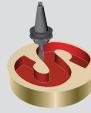




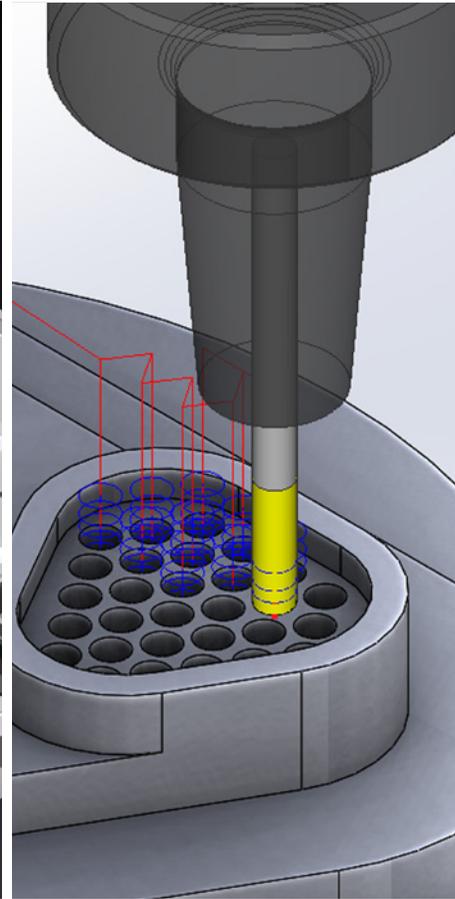
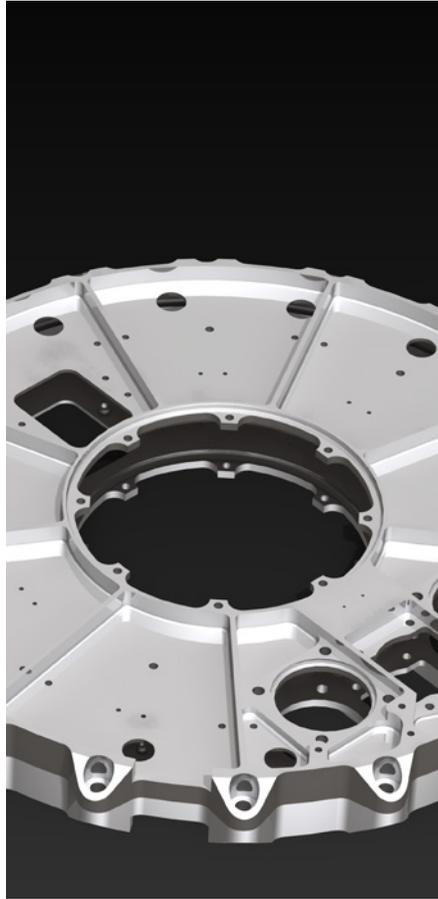
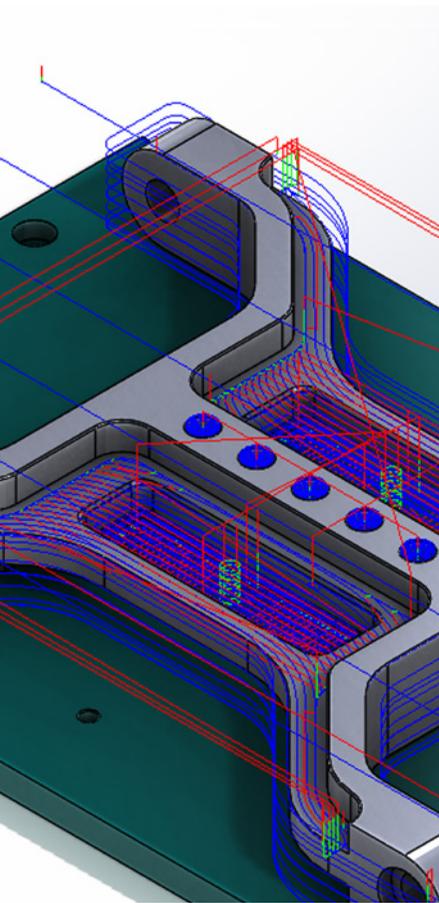
SolidCAM + SolidWorks

The complete integrated Manufacturing Solution



SolidCAM

iMachining – The Revolution in CAM!



SolidCAM Milling Training Course: **2.5D Milling**

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

www.solidcam.com



SolidCAM 2013

Milling Training Course

2.5D Milling

©1995-2013 SolidCAM

All Rights Reserved.

Contents

1. Introduction

1.1 About this course.....	3
1.2 Basic Concepts	5
1.3 Process Overview	5

2. CAM-Part Definition

Exercise #1: CAM-Part Definition	9
--	---

3. SolidCAM 2.5D Operations

Exercise #2: Cover Machining.....	30
Exercise #3: Cover Machining.....	84
Exercise #4: Bracket Machining	153
Exercise #5: Electronic Box Machining.....	155
Exercise #6: Clamp Machining.....	158
Exercise #7: Basic Part Machining.....	166
Exercise #8: Cover Machining.....	170
Exercise #9: Mounting Base Machining.....	176
Exercise #10: Support Machining.....	185

4. Indexial 4-Axis Milling

Exercise #11: Frame Machining.....	197
Exercise #12: Mounting Machining.....	216

5. Indexial 5-Axis Milling

Exercise #13: Clamp Machining.....	227
------------------------------------	-----

6. ToolBox Cycles

Exercise #14: Standard Cycles Machining.....	236
--	-----

7. Automatic Feature Recognition

Exercise #15: Pocket Recognition.....	251
Exercise #16: Mounting Box Machining.....	258
Exercise #17: Drill Recognition	260
Exercise #18: Electronic Box Machining.....	284

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 2.5D Milling technologies. This course covers the basic concepts of SolidCAM 2.5D 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.

Course design

This course is designed around a task-based approach to training. With the guided exercises you will learn the commands and options necessary to complete a machining task. The theoretical explanations are embedded into these exercises to give an overview of the SolidCAM 2.5D Milling capabilities.

Using this training book

This training book is intended to be used both in a classroom environment under the guidance of an experienced instructor and as self-study material. It contains a number of laboratory exercises to enable you to apply and practice the material covered by the guided exercises. The laboratory exercises do not contain step-by-step instructions.

About the CD

The CD supplied together with this book contains copies of various files that are used throughout this course. The **Exercises** folder contains the files that are required for guided and laboratory exercises. The **Built Parts** folder inside the **Exercises** contains the final manufacturing projects for each exercise. Copy the complete **Exercises** folder on your computer. The SolidWorks files used for the exercises were prepared with SolidWorks 2013.

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 indicate the beginning of the exercise action.
 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 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.

1.3 Process Overview

The major stages of the SolidCAM manufacturing project creation process are the following:

CAM-Part definition

This stage includes the definition of the global parameters of the Manufacturing Project (CAM-Part). You have to define a number of Coordinate Systems that describe the positioning of the part on the CNC-machine.

Optionally, you can define the Stock model and Target model to be used for the rest material calculation. The Stock model describes the initial state of the workpiece that has to be machined. The Target model describes the one that has to be reached after the machining. After every operation, SolidCAM calculates how much material was actually removed from the CAM-Part and how much material remains unmachined (rest material). The rest material information enables SolidCAM to automatically optimize the tool path and avoid the air cutting.

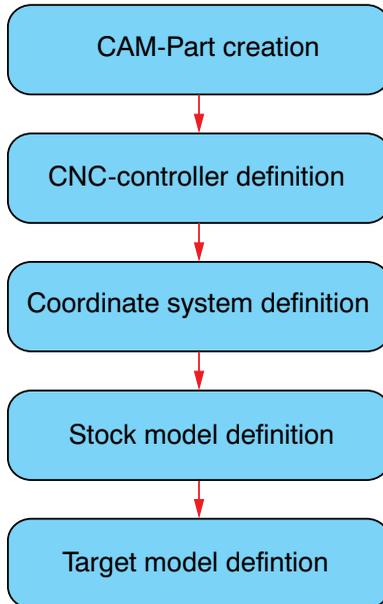
Operations definition

SolidCAM enables you to define a number of milling operations. During an 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 includes the following stages:



- **CAM-Part creation.** At this stage, you have to define the CAM-Part name and location. SolidCAM defines the necessary system files and a folder to allocate the place to store SolidCAM data.
- **CNC-controller definition.** Choosing a CNC-controller is a necessary step. The controller type influences the Coordinate System definition and the Geometry definition.
- **Coordinate system definition.** You have to define the Coordinate System – the origin for all machining operations of the CAM-Part.
- **Stock model definition.** SolidCAM enables you to define the stock model that describes the initial state of the workpiece to be machined.
- **Target model definition.** SolidCAM enables you to define the model of the part in its final state after the machining.

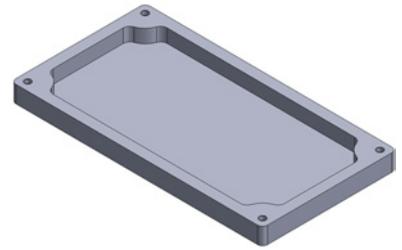
The following exercises describe the full process of the CAM-Part definition. It is recommended to go through the stages in order to understand how the CAM-Part features are built. For this purpose, you have to turn off the automatic CAM-Part definition.

Before you start, select **SolidCAM Settings** command from the **SolidCAM** menu. In the left pane, select **CAM-Part > Automatic CAM-Part definition**. In the right pane, click the **Milling** tab and clear the following check boxes: **Use default CNC-Controller**, **Don't show CoordSys manager after MAC1 Pos1**, **Create MAC1-1 position automatically**, **Definition of Stock**, and **Definition of Target**. Click **OK** to confirm your choice.

These settings can be turned back on at any time.

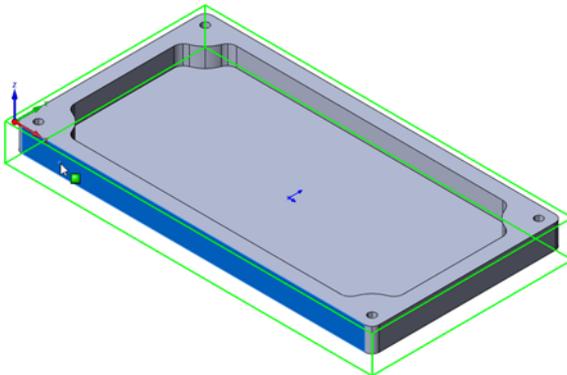
Exercise #1: CAM-Part Definition

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

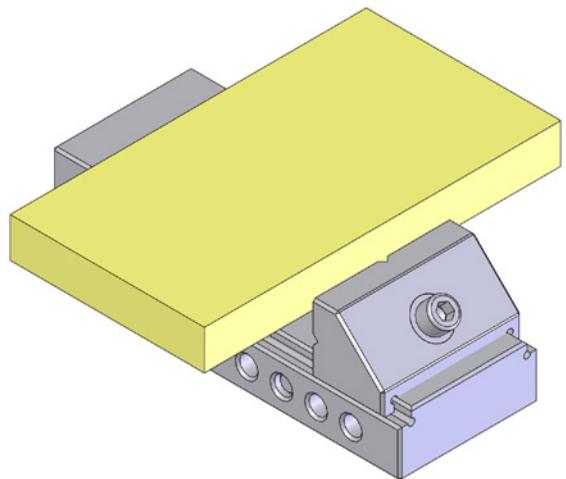


When you start to program a CAM-Part, you have to decide what workpiece you are going to use. This decision determines the number and the type of operations that are used to reach the final part shape.

In this exercise, the box stock is used. The box dimensions include offsets from the actual model.



At the next stage, you have to decide on what type of CNC-machine you are going to use (3-, 4- or 5-axis). In this exercise, a 3-axis CNC-machine is chosen for the machining. With a CNC-machine of this type, all the required faces of the cover part can be machined using a single positioning.

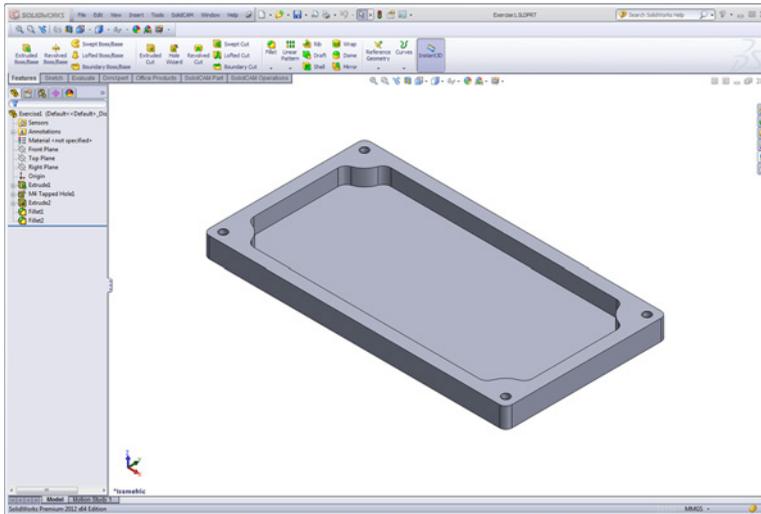




1. Load the SolidWorks model

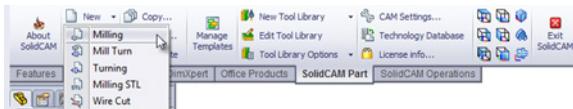
Load the **Exercise1.sldprt** model located in the **Exercises** folder.

This model contains a number of features forming the solid body of the cover.

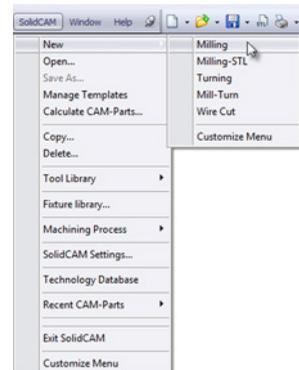


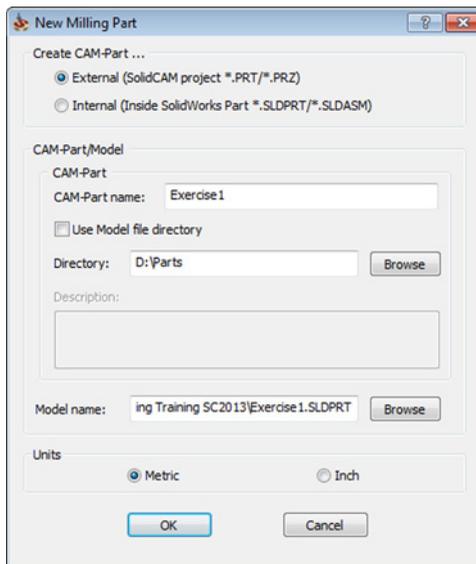
2. Start SolidCAM

To activate SolidCAM, click the **SolidCAM** field in the main menu of SolidWorks and choose **Milling** from the **New** submenu or click **New > Milling** on the **SolidCAM Part** toolbar.



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





New Milling 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, 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 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.

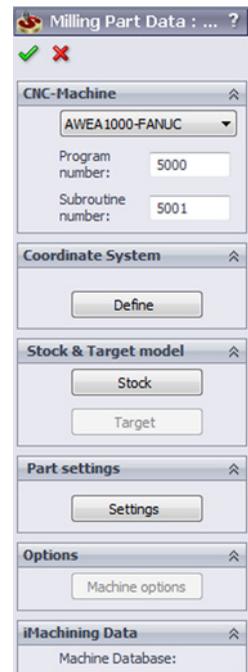


Every time the CAM-Part is opened, SolidCAM automatically checks the correspondence of the dates of the CAM-Part and the original SolidWorks design model. When the date of the original SolidWorks model is later than the date of the CAM-Part creation, this means that the SolidWorks original model has 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

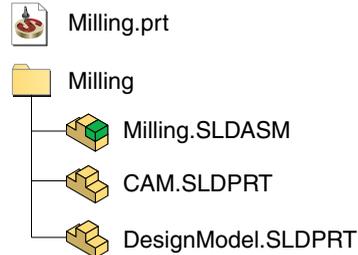
Choose the **External** mode of CAM-Part creation. When the **Directory**, **CAM-Part Name** and **Model Name** have been defined, click the **OK** button to confirm the CAM-Part creation. The CAM-Part is defined, and its structure is created. The **Milling Part Data** dialog box is displayed.





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 **Milling**.



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

SolidCAM copies the original SolidWorks model to the **Milling** subdirectory and creates a SolidWorks assembly that has the same name as the CAM-Part (**Milling.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.

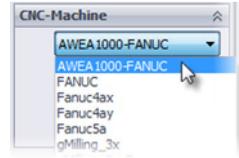
The SolidCAM CAM-Part uses the assembly environment of SolidWorks. This enables you to create auxiliary geometries (i.e. sketches) without making changes in the original design model. You can also insert some additional components into the assembly file such as stock model, CNC-machine table, clamping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



4. Choose the CNC-Machine

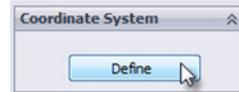
Select the CNC-machine controller. Click the arrow in the **CNC-Machine** area to display the list of post-processors installed on your system.

In this exercise, use a 3-Axis CNC-machine with the **AWEA1000-Fanuc** CNC-controller. Choose the **AWEA1000-Fanuc** CNC-controller from the list.



5. Start the Coordinate System definition

Click the **Define** button in the **Coordinate System** area to define the Machine Coordinate System.

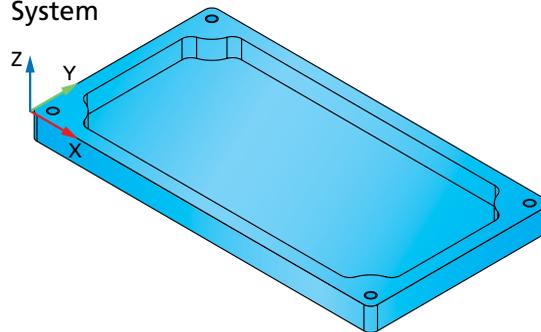


To complete the CAM-Part definition, you need to define the **Machine Coordinate System**.

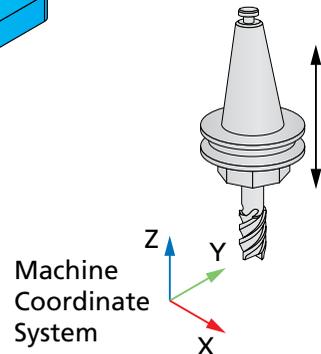
The **Machine Coordinate System** defines the origin for all machining operations of the CAM-Part. It corresponds with the built-in controller functions.

You can define the Coordinate System origin position and 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.

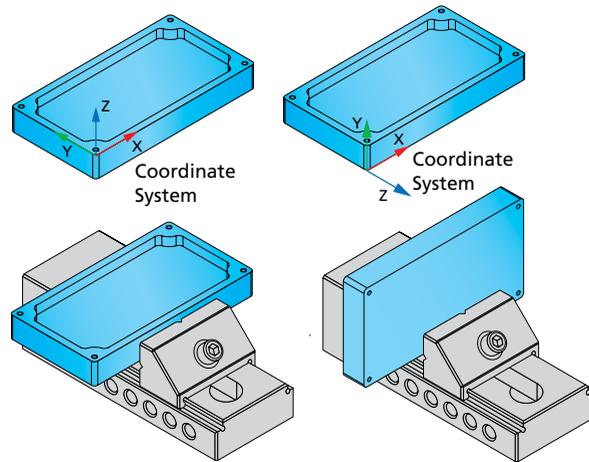
Coordinate System



The Z-direction of the Machine Coordinate System is parallel to the revolution axis of the tool.



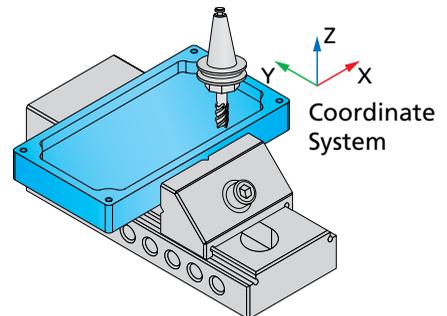
In SolidCAM, the tool approaches from the positive direction of the Z-axis (like on a vertical CNC-machine).



For 3-Axis CNC milling machines, each Machine Coordinate System means separate clamping. If you need to machine the part from different sides, use several Machine Coordinate Systems with the Z-axis oriented normally to the machined sides.

In this exercise, it is enough to define one Machine Coordinate System with the Z-axis oriented upwards.

Such coordinate system enables you to machine the part with a single clamping.



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



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.

Define

This method enables you to define the Coordinate System by selecting points. You have to define the origin and the direction of the X- and Y-axes.

Select Coordinate System

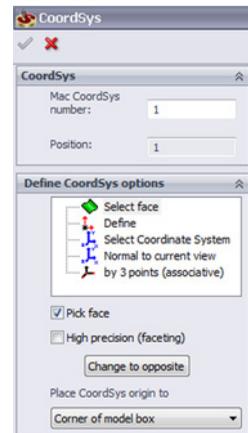
This method enables you to choose the SolidWorks Coordinate System defined in the design model file as the CoordSys. The CoordSys origin and axes orientation are the same as those of the original SolidWorks Coordinate System.

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 Coordinate System by selecting any 3 points.

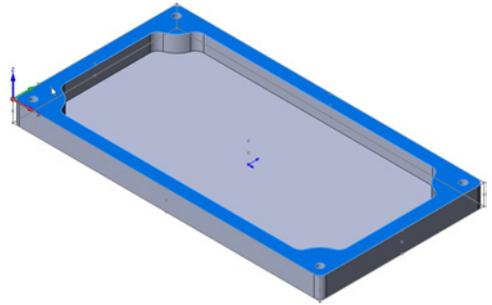




6. Select the model face

With the **Select Face** method chosen, click on the model face as shown.

The CoordSys origin is automatically defined in the corner of the model box. The Z-axis of the CoordSys is normal to the selected face.

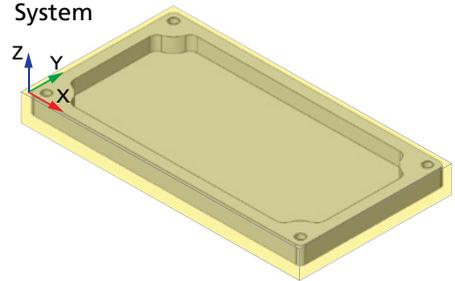


Model box

SolidCAM calculates the box surrounding the model. The upper plane of the model box is parallel to the XY-plane of the defined CoordSys.

The CoordSys is located in the corner of the model box.

Coordinate System



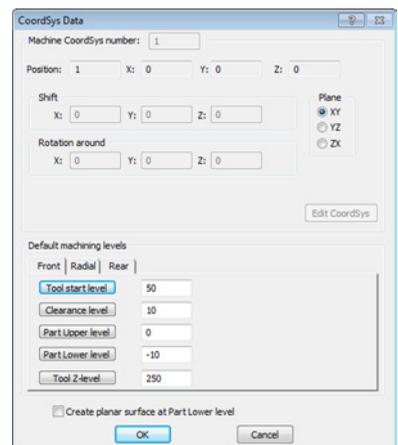
Confirm by clicking the  button. The Coordinate System is defined.

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



7. CoordSys Data

This dialog box enables you to define the 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 machine controller of your machine.

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

- **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 its output to the CCode 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 is the angle of rotation around the main axes X, Y and Z.

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.

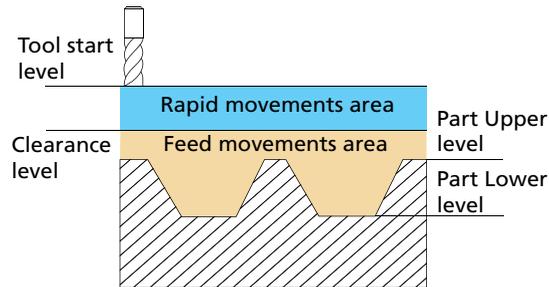
The **Rear** tab displays levels for milling from the negative Z-direction. The negative Z-direction can be used in case of milling of the part from the back side with the same Coordinate System in the main spindle or in case of using the back spindle.

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 milled.
- The **Part Lower level** defines the lower surface level of the part to be milled.



- The **Tool Z-level** parameter defines 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 the **OK** button.

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

Confirm the **CoordSys Manager** dialog box with the  button. The **Milling Part Data** dialog box is displayed again.

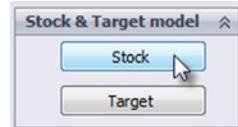




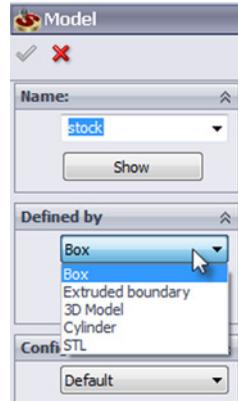
8. Define the Stock model

For each Milling project, you can define the Stock model, which is the workpiece that is placed on the machine before you start machining the CAM-Part.

Click the **Stock** button in the **Stock & Target model** section of the **Milling Part Data** dialog box.



The **Model** dialog box is displayed. This dialog box enables you to choose the mode of the Stock model definition.



Stock Model Definition Modes

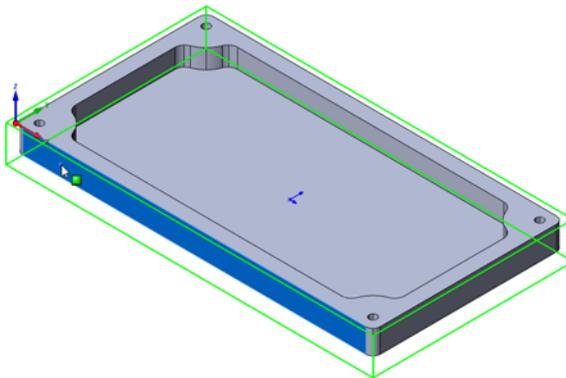
- **Box** – in this mode SolidCAM automatically determines the box surrounding the model.
- **Extruded boundary** – this mode enables you to define the 2D stock geometry by selecting a chain of geometrical elements (lines, arcs, splines, edges, etc.).
- **3D Model** – this mode enables you to define the stock model via 3D model selection.
- **Cylinder** – this mode enables you to define the stock model as a cylinder (or a tube) surrounding the selected solid model.
- **STL file** – this mode enables you to define the stock model based on a STL file that exists in your system. When you choose this mode, the **STL file** section becomes available. By clicking the **Browse** button, you can choose the STL file for the stock definition.

Choose the **Box** mode from the **Defined by** list. The appearing dialog box enables you to select a solid body for the surrounding box calculation.

Optionally, offsets from the model can be defined. In this exercise, define the stock model offsets as follows:

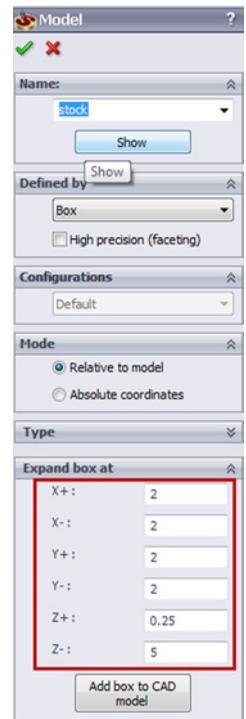
- For the **X+**, **X-**, **Y+** and **Y-** offsets, use the default values of **2**
- For the **Z+** offset, set the value to **0.25**
- For the **Z-** offset, set the value to **5**

Click on the solid body. One of its faces is highlighted.



SolidCAM automatically generates the surrounding box.

Confirm the Stock model definition by clicking the  button. The **Milling Part Data** dialog box is displayed.



9. Define the Target model

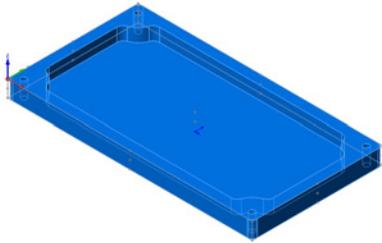
SolidCAM enables you to define the Target model, which is the final shape of the CAM-Part after the machining. SolidCAM uses the Target model for gouge checking in the **SolidVerify** simulation.

Click the **Target** button in the **Stock & Target model** area of the **Milling Part Data** dialog box.



The **Model** dialog box is displayed.

By default, when you create a new CAM-Part, stock and target models are defined automatically. If you have not changed the default settings, the solid body is highlighted, and the target model is already chosen in the **Type** section.



Click on the solid body to clear the selection. Notice that the **Solid 1** icon is also removed from the **Type** section. Click on the solid body once again. The face is selected, the **Solid 1** icon appears in the **Type** section, and the target model is defined.

Confirm the **Model** dialog box with the  button.

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

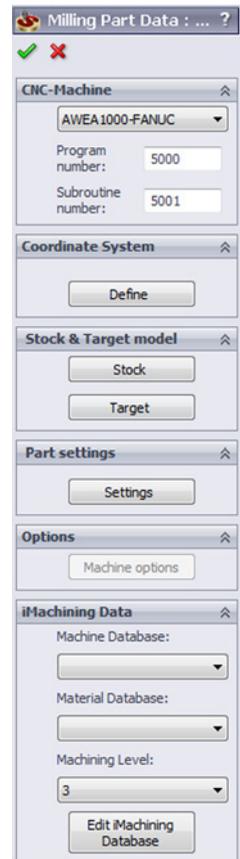
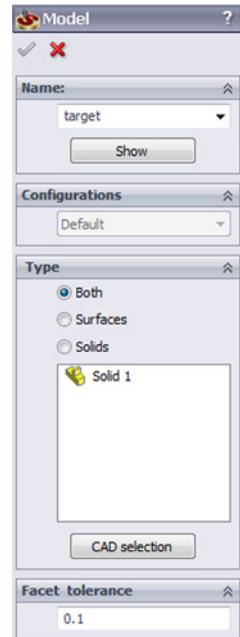


10. Save the CAM-Part data

Confirm the **Milling Part Data** dialog box by clicking the  button.

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

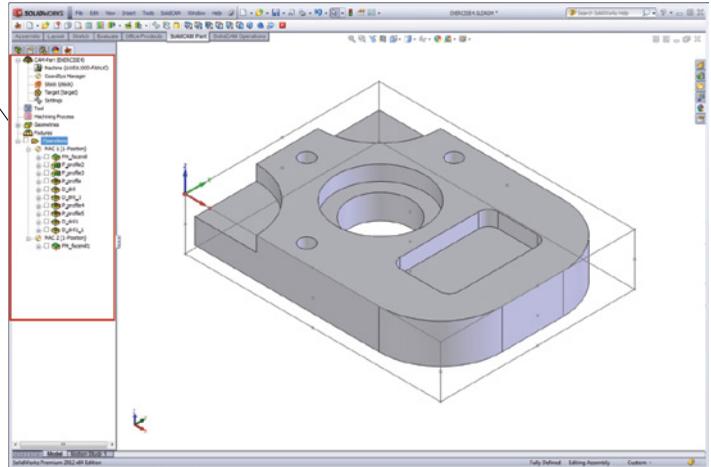
At this stage, the definition of the CAM-Part is finished. The definition of Milling operations is covered in the following exercise using this CAM-Part.





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

SolidCAM
Manager



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

CAM-Part header

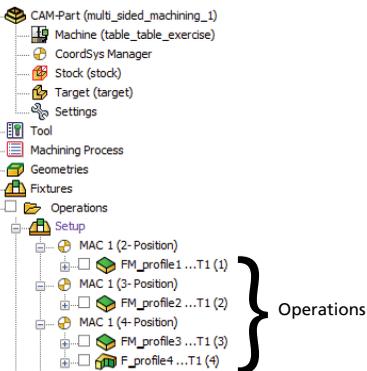
Tool header

Machining Process header

Geometries header

Fixtures header

Operations header



- **CAM-Part header**

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

The **Machine** subheader is located under the CAM-Part header. Double-click this subheader to review your machine configuration and parameters.

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 **Stock** and **Target** subheaders are located under the **CAM-Part** header. Double-click these subheaders to load the **Stock model/Target model** dialog boxes that enable you to change the definition of the Stock/Target models.

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.

- **Fixtures header**

This header displays available fixtures. By right-clicking it, you can display the menu to define and manage your fixtures.

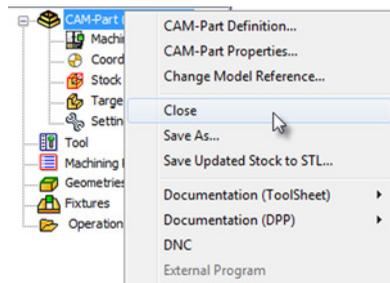
- **Operations header**

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

11. Close the CAM-Part

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

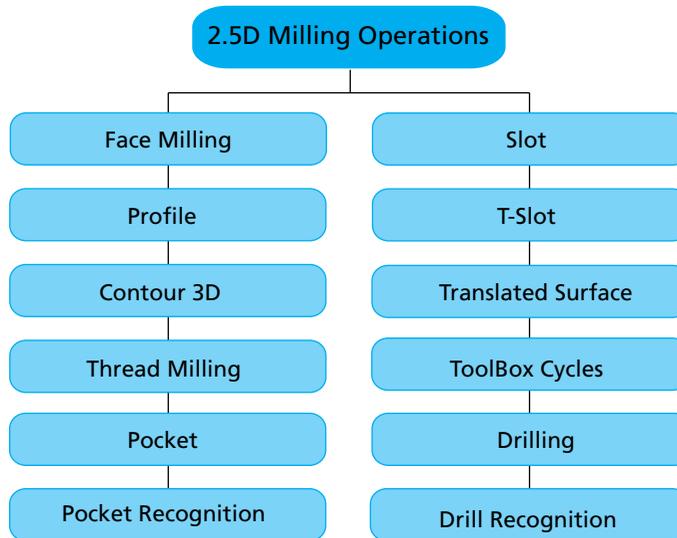
The CAM-Part is closed.



SolidCAM 2.5D Operations

3

SolidCAM offers you the following types of 2.5D Milling operations:



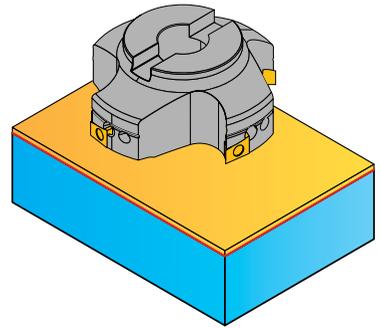
In SolidCAM, an operation is a single machining step. A workpiece is usually manufactured using several machining steps and technologies. For each of these steps you can define a separate operation. An operation can be very complex, but it always uses one tool, one major geometry and executes one machining type, e.g. Profile Milling or Drilling. You can edit any single machining operation, change the operation sequence and generate the GCode, combining and splitting the operation list of your CAM-Part.

The Machining Geometry has to be defined for each operation. The Geometry prompts SolidCAM what and where you want to machine.

- A Geometry for Profile, Pocket, Contour 3D, Slot and T-Slot operations, and Toolbox cycles consists of a number of chains. Chain geometries are defined by selecting the following entities: edges of models, 2D curves, 3D curves, circles, lines and splines. Each chain is composed of one or more entities and defines an open or closed contour.
- A Geometry for Face Milling operations can be defined by selecting solid models, faces or chains of model elements.
- A Geometry for Drilling and Thread Milling operations consists of one or more points (drilling centers) that can be defined by a number of methods directly on the solid model.
- A Geometry for Drill Recognition and Pocket Recognition operations is determined automatically by SolidCAM Automatic Feature Recognition functionality.

Face Milling Operation

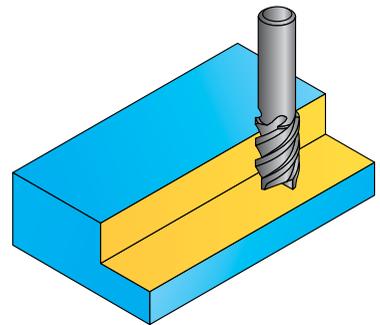
This operation enables you to machine large flat surfaces with face mill tools.



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:

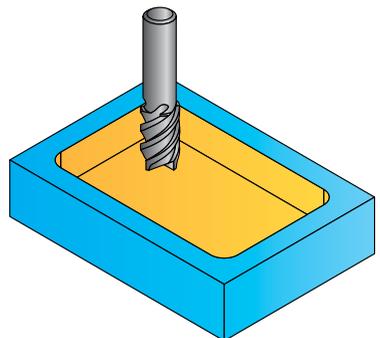
- Milling a single profile to the specified constant or variable depth in one step or in several user-defined down steps.
- Concentric profiles to the specified constant or variable depth; this type of profiling generates several concentric profiles that start from the defined clear offset distance from the profile, and finish on the profile geometry, thus clearing the area around the profile to a constant depth.



Pocket Operation

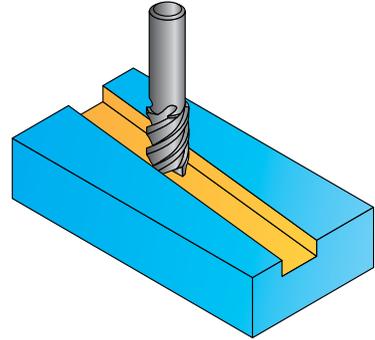
In pocket milling, you have to remove material from the interior of a closed geometry. SolidCAM offers two types of pocketing:

- 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.
- 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.



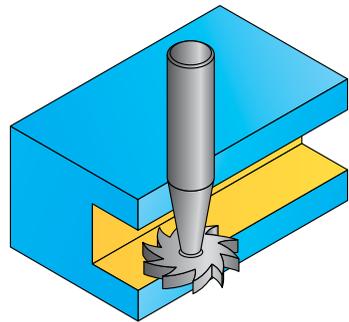
Slot Operation

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.



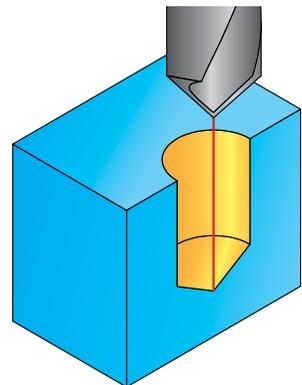
T-Slot Operation

This operation enables you to machine slots in vertical walls with a slot mill tool.



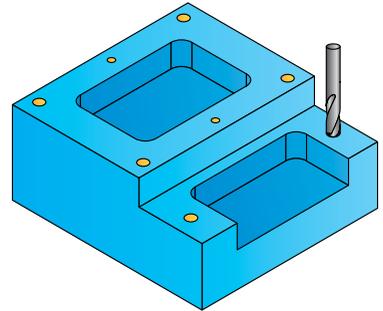
Drilling Operation

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).



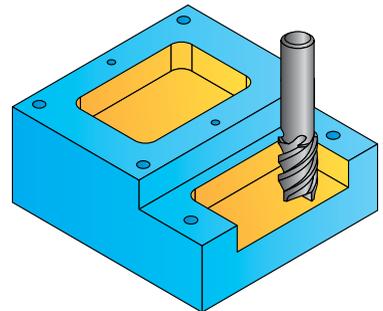
Drill Recognition

This Operation carries out a highly efficient drill recognition and geometry creation with the functionality of the AFRM-module (Automatic Feature Recognition and Machining). In this operation drilling on different levels can be carried out. The drilling levels are automatically recognized but may be edited by the user.



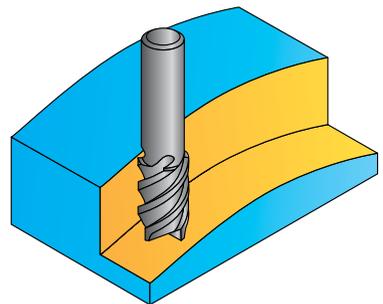
Pocket Recognition

This Operation recognizes automatically pocket features at the target model and creates the necessary machining.



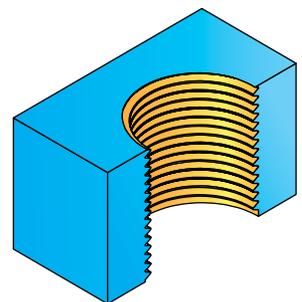
Contour 3D Operation

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.



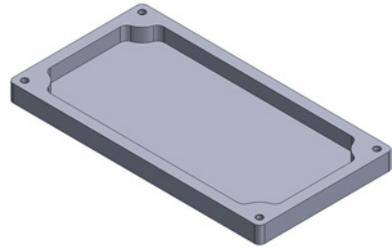
Thread Milling Operation

This operation enables you to generate a helical tool path for the machining of internal and external threads with thread mills.



Exercise #2: Cover Machining

In this exercise, you use the CAM-Part defined in the **Exercise #1**. You have to define several 2.5D operations in order to machine the model external faces, pocket and holes in the corners.

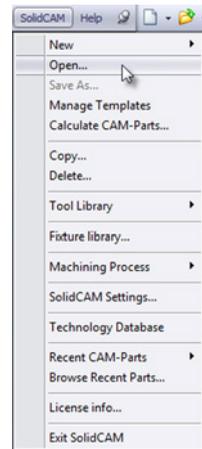


In the process of definition of operations, you have to define the machining geometry, the tool and several technological parameters.



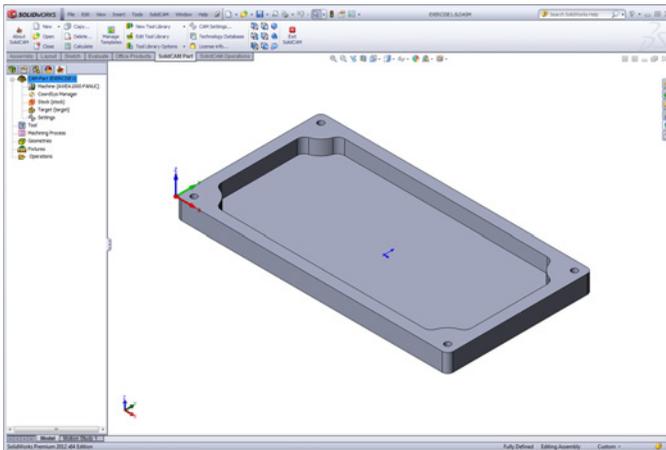
1. Open the CAM-Part

In the menu, choose **SolidCAM > Open**, or click **Open** on the **SolidCAM Part** toolbar.



In the browser window, choose **Exercise1** – the CAM-Part prepared in the previous exercise.

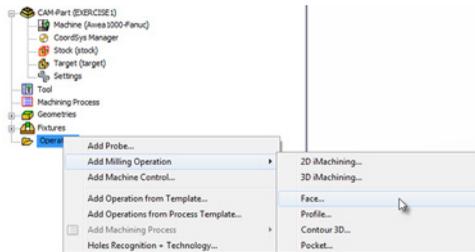
The CAM-Part is loaded.



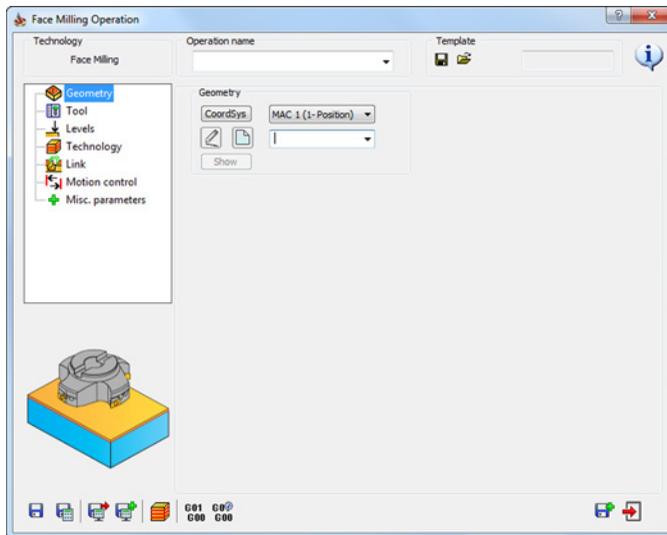


2. Add a Face Milling operation

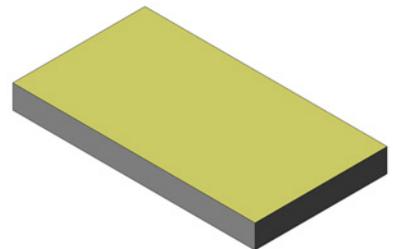
In **SolidCAM Manager**, right-click the **Operations** header and choose **Face** from the **Add Milling Operation** submenu.



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



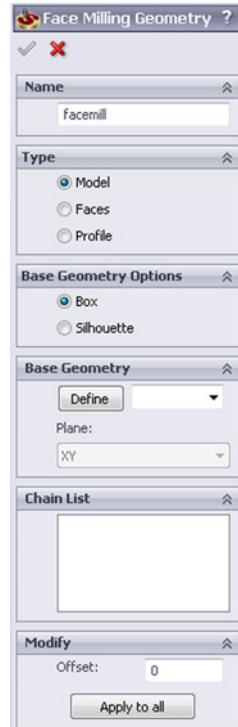
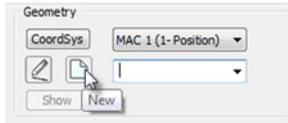
In this operation, the upper face is machined.





