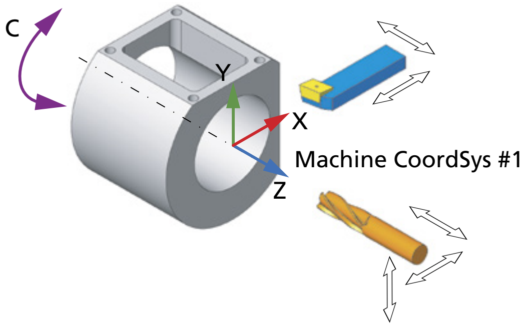
Coordinate Systems

SolidCAM enables you to define **Machine Coordinate System #1** with the Z-axis directed along the revolution axis.

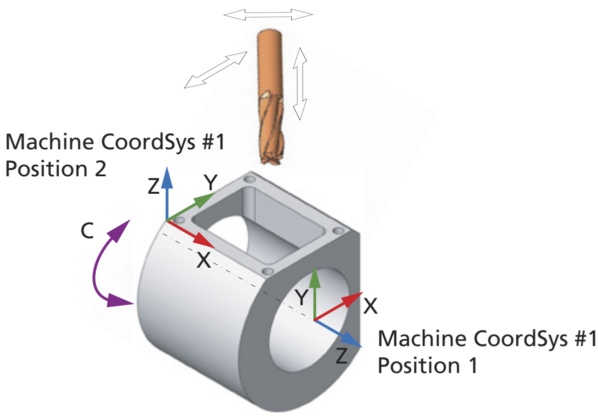


This Machine Coordinate System is used for Turning operations. The turning tool movements are located in the ZX-plane.

The **Machine Coordinate System #1** is also used for facial milling, where the axis of the milling tool is parallel to the Z-axis of the Machine Coordinate System.



To perform the indexial milling of the upper face, the pocket and the drills, you have to define a new indexial position under the **Machine CoordSys #1**.



5. Define the Stock model

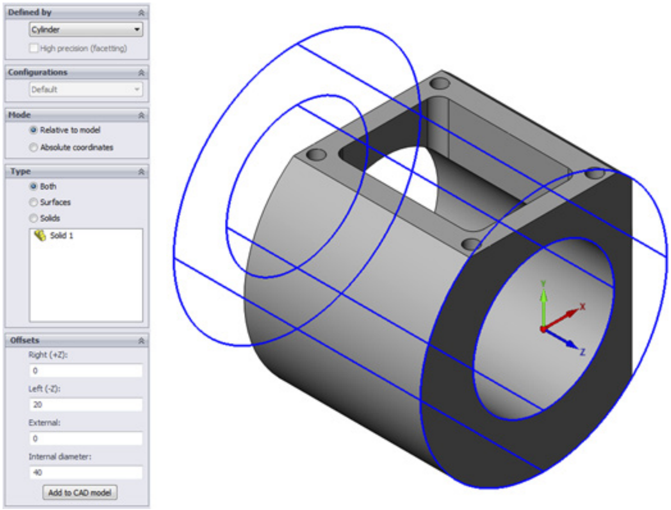
In the **Mill-Turn Part Data** dialog box, click the **Stock** button. The **Model** dialog box is displayed in the SolidWorks **PropertyManager** area.

Choose the **Cylinder** option and click on the solid body to select it.

Specify the following parameters in the dialog box:

- Set the **Right (+Z)** value to **2**. This offset enables you to machine the front face of the stock material.
- Set the **Left (-Z)** value to **20**. This offset enables you to have stock material at the back side of the model for clamping.
- Set the **External** to **2**.

- Set the **Internal diameter** to **36**. The stock used in this exercise is a tube with this internal diameter.

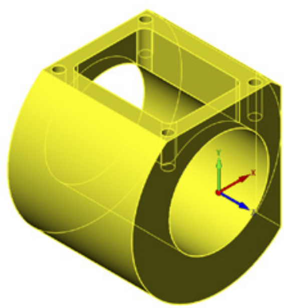


Set the **Facet tolerance** value to **0.01**.

Confirm the parameters with the button.

The **Mill-Turn Part Data** dialog box is displayed.

6. Define the Target model



In the **Mill-Turn Part Data** dialog box, click the **Target** button.

The **Model** dialog box is displayed.

Click on the solid body. It is highlighted.

In the **Generate Envelope/Section** section, select the **Envelope** option.

Confirm the selection with the button. The **Mill-Turn Part Data** dialog box is displayed.

7. Save the CAM-Part

Click in the **Mill-Turn Part Data** dialog box. The CAM-Part is saved and the **SolidCAM Manager** tree is displayed.

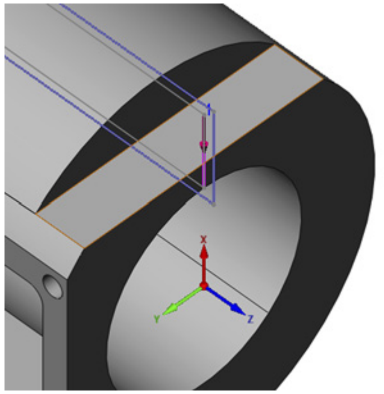
At this stage, the CAM-Part definition is finished. Now you have to define a number of operations to machine the part.

8. Add an operation

Right-click the **Operations** header in the **SolidCAM Manager** tree and choose **Turning** from the **Add Turning Operation** submenu.

The **Turning Operation** dialog box is displayed.

9. Define the Geometry



In the **Submachine** section, select the **upper rotary turret - main spindle** option from the list.

Click on the **Geometry** section.

The **Geometry Edit** dialog box is displayed.

Define the geometry as shown on the picture.

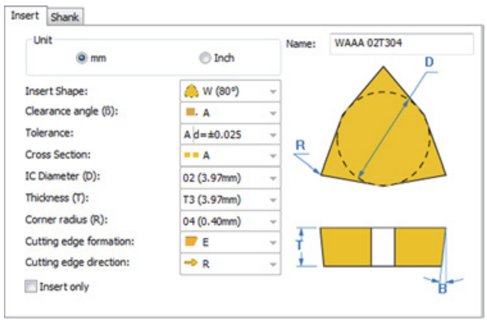
10. Define the Tool

Click the **Select** button on the **Tool** page.

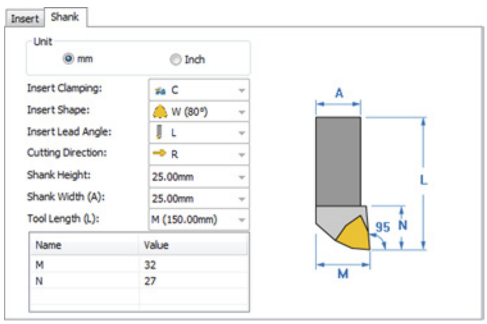
The **Part Tool Table** is displayed. Define a new turning tool.

Click and select **Ext.Turning** tool from the **Composite Tools** section.

Define the tool with the following parameters:



- Set the **Insert Shape** to **W**
- Set **Thickness** to **T3**
- Set the **Corner radius** to **04**



- Set the **Insert Lead Angle** to **L**
- Set the **M** value to **32**

Click **Select** to choose the tool for the operation.

11. Define the machining parameters

Switch to the **Technology** page of the **Turning Operation** dialog box.

In the **Mode** section, select the **Face front** option to machine the front face of the model.

Make sure that the **Rough** option is chosen in the **Work type** section.

Switch to the **Semi-finish/finish** tab. In the **Finish** section, select the **ISO-Turning method**.

12. Save and calculate

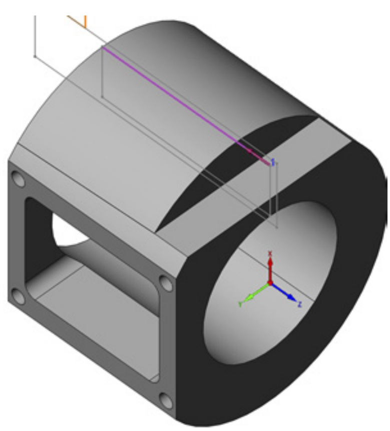
Click the **Save & Calculate** icon to save the operation data and generate the tool path.

13. Add an operation

In the **Turning Operation** dialog box, click **Save & Copy** to start a new operation. All data defined in the previous operation is copied to a new operation.

14. Define the Geometry

In the **Geometry** page, click to define a new geometry. Click on the entity of the Envelope sketch as shown. The selected entity is highlighted.



Click the **Accept Chain** button in the **Geometry Edit** dialog box to confirm the chain selection.

Close the dialog box with the button.

The geometry is defined for the operation.

15. Define the Tool

Use the same tool as in the previous operation.

16. Define the machining parameters

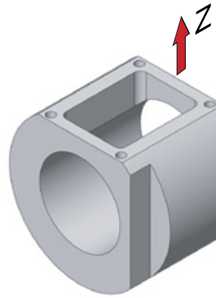
In the **Technology** page, change the **Mode** to **Long external** .

The rest of the parameters remain the same.

17. Save and calculate

Click the **Save & Calculate** icon to save the operation data and generate the tool path.

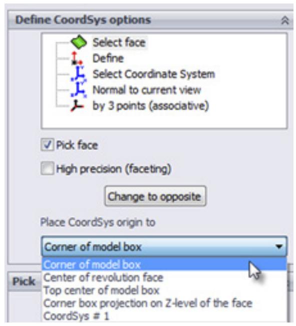
18. Add a new CoordSys position



To machine the top face, you have to define an additional Coordinate System position related to the **Machine CoordSys #1**. The Z-axis of this CoordSys position has to be normal to the top face.

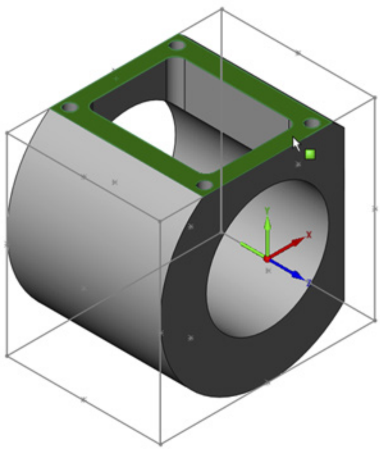
Double-click the **CoordSys Manager** header in the **SolidCAM Manager** tree. The **CoordSys Manager** dialog box is displayed. Right-click the **MAC 1** item and choose **Add** from the menu.

The **CoordSys** dialog box is displayed. Note that the **Mac CoordSys number** is **1** and the **Position** number is **2**. Now SolidCAM can define **CoordSys Position #2** related to **Machine Coordinate System #1**.



In the **Place CoordSys origin to** list, choose **Corner of model box**.

Select the **High precision** check box and click on the model face as shown.

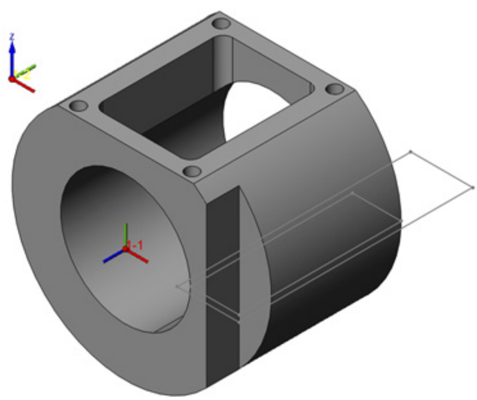


The CoordSys position origin is defined in the corner of the model box. The Z-axis of the position is normal to the selected face.

Click ✓ to confirm the CoordSys position definition.

The **CoordSys Data** dialog box is displayed.

The **Part Upper level** and **Part Lower level** values are determined automatically according to the solid model.



Confirm the **CoordSys Data** dialog box with the **OK** button. The **CoordSys Manager** dialog box is displayed again. Close it with ✓.

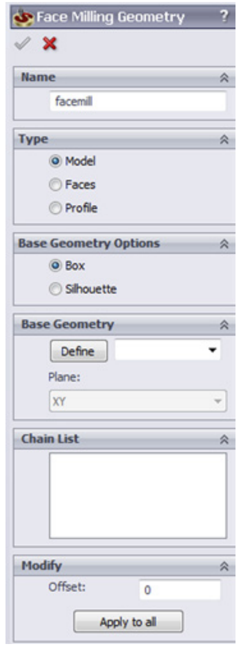
19. Add a new Face Milling operation

Add a new Face Milling operation to machine the model face using **CoordSys Position #2**. This operation enables you to machine large flat surfaces with face mill tools.

Right-click the **Operations** header in the **SolidCAM Manager** tree and choose **Face** from the **Add Milling Operation** submenu. The **Face Milling Operation** dialog box is displayed.

20. Define the Geometry

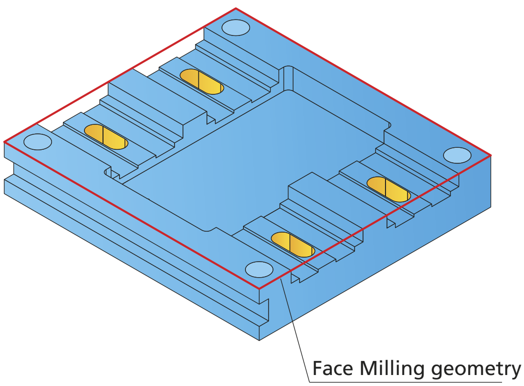
On the **Geometry** page of the **Face Milling Operation** dialog box, choose the **Machine CoordSys #1 Position #2**. Click the button. The **Face Milling Geometry** dialog box is displayed.



The **Type** section enables you to define the face milling geometry using the following methods:

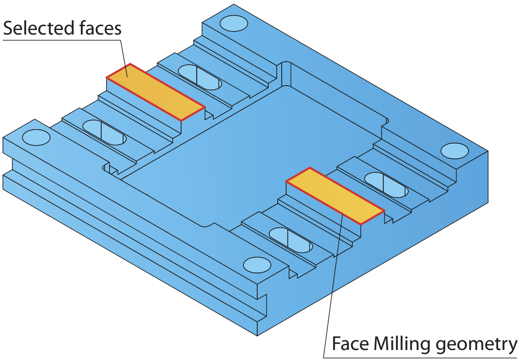
- **Model**

  This option generates a rectangle located at the XY-plane and surrounding the Target model and selects it for the Face Milling geometry. The rectangle chain is displayed in the **Chain List** section.



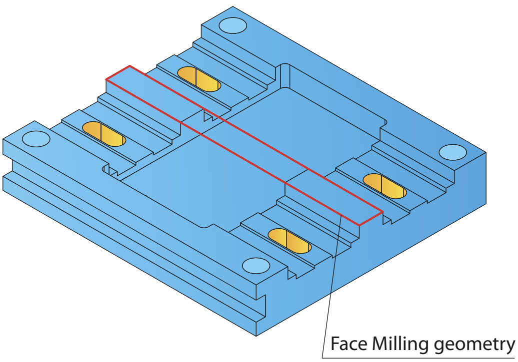
- **Faces**

  This option enables you to define the Face Milling geometry by face selection. The **Define** button and the related combo-box enable you either to define a new faces geometry with the **Select Faces** dialog box or to choose an already defined geometry from the list. When the model faces are selected, SolidCAM generates a number of chains surrounding the selected faces. These chains are displayed in the **Chain List** section.



- **Profile**

  This option enables you to define the Face Milling geometry by a profile. The **Define** button and the related combo box enable you either to define a new profile geometry with the **Geometry Edit** dialog box or to choose an already defined geometry from the list. The defined chains are displayed in the **Chain List** section.



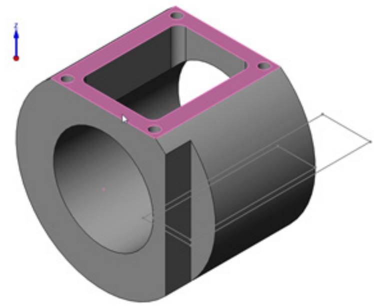
In the **Type** section, choose the **Faces** option and click the **Define** button. The **Select Faces** dialog box is displayed. This dialog box enables you to define the face milling geometry by selecting the model faces.

Select the model face as shown.

Confirm the **Select Faces** dialog box with .

The **Face Milling Geometry** dialog box is displayed. In the **Modify** section, set the **Offset** from the machined face to **1**.

Confirm the **Face Milling Geometry** dialog box by clicking .



21. Define the Tool

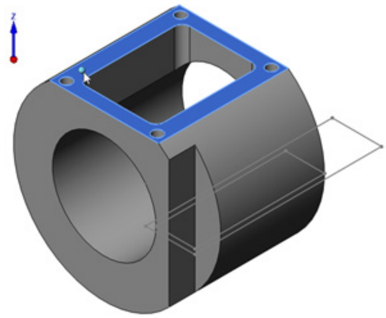
Add a new **End Mill** tool to the **Part Tool Table**.

Set the **Diameter** of the tool to **16**.

Choose the **BT40 ER40x80** tool holder from the Global holders table and click the **Select** button to choose the tool for the operation.

22. Define the Milling levels

Click the **Face depth** button on the **Levels** page. Define the **Lower level** directly on the solid model as shown.



23. Define the machining strategy

On the **Technology** page, make sure that the **Hatch** machining strategy is chosen. This strategy enables you to machine the face in a linear pattern.

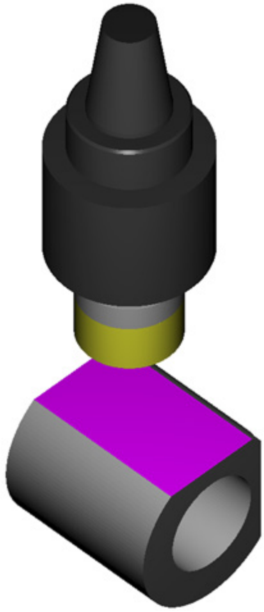
24. Save and Calculate

Click the **Save & Calculate** icon to save the operation data and calculate the tool path.

25. Simulate

Simulate the tool path with the **SolidVerify** mode.

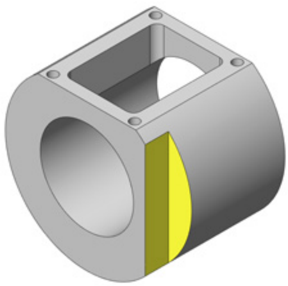
When the simulation is finished, close the **Face Milling Operation** dialog box.



26. Define a Profile operation

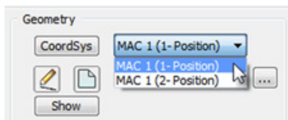
Define a new Profile operation to machine the step displayed on the illustration.

The **Profile Operation** dialog box is displayed.



27. Define the Geometry

In the **Geometry** page, choose **MAC 1 (1-Position)** from the **Coordsys** options. Click to start the definition. The **Geometry Edit** dialog box is displayed.

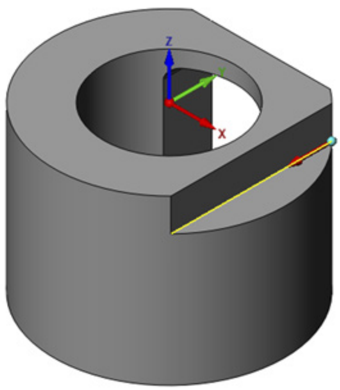


Select the model edge as shown.

Click **Accept chain** in the **Geometry Edit** dialog box to confirm the chain selection.

Close the dialog box with .

The geometry is defined for the operation.



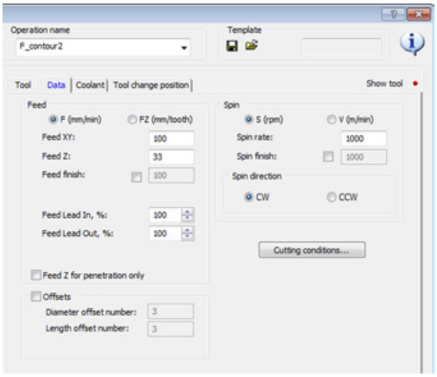
28. Define the Tool

Click the **Select** button on the **Tool** page.

The **Part Tool Table** is displayed.

Add a new **End Mill** tool. Set the **Diameter** to **16**. Choose the **BT40 ER16x70** tool holder from the **Global** holders table.
Click the **Select** button to choose the defined tool for the operation.

Switch to the **Data** tab on the **Tool** page to define the tool spin parameters for the operation.



Spin

This parameter defines the spinning speed of the tool. It defines two spin values:

- **Spin rate** – normal spin rate used in rough milling;
- **Spin finish** – finish spin rate used in finish milling.

The **Spin finish** check box enables you to optionally define different values for **Spin rate** and **Spin finish**. When this check box is selected, the corresponding edit box is available so that you can edit its value. When this check box is not selected, the specified Spin rate value is used for both rough and finish machining.

The spin value can be defined in two types of units: **S** and **V**.

S is the default and signifies **Revolutions per Minute. V** signifies material cutting speed in **Meters per Minute** in the **Metric** system or in **Feet per Minute** in the **Inch** system; it is calculated according to the following formula:

V = (S * PI * Tool Diameter) / 1000

Feed

This parameter defines the feed rate of the tool. It defines three feed values:

- **Feed XY** – the feed rate in the XY-plane.
- **Feed Z** – the feed rate in the Z direction.
- **Feed finish** – the feed rate used for finish milling.

The **Feed finish** check box enables you to optionally define different values for **Feed XY** and **Feed finish**. When this check box is selected, the corresponding edit box is available so that you can edit its value. When this check box is not selected, the specified **Feed XY** value is used for both rough and finish machining.

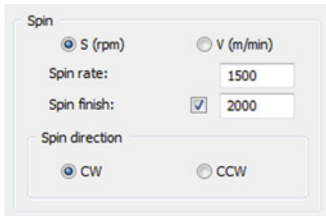
The feed value can be defined in two types of units: **F** and **Fz**.

F is the default and signifies **Units per minute**. **Fz** signifies **Units per tooth** and is calculated according to the following formula:

Fz = F/(Number of Flutes * S)

Define the parameters as follows:

- Set the **Spin rate** parameter to **1500**
- Select the check box and set the **Spin finish** parameter to **2000**

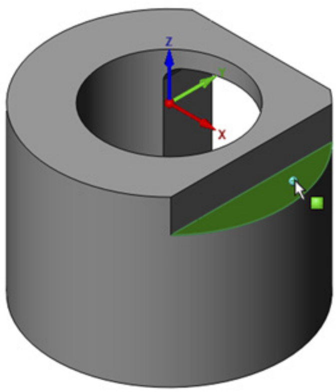


29. Define the Milling levels

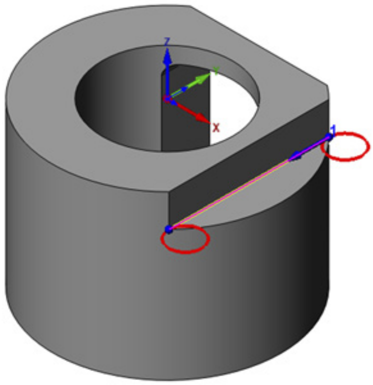
Click the **Profile depth** button on the **Levels** page to define the machining depth directly on the solid model.

Click on the model face as shown.

Confirm the selection and close the **Pick Lower level** dialog box with .



30. Specify the technological parameters



In the **Modify** section of the **Technology** page, click the **Geometry** button to check the tool location relative to the profile. If the tool is located at the wrong side, change the **Tool side** to **Left**.

Select the **Rough** check box in order to perform the rough machining of the profile. Set the **Step down** value to **5**. The material is removed in two Z-cuts.