3. Define the Face Milling geometry

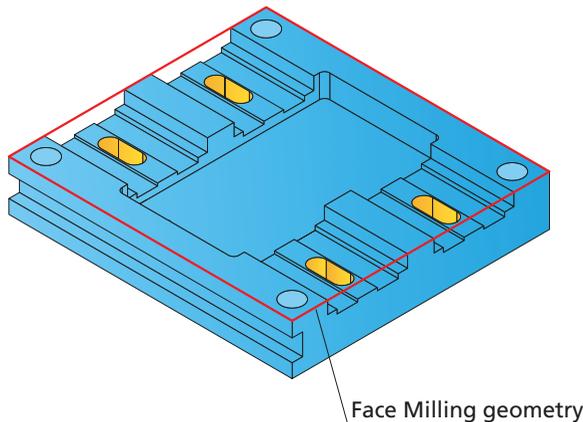
Click the **New** button (📄) in the **Geometry** page. The **Face Milling Geometry** dialog box is displayed.



The **Type** section enables you to define the face milling geometry using the following methods:

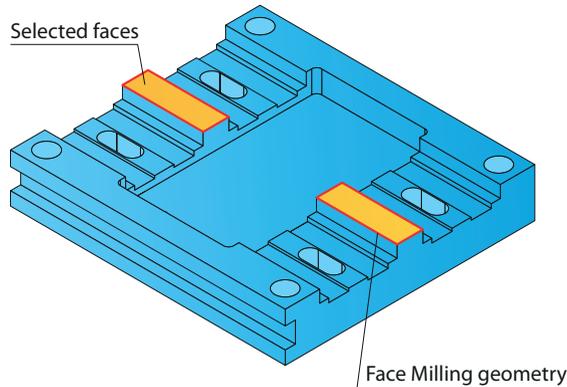
- **Model**

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.



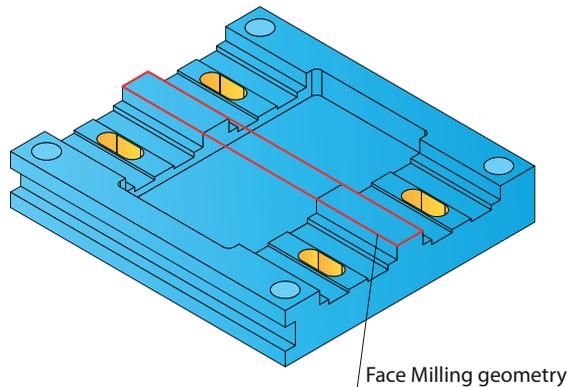
- **Faces**

This option enables you to define the Face Milling geometry by face selection. The **Define** button and the related box enable you either to define a new faces geometry with the **Select Faces** dialog box or 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 box enable you either to define a new profile geometry with the **Geometry Edit** dialog box or choose an already defined geometry from the list. The defined chains are displayed in the **Chain List** section.



In the **Type** section, use the default **Model** option for the Face Milling geometry definition. Click the **Define** button. The **3D Geometry** dialog box is displayed.

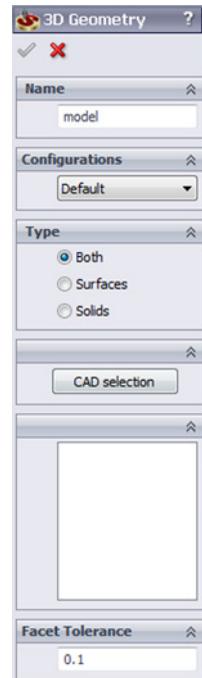
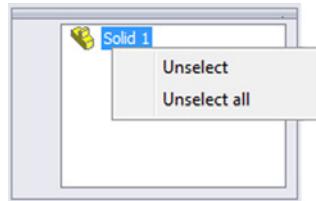


This dialog box enables you to define the 3D Model geometry by selecting the following types of model elements:

- **Solid** – only solid objects are selected;
- **Surfaces** – only surfaces are selected;
- **Both** – both surfaces and solids will be selected.

The **CAD selection** button enables you to select the 3D geometry with SolidWorks tools.

You can select an object by clicking on it. When an object is selected, its icon is displayed in the list in the bottom of the dialog box. To unselect the object, click on it again or right-click its icon in the list of selected elements and choose **Unselect** from the menu. To remove selection from all objects in the list, click **Unselect all**.

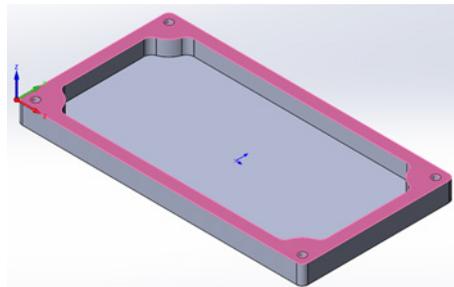


Click on the solid model to select it. The model is highlighted, and its icon appears in the list. Confirm the **3D Geometry** dialog box by clicking . The **Face Milling Geometry** dialog box is displayed again.

The rectangle is generated surrounding the target model at the XY-plane.

Confirm the **Face Milling Geometry** dialog box by clicking .

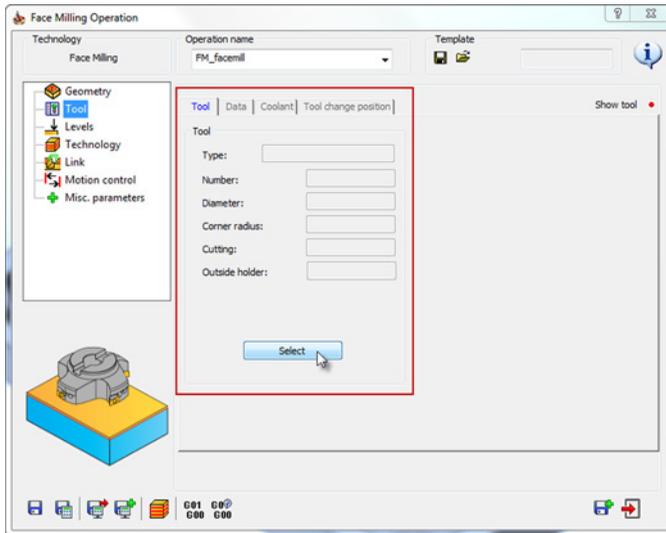
The geometry is defined for the operation.



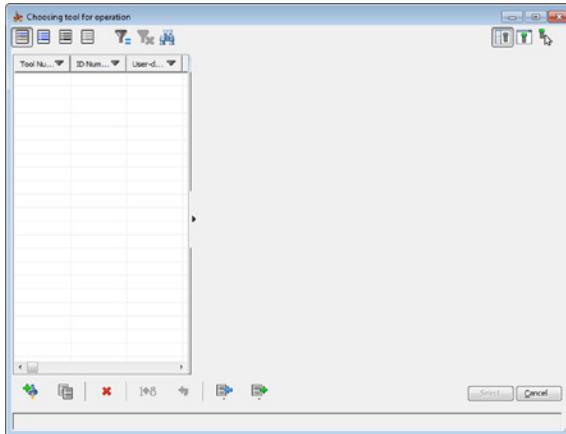


4. Define the Tool

Switch to the **Tool** page of the **Face Milling Operation** dialog box. Start the tool definition by clicking the **Select** button.



The **Choosing tool for operation** dialog box with the **Part Tool Table** is displayed.



The **Part Tool Table** contains all tools available for use to machine a specific CAM-Part. The **Part Tool Table** is stored within the CAM-Part.

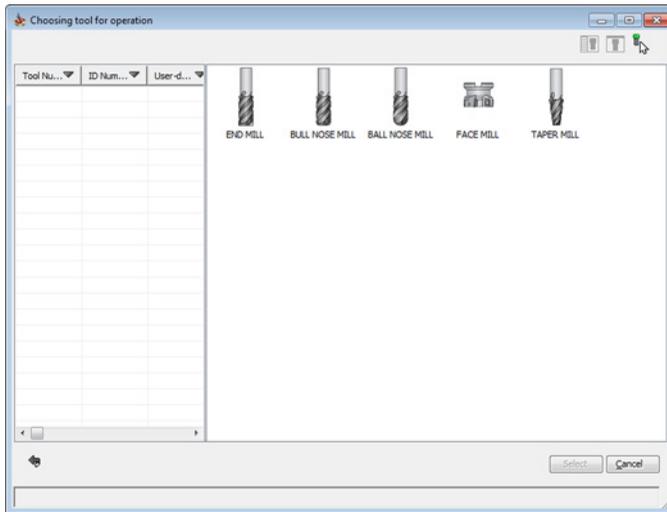
This dialog box enables you to manage the tools contained in the **Part Tool Table**.



Currently, the **Part Tool Table** is empty. Define a new tool suitable for face milling.

Click the **Add Milling Tool** button () to start the tool definition.

The new pane containing available tools is displayed.

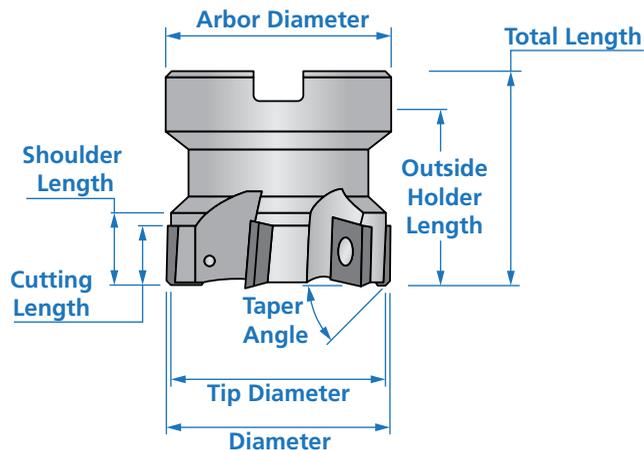


This dialog box enables you to add a new tool to the tool library choosing from the tools available for the current operation. In this operation, a face mill of $\text{Ø}100$ will be used.



Face mill

This tool type is used for machining of large flat surfaces. A tool of this type is defined with the parameters shown in the image.



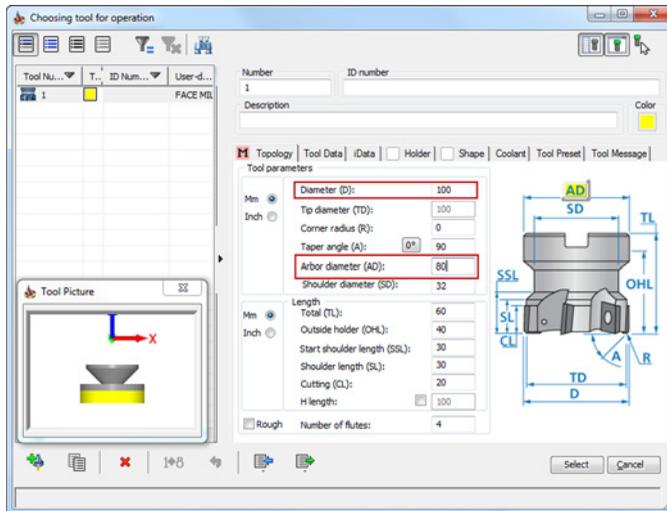


Choose the **Face mill** tool.

In the new pane, define the following parameters:

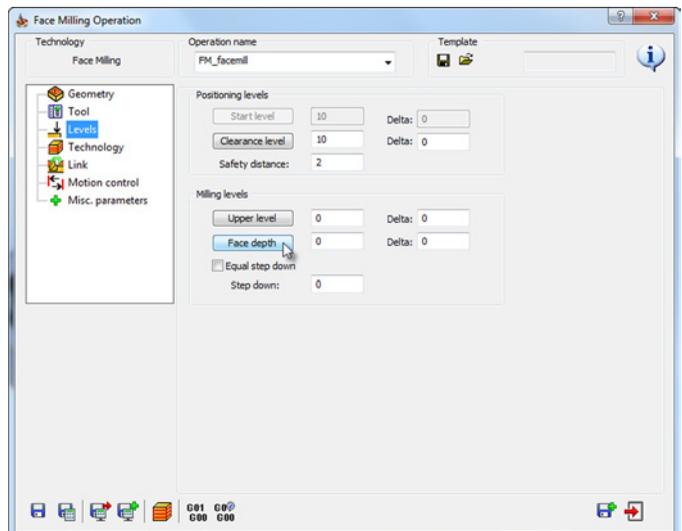
- Set the **Diameter** to **100**
- Set the **Arbor Diameter** to **80**

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

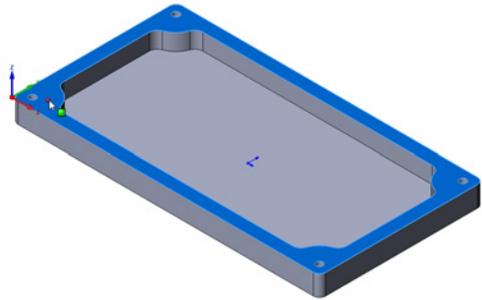


5. Define the Face depth

Switch to the **Levels** page of the **Face Milling Operation** dialog box. Click the **Face depth** button in the **Milling levels** area. This button enables you to define the Operation Lower level directly on the solid model. The depth is calculated automatically as the difference between the Z-values of the Operation Upper and Lower levels.



The **Pick Lower level** dialog box is displayed.



Select the model face as shown.

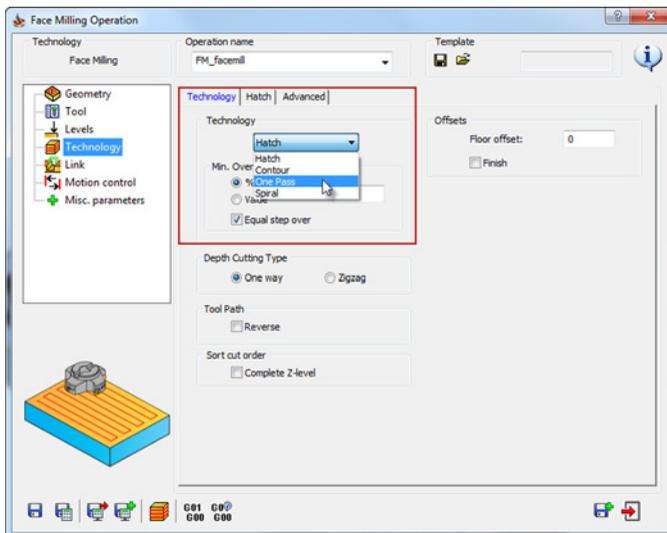
The lower level value (0) is determined and displayed in the **Pick Lower level** dialog box. Confirm this dialog box with .

The **Face depth** value is displayed in the **Milling levels** area. The pink background of the edit box means that the parameter is associative to the model. Associativity enables the selected level to be synchronized with the solid model changes; SolidCAM automatically updates the CAM data when the model is modified.



6. Define the technological parameters

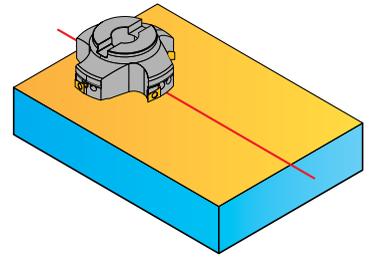
Switch to the **Technology** page of the **Face Milling Operation** dialog box. In the **Technology** section, choose the **One Pass** option.





One Pass machining technology

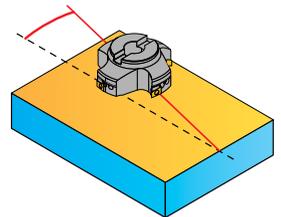
SolidCAM performs the face milling in one pass. The direction and location of the pass are calculated automatically according to the face geometry, in order to generate an optimal tool movement with the tool covering the whole geometry.



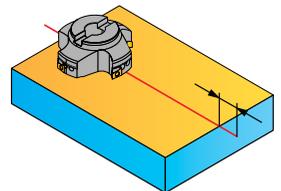
Selecting the **One pass** option automatically opens the **One pass** tab that enables you to define the machining parameters.



The **Hatch angle** section enables you to define the angle between the tool path and the geometry.



The **Extension** section enables you to define the tool path extension over the face edges. The extension can be defined either by percentage of the tool diameter (the **% of tool diameter** option) or by value (the **Value** option).



7. Save and Calculate

Click the **Save & Calculate** () button.

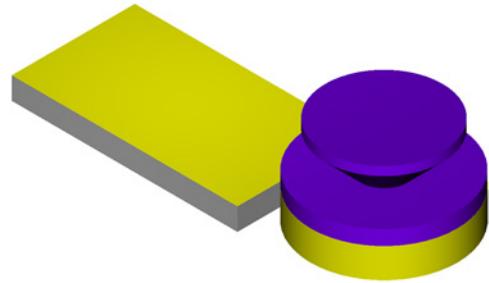
The Face Milling operation data is saved, and the tool path is calculated.



8. Simulate

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

Switch to the **SolidVerify** page and start the simulation with the  button.



The solid stock model defined in **Exercise #1** is used in the **SolidVerify** simulation mode. During the machining simulation process, SolidCAM subtracts the tool movements (using solid Boolean operations) from the solid model of the stock. The remaining machined stock is a solid model that can be dynamically zoomed or rotated. It can also be compared to the target model in order to show the rest material.

During the simulation, you can rotate , move , or zoom  the model. Use these options to see the machining area in details.

The **Single step** mode can be used to simulate the next tool movement by clicking the  button or by using the space bar on your keyboard.

Close the simulation with the  button. The **Face Milling Operation** dialog box is displayed. Close this dialog box with the **Exit**  button.

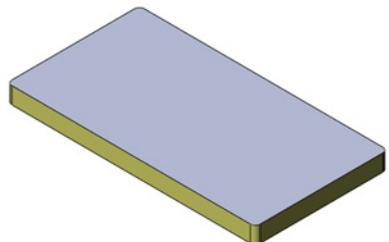
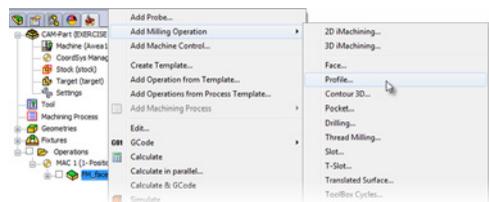


9. Add an operation

Right-click the Face Milling operation entry in **SolidCAM Manager** and choose **Profile** from the **Add Milling Operation** submenu.

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

In this operation, the external profile is machined.





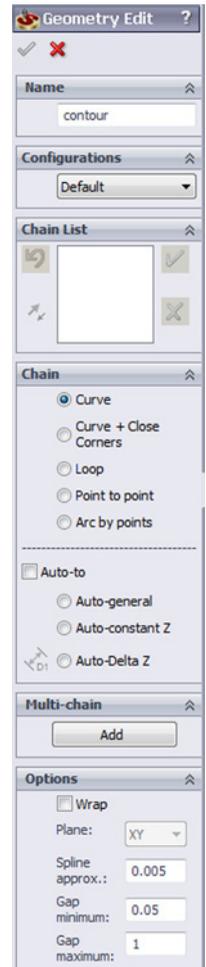
10. Define the Geometry

The first step of definition of each operation is the Geometry selection. At this stage, you have to define the Geometry for the Profile operation using the solid model geometry.

Click  in the **Geometry** page of the **Profile Operation** dialog box.

The **Geometry Edit** dialog box is displayed in the SolidWorks **PropertyManager** area. This dialog box enables you to add and edit geometry chains.

When this dialog box is displayed, you can select solid model entities for the Geometry definition.



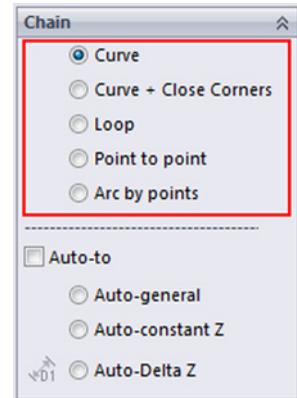
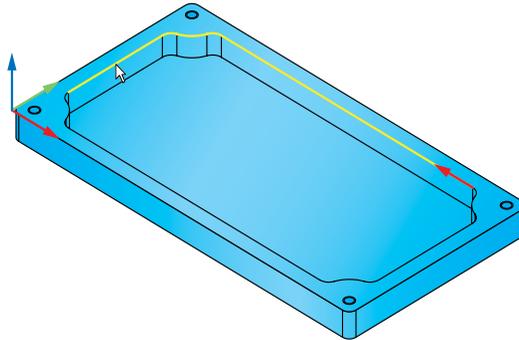


Chain Selection Options

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

Curve

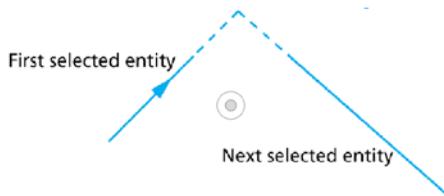
This option enables you to create a chain of existing curves and edges by selecting them one after the other.



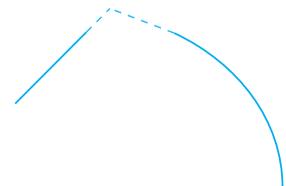
Associativity: SolidCAM keeps the associativity to any edge or sketch entity. Any change made to the model or sketch automatically updates the selected geometry.

Curve + Close corners

This option enables you to close the gaps between successive chain entities irrespective of the **Gap minimum** and **Gap maximum** values (defined in the **Units** section of **SolidCAM Settings**) by virtually extending the entities up to their intersection.



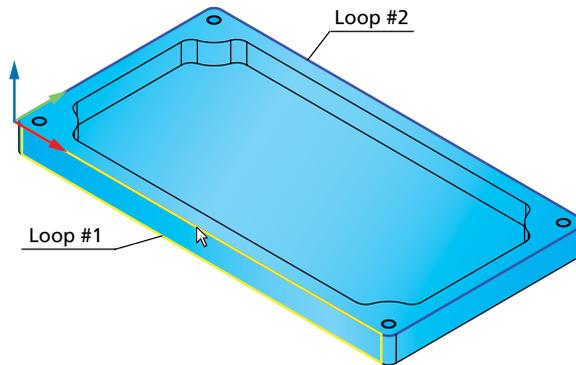
Splines and arcs are extended by lines tangential to the arc/spline at its end point.



Associativity: When the model used for the geometry definition is modified, SolidCAM enables you to synchronize the geometry with the updated model. During the synchronization, SolidCAM determines gap areas created using the **Curve + Close Corners** option and regenerates the extension of the chain elements so as to close the gaps.

Loop

This option enables you to select a loop by picking one of the model edges.

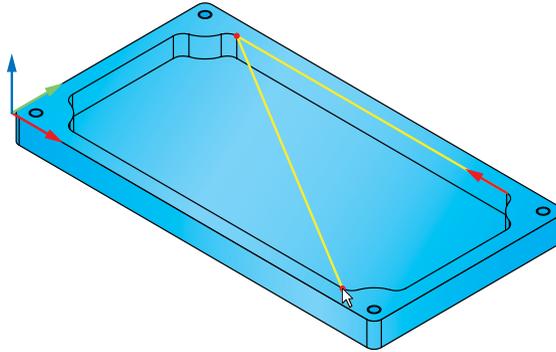


1. Pick an edge shared by two model faces. Two faces to which this edge belongs are determined, and their loops are highlighted. The first determined loop is considered to be the primary and is highlighted with yellow color. The second loop is considered to be the secondary and is highlighted with blue color.
2. Choose one of the loops. Click on any other edge forming the face. You are prompted to accept the chain that is now highlighted with yellow color. Accept the chain with the **Yes** button. A closed geometry chain is defined on this loop, and the secondary loop is rejected.

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.



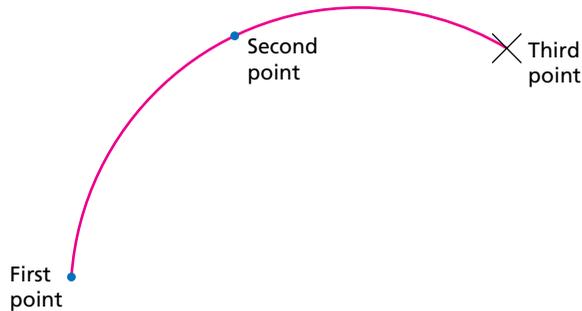
You cannot select a point that is not located on a SolidWorks entity (if you need to select such a point, add a planar surface under the model and select the points on that surface).

The following rules apply to the virtual line selection using the **Point to point** option:

- When you select a virtual line between two edges, the line behaves as a spring. Whenever the model is changed and synchronized, the geometry is updated with the model.
- When you select a sequence of several virtual lines, only the points connected to model edges or sketch elements are updated, but all other points stay fixed at the defined X-, Y- and Z-positions.

Arc by points

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



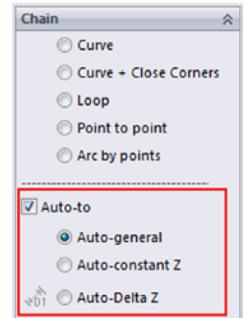
Associativity: SolidCAM does not keep the associativity to any selected arcs by points. SolidCAM saves the X-, Y- and Z-coordinates of the selected points. Any change made to the model or sketch does not update the selected geometry.

Automatic selection options

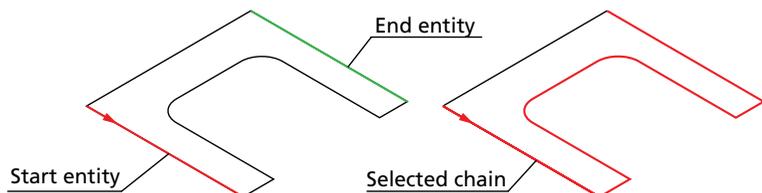
SolidCAM automatically determines the chain entities and close the chain contour. The Auto select mode offers the following options:

Auto-to

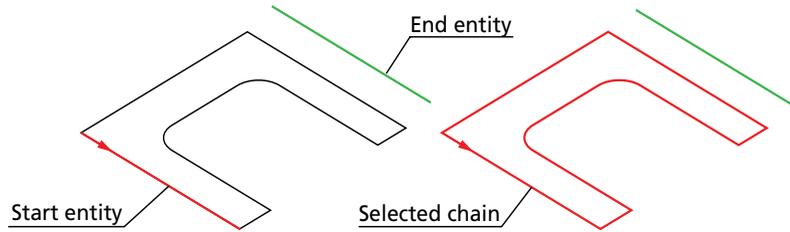
The chain is selected by specifying the start curve, the direction of the chain and the element up to which the chain is created. SolidCAM enables you to choose any model edge, vertex or sketch entity to determine the chain end.



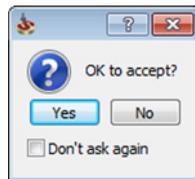
When the end item is chosen, SolidCAM determines the chain according to the rules of the chosen selection mode (**Auto-general**, **Auto-constant Z** or **Auto-Delta Z**). The chain selection is terminated when the selected end item is reached.



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.



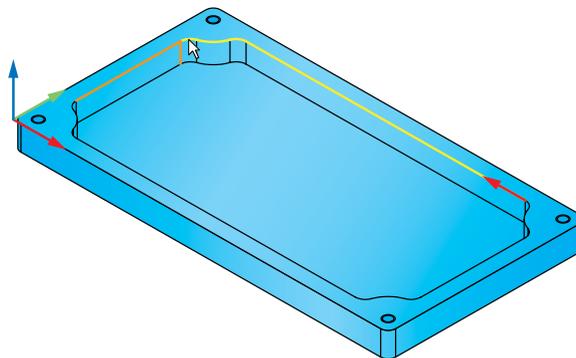
The confirmation message is displayed.



The **Auto-to** option is useful if you do not want to define a closed chain, but an open chain up to a certain element.

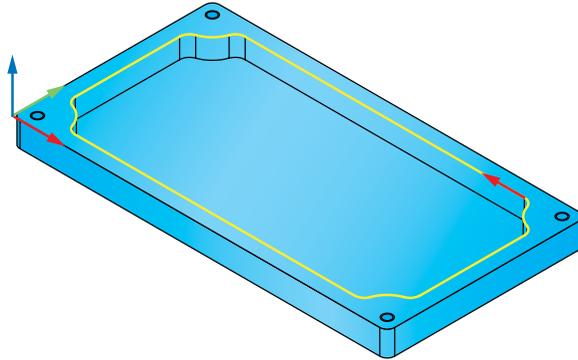
Auto-general

SolidCAM highlights all the 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 found 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 **SolidCAM Settings**.

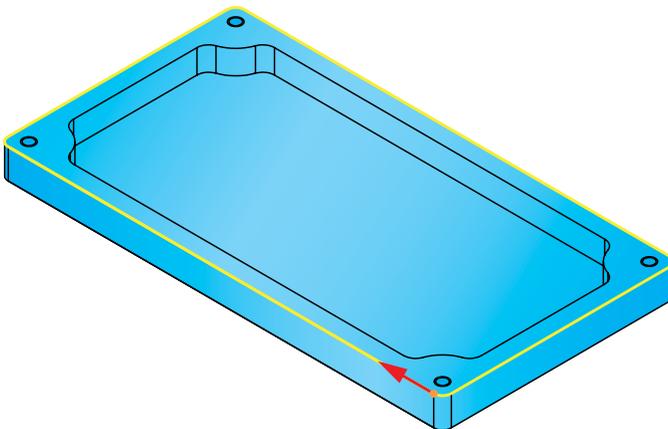
**Auto-Delta Z**

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



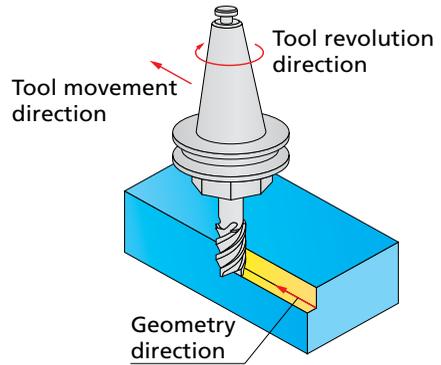
In this exercise, the geometry must be defined as shown.

The red arrow indicates the direction of the geometry.

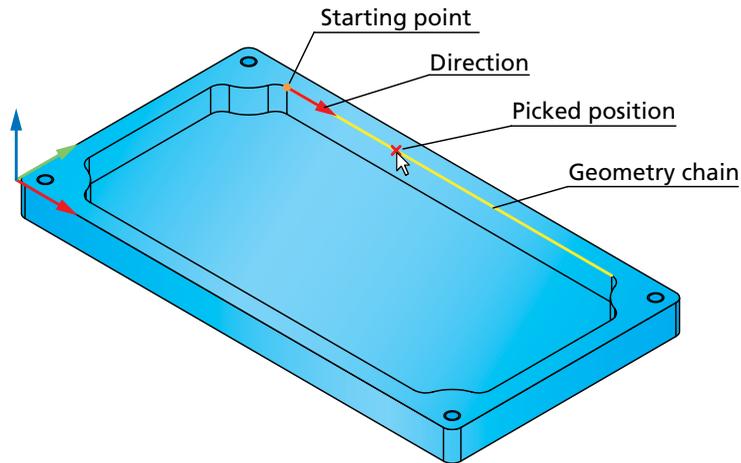




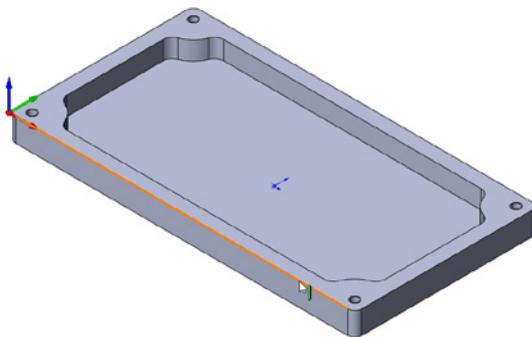
In SolidCAM operations, the direction of the chain geometry is used for the tool path calculation. In Profile milling, the tool moves in the direction of the geometry by default. In this exercise, the combination of the geometry direction and the clockwise direction of the tool revolution enables you to perform climb milling.



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

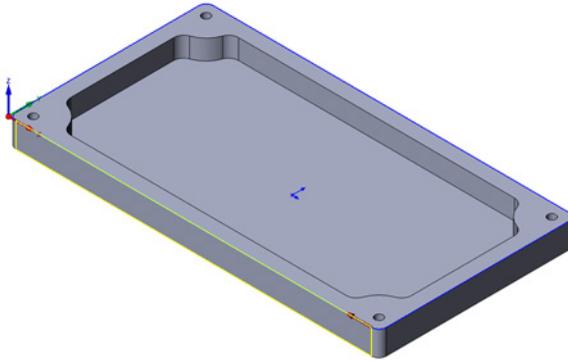


Choose the **Loop** option in the **Chain** section and click on the model edge as shown.



Notice that the picked position must be close to the start point of the geometry.

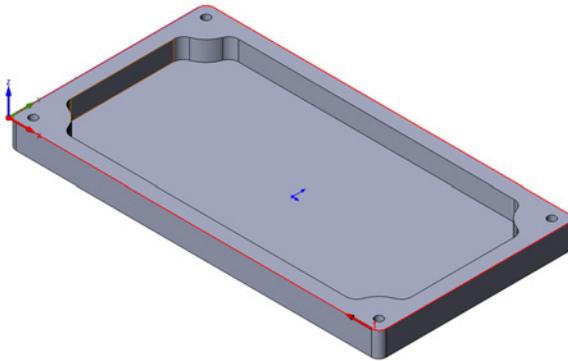
Two faces shared by the selected edge are determined, and their loops are highlighted. The red arrow indicates the direction of the selected chain.



Click the secondary chain highlighted with blue color to choose it for geometry definition.

The confirmation message is displayed. Confirm it with **Yes**.

The picked chain is now highlighted with red color, and the second chain is rejected.



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



At this stage, the Geometry is defined. Confirm the Geometry selection with . The **Profile Operation** dialog box is displayed.

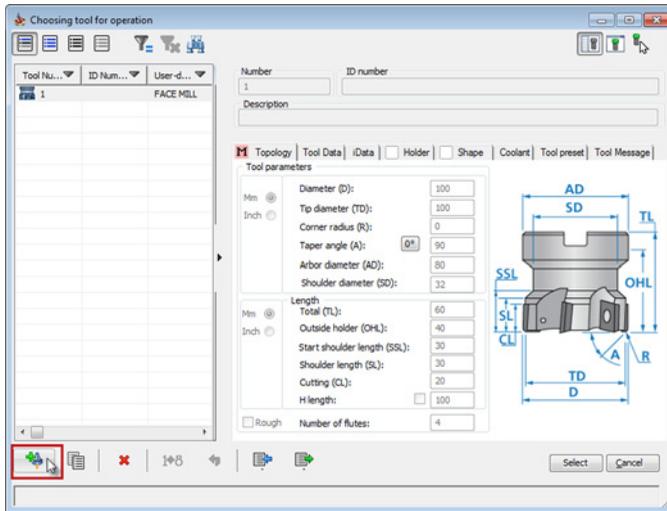


11. Define the Tool

At this stage, you have to define the tool for the Profile milling.

Switch to the **Tool** page of the **Profile Operation** dialog box and click the **Select** button.

The **Part Tool Table** with the tool used in the previous operation is displayed.



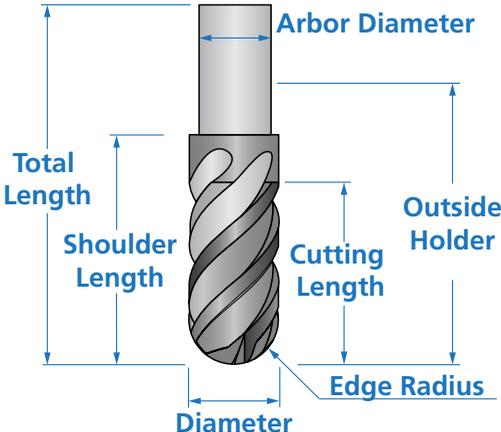
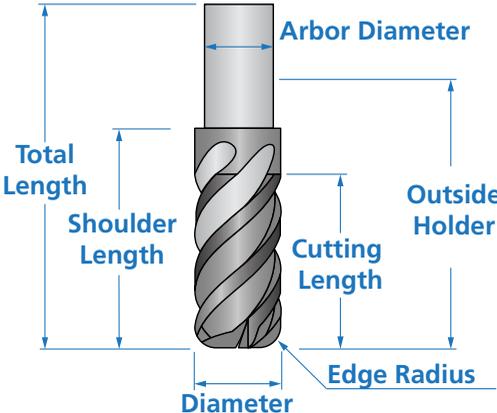
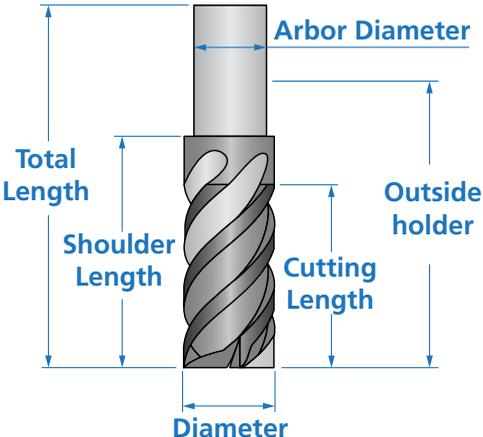
Click the **Add Milling Tool** button () to start the tool definition.

In this Profile operation, an **End mill** of **Ø16** will be used.



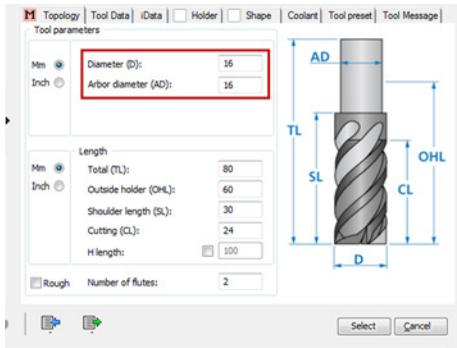
End Mill/Bull Nose Mill/Ball Nose Mill

These tool types are used for the definition of rough/finish mills. The tool shapes and basic parameters are shown below:





Click the **End Mill** tool to choose it for the operation. In the **Tool parameter** section, under **Topology**, set the **Diameter** value to **16**.

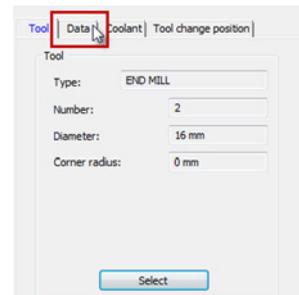
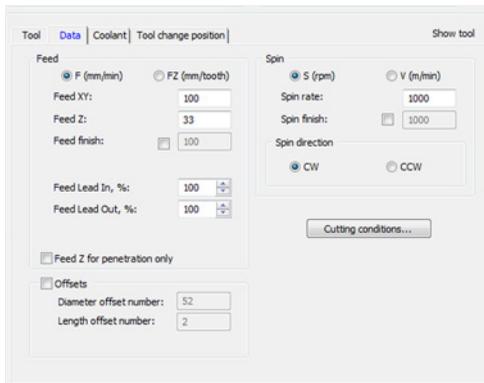


Click the **Select** button to confirm the tool parameters and choose the tool for the operation.

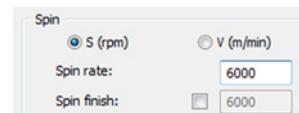


12. Define the Feed and Spin parameters

Switch to the **Data** tab in the **Tool** section of the **Profile Operation** dialog box.



Set the **Spin rate** (used in rough milling) value to **6000**. The **Spin finish** (used in finish milling) value is automatically set to **6000**.



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.

Set the **Feed XY** (feed rate for XY movements) to **1500** and the **Feed Z** (feed rate for Z movements) to **250**. Select the check box near the **Feed finish** (feed rate for finish milling) parameter and set the value to **1800**.



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.



13. Define the Milling levels

Switch to the **Levels** page of the **Profile Operation** dialog box.

SolidCAM enables you to define the milling levels using the solid model data.

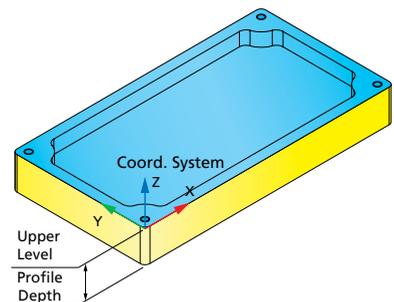


Upper Level

This parameter defines the Z-level at which the machining starts.

Profile Depth

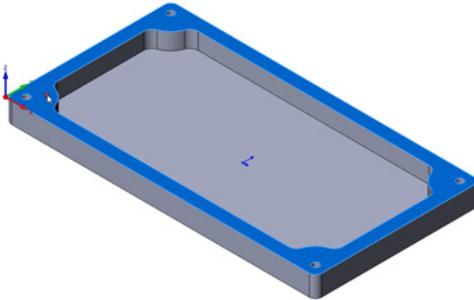
This parameter defines the Z-level below which the tool does not mill. This plane is not penetrated in any milling strategy.





Click the **Upper level** button in the **Milling levels** area. The **Pick Upper level** dialog box is displayed.

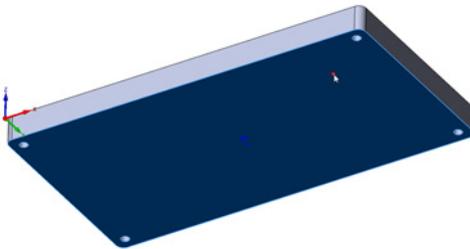
Select the model face as shown.



The **Upper Level** value (**0**) is determined and displayed in the **Pick Upper level** dialog box. Confirm this dialog box by clicking .

Click the **Profile depth** button in the **Milling levels** area. The **Pick Lower level** dialog box is displayed.

Pick the bottom edge of the model as shown.



The **Lower level** value (**-10**) is determined and displayed in the **Pick Lower level** dialog box. Confirm this dialog box with .

Now you need to define the **Delta** depth.



The **Delta** depth parameter defines the offset for the cutting depth that can be changed with its associativity preserved. The **Delta** value is always relative to the **Profile Depth** defined for the operation.



Set the **Delta** value to **-1**.

The screenshot shows a dialog box with two sections: 'Positioning levels' and 'Milling levels'. In the 'Positioning levels' section, 'Start level' is 10, 'Clearance level' is 10, and 'Safety distance' is 2. In the 'Milling levels' section, 'Upper level' is 0 and 'Profile depth' is 10. The 'Delta' value for 'Profile depth' is set to -1, which is highlighted with a red box.

The milling levels are defined.



14. Define the technological parameters

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

The screenshot shows the 'Technology' page of the 'Profile Operation' dialog box. The 'Operation name' is 'F_contour'. The 'Technology' page is divided into several sections: 'Modify' (Tool side: Left, Geometry; checkboxes for Ignore geometry self-intersect and Compensation), 'Depth type' (Constant, Pick), 'Depth cutting type' (One way, Zigzag), 'Rest material/Chamfer' (None), 'Use fillet size for last cut' (Internal: 0, External: 0), 'Offsets' (Wall offset: 0, Floor offset: 0), 'Equal step down', 'Rough' (Step down: 0), 'Finish' (Number of passes: 1, Extension/Overlap: 0, Step down: 0), 'Clear offset' (Offset: 0, Step over: 0), 'One way', 'Zigzag', 'Complete Z-level', and 'Sort by chains'.

First, you need to make sure that the tool position relative to the geometry is correct. In the **Modify** section, check the **Tool side** option.



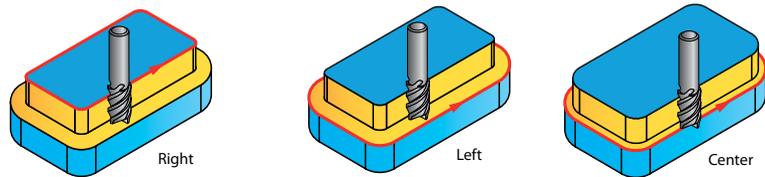
Modify

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.

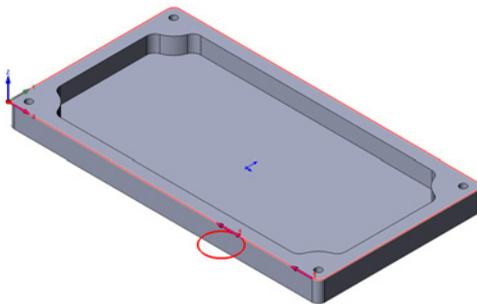
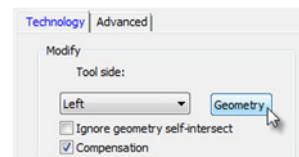
Left – the tool cuts on the left side of the profile geometry.

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 this case, the default **Left** option meets the requirements of climb milling. Click the **Geometry** button to check the tool position.



This tool position is correct. Click  in the **Modify Geometry** dialog box. The **Profile Operation** dialog box is displayed again.

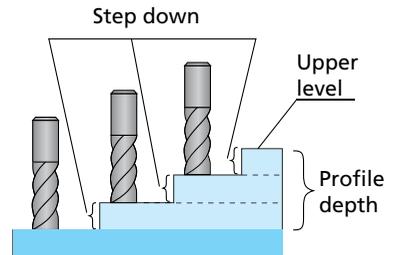
SolidCAM enables you to perform the rough and finish machining of the profile in a single Profile operation.

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 **5**. With this value, SolidCAM performs two cuts at the following Z-levels: **-5, -10**; the last cut is performed at the Z-level defined by **Profile depth**.

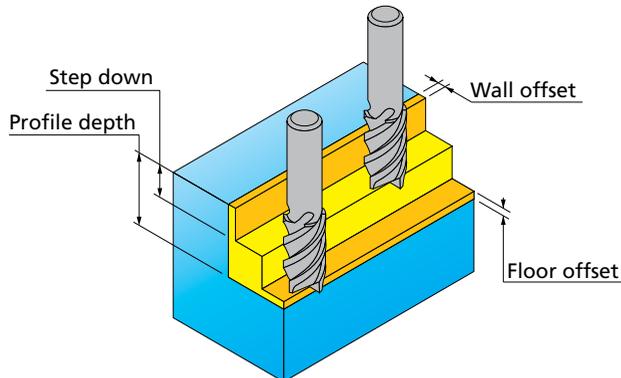


Now you need to define the wall offset that will remain after the roughing passes.

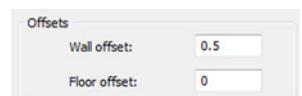


Offsets

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** value to **0.5**. The allowance of 0.5 mm will be left on the model walls during the profile roughing. This allowance will be 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.

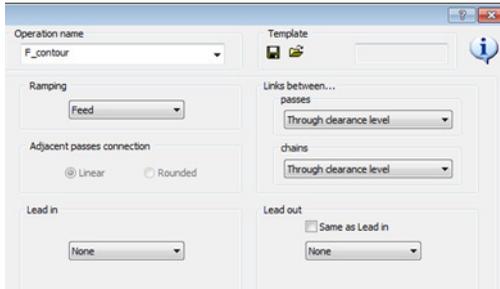


Finish
Number of passes: 1
Extension/Overlap: 0
Step down: 0



15. Define the Lead in and 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.



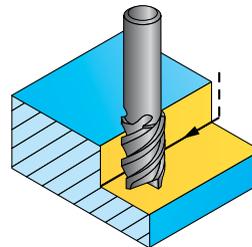
Profile Lead in and Lead out

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:

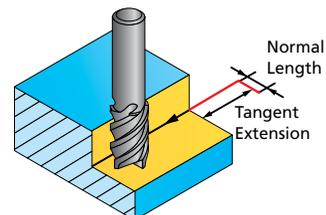
- **None**

The tool leads in to and out from the milling level exactly adjacent to the start point of the profile.



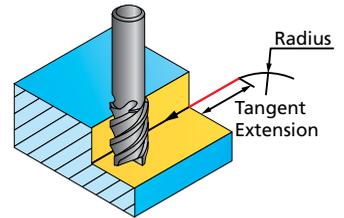
- **Normal**

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 **Normal length** field. The distance between the normal and material is set in the **Tangent extension** field.



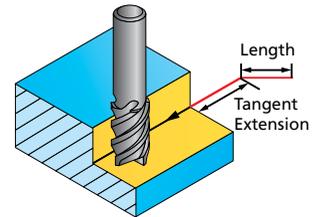
- **Arc**

The tool leads in to and out from the profile with a tangential arc. The arc radius can be set in the **Radius** field. The length of the extension can be set in the **Tangent extension** field. The arc angle is set in the **Arc angle** field.



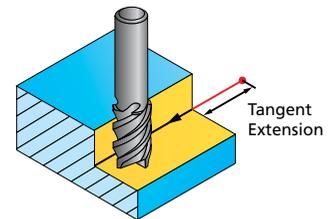
- **Tangent**

The tool leads in/out on a line tangent to the profile. The length of the tangent can be set in the **Length** field. The distance to the material can be set in the **Tangent extension** field.



- **Point**

The tool leads in/out from a user-defined 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. The distance between the point and material is set in the **Tangent extension** field.



- **User-defined**

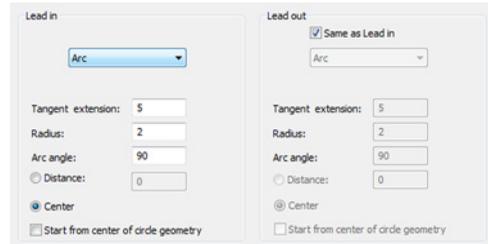
The tool leads in/out from a user-defined position. When you select this option, you can define a geometry of the tool approach to the material.

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, then set the **Tangent extension** value to **5** and the **Radius** value to **2**.

Under **Lead out**, select the **Same as Lead in** check box.

The definition of the basic technological parameters of profile milling is finished.



16. Calculate the Tool path

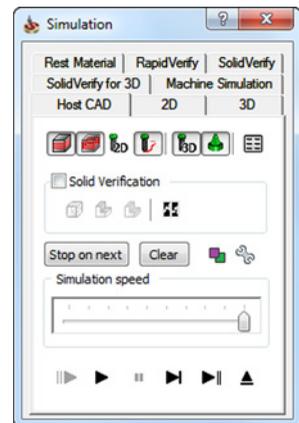
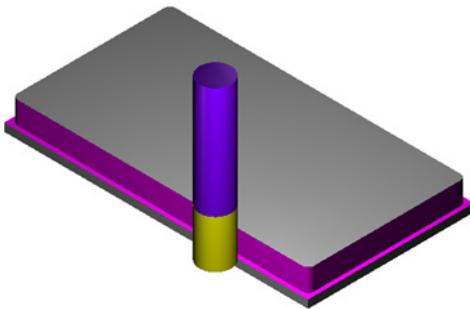
Click the **Save & Calculate** button. The Profile operation data is saved, and the tool path is calculated.



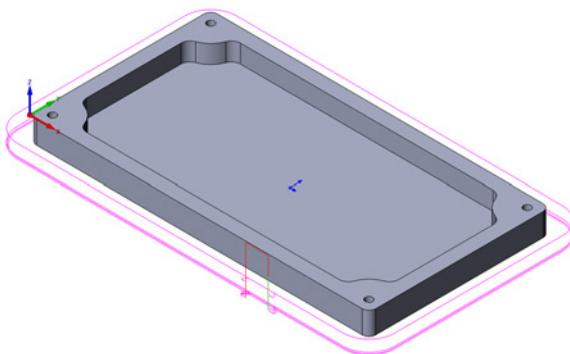
17. Simulate the operation

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 **Play** button.

When the simulation is finished, play the it step by step using the button.



Switch to the **Host CAD** simulation mode and click the **Play** button.





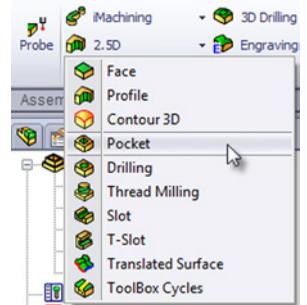
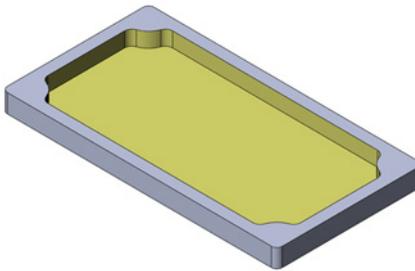
The **Host CAD** simulation mode enables you to display the tool path directly on the model in the SolidWorks window. Since all the **View** options of SolidWorks are active during the simulation, you can see the tool path from different perspectives and zoom on a certain area of the model.

Close the simulation with the **Exit**  button. The **Profile Operation** dialog box is displayed.

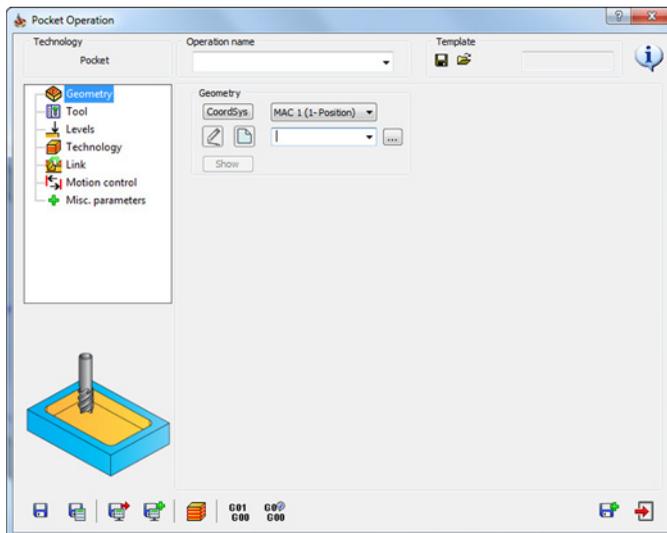


18. Add a Pocket operation

The Pocket operation is used for the internal pocket machining. On the **SolidCAM Operations** toolbar, click **2.5D > Pocket**.



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



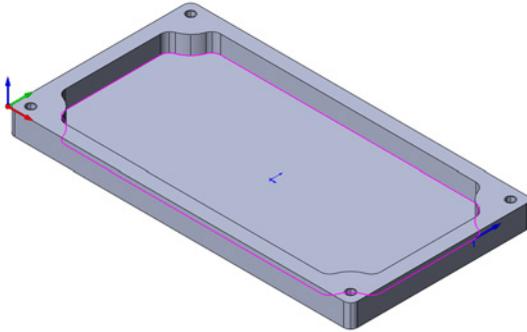


19. Define the Geometry

The geometry for a Pocket operation is generally represented by closed chains. In this exercise, you have to define a chain using the solid model edges.

Click the  button in the **Geometry** page to start the geometry definition. The **Geometry Edit** dialog box is displayed.

Using the **Loop** option, define the chain as shown.



Confirm the geometry definition by clicking .

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



20. Define the Tool

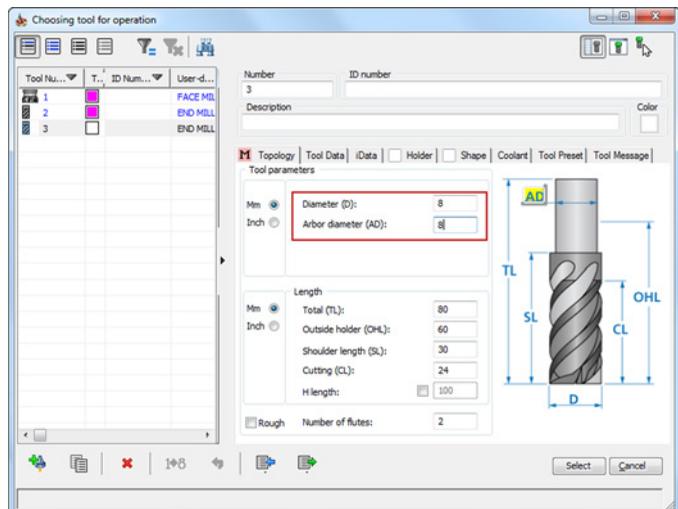
Switch to the **Tool** page and click the **Select** button.

The **Part Tool Table** is displayed.

Click  to start the tool definition. The **Milling Tools** table is displayed.

In this Profile operation, an **End mill** of $\varnothing 8$ will be used.

Choose the **End mill** tool for the operation. In the **Topology** page, set the **Diameter** value to 8.



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

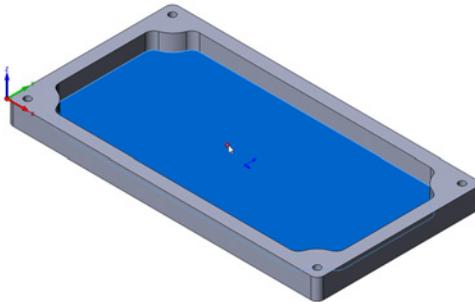
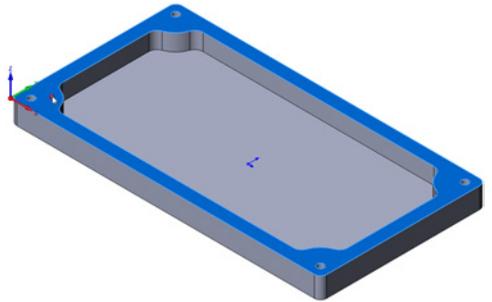


21. Define the Milling levels

Switch to the **Levels** page of the **Pocket Operation** dialog box and define upper and lower levels of machining directly on the solid model.

Define the **Upper level** as shown. The **Upper level** value (**0**) is determined.

Define the **Pocket depth** by clicking on the pocket bottom face as shown. The **Pocket depth** value (**8**) is determined.



Set the **Step down** value to **4** to perform the pocket machining in two equal steps.

Milling levels			
Upper level	0	Delta:	0
Pocket depth	8	Delta:	0
<input type="checkbox"/> Equal step down			
Step down:	4		



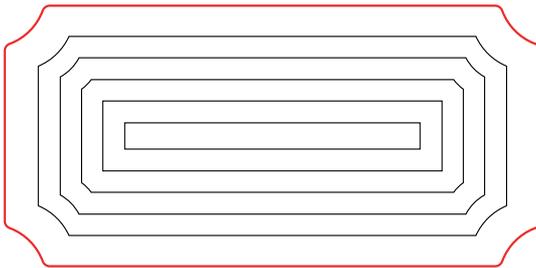
22. Define the technological parameters

Switch to the **Technology** page of the **Pocket Operation** dialog box. In the **Offsets** section, set the **Wall offset** and the **Floor offset** values to **0.2**. These offsets remain unmachined during roughing and are removed with the further finishing.

In the **Finish** area, select the **Wall** and **Floor** check boxes. These options enable you to perform finishing of the **Wall offset** and **Floor offset** that remain after the roughing.

The screenshot shows two sections of the software interface. The top section, titled 'Offsets', contains three input fields: 'Wall offset' with a value of 0.2, 'Island offset' with a value of 0, and 'Floor offset' with a value of 0.2. The bottom section, titled 'Finish', has two checked checkboxes for 'Wall' and 'Floor'. Below these are two radio buttons for 'On': 'Geometry' (selected) and 'Offset'. There are also two radio buttons for 'Depth': 'Total depth' and 'Each step down' (selected). A checkbox for 'Wall finish only' is present and unchecked.

Define the machining strategy. Make sure that the default **Contour** option is chosen in the **Technology** section. When the **Contour** strategy is chosen, the tool moves on offsets parallel to the pocket contour.



The screenshot shows the 'Technology' section of the software interface. The 'Technology' dropdown menu is set to 'Contour' and is highlighted with a red box. Below it, the 'Min. Overlap' section has two radio buttons: '% of tool diameter' (selected) and 'Value' (with a value of 50). A checkbox for 'Equal step over' is checked.

Switch to the **Contour** tab to display the Contour parameters. This page enables you to define the parameters of the Contour strategy.

The screenshot shows the 'Contour' tab of the software interface. It contains several sections: 'Start from' with 'Inside' selected; 'Corner' with a dropdown set to 'None' and 'Corner radius' and 'Min. corner radius' both set to 0; 'Direction' with 'Climb' selected; and 'Adjacent passes connection' with a dropdown set to 'Linear'. At the bottom, there are two checkboxes: 'Exit material' and 'Connect islands', both of which are unchecked.



Contour parameters

Start from

- **Inside**

This option allows you to work in a pocket area starting from the middle of the pocket and cutting towards the outside border of the pocket.

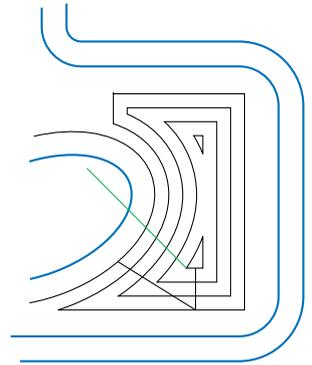
- **Outside**

This option allows you to work in a pocket area starting from the outside border of the pocket and cutting towards the middle of the pocket.

Corner

- **None**

The tool path is formed with sharp corners between one direction and the next.



- **Fillet**

This option connects each direction with a given radius allowing for a smoother transition between path directions.

The **Min. corner radius** option enables you to define the minimal value of the corner radius.

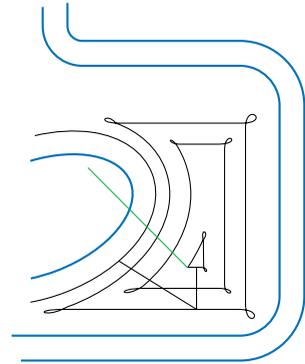


If the given radius is too large for a specific corner, it produces the largest possible radius at that point.

Sometimes the fillet option can leave some material. This particularly happens if the given radius is large.

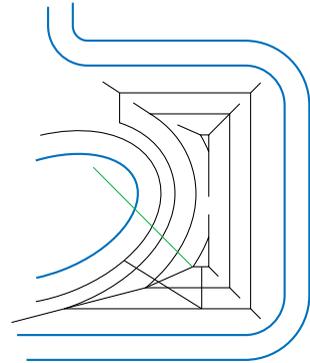
- **Loop**

This option ensures that the connecting points on the path are cleaned using a smooth transition. The tool path forms a loop in the corner, preventing an abrupt change of direction.



- **Sharp**

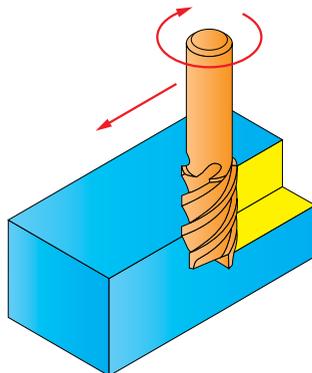
This option ensures that using a short and simple movement cleans the connecting points on the path. Although this produces a sharp movement by the tool, the path itself is slightly shorter than the smooth corner option. This can help cut down on machining time.



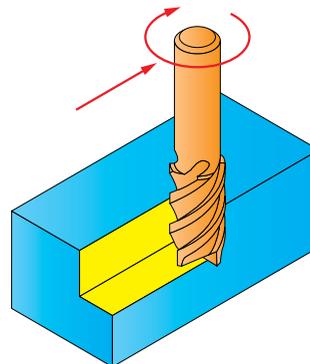
This option is *not* recommended for high-speed cutting.

Direction

This option enables you to choose climb or conventional milling for the roughing operation.



Climb milling



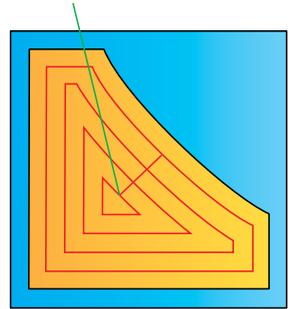
Conventional milling

Adjacent passes connection

This option enables you to choose the method of the tool movement within the pocket from one tool path pass to the next one.

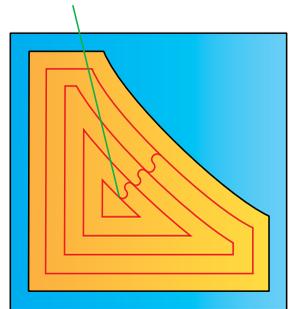
- **Linear**

The tool performs normal approach from one tool path pass to the next one.



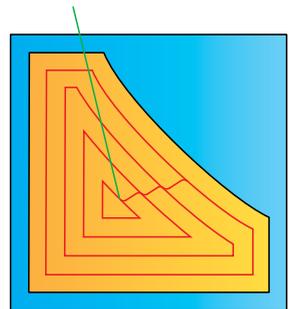
- **Rounded**

The tool performs an arc movement from one tool path pass to the next one. The connection points are located on the line, which is normal to the tool path.



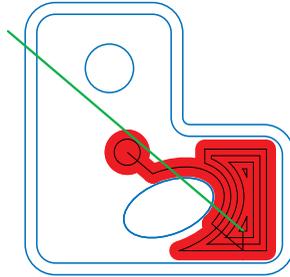
- **Smooth**

The tool performs an arc movement from one tool path pass to the next one. The connection points may not lie on one line forming an angle with the line normal to the tool path.



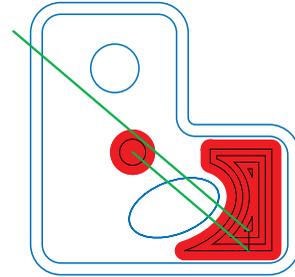
Exit material

This option controls the tool movements between the working areas.



Exit material check box not selected

When the tool moves from one working area to the next, it moves through the full material around the island to get to the next working area as shown above.

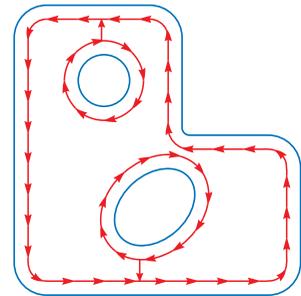


Exit material check box selected

The tool exits the material and travels rapidly above the material to the next working area as shown above. The lead in path is the Lead in you define.

Connect islands

This option enables you to keep the same cutting direction (conventional or climb milling) throughout the entire tool path where possible. This is particularly important in high-speed cutting.

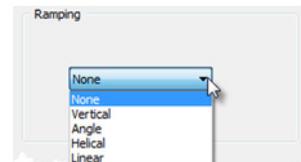


This operation will be performed with the default Contour parameters.



23. Define the Ramping strategy

Switch to the **Link** page of the **Pocket Operation** dialog box. Define the strategy with which the tool is plunging into the material during the pocket roughing.





The following **Ramping** strategies are available for rough machining of pockets:

- **None**

The tool enters the material vertically at the pocket start point chosen automatically by the SolidCAM pocket algorithm.

- **Vertical**

The tool enters the material vertically at a user-defined position. From this position, the tool moves to the pocket start point calculated by the pocket algorithm. Click the **Data** button to specify the position where the tool plunges into the material.

- **Angle**

The tool moves to the pocket start point at a specified ramp angle. The start point must be selected using the **Data** button. Enter the ramping angle value into the **Angle** edit box of the **Angle ramping** dialog box.

SolidCAM does not check the ramping movement against the pocket contour. Check the tool path simulation to make sure that the tool does not gouge the pocket walls or islands.

- **Helical**

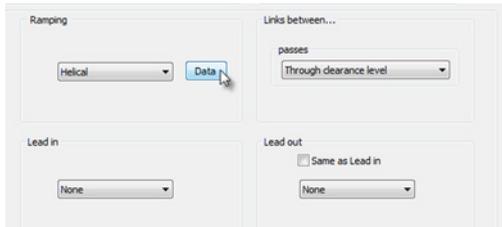
The tool descends from the safety distance above the **Upper level** to the material in a circular motion until the step down is reached using the user-defined radius. When the tool reaches the step down depth, it machines all the material at the step down depth. Click the **Data** button to set the helical ramping parameters.

- **Linear**

The **Linear** ramping follows the same rules as the **Helical** ramping. The difference is that the descent is performed in a linear zigzag fashion rather than in a circular one.

Choose the **Helical** option and click the **Data** button.

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.



Helical Ramping Parameters

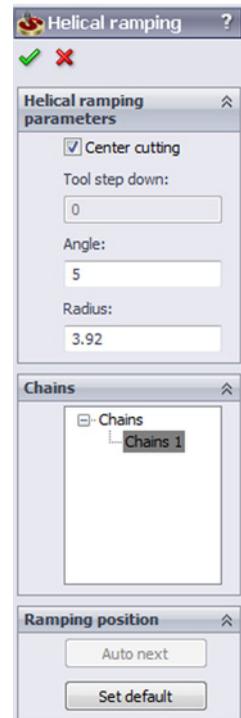
The **Tool step down** parameter defines the distance between each two adjacent turns of the tool helical movement.

The **Angle** parameter defines the ramping angle.

The **Radius** parameter defines the radius of the descending helix.

Center cutting

If your tool has center cutting capabilities, select the **Center cutting** check box. In the **Angle** field, enter the descent angle that you would like the tool to follow. In the **Radius** field, enter the radius of the tool path helix.



The working order is as follows:

- The tool descends from the safety distance above the Upper level to the material in a circular motion until the step down is reached using the user-defined radius.
- When the tool reaches the step down depth, it machines all the material at the step down depth.

- When the machining is completed at this depth, the tool goes up to the Clearance level.
- At this stage, the tool repositions itself at the Z safety distance above the previous step down depth and repeats the Helical Lead in to the next working depth.
- This process repeats itself until the final depth has been machined.

If the tool does not have center cutting capabilities, do not select the **Center cutting** check box. In the **Tool step down** field, enter the depth of the step down of the tool. The result of this action is the same as with a center cutting tool except for the following differences:

- The tool descends from the safety distance above the Upper level to the material in a circular motion up to the Tool Step down.
- At this point, the tool clears a circular path. Then it descends to the next Tool step down.
- This is repeated until the tool reaches the CAM-Part step down depth.
- When the tool reaches the step down depth, it machines all the material on that step down depth.
- When the machining is completed at this depth, the tool goes up to the Clearance level.
- At this stage, the tool repositions itself at the Z safety distance above the previous step down depth and repeat the Helical Lead in to the next working depth.
- This process repeats itself until the final depth has been machined.

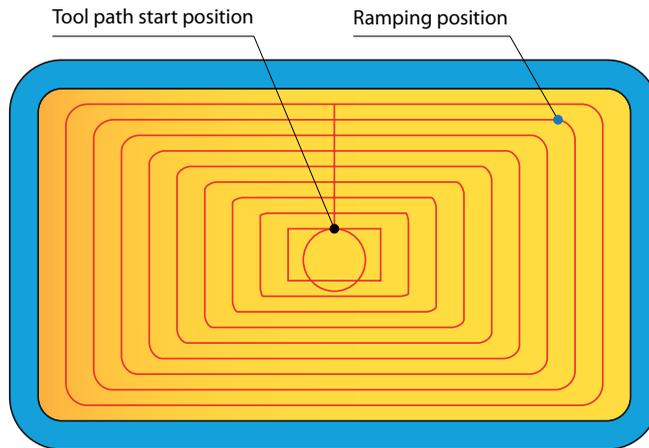
Chains

This section displays the list of all geometry chains defined for the operation. All the chain entries are displayed under the **Chains** header. You can select chain entries in the list.

When the **Chains** header is selected, SolidCAM displays the tool path and default ramping positions for all of the chains. The circles represent the default helical ramping movement defined for each chain.

When a chain entry is selected, SolidCAM displays the tool path and the default ramping position for this chain. This position is automatically defined at the start position of the tool path segment relevant for the current chain. You can change this position by picking a point on the model or by entering the new position coordinates into the **X, Y, Z** dialog box. The schematic circle facilitates the definition of the position.

When the position is picked, it is marked on the model with a blue dot. The coordinates of the picked point are displayed in the **X, Y, Z** dialog box. The circle of the tool path color represents the helical movement of the tool plunging.



When the ramping position is defined, the tool descends into the material at the specified ramping position with helical movements according to the defined parameters. When it reaches the level of the first cutting pass, it moves to the start position of the tool path and performs machining of the pocket.

Ramping Position

The **Set default** button enables you to replace the currently defined ramping position for the selected chain by the default ramping position located at the tool path start position.

The **Auto next** button provides you with the selection mode that enables you to define the ramping positions for all of the chains one by one.

Confirm the dialog box with .



24. Define the Lead in and Lead out

The **Lead in** and **Lead out** strategies are used for defining the horizontal approach/retreat of the tool to/from the geometry in the finish machining of the part walls.



Pocket Lead in and Lead out

The following **Lead in** and **Lead out** strategies are available for pocket machining:

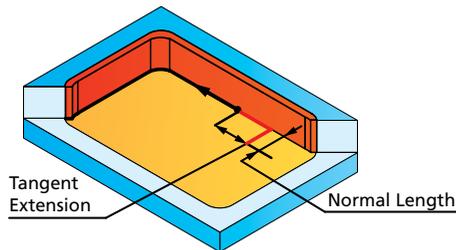
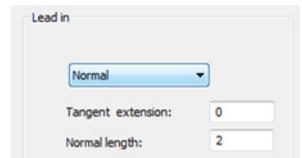
- **None**

The tool leads in to and out from the milling level exactly adjacent to the start point of the geometry.



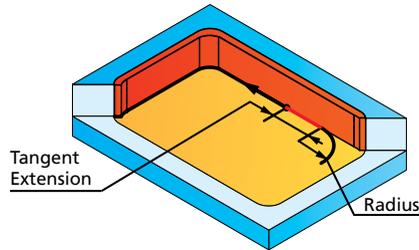
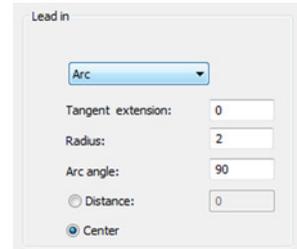
- **Normal**

The tool leads in and out at the last point of the geometry with a movement normal to the pocket contour. The length of the normal can be set in the **Normal length** field. The distance between the normal and start of the geometry is set in the **Tangent extension** field.



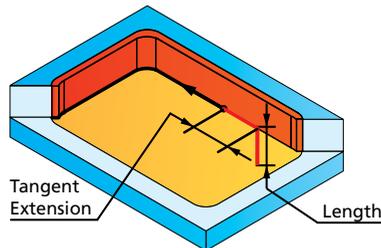
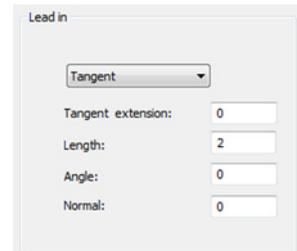
- **Arc**

The tool leads in to and out from the geometry with a circular motion, tangent to the last entity of the pocket contour. The arc radius can be set in the **Radius** field. The length of the extension can be set in the **Tangent extension** field. The arc angle is set in the **Arc angle** field. The arc angle is set in the **Arc angle** field.



- **Tangent**

The tool leads in to and out from the pocket wall in a movement tangent to the last profile entity. The length of the tangent can be set in the **Length** field. The distance to the material can be set in the **Tangent extension** field.

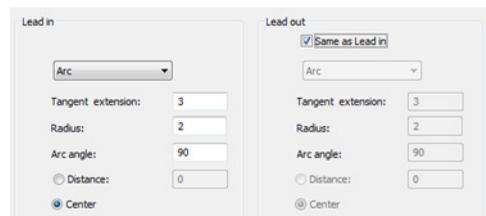


When the **Same as Lead in** check box is selected, the strategy and parameters defined for **Lead in** are used for **Lead out**.



Under **Lead in**, choose the **Arc** option from the list and set the **Tangent extension** value to **3** and the **Radius** value to **2**.

Under **Lead out**, select the **Same as Lead in** check box.





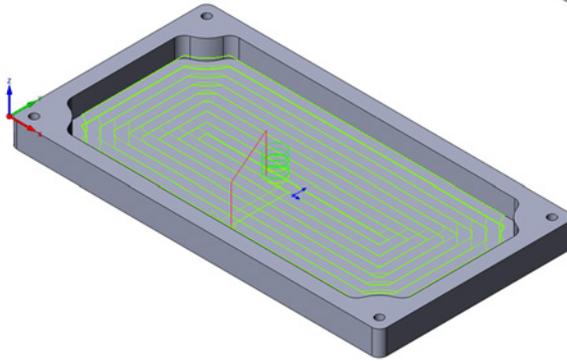
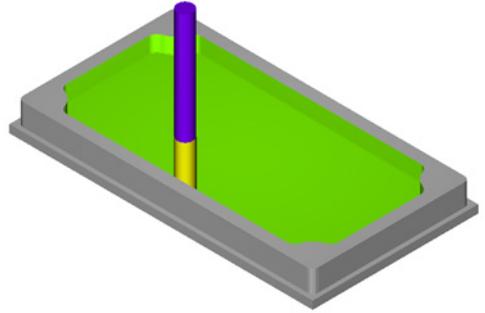
25. Calculate the Tool path

Click the **Save & Calculate** button. The Pocket operation data is saved and the tool path is calculated.



26. Simulate the operation

Simulate the operation in the **SolidVerify** and **Host CAD** simulation modes.

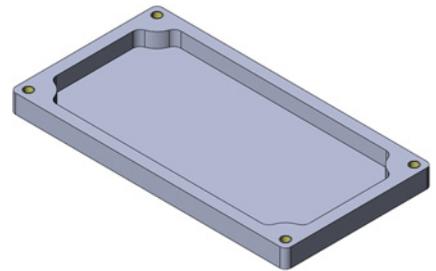


27. Add a Drilling operation

This Drilling operation is used to perform the preliminary center drilling of the four holes in the corners of the model.

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.





28. Define the Drill geometry

In the **Geometry** area, click the  button.

The **Drill Geometry Selection** dialog box is displayed in the SolidWorks **PropertyManager** area. This dialog box enables you to select the geometry for drilling directly on the solid model.



SolidCAM enables you to select the drill centers using the following options:

- **Pick position**

You can define drill centers one by one directly on the solid model.

- **3 Points on circumference**

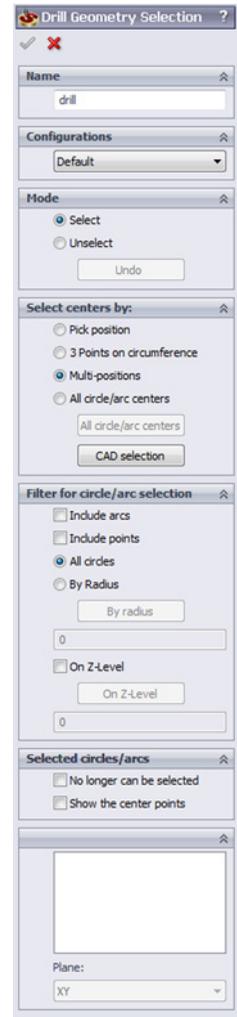
Usually, all curves and arcs of imported models are converted into splines by the exporting CAD system. Due to the nature of spline curves or surface boundaries, you cannot pick a center position like you could on a circle or an arc. SolidCAM calculates the center position of an arc defined by three points positioned on the spline edges. This facilitates selecting drill centers on spline surfaces.

- **Multi-positions**

You can select the model face. SolidCAM automatically recognizes all arcs/circles located on the selected face and selects the center points as drill positions.

- **All circle/arc centers**

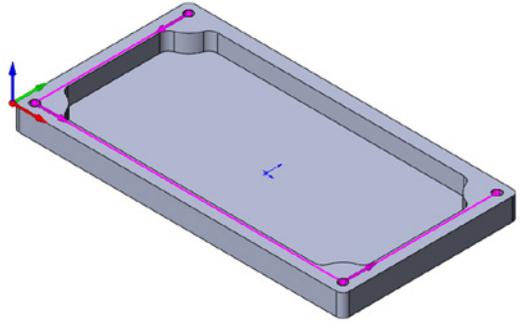
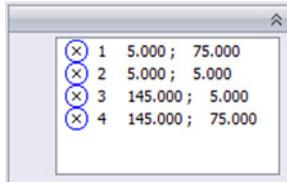
SolidCAM searches the solid model for arcs and circles and adds all center points as drill positions to the geometry.





Choose the **All circle/arc centers** option and click the corresponding button.

Four drill positions are selected. Their coordinates are displayed in bottom part of the **Drill Geometry Selection** dialog box.



Click to confirm the geometry selection. The **Drilling Operation** dialog box is displayed.

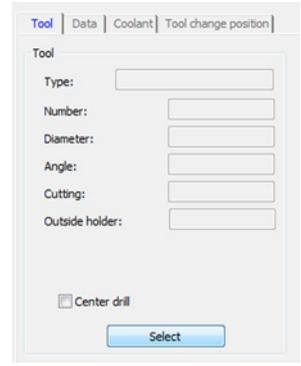


29. Define the Tool

Switch to the **Tool** page of the **Drilling Operation** dialog box and click the **Select** button.

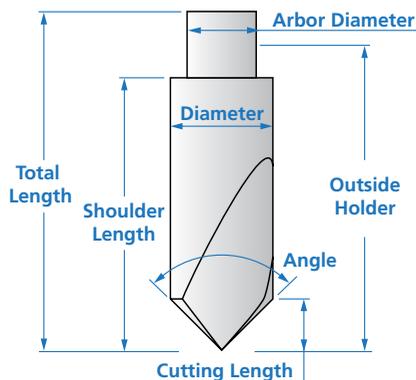
The **Part Tool Table** is displayed.

Click to start a new drilling tool definition. From the **Drilling Tools** section, choose the **Spot drill** tool for the operation.



Spot Drill

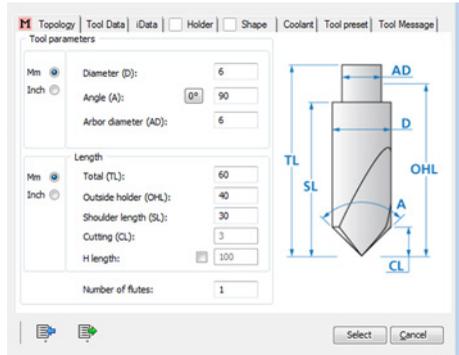
This tool type is used for center drilling and chamfering in Drilling operations. A tool of this type is defined with the parameters shown in the image.





A **Spot drill** of $\varnothing 6$ is used in this operation.

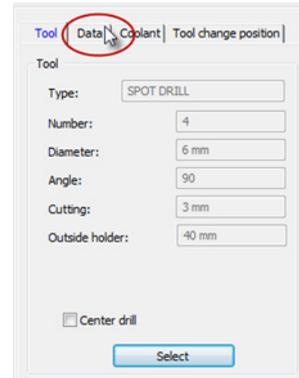
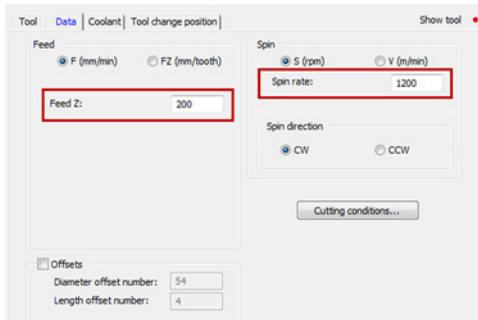
Click the **Select** button to choose the tool for the operation.



The **Drilling Operation** dialog box is displayed. Click the **Data** tab.

Define the spin and the feed for the operation.

- Set the **Spin rate** to **1200**
- Set the **Feed Z** to **200**



30. Define the center drilling depth

Switch to the **Levels** page of the **Drilling Operation** dialog box. Click the **Drill depth** button and select the upper face of the model. The **Drill depth** value (0) appears in the relevant edit box.

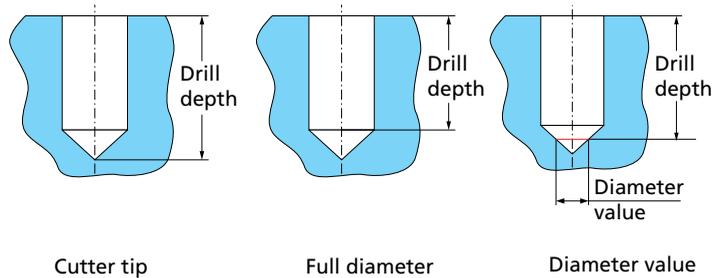
To perform the drilling down to the specified diameter of the tool, use the **Depth type** option.





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**

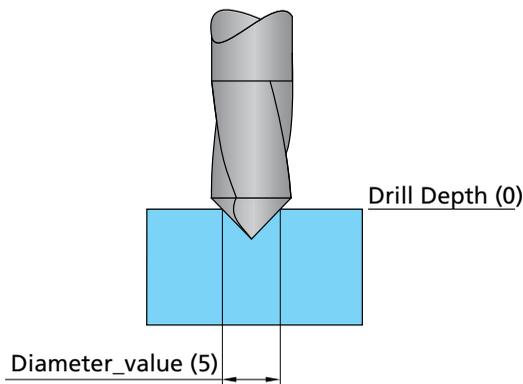
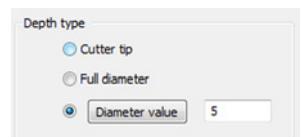
The drill reaches the defined drill depth with the drill cone diameter specified in the edit box.

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.



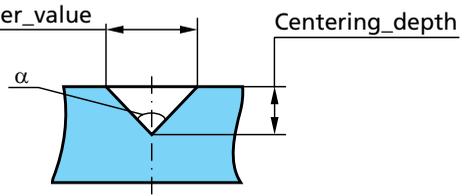
Choose the **Diameter value** option and set the value to **5**.

In this manner, the drilling is performed till the tool diameter of 5 mm is reached at the depth of 0.



In accordance with these parameters, SolidCAM calculates the centering depth using the following formula:

$$\text{Centering_depth} = \frac{\text{Diameter_value} / 2}{\tan(\alpha/2)}$$



If **Diameter_value** is 5 mm, and α is 90° , the centering depth is calculated as follows:

$$\text{Centering_depth} = \frac{5/2}{\tan(90/2)} = 2.5$$



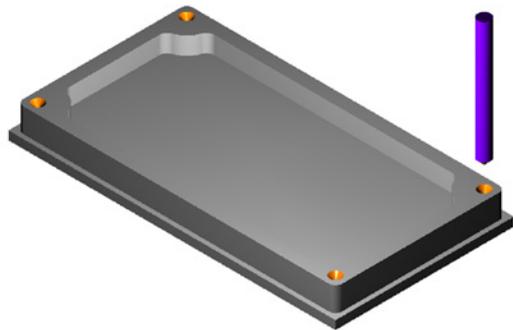
31. Calculate the tool path

Click the **Save & Calculate** button. The Drilling operation data is saved and the tool path is calculated.



32. Simulate the operation

Simulate the operation in the **SolidVerify** simulation mode.



33. Add a Drilling operation

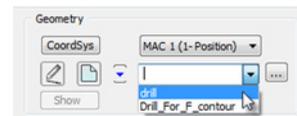
Add another Drilling operation to perform the through drilling of the holes.

Right-click the last Drilling operation and choose **Drilling** from the **Add Milling Operation** submenu. The **Drilling Operation** dialog box is displayed.



34. Define the Geometry

This operation is using the geometry that was defined in the previous center drilling operation. Choose the **Drill** geometry from the list in the **Geometry** area.

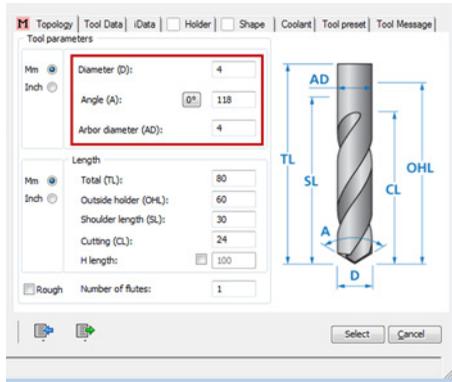


Each geometry defined in SolidCAM has a unique name. When the geometry is being defined, it is assigned a default name that can be changed. Using this name, you can choose the geometry for a specific operation.



35. Define the Tool

Define a new **Drill** tool of $\varnothing 4$ for this operation.



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

The **Drilling Operation** dialog box is displayed. Click the **Data** tab in the **Tool** page.

Define the spin and feed for the operation.

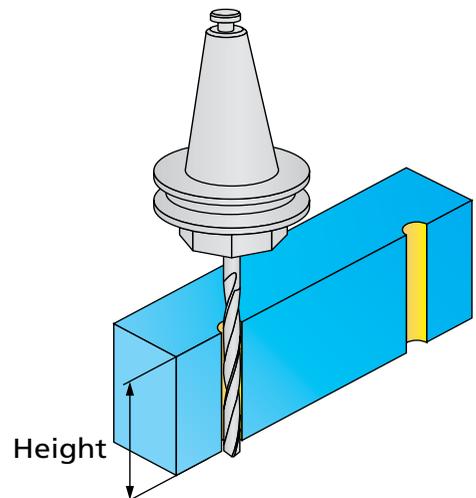
- Set the **Spin rate** to **3000**
- Set the **Feed Z** to **360**



36. Define the Drilling depth

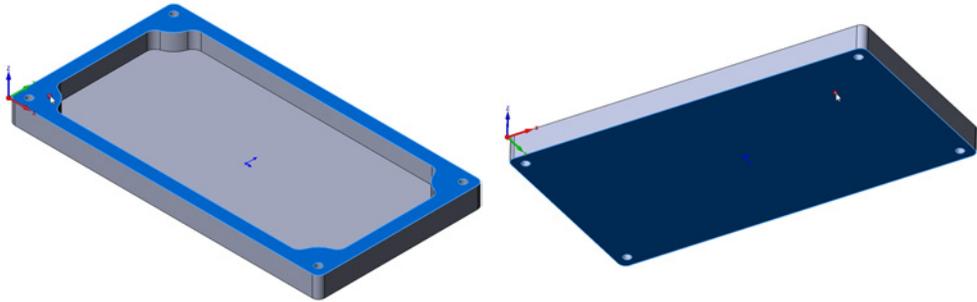
The overall height of the model is 10 mm plus the 5 mm bottom offset defined for the stock. The drilling has to be performed deeper than this depth in order to enable the tool to exit from the material and perform the through drilling.