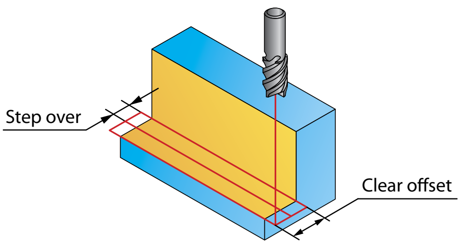
Set the **Wall offset** and **Floor offset** to **0.2**. The offsets of 0.2 mm will be left on the profile wall and floor during machining. These offsets will be removed in a separate cut in the end of the profile machining.

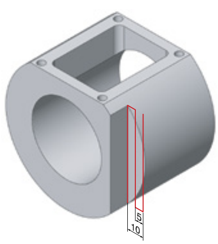
Select the **Clear offset** check box.

Clear offset

This option generates several concentric profiles with a constant depth that start at the defined clear offset distance from the profile and finish at the geometry of the profile, thus clearing the area around the profile.

- **Offset** defines the distance from the geometry, at which the milling starts. The **Clear offset** value should be equal or larger than the **Wall offset** value. The tool starts milling the profile at the distance defined as **Clear offset** and finishes at the distance defined as **Wall offset**.
- **Step over** defines the overlap of adjacent tool paths; it determines the offset between two successive concentric profiles.





Set the **Offset** value to **10** and the **Step over** value to **5**. Choose the **Zigzag** option. The profile will be machined in two equidistant profiles located at the offset of 5 mm.

Select the **Finish** check box to perform the finish machining of the profile. Set the **Step down** value to **5**.

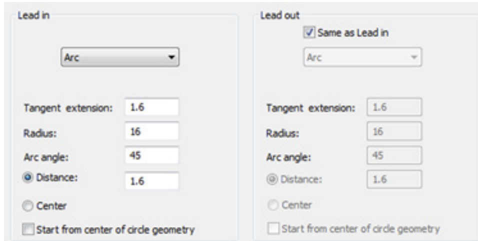
31. Define the Lead in and Lead out

In the **Link** page, choose the **Arc** option for **Lead in**. This option enables the tool to approach the material from outside by an arc.

Define the following parameters:

- Set the **Tangent extension** value to **1.6**
- Set the **Radius** value to **16**
- Set the **Arc angle** value to **45**
- Set the **Distance** value to **1.6**

In the **Lead out** section, select the **Same as lead in** option.



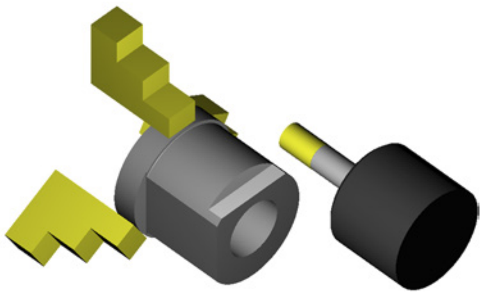
32. Save and calculate

At this stage, the definition of the technological parameters is finished. Click the **Save & Calculate** icon to save the operation data and generate the tool path.

33. Simulate the operation

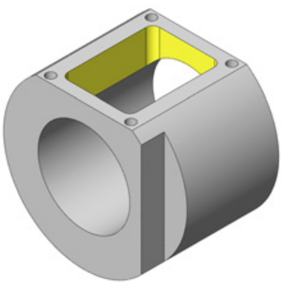
Simulate the operation in the **SolidVerify** mode.

When the simulation is finished, close the **Profile Operation** dialog box with the **Exit** button.

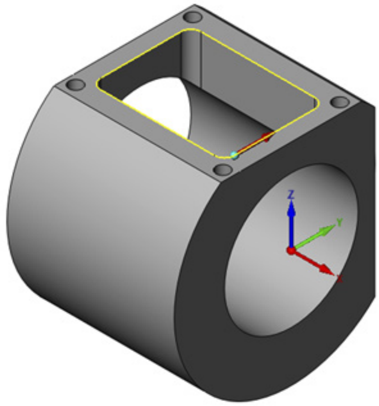


34. Add a Pocket operation

Define a new Pocket operation to machine the through pocket located on the face machined earlier.



35. Define the Geometry



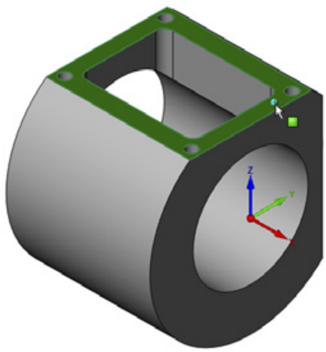
Choose **Machine CoordSys #1 Position #2** for the operation.

Define the Geometry for the Pocket operation as shown. In the **Geometry Edit** dialog box, use the **Auto-constant Z** option to complete the chain.

36. Define the Tool

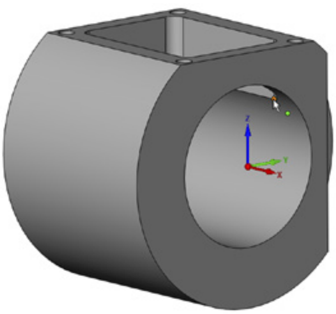
In the **Part Tool Table**, define a new **End mill** tool with default parameters.

37. Define the Milling levels



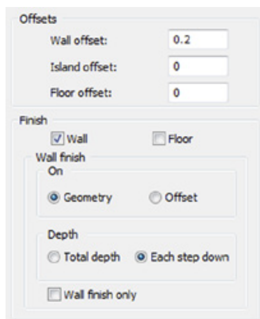
Define the **Upper level** on the face machined in the previous operation as shown.

Click the **Pocket depth** button and pick the point on the model as shown.



Set the **Step down** to **6** and select the **Equal step down** check box.

38. Define the Wall offset



Switch to the **Technology** page of the **Pocket Operation** dialog box. In the **Offsets** section, set the **Wall offset** to **0.2**.

In the **Finish** section, select the **Wall** check box.

With the specified **Wall offset** value, the offset of 0.2 mm is left on the wall of the pocket. This offset is removed at the end of pocket machining in a separate cut.

39. Define the Ramping parameters

In the **Ramping** section of the **Link** page, choose the **Helical** option from the list. This option enables the tool to plunge into the material along a helix at the automatically calculated point.

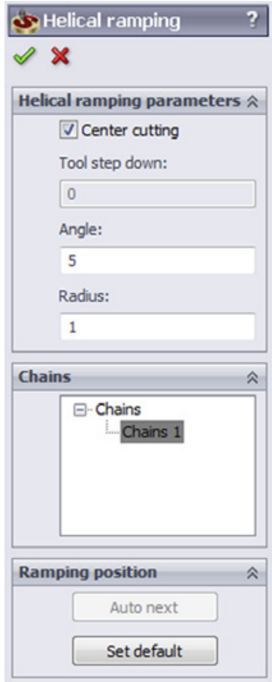
Click the **Data** button to define the Helical Ramping parameters.

The **Helical ramping** dialog box is displayed. This dialog box enables you to define the ramping position and the related parameters for each chain used in the Pocket operation.

The tool chosen for the operation has center cutting capabilities, therefore make sure that the **Center cutting** check box is selected.

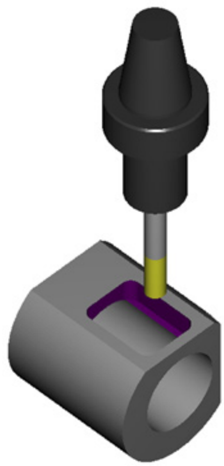
Set the **Angle** value to **5** and the **Radius** value to **1**.

Confirm the **Helical Ramping** dialog box with ✓.



40. Save and Calculate

Click the **Save & Calculate** icon in order to save the operation data and calculate the tool path.



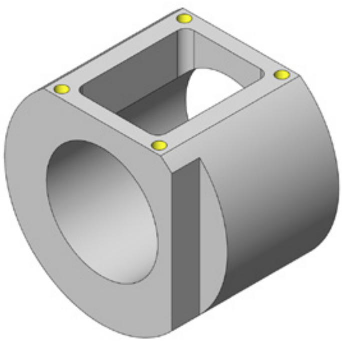
41. Simulate

Simulate the tool path with the **SolidVerify** mode.

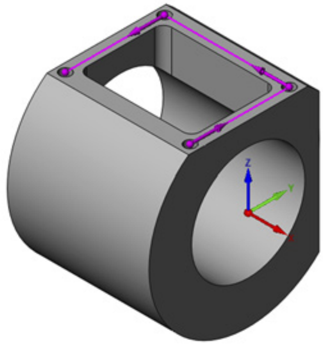
When the simulation is finished, close the **Pocket Operation** dialog box.

42. Define a Drilling operation

Define a new Drilling operation to machine four holes **Ø4** located on the top face of the model.



43. Define the Geometry

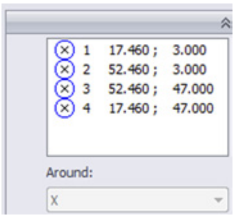


Choose **Machine CoordSys #1 Position #2** used in the previous operation.

In the **Geometry** area, click to start the geometry definition.

The **Drill Geometry Selection** dialog box is displayed. In the **Select centers by** area, choose the **Multi-positions** option and click on the top face of the model as shown.

SolidCAM selects all drill centers on the selected face. The coordinates of the selected hole centers are displayed in the **Drill Geometry Selection** dialog box.



Confirm the geometry selection with the button.

44. Define the Tool

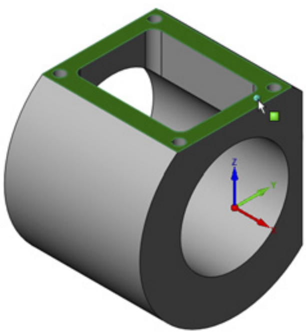
Click the **Select** button on the **Tool** page. The **Part Tool Table** dialog box is displayed.

Add a **Drill** tool.

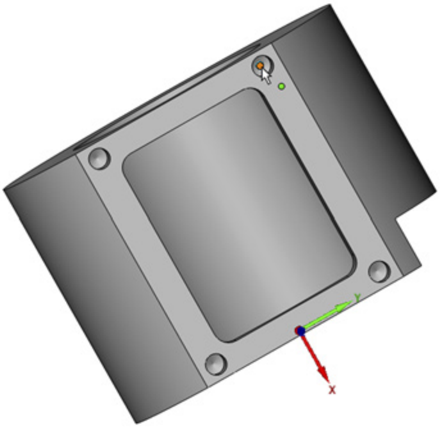
Set the **Diameter** to **4**. Choose the **BT40 ER16x70** tool holder from the Global holders table and click the **Select** button to choose the tool for the operation.

45. Define the Milling levels

Define the **Upper level** on the top face of the model as shown.

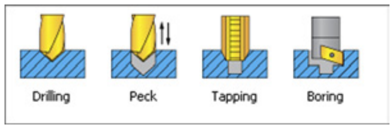


In the same manner, define the **Drill depth** by picking the drill cone vertex on the solid model.



46. Define the Drill cycle

On the **Technology** page, click the **Drill cycle type** button to choose the drill cycle.

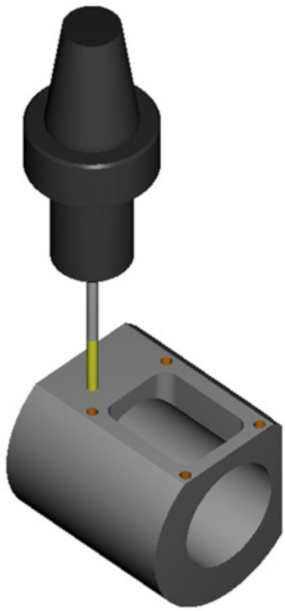


The drill cycles panel is displayed.
Choose the **Peck** option.

Click the **Data** button in the **Drill cycle** area to define the pecking parameters.

The **Drill Options** dialog box is displayed. This dialog box enables you to define the parameters of the chosen canned drill cycle.

Set the **Step down** to **2** and click **OK** to confirm the dialog box.



47. Save and Calculate

Click the **Save & Calculate** icon to save the operation data and calculate the tool path.

48. Simulate

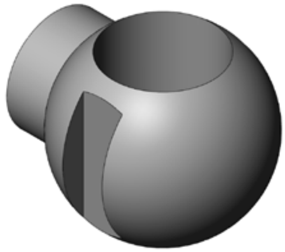
Simulate the tool path in the **SolidVerify** mode.

When the simulation is finished, close the **Drilling Operation** dialog box.

At this stage, the exercise is finished.

Exercise #17: Joint Part Machining

Define the CAM-Part and operations for the machining of the joint part presented on the illustration on a Mill-Turn CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Turning
- Indexial 4-axis milling on 4-axis machines

The SolidWorks model of the Joint (**Exercise17.sldprt**) is located in the **Exercises** folder.

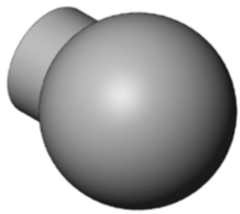
The following steps have to be implemented to reach the final CAM-Part:

1. Define the CAM-Part

At this stage, you have to define the CAM-Part, the CNC-controller, the Coordinate System, the Stock model and the Target model.
The **NTX1000** CNC-controller has to be chosen for this exercise.

2. Define the clamping fixture

Define the fixture for clamping of the part on the CNC-machine table.

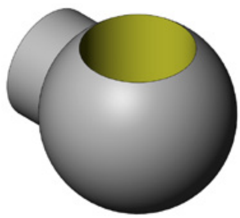


3. External turning

Machine the external turning faces using the automatically generated envelope.

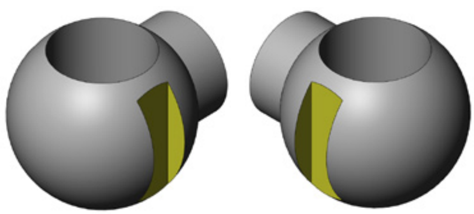
4. Drilling

Machine the hole located on the spherical surface.

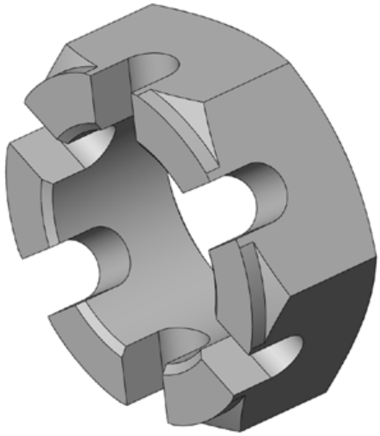


5. Side slots machining

Machine the two slots using the SolidCAM indexial milling capabilities.



Exercise #18: Slotted Nut Machining



Define the CAM-Part and operations for the machining of the slotted nut presented on the illustration on a Mill-Turn CNC-machine.

This exercise reinforces the following skills:

- CAM-Part definition
- Turning
- Indexial 4-axis machining

The SolidWorks model of the Slotted Nut (**Exercise18.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

1. Define the CAM-Part

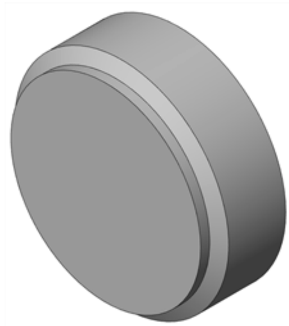
At this stage you have to define the CAM-Part, the CNC-controller, the Machine Coordinate System, the Stock model and the Target model.
The **NTX1000** CNC-controller has to be chosen for this exercise.

2. Define the clamping fixture

Define the fixture for clamping of the part on the CNC-machine table.

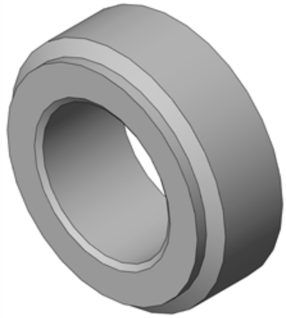
3. External turning

Define a Turning operation to obtain the following revolution body.



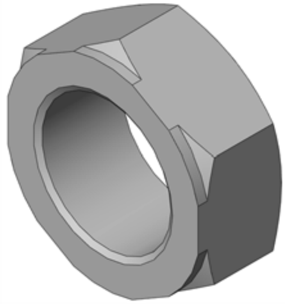
4. Internal turning

Machine the internal faces of the model.



5. Machine the side faces

Machine the side faces of the part.

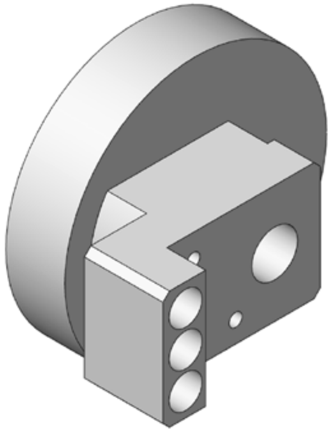


6. Machine the slots

Machine the slots located on the sides of the part.

Exercise #19: Connector Part Machining

Define the CAM-Part and operations for the machining of the Connector part presented on the illustration on a 4-axis Mill-Turn CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Turning
- Facial milling
- Indexial 4-axis machining on 4-axis CNC-machines

The SolidWorks model of the Connector part (**Exercise19.sldprt**) is located in the **Exercises** folder.

The following steps have to be implemented in order to reach the final CAM-Part:

1. Define the CAM-Part

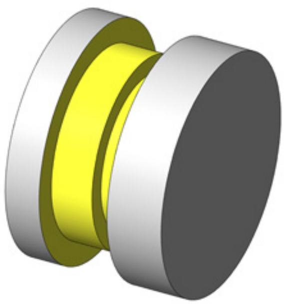
Define the CAM-Part, the CNC-controller, the Coordinate System, the Stock model and the Target model.
The **NTX1000** CNC-controller has to be chosen for this exercise.

2. Define the clamping fixture

Define the fixture for clamping of the part on the CNC-machine table.

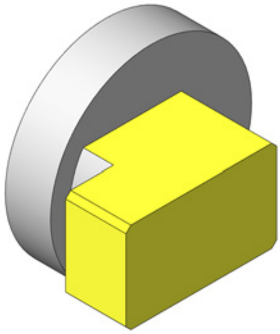
3. External turning

Machine the external turning faces using the automatically generated envelope.



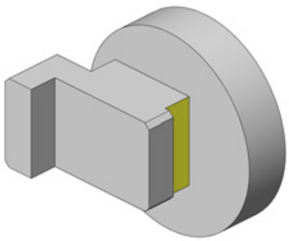
4. Facial milling

Machine the highlighted faces using the SolidCAM facial milling capabilities.



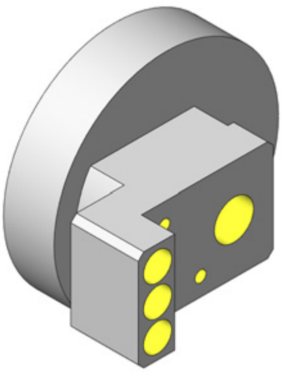
5. Slot machining

Machine the highlighted model faces.



6. Drilling

Drill all the holes.



4.4 Using MCO on Mill-Turn CNC-machines

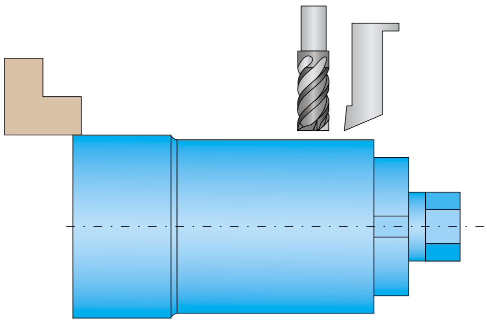
SolidCAM supports the use of the back spindle in the Mill-Turn module. This functionality enables you to use the back spindle of your CNC-machine for Turning and Milling machining with the part clamped in the main or back spindle. The part is transferred between the spindles, thus enabling you to complete the part machining from both sides in a single setup.

Each milling or turning operation can be performed with the part clamped in the main or back spindle.

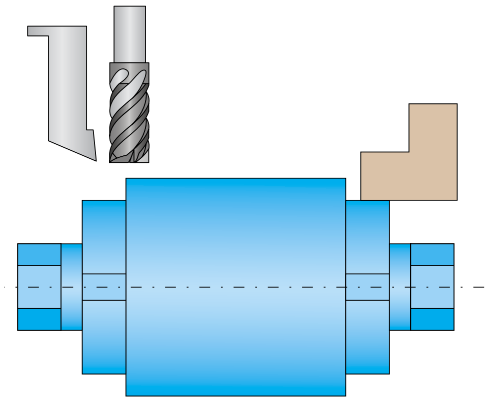
Operation sequence #1: General use

There are two optional stages in this operation sequence:

1. **SolidCAM Operations with Main spindle**. At this stage, only the main spindle is used for milling and turning operations; this is the standard way of working in SolidCAM when the back spindle does not exist.



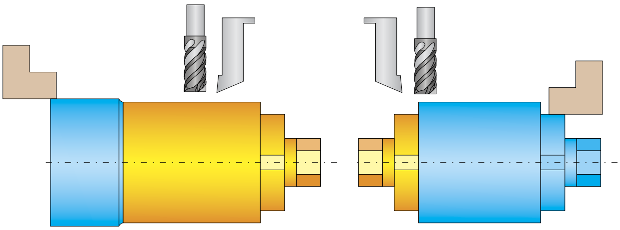
2. **SolidCAM Operations with Back spindle**. At this stage, the operations are performed with the back spindle only.



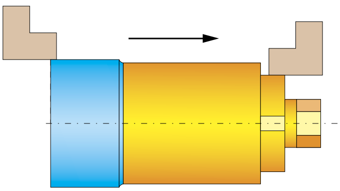
Operation sequence #2: Optimized tool use

SolidCAM enables you to define a combined sequence of operations that uses intermittently the main and back spindles. Each operation in the sequence is related either to the main or to the back spindle. **The Machine Control Operation** enables you to define the transfer of the machined part between the main and back spindles.

In this sequence, the combined operations with the main and back spindle use enable you to avoid unnecessary tool change (in cases when the same tool can work with the main and back spindles). In the beginning or end of all operations, the finished part is ejected from the back spindle and the partly machined part is transferred from the main spindle to the back spindle.



Machining with the main and back spindle



Transferring the part to the back spindle

Exercise #20: Machining with Back Spindle

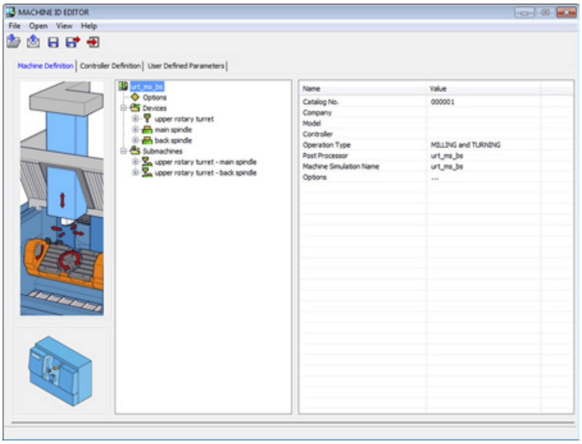
This exercise covers the process of Machine Control Operation definition in order to transfer a part from the main spindle to the back spindle.

1. Load the CAM-Part

Load the **Exercise16.prz** CAM-Part prepared earlier.

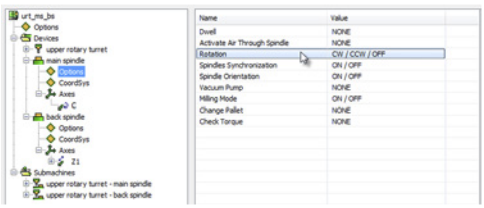
2. Open the Machine ID Editor

Double-click the **Machine** header in **SolidCAM Manager**. The **Machine ID Editor** dialog box is displayed.