Switch to the **Levels** page.



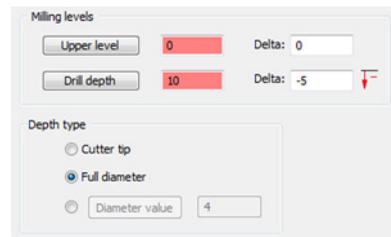
Define the **Upper level** by clicking on the top face of the model as shown.

Define the **Drill depth**. Rotate the model and select the bottom face as shown.



Since the **Z**-offset defined for the stock model is 5 mm, set the **Delta** value to **-5**.

To perform the through drilling, choose the **Full diameter** option in the **Depth type** area. With this option, the drilling is performed until the full diameter is reached at the specified drill depth. This means that the conical part of the tool exits from the material.

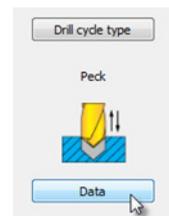
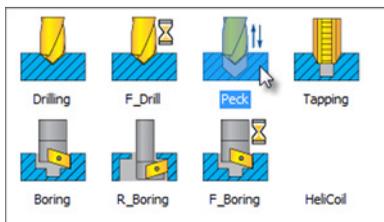


37. Define the Drilling type

SolidCAM enables you to use a number of drill canned cycles supported by your CNC-controller. In this operation, the pecking canned cycle is used for chip breaking. With this cycle, the chip breaking is accomplished by slight retracts of the tool during the drilling process.

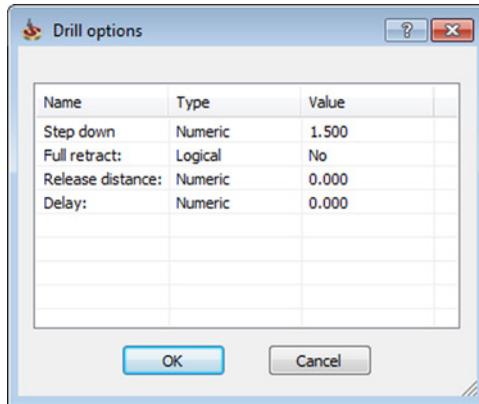
Switch to the **Technology** page and click the **Drill cycle type** button. Available drill cycles are displayed.

Click the **Peck** button. The cycle is chosen for the operation.



Click the **Data** button to define the pecking parameters. The **Drill Options** dialog box is displayed.

Set the **Step down** to **1.5** in order to define the depth of each pecking movement. Confirm the data with the **OK** button.



38. Calculate the tool path

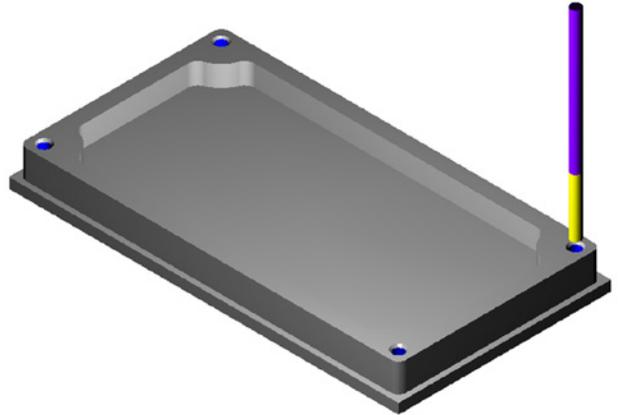
Click the **Save & Calculate** button.

The Drilling operation data is saved, and the tool path is calculated.



39. Simulate the operation

Simulate the operation in the **SolidVerify** simulation mode.

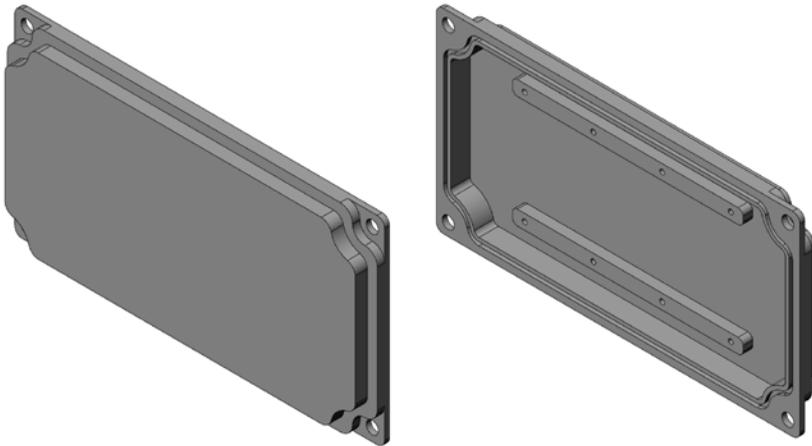


Since in the previous operation the drilling diameter was greater than that in this operation, the drilling results in a chamfer on the drilled holes.

Now you have successfully finished the exercise.

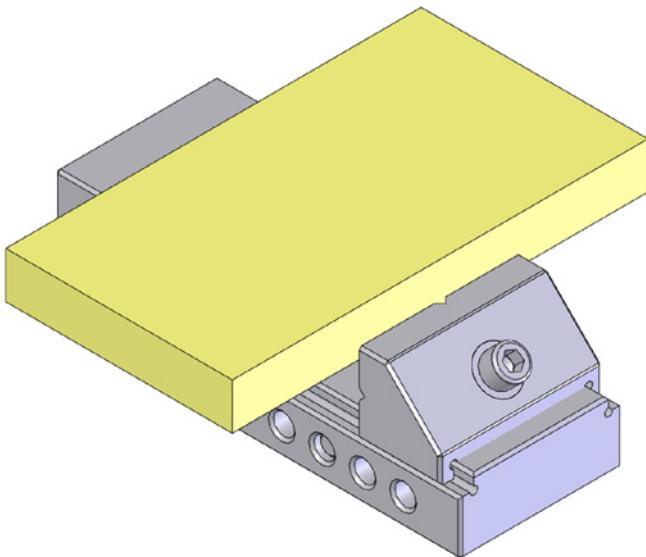
Exercise #3: Cover Machining

In this exercise, a number of SolidCAM 2.5D operations are used for the cover part machining.

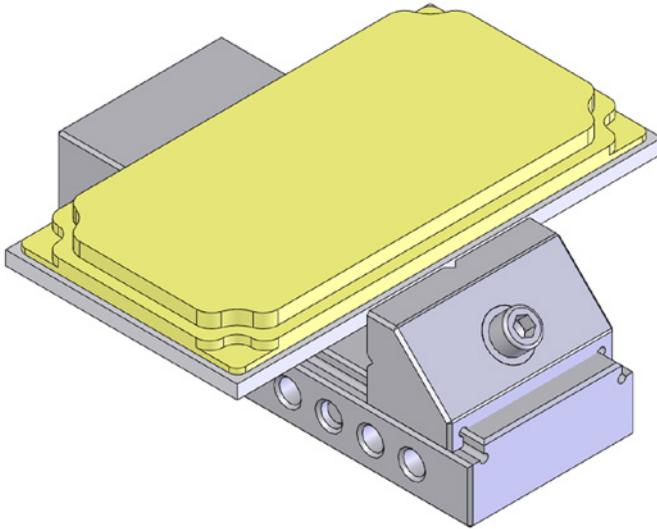


The cover is machined on the 3-Axis milling CNC-machine using the machining vice. The part is machined using two setups.

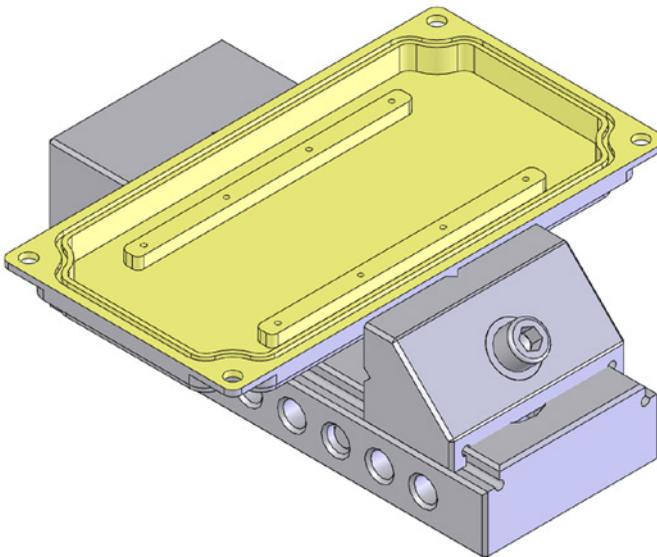
At the first stage, the workpiece is positioned in the vice as shown below.



The highlighted faces are machined.



At the next stage, the rest of the cover faces are machined using the second positioning.

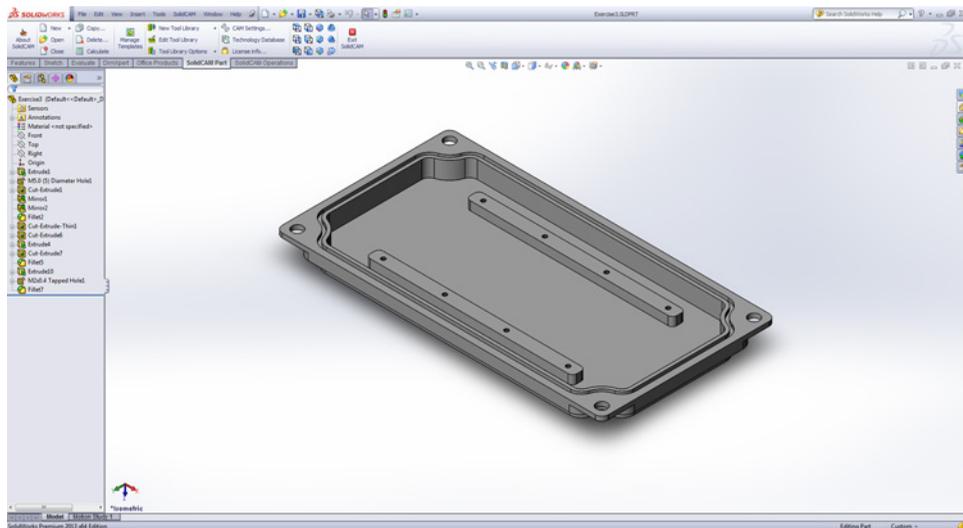




1. Load the SolidWorks model

Load the **Exercise3.sldprt** model that is located in the **Exercises** folder.

This model contains a number of features forming the solid body of the cover.



2. Start SolidCAM

To activate SolidCAM, click the **SolidCAM** field in the main menu of SolidWorks and choose **Milling** from the **New** submenu or click **New > Milling** on the **SolidCAM Part** toolbar.

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



3. Confirm the CAM-Part creation

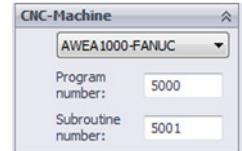
Choose the **External** mode of CAM-Part creation. Confirm the default **Directory**, CAM-Part **Name** and **Model Name** by clicking the **OK** button in the **New Milling Part** dialog box. The CAM-Part is defined.



4. Choose the CNC-controller

When the CAM-Part is defined, the **Milling Part Data** dialog box is displayed.

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



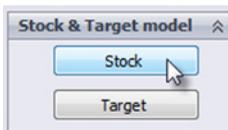
In this exercise, you use a 3-Axis CNC-machine with the **AWEA1000-Fanuc** CNC-controller. Choose the **AWEA1000-Fanuc** CNC-controller from the list.



5. Define the Stock model

In this exercise, you have to define the Stock model before you define the Coordinate System in order to use the workpiece for the CoordSys definition.

Click the **Stock** button in the **Stock & Target model** section to start the definition process.



The **Model** dialog box is displayed.

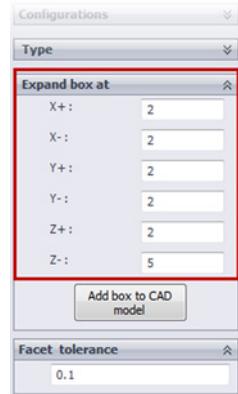
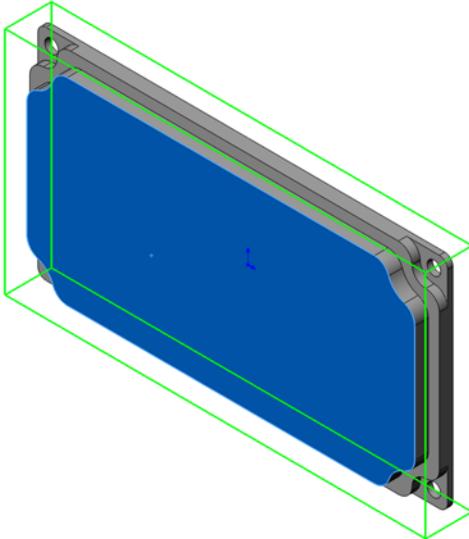


When the Coordinate System is not defined, SolidCAM enables you to use only the **Box (Auto)** method of the Stock model definition.



SolidCAM generates the stock box surrounding the model with the specified allowances. In the **Expand box at** section, set the value of the **Z-** parameter direction to **5**. This allowance is used for the first clamping. Set the value of **2** for the rest of the directions.

Click on the model. The face is highlighted, and the box surrounding the model is displayed.



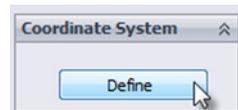
Click the **Add box to CAD model** button. The stock model box is added into the **CAM** component of the CAM-Part assembly as a **3D Sketch** feature.

Confirm the **Model** dialog box with . The **Milling Part Data** dialog box is displayed.



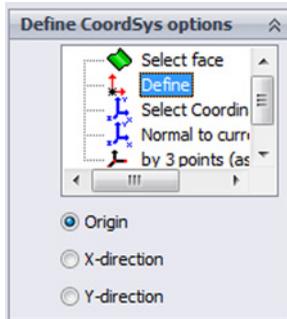
6. Define the Coordinate System

Click the **Define** button in the **Coordinate System** area to start the Coordinate System definition.



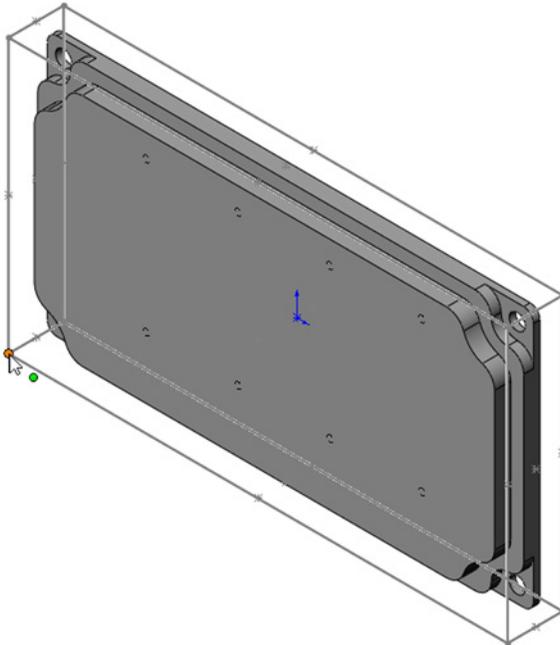
The **CoordSys** dialog box is displayed.

In the **Define CoordSys options** list, choose the **Define** option.

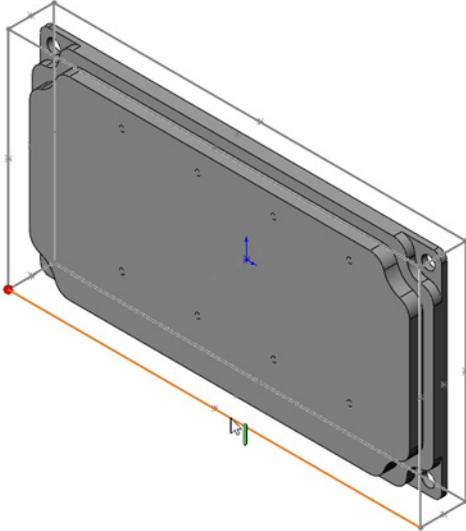


This mode enables you to define the Coordinate System by picking three points on the solid model. At first, you have to define the Coordinate System origin location and then the points for the X- and Y-directions.

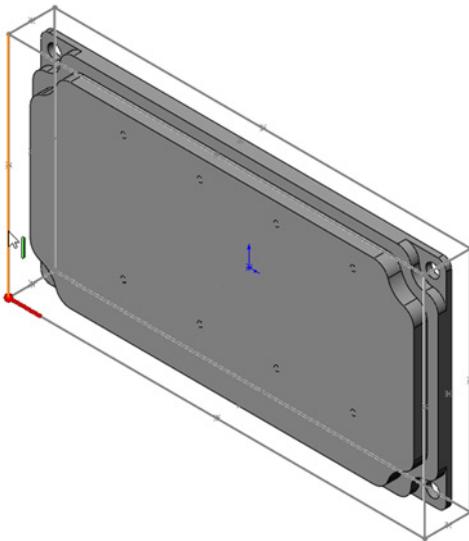
Pick the origin point in the stock box corner as shown.



Click on the stock model edge as shown to define the X-axis of the Coordinate System.

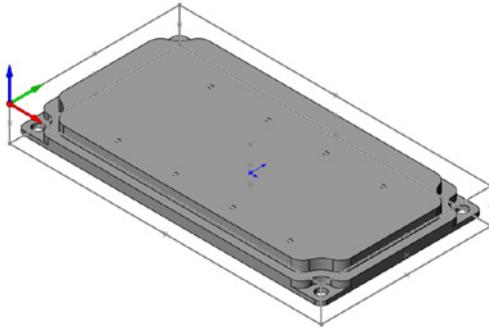


Click on the stock model edge as shown to define the Y-axis of the Coordinate System.

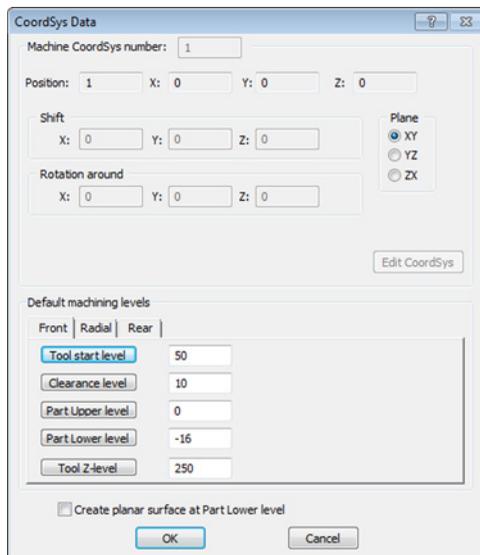


When a point is selected, the next button is automatically activated. If you miss the selection, you can at any time select the button you want to define and continue automatically to the next button.

Confirm the Coordinate System definition with . The model is rotated, and the Coordinate System is displayed.



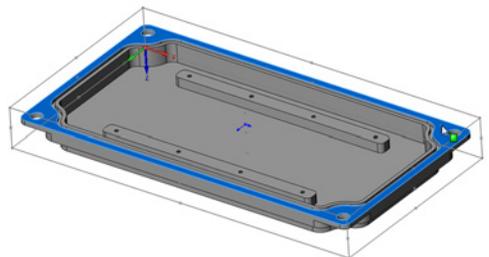
The **CoordSys Data** dialog box is displayed.



Define the Part Lower level directly on the solid model. This parameter defines the lower surface level of the part to be milled.

Click the **Part Lower level** button.

Rotate the model and select the lower face that is milled using the first positioning as shown. The Z-coordinate of the face (-12) is displayed in the **Pick Part Lower level** dialog box.



Confirm this dialog box by clicking .

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

The icon of the defined Coordinate System is displayed in the **CoordSys Manager** dialog box.

Confirm the **CoordSys Manager** dialog box with . The **Milling Part Data** dialog box is displayed again.



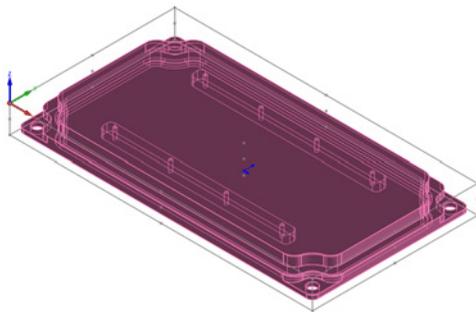
7. Define the Target model

Click the **Target** button in the **Stock & Target** section of the **Milling Part Data** dialog box.

The target **Model** dialog box is displayed.

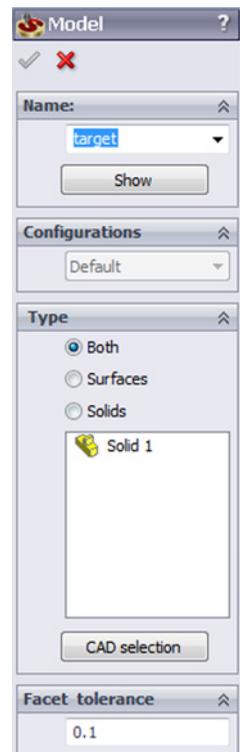
This dialog box enables you to define a 3D model for the Target.

The default target model is chosen. Click twice on the solid body to clear the selection and select a new target model. The solid body is highlighted..



Confirm the **Model** dialog box with .

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



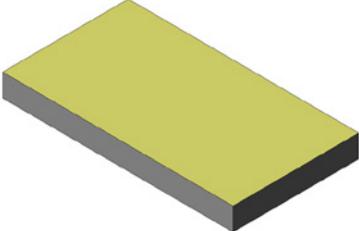
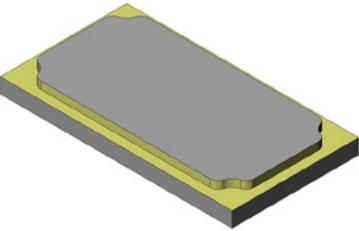
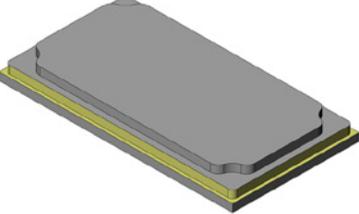
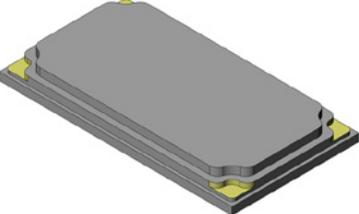
8. Save the CAM-Part data

In the **Milling Part Data** dialog box, click .

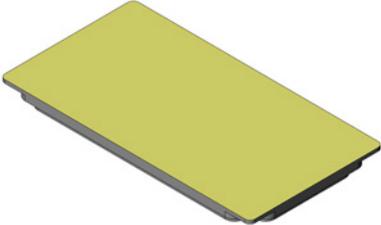
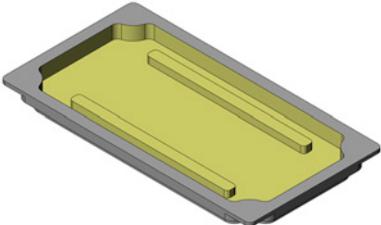
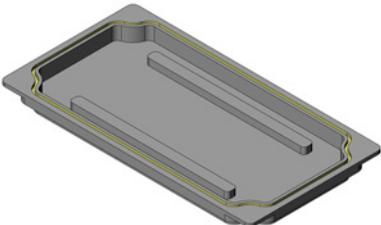
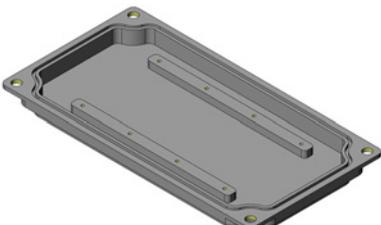
The **Milling Part Data** dialog box is closed, and **SolidCAM Manager** is displayed. The defined CAM-Part is saved.

At this stage, the definition of the CAM-Part is finished.

At the next stage, you have to define several operations to machine the cover part. Using the first defined Coordinate System (first clamping), you have to perform the following operations:

Upper face machining	
Upper profile machining	
Lower profile machining	
Hole pads machining	

Then the part has to be rotated and clamped again. With the second clamping, the following operations are performed:

Upper face machining	
Pocket machining	
Slot machining	
Holes machining	



9. Add a Face Milling operation

In **SolidCAM Operations** toolbar, click **2.5D > Face**. The **Face Milling** operation is used for the upper face machining.

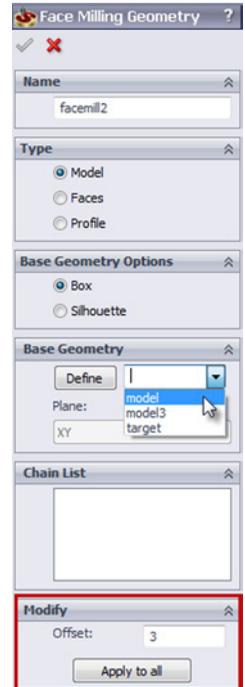
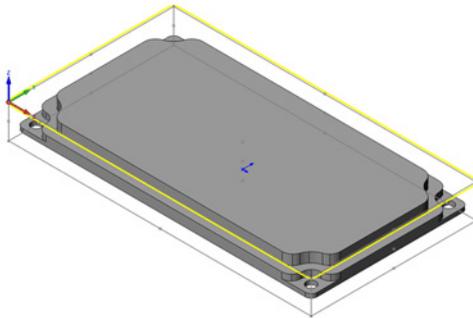
The **Face Milling Operation** dialog box is displayed.



10. Define the Face Milling geometry

Click the  button in the **Geometry** page. The **Face Milling Geometry** dialog box is displayed.

Using the default **Model** option, click the **Define** button and pick on the solid model to select a face.



In the **Face Milling Geometry** dialog box, define the 3 mm offset to machine over the stock edges. In the **Modify** section, set the **Offset** value to **3**.

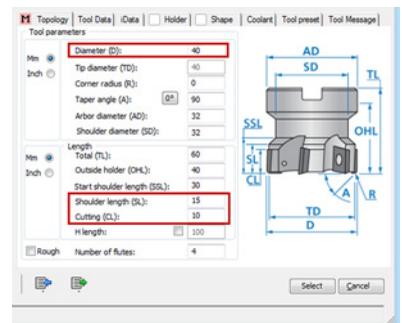
Confirm the **Face Milling Geometry** dialog box by clicking . The geometry is defined for the operation.



11. Define the Tool

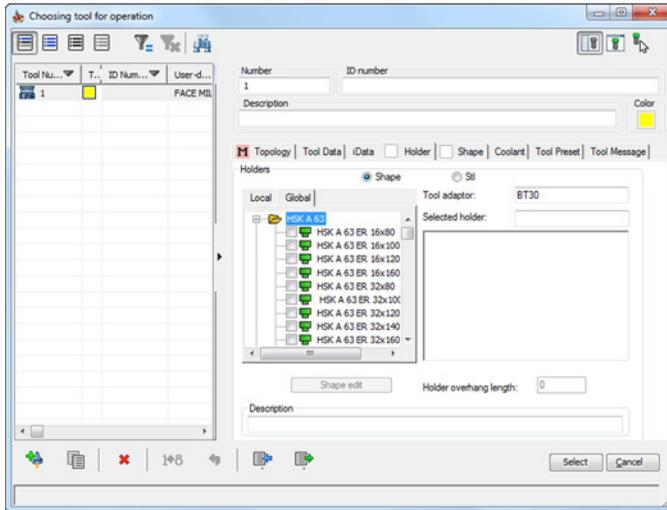
Define a new **Face mill** of $\text{Ø}40$ for this operation and edit the following parameters:

- Set the **Shoulder length** to **15**
- Set the **Cutting length** to **10**



SolidCAM enables you to define a variety of tool holders to help you check and prevent all possible collisions between the tool holding system and the workpiece. This feature also enables you to see a more realistic simulation in the **SolidVerify** simulation.

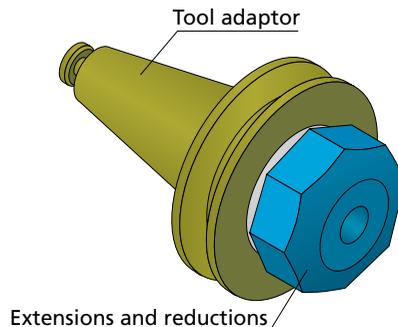
Switch to the **Holder** page in the **Choosing tool for operation** dialog box.



The **Global** holders table is supplied within SolidCAM. This table contains a number of frequently used tool holder components. You can make changes in the **Global** table.

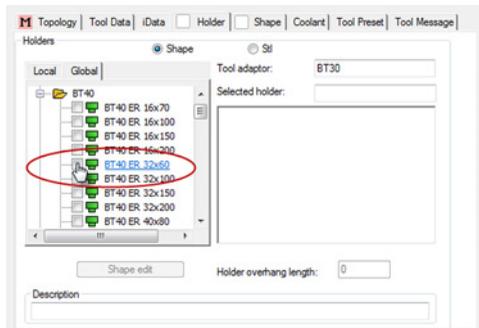


The SolidCAM tool holder is defined by combining two components. The first component is the tool adaptor mounted on the spindle unit of the milling machine. The second component can consist of various types of extensions and reductions like collet chucks, arbors, shanks and other components that you may have.



SolidCAM enables you to define the Tool adaptor in the *.vmid file under **Devices > Spindle > Stations**.

In the **Global** table, choose the **BT40 ER32x60** collet chuck by clicking its check box. This collet chuck is suitable for the chosen tool diameter (40 mm).

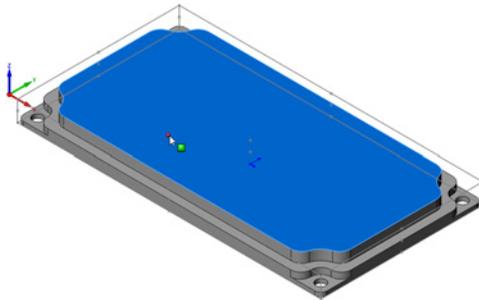


Choose the defined tool for the operation by clicking the **Select** button. The **Face Milling Operation** dialog box is displayed.



12. Define the Face depth

Switch to the **Levels** page of the **Face Milling Operation** dialog box. Click the **Face depth** button in the **Milling levels** area and select the model face as shown.

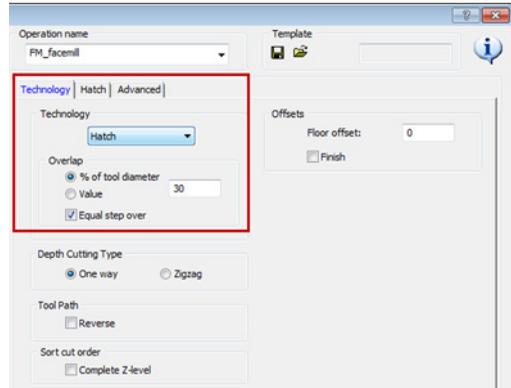


The **Face depth (2)** is calculated.



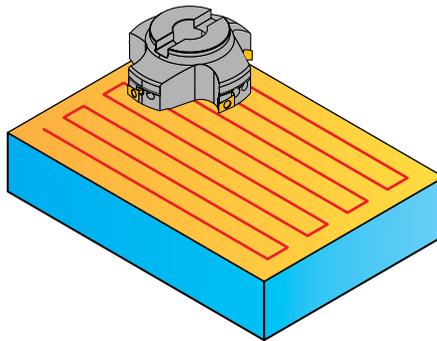
13. Define the technological parameters

Switch to the **Technology** page of the **Face Milling Operation** dialog box. In the **Technology** section, use the default **Hatch** option.

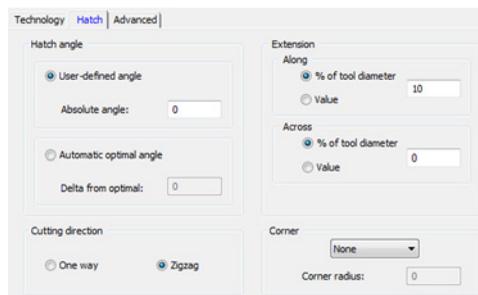


Hatch Machining Technology

The machining is performed in a linear pattern.

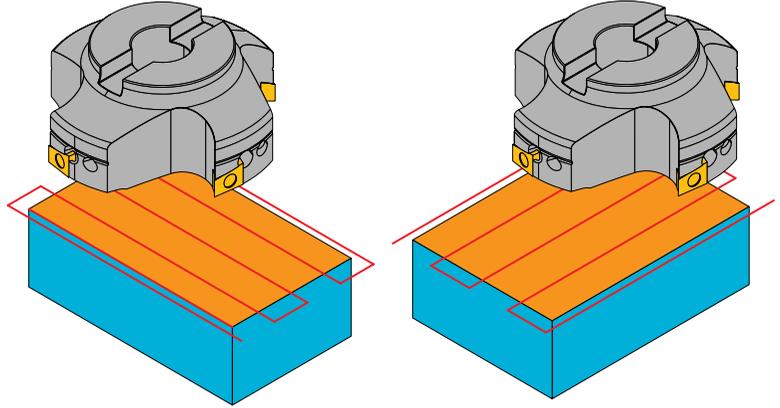


The **Hatch** page enables you to define the hatching parameters.

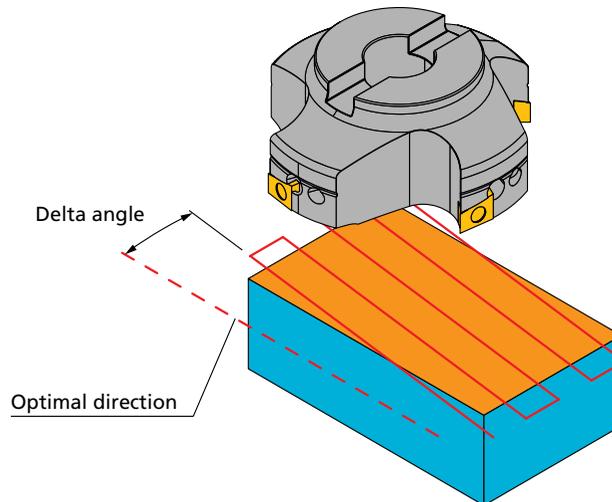


- **Hatch angle**

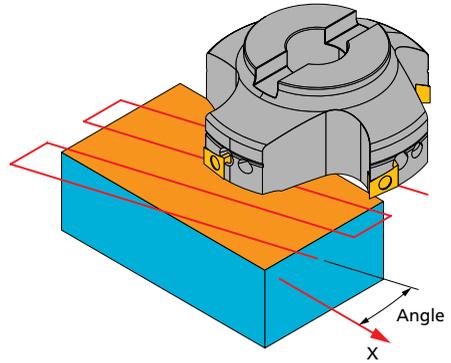
When the **Automatic optional angle** option is chosen, SolidCAM automatically calculates the optimal angle of hatching to expedite the machining. The tool path always follows the length of the face no matter what angle the machined surface is facing.



The **Delta from optimal** parameter enables you to change the hatching angle.



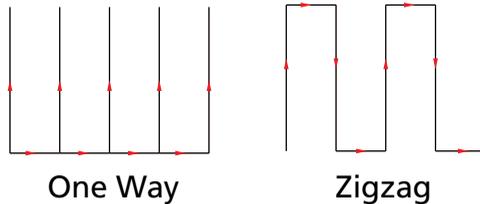
When the **User-defined angle** option is chosen, you can determine the angle of the tool path.



- **Cutting direction**

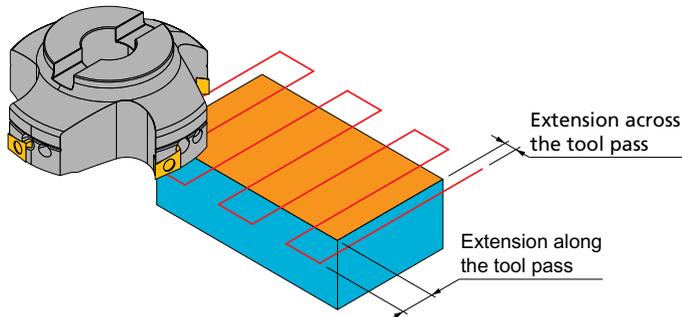
The **One Way** option enables you to create the tool path with only one-directional movements.

The **Zigzag** option enables you to create the tool path with bidirectional movements.



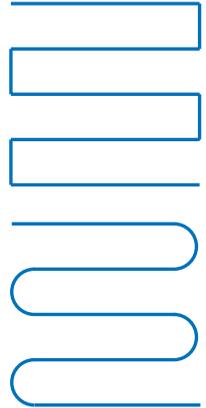
- **Extension**

During face milling, the tool path is extended over the edges of the machined face. The **Extension** section enables you to define the extension both along the tool path (the **Along** section) and across the tool path (the **Across** section). The extension can be defined either by percentage of the tool diameter (the **% of tool diameter** option) or by value (the **Value** option).



- **Corner**

When the **None** option is chosen, the tool path is formed with sharp corners between one direction and the next.



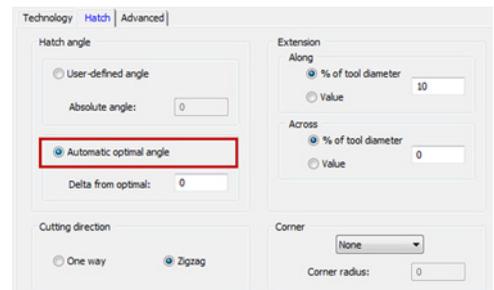
The **Fillet** option connects each direction with a given radius allowing for a smoother transition between path directions.



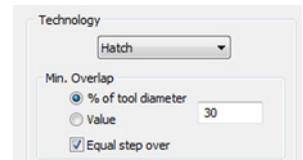
Click the **Data** tab to define the machining parameters.

In the **Hatch angle** section, switch to the **Automatic optimal angle** option.

Switch back to the **Technology** tab.

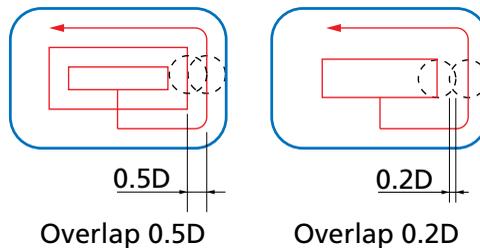


Notice that in the **Overlap** section the **% of tool diameter** option is chosen and the value is set to **30**, and the **Equal step over** check box is selected.



Overlap

This section enables you to define the tool overlapping between two successive passes. This option can be defined as **Percentage of the tool diameter** or as a **Value**.

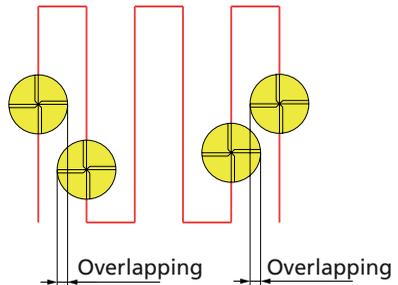
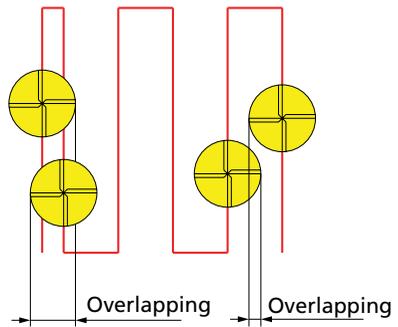


Equal Step over

This option enables you to generate the hatch tool path spaced evenly with the automatically calculated **Overlap** value, which is nearest to the specified **Min. Overlap** value but not smaller than this value.

When this check box is not selected, the distance between the last pass and the one before it can be smaller than that between all of the other passes.

When this check box is selected, the evenly spaced hatch tool path is generated. The overlap between two successive passes is not smaller than the specified **Min. Overlap** value.

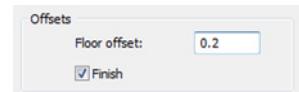


This option is available only for the **Hatch** strategy.



Define the roughing offset that remains on the floor of the face. This offset is left unmachined during the rough face machining and removed during the face finishing.

In the **Offsets** section, set the **Floor offset** value to **0.2**. Select the **Finish** check box to perform finishing of the face in this operation. This check box enables you to remove the remaining offset with the last cutting pass.

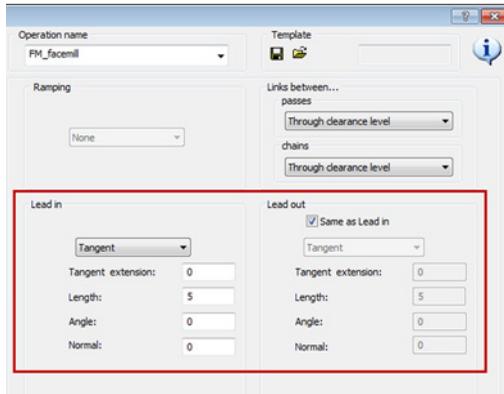


14. Define the Lead in and Lead out

Switch to the **Link** page of the **Face Milling** dialog box to define the way the tool approaches the material and retreats away.

In the **Lead in** section, choose the **Tangent** option. This option enables the tool to approach the material on a line tangent to the profile. In the **Length** field, set the length of the tangent to **5**.

In the **Lead out** section, select the **Same as Lead in** check box.



15. Save and Calculate

Click the **Save & Calculate** button.

The Face Milling operation data is saved, and the tool path is calculated.

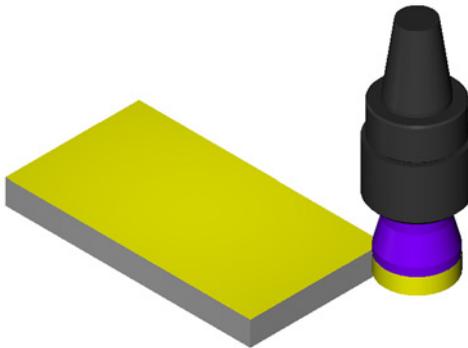


16. Simulate

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

Switch to the **SolidVerify** page and start the simulation with the  button.

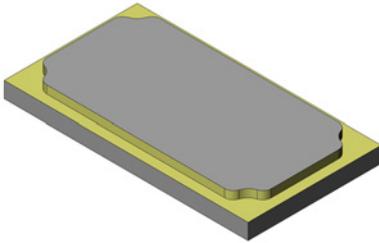
Close the simulation with the  button. The **Face Milling Operation** dialog box is displayed. Close this dialog box with the **Exit** button.





17. Add a Profile operation

At this stage, you have to define a Profile operation in order to machine the upper profile of the cover.



In **SolidCAM Manager**, add a new Profile operation.



18. Define the Geometry

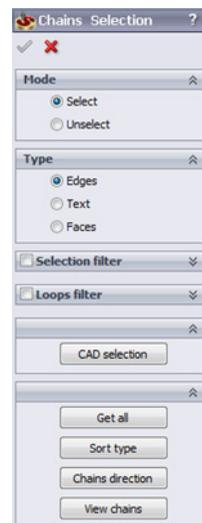
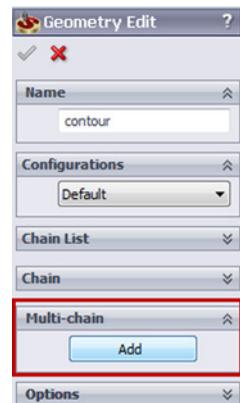
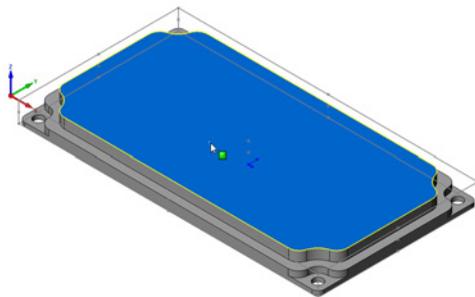
Click the  button in the **Geometry** page of the **Profile Operation** dialog box. The **Geometry Edit** dialog box is displayed.

Click the **Add** button in the **Multi-chain** section.

The **Chains Selection** dialog box is displayed. This dialog box enables you to pick a number of chains from the model by selecting the model elements. SolidCAM automatically creates chains from the selected elements.

Click on the top face of the model as shown.

The face is selected, and its boundary is highlighted.