The machine defined for this part machining consists of two tables called **main spindle** and **back spindle**, and one turret called **upper rotary turret**.

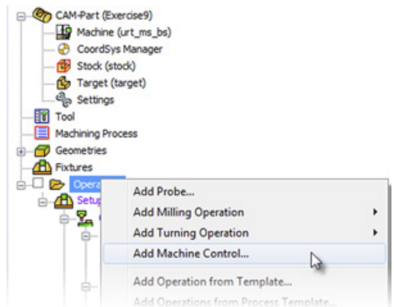
Double-click the **main spindle** entry to open the tree. Click **Options** to review the spindle definition in the right pane.



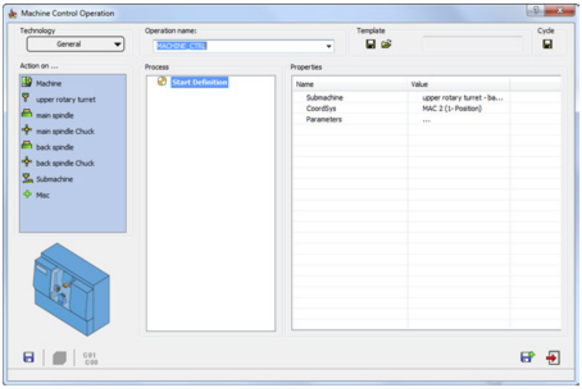
Click **Exit** ( ) to close the **Machine ID Editor** dialog box.

3. Add a Machine Control Operation (MCO)

Right-click the **Operations** header in **SolidCAM Manager** and choose **Add Machine Control** from the menu.



The **Machine Control Operation** dialog box is displayed.



Machine Control Operation

This operation enables you to manage and use the devices of the CNC-machine.

- **Action on**

  This section contains a list of available CNC-machine devices. Choose the device to be managed and drag-and-drop it into the Process section.

- **Process**

  In this section the various machine devices are added for performing actions such as movements, opening/closing the chuck jaws, tool change, etc.

- **Properties**

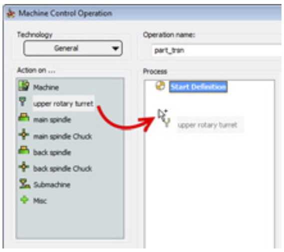
  This section contains a list of parameters of devices to be modified.

Using this operation, the part will be moved from the main spindle to the back spindle. The procedure is as follows: the jaws of the back spindle chuck will open, the back spindle will be moved to the part, its jaws will take the part and close, the jaws of the main spindle chuck will open to release the part, the back spindle will be returned to its initial position.

In the **Operation name** field, type in a new name: **part transfer**. This name will appear later in the **SolidCAM Manager** tree.

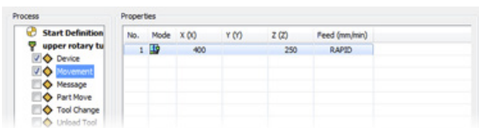
Click the **Start Definition** entry. Under **Properties**, select **MAC-1 Position #1** for **CoordSys**. The CAM-Part is located in the main spindle, therefore choose **upper rotary turret - main spindle** for the **Submachine**.

First, the Turret should retract to a safety position in order to avoid possible collisions with the back spindle. Drag and drop the **upper rotary turret** item from the **Action on** pane to the **Process** pane.



Click the **Movement** item located under the **upper rotary turret** item and define the movement properties in the right pane as follows:

- In the **Mode** cell, choose **MA**–Machine Coordinate mode ( )
- In the **X** cell, type in **400**
- In the **Z** cell, type in **250**
- In the **Feed** cell, choose **RAPID**



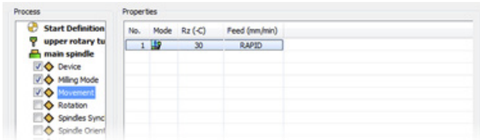
The first table, **main spindle**, should also be prepared to the operation. The angles of the two tables should be synchronized to enable the part transfer with an exact angle.

Drag and drop the **main spindle** item from the **Action on** pane to the **Process** pane.

Select the **Milling Mode** item to activate the 4th axis of the machine.

Select the **Movement** item and define the movement properties as follows:

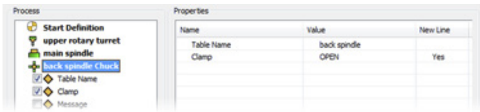
- In the **Mode** cell, choose **MA** ()
- In the **Rz** cell, type in **30**
- In the **Feed** cell, choose **RAPID**



Now you have to open the back spindle clamp.

Drag and drop the **back spindle Chuck** item from the **Action on** pane to the **Process** pane.

Select the **Clamp** item to enable the clamp opening.

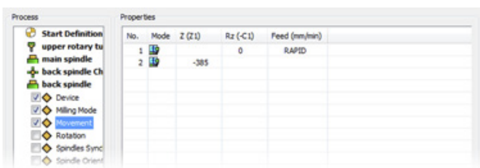


Similar to the first table, the second table, **back spindle**, should be prepared for the operation. Drag and drop the **back spindle** item from the **Action on** pane to the **Process** pane.

Select the **Milling Mode** item to activate the 4th axis of the machine.

Select the **Movement** item and define the movement properties as follows:

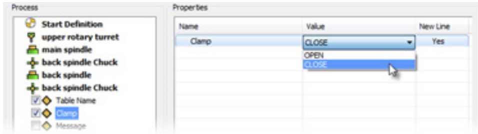
- For the first movement, in the **Mode** cell, choose **MA** ()
- In the **Rz** cell, type in **0**
- In the **Feed** cell, choose **RAPID**
- For the second movement, in the **Mode** cell, choose **MA** ()
- In the **Z** cell, type in **-385**



With the last movement, the back spindle approaches to the part in order to catch it. Now the back spindle chucks should be closed.

Define a new action for the **back spindle Chuck**.

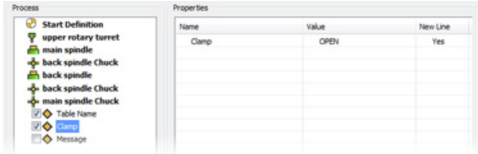
Select the **Clamp** item and choose the **CLOSE** option in the **Value** cell.



At the next action, the clamp of the main spindle should be released.

Define a new action for the **main spindle Chuck**.

Select the **Clamp** item and make sure the **OPEN** option is chosen in the **Value** cell.



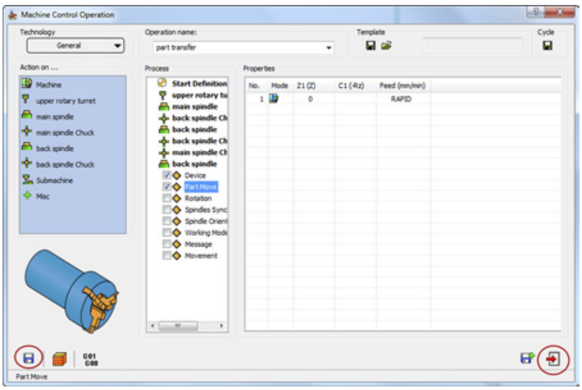
During the next action, the second table with the clamped part should retract to the initial position.

Define a new action for the **back spindle**.

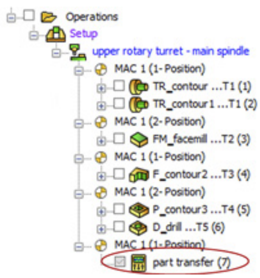
Select the **Part Move** item and define the movement properties as follows:

- In the **Mode** cell, choose **MA** ()
- In the **Z** cell, type in **0**
- In the **Feed** cell, choose **RAPID**

Click the **Save** icon to save the operation. Click the **Exit** icon to close the **Machine Control Operation** dialog box.



The **part transfer** operation appears in **SolidCAM Manager**.

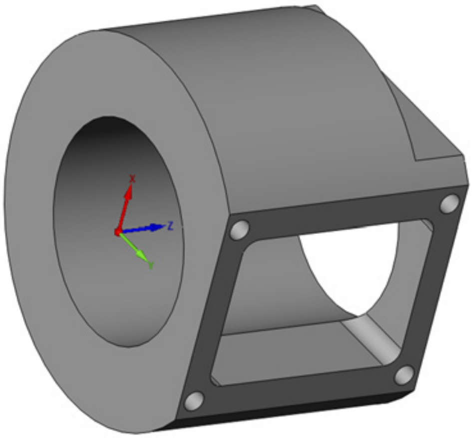


4. Perform turning with the back spindle

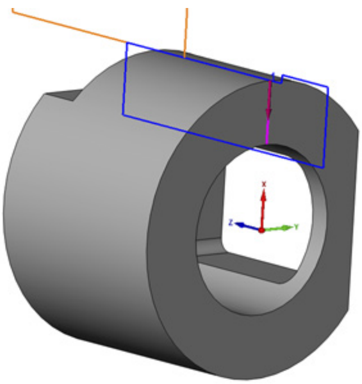
Add a Turning operation to machine the back face.

Define an additional Coordinate System for turning with the black spindle.

In the **Submachine** section, select **upper rotary turret - back spindle**. Choose the **Machine Coordinate System #2 Position #1**.



Define the geometry as shown.



Add a new **Ext. Turning** composite tool with the following parameters:

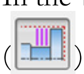
For the **Insert**:

- Set the **Insert Shape** to **W**
- Set the **IC Diameter** to **02**
- Set **Thickness** to **T3**
- Set the **Corner radius** to **04**

For the **Shank**:

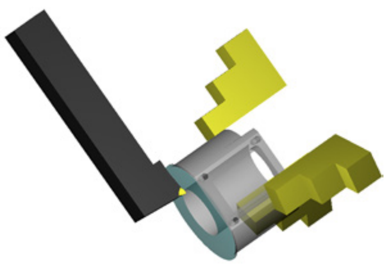
- Set the **Insert Lead Angle** to **L**
- Set the **M** value to **32**

Set **Mounting** to .

In the **Technology** page, select the **Face back** mode ().

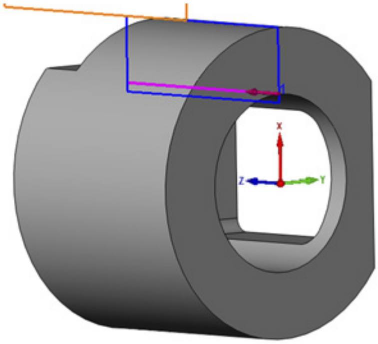
In the **Finish/Semi-finish** tab, select **ISO-Turning method** in the **Finish** section.

Save, calculate and simulate the operation in the SolidVerify mode.



5. Perform internal turning with the back spindle

Add a Turning operation to machine the internal part of the model.



In the **Submachine** section, select **upper rotary turret - back spindle**. Choose the **Machine Coordinate System #2 Position #1**. Define the geometry as shown.

Add a new **Int. Turning** composite tool with the following parameters:

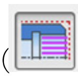
For the **Insert**:

- Set the **IC Diameter** to **09**
- Set **Thickness** to **T3**
- Set the **Corner radius** to **04**

For the **Shank**:

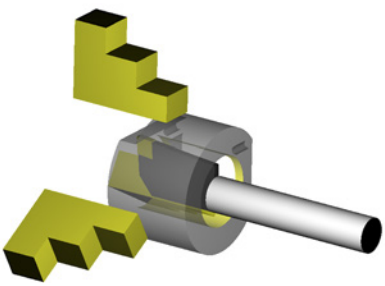
- Set the **Shank Diameter** to **20.00**
- Set the **Tool Length** to **150**

Set **Mounting** to  .

In the **Technology** page, select the **Face back** mode ().

In the **Finish/Semi-finish** tab, select **ISO-Turning method** in the **Finish** section.

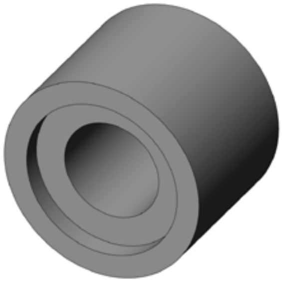
Save, calculate and simulate the operation in the SolidVerify mode.



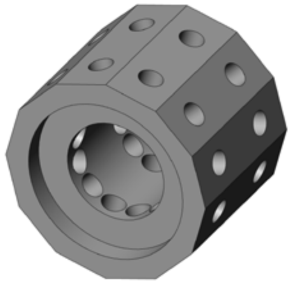
At this stage, the exercise is finished.

Exercise #21: Indexial Milling on 4-Axis CNC-Machines

This exercise covers the SolidCAM functionality of indexial 4-axis milling using 3-axis Mill-Turn CNC-machines. The exercise uses a CAM-Part with previously defined Turning operations.



The CAM-Part after the turning



The complete CAM-Part after the milling

The milling is performed in Profile and Drilling operations to machine each dodecahedron face and drill two holes on it. The circular pattern of these two operations is used to machine the other faces and holes.

1. Open the CAM-Part

Open the **Exercise21.prz** file.

2. Simulate the CAM-Part

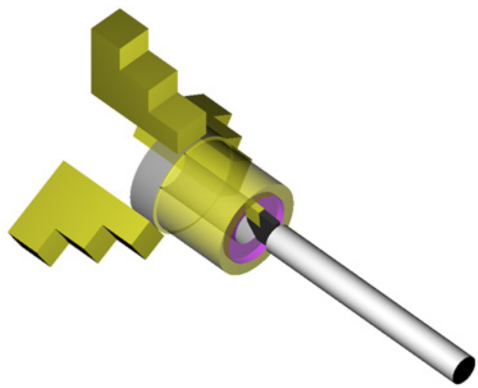
This CAM-Part contains a number of turning operations for machining of the external and internal faces.

Right-click the **Operations** header in **SolidCAM Manager** and choose **Simulate** from the menu. This command enables you to simulate the complete CAM-Part.

The **Simulate** command is also available for each operation separately by right-clicking on the operation icons in **SolidCAM Manager**.

Play the **SolidVerify** simulation.

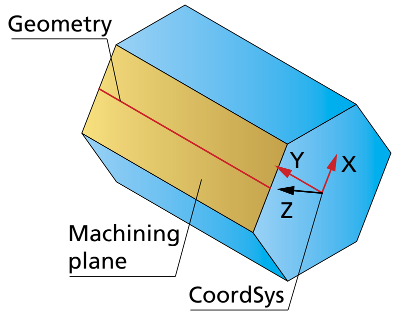
Close the **Simulation** control panel and return to **SolidCAM Manager**.



3. Define an additional CoordSys position

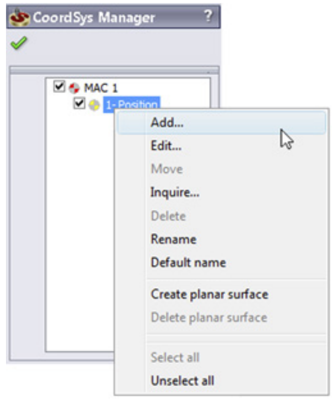
At this stage, you need to define an additional CoordSys position to perform milling of the dodecahedron face.

SolidCAM enables you to define a number of CoordSys positions for indexial 4-axis milling directly on the solid model. These positions correspond to the **Machine Coordinate System #1**. The geometry for the machining has to be located in the plane parallel to the **XY**-plane of such CoordSys position.



In the **SolidCAM Manager** tree, double-click the **CoordSys Manager** header.

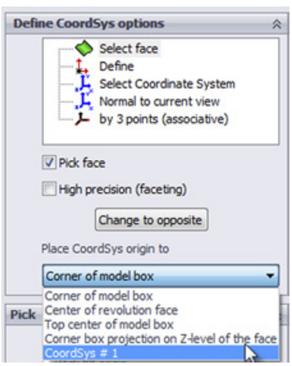
The **CoordSys Manager** dialog box is displayed in the SolidWorks **PropertyManager** area.

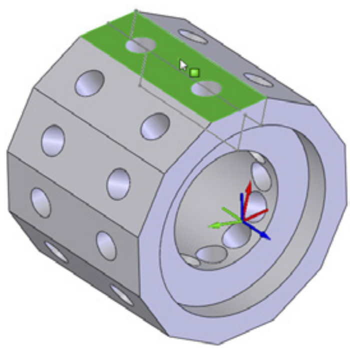


In the **CoordSys Manager** dialog box, right-click the **Machine Coordinate System #1 Position 1** and choose the **Add** command from the menu.

The **CoordSys** dialog box is displayed.

In the **Define CoordSys options** section, choose the **CoordSys #1** option from the **Place CoordSys origin to** list.





Rotate the model and click on its face as shown.
Click **Change to opposite**.

Note that you have to select the face that contains the line sketch that will be used in the next steps for the milling geometry definition.

The system automatically creates the CoordSys position in which the Z-axis is normal to the selected faces, and the axes X and Y are defined in accordance with the CNC-machine capabilities.

Click to confirm the **CoordSys** dialog box. Confirm the **CoordSys Data** dialog box with the **OK** button.

The defined **CoordSys Position #2** is displayed in the **CoordSys Manager** dialog box.

Close this dialog box with .

4. Add a Profile operation

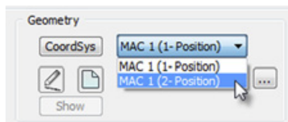
Right-click the last Turning operation icon in **SolidCAM Manager** and choose **Profile** from the **Add Milling Operation** submenu.

The **Profile Operation** dialog box is displayed.

5. Choose the Coordinate System position

On the **Geometry** page, choose the **Position #2** of the **Machine Coordinate System #1** from the **CoordSys** list.

This CoordSys position is used for the profile milling of the dodecahedron face.



6. Define the Geometry

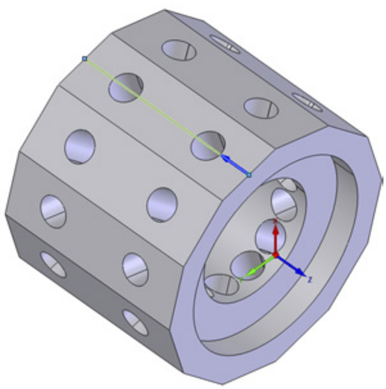
Click . The **Geometry Edit** dialog box is displayed.

Select the predefined sketch line as shown.

Make sure that the selected geometry chain direction is from the model front face backwards.

Click the **Accept chain** button to confirm the chain selection.

Close the **Geometry Edit** dialog box with . The geometry is defined.



7. Define the Tool

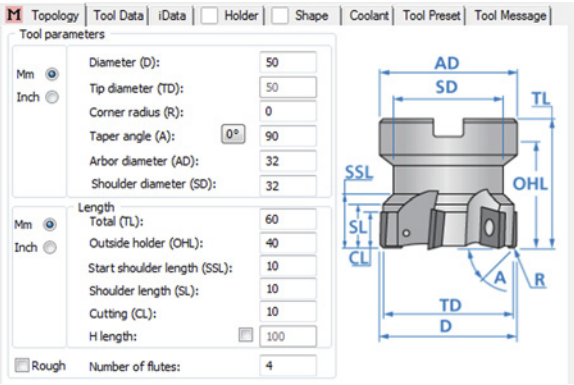
Click the **Select** button on the **Tool** page to choose a milling tool for the operation.

The **Part Tool Table** is displayed.

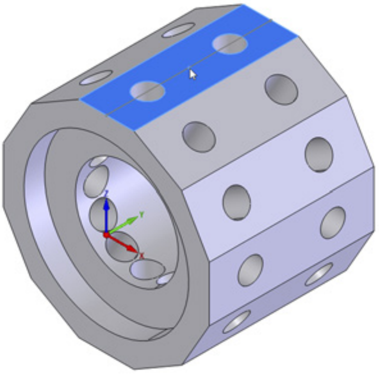
Click and choose the **Face Mill** tool. Define the following parameters:

- Set the **Diameter** to **50**
- Set the **Start shoulder length** to **10**
- Set the **Shoulder length** to **10**
- Set the **Cutting** length to **10**

Click **Select** to choose the tool for the operation.



8. Define the machining levels

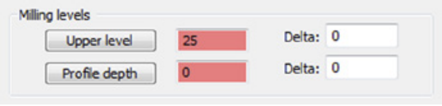


Click the **Upper level** button on the **Levels** page. This button enables you to define the plane where the machining will start directly on the solid model.

Click on the model face that contains the profile geometry as shown.

The value (**25**) of the **Upper level** is displayed in the **Pick Upper level** dialog box. Confirm your selection with .

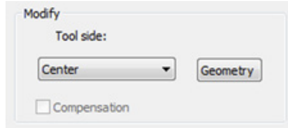
Click the **Profile depth** button. This option enables you to define the machining depth directly on the solid model.



Click on the same model face. The value of the **Lower level** is displayed. Confirm your selection with .

The **Profile depth** is calculated automatically according to the **Upper** and **Lower level** values. The **0** value of the **Profile depth** means that the machining is performed in a single Z-step.

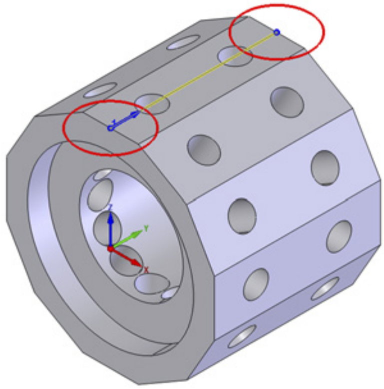
9. Define the Tool side



Define the tool position relative to the selected geometry.

Switch to the **Technology** page of the **Profile Operation** dialog box. Choose the **Center** option from the **Tool side** list. When this option is chosen, the center of the tool moves on the profile geometry.

Click the **Geometry** button. As you can see, the tool completely covers the face to be milled.



Click to return to the **Profile Operation** dialog box.

On the **Technology** page, select the **Rough** check box.

10. Define the Lead in and the Lead out

On the **Link** page, choose the **Tangent** option for both **Lead in** and **Lead out**.

For **Lead in**, set the **Length** value to **10**. The tool enters the material tangentially to the geometry from outside of the material.

For **Lead out**, set the **Length** value to **1**. This tangential lead out movement enables the tool to completely machine the geometry.

11. Calculate the operation

Click the **Save & Calculate** icon to save the operation data and calculate the tool path.

12. Simulate the operation

Click the **Simulate** button and choose the **SolidVerify** mode for the simulation. Play the simulation.

Close the simulation and exit from the **Profile Operation** dialog box.

The **SolidCAM Manager** dialog box is displayed.

13. Add a Drilling operation

Right-click the last Profile operation in **SolidCAM Manager** and choose **Drilling** from the **Add Milling Operation** submenu. The **Drilling Operation** dialog box is displayed.

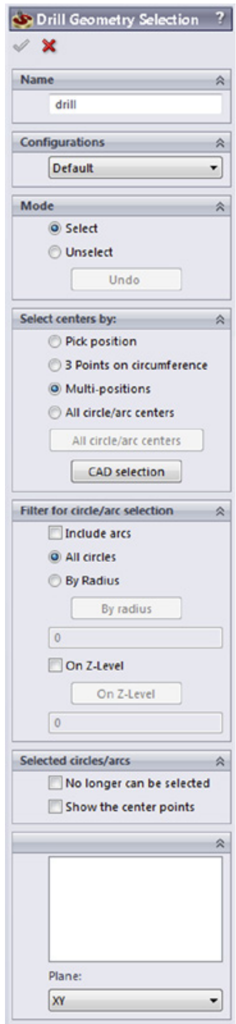
14. Choose the Coordinate System position

On the **Geometry** page, choose the same CoordSys position (**Machine CoordSys #1 Position #2**) that was used in the previous operation.