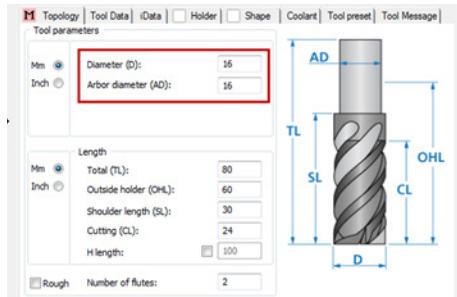
Click  to choose the selected chain as the geometry. The **Geometry Edit** dialog box is displayed.

Confirm the geometry definition with . The geometry is defined for the operation.



19. Define the Tool

Add a new **End mill** tool of **Ø16** for the operation.

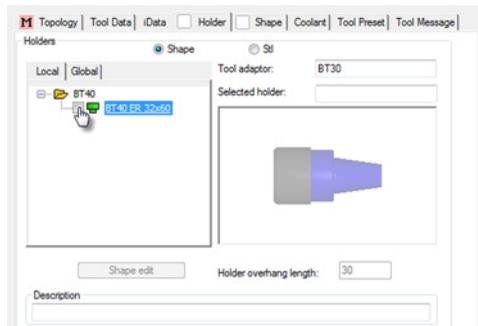


Choose the tool holder. Switch to the **Holder** page and select the **Use holder** check box. Click the **Local** holders tab.



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.

Choose the **BT40 ER32x60** Collet chuck.

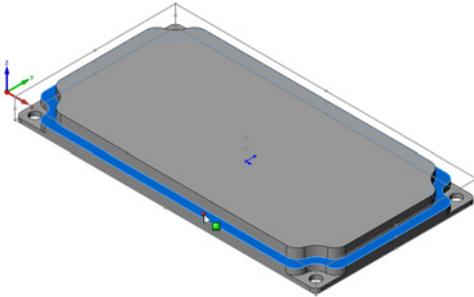


Confirm the tool selection by clicking the **Select** button.



20. Define the Profile depth

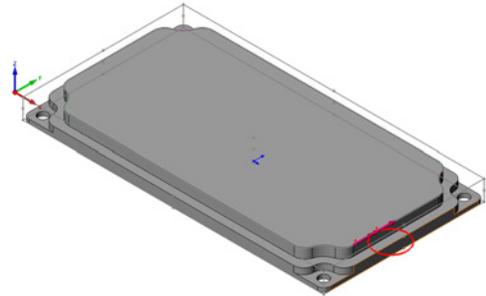
Switch to the **Levels** page. In the same manner as explained in previous steps, define the **Profile depth** by clicking on the model face as shown below.



21. Define the technological parameters

Switch to the **Technology** page. In the **Modify** section, set the **Tool side** to **Right**. Click the **Geometry** button to check the tool position relative to the geometry.

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

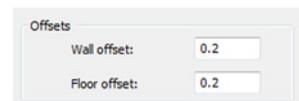


Now you have to define the roughing and finishing parameters. SolidCAM Profile operation enables you to perform the rough and finish machining in the single operation.

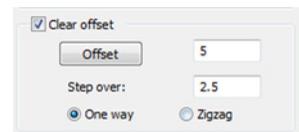
Select the **Rough** check box. Set the **Step down** value to **3.5**. The profile is machined in two equal Z-steps.



In the **Offsets** section, set the **Wall offset** and the **Floor offset** to **0.2**. These allowances are removed during the finish machining.



Select the **Clear offset** check box. Set the **Offset** value to **5** and the **Step over** value to **2.5**.

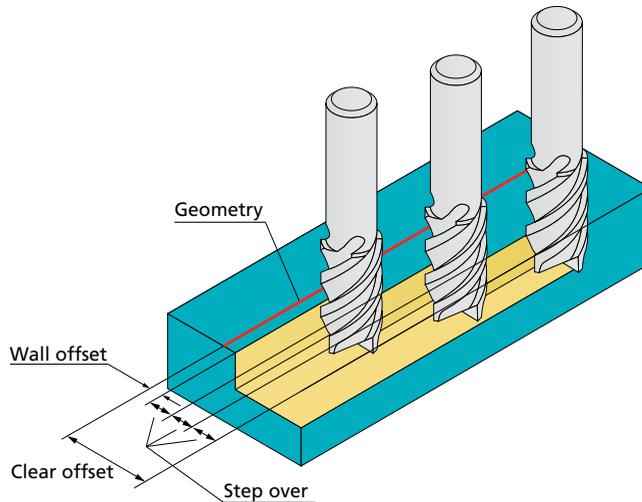




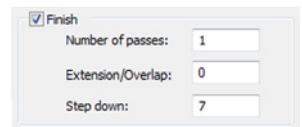
Clear Offset

This option generates several concentric profiles with a constant depth that start from the defined clear offset distance from the profile and finish up to the geometry of the profile, thus clearing the area around the profile.

The **Offset** defines the distance from the geometry at which the milling starts. The **Clear offset** value should be equal to or larger than the **Wall offset** value. The tool starts milling the profile at the distance defined by the **Clear offset** and finishes at the distance defined by the **Wall offset**; the overlap of the adjacent tool paths is defined by the **Step over** parameter. The **Step over** parameter defines the overlap of adjacent tool paths. It determines the offset between two successive concentric profiles.

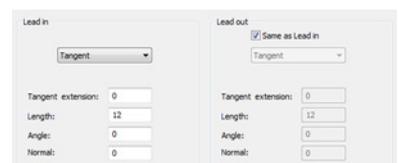


In the **Finish** section, set the **Step down** value to **7**. The 0.2 mm allowances that remain after the roughing are removed in one step.



22. Define the Lead in and Lead out

Switch to the **Link** page. In the **Lead in** section, choose the **Tangent** option. With this option, the tool approaches the material tangentially to the geometry in the start point. Set the **Length** value to **12**.



In the **Lead out** section, select the **Same as Lead in** check box.



23. Save and Calculate

Click the **Save & Calculate** button.

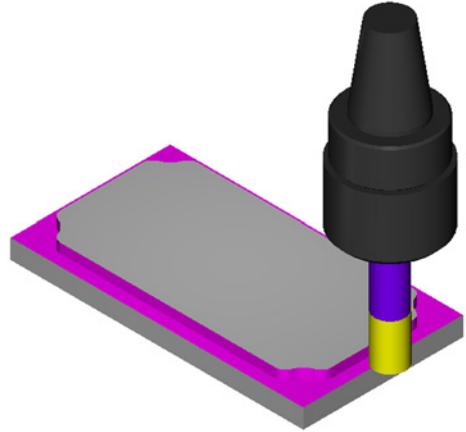
The Profile operation data is saved and the tool path is calculated.



24. Simulate

Click the **Simulate** button in the **Profile Operation** dialog box. The **Simulation** control panel is displayed.

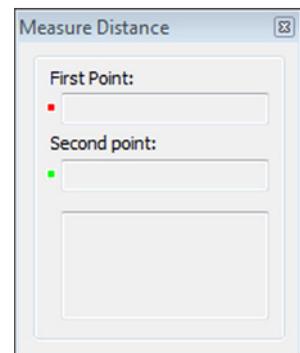
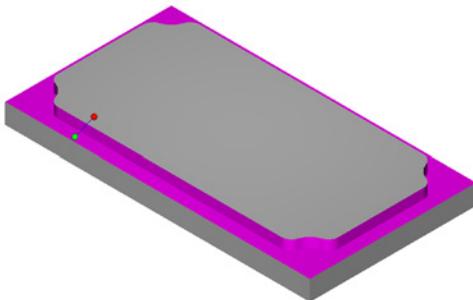
Switch to the **SolidVerify** page and start the simulation with the  button.



The **SolidVerify** simulation mode enables you to measure distances directly on solid bodies in the **SolidVerify** window. This feature enables checking the linear dimensions of the part during simulation.

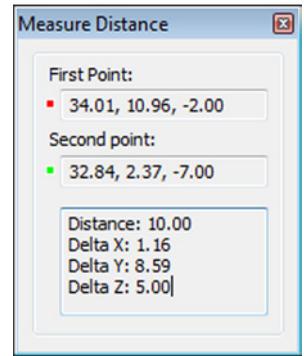
Click the **Measure**  button on the toolbar. The **Measure Distance** dialog box is displayed.

Click on the top face of the cover and then on the horizontal face machined in the current operation.



The coordinates of the pocket points and the result distance are displayed in the **Measure Distance** dialog box.

SolidCAM displays the measured distance value and **Delta** values for the X-, Y- and Z-axes. In this case, the **Delta Z** parameter displays the depth of the machined face relative to the cover top face (5).



Close the simulation with the  button. The **Profile Operation** dialog box is displayed.

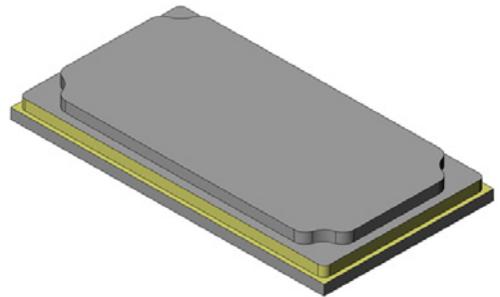
Close the **Profile Operation** dialog box with the **Exit** button.



25. Add a Profile operation

At this stage, you have to define a Profile operation in order to machine the lower profile of the cover.

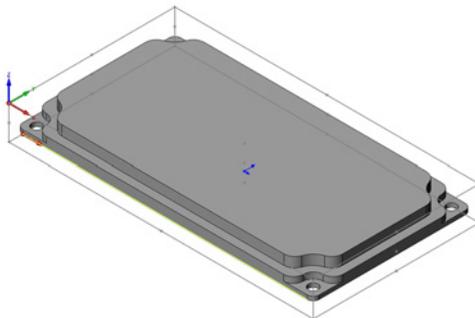
In **SolidCAM Manager**, add a new Profile operation.



26. Define the Geometry

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

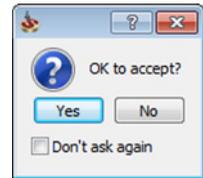
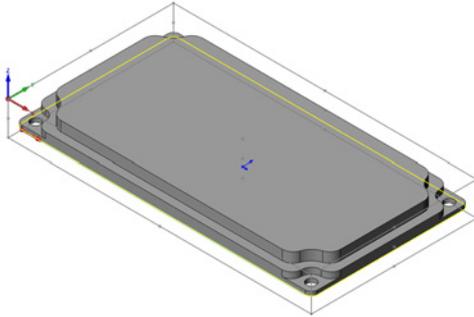
Click on the workpiece edge as shown to define the first entity of the chain.



In the **Chain** section, choose the **Auto-constant Z** option. The closed chain is automatically selected.

The confirmation message is displayed.

Confirm it by clicking **Yes**.



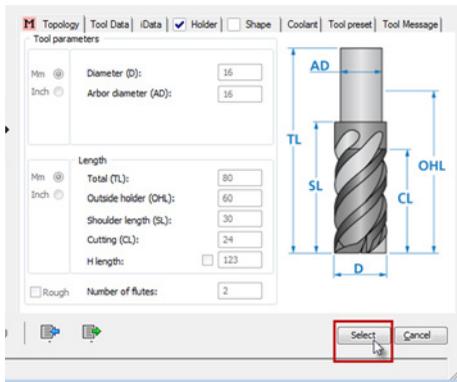
Confirm the geometry with .



27. Define the Tool

In this operation, use the same tool as in the previous operation (**Ø16 End mill**).

Click the **Select** button in the **Tool** page. The **Part Tool Table** is displayed. Choose the previously defined **Tool #2** and click the **Select** button.

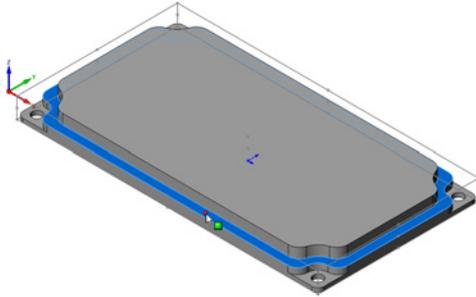




28. Define the Profile depth

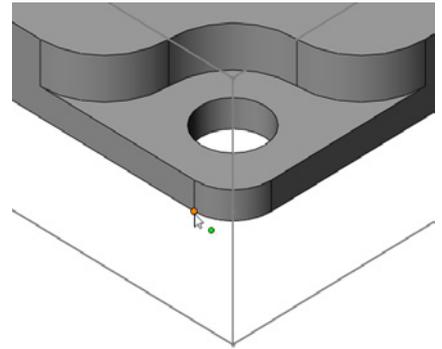
You have to define a new Upper level for the operation taking into account the already machined faces. Switch to the **Levels** page and click the **Upper level** button in the **Milling levels** area.

Define the **Upper level** by clicking on the model face as shown.



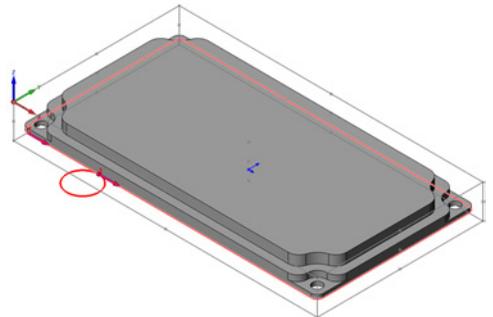
Confirm the definition of the **Upper level** with .

In the same manner as explained in the previous steps, define the **Profile depth** by clicking on the model vertex as shown.



29. Define the technological parameters

Switch to the **Technology** page of the **Profile Operation** dialog box. In the **Modify** section, set the **Tool side** to **Right**. Click the **Geometry** button to check the tool position relative to the geometry.



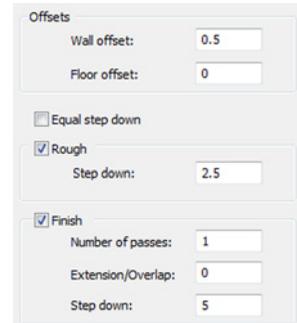
Close the **Modify Geometry** dialog box with the  button.

Now you have to define the parameters of profile roughing and finishing.

Select the **Rough** check box. Set the **Step down** value to **2.5**. The profile is machined in two equal Z-steps.

In the **Offsets** section, set the **Wall offset** value to **0.5**. This allowance is removed during the finish machining.

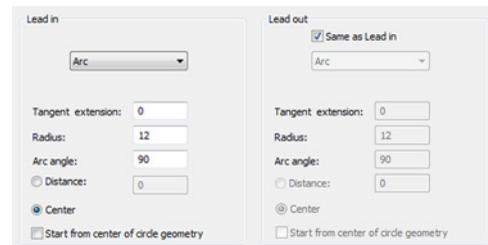
Select the **Finish** check box and set the **Step down** value to **5**. The 0.5 mm allowance that remains after the roughing is removed in one step.



30. Define the Lead in and Lead out

Switch to the **Link** page of the **Profile Operation** dialog box. In the **Lead in** section, choose the **Arc** option. The tool approaches the material tangentially to the geometry at the start point. Set the **Radius** value to **12**.

In the **Lead out** section, select the **Same as Lead in** check box.



31. Save and Calculate

Click the **Save & Calculate** button.

The Profile operation data is saved, and the tool path is calculated.

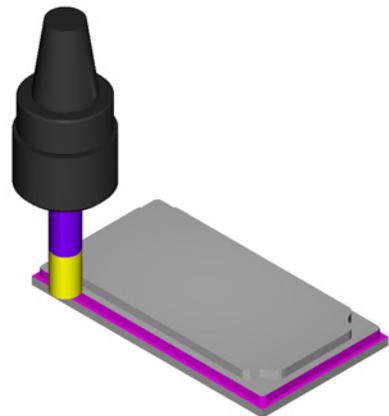


32. Simulate

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.

Close the simulation with the  button. The **Profile Operation** dialog box is displayed. Close the **Profile Operation** dialog box with the **Exit** button.

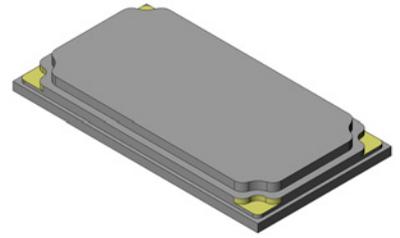




33. Add a Profile operation

At this stage, you have to define a new Profile operation to machine four hole pads.

In **SolidCAM Manager**, add a new Profile operation.

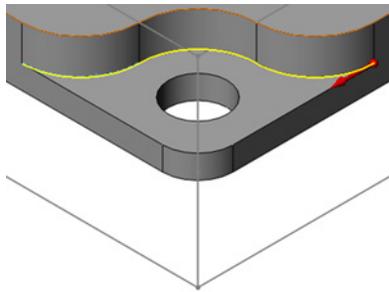
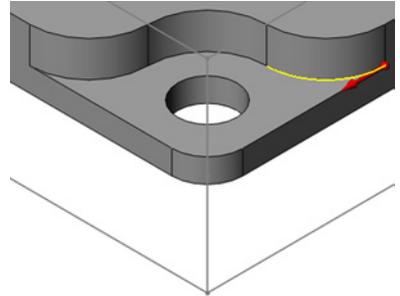


34. Define the Geometry

In the **Geometry** page, click the  button. The **Geometry Edit** dialog box is displayed.

Click on the model edge as shown.

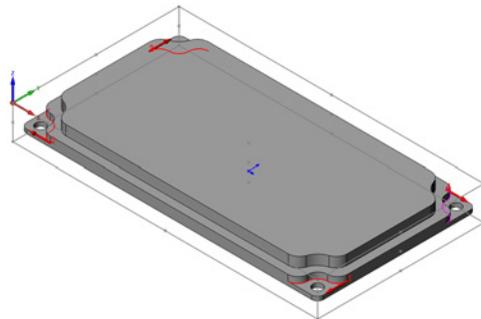
Click on the next model edges as shown below to complete the chain.



In the **Chain List** section, click  to confirm the chain selection.

In the same manner, define the geometry for the rest of the pads. Make sure that all the selected chains have the same direction.

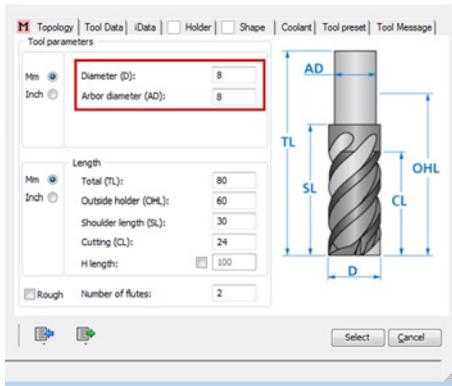
Confirm the geometry definition by clicking .





35. Define the Tool

Define a new tool for the operation. Use an **End mill** of $\text{Ø}8$ with the **BT40 ER32x60** tool holder.

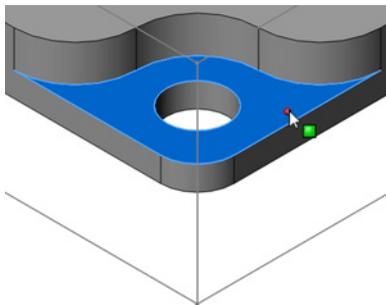
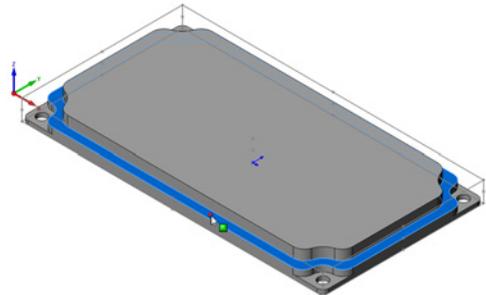


36. Define the Milling levels

In this operation, the machining starts at the Z-level of the already machined faces. The upper level has to be defined.

Define the **Upper level** by clicking on the already machined model face as shown.

In the same manner as explained earlier, define the **Profile depth** by clicking on the pad face as shown.





37. Define the technological parameters

Select the **Rough** check box. Set the **Step down** value to **3**. The profile is machined in one Z-step.

In the **Offsets** section, set the **Wall offset** value to **0.5**. This allowance is removed during the finish machining.

Select the **Clear offset** check box. Set the **Offset** value to **5** and the **Step over** value to **4.5**.

Select the **Finish** check box and set the **Step down** value to **3**. The 0.5 mm allowance remaining after the roughing is removed in one step.



38. Define the Lead in and Lead out

Set the **Arc** option for **Lead in**. Set the **Radius** value to **2**. In the **Lead out** section, select the **Same as Lead in** check box. The tool leads in/leads out to the profile with a tangential arc.



39. Save and Calculate

Click the **Save & Calculate** button.

The Profile operation data is saved and the tool path is calculated.

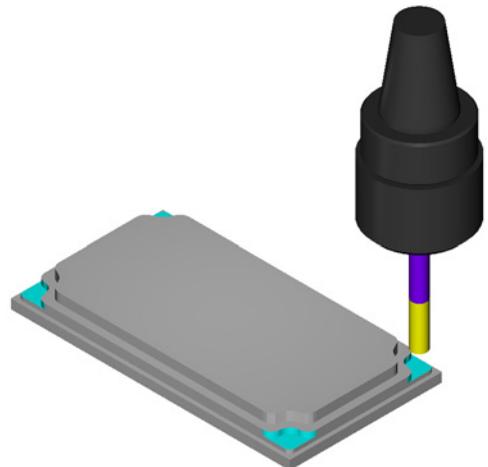


40. Simulate

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.

Close the simulation with the  button. The **Profile Operation** dialog box is displayed.



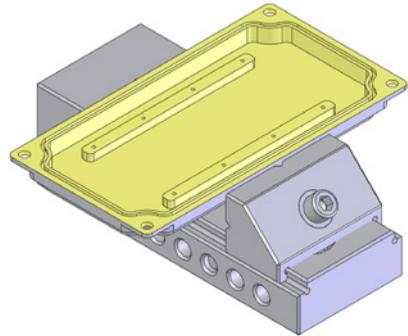
Close the **Profile Operation** dialog box with the **Exit** button.

At this point, the machining of the external cover faces is finished. At the next stages you have to machine the internal faces.



41. Define a New Coordinate System

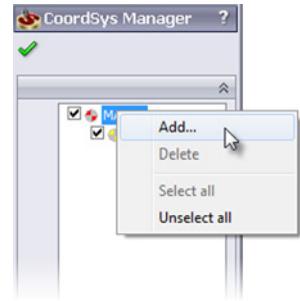
The machining of the internal model faces requires another positioning. The part has to be rotated and clamped in a vice as shown.



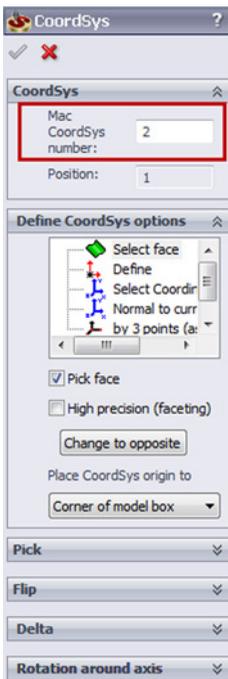
SolidCAM enables you to define the second Coordinate System.

Double-click the **CoordSys Manager** header in **SolidCAM Manager**. The **CoordSys Manager** dialog box is displayed.

Right-click the **MAC 1** item in the list and choose the **Add** option from the menu. The **CoordSys** dialog box is displayed.



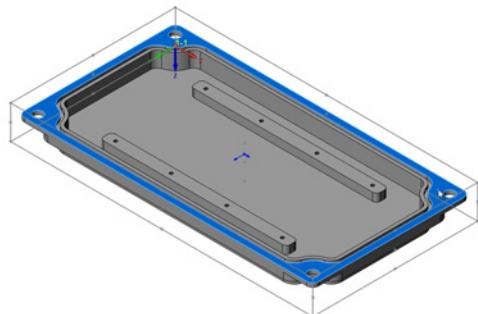
In the **Mac CoordSys Number** field, set the value to **2**. Changing of the **Mac CoordSys number** means that a new clamping is used.



Make sure that the default **Select Face** mode is chosen.

When this mode is chosen, SolidCAM enables you to define a new Coordinate System by selecting a planar face. In this case, the Z-axis of the Coordinate System is normal to the selected face.

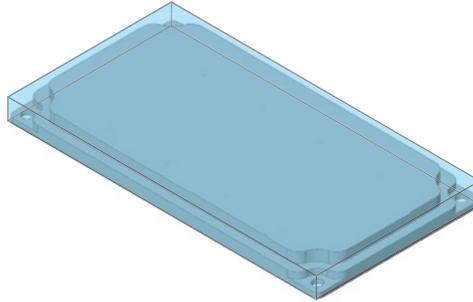
Rotate the model and click on its bottom face as shown below.



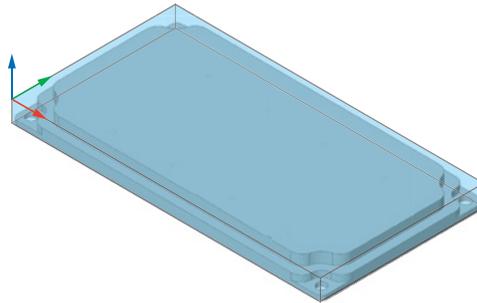
The Coordinate System origin is placed according to the **Place CoordSys origin to** setting. Make sure that the **Corner of model box** option is chosen.



In this case, the box surrounding the model is calculated.



The upper plane of the model box is parallel to the XY-plane of the defined CoordSys. The CoordSys is located in the corner of the model box with the following coordinates: $(X_{MIN}, Y_{MIN}, Z_{MAX})$.

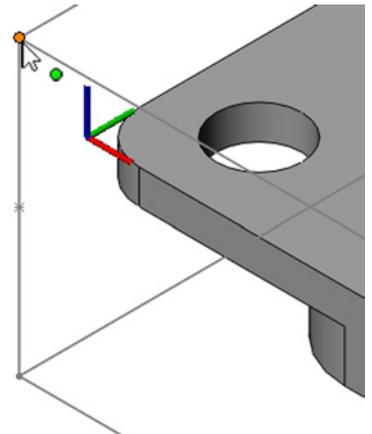


Now you have to move the origin of the Coordinate System from the automatically defined position to the corner of the workpiece.

Select the **Pick origin** check box in the **Pick** section of the **CoordSys** dialog box.



Click on the corner of the workpiece (stock model) as shown to choose it for the origin. The origin is moved to the new location.



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

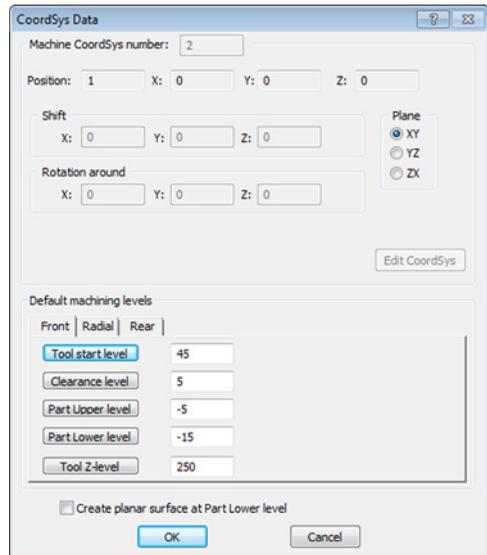
The **CoordSys Data** dialog box is displayed.

SolidCAM automatically determines the **Part Upper level** and the **Part Lower level** parameters relative to the Coordinate System location.

Confirm the dialog box with the **OK** button.

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

Two Machine Coordinate Systems are listed in the **CoordSys Manager** dialog box. Using them you can program operations for different positions (clamping). The **Machine Coordinate System #2** is used for the machining of the back face and the internal faces.

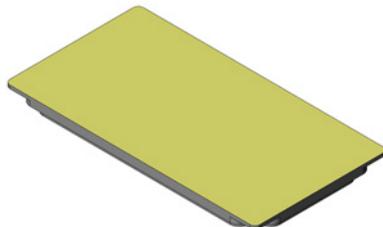


Click  to confirm the **CoordSys Manager** dialog box.



42. Add a new Face Milling operation

Add a new Face Milling operation to perform the back face milling.





43. Define the Geometry

Since this operation is performed with the second Coordinate System position, choose the **Machine Coordinate System #2**.

Choose **MAC 2** from the **CoordSys** list in the **Geometry** section.



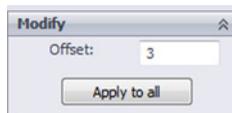
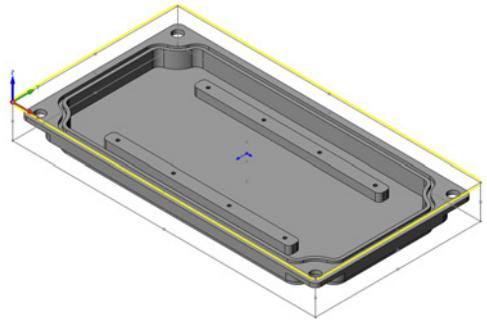
Click the  button to start the geometry definition. The **Face Milling Geometry** dialog box is displayed.

In the **Base Geometry** section, use the default **Model** option for the Face Milling geometry definition. Click the **Define** button. The **3D Geometry** dialog box is displayed.

Click on the solid model to select it. The model is highlighted, and its icon appears in the list. Confirm the **3D Geometry** dialog box by clicking . The **Face Milling Geometry** dialog box is displayed again.

The rectangle is generated surrounding the **Target model** at the XY-plane.

Define the 3 mm offset to extend the machined surface over the stock edges. In the **Modify** section, set the **Offset** value to **3**.



Click  to confirm the **Face Milling Geometry** dialog box. The geometry is defined for the operation.



44. Define the Tool

Use the **Ø40 Face mill** used in previous operations.

Click the **Select** button in the **Tool** page. The **Part Tool Table** is displayed. Choose the previously defined tool and click the **Select** button.

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

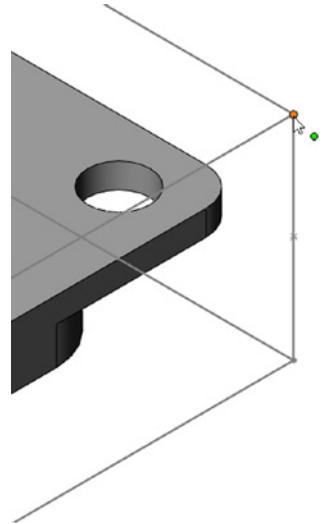
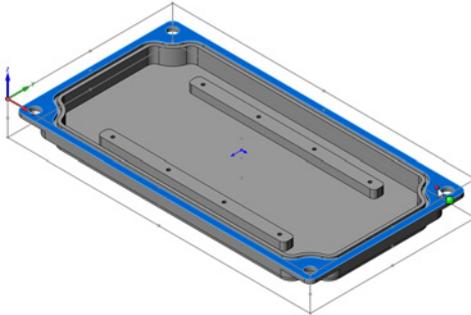


45. Define the Milling levels

In this operation, the machining starts on the top face of the workpiece (**Z=0** of the **Machine Coordinate System #2**) and ends on the back face of the cover part (**Z=-5** of the **Machine Coordinate System #2**).

Define the **Upper level** by clicking on the workpiece corner as shown.

Define the **Face depth** directly on the solid model by clicking on its bottom face as shown below.



Set the **Step down** to **2.5**.

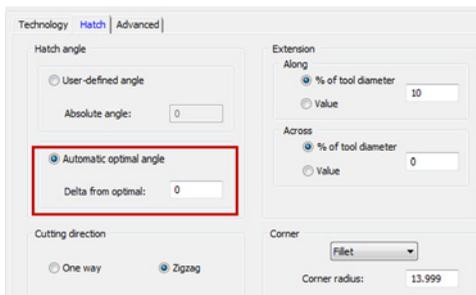


46. Define the technological parameters

Switch to the **Technology** page of the **Face Milling Operation** dialog box. In the **Technology** section, choose the **Hatch** option and click the **Hatch** tab.



In the **Hatch angle** section, switch to the **Automatic optimal angle** option.



Switch back to the **Technology** tab. Note that in the **Overlap** section the **% of tool diameter** option is chosen, and the value is set to **30**.

Define the roughing offset that remains on the floor of the face. This offset is being left unmachined during the rough face machining and is removed during the face finishing.

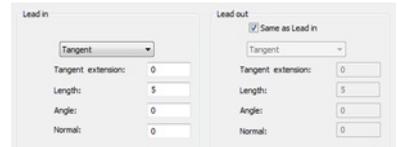
In the **Offsets** section, set the **Floor offset** value to **0.2**. Select the **Finish** check box to perform finishing of the face in this operation. This check box enables you to remove the remaining offset with the last cutting pass.



47. Define the Lead in and Lead out

Switch to the **Link** page of the **Face Milling Operation** dialog box to define the way the tool approaches the material and retreats away.

In the **Lead in** section, choose the **Tangent** option. This option enables the tool to approach the material on a line tangent to the profile. In the **Length** field, set the length of the tangent to **5**.



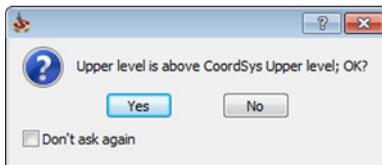
In the **Lead out** section, select the **Same as Lead in** check box.



48. Save and Calculate

Click the **Save & Calculate** button.

SolidCAM notifies you that the upper level chosen for the operation is above the CoordSys Upper level.

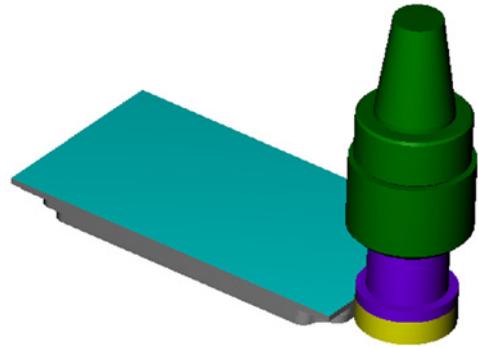


Confirm this message with the **Yes** button. The operation data is saved, and the tool path is calculated.



49. Simulate

Simulate the operation in the **SolidVerify** mode.

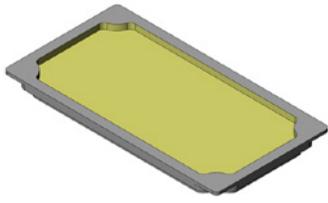
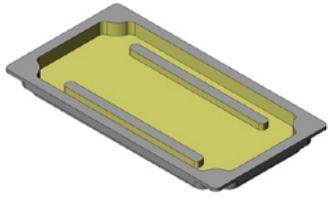
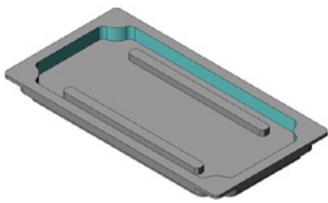


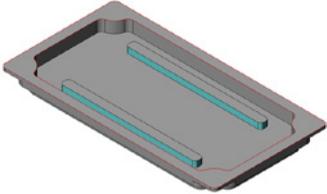
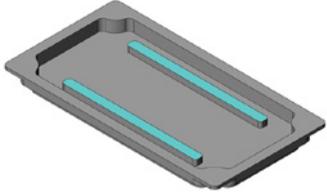
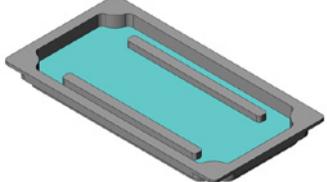
The bottom face machining is finished. Now you have to perform the pocket machining.



Pocket machining overview

The pocket is machined in several technological steps:

	The rough machining of the upper part of the pocket. The machining is performed until the Z-level of the pads is reached.
	The rough machining of the pocket with islands (pads). The machining is performed from the upper face of the pads till the pocket floor. At this stage, two operations are used to perform the machining with two tools of big and small diameter.
	The finish machining of the outside wall of the pocket.

	<p>The finish machining of the island walls.</p>
	<p>The finish machining of the island top face.</p>
	<p>The finish machining of the pocket floor.</p>



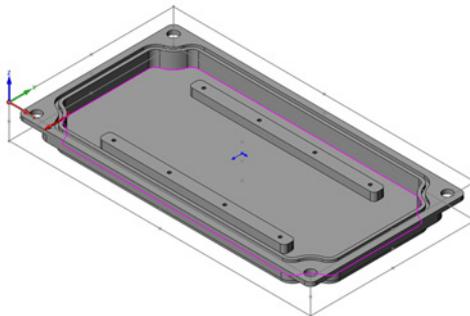
50. Add a Pocket operation

Add a new Pocket operation to perform the rough machining of the upper part of the pocket down to the pads height.



51. Define the Geometry

Choose the **Machine Coordinate System #2** and define the geometry as shown below using the **Auto-constant Z** option.





52. Define the Tool

Choose the already defined **Tool #2 (Ø16 End mill)** from the **Part Tool Table** for the pocket rough machining operation.

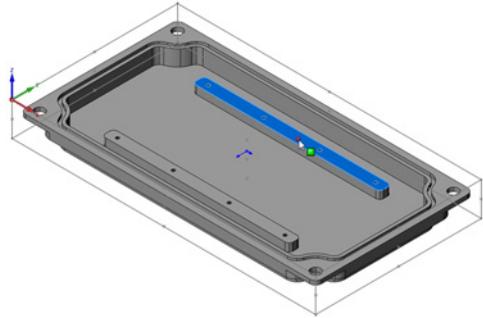


53. Define the Pocket depth

Define the **Pocket Depth** directly on the solid model. Use the top face of the pads for the definition.

SolidCAM uses the constant-Z passes for the Pocket generation. The **Step down** parameter enables you to define the distance between each two successive Z-levels.

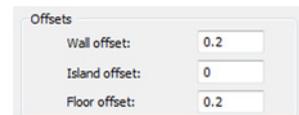
Set the **Step down** value to **2**. The pocket is machined in two Z-levels.



54. Define the technological parameters

Switch to the **Technology** page of the **Pocket Operation** dialog box. Make sure that the default **Contour** option is chosen in the **Technology** section.

In the **Offsets** section, set the **Wall offset** and the **Floor offset** values to **0.2**. These offsets remain unmachined during roughing and are removed with the further finishing.

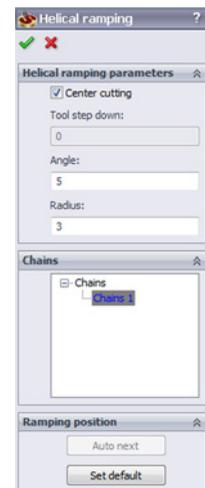


55. Define the Ramping strategy

Switch to the **Link** page of the **Pocket Operation** dialog box. In the **Ramping** section, choose the **Helical** option.

Click the **Data** button. The **Helical ramping** dialog box is displayed.

Set the **Radius** of the descent helix to **3** and confirm the dialog box with .



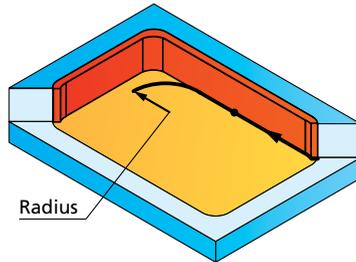


56. Define the Lead out

In the **Lead out** section, choose the **Arc** option and set the **Radius** value to **2**.



With a circular motion tangent to the last entity of the pocket contour, the tool retreats from the profile. The radius of the arc must be specified.



57. Save and Calculate

Click the **Save & Calculate** button.

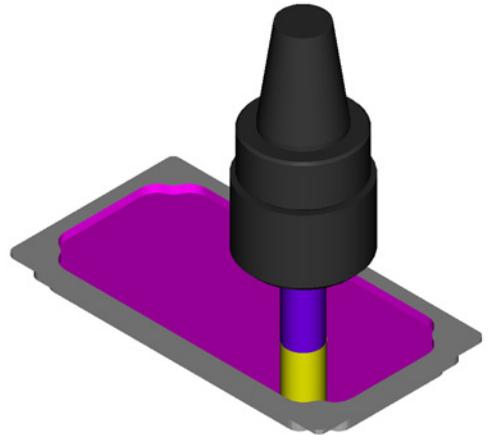
The Pocket operation data is saved and the tool path is calculated.



58. Simulate

Perform the simulation of the Pocket operation in the **SolidVerify** mode.

During the simulation, notice the helical lead in movement.



59. Add a new Pocket operation

Add a new Pocket operation to machine the bottom part of the pocket including two islands (pads for the circuit board installing).

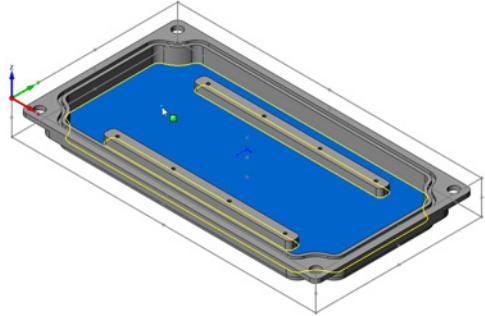


60. Define the Geometry

Choose the **Machine Coordinate System #2** for the operation.

In the same manner as explained in the **Step #18** of this exercise, define the geometry by clicking on the pocket bottom face as shown below.

SolidCAM automatically determines the edges of the selected face and defines chains on them. The first chain is the external boundary of the pocket. All closed chains inside the first chain of each pocket are automatically treated as pocket islands. Overlapping chains are milled as separate pockets, not as islands. To select multiple pockets with islands, continue adding chains to the geometry.



61. Define the Tool

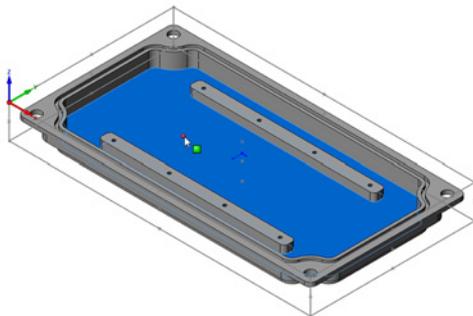
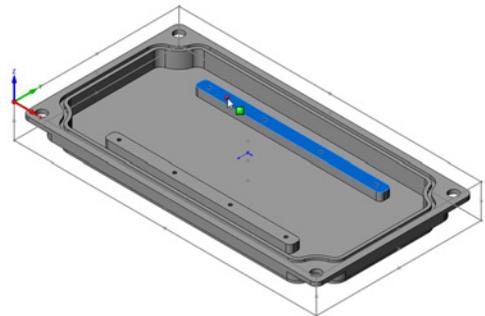
Choose the **Tool #2 (Ø16 End mill)** from the **Part Tool Table** for the operation.



62. Define the Milling levels

The machining in this operation starts at the Z-level of the top faces of the pads and ends on the bottom face of the pocket.

Define the **Upper level** by selecting the top face of the pads as shown.



Define the **Lower level** by selecting the bottom face of the pocket.

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



63. Define the technological parameters

Make sure that the default **Contour** option is chosen in the **Technology** section.

In the **Offsets** section of the **Technology** page, set the **Wall offset**, **Island offset** and **Floor offset** values to **0.2**. These offsets remain unmachined during roughing and are removed in the next finishing operations.

Offsets	
Wall offset:	<input type="text" value="0.2"/>
Island offset:	<input type="text" value="0.2"/>
Floor offset:	<input type="text" value="0.2"/>



Wall offset – the roughing offset remaining on the wall of the pocket.

Island offset – the roughing offset remaining on the pocket's islands.

Floor offset – the roughing offset remaining on the floor of the pocket.



64. Define the Ramping strategy

In the **Ramping** section of the **Link** page, choose the **Helical** option. Click the **Data** button. The **Helical ramping** dialog box is displayed. Set the **Radius** of the descent helix to **3** and confirm the dialog box with the **OK** button.



65. Define the Lead out

In the **Lead out** section, choose the **Arc** option. Set the **Radius** value to **2**.



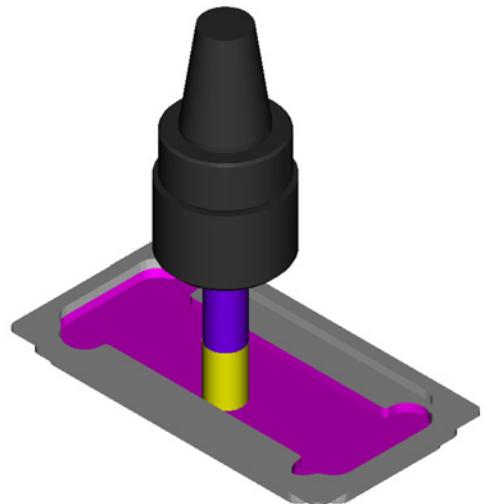
66. Save and Calculate

Click the **Save & Calculate** button. The Pocket operation data is saved, and the tool path is calculated.



67. Simulate

Perform the Pocket operation simulation in the **SolidVerify** mode.



Make a note that the pocket geometry is not completely machined because the tool is too large to reach some model areas. In order to complete the machining, you have to perform an additional Pocket operation with a tool of a smaller diameter in the areas that were not machined in the current Pocket operation.