15. Define the Drill geometry

Click to start the geometry definition.

The **Drill Geometry Selection** dialog box is displayed. This dialog box enables you to select the drill positions.



Select/Unselect drill centers

You can switch between the **Select/Unselect** modes to add or remove drill positions from the geometry. The **Undo/Redo** button deletes or restores the last selection. All selected points are shown in the list at the bottom of the dialog box. To remove a point from the list, right-click on its name in the list and choose the **Delete** option from the menu, or the **Delete All** option to remove all points.

Selecting modes (Select centers by)

You can add drill positions to the current geometry using the following selection options:

- **Pick position**

  You can define the drill positions one by one using the CAD point selection options.

- **3 Points on circumference**

  Usually, all curves and arcs of imported models are converted into splines by the exporting CAD system. Due to the nature of spline curves or surface boundaries, you cannot pick a center position like you could on a circle or an arc. SolidCAM calculates the center position of an arc defined by three points positioned on the spline edges. This facilitates selecting drill centers on spline surfaces.

- **Multi-positions**

  You can select circular edges of the solid model. The drill position is automatically defined in the circular edge center.

- **All circle/arc centers**

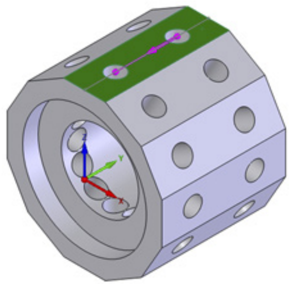
  SolidCAM searches the solid model for arcs and circles parallel to the XY-plane of the current Coordinate System and adds all center points as drill positions to the geometry.

CAD selection

SolidCAM enables you to select the drill geometry with the Host CAD tools.

For more information about the **Drill Geometry Selection** dialog box functionality, refer to the **SolidCAM Milling User Guide**.

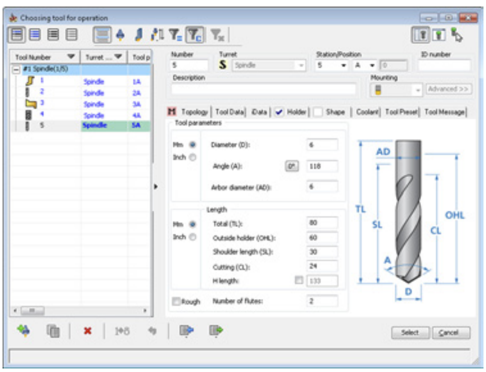
Use the default **Multi-positions** option in the **Select centers by** field. Click on the model face as shown. Two circular edges are recognized on the chosen face and their center points are selected.



Click ✔ to confirm the geometry definition.

16. Define the Tool

Click the **Select** button on the **Tool** page. The **Part Tool Table** is displayed.

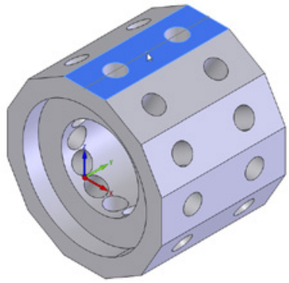


Add a **Drill** tool to the **Part Tool Table** and choose the **BT40 ER16x70** tool holder from the **Local Holders Table**.

Click **Select** to choose the tool for the operation.

The **Drilling Operation** dialog box is displayed.

17. Define the Drilling levels



On the **Levels** page, click the **Upper level** button to define the upper machining level directly on the solid model.

Click on the face as shown.

Confirm the **Pick Upper level** dialog box by clicking .

Set the **Drill depth** value to **15** and the **Delta** value to -**1**.

The **Delta** parameter defines the offset for the cutting depth that can be changed with its associativity preserved. The **Delta depth** value is always relative to the **Depth** defined for the operation.

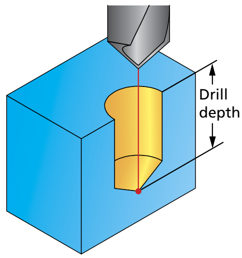
Define the **Depth type**.

Depth type

This option enables you to deepen the drilled hole in order to obtain a given diameter at the specified drill depth.

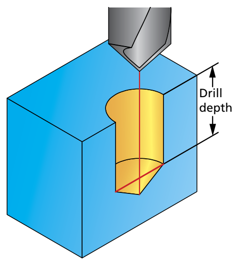
- **Cutter tip**

  The drill tip reaches the defined drill depth.



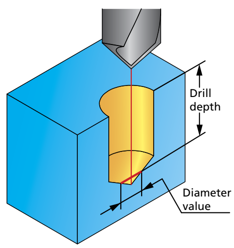
- **Full diameter**

  The drill reaches the defined drill depth with the full diameter.



- **Diameter value**

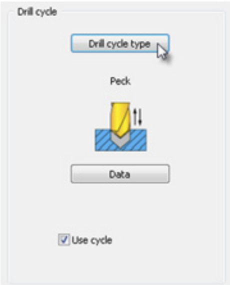
  The drill reaches the defined drill depth with the drill cone diameter specified in the edit box.



The **Diameter value** can vary from **0** all the way up to the drill tool diameter. A value greater than the drill tool diameter is automatically decreased to the drill tool diameter.

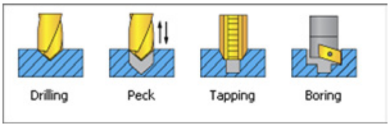
In the **Depth type** section, choose the **Full diameter** option

18. Define the drilling cycle



Switch to the **Technology** page of the **Drilling Operation** dialog box and click the **Drill cycle type** button.

The **Drill Cycle** dialog box is displayed.



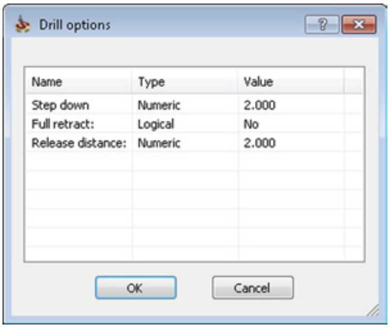
SolidCAM enables you to use the following types of drill canned cycles supported by the post-processor of the current CNC-controller:

Drilling. The drill travels in one single motion to the specified depth and retracts.
Peck. The drill travels in steps to the specified depth. At each depth step the tool rapidly retracts to the safety distance. Then the tool rapidly returns to the point where it continues drilling. This sequence is repeated until it reaches the final depth.
Tapping. The tap travels in one single motion to the specified depth and then retracts reversing its spin direction.
Boring. The boring tool travels in one single motion to the specified depth, stop its spin motion and retracts rapidly.

Choose the **Peck** option.

In the **Drill cycle** section, click the **Data** button to specify the pecking parameters.

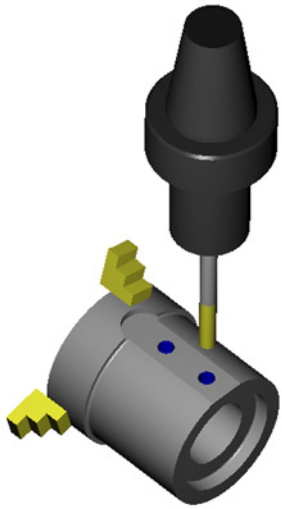
The **Drill options** dialog box is displayed. This dialog box enables you to define the parameters of the chosen drill canned cycle.



Step down

This option defines the depth that you want the tool to travel at each step.

Set the **Step down** to **2** and click **OK** to confirm the dialog box.



19. Save and calculate the operation

Click **Save & Calculate** to save the operation data and calculate the drilling tool path.

20. Simulate the operation

Click **Simulate**. The **Simulation** control panel is displayed.

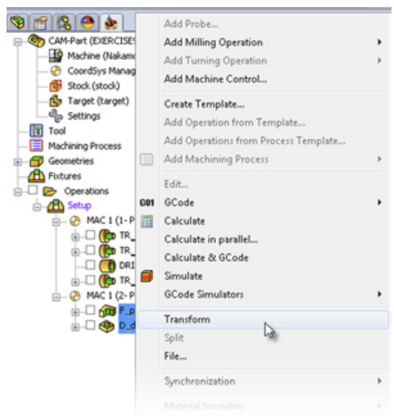
Simulate the tool path in the **SolidVerify** mode.

Close the **Simulation** control panel. Close the **Drilling Operation** dialog box.

21. Define the pattern of operations

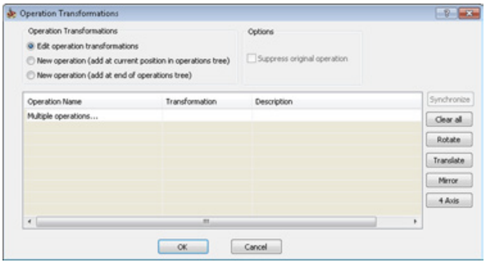
At this stage, the definition of the operations is finished. Now you have to define a circular pattern of operations around the revolution axis of the CAM-Part in order to machine all of the dodecahedron faces.

In the **SolidCAM Manager** tree, select the last two operations (Profile and Drilling). Right-click these operations and choose **Transform** from the menu.



The **Operation Transformations** dialog box is displayed. This dialog box enables you to perform various transformations such as rotating, translating, and mirroring of the selected operations.

Operation transformations



This section offers you the choice of whether the original operation will be transformed or its copy will be added for transformation and, in the latter case, enables you to define the placing of the new operation in the **SolidCAM Manager** tree.

The following options are available:

- **Edit operation transformations**

  This option enables you to perform transformation of the selected operation.

- **New operation (add at current position in operations tree)**

  This option enables you to add a copy of the selected operation and apply the transformation to the copied operation, without effect on the original operation. The transformed copy is inserted immediately after the original operation in the **SolidCAM Manager** tree. The original operation can be suppressed by selecting the appropriate check box in the **Options** section.

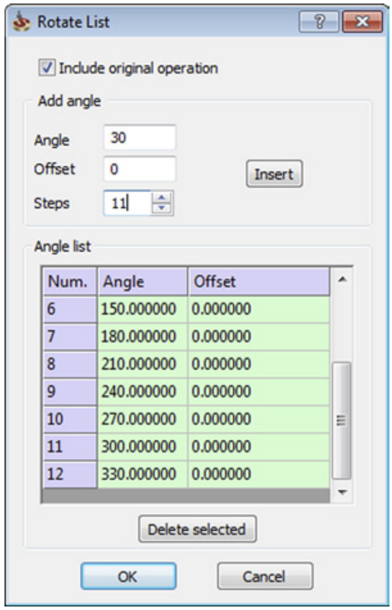
- **New operation (add at end of operations tree)**

  This option also enables you to add a copy of the selected operation and apply the transformation to the copied operation, without effect on the original operation. The transformed copy is inserted at the end of the operation list in the **SolidCAM Manager** tree. The original operation can be suppressed by selecting the appropriate check box in the **Options** section.

In the **Operation transformations** section, use the default **Edit operation transformations** option.

In this exercise, you need to define a circular pattern of the selected operations around the 4th axis. Click the **4 Axis** button located in the right part of the dialog box. The **Rotate List** dialog box is displayed.

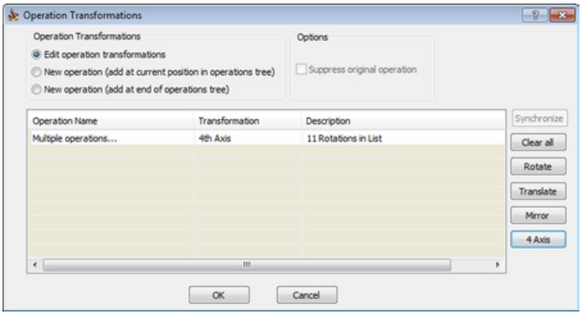
Select the **Include original operation** check box. When this check box is selected, the original operation tool path is included in the transforming action. In the **Add angle** section, set the **Angle** value to **30°**. This parameter defines the rotation angle for each pattern step. Enter **11** for the number of **Steps**.



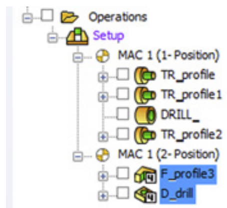
Click the **Insert** button. The specific value of the rotation angle is calculated for each step of the pattern.

Confirm the **Rotate List** dialog box with **OK**. The operations to be transformed appear in the **Operation Transformations** dialog box.

Confirm the dialog box with **OK**.



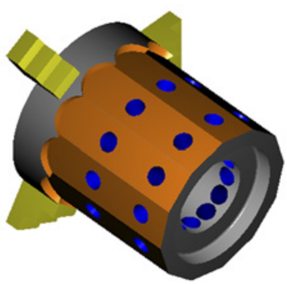
Note that the Profile and Drilling operations icons are marked with the 4 icon. This icon means that the operations were transformed using the C-axis.



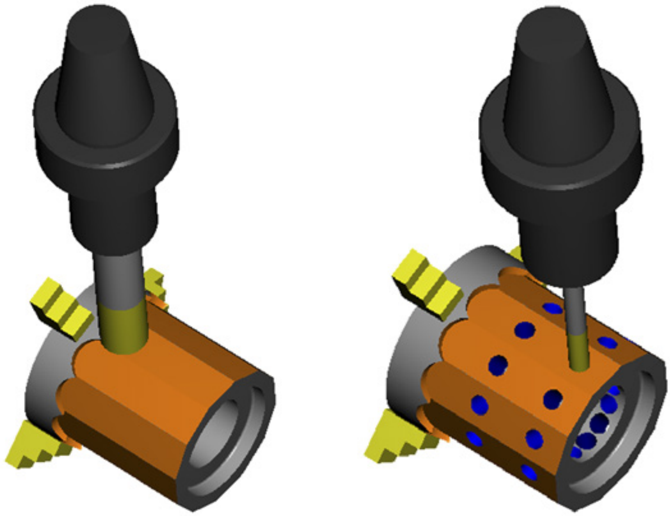
22. Simulate the operations

Right-click the selected Profile and Drilling operations and choose **Simulate** from the menu.

Simulate the tool path in the **SolidVerify** mode.



Note that the indexial machining of the part is performed as follows: cutting passes in milling operations are performed only along the Z-axis of the Machine Coordinate System, without movements in the XY-plane; drilling motion is performed along the X-axis of the Machine Coordinate System. The transformation of operations around the Z-axis is performed using the C-axis. Therefore, the GCode generated for this part can be used on any 3-axis CNC-machine (and also on machines with more than three coordinate axes).



Exercise #22: Indexial Milling on 5-Axis CNC-Machines

This exercise covers the process of CAM-Part definition and Milling on 5-axis Mill-Turn CNC-machines (supporting the XYZCB coordinates). The following flange model is used.

Since the CAM-Part definition process was explained in detail in previous exercises, some details of the CAM-Part definition process are omitted in this exercise.

1. Load the SolidWorks model

Load the **Exercise22.sldprt** model located in the **Exercises** folder.

This model contains a number of features forming the solid body and a few sketches that are used for the CAM-Part definition.

2. Start a new SolidCAM Mill-Turn part

Click **SolidCAM** in the main menu of SolidWorks and choose **Mill-Turn** from the **New** submenu. SolidCAM is started, and the **New Mill-Turn Part** dialog box is displayed. Confirm this dialog box with **OK**.

The **Mill-Turn Part Data** dialog box is displayed.

3. Define the CNC-controller

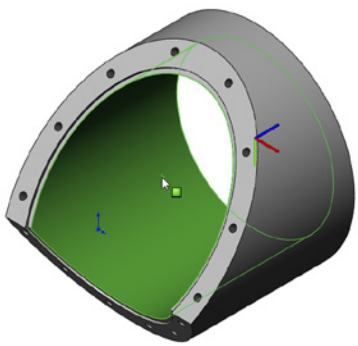
Choose the **ust_us_bs** CNC-controller from the list.

4. Define the Coordinate System

Click the **CoordSys** button to start the Coordinate System definition.

The **CoordSys** dialog box is displayed.

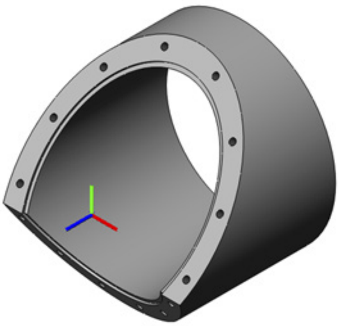
Click the cylindrical face of the model as shown.



The Z-axis of the Machine CoordSys is collinear with the revolution axis of the part.

Click the **Change to opposite** button in the **CoordSys** dialog box.

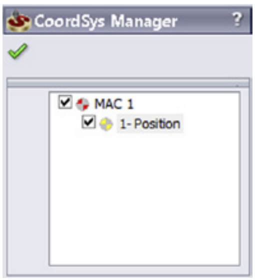
The Machine Coordinate System is defined on the model front face.



Click  to confirm the Machine Coordinate System definition.

The **CoordSys Data** dialog box is displayed. Close it with **OK**.

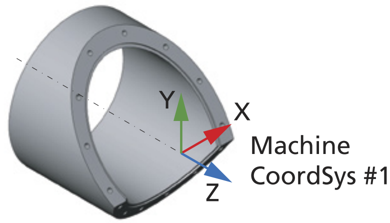
The **CoordSys Manager** dialog box is displayed in the SolidWorks **PropertyManager** area.



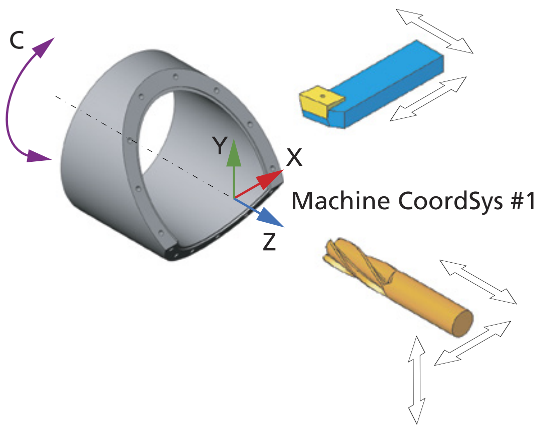
Coordinate Systems for 5-axis Mill-Turn machines

SolidCAM enables you to define **Machine CoordSys #1** with the Z-axis directed along the revolution axis.

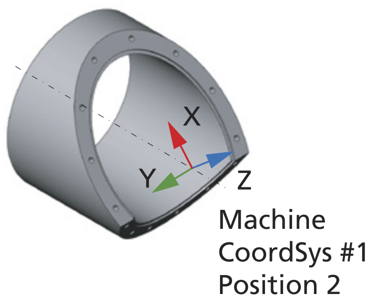
This **Machine Coordinate System** is used for Turning operations; the turning tool movements are performed in the ZX-plane.

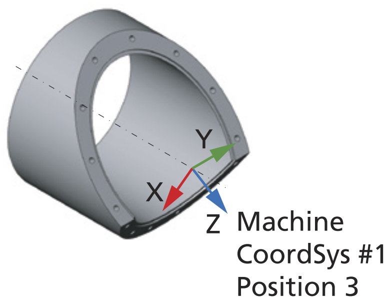


Machine CoordSys #1 is used also for facial milling, where the axis of the milling tool is parallel to the Z-axis of the Machine Coordinate System.



To perform indexial milling of the inclined faces, you need to define two additional positions for **Machine CoordSys #1**.





Confirm the **CoordSys Manager** dialog box with the button.

The **Mill-Turn Part Data** dialog box is displayed.

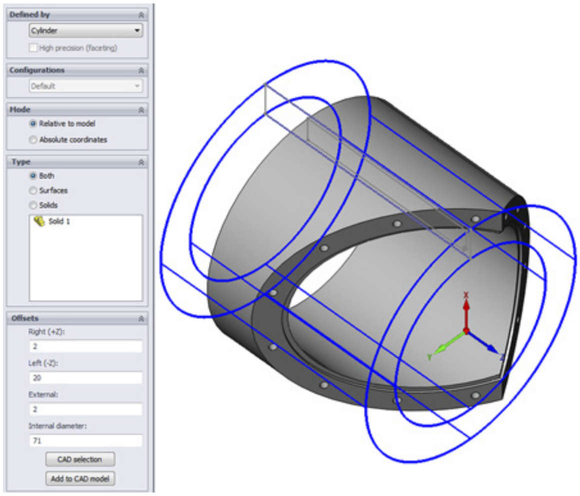
5. Define the Stock model

Click the **Stock** button in the **Mill-Turn Part Data** dialog box.

Choose the **Cylinder** option in the **Model** dialog box and click on the solid body to choose it for the Stock definition.

Define the following offsets:

- Set the **Right (+Z)** offset to **2** and **Left (-Z)** offset to **20**
- Set the **External** offset to **2**
- Set the **Internal diameter** to **71**



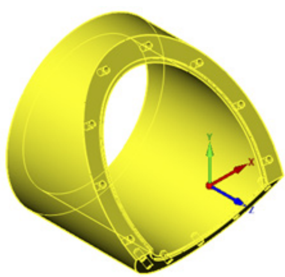
Set the **Facet tolerance** value to **0.01**.

Confirm the **Model** dialog box with the button.

6. Define the Target model

Click the **Target** button.

Select the solid body as the Target model.



7. Save the CAM-Part definition

In the **Mill-Turn Part Data** dialog box, click the **Save & Exit** button to confirm the CAM-Part definition.

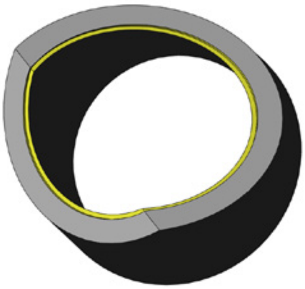
The **SolidCAM Manager** tree is displayed.

Technological overview

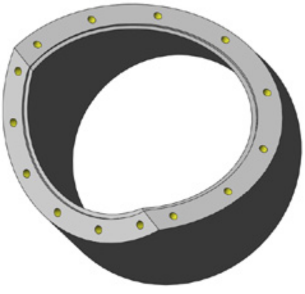
The current CAM-Part requires the following operations:

- **Facial milling** – machining of two inclined faces.

- **Step machining** – machining of the steps around the inclined faces.



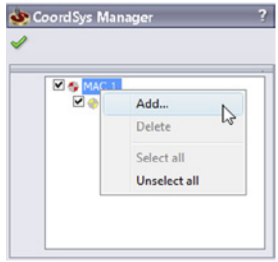
- **Holes machining** – machining of the pattern of holes **Ø3** located on the inclined faces.



All of these operations are performed using B-axis rotation, therefore additional Coordinate System positions of the **Machine CoordSys #1** have to be defined in order to perform the machining.

8. Define additional CoordSys positions

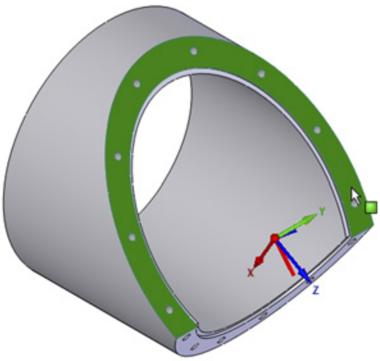
In the **SolidCAM Manager** tree, double-click the **CoordSys Manager** header to display the **CoordSys Manager** dialog box.

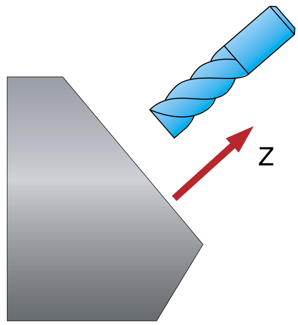


In the **CoordSys Manager** dialog box, right-click the **Machine CoordSys #1** item and choose the **Add** command from the menu.