68. Add a new Pocket operation

Add a new Pocket operation. SolidCAM enables you to use the existing operations as templates for new ones. In this case, the last created Pocket operation is used to define a new Pocket operation from the template. In the **Operation name** section, choose the last defined operation name (**P_contour4**) from the list.

All the parameters of the chosen operation are copied to the current one.



69. Change the Tool

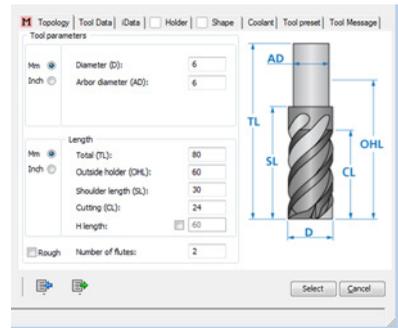
Define a new tool for the operation.

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

This dialog box displays the parameters of the currently active **Tool #2 (Ø16 End mill)**. Since this tool is used in several operations, its parameters cannot be edited.

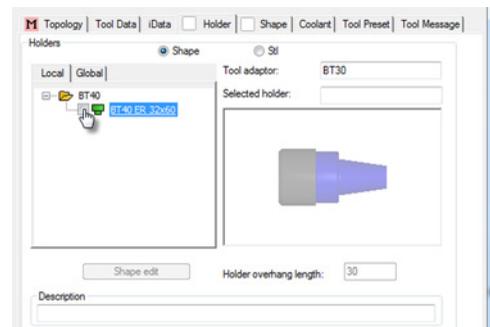
Click  to define a new tool. Choose the **End mill** tool from the **Tool type** dialog box.

A new **Tool #4** is added with the default parameters. Choose the tool holder.



In the **Holder** page, click the **Local** tab. Choose the **BT40 ER32x60** collet chuck from the list.

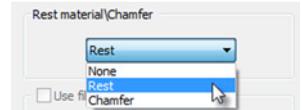
In the **Part Tool Table**, click the **Select** button to choose the defined tool (**Ø6 End mill**) for the operation.



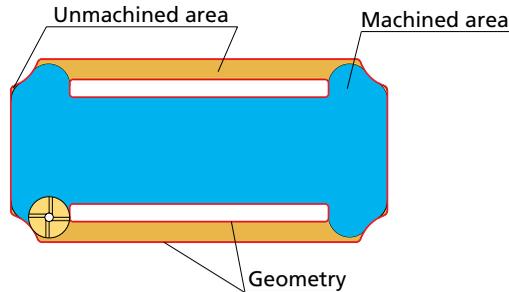


70. Define the Rest material parameters

In the **Rest Material/Chamfer** section of the **Technology** page, choose the **Rest** option from the list.



During the machining, when a large tool is used, the tool leaves material in areas that it cannot enter.



The **Rest material** option enables you to remove the material from these areas without defining a new geometry.

The new **Rest** tab appears and opens the page automatically.

Notice that the **Separate areas** option is chosen by default in the **Milling type** box. When this option is chosen, SolidCAM performs the machining only in areas that were not machined with the previous tool.



Define the diameter of the end mill that was used in the previous operation. Click the **Previous tool diameter** button. The **Part Tool Table** is displayed with all the tools used in the operations of the current CAM-Part. Choose the **Ø16 End mill** used in the previous operation and click the **Select** button.

In the **Previous wall offset** field, set the value of **0.2**. This offset was defined in the previous Pocket operation.

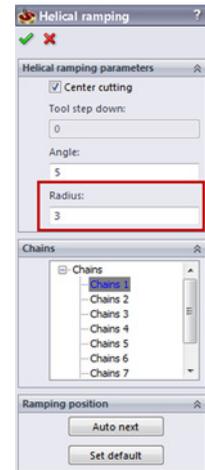
Define the **Extension/Overlap** parameter – the overlap distance that you would like to start from and end with the previous larger end mill. Set the **Extension/Overlap** value to **1**.



71. Define the Ramping strategy

In the **Ramping** section of the **Link** page, choose the **Helical** strategy to define how the tool enters into the material. Click the **Data** button. The **Helical ramping** dialog box is displayed. Set the Z-entry helix **Radius** to **3**. Confirm the dialog box with .

With the defined parameters, the tool machines all the areas that were not machined by the previous tool. The 0.2 mm allowance remains for further finishing. The machining area is extended by 1 mm to overlap the previously machined area.



72. Save and Calculate

Click the **Save & Calculate** button.

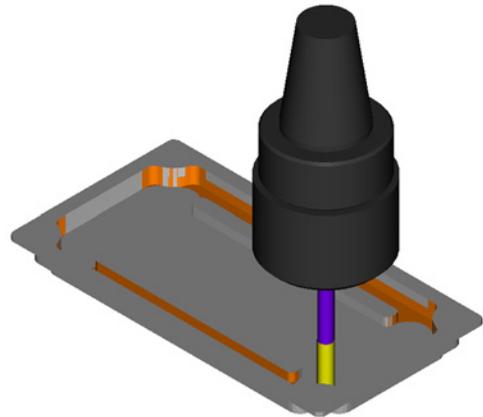
The Pocket operation data is saved, and the tool path is calculated.



73. Simulate

Play the simulation of the Pocket operation in the **SolidVerify** mode.

Notice that the machining is performed only in the areas that were not machined in the previous operation.

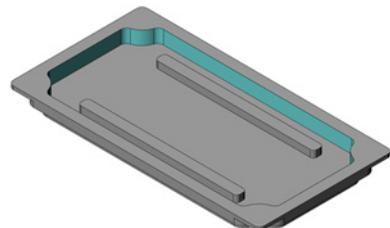


At this stage, the rough machining of the pocket is completed, and you have to program the finishing operations.



74. Add a Profile operation

A Profile operation is used for the finish machining of the pocket walls.



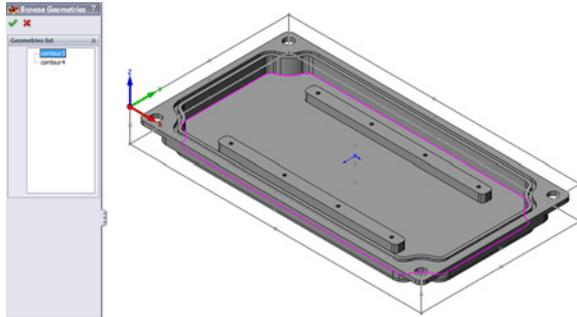
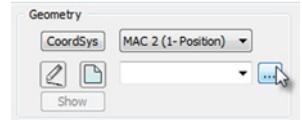


75. Define the Geometry

In the **Geometry** page, click the **Browse** (...) button to view the geometries defined for the current Coordinate System.

The **Browse Geometries** dialog box is displayed with a list of geometries defined for this CAM-Part associated with the current Coordinate System. You can click entries in the list to display the corresponding geometries on the model.

Choose the **Contour3** geometry from the list.



Confirm the **Browse Geometries** dialog box by clicking .



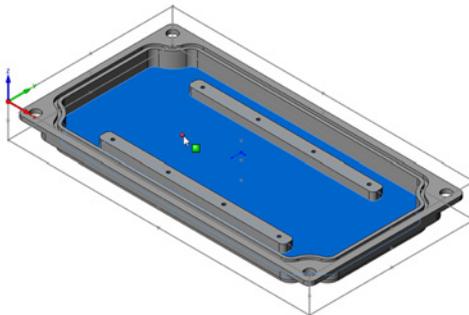
76. Define the Tool

Choose the already existing **Tool #4 (Ø6 End mill)** from the **Part Tool Table**.



77. Define the Profile depth

Define the **Profile depth** by clicking on the bottom face of the pocket.





78. Define the technological parameters

In the **Modify** section, choose the **Right** option from the **Tool side** area.



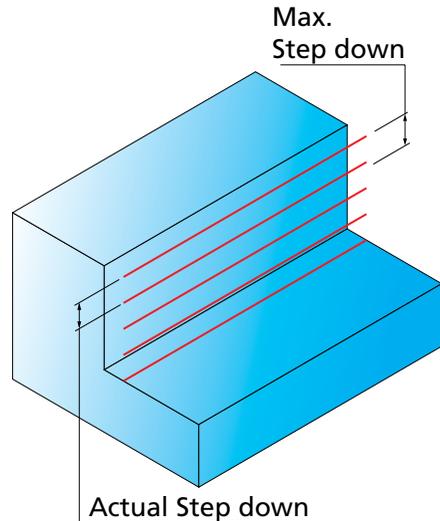
In this operation, use the **Equal step down** option to keep an equal distance between all Z-levels.



Equal Step down

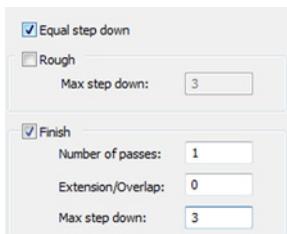
This option enables you to perform all cuts at an equal Z-level distance one from the other. SolidCAM automatically calculates the actual step down to keep an equal distance between all passes.

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.

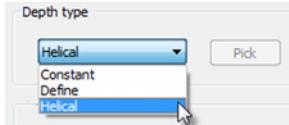


Select the **Equal step down** check box.

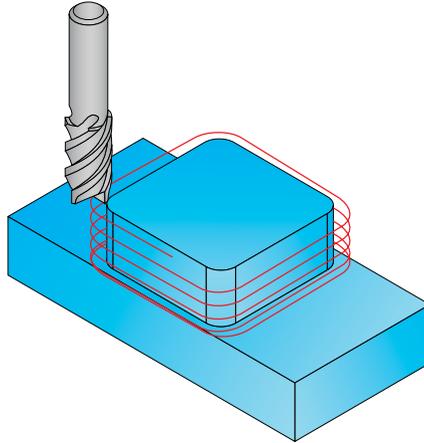
Select the **Finish** check box and set the **Max. Step down** value to **3**.



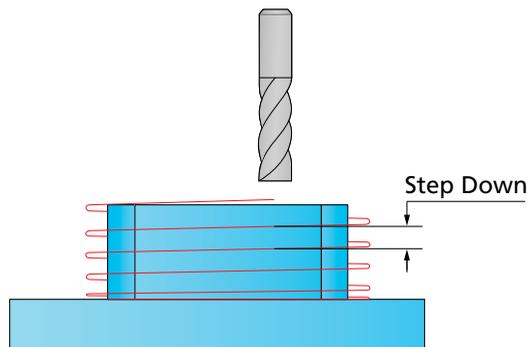
In the **Depth type** area, choose the **Helical** option.



The **Helical** milling option enables you to enhance the productivity of profile machining by avoiding lead in/out movements at each Z-level and eliminating unnecessary rapid movements.



With the **Helical** option, the tool performs spiral movements around the geometry with continuous lowering along the Z-axis. For each turn around the geometry, the tool moves downward along the Z-axis according to the step down value. When the Profile depth is reached by the spiral movements, SolidCAM performs the last cut with the constant-Z movement at the Profile depth.





79. Define the Lead in and Lead out

Choose the **Normal** option in the **Lead in** section of the **Link** page and set the **Normal length** value to **5**. In the **Lead out** section, select the **Same as Lead in** check box. The tool approaches the material with the movement normal to the pocket contour and retreats in the same way.



80. Save and Calculate

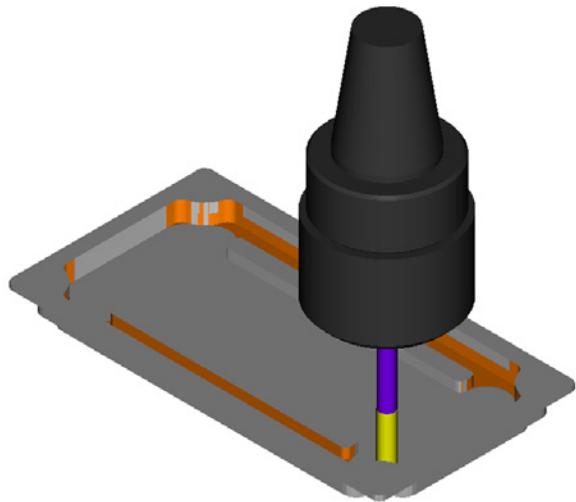
Click the **Save & Calculate** button.

The Profile operation data is saved, and the tool path is calculated.



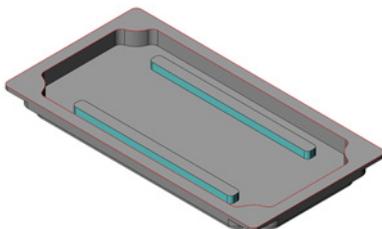
81. Simulate

Play the simulation in the **SolidVerify** mode.



82. Add a Profile operation

Add a new Profile operation to machine the walls of the islands.

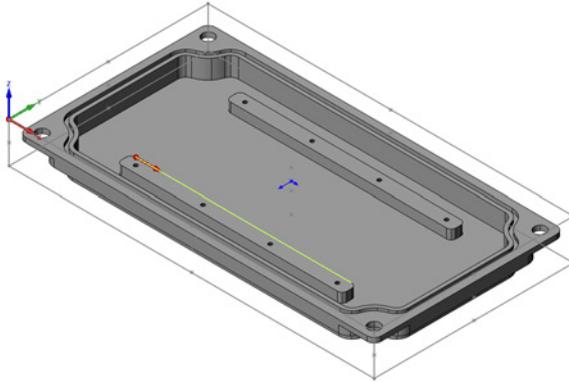




83. Define the Geometry

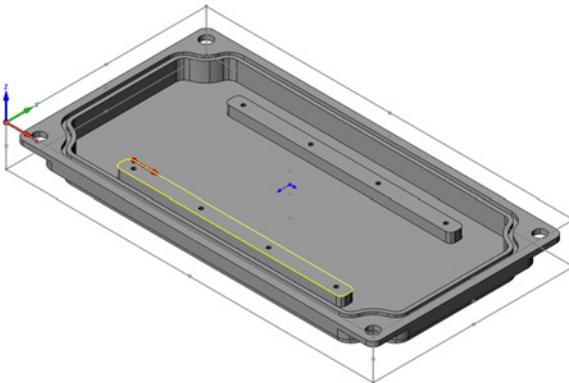
Choose the **Machine Coordinate System #2** for the operation and click the  button in the **Geometry** page. The **Geometry Edit** dialog box is displayed.

Select the edge of the island as shown below.



Make sure that the chain direction is clockwise. Such direction enables you to perform the climb milling of the profile.

In the **Chain** section, choose the **Auto-constant Z** option. The chain is automatically completed.

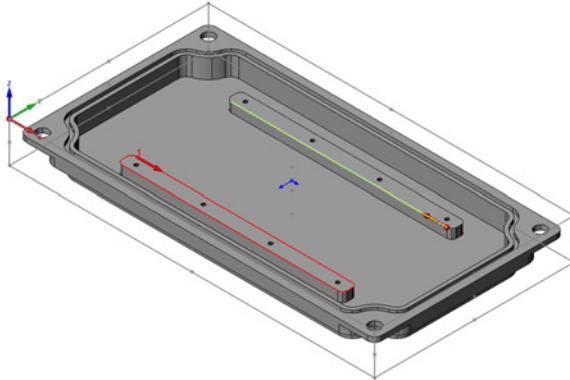


The confirmation message is displayed.



Confirm it by clicking **Yes**. The first chain is defined.

Select the edge of the other island as shown below.



Make sure that the chain direction is clockwise. Such direction enables you to perform the climb milling of the profile.

In the **Chain** section, choose the **Auto-constant Z** option. The chain is automatically completed and the confirmation message is displayed. Confirm it by clicking the **Yes** button.



The first selected entity in the geometry chain defines the approach location for the whole chain. In this case, the internal edges of the pads are chosen in order to prevent the collision between the tool and pocket wall during the lead in movement.

Confirm the Geometry definition with .



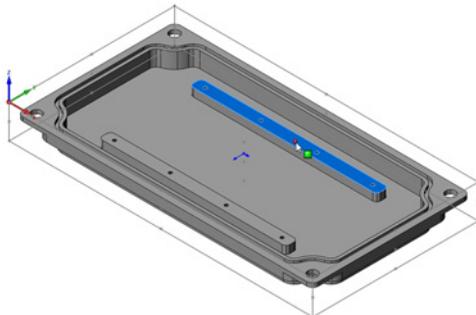
84. Define the Tool

Choose the **Tool #4 (Ø6 End mill)** for the operation.

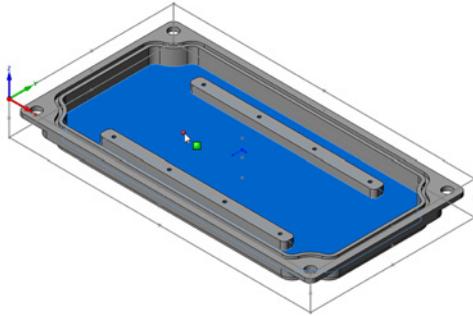


85. Define the Milling levels

Define the operation **Upper level** directly on the solid model by clicking on the top face of the pad as shown below.



Define the **Profile depth** by clicking on the bottom face of the pocket.



86. Define the technological parameters

In the **Technology** page, make sure the **Finish** check box is selected and set the **Step down** value to **4**.

In the **Depth Type** area, set the **Helical** option.



87. Define the Lead in and Lead out

Choose the **Arc** option in the **Lead in** section and set the **Radius** value to **2**. In the **Lead out** section, select the **Same as Lead in** check box. The tool approaches the material with an arc movement tangential to the geometry and retreats in the same way.



88. Save and Calculate

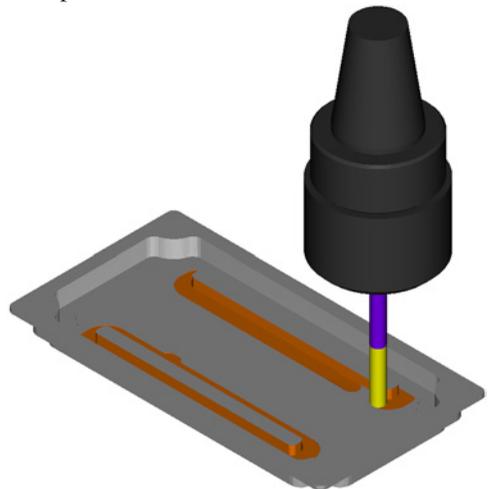
Click the **Save & Calculate** button.

The Profile operation data is saved, and the tool path is calculated.



89. Simulate

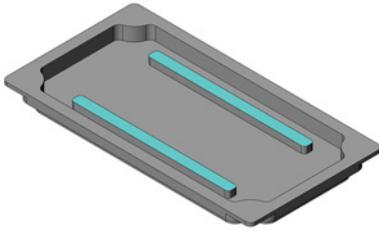
Play the simulation in the **SolidVerify** mode.





90. Add a Face Milling operation

Add a new Face Milling operation to machine the top faces of the pads.



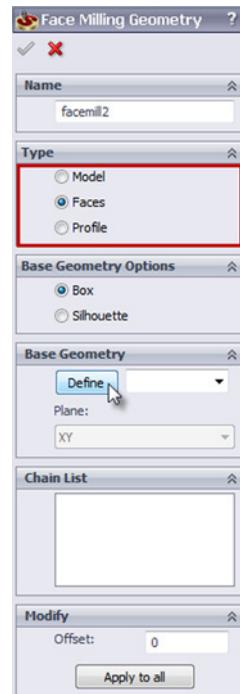
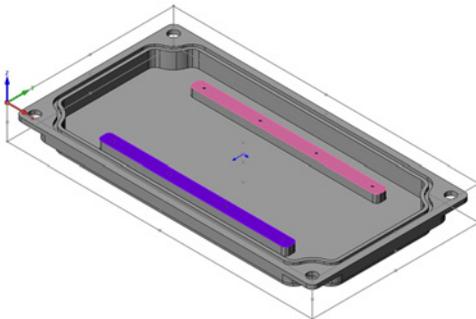
91. Define the geometry

In the **Geometry** page, choose the **Coordinate System #2 (MAC 2)** and click  to define the geometry for this milling operation.

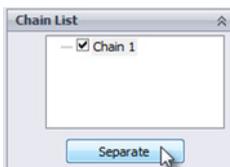
In the **Face Milling Geometry** dialog box, choose the **Faces** option in the **Type** section and then click the **Define** button.

The **Select Faces** dialog box is displayed.

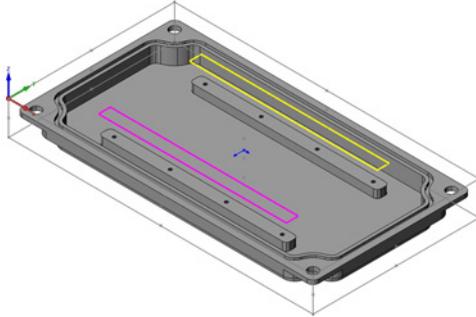
Click on the surface of the tow pads and confirm your selection by clicking .



The **Face Milling Geometry** dialog box is displayed again. In order to machine the two faces separately, click the **Separate** button in the **Chain List** section.



Two separate chains are created.



Confirm the geometry definition by clicking .



92. Define the tool

Choose the **Tool # 3 (Ø8 End mill)** for the operation.



93. Milling levels

Define the operation **Upper level** directly on the solid model by clicking on the top face of the pad.

Define the **Face depth** value by clicking on the same face.



94. Define the technological parameters

In the **Technology** page, choose the **One Pass** strategy from the list.



95. Save and Calculate

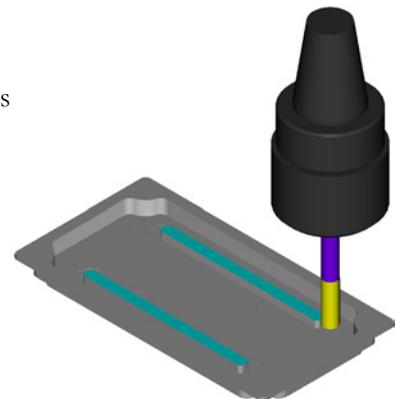
Click the **Save & Calculate** button.

The operation data is saved, and tool path is calculated.



96. Simulate

Play the simulation in the **SolidVerify** mode.

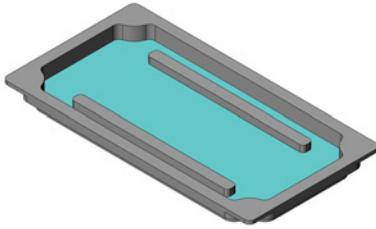


At the next stage you have to machine the pocket floor.



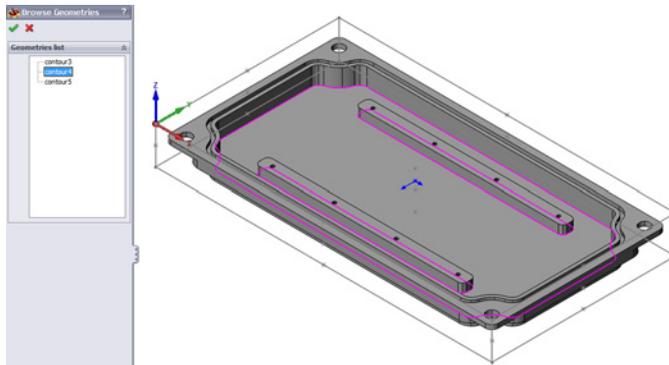
97. Add a Pocket operation

Define a new Pocket operation to perform the finish machining of the pocket floor.



98. Define the geometry

Use the already defined geometry **contour4**.



99. Define the tool

Choose the **Tool #4 (Ø6 End mill)** for the operation.



100. Define the Milling levels

Define the operation **Upper level** directly on the bottom face of the pocket. Define the **Pocket depth** value by clicking on the same face to perform the machining in a single Z-level.



101. Define the technological parameters

Make sure that the default **Contour** strategy is chosen in the **Technology** page.

In the **Offsets** section, set the **Wall offset** and **Island offset** values to **0.1**. Such offset prevents the contact between the tool and the already finally machined wall and island surfaces during the floor machining.

**102. Define the Lead out**

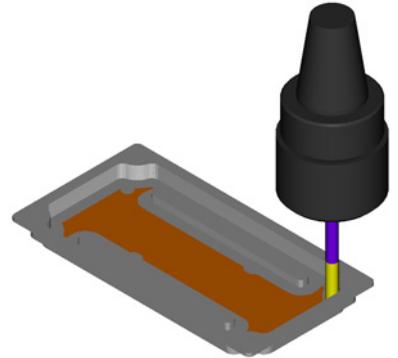
Choose the **Arc** option from the **Lead out** list. Set the **Radius** value to **2**.

**103. Save and Calculate**

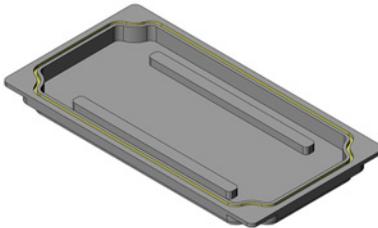
Click the **Save & Calculate** button. The Profile operation data is saved and the tool path is calculated.

**104. Simulate**

Play the simulation in the **SolidVerify** mode.

**105. Add a Slot operation**

Add a new Slot operation to machine the slot on the bottom face of the cover model.



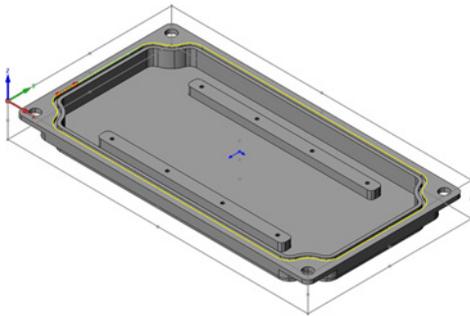
In the **SolidCAM Operations** toolbar of the **CommandManager**, choose **2.5D > Slot**.

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



106. Define the Geometry

Choose the **Machine Coordinate System #2** for the operation and define the geometry for the slot as shown.



Make sure that the chain direction is clockwise to perform the climb milling.



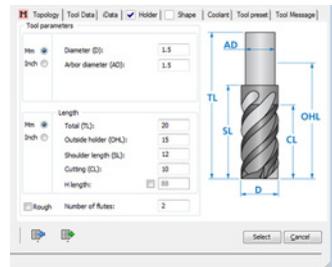
107. Define the Tool

Define a new tool in the **Part Tool Table (Ø1.5 End mill)**.

Edit the following tool parameters:

- Set the **Total tool length** to **20**
- Set the **Outside holder length** to **15**
- Set the **Shoulder length** to **12**
- Set the **Cutting length** to **10**

Choose the **BT40 ER16x70** collet chuck from the **Global** holders table for the operation.



108. Define the technological parameters

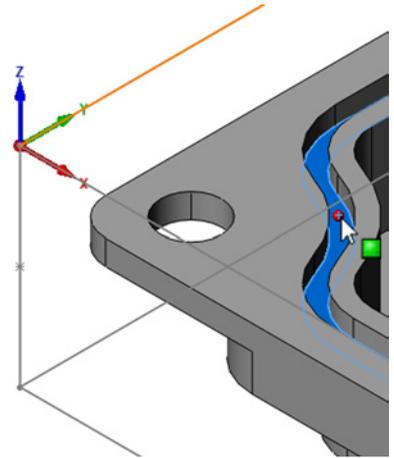
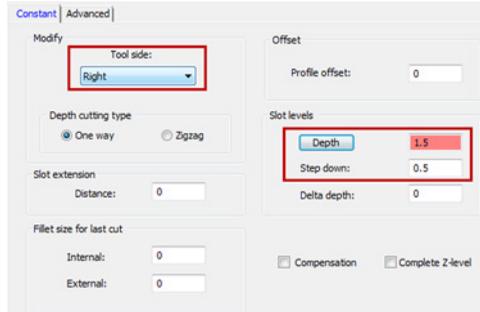
In the **Tool side** area, choose the **Right** option to machine the slot at the right side of the selected geometry.

In the **Slot levels** area, click the **Depth** button to define the slot depth directly on the solid model.

Click on the bottom face of the slot and confirm the definition with .

The **Slot depth** (1.5 mm) is determined.

Set the **Step down** value to **0.5**.



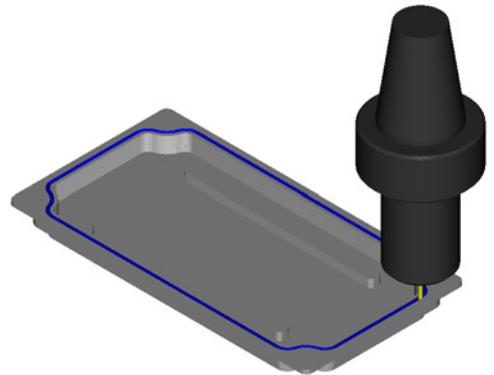
109. Save and Calculate

Click the **Save & Calculate** button. The Slot operation data is saved and the tool path is calculated.



110. Simulate

Play the simulation in the **SolidVerify** mode.



At the final stage of the cover part, you have to perform several operations to machine the holes located on the bottom face and pads.



111. Define a Drilling operation

Define a new Drilling operation for the center drilling of the holes located on the bottom face of the cover.



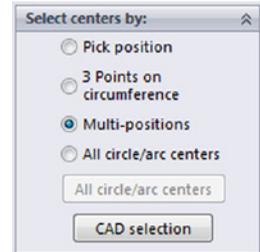
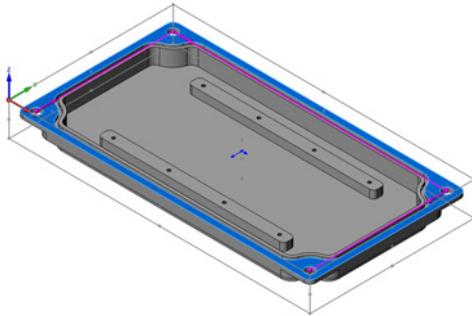
112. Define the Geometry

In the **Geometry** page, choose the **Machine Coordinate System #2** for the operation and click the  button to start the definition of drill positions.

The **Drill Geometry Selection** dialog box is displayed.

Choose the **Multi-positions** option in the **Select centers by** section.

Click on the bottom model face as shown below.



All the circular edges located on the selected face are collected, and the drill positions are determined. Confirm the geometry selection with .

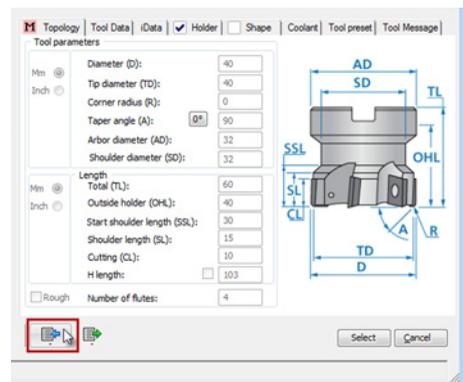


113. Define the Tool

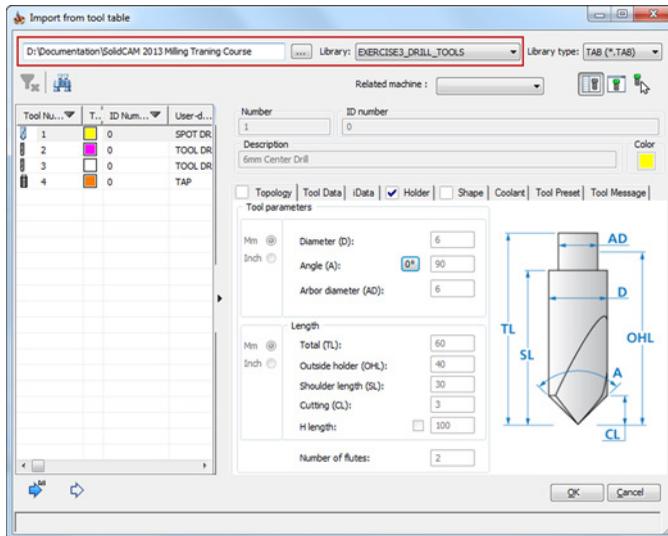
In the **Tool** page, click the **Select** button. The **Part Tool Table** is displayed.

In the drilling operations of this exercise, the tools imported from an external tool library are used. The **Exercise3_Drill_Tools** library contains several drilling tools required for this exercise. This library is located in the **Exercises** folder you have copied to your hard drive.

In the **Part Tool Table**, click the **Import Tool(s)** button and choose **Import** to import the tool from the previously defined tool library.



The **Import from Tool Table** dialog box is displayed. Click the **Browse** button (⋮) and choose the location of the **Exercise3_Drill_Tools** tool library. Set it in the **Library** list.



Click the **Import All Tools** button (⏏) located in the left bottom part of the dialog box.

The tools are imported into the **Part Tool Table**.

Click **OK** to exit from the tool library.

In the **Part Tool Table**, choose the **Tool #6 (Ø6 Spot drill)** for the operation.

Confirm the tool definition by clicking the **Select** button.

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

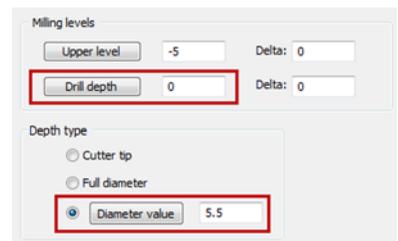


114. Define the Drilling depth

In the **Milling levels** area, set the **Drill depth** to **0**.

To perform center drilling, use the **Depth type** option. In the **Depth type** section, choose the **Diameter value** option and set the value to **5.5** mm.

The center drilling operation is performed until the specified diameter is reached at the upper level (the top face of the model). The upper diameter of the center drilling cone is greater than the drilled hole diameter; this means that the 0.25 mm chamfer is performed at the hole edge.





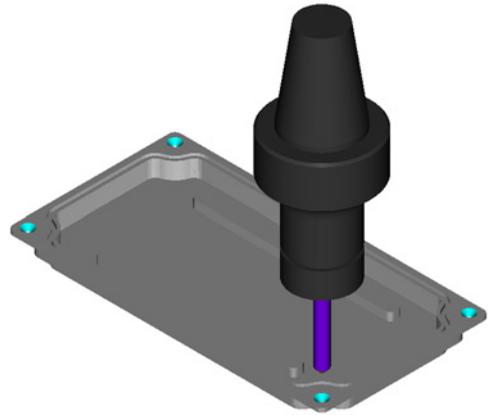
115. Save and Calculate

Click the **Save & Calculate** button. The Drilling operation data is saved and the tool path is calculated.



116. Simulate

Play the simulation of the center drilling in the **SolidVerify** mode.



117. Add a Drilling operation

In **SolidCAM Toolbar**, add a new Drilling operation to drill the holes center-drilled at the previous stage.

Choose the same **Drill** geometry used in the previous operation.

Choose the **Tool #7 (Ø5 Drill)** from the **Part Tool Table**.

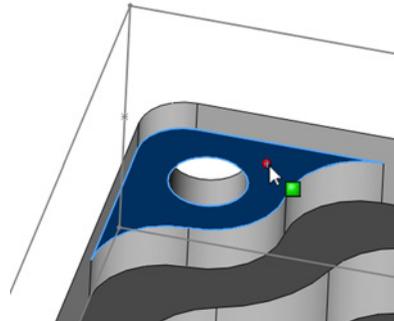


118. Define the Drilling depth

In the **Levels** page, define the **Upper level** by clicking on the model upper face. Define the **Drill depth** by clicking on the model face as shown.

Set the **Delta** depth value to **-1**.

To perform through drilling, use the **Depth type** option. In the **Depth type** section, choose the **Full diameter** option. The drilling is performed until the given diameter is reached at the specified Drill depth. This means that the conical part of the tool exits from the material.



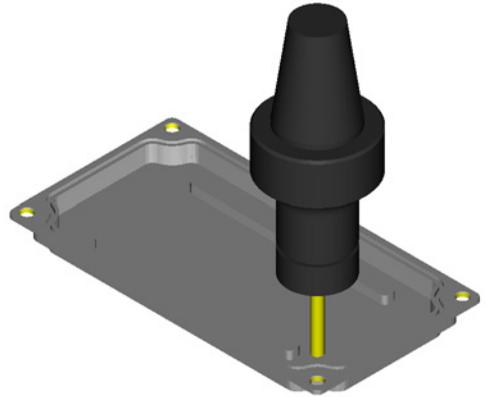
119. Save and Calculate

Click the **Save & Calculate** button. The Drilling operation data is saved and the tool path is calculated.



120. Simulate

Play the simulation of the drilling in the **SolidVerify** mode.



At the next stage, you have to perform a number of drilling operations (center drilling, drilling, tapping) for eight holes located on the pads top faces.



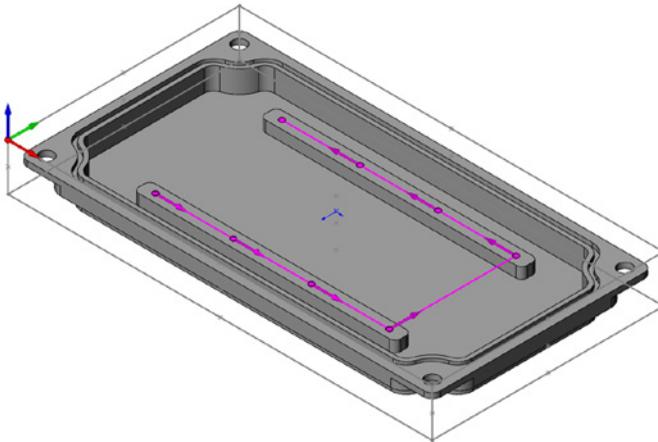
121. Add a Drilling operation

Add a new Drilling operation to perform the center drilling of the pad holes.



122. Define the Geometry

In the same manner as explained in **Step #112**, define the geometry for the drilling. Use the **Machine Coordinate System #2** for the operation.



123. Define the Tool

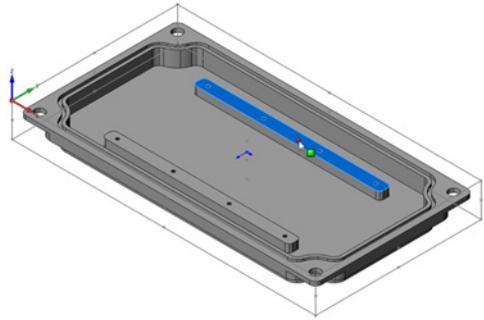
Choose the **Tool #6** ($\text{\O}6$ Spot drill) from the **Part Tool Table** for the operation.



124. Define the Drilling depth

In the **Levels** page, define the **Upper level** by clicking on the top face of the pad.

Set the **Drill depth** value to **1**.



125. Save and Calculate

Click the **Save & Calculate** button. The Drilling operation data is saved, and the tool path is calculated.



126. Add a Drilling operation

Add a new Drilling operation to perform the drilling of the pad holes.

In the **Geometry** area, choose the **Drill1** geometry defined in **Step #122**.

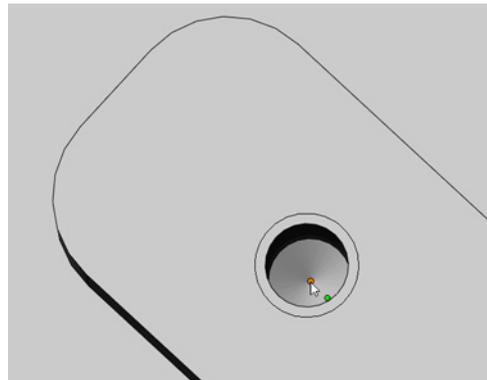
Choose the **Tool #8 (Ø1.6 Drill)** for the operation. This tool enables you to perform preliminary drilling for the M2 tapping.



127. Define the Milling levels

In the **Levels** page, define the **Upper level** in the same manner as explained in **Step #124**.

Define the **Drill depth** directly on the solid model. Pick the lower vertex of the hole cone as shown to define the depth.



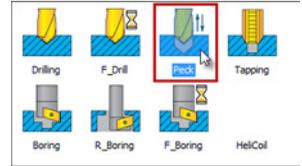


128. Define the Drill cycle type

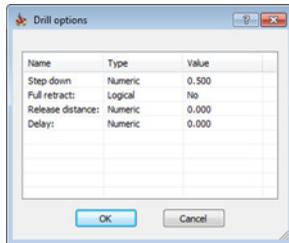
Click the **Drill cycle type** button in the **Technology** page. The Drill Cycles panel is displayed.

Choose the **Peck** type to perform the pecking canned cycle.

Click the **Data** button to define the pecking parameters.



The **Drill options** dialog box is displayed. Set the **Step down** value for pecking to **0.5**.



Confirm the dialog box with **OK**.



129. Save and Calculate

Click the **Save & Calculate** button. The operation data is saved, and the tool path is calculated.

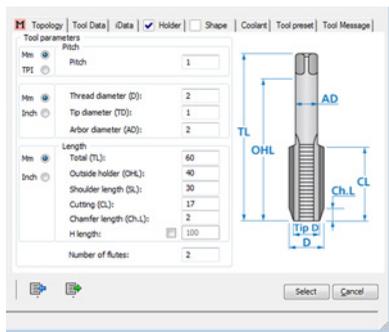


130. Add a Drilling operation

Add a new Drilling operation to perform the tapping of the pads holes.

In the **Geometry** area, choose the **Drill1** geometry defined in **Step #122**.

Choose the **Tool #9 (Ø2 Tap)** for the operation.

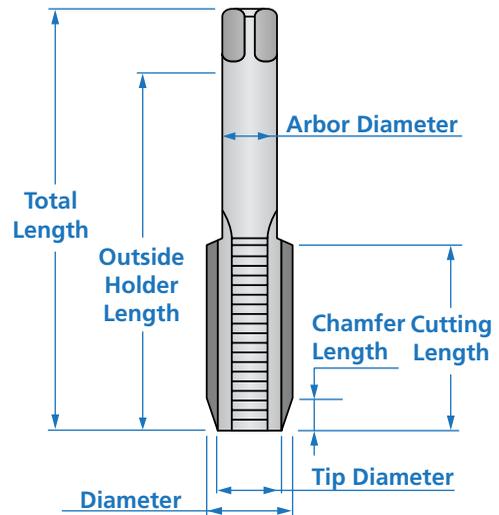




Tap Tool

This tool is intended for the machining of internal threads in Drilling operations.

The tool consists of two parts: cylindrical and conical, and is defined by the following major parameters presented on the illustration.



When the **Tap** tool is used in the operation, the **Tapping drill cycle** is chosen automatically.



131. Define the Drilling depth

In the **Levels** page, define the **Upper level** in the same manner as explained in **Step #124**.

Set the **Drill Depth** to **4**.



132. Save and Calculate

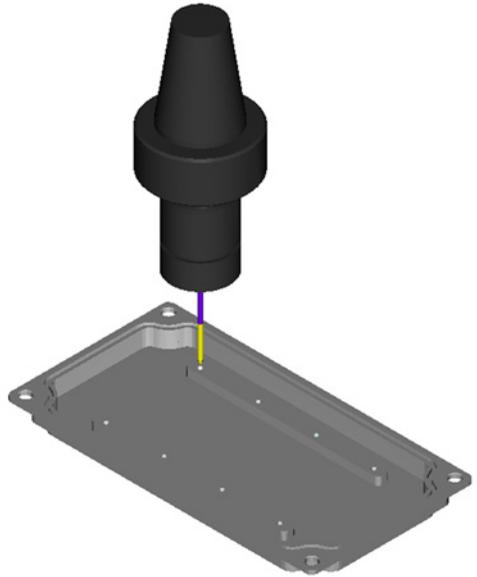
Click the **Save & Calculate** button. The Drilling operation data is saved and the tool path is calculated.



133. Simulate

Click the **Simulate** button. The **Simulation** control panel is displayed.

SolidCAM enables you to simulate several operations together. Holding the **Shift** key, select the last three operations in **SolidCAM Manager** (center drilling, drilling and tapping of the pad holes) and play the simulation.



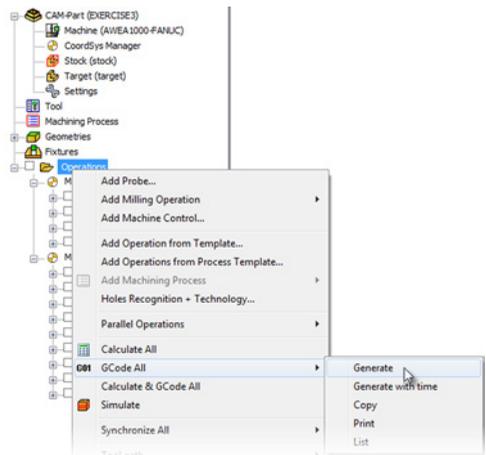
At this stage, programming of the machining operations is completed. Now you have to generate the GCode to send it to the CNC-machine controller.



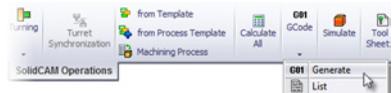
134. Generate the GCode

SolidCAM enables you to generate the GCode for several operations or for the complete CAM-Part.