The **CoordSys** dialog box is displayed. In the **Place CoordSys origin to** list, choose **CoordSys #1**.

Click on the model face as shown.

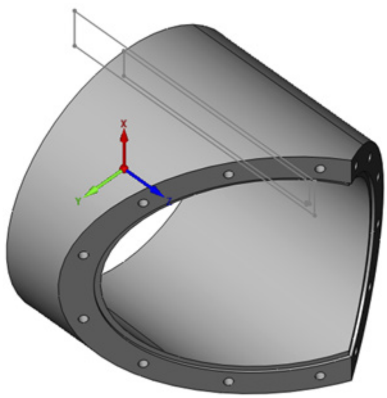




The **CoordSys Position #2** is defined at the origin of the **Machine CoordSys #1** with the Z-axis normal to the selected face.

Confirm the **CoordSys** dialog box with the button. Close the **CoordSys Data** dialog box with the **OK** button.

In the same manner, add the **CoordSys Position #3** related to **Machine CoordSys #1** to machine the second inclined face.



Add a new **Machine Coordsys # 2** in order to machine the back side of the model with the back spindle.

Confirm the definition of the Coordinate System positions and close the **CoordSys Manager** dialog box with the button.

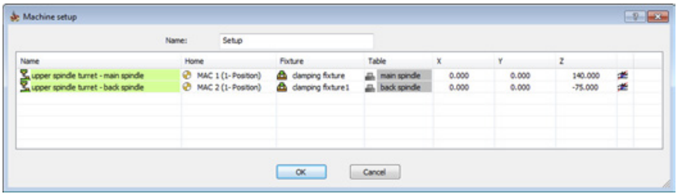
The **SolidCAM Manager** tree is displayed.

9. Define the Machine setup

Define two Machine setup positions: one with the main spindle and another with the back spindle.

Define two fixtures for each of the positions.

Set the model shift along the Z-axis. For the main spindle, set the **Z** shift to **140**. For the back spindle, set the **Z** shift to **-75**.



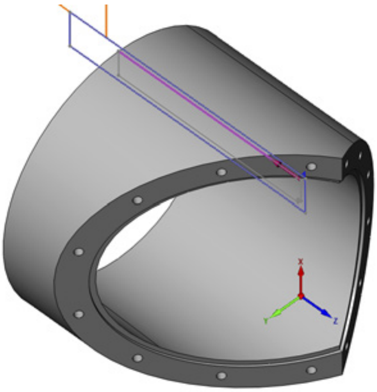
10. Add a Turning operation

Add a new Turning operation to machine the external faces of the model.

In the **Submachine** section, choose the **upper spindle turret - main spindle** option.

In the **Geometry** section, choose **Machine CoordSys #1 Position #1** from the list and click .

The **Geometry Edit** dialog box is displayed. Select the sketch entity as shown.



11. Define the Tool

Add a new **Ext. Turning** composite tool with the following parameters:

For the **Insert**:

- Set the **IC Diameter** to **09**
- Set **Thickness** to **T3**
- Set the **Corner radius** to **04**

For the **Shank**:

- Set the **Insert Lead Angle** to **L**
- Set the **M** value to **32**

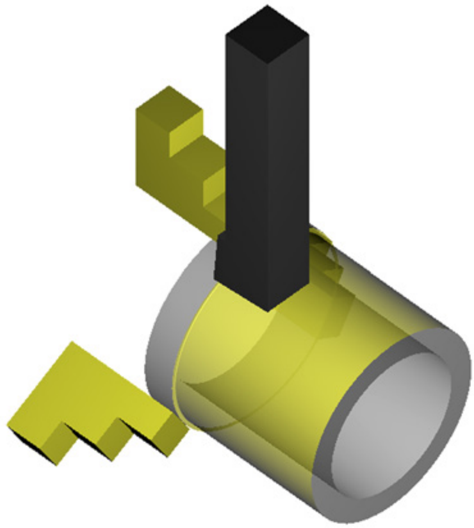
Set **Mounting** to .

12. Define the technological parameters

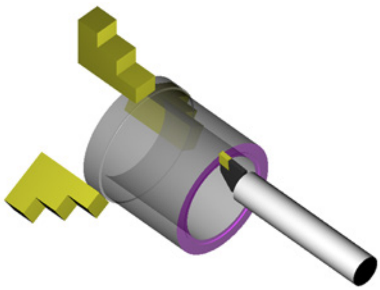
In the **Technology** page, select the **Long external** mode.

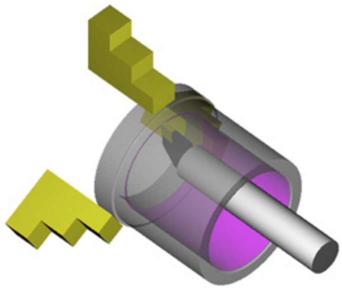
In the **Semi-finish/finish** tab, select **ISO-Turning method** in the **Finish** section.

Save, calculate and simulate the operation in the SolidVerify mode.



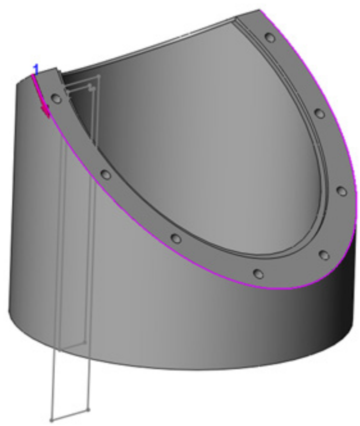
Add two more turning operations to machine the front face and the internal part of the model. Simulate the operations in the SolidVerify mode.





13. Add a Profile operation

Add a new Milling Profile operation to machine the inclined face.



14. Define the Geometry

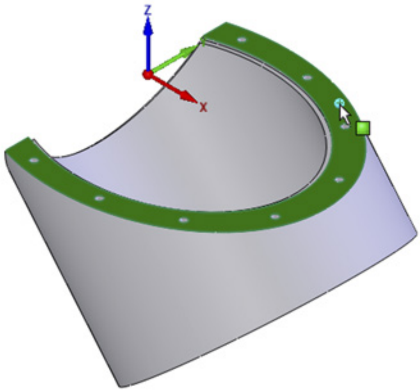
In the **Submachine** section, choose the **upper spindle turret - main spindle** option.

In the **Geometry** section, choose **Machine CoordSys #2 Position #1** from the list and click .

The **Geometry Edit** dialog box is displayed. Select the model edge as shown.

15. Define the Tool

Define a new **Ø16 End Mill** for the operation.



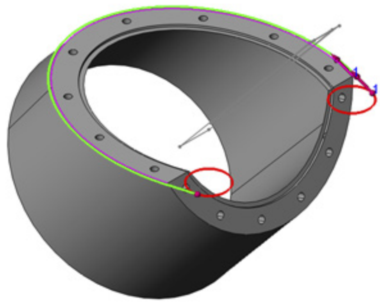
16. Define the Milling levels

Click the **Profile depth** button.

Define the Profile depth directly on the solid model by clicking on the face as shown.

Confirm the definition with the button.

17. Define the Tool side

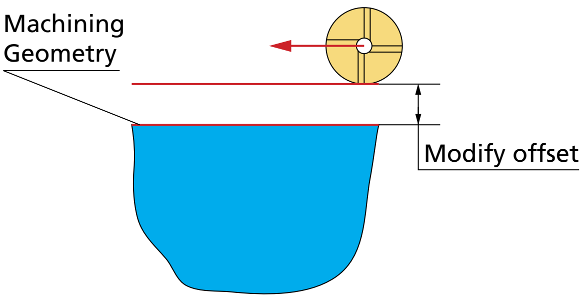


Switch to the **Technology** page of the **Profile Operation** dialog box. In the **Modify** section, click the **Geometry** button to check the tool location relative to the selected geometry.

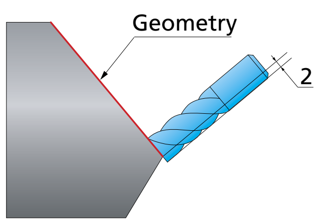
The tool has to be located inside the geometry. If it is not located correctly relative to the geometry, change the tool side.

In the **Modify Offset** section, define the offset from the geometry.

The **Offset** parameter enables you to define the offset for the machining geometry. The machining is performed at the specified offset.



Set the **Modify offset** value to **-1**. With a negative offset value, SolidCAM performs the finishing cut that overlaps the geometry.



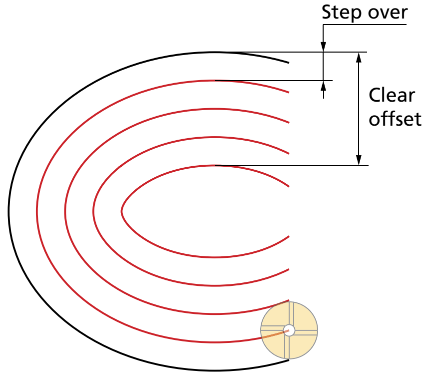
18. Define the Roughing parameters

On the **Technology** page of the **Profile Operation** dialog box, select the **Rough** check box.

Set the **Wall offset** to **0.5**.

In the **Clear offset** area, set the **Offset** value to **20** and the **Step over** value to **10**.

Select the **Finish** check box and set the **Number of passes** to **1**.



19. Define the Lead in and the Lead out

On the **Link** page, choose the **Feed** option for **Ramping**.

Choose the **Tangent** option for **Lead in** and set the **Length** value to **10**. For **Lead out**, select the **Same as Lead in** check box.

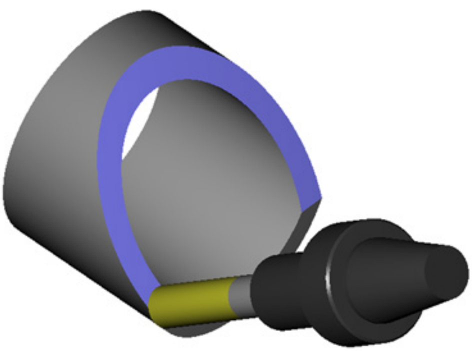
20. Save and Calculate

Click the **Save & Calculate** icon to save the operation parameters and calculate the tool path.

21. Simulate the operation

Simulate the operation in the **SolidVerify** mode.

Close the **Profile Operation** dialog box with the **Exit** button.



22. Add a Profile operation

Add a new Profile operation to machine the step located on the machined inclined face.

23. Define the Geometry

In the **Submachine** section, choose the **upper spindle turret - main spindle** option.

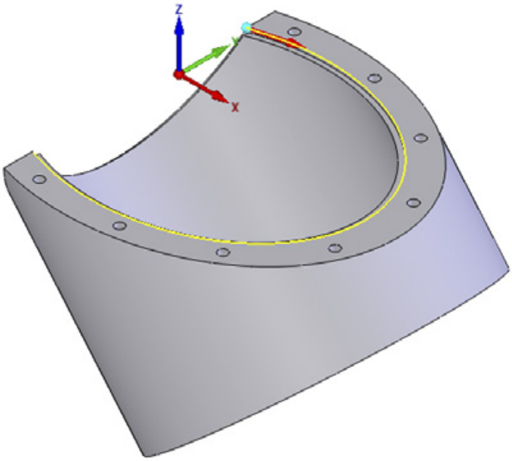
In the **Geometry** section, choose the **Machine CoordSys #1 Position #2**.

Click . The **Geometry Edit** dialog box is displayed.

Select the model edge as shown.

Confirm the edge selection with the **Accept Chain** button in the **Geometry Edit** dialog box.

Confirm the geometry definition with the button.



24. Define the Tool

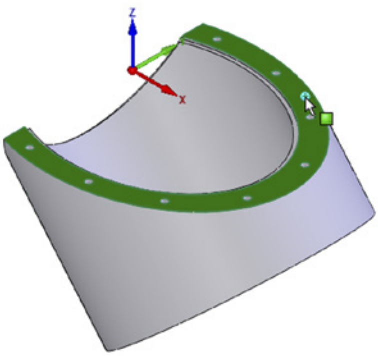
In the **Part Tool Table**, choose the tool used in the previous operation (**Ø16 End Mill**).

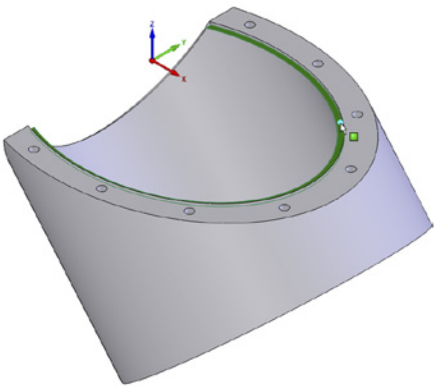
25. Define the Milling levels

Define the **Milling levels** directly on the solid model.

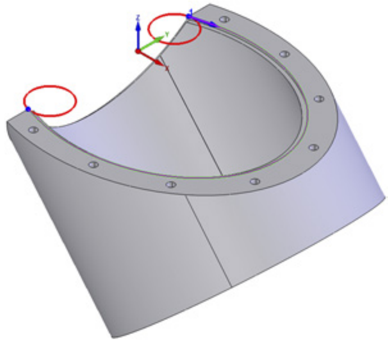
Use the inclined face for the **Upper level** definition.

Use the lower face of the step for the **Profile depth** definition.





26. Define the Tool side



In the **Tool side** section of the **Technology** page, click the **Geometry** button to check the tool position relative to the selected geometry.

The tool has to be located inside the geometry; if it is not located correctly, change the tool side.

In the **Profile Operation** dialog box, select the **Finish** check box and set the **Step down** value to **1**. With this value, the step is machined in a single cut.

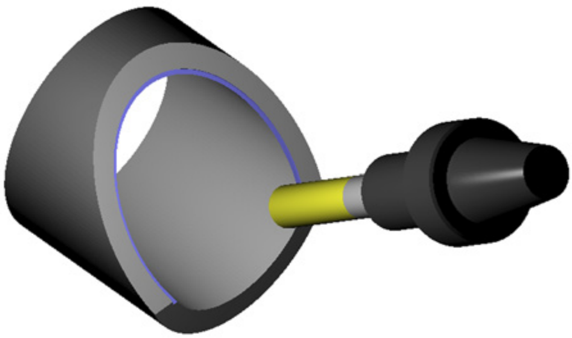
27. Define the Lead in and the Lead out

On the **Link** page, choose the **Tangent** option for **Lead in** and set the **Value** to **10**. For **Lead out**, select the **Same as Lead in** check box.

28. Save and Calculate

Click the **Save & Calculate** icon to save the operation parameters and calculate the tool path.

29. Simulate the operation



Simulate the operation in the **SolidVerify** mode.

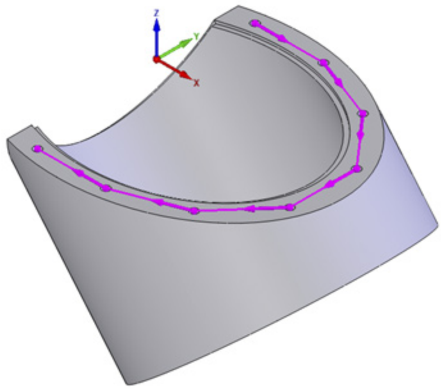
30. Add a Drilling operation

Add a new Drilling operation to machine holes **Ø3** located on the inclined face.

31. Define the Geometry

In the **Submachine** section, choose the **upper spindle turret - main spindle** option.

In the **Geometry** section, choose **Machine CoordSys #1 Position #2** for the operation.



Click to define the Geometry. The **Drill Geometry Selection** dialog box is displayed.

In the **Select centers by** field, choose the **Multi-positions** option. Select the inclined face as shown. SolidCAM automatically determines all centers of the holes located on the selected face.

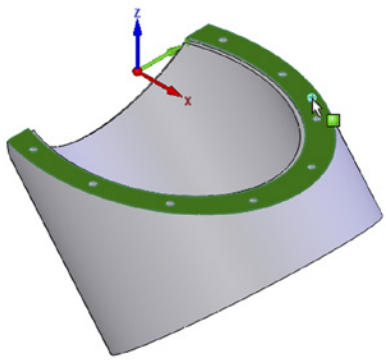
Confirm the geometry selection by clicking .

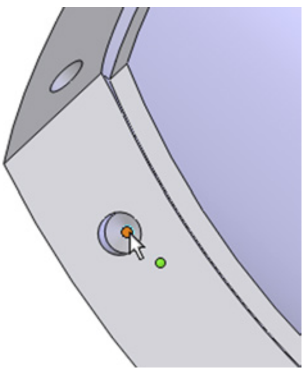
32. Define the Tool

Define a new drilling tool **Ø3** for the operation.

33. Define the Milling levels

Define the **Upper level** and the **Drill depth** directly on the solid model.
Define the **Upper level** by clicking on the inclined face of the model as shown.
Pick the drill cone vertex to define the **Drill depth**.





34. Define the Drill cycle

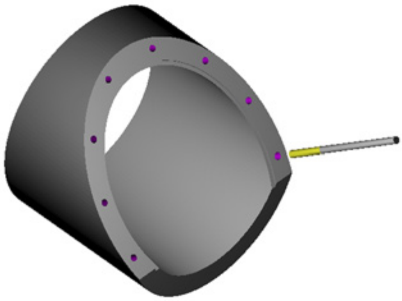
In the **Technology** page, choose the **Peck** drill cycle for the operation.
In the **Drill Options** dialog box, set the **Step down** to **2**.

35. Save and Calculate

Click the **Save & Calculate** icon to save the operation parameters and calculate the tool path.

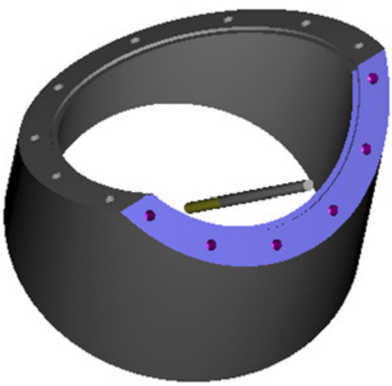
36. Simulate the operation

Simulate the operation in the **SolidVerify** mode.



37. Define the machining of the other side

Repeat **Steps #13-36** for the other inclined face of the model.
Use **Machine CoordSys #1 Position #3** for the operations.



38. Add an MCO Operation

Right-click the **Operations** header in **SolidCAM Manager** and choose **Add Machine Control** from the menu.

The **Machine Control Operation** dialog box is displayed.

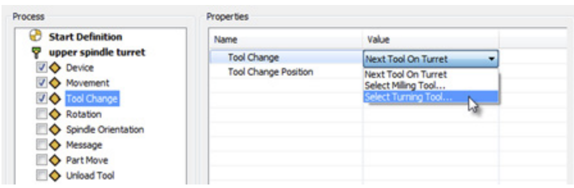
Click the **Start Definition** entry. Under **Properties**, select **MAC-1 Position #1** for **CoordSys**. The CAM-Part is located in the main spindle, therefore choose **upper spindle turret - main spindle** for the **Submachine**.

First, the Turret should retract to a safety position in order to avoid possible collisions with the back spindle. Drag and drop the **upper spindle turret** item from the **Action on** pane to the **Process** pane.

Click the **Movement** item located under the **upper spindle turret** item and define the movement properties in the right pane as follows:

- In the **Mode** cell, choose **MA**–Machine Coordinate mode ( )
- In the **X** cell, type in **400**
- In the **Z** cell, type in **250**
- In the **Feed** cell, choose **RAPID**

Click the **Tool Change** item located under the **upper spindle turret** item and define the tool change properties in the right pane. In the **Tool Change** row, double-click **Next Tool On Turret**. In the list, choose the **Select Turning tool** option.



The **Part Tool Table** is displayed with all the tools used for machining the part. Click to define a new turning tool.

From **Composite Tools**, choose the **Ext. Grooving** tool. Define the tool with the following parameters:

- For the **Insert**, use the default parameters
- For the **Shank**, set the **Cutting direction** to **R (Right)**.

Click the **Select** button. The tool is defined and prepared for the next operation.

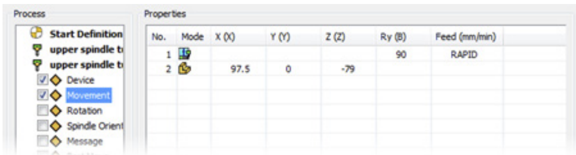
Define one more action for the **upper spindle turret**. Drag and drop the **upper spindle turret** item from the **Action on** pane to the **Process** pane.

Select the **Movement** item located under the **upper spindle turret** item and define the movement properties in the right pane as follows:

- In the **Mode** cell, choose **MA** ()
- In the **Ry** cell, type in **90**
- In the **Feed** cell, choose **RAPID**

Start a second line of definitions for the Part Coordinate System:

- In the **Mode** cell, choose **PA** ()
- In the **X** cell, type in **97.5**
- In the **Z** cell, type in **-79**



The next action you need to perform is to open the jaws of the back spindle chuck.

Drag and drop the **back spindle Chuck** item from the **Action on** pane to the **Process** pane.

Select the **Clamp** item to enable the clamp opening.

The second table, **back spindle**, should be prepared for the operation. Drag and drop the **back spindle** item from the **Action on** pane to the **Process** pane.

Select the **Spindles Synchronization** item to enable synchronization with other devices.

Select the **Movement** item and define the movement properties as follows:

- For the first movement, in the **Mode** cell, choose **MA** ()
- In the **Z** cell, type in **-360**
- In the **Feed** cell, choose **RAPID**

With the previous movement, the back spindle approaches to the part in order to catch it. Now the back spindle chucks should be closed.

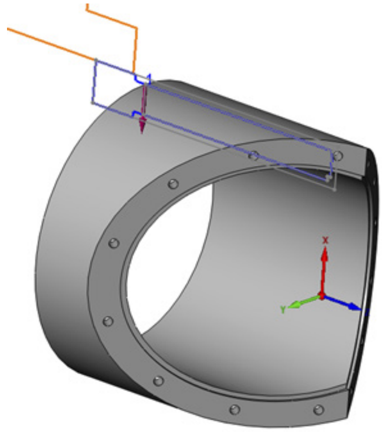
Define a new action for the **back spindle Chuck**.

Select the **Clamp** item and choose the **CLOSE** option in the **Value** cell.

At this stage, the part is held both in the main and back spindles.

Click the **Save** button to save the operation. Click the **Exit** button to close the **Machine Control Operation** dialog box.

39. Perform cutting off



Add a Cutoff operation to perform parting of the machined part.

With the **MAC-1 Position #1**, define the geometry as shown.

Note the direction of the geometry. In this operation, the geometry directed to the model rotation axis should be used.

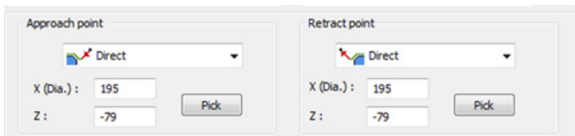
Use the **External grooving** tool defined in the previous operation.

In the **Technology** page, use the default **Long external** mode. In the **Offset** section, set the **Offset Z** to **-1**.

In the **Link** page, choose the **Direct** option for the **Approach** and **Retract points**.

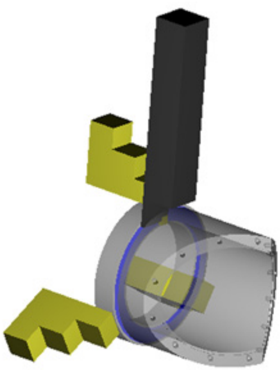
The point coordinates are the same as defined for the movement in the previous operation:

- Set **X (Dia.)** to **195**
- Set **Z** to **-79**



Click the **Save & Calculate** icon to save the operation parameters and calculate the tool path.

Simulate the operation in the **SolidVerify** mode.



At this stage, the part should be machined from the rear side to perform the finishing cut of the back face.

40. Add an MCO Operation

Right-click the **Operations** header in **SolidCAM Manager** and choose **Add Machine Control** from the menu.

The **Machine Control Operation** dialog box is displayed.

Click the **Start Definition** entry. Under **Properties**, select **MAC-1 Position #1** for **CoordSys**. The CAM-Part is located in the main spindle, therefore choose **upper spindle turret - main spindle** for the **Submachine**.

Drag and drop the **back spindle** item from the **Action on** pane to the **Process** pane.

Click the **Part Move** item located under the **back spindle** item and define the movement properties in the right pane as follows:

- In the **Mode** cell, choose **MA** ( )
- In the **Z** cell, type in **0**
- In the **Feed** cell, choose **RAPID**

Click the **Save** button to save the operation. Click the **Exit** button to close the **Machine Control Operation** dialog box.

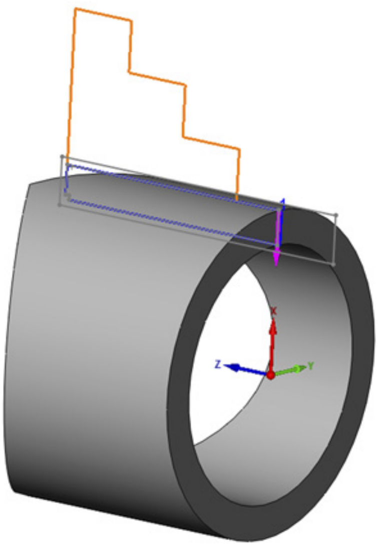
41. Perform back face turning

Add a Turning operation to machine the back face of the model.

In the **Submachine** section, choose **upper spindle turret - back spindle**.

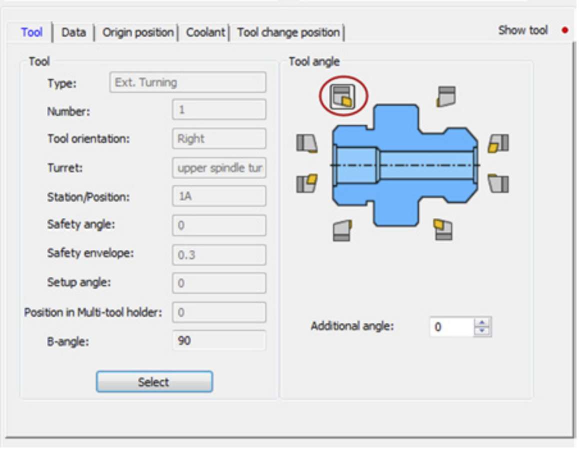
With the **MAC-2 Position #1**, define the geometry as shown.

Note the direction of the geometry. In this operation, the geometry directed to the model rotation axis should be used.



Use the tool #1 (**Ext. Turning**) defined earlier.

In the **Tool** page, set **Tool angle** as shown.

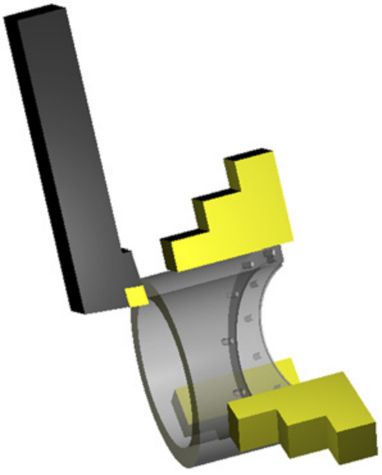


In the **Technology** page, use the **Face back** mode. In the **Semi-finish/finish** page, choose the **ISO-Turning method** in the **Finish** section.

Click the **Save & Calculate** icon to save the operation parameters and calculate the tool path.

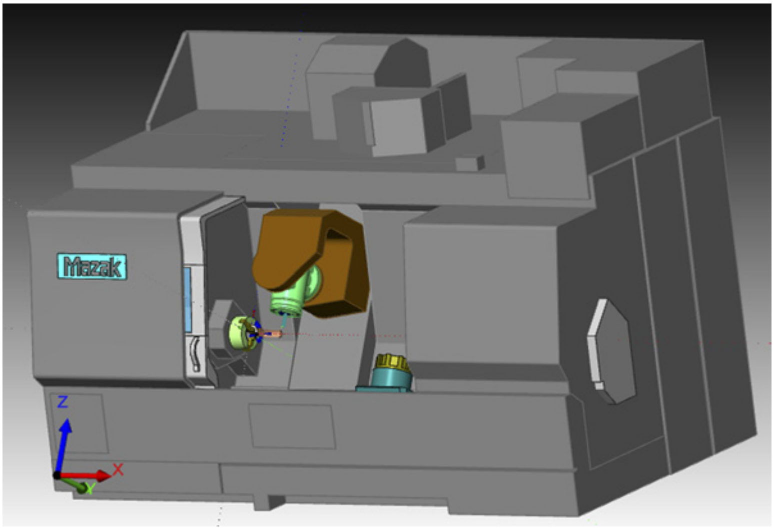
Simulate the operation in the **SolidVerify** mode.

At this stage, the exercise is finished.

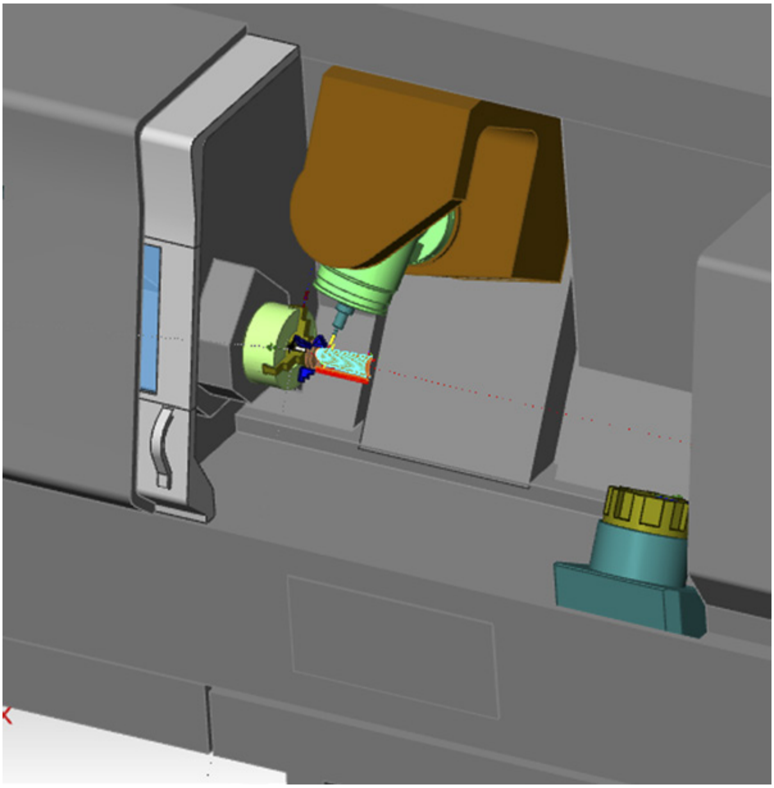


5 Simultaneous 5-Axis Milling on Mill-Turn CNC-Machines

SolidCAM enables you to perform Simultaneous 5-Axis machining on Mill-Turn CNC-machines.

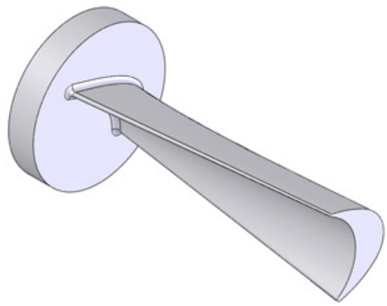


For more detailed explanation on the Simultaneous 5-Axis operations, refer to the **SolidCAM Sim. 5-Axis Machining User Guide**.



Exercise #23: Turbine Blade Machining

This exercise covers the process of CAM-Part and operations definition for Simultaneous 5-Axis machining of the part on a Mill-Turn CNC-machine.



1. Load the SolidWorks model

Load the **Exercise23.sldprt** model located in the **Exercises** folder.

2. Start the Mill-Turn part definition

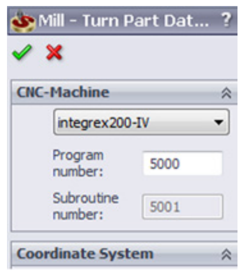
Start SolidCAM and confirm the **New Mill-Turn Part** dialog box.

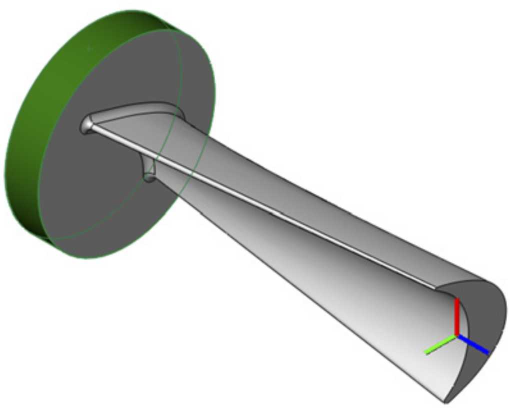
In the **Mill-Turn Part Data** dialog box, choose the **Mill_Turn_5_Axis** CNC-controller from the list.

3. Define the Coordinate System

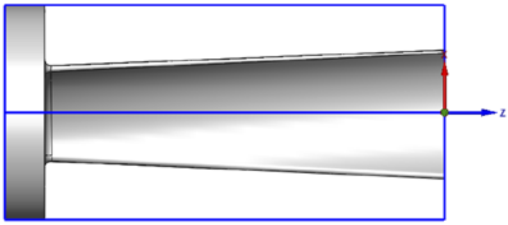
Define the Coordinate System as shown using the **CoordSys** dialog box.

This Coordinate System will be used for Milling and Turning operations.





4. Define the Stock model



Define the Stock model using the **Cylinder** option with all of the offsets set to **0**.

Set the **Facet tolerance** value to **0.01**.

5. Define the Target model

For further turning geometry definition, you will need the envelope of the part. To generate the Envelope sketch during the Target model definition, choose the **Envelope** option under **Generate Envelope/Section** in the **Model** dialog box.

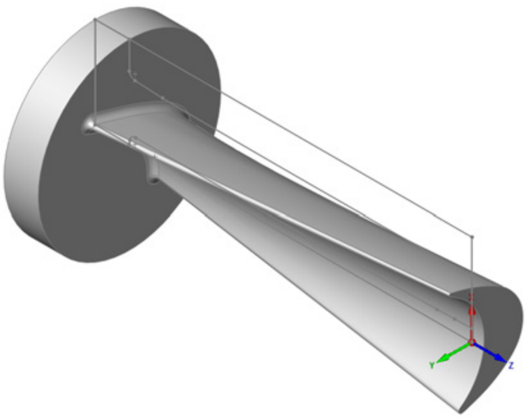
Set the **Facet tolerance** value to **0.01**. Select the solid body for the Target model definition.

During the Target model definition, SolidCAM creates the Envelope sketch in the **CAM** component of the CAM-Part assembly.

6. Save the CAM-Part

Confirm the **Mill-Turn Part Data** dialog box to save the CAM-Part.

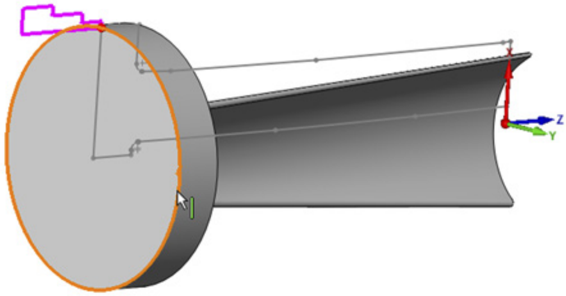
Now that the CAM-Part is defined, you have to define the machine setup for machining visualization during the machine simulation. After that you can set properties in a number of operations for machining of the part using the Turning and Simultaneous 5-Axis Milling capabilities.



7. Define the clamping fixture

If the automatic definition of a fixture is enabled in the Automatic CAM-Part definition section of SolidCAM settings, edit the clamping fixture parameters according to the following values.

If the automatic definition is disabled, define the clamping fixture as follows.



Use the **Chuck (Standard)** option.

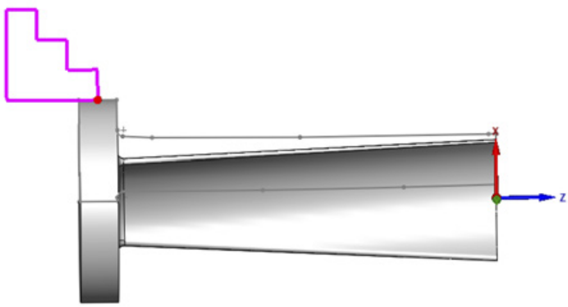
To specify the **Clamping diameter**, pick the edge of the back face of the model as shown. The value of **80** is displayed in the **Clamping diameter** edit box.

To define the **Axial position**, pick the same model edge. The value of **-165** is displayed in the **Axial position** edit box. Change this value to **-157.5** to make the fixture hold the part's end by 7.5 mm.

Define the dimensions of the chuck as follows:

- Set the **Jaw width (JW)** and the **Jaw height (JH)** values to **36**.
- Set the **Step width (SW)** and the **Step height (SH)** values to **12**.

The clamping fixture is defined.



8. Define the Machine Setup

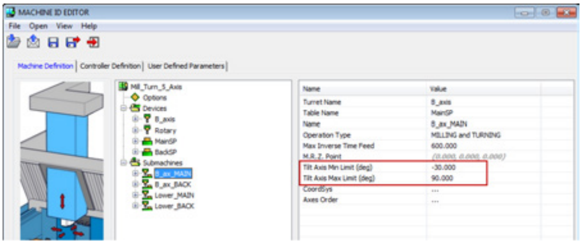
Define the Machine setup position using the clamping fixture defined in the previous step.

Set the model shift along the Z-axis. For **B_ax_MAIN** submachine, choose the **clamping fixture** as a fixture and set the **Z** shift to **220**.

9. Review the Machine ID

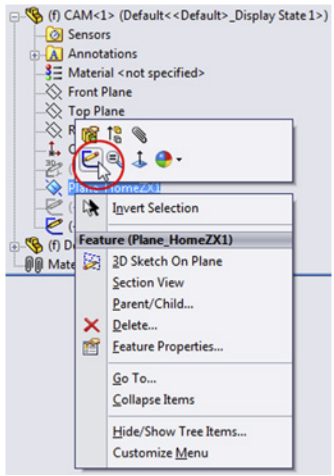
Double-click the **Machine (Mill_Turn_5_Axis)** header in the **SolidCAM Manager** tree. The **Machine ID Editor** dialog box is displayed. This dialog box enables you to check the CNC-machine parameters.

In the **Machine Definition** page, click **B_ax_MAIN** under **Submachines**.



Take a note that the tilt axis limits are set in the range between -30° and 90°. These values are used later in the **Motion Control** section of 5-Axis operations for defining the proper angle pair.

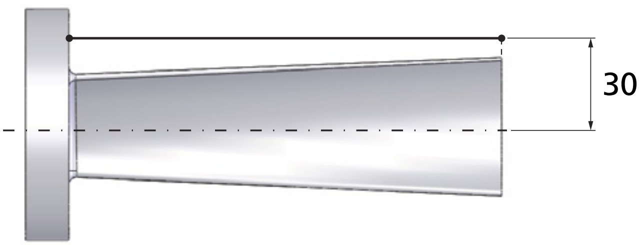
10. Perform external turning



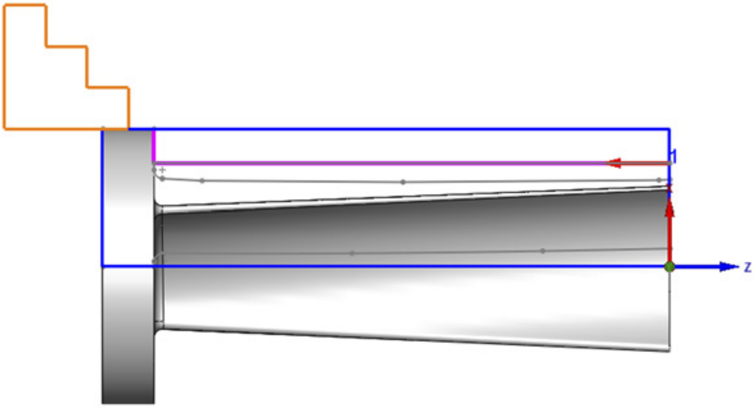
At this stage, you need to perform external turning of the part.

To define the geometry for this operation, first prepare a sketch on the **ZX-**plane of the Coordinate System. In the **FeatureManager** Design Tree, expand the **CAM** component. Right-click the **Plane_HomeZX1** item and select the **Sketch** icon in the menu.

Draw a horizontal line at the distance of **30** mm from the part centerline as shown.

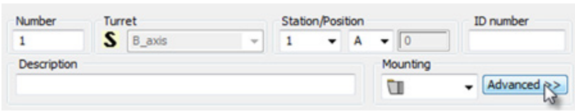


Add a Turning operation. Using the sketched line and the vertical fragment of the Envelope sketch, define the geometry for the operation. Set the **Start** and the **End extension** values to **2**.

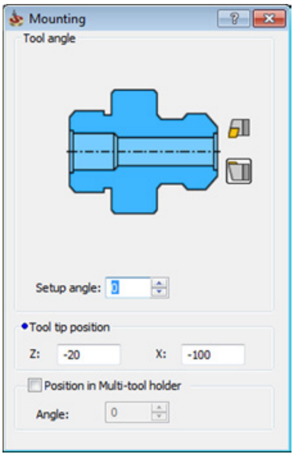


Add an **External Rough** tool and set the lengths of the cutting edges **D1** and **D2** to **5**.

Define the tool mounting. Click the **Advanced** button in the **Part Tool Table** dialog box.



The **Mounting** dialog box is displayed.



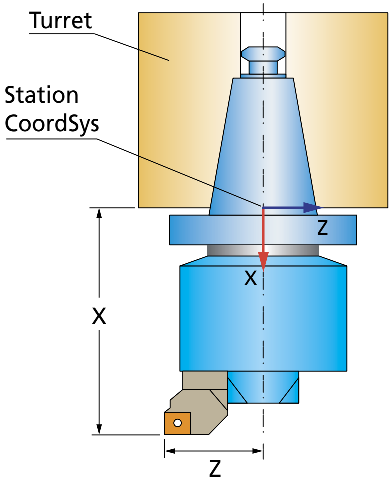
Mounting of turning tools on turrets of the **Spindle** type is defined by two major parameters: the orientation of the tool's cutting insert face (front  or back ) and the default tool direction vector defined in the Machine ID file.

Use the default tool orientation defined in the **Mounting** dialog box.

For correct mounting of the tool, you need to specify the **Tool tip position** parameters.

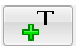
Tool tip position

This option enables you to define the tool tip position relative to the station Coordinate System for Machine Simulation.

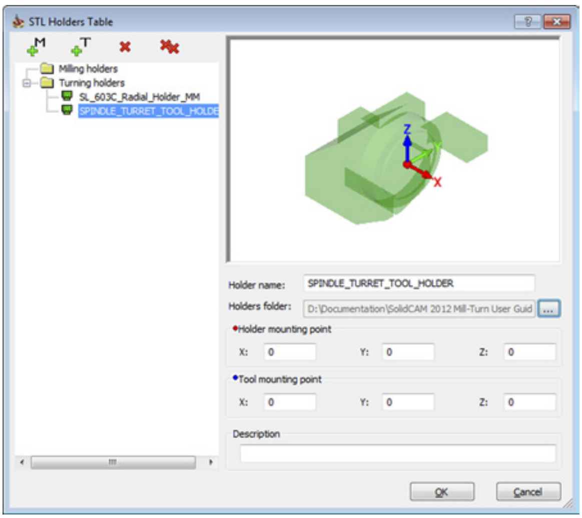


Set the **Z** value to **-90** and the **X** value to **10**.

Save and exit the operation to import an STL holder that can be used in the operation.

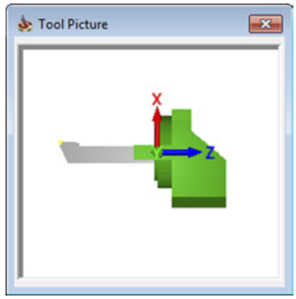
In the **SolidCAM** main menu, click **Tool Library > Tool STL Holders**. The **STL Holders Table** dialog box is displayed. Click  to add a new holder.

Using the **Browse** dialog box, locate the **SPINDLE_TURRET_TOOL_HOLDER.stl** file in the **Exercises** folder and double-click it.



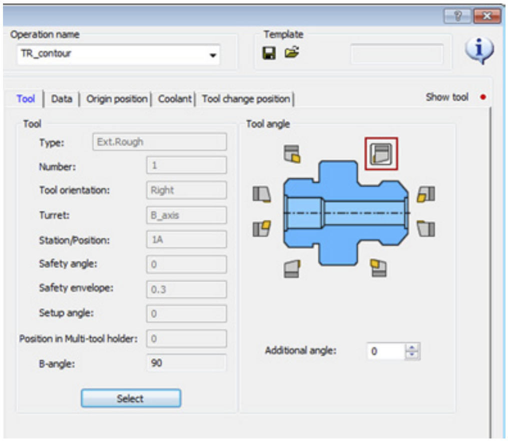
Click **OK** to close the **STL Holders Table**.

Open again the **TR_contour** operation. In the **Tool** page, click **Select** to continue the tool definition. Under Holder, choose the **SPINDLE_TURRET_TOOL_HOLDER** from the **Global** list. Using the **Flip** buttons, set the holder orientation as shown.



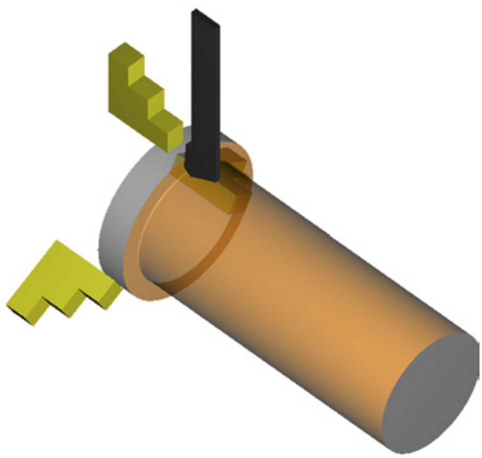
Click the **Select** button to confirm the tool definition.

In the **Tool** page, set the **Tool angle** as shown.



On the **Technology** page of the **Turning Operation** dialog box, make sure that **Mode** is set to **Long external** () to perform external longitudinal turning.

Save and calculate the operation. Simulate it in the **SolidVerify** mode.