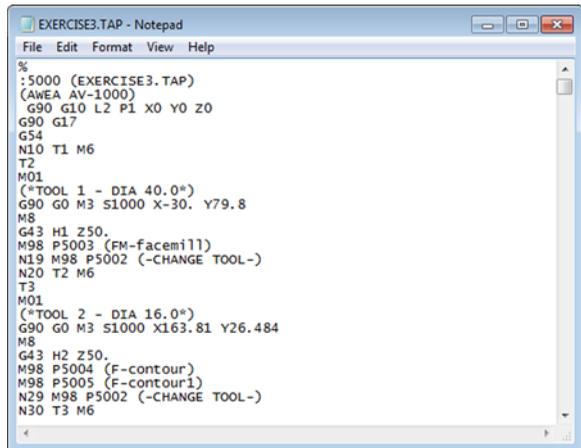
Right-click the **Operations** header in **SolidCAM Manager** and choose the **Generate** command from the **GCode All** submenu.



Alternatively, you can choose **GCode > Generate** icon from the **SolidCAM Operations** toolbar.



The GCode is generated for the complete CAM-Part. The generated GCode is displayed in the **Notepad** window.

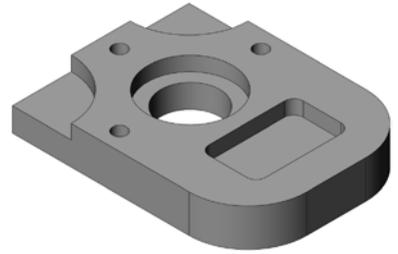


```
EXERCISE3.TAP - Notepad
File Edit Format View Help
%
: S000 (EXERCISE3.TAP)
(AWEA AV-1000)
G90 G10 L2 P1 X0 Y0 Z0
G90 G17
G54
N10 T1 M6
T2
M01
(*TOOL 1 - DIA 40.0*)
G90 G0 M3 S1000 X-30. Y79.8
M8
G43 H1 Z50.
M98 P5003 (FM-facemill)
N19 M98 P5002 (-CHANGE TOOL-)
N20 T2 M6
T3
M01
(*TOOL 2 - DIA 16.0*)
G90 G0 M3 S1000 X163.81 Y26.484
M8
G43 H2 Z50.
M98 P5004 (F-contour)
M98 P5005 (F-contour1)
N29 M98 P5002 (-CHANGE TOOL-)
N30 T3 M6
```

At this stage, the exercise is completed.

Exercise #4: Bracket Machining

Define the CAM-Part and operations for the machining of the bracket presented on the illustration on a Milling CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Face milling
- Pocket machining
- Profile machining
- Drilling

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

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

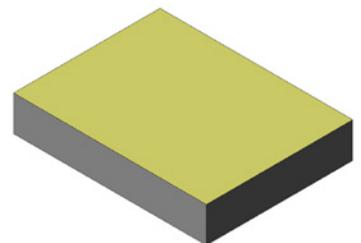
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller, Machine Coordinate System, the Stock and Target models.

The **AWEA1000-Fanuc** CNC-controller has to be chosen for this exercise.

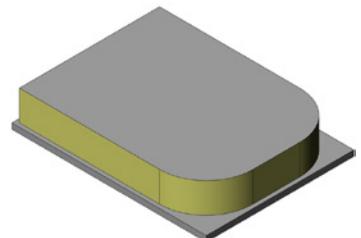
2. Machine the top face

Define a Face Milling operation to machine the top face of the bracket.



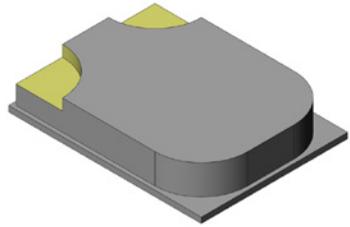
3. Machine the side faces

Perform the Profile rough and finish machining of the bracket side faces.



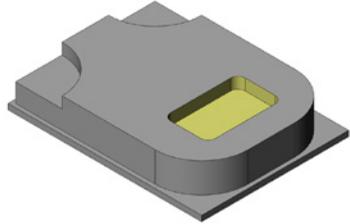
4. Machine the open pockets

Define a Profile operation to perform the machining of the open areas as shown.



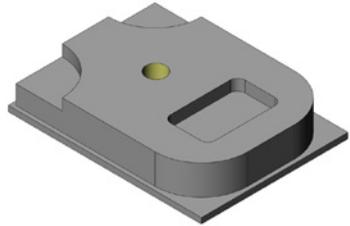
5. Machine the pocket

Define a Pocket operation to perform rough and finish machining of the pocket located on the top face of the bracket.



6. Pre-machine the cylindrical hole

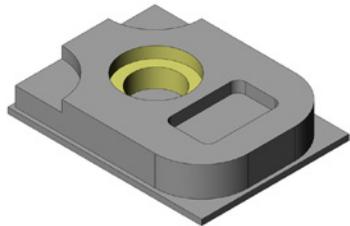
Define two Drilling operations to perform the center drilling and preliminary drilling for the further pocket operation.



7. Machine the cylindrical hole

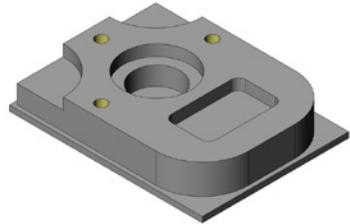
Perform two Pocket operations to machine the faces of the through hole.

Use the pre-drilled hole to perform the safe lead in movement of the tool into the material.



8. Machine the holes

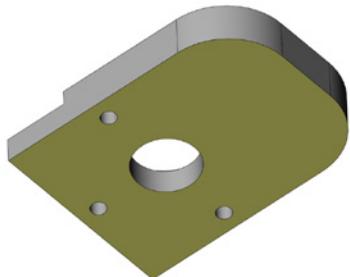
Perform center drilling and drilling of the holes.



9. Machine the bottom face

Define a Face Milling operation to machine the bottom face of the bracket.

This operation requires different clamping, so define and use a new **Coordinate System** with the Z-axis oriented normally to the bottom face.

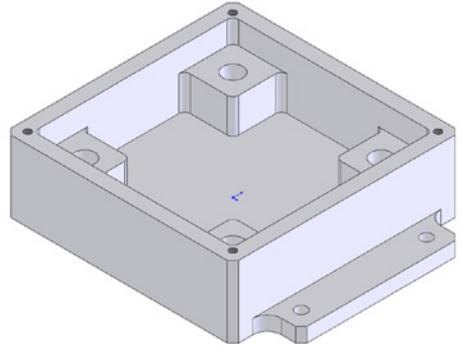


Exercise #5: Electronic Box Machining

Define the CAM-Part and operations for machining of the electronic box presented on the illustration on Milling CNC-machine.

This exercise reinforces the following skills:

- CAM-Part definition
- Face milling
- Pocket machining
- Profile machining
- Rest material machining
- Drilling



The SolidWorks model of the Electronic Box (**Exercise5.sldprt**) is located in the **Exercises** folder.

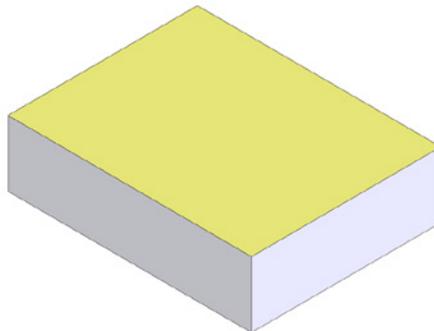
The following steps have to be implemented in order to reach the final CAM-Part:

1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**AWEA1000-Fanuc**), Machine Coordinate System, the Stock and Target models.

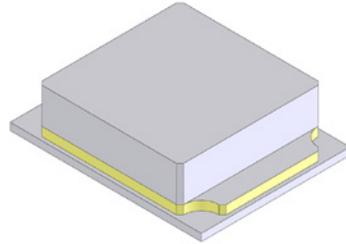
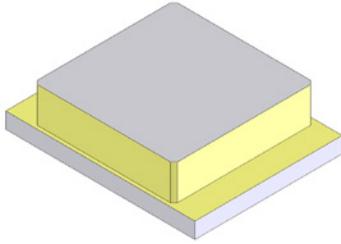
2. Machine the top face

Define a Face Milling operation to machine the top face of the electronic box part with the **Hatch** strategy.



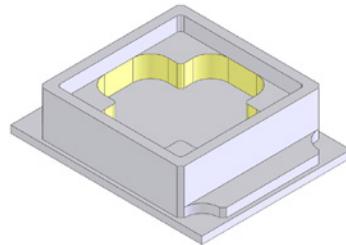
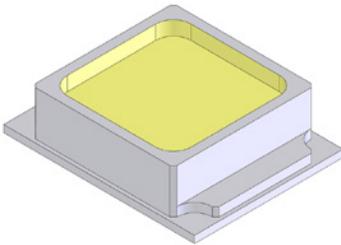
3. Machine the side faces

Define two Profile operations to perform rough machining of the external faces.



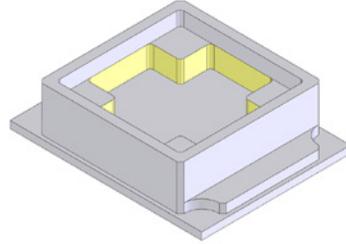
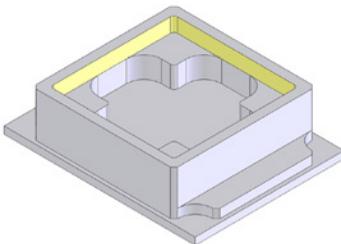
4. Machine the pocket

Define two Pocket operations to perform rough machining of the internal pocket faces with a tool of large diameter.

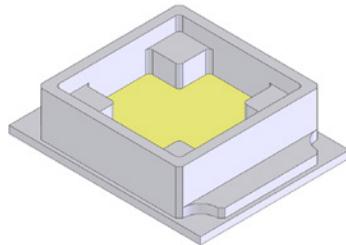
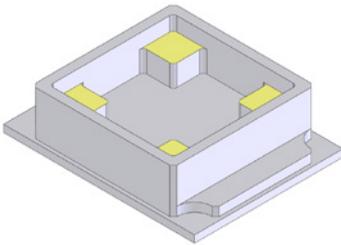


5. Perform the pocket finish machining

Define a number of Pocket operations to perform finish machining of the internal pocket faces. Choose the Tool according to the minimal radius of the model fillets.

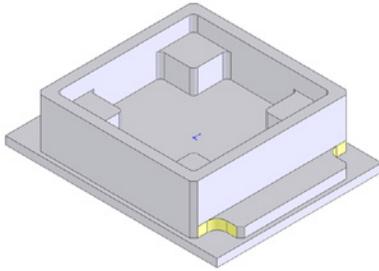


During the wall finish machining, use the **Rest Material** strategy to remove the bulk of material in the corners.

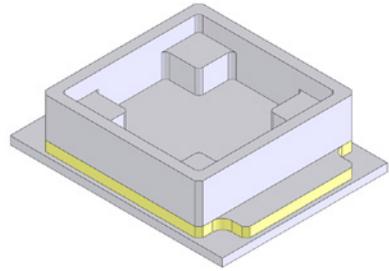
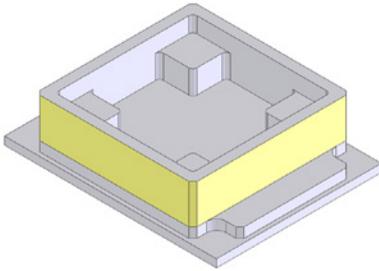


6. Perform finish machining of the external faces

At the first stage, perform the rest material machining of the corner faces to remove the bulk of the material.

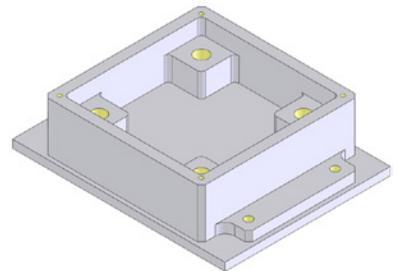


Define a number of Profile operations to machine the external faces.



7. Machine the holes

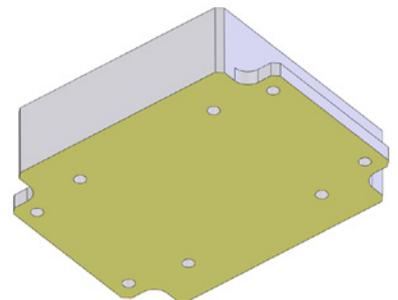
Define a number of Drilling operations to perform the center drilling, drilling, and tapping of the holes. Use a Pocket operation for counterboring of the holes.



8. Machine the bottom face

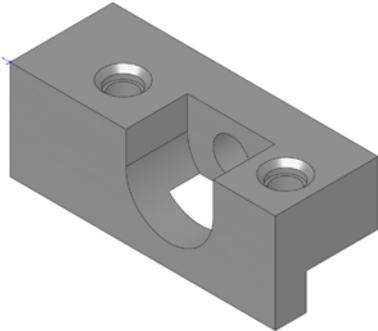
Define a Face Milling operation to machine the bottom face of the electronic box part using the **Hatch** strategy.

This operation requires different clamping, so define and use a new Coordinate System with the Z-axis oriented normally to the bottom face.



Exercise #6: Clamp Machining

Define the CAM-Part and operations for the machining of the clamp part presented on the illustration on a Milling CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Multi-sided machining using 2.5-axis CNC-machine
- Profile machining
- Open pocket machining
- Drilling

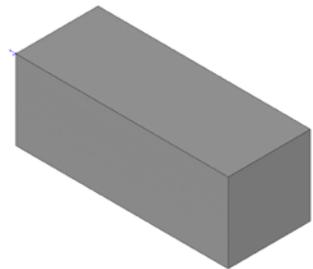
The SolidWorks model of the Clamp (**Exercise6.sldprt**) is located in the **Exercises** folder.

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

1. Define the CAM-Part

At this stage you have to define the CAM-Part, CNC-controller (**AWEA1000-Fanuc**), Machine Coordinate System, the Stock and Target models.

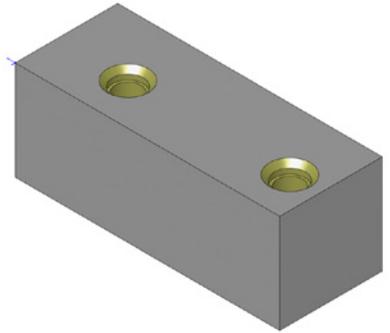
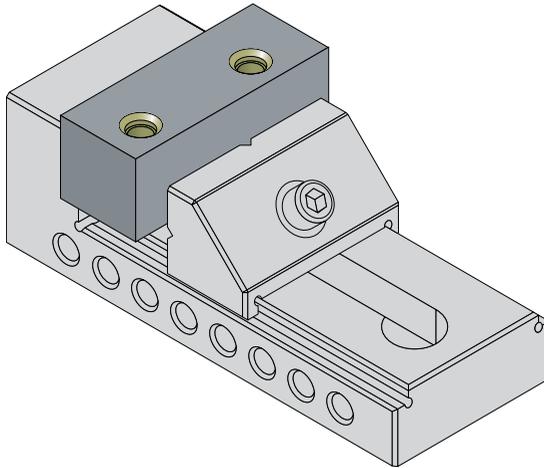
Make a note that the workpiece used for this part is the box exactly surrounding the clamp model without an offset.



2. Holes machining

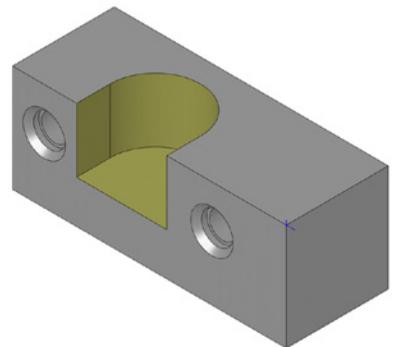
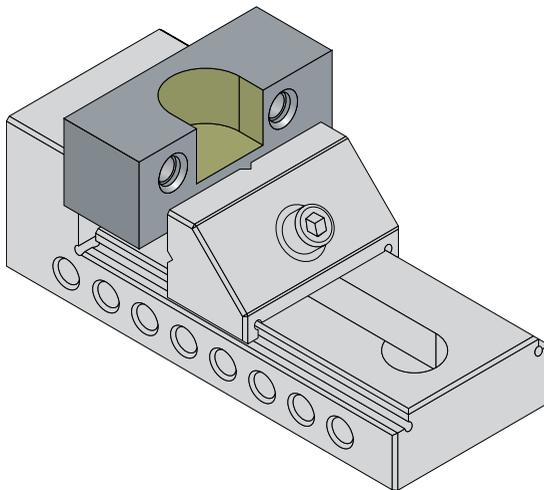
Perform a number of Drilling operations to machine the holes.

Use the appropriate Machine Coordinate System defined for the positioning shown below.



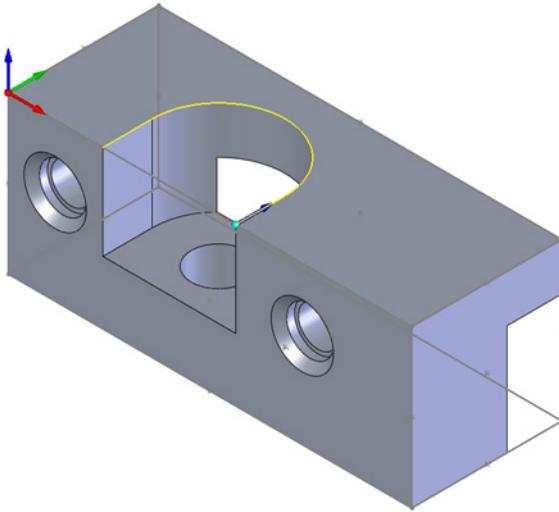
3. Open pocket machining

Add a Machine Coordinate System for the positioning shown below to machine the open pocket.

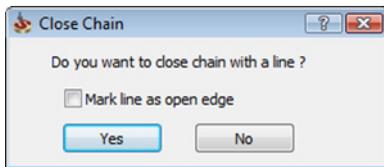


Add a new Pocket operation. In this operation, the open pocket machining strategy will be used.

Select the open pocket geometry chain as shown.



Click  to confirm the selected chain. The **Close Chain** dialog box is displayed.



Close Chain dialog box

This dialog box is displayed when you confirm an open chain definition in the **Geometry Edit** dialog box.

When you confirm this dialog box with the **Yes** button, the chain is closed with a line that connects the start and the end points of the chain. The **Mark line as open edge** check box enables you to mark the connecting line as open edge to perform open pocket machining.

When you click the **No** button, SolidCAM returns to the geometry definition and enables you to close the chain manually by the model element selection.

Select the **Mark line as open edge** check box and confirm the dialog box with **OK**. The open edge is marked.

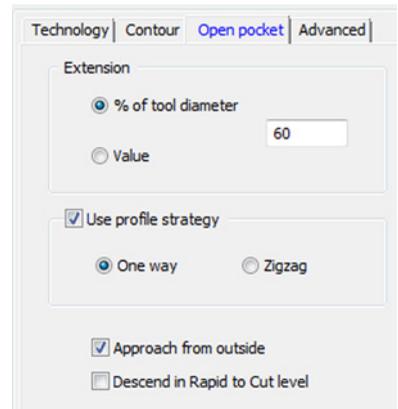
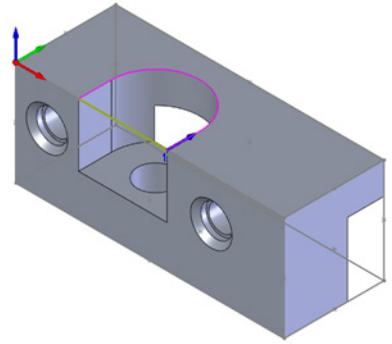
Confirm the **Geometry Edit** dialog box by clicking .

Choose the **Tool #4 (Ø6 End mill)** for the operation.

Select the upper face of the model as the **Upper level** and the pocket floor as the **Pocket depth**.

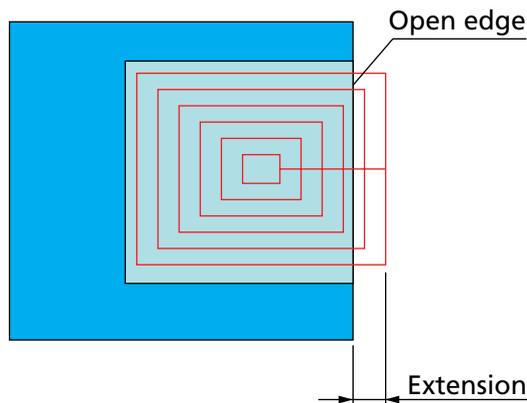
In the **Technology** page, only the **Contour** machining strategy is available. Switch to the **Open pocket** tab to define the parameters of the open pocket machining.

Define the extension of the tool path beyond the open edge.



Extension

This section enables you to define the overlapping between the tool and the open edges. The overlapping can be defined as percentage of the tool diameter (the **% of tool diameter** option) or as a value (the **Value** option).



Set the **Extension** value to **60% of the tool diameter**. Since the diameter of the tool used in this operation is 6 mm, the tool path is extended by 3.6 mm over the open pocket edge.



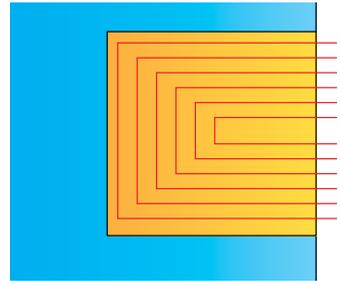
Note that the **Extension** value cannot be smaller than the radius of the tool.

Select the **Use profile strategy** check box.



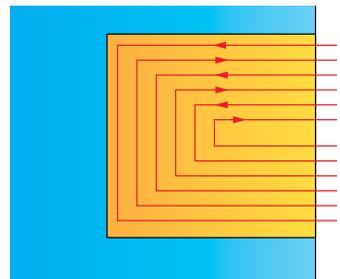
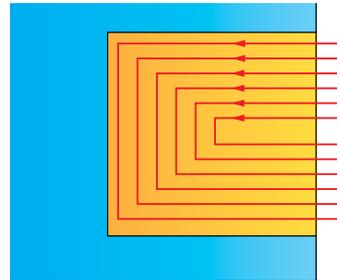
Use Profile Strategy

This option enables you to perform the open pocket machining in a Profile manner. The tool path at a specific Z-level consists of several equidistant profiles starting from outside the model at the distance defined by the **Extension** parameter. The tool moves on offsets parallel to the pocket geometry.



The following options enable you to define the tool path direction and linking:

- When the **One way** option is chosen, the tool finishes one profile pass, then rapidly moves (**GO**) to the safety distance and then to the start of the next cutting pass. The cutting direction (either climb or conventional) is preserved for each cutting pass.
- When the **Zigzag** option is chosen, the tool finishes one profile pass and then moves directly to the next pass. The machining is performed without leaving the material, thus constantly switching between climb and conventional milling.



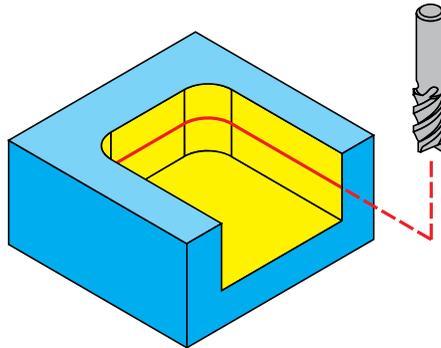
Use the default **One way** option.

Select the **Approach from outside** check box.



Approach from Outside

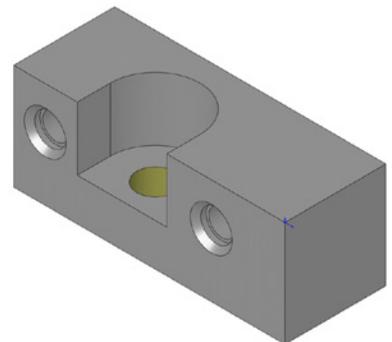
This option enables the tool to approach from outside of the material in the open pocket areas, if possible. Such approach enables you to decrease the tool loading when plunging into the material. This option enables you to perform the approach movement from the automatically calculated point outside of the material. The tool moves to the required depth outside of the material and then plunges into the material.



Switch to the **Technology** tab. In the **Offsets** section, set the **Wall offset** and the **Floor offset** to **0.2**. To remove these offsets with the finishing cut, select the **Wall** and **Floor** check boxes in the **Finish** section.

4. Hole machining

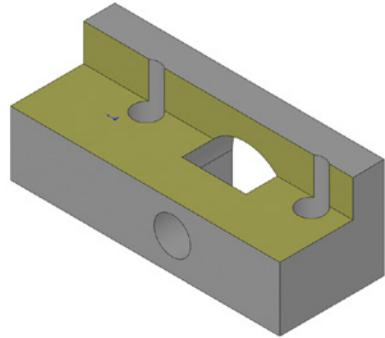
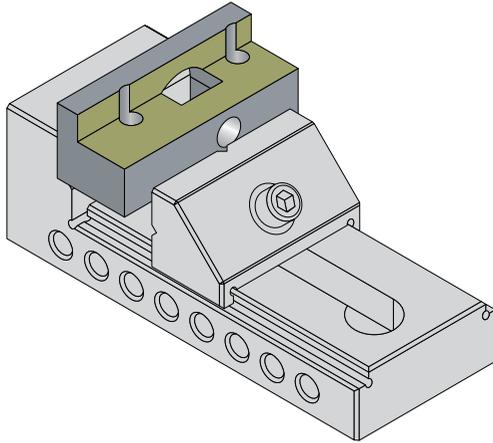
Define two Drilling operations to perform center drilling and drilling of the hole located at the bottom of the open pocket. Use the same Machine Coordinate System (positioning) as in the previous operation.



5. Step machining

Define a Profile operation to machine the step.

Define the Machine Coordinate System for the positioning shown below.



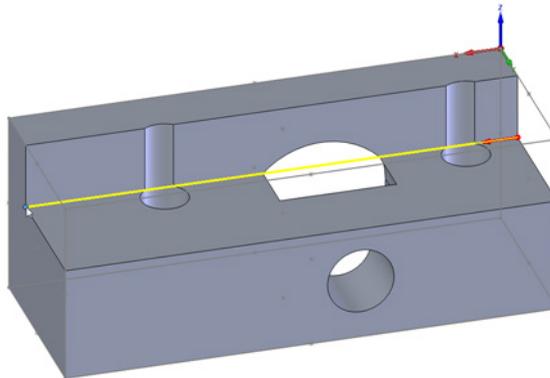
Use the **Point to point** option in the **Geometry Edit** dialog box to define the geometry.



Point to Point Mode

This option enables you to connect the specified points. The points are connected by a straight line. This option is useful when it is impossible to define the geometry using the model edges.

In this exercise, it is impossible to define the straight geometry for the Profile Milling using the model edges. The **Point to Point** option enables you to define the geometry by two points located on the model vertices.

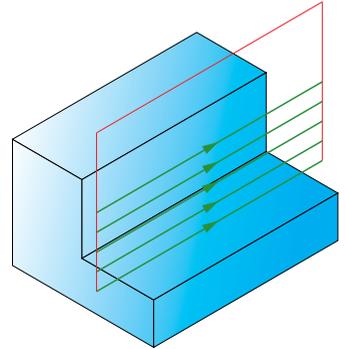


Use the **Zigzag** cutting type to connect the successive Z-level passes directly from the end of a pass to the beginning of the next pass.

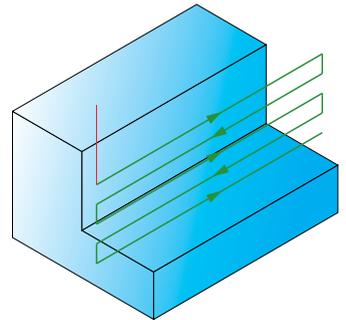


Depth Cutting Type

When the **One Way** option is chosen, the cutting passes are oriented in the same direction, and the connection between them is performed through the **Operation Clearance level**. At the end of each pass the tool performs a retreat movement to the **Operation Clearance level**, a horizontal movement at rapid feed to the start point of the next pass and then descends to the Z-level of the next pass. The same cutting direction (climb or conventional) is kept along the whole tool path.



When the **Zigzag** option is chosen, the tool path is performed in a zigzag manner, with the tool path direction changing from one pass to the next. The passes located at two successive Z-levels are connected directly from the end of one pass to the beginning of the next pass.



The **Zigzag** option cannot be used together with the **Clear offset** technology.

Exercise #7: Basic Part Machining

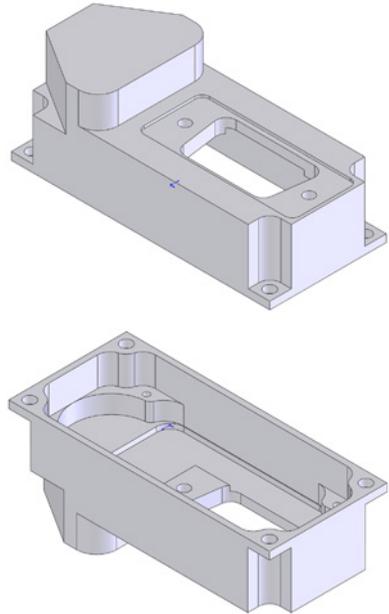
Define the CAM-Part and operations for the machining of the basic part presented on the illustration on a Milling CNC-machine.

This exercise reinforces the following skills:

- CAM-Part definition
- Pocket machining
- Profile machining
- Rest material machining
- Face milling
- Drilling

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

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

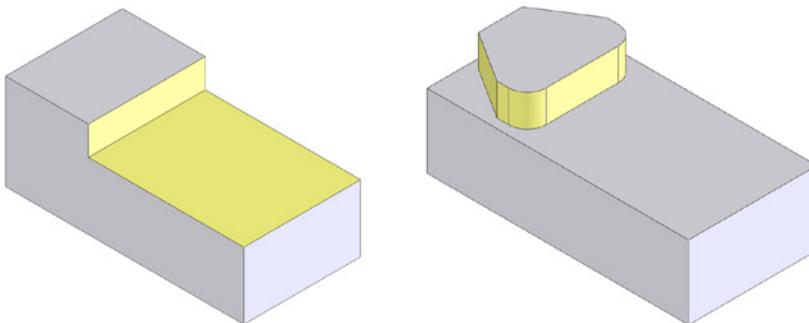


1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**AWEA1000-Fanuc**), Machine Coordinate System, the Stock and Target models.

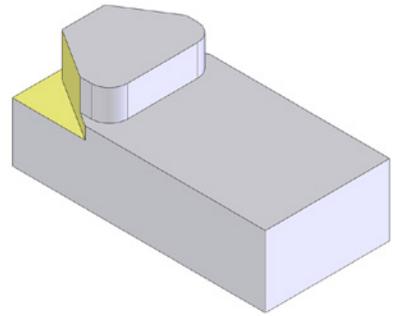
2. Upper cusp machining

Define two Profile operations to remove the bulk of material and perform the rough and the finish machining of the cusp.



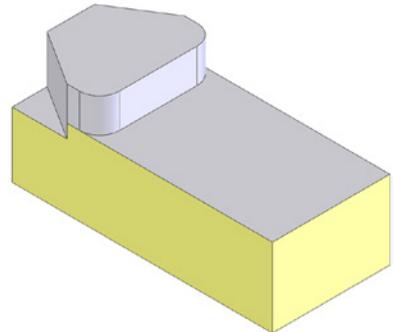
3. Step machining

Perform the rough and finish machining of the step faces using the Profile operation.



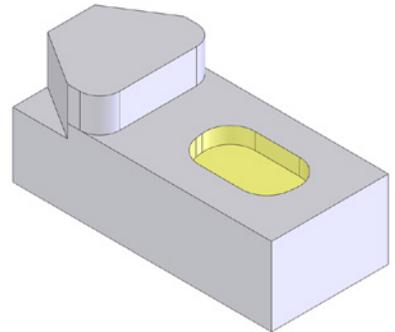
4. External contour machining

Define a Profile operation to perform the rough and finish machining of the external model faces.



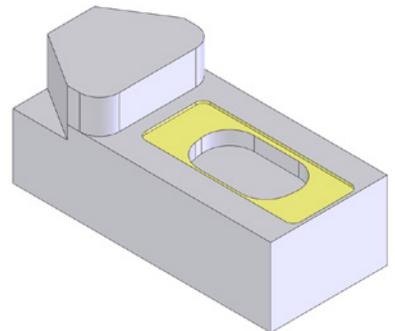
5. Connector opening machining

Define a Pocket operation to perform the rough machining of the connector opening faces.



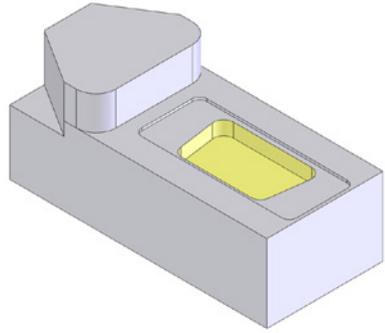
6. Connector pocket machining

Define a number of Pocket and Profile operations to perform the roughing and finishing of the connector pocket.



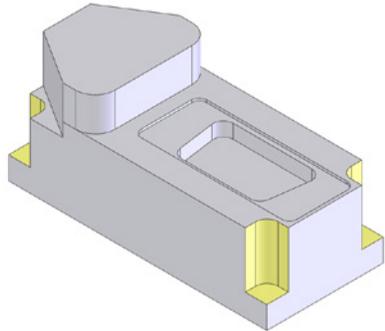
7. Connector opening finish machining

Use the Pocket rest material strategy to perform the finish machining of the connector opening faces.



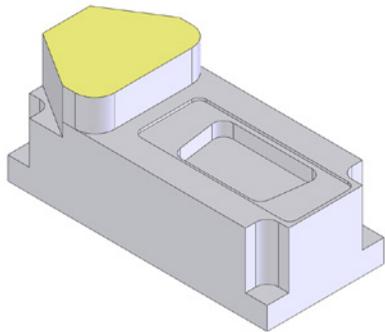
8. Machine screw head areas

Define a Profile operation to perform the rough and finish machining of the screw head areas.



9. Top face machining

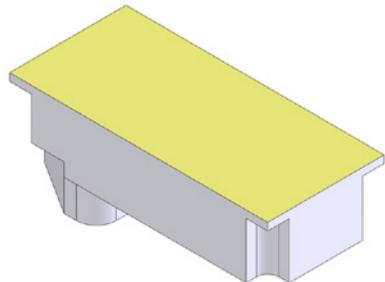
Use the Face milling **Hatch** strategy to perform the rough and finish machining of the top face.



10. Bottom face machining

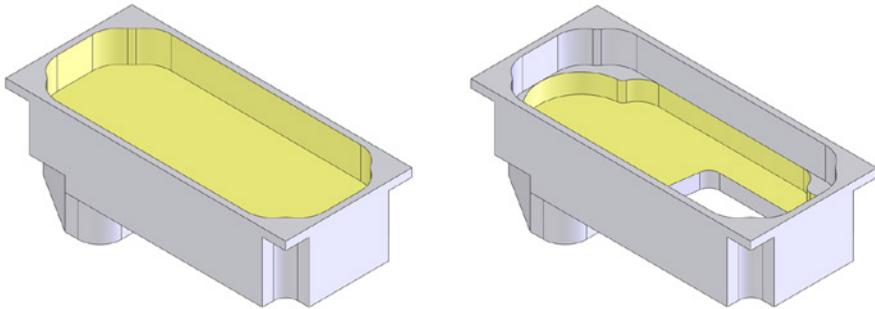
Use the Face milling **Hatch** strategy to perform the rough and finish machining of the bottom face of the Cover part.

Note that this operation and the following ones use a new positioning and therefore you have to define a new Machine Coordinate System.



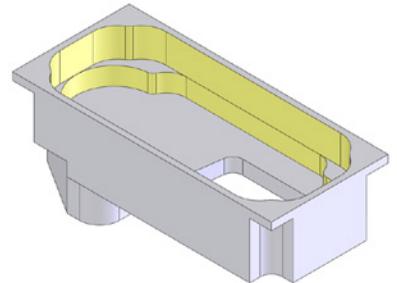
11. Pocket rough machining

Use a Pocket operation to perform the rough machining of the highlighted faces.



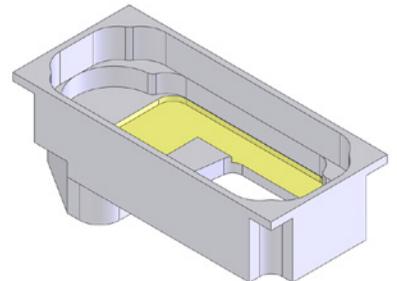
12. Wall finish machining

Define two Profile operations to perform the finish machining of the wall faces of the pockets.



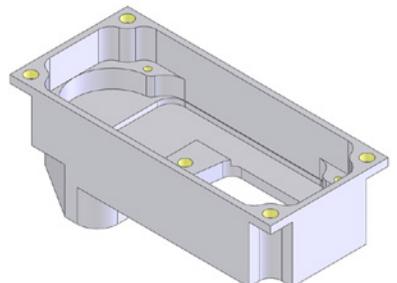
13. Bottom pocket machining

Perform the rough and finish machining of the bottom pocket with the island.



14. Hole machining

Define a number of Drilling operations to machine holes located on the cover part faces.



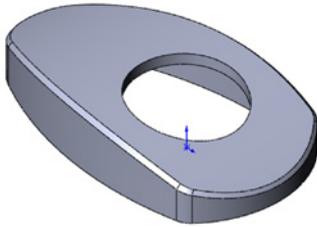
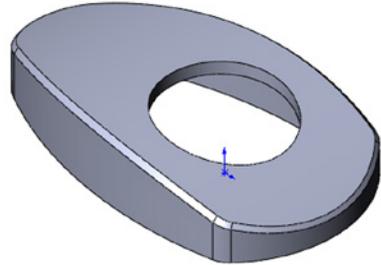
Exercise #8: Cover Machining

Machine the hole and chamfer on the pre-machined cover part presented on the illustration on a Milling CNC-machine.

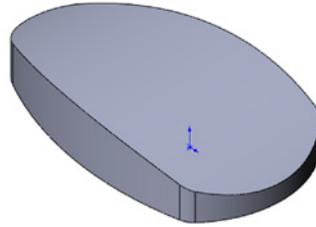
This exercise reinforces the following skills:

- 3D contour machining

The part file (**Exercise8.prz**) is located in the **Exercises** folder. This file contains the defined CAM-Part. The model used for the part definition contains two configurations: **Default** (model with the hole and chamfer features) and **Stock** (hole and chamfer features suppressed).



Default



Stock

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



1. Open the CAM-Part

Open the **Exercise8.prz** file located in the **Exercises** folder.



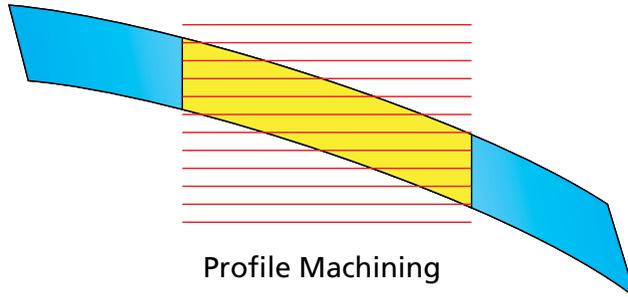
2. Add a Contour 3D operation

Add a new Contour 3D operation to perform machining of the hole on the upper face of the part.

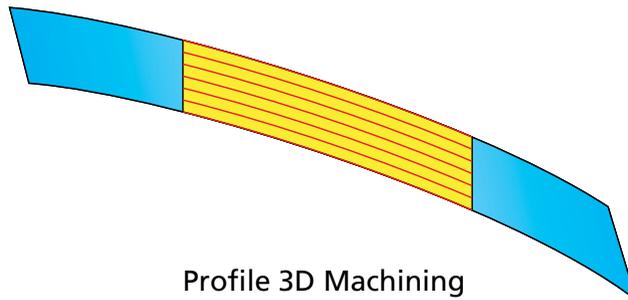
The **Contour 3D Operation** dialog box is displayed.



A through hole located on a convex surface can be machined in several methods, such as Profile machining and Contour 3D machining. Profile milling used for machining of such holes results in a great number of tool path passes with unavoidable air cutting, because it yields a planar tool path parallel to the XY-plane. Contour 3D machining enables you to generate a curved tool path in three-dimensional space. Therefore, in this exercise the **Contour 3D** operation is used for hole machining to optimize the tool path and avoid redundant tool movements.



Profile Machining

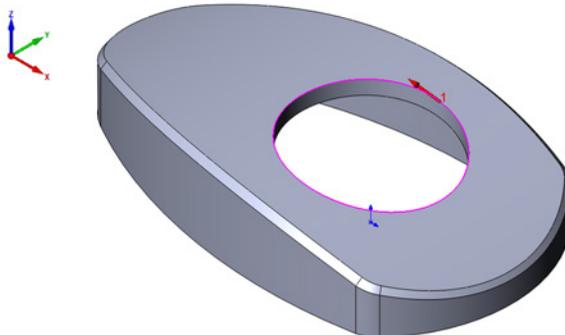


Profile 3D Machining



3. Define the Geometry

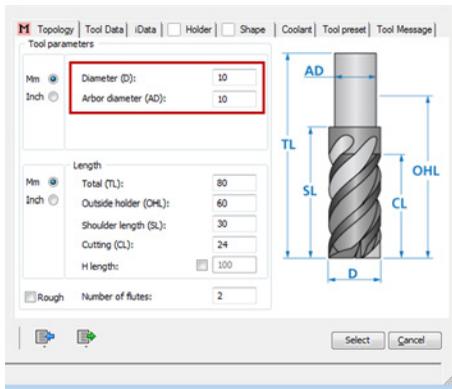
Define the geometry chain on the upper edge of the hole as shown.





4. Define the Tool

Add a new **Ø10 End mill** tool for the operation.

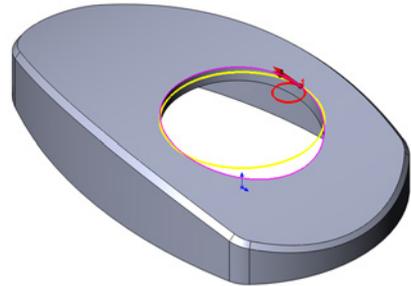


5. Define the technological parameters

In the **Modify** section, click the **Geometry** button to check the position of the tool relative to the geometry.



In this case, the default **Left** option set for **Tool side** meets the requirements of the climb milling. Close the **Modify Geometry** dialog box by clicking the  button.



In the **Offsets** section, set the **Wall offset** value to **0.2**. The allowance of 0.2 mm

will be left on the hole wall during the roughing. This allowance will be removed with a separate finishing cut in the end of the machining.

Select the **Rough** check box. Set the **Step down** to **2**. The hole will be machined in three equidistant rough passes.

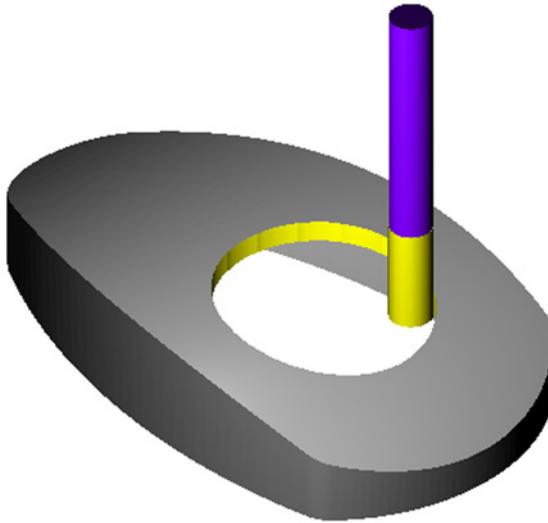
Select the **Finish** check box. Set the **Step down** to **6**. The finishing will be performed in a single pass.

**6. Define the Lead in and Lead out**

For **Lead in**, choose the **Arc** option and use the default **Radius** value of **2**. For **Lead out**, select the **Same as Lead in** check box.

**7. Simulate the operation**

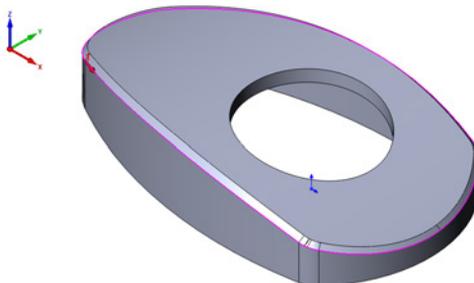
Perform the simulation of the operation in the **SolidVerify** mode.

**8. Add a Contour 3D operation**

Add a new Contour 3D operation to perform chamfering of the upper face of the cover.

**9. Define the Geometry**

Define the geometry chain on the upper edge of the face as shown.





10. Define the Tool

Add a new **Ø10 Spot drill** tool for the operation.