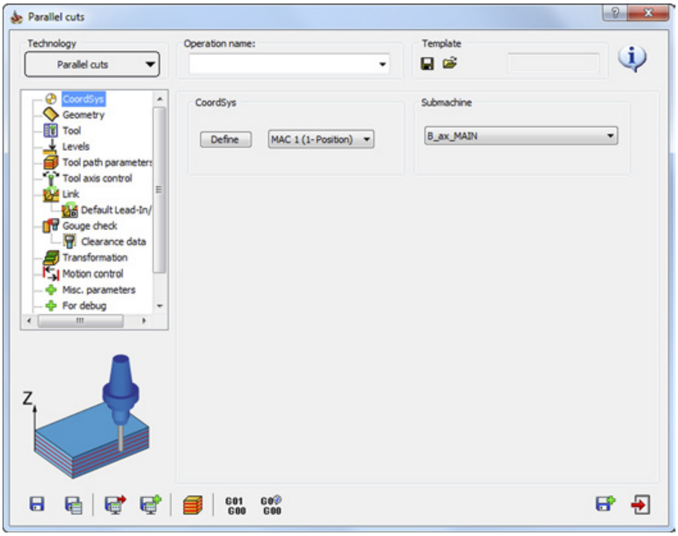


11. Define a 5-Axis operation

At this stage, you need to define a Sim. 5-Axis operation to perform rough machining of the turbine blade.

Right-click the Turning operation entry in the **SolidCAM Manager** tree and choose the **Sim. 5-Axis Milling** item from the **Add Milling Operation** submenu.
The **Parallel cuts** dialog box is displayed. This dialog box enables you to define the parameters of various strategies in the simultaneous 5-axis machining.

Use the default **Parallel cuts** strategy to perform a 5-axis operation.

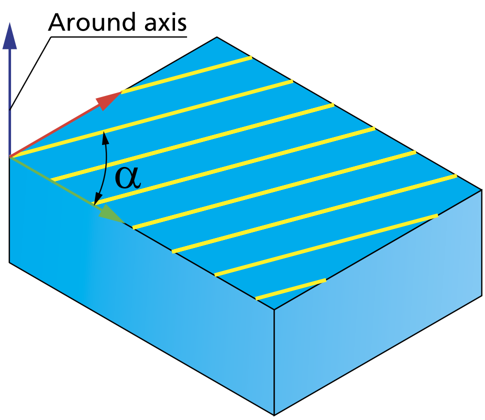


Parallel cuts

This strategy creates tool path cuts parallel to each other. It offers you two ways the tool path cuts can be performed:

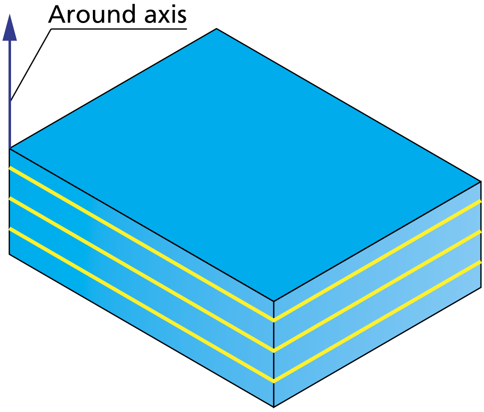
- **Linear**

  When this option is chosen, the tool path cuts are generated in the **Linear** manner, and the axis chosen from the **Around axis** list is normal to the plane of machining.



- **Constant**

  When this option is chosen, the tool path cuts are generated in the **Constant Z** manner around the axis chosen from the **Around axis** list.



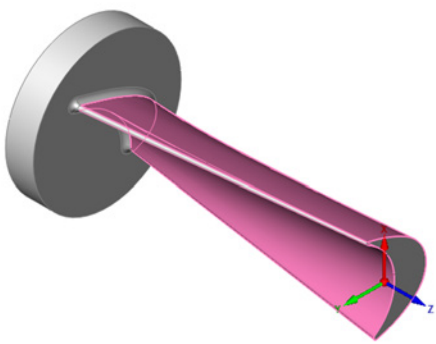
12. Define the geometry

Switch to the **Geometry** page to define the geometry for machining.

Under **Work type**, select the **Constant** option and choose **Z-Axis** from the **Around axis** list.

Now you have to define the geometry by selecting the **Drive surface** – the model faces that will be machined in this operation.

Under **Geometry**, click  in the **Drive surface** section. The **Select Faces** dialog box is displayed. This dialog box enables you to define the geometry by selecting the model faces.

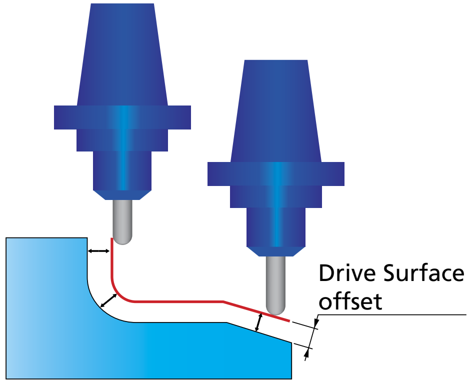


Select the model faces as shown.

Define the **Drive surface offset**.

The **Drive surface offset** parameter enables you to define a machining allowance for the drive surface. The machining is performed at the specified distance from the drive surface.

The offset is three-dimensional and expands the faces in every direction.



Set the **Drive surface offset** value to **0.2**.

In the **Area** section, make sure that the **Full, avoid cuts at exact edges** option is chosen. This option enables you to generate the tool path on the whole drive surface avoiding the drive surface edges. With this option, the minimal distance between the edge and the tool path is equal to half of the **Max. Step over** value.

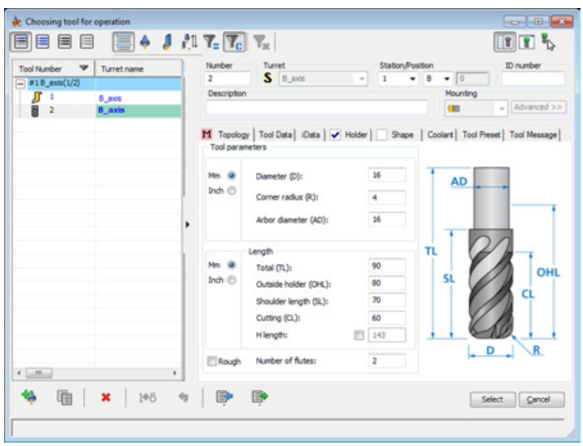
The geometry parameters are defined.

13. Define the tool

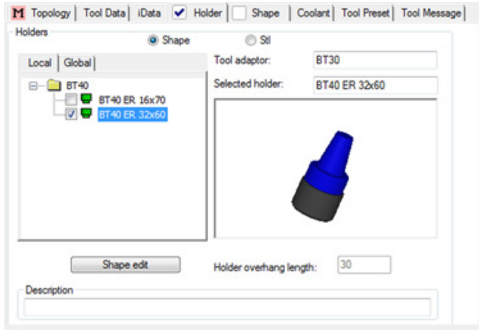
Switch to the **Tool** page to define the tool for the operation.

Add a **Bull Nose Mill** tool. Edit the tool parameters as follows:

- Set the **Diameter** to **16**
- Set the **Corner radius** to **4**
- Set the **Total** length to **90**
- Set the **Outside holder** length to **80**
- Set the **Shoulder** length to **70**
- Set the **Cutting** length to **60**



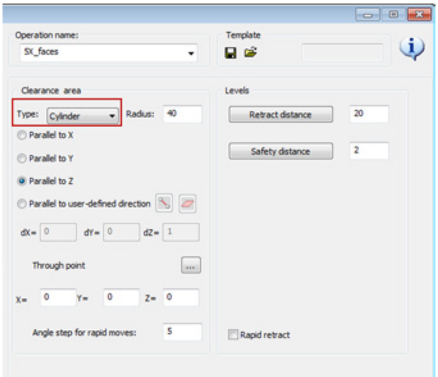
Use the **BT40 ER32x60** tool holder.



14. Define the machining levels

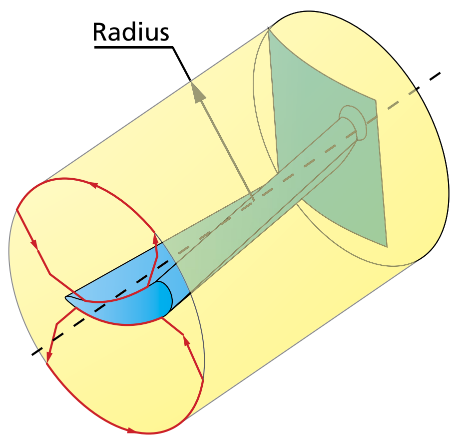
Switch to the **Levels** page to define the machining levels for the operation.

Under **Clearance area**, choose the **Cylinder** option from the **Type** list.



The **Clearance area** is the area where the tool movements can be performed with the rapid feed safely without contacting the material.

The **Cylinder** option enables you to define the Clearance area as a cylindrical surface enclosing the Drive surface. The tool performs a retract movement to the Clearance cylinder, and then a rapid movement along the cylinder surface.



The **Radius** parameter enables you to specify the cylinder radius.

The **Parallel to X**, **Parallel to Y** and **Parallel to Z** options enable you to define the cylinder axes only parallel to one of the Coordinate System axes (X, Y or Z).

Set the **Radius** value to **40** and make sure that the **Parallel to Z** option is chosen.

15. Define the tool path parameters

Switch to the **Tool path parameters** page.

In the **Surface quality** tab, under **Surface edge merge distance**, choose **As value** and set the value to **0.5**.

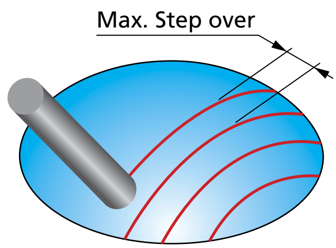
Surface edge merge distance

SolidCAM generates first tool paths for individual surfaces. Then they are merged together to form the complete tool path. The decision about merging is based on the **Surface edge merge distance** parameter. If all surface paths on a tool path slice are merged, SolidCAM checks if a closed surface path can be built by connecting the start to the end.

The **Surface edge merge distance** parameter can be defined either as a numeric value (the **As value** option) or as a percentage of the tool diameter (the **% of tool diameter** option). In both cases, this limit value must be greater than or equal to the **Cut tolerance** value.

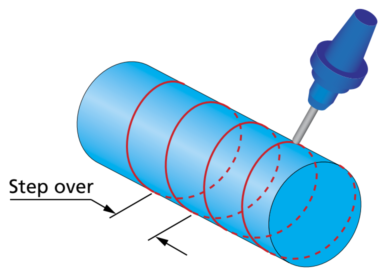
Under **Surface quality**, set the **Maximum step over** value to **5**. This parameter defines the maximum distance between two consecutive cuts.

Switch to the **Sorting** tab. Choose **Spiral** for **Cutting method**.



When the **Spiral** option is chosen, SolidCAM generates a spiral tool path around the drive surface according to the chosen pattern.

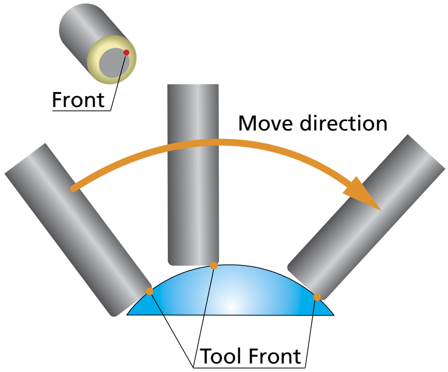
The spiral pitch is defined by the **Max. Step over** parameter.



Under **Tool contact point**, choose **At front**.

The **Tool contact point** section enables you to define the point on the tool surface that contacts with the drive surfaces during the machining.

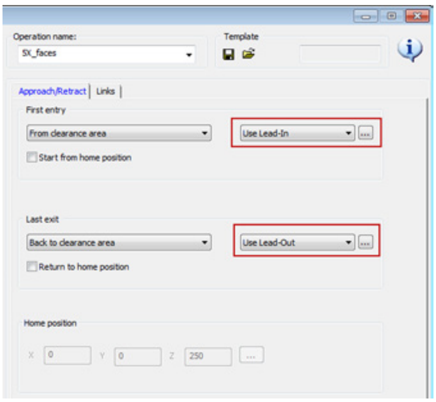
When the **At front** option is chosen, the tool contact point is located in the beginning of the tool corner radius (for bull nosed tools) in the direction of the tool movement. All changes to tool orientation are performed around the contact point and may cause gouges. It is recommended to use the **Gouge checking** options to avoid possible gouges.



16. Define the tool path linking

Switch to the **Link** page to define the linking of the tool path.

In the **Approach/Retract** tab, choose the **Use Lead In** option from the list in the **First entry** section and the **Use Lead-Out** option from the list in the **Last exit** section.



The **First entry** section enables you to define the first approach of the tool to the cutting area. The **Last exit** section enables you to define the last retreat of the tool from the cutting area after the machining. SolidCAM enables you to use pre-defined **Lead In/Lead Out** strategies to perform the movements between cutting levels.

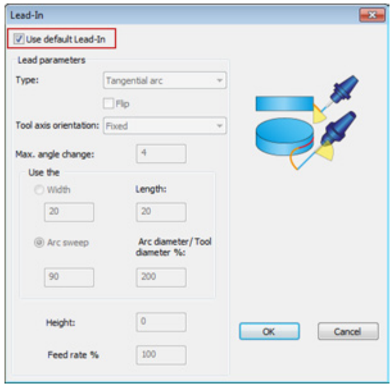
- **Use Lead-In**

  The approach movement is performed to the cutting level using the specified **Lead-In** options.

- **Use Lead-Out**

  The approach movement is performed from the cutting level using the specified **Lead-Out** options.

Define the **Lead-in** and **Lead-out** strategies. Click ... in the **First entry** section.



The **Lead-In** dialog box is displayed. This dialog box enables you to define the strategy of approach movement.

Make sure that the **Use default Lead-In** check box is selected. The default Lead-in strategy will be used for approach movement. You can change this strategy on the **Default Lead In/Out** page of the **5-Axis operation** dialog box.

Confirm the dialog box by clicking **OK**.

Click ... in the **Last exit** section.

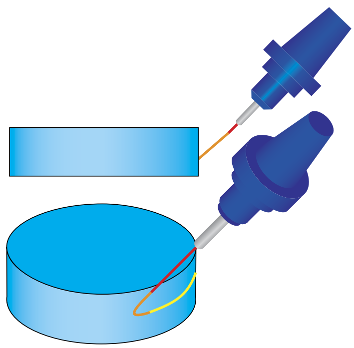
The **Lead-Out** dialog box is displayed. This dialog box enables you to define the strategy of retreat movement.

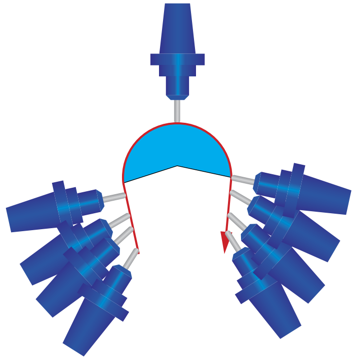
Clear the **Use default Lead-Out** check box. The Lead-out parameters can be edited.

For **Type**, choose **Vertical tangential arc**.

Vertical tangential arc

SolidCAM performs the approach/retreat movement with an arc tangential to the drive surface located in the plane of the tool axis. Note that when the side tilting options are used, the plane of vertical tangential arc is not changed.





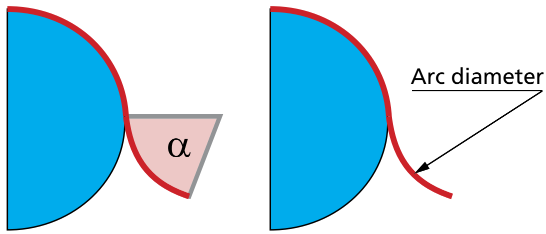
For **Tool axis orientation**, choose **Variable**. When this option is chosen, the tool axis orientation continuously changes during the approach/retreat movement. This option enables you to avoid marks on the part surface caused by the tool rotation at the start/end point of the cutting pass.

In the **Use the** section, make sure that **Arc sweep/Arc diameter** option is chosen. Set the **Arc diameter/Tool diameter** value to **300**.

Arc sweep/Arc diameter

SolidCAM enables you to specify the diameter of the approach/retreat arc using the percentage of the **Arc diameter** to the **Tool diameter**. For example, when a tool with the diameter of 10 mm is used and the percentage parameter is set to 200%, the resulting approach/retreat arc diameter is 20 mm.

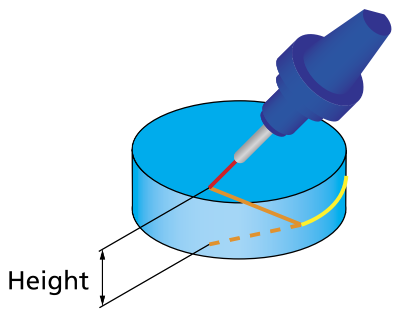
The **Arc sweep** parameter enables you to define the angle of the approach/retreat arc segment.



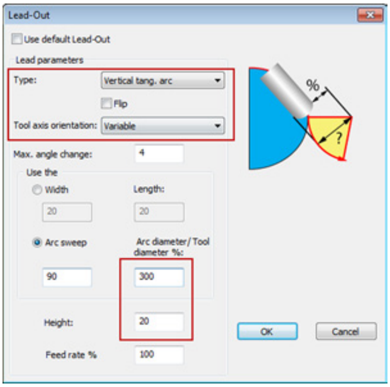
Set the **Height** value to **20**.

Height

This parameter enables you to define the incremental height of the first point of the approach/retreat movement. When the approach movement is performed with a line, the **Height** parameter enables you to perform inclined ramp approach. When the approach movement is performed with an arc, the **Height** parameter enables you to perform a helical approach.



Confirm the dialog box by clicking **OK**.



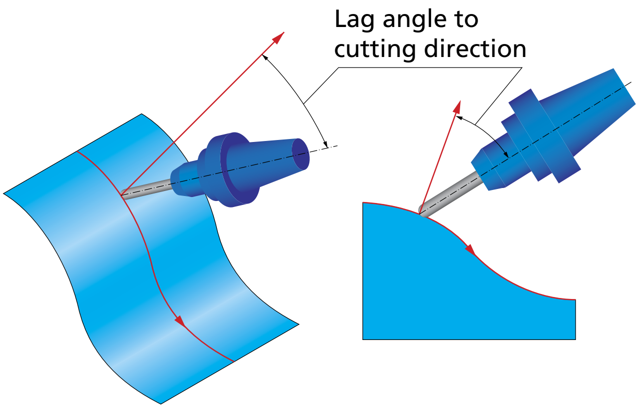
17. Define the tool tilting parameters

Switch to the **Tool axis control** page to define the parameters related to the tilting of the tool during the machining.

In the **Tool axis direction** section, make sure that the **Tilted relative to cutting direction** option is chosen. Under **Angles**, set the **Lag angle to cutting direction** value to **20** and the **Tilt angle at side of cutting direction** value to **5**.

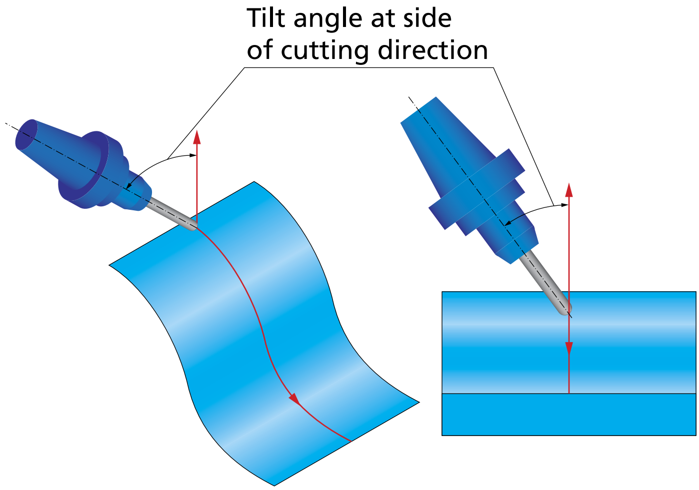
Tilted relative to cutting direction

This option enables you to define the tool tilting relative to the cutting direction. The **Lag angle to cutting direction** parameter enables you to define the tool tilting in the direction of the cutting pass. This parameter is

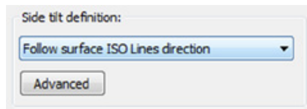


measured relative to surface normal.

The **Tilt angle at side of cutting direction** parameter enables you to define the tool inclination in the direction determined by **Side tilting** options. The **Tilt angle at side of cutting direction** parameter is measured relative to surface normal.

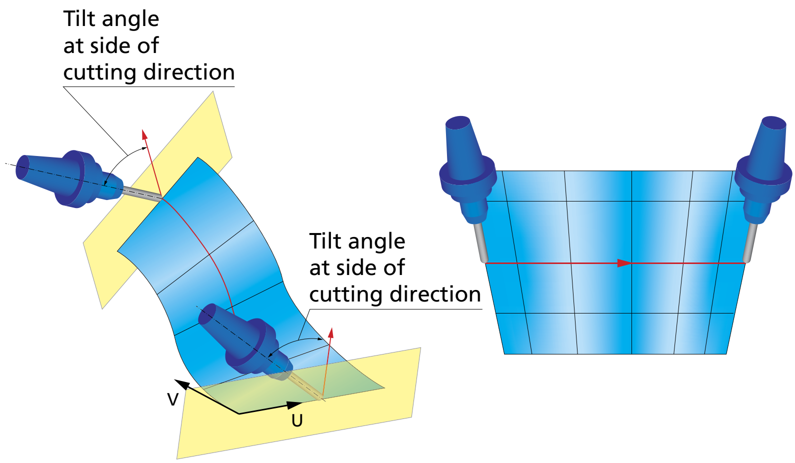


Under **Side tilt definition**, choose the **Follow surface ISO lines direction** option.



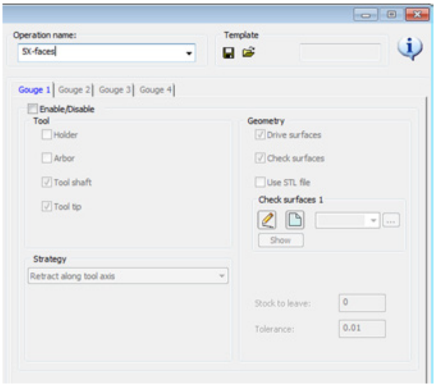
Follow surface ISO lines direction

The direction of the side tilting is chosen according to the direction of the U- and V-vectors of the drive surface.



18. Define the gouge checking parameters

Switch to the **Gouge check** page. It contains four tabs in which you can define four different sets of gouge checking parameters. In each set you have to choose components of the tool holding system and model faces to check for possible collisions between them. You also have to define the strategy to avoid the possible collisions.

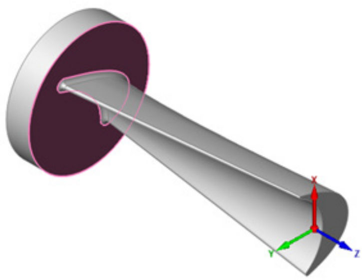


In the **Gouge 1** tab, select the **Enable** check box. In the **Tool** section, select all of the check boxes corresponding to the components of the tool.

In the **Geometry** section, clear the **Check surfaces** check box. In this set, the collisions between the drive surfaces and the tool will be checked.

Switch to the **Gouge 2** tab and select the **Enable** check box. In the **Tool** section, select all of the check boxes corresponding to the components of the tool.

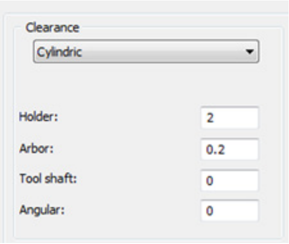
In the **Geometry** section, clear the **Drive surfaces** check box. In this set, the collisions between the check surfaces and the tool will be checked. To define the check surfaces, click and select the model face as shown.



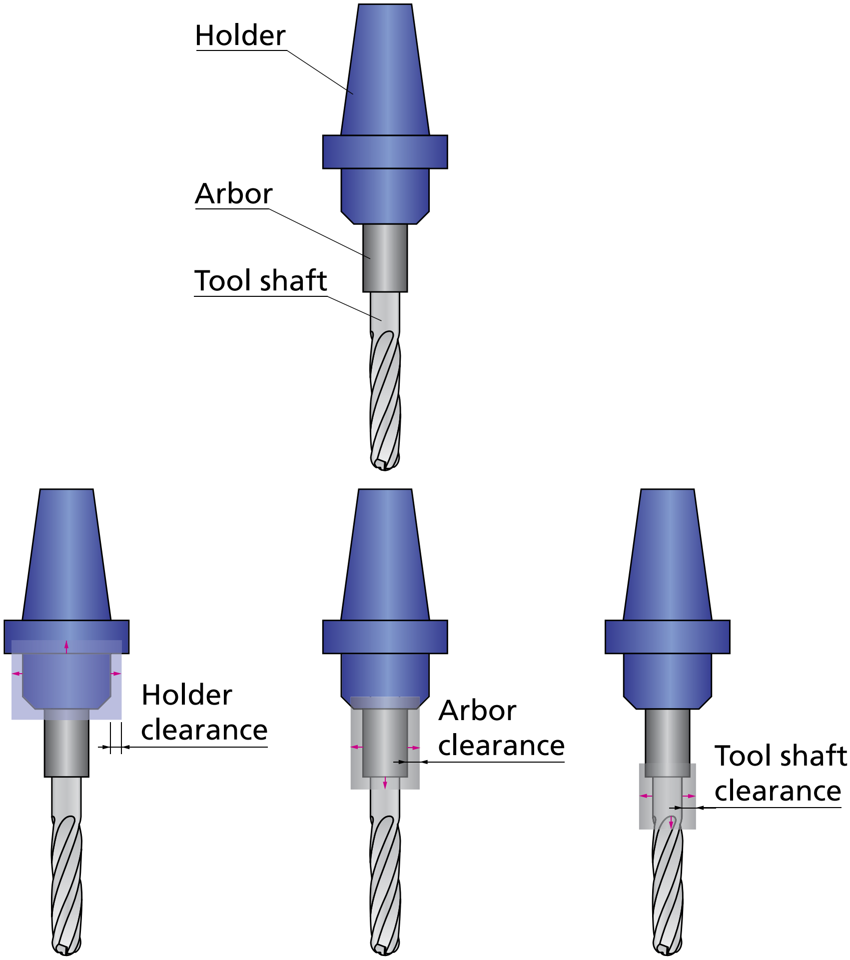
Set the **Stock to leave** value to **0.2**. This parameter enables you to define the allowance for the check surfaces, i.e. the distance the tool will keep from the check surface.

Under **Strategy**, choose **Leaving out gouging points**. When this strategy is used, SolidCAM trims the segments of the tool path where the collisions are detected. The tool path updated by trimming will not contain gouges. Use the default **Don't trim Tool path** option. With this option, only the colliding segments of the tool path are trimmed out.

Switch to the **Clearance data** page. In the **Clearance** section, set the **Arbor** value to **0.2** and the **Tool shaft** value to **0**.



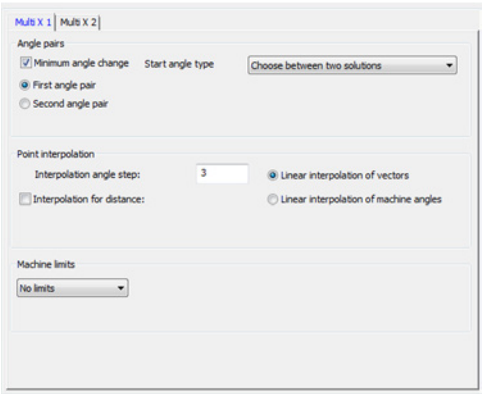
SolidCAM considers the tool holder, the arbor and the tool shaft to have a cylindrical shape. The **Arbor** clearance parameter defines the offset applied to the arbor cylinder from all sides. The **Holder** clearance parameter defines the offset applied to the tool holder cylinder from all sides. The **Tool Shaft** clearance parameter defines the offset applied to the tool shaft cylinder from all sides.



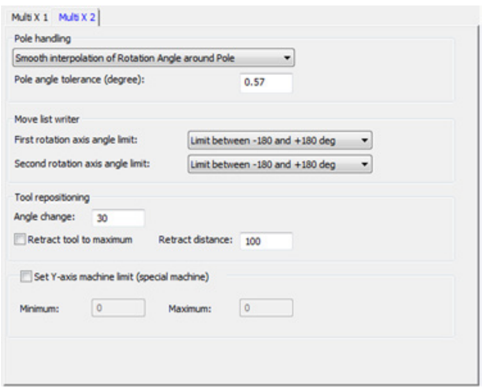
19. Define the motion control parameters

Switch to the **Motion control** page.

In the **Multi X 1** tab, use default settings for **Angles pairs**. The **Rotation angle value** is set to **30** to be within the range of minimal and maximal tilt axis. Set the **Interpolation angle step** value to **3**. Using this parameter, SolidCAM enables you to interpolate the angular movements. A new interpolated tool axis position is defined at each angle defined by the **Interpolation angle step** parameter.



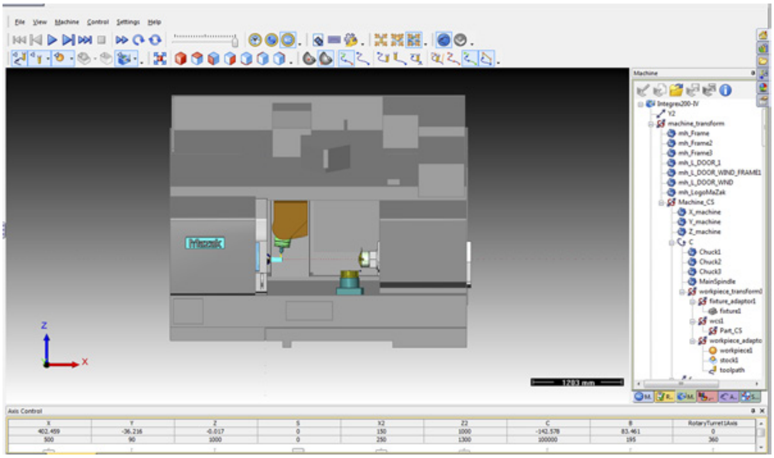
Switch to the **Multi X 2** tab. In the **Move list writer** section, choose the **Limit between -180 and +180 deg** option for both **First** and **Second rotation axis angle limit** parameters. With this option, the angle coordinates in the output tool path are limited by the range from -180° to 180°.



20. Save, calculate and simulate the operation

Save and calculate the operation.

Perform the simulation in the **Machine simulation** mode. This mode enables you to perform the machining simulation and tool path verification using the kinematics of the CNC-machine.



Play the simulation by clicking on the **Play** button.

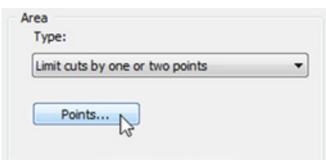
21. Define a 5-Axis operation

Now you need to define an additional Sim. 5-Axis operation to perform machining of the area close to the blade base.

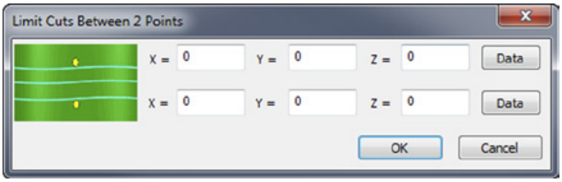
Right-click the last Sim. 5-Axis operation in the **SolidCAM Manager** tree and choose **Copy** from the menu. Right-click this operation again and choose **Paste**. The copied operation is pasted after the previous one. You need to edit several parameters of this operation.

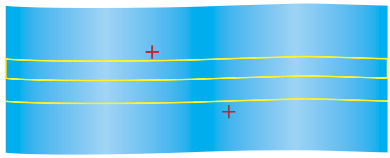
22. Define the geometry

Switch to the **Geometry** page to define the geometry for machining. In the **Area** section, choose the **Limit cuts by one or two points** option from the **Type** list. This option enables you to limit the tool path by one or two points.



Click the **Points** button to define the limiting points using the **Limit Cuts Between 2 Points** dialog box. This dialog box enables you to enter the coordinates of the limiting points or define them directly by clicking on the solid model.

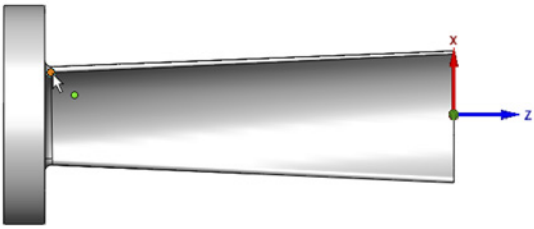




To define the first point, click the **Data** button in the **Limit Cuts Between 2 Points** dialog box. The **Select First Limit Point** dialog box is displayed.

Select the first point on the model as shown. The coordinates of the selected point are displayed in the **Select First Limit Point** dialog box.

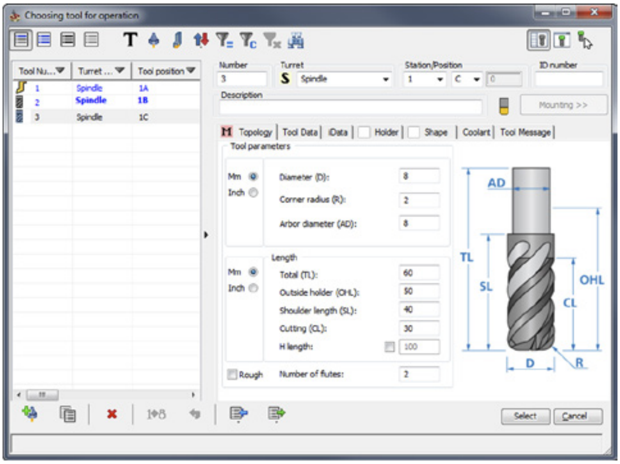
Define the second point as shown.



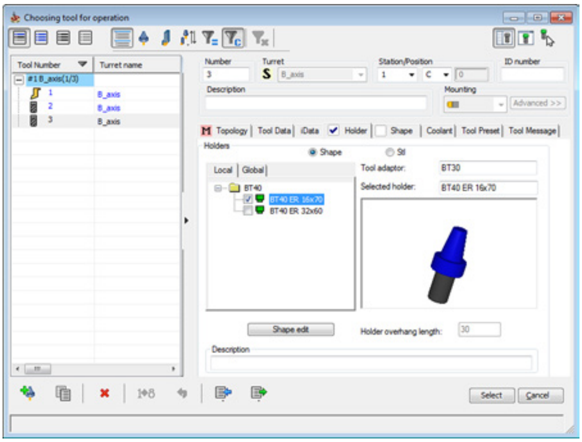
23. Define the tool

Add a **Bull Nose Mill** tool. Edit the tool parameters as follows:

- Set the **Diameter** to **8**
- Set the **Corner Radius** to **2**
- Set the **Total** length to **60**
- Set the **Outside holder** length to **50**
- Set the **Shoulder** length to **40**
- Set the **Cutting** length to **30**



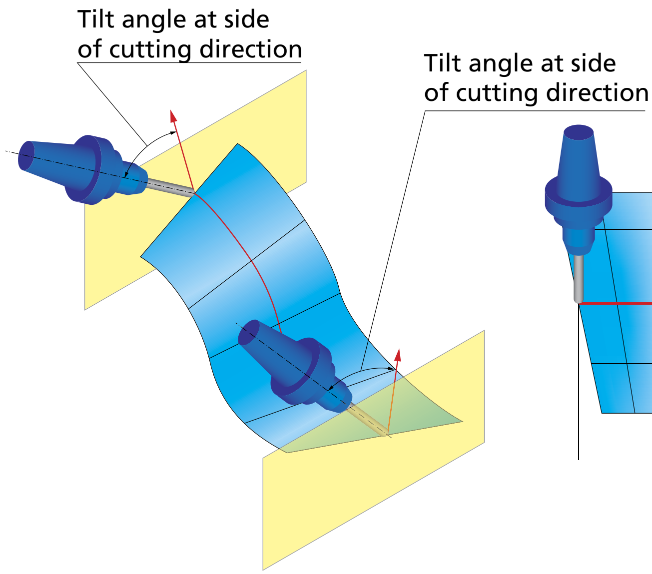
Use the **BT40 ER16x70** tool holder.



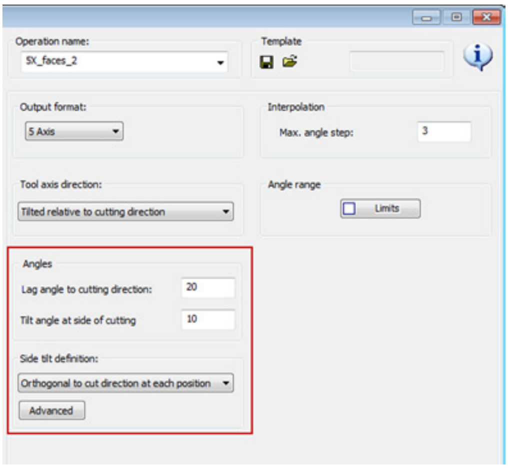
24. Define the tool tilting parameters

Switch to the **Tool axis control** page to define the parameters related to the tilting of the tool during the machining.

In the **Angles** section, set the **Tilt angle at side of cutting direction** value to **10**.

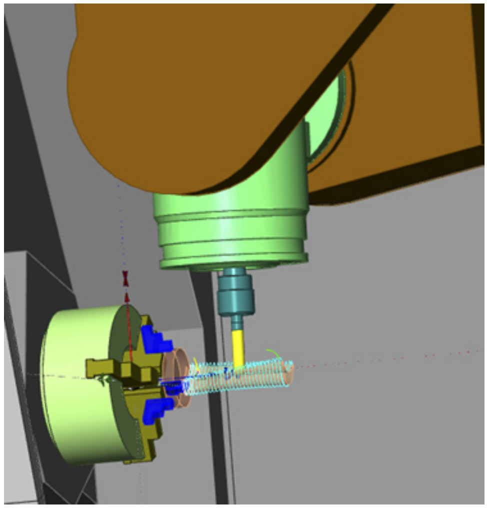


Under **Side tilt definition**, choose the **Orthogonal to cut direction at each position** option. When this option is chosen, the plane of the side tilting is orthogonal to the tool path direction for each cutting position.



25. Save, calculate and simulate the operation

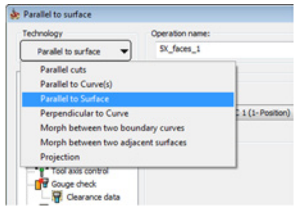
Save and calculate the operation. Perform the simulation in the **Machine simulation** mode. Play the simulation by clicking the **Play** button.



26. Define a Sim. 5-Axis operation

Define an additional Sim. 5-Axis operation to perform finishing of the blade.

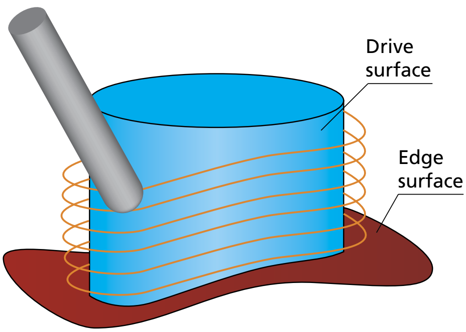
In the **Technology** section, choose the **Parallel to surface** option from the list.



Parallel to surface

This option enables you to generate the tool path on the drive surface parallel to the specified check surface.

The **Edge surface** section enables you to define the check surface geometry for the tool path generation.



The drive and check surfaces have to be adjacent, i.e. they must have a common edge.

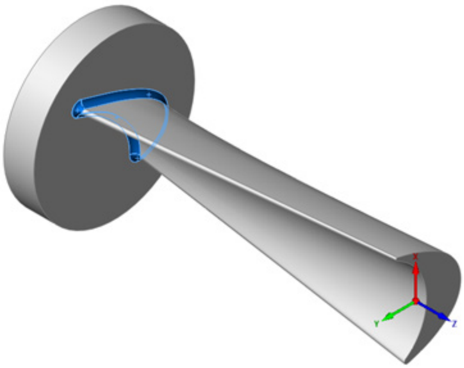
Depending on the defined **Tool tilting**, it is recommended to activate the gouge checking to make sure that the check surface will not be gouged.

27. Define the geometry

Switch to the **Geometry** page to define the geometry for machining.

In the **Drive surface** section, choose the **Faces** geometry defined in the previous operations.

In the **Edge surface** section, click to define the check surface geometry. Select the model faces as shown.

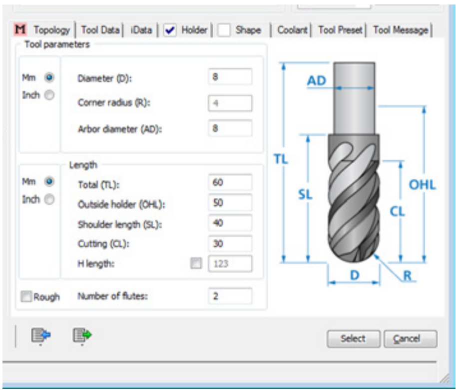


28. Define the tool

Add a **Ball Nose Mill** tool. Edit the tool parameters as follows:

- Set the **Diameter** to **8**
- Set the **Total** length to **60**
- Set the **Outside holder** length to **50**
- Set the **Shoulder** length to **40**
- Set the **Cutting** length to **30**

Use the **BT40 ER16x70** tool holder.



29. Define the machining levels

Switch to the **Levels** page and define the machining levels in the same manner as explained in **Step #14**: choose the **Cylinder** type and set the **Radius** to **40**. Make sure that the **Parallel to Z** option is chosen.

30. Define the tool path parameters

Switch to the **Tool path parameters** page.

Under **Surface edge merge distance**, choose **As value** and set the value to **0.5**.

In the **Sorting** tab, choose **Spiral** for **Cutting method**.

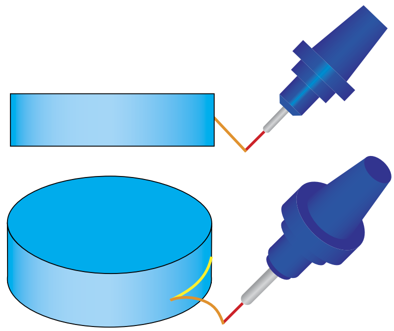
31. Define the tool path linking

Switch to the **Link** page to define the linking of the tool path.

For **First entry**, choose the **Use Lead-In** option from the list. Choose the **Reverse tangential arc** strategy for approach movement.

Reverse tangential arc

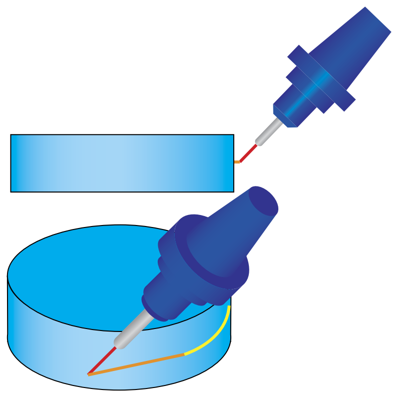
The approach/retreat movement is performed with an arc tangential to the drive surface, like the **Tangential arc** option, but with the direction of the approach/retreat arc reversed. The approach/retreat arc is generated in the plane orthogonal to the tool axis.



For **Last exit**, choose the **Use Lead-Out** option from the list. Choose the **Tangential line** strategy for retreat movement.

Tangential line

SolidCAM performs the approach/retreat movement with a line tangential to the cutting pass.



32. Define the tool tilting parameters

Switch to the **Tool axis control** page to define the parameters related to the tilting of the tool during the machining.

In the **Tool axis direction** section, make sure that the **Tilted relative to cutting direction** option is chosen. Set the **Lag angle to cutting direction** value to **20** and the **Tilt angle at side of cutting direction** value to **10**.

Under **Side tilt definition**, choose the **Follow surface ISO lines direction** option.

33. Define the gouge checking parameters

Switch to the **Gouge check** page and define the parameters in the same manner as explained in **Step #18**.

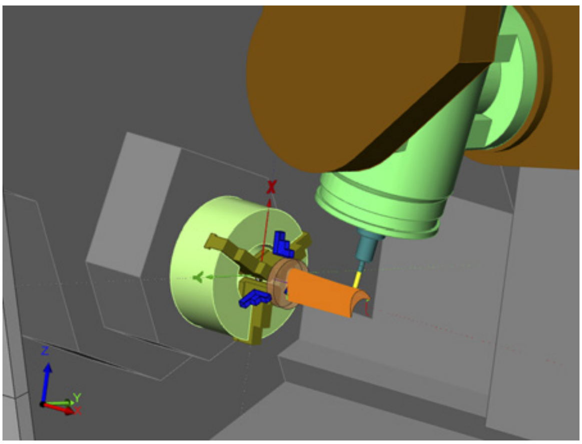
34. Define the motion control parameters

Switch to the **Motion control** page and define the parameters in the same manner as explained in **Step #19**.

35. Save, calculate and simulate the operation

Save and calculate the operation. Perform the simulation in the **Machine simulation** mode. Play the simulation by clicking the **Play** button.

At this stage, the exercise is completed.



Congratulations!

You have successfully finished the **SolidCAM Turning and Mill-Turn Training Course**.

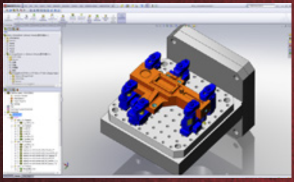
SolidCAM + SolidWorks

The complete integrated Manufacturing Solution

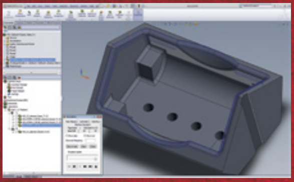
SOLIDWORKS Certified Gold Product

SolidCAM

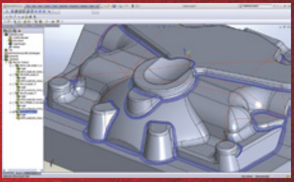
iMachining – The Revolution in CAM!



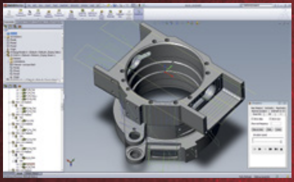
2.5D Milling



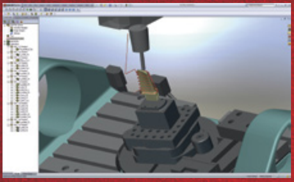
HSS (High-Speed Surface Machining)



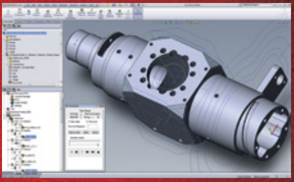
HSM (High-Speed Machining)



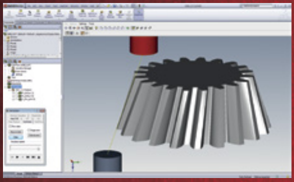
Indexed Multi-Sided Machining



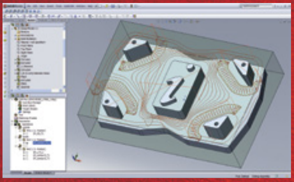
Simultaneous 5-Axis Machining



Turning and Mill-Turn up to 5-Axis



Wire EDM



iMachining

Service and Support

YouTube Broadcast Yourself™

www.youtube.com/SolidCAMProfessor

www.youtube.com/SolidCAMiMachining

www.facebook.com/SolidCAM

www.solidcam.com