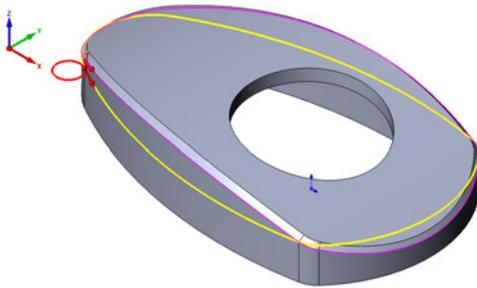
11. Define the Milling levels

In the **Levels** page, set **Depth** to **0**.



12. Define the technological parameters

In the **Modify** section of the **Technology** page, set the **Tool side** to **Right**. Click the **Geometry** button to check the tool position relative to the geometry.

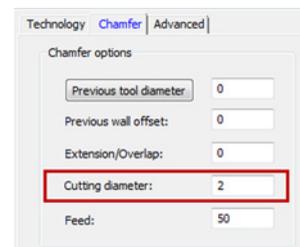
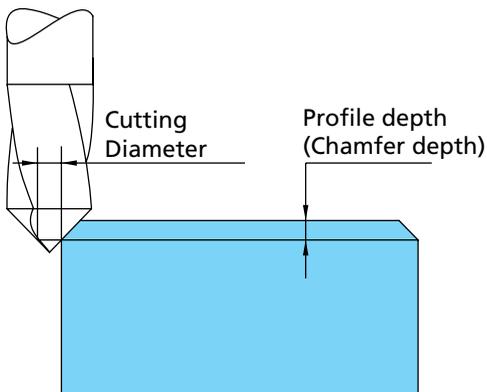


Close the **Modify Geometry** dialog box by clicking .

Select the **Rough** check box. Set the **Step down** to **0**.

In the **Chamfer** section, select the check box and switch to the **Chamfer** tab.

Set the **Cutting diameter** value to **2**.

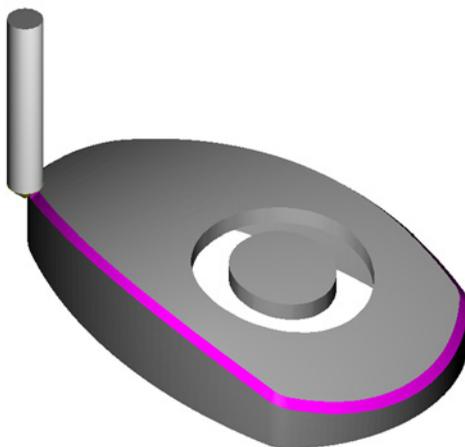


**13. Define the Lead in and Lead out**

For **Lead in**, choose the **Arc** option and use the default **Radius** value of **2**. For **Lead out**, select the **Same as Lead in** check box.

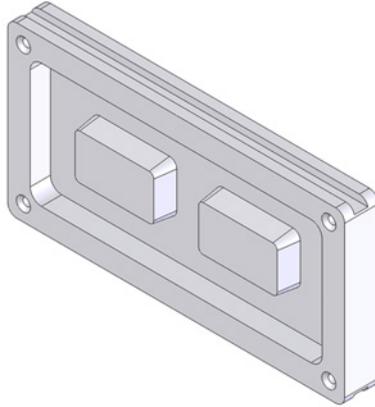
**14. Simulate the operation**

Perform the simulation of the operation in the **SolidVerify** mode.



Exercise #9: Mounting Base Machining

Define the CAM-Part and operations for the machining of the mounting base presented on the illustration on a Milling CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Pocket machining
- Drafted profile machining
- T-slot milling
- Drilling

The SolidWorks model of the Mounting base (**Exercise9.sldprt**) is located in the **Exercises** folder.

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



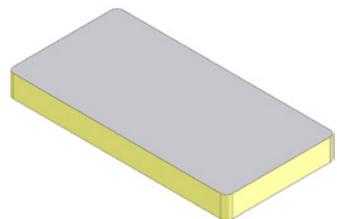
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**Awea1000-Fanuc**), Machine Coordinate System, the Stock and Target models.



2. Contour machining

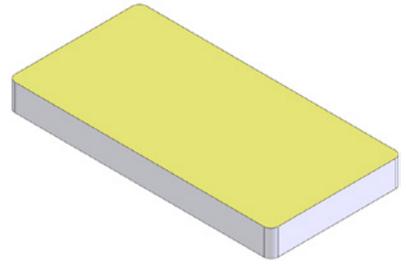
Define a Profile operation to machine the mounting base external faces.





3. Top face machining

Define a Face Milling operation to machine the top face of the mounting base.

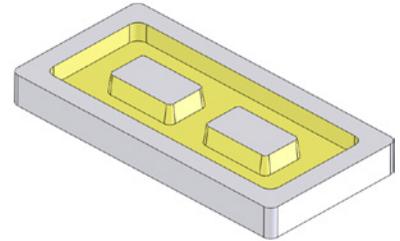


4. Pocket machining

Define a Pocket operation to perform rough and finish machining of the pocket faces.



Use the lower edges of the islands for the internal chains definition, because the walls are drafted.



Use a **Bull nose tool** of $\varnothing 8$ with **Corner radius** of **0.2** mm for machining the pocket.



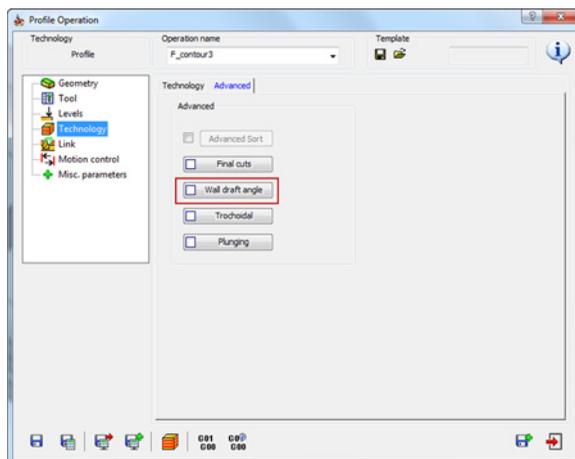
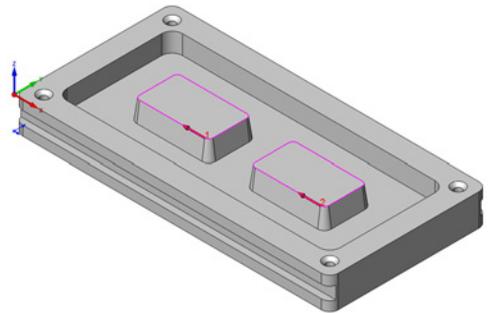
5. Profile machining for islands

Define a Profile operation for two draft angled walls at the islands. Use the **Bull nose tool** of $\varnothing 8$ already defined in **Step #4**.

Define the geometry as shown.

Select the **Finish** option in the **Technology** page and set the **Step down** value to **0.1**.

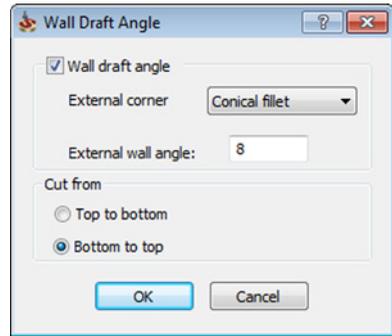
Switch to the **Advanced** tab and click the **Wall draft angle** button.



In the **Wall Draft Angle** dialog box, check the same option.

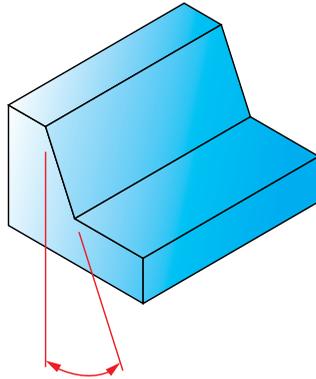
Set the **External wall angle** to **8**.

In the **Cut from** area select the **Bottom to top** option.



Wall Draft Angle

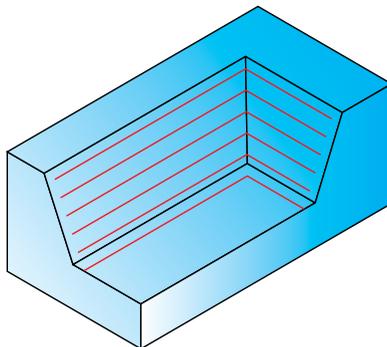
When this check box is selected, the inclined wall machining is performed.



- **External wall angle**

This parameter defines the draft angle of the wall measured from the Z-axis direction as shown.

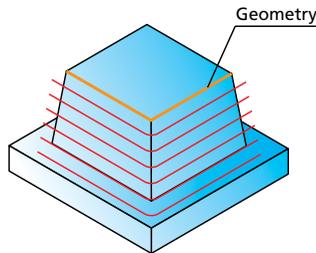
For the inclined wall machining, each cutting pass located at a specific Z-level is generated according to the specified **External wall angle** value.



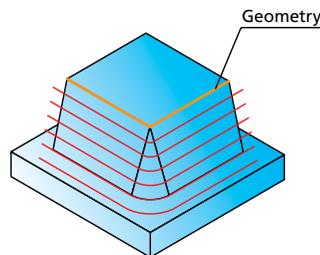
- **External corner type**

This option enables you to define how the cutting passes are connected during the external corners machining. There are three ways to connect between the cutting passes:

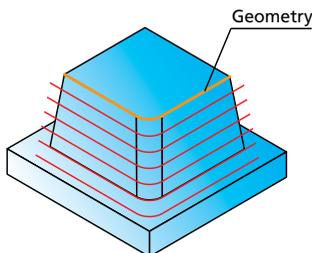
Sharp corner: the tool path is calculated to perform the machining of a sharp corner. This option affects only the machining of geometry sharp corners.



Conical fillet: the tool path is calculated to perform the machining of the corner with a conical fillet; the radius of the tool path rounding increases from one pass to the next. This option affects only the machining of sharp geometry corners.



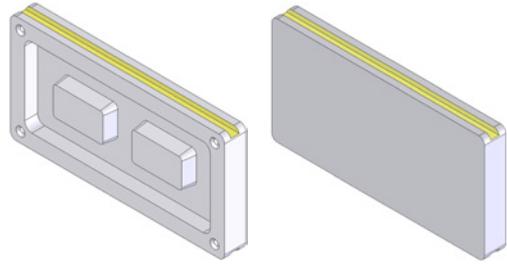
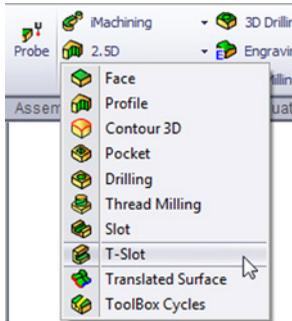
Cylindrical fillet: this option affects the machining of both of sharp and filleted corners of the geometry. The machining of sharp corners is performed similar to the Conical fillet option. For the filleted corners, the tool path is calculated to perform the machining of the corner with a cylindrical fillet; the radius of the tool path rounding is the same for all the cutting passes.





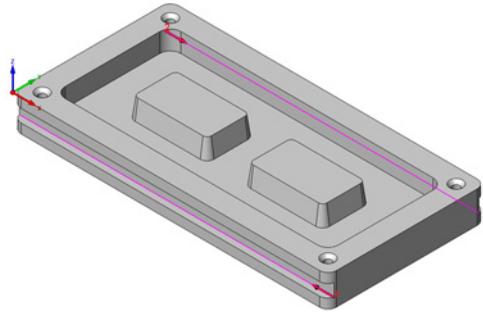
6. T-Slot machining

In **SolidCAM Operations** toolbar, select **2.5D > T-Slot**.



The **T-Slot Operation** dialog box is displayed.

Define two geometry chains as shown. Make sure the direction of the chains geometry is the same as on the illustration.



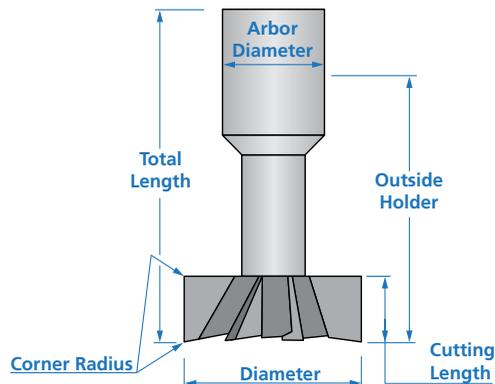
Add a new **Slot mill** tool.



Slot Mill

This type of tool can be used for all types of machining from simple 2.5D undercuts up to 5-axis simultaneous machining.

A tool of this type is defined with the parameters shown in the image.



Edit the tool parameters as follows:

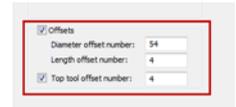
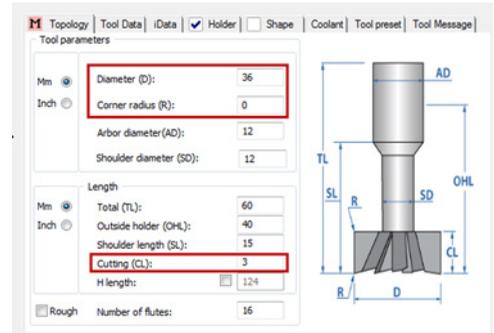
- Set **Diameter** to **36**
- Set **Corner radius** to **0**
- Set **Cutting length** to **3**

Add the **BT40 ER32x60** tool holder.

Confirm the tool definition by clicking the **Select** button.

In the **Tool** page, click the **Data** tab.

Select the **Offsets** option and the **Top tool offset number** check box.



The **Top Tool Offset Number** defines the register number of the upper cutting edge of the T-slot mill. This option enables SolidCAM to automatically take into account the minor size differences between the defined tool and the one actually used for cutting the workpiece, if there are any.

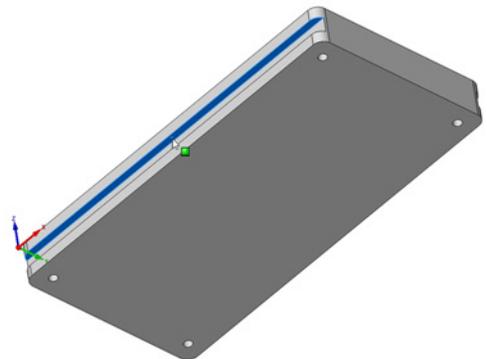
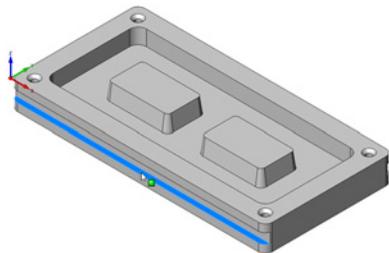


7. Define the Milling levels

Define the **Upper level** for slot machining by clicking on the model face as shown.

Confirm the value (**-4.5**) in the **Pick Upper level** dialog box.

Define the **T-slot depth** of the T-slot by clicking on the model as shown.



The depth value calculated in the **T-slot depth** area is **6**.



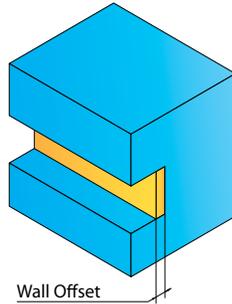
8. Define the technological parameters

Define the rough parameters in the **Technology** page. To perform roughing of the slot, select the **Rough** check box.

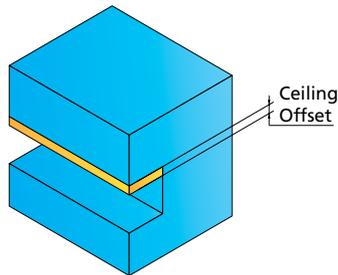
To perform the rough machining of the slot, you have to specify offsets on the slot geometry. These offsets are removed with the finishing cut.



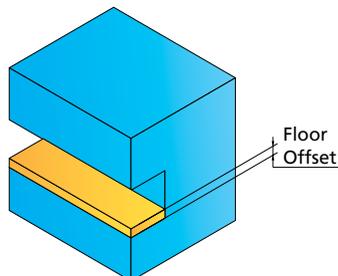
The **Wall offset** option defines the offset that remains on the slot wall after roughing.



The **Ceiling offset** option defines the offset that remains on the slot ceiling after roughing.



The **Floor offset** option defines the offset that remains on the slot floor after roughing.





Set all of the offset to **0.2**

Offsets	
Wall offset:	0.2
Ceiling offset:	0.2
Floor offset:	0.2

Define the finishing parameter.

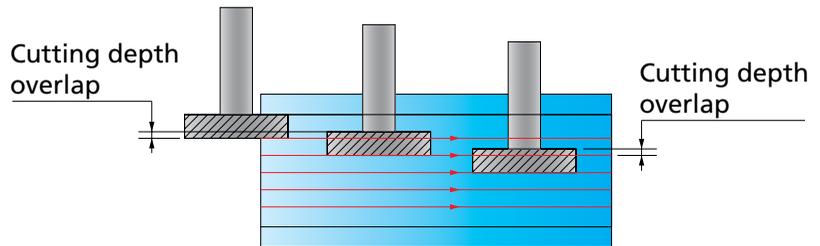
Select the **Finish** check box. Notice that the **Ceiling**, **Wall** and **Floor** check boxes are selected by default. These check boxes define the offsets (Ceiling, Wall and Floor, respectively) to be removed with the finishing pass. Set the **Cutting depth overlap** value for finishing to **2**.

Finish		
<input checked="" type="checkbox"/> Ceiling	<input checked="" type="checkbox"/> Wall	<input checked="" type="checkbox"/> Floor
Number of passes:	1	
Extension/Overlap:	0	
Cutting depth overlap:	2	
Cut from:	Top to bottom	



Cutting depth overlap

This parameter defines the overlap of each two adjacent tool paths in both rough and finish machining of the slot.



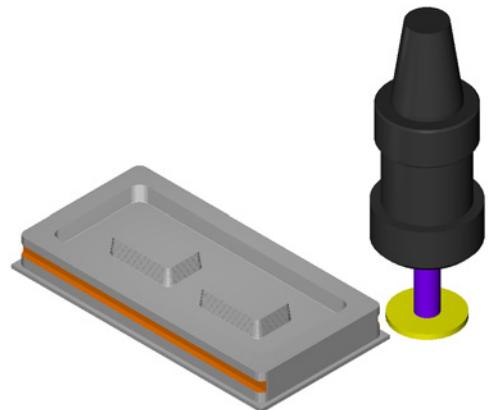
9. Define the Lead in and Lead out

For **Lead in**, select the **Tangent** option and set the **Tangent extension** to **12**. For **Lead out**, select the **Same as Lead in** check box.



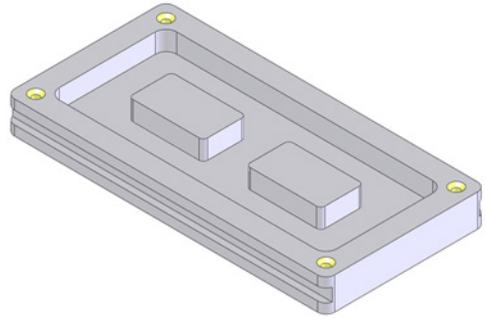
10. Simulate the operation

Perform the simulation of the operation in the **SolidVerify** mode.



11. Holes machining

Define several Drilling operations to perform the machining of the countersink holes located on the top face of the mounting plate.

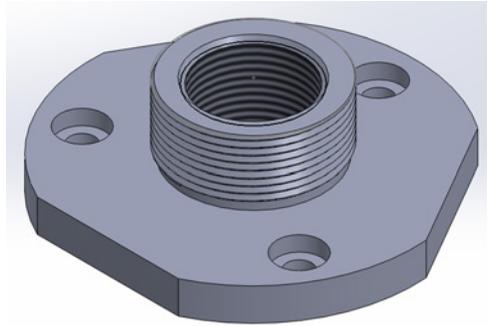


Exercise #10: Support Machining

Machine the threads on the pre-machined support part presented on the illustration on a Milling CNC-machine.

This exercise reinforces the following skills:

- Threading



The part file (**Exercise10.prz**) is located in the **Exercises** folder. This file contains the defined CAM-Part.

The following steps have to be implemented in order to perform the internal and external threading:



1. Open the CAM-Part

Open the **Exercise10.prz** file located in the **Exercises** folder.



2. Add a Thread Milling Operation

Add a new Thread Milling operation to perform threading of the internal surface of the part.

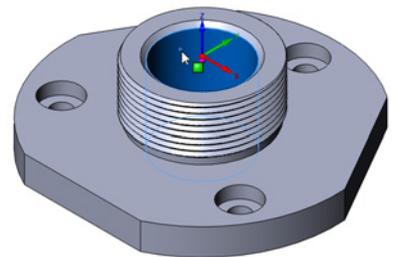
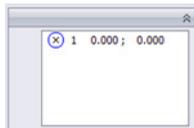
The **Thread Milling Operation** dialog box is displayed.



3. Define the Geometry

Define the geometry by clicking on the internal surface of the part.

The thread milling position is selected, and its coordinates are displayed in the **Drill Geometry Selection** dialog box.





4. Define the Tool

Switch to the **Tool** page of the **Thread Milling Operation** dialog box and click the **Select** button.

The **Part Tool Table** dialog box is displayed.

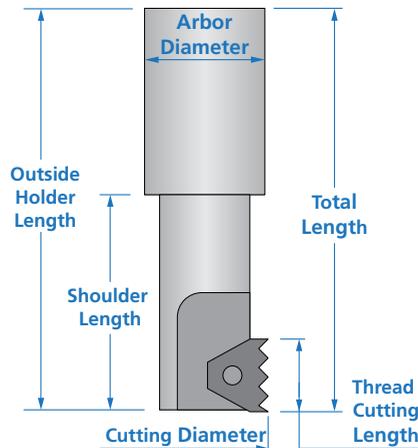
Click  to start a new threading tool definition. The available tools are displayed.

Choose the **Thread Mill** tool for the operation.



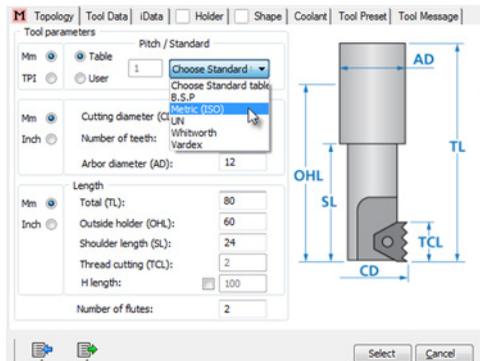
Thread Mill

This tool type is used for threading in Thread Milling operations. A tool of this type is defined with the parameters shown in the image.

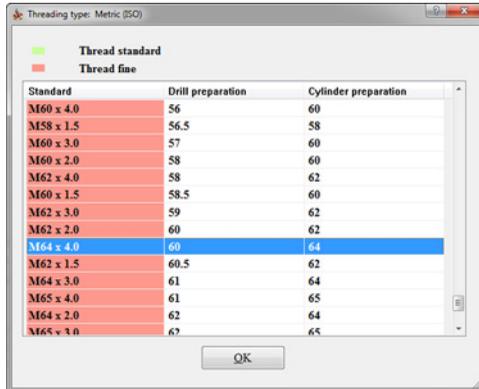


In the **Pitch/Standard** area under **Tool Parameters**, click the arrow near **Choose standard table** and select the **Metric (ISO)** option from the list.

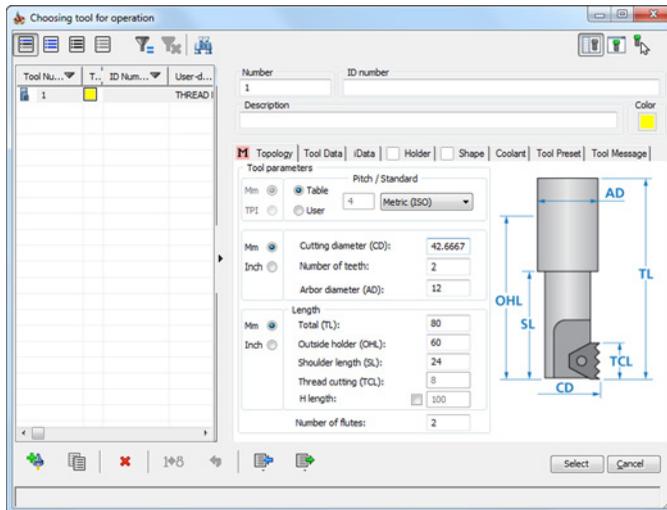
The **Threading Type: Metric (ISO)** dialog box is displayed.



Select the **M64 x 4.0** standard from the list and click **OK**.



Use the default tool parameters.

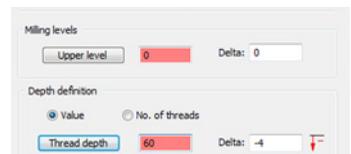


5. Define the Milling levels

Define the operation **Upper level** directly on the solid model by clicking on the top of the part.

Define the **Thread depth** by clicking on the bottom face of the part.

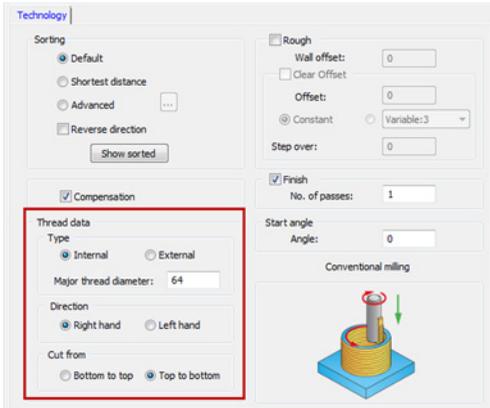
In the **Depth definition** area, set the **Delta** value to **-4**.





6. Define the technological parameters

In the **Technology** page, make sure the default **Internal** type is chosen in the **Thread Data** area. Set the **Major thread diameter** value to **64**. In the **Cut from** section, select **Top to bottom**.



7. Define the Lead in

Switch to the **Link** page. Under **Lead in/out arc**, clear the **Start from the center** check box.



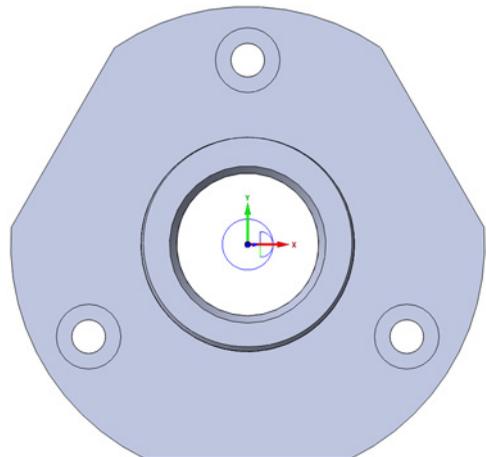
Start from the center

This option enables you to start threading from the center of the geometry. When this option is cleared, the tool starts machining from a safety distance, which allows to avoid air cutting and save machining time.

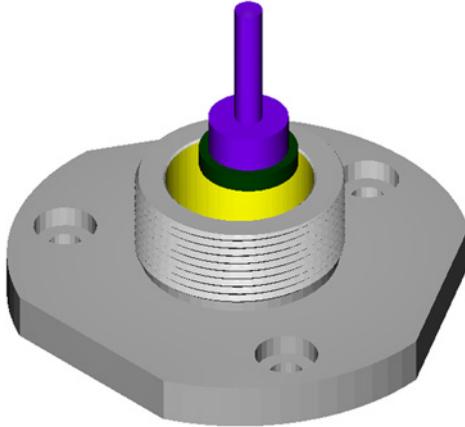


8. Simulate the operation

Perform the simulation of the operation in the **Host CAD** mode. Set the CAM-Part in the **Top** view to observe that the tool approaches the material not in the center.



Perform the simulation of the operation in the **SolidVerify** mode.



9. Add a Thread Milling Operation

Add a new Thread Milling operation to perform small internal threads.



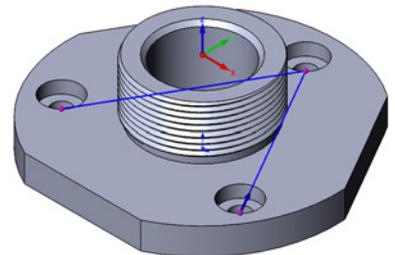
10. Define the Geometry

Define the geometry by clicking on the upper surface of the model containing the holes.



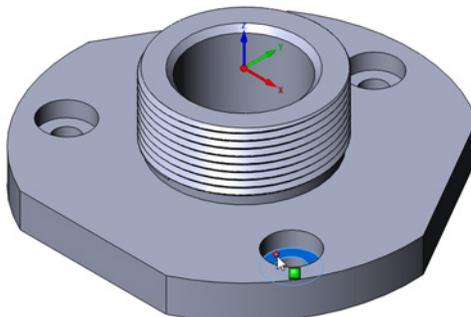
11. Define the Tool

Define a new **Thread mill** selecting the **M14x1.5** standard from the **Metric (ISO)** table.



12. Define the Milling levels

Define the operation **Upper level** directly on the solid model by clicking on the small pocket surface as shown.

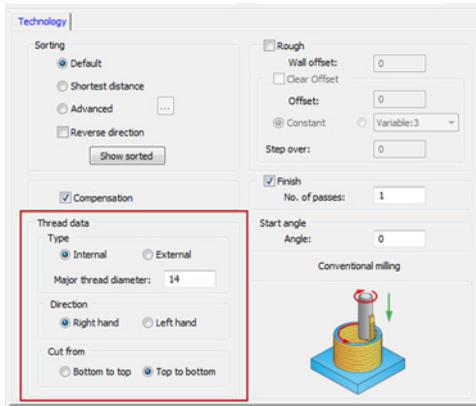


Define the **Thread depth** by clicking on the model bottom face.



13. Define the technological parameters

In the **Technology** page, make sure the default **Internal** type is chosen in the **Thread data** area. Set the **Major thread diameter** value to **14**. In the **Cut from** section, select **Top to bottom**.



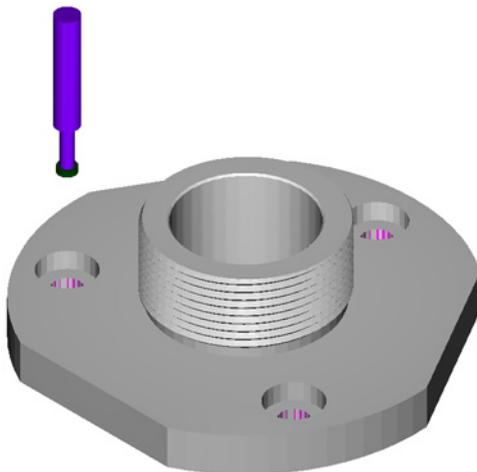
14. Define the Lead in

Switch to the **Link** page. Under **Lead in/out arc**, make sure that the **Start from the center** check box is selected.



15. Simulate the operation

Perform the simulation of the operation in the **SolidVerify** mode.





16. Add a Thread Milling operation

Add a new Thread Milling operation to perform threading of the external surface of the part.



17. Define the Geometry

Use the **drill** geometry defined in the first operation.

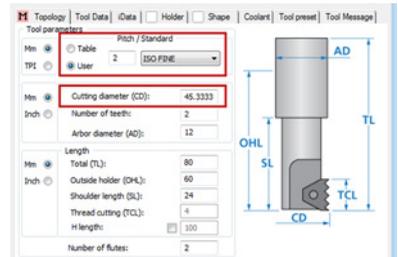


18. Define the Tool

Define a new **Thread mill**.

In the **Tool parameters** area, under **Pitch / Standards**, select **User** and set **3** for the **Pitch** value. From the list, select the **ISO FINE** standard.

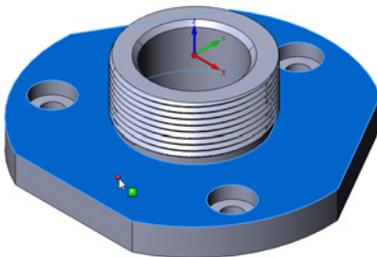
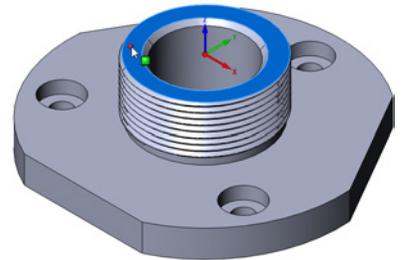
- Set the **Cutting diameter** to **45.3333**



19. Define the Milling levels

Define the operation **Upper level** directly on the solid model by clicking on the top face of the part as shown.

Define the **Thread depth** by clicking on the model surface as shown.

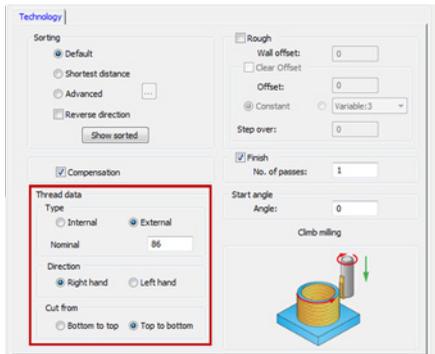




20. Define the technological parameters

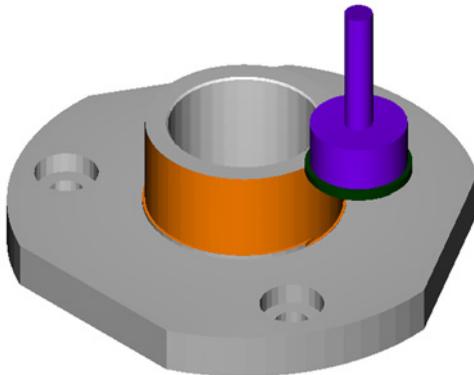
In the **Technology** page, choose the **External** type in the **Thread data** area. Set the **Major thread diameter** value to **86**.

In the **Cut from** area, choose the **Top to bottom** option.



21. Simulate the operation

Perform the simulation of the operation in the **SolidVerify** mode.



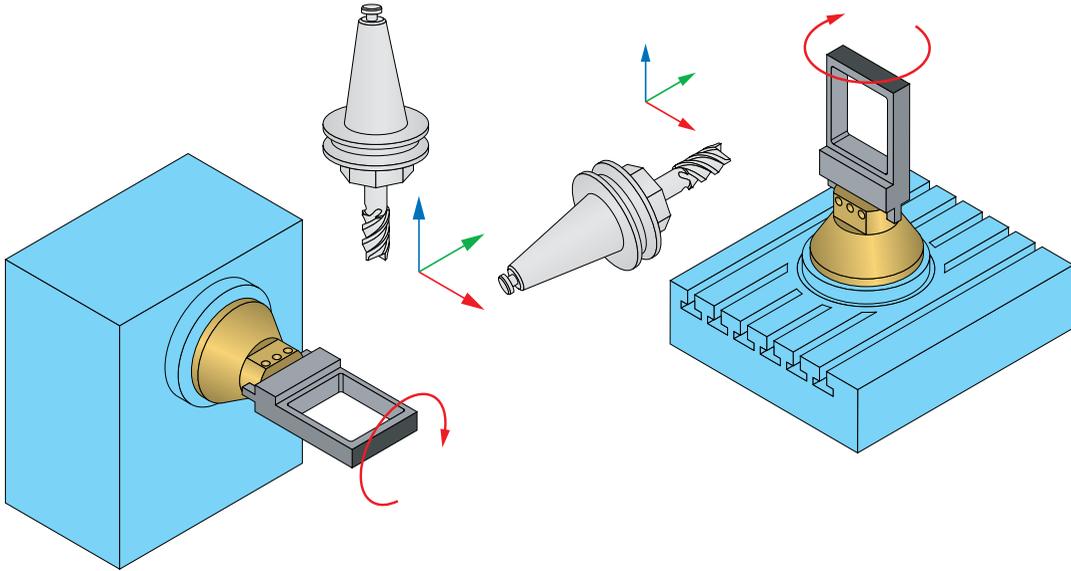
At this stage, the exercise is completed.

Indexial 4-Axis Milling

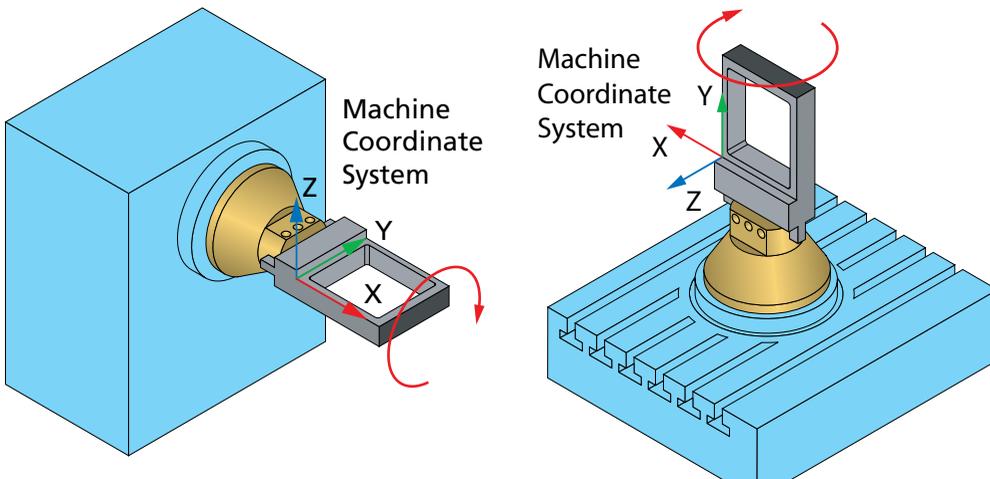
4

SolidCAM enables you to perform indexial 4-axis milling on vertical or horizontal CNC-machines with a 4th axis rotary table.

The 4th axis rotation is used for the positioning of the part. When the part is positioned to the required orientation, the 4th axis is locked and the machining is performed using 3-axis movements.

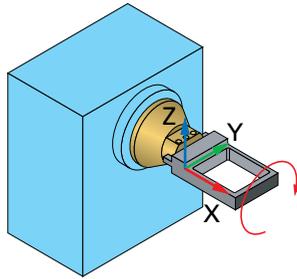


For each clamping, SolidCAM enables you to define the Machine Coordinate System with X-axis (for vertical machines) or Y-axis (for horizontal machines) parallel to the revolution axis of the rotary table.

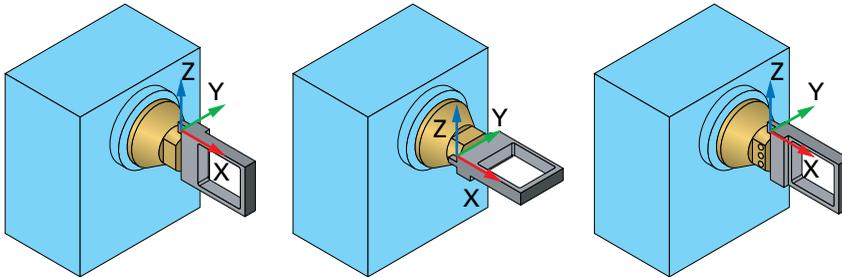


Within one clamping (Machine Coordinate System), SolidCAM enables you to define a number of positions around the revolution axis. For vertical machines the X-axis of these positions is parallel to the X-axis of the Machine Coordinate System.

Machine Coordinate System

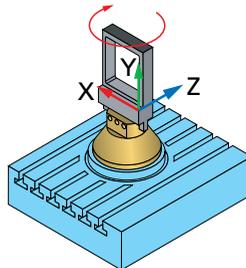


Positions

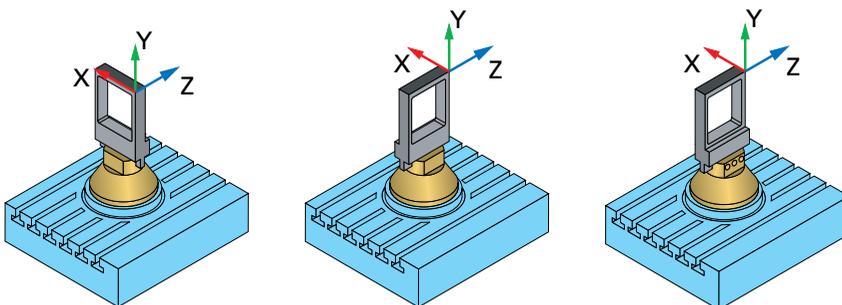


For horizontal machines, the Y-axis of these positions is parallel to the Y-axis of the Machine Coordinate System.

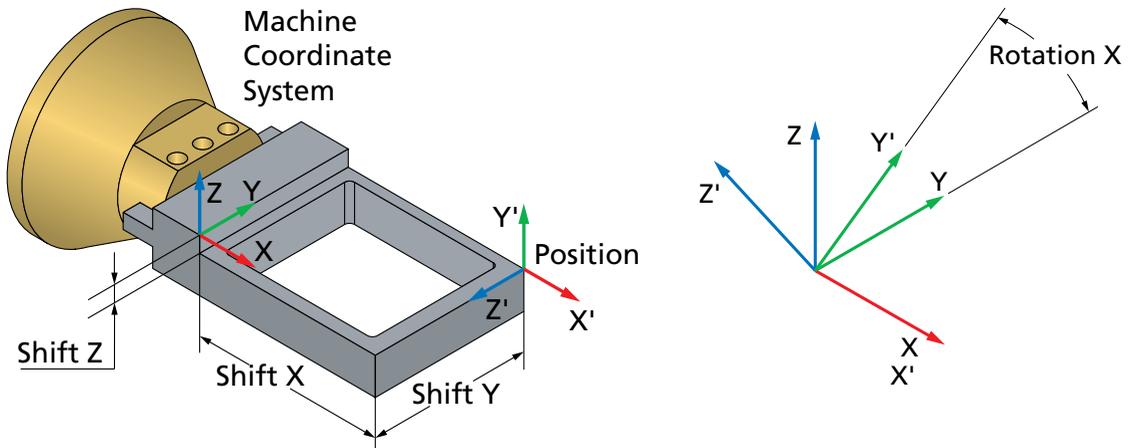
Machine Coordinate System



Positions



For each position, SolidCAM automatically determines the **Shift** and the **Rotation** parameters relative to the Machine Coordinate System.



The **Shift** components (**Shift X**, **Shift Y** and **Shift Z**) describe the shifting of the position relative to the Machine Coordinate System.

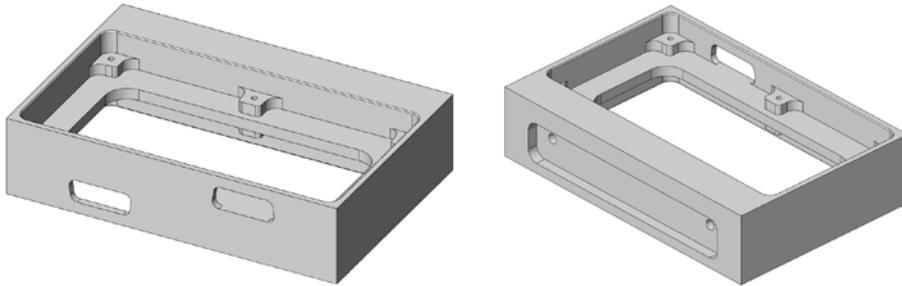
The **Rotation** components (**Rotation around X**, **Rotation around Y** and **Rotation around Z**) describe the axis orientation of the position relative to the Machine Coordinate System.

Positions can be defined only around the revolution axis. For vertical machines only positions around X-axis can be added, so the **Rotation around Y** and **Rotation around Z** values are **0**. The **Rotation around X** parameter can be different from **0**. For horizontal machines only positions around Y-axis can be added, so the **Rotation around X** and **Rotation around Z** values are **0**. The **Rotation around Y** parameter can be different from **0**.

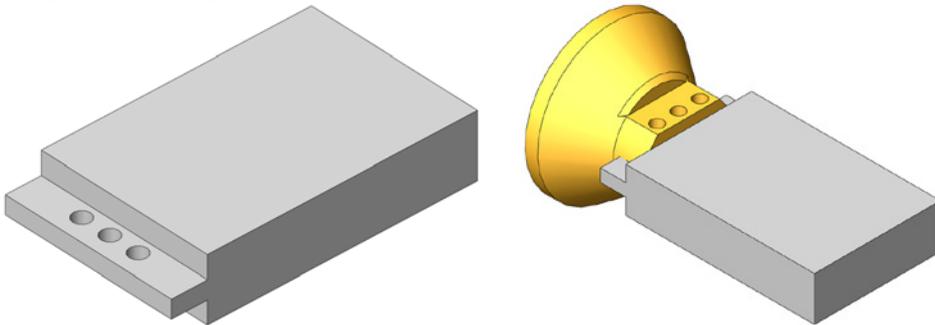
Machine Coordinate Systems and their positions are used for the geometry definition. Before the geometry definition you have to choose the Coordinate System position. The geometry has to be located in the plane parallel to the XY-plane of the chosen CoordSys.

Exercise #11: Frame Machining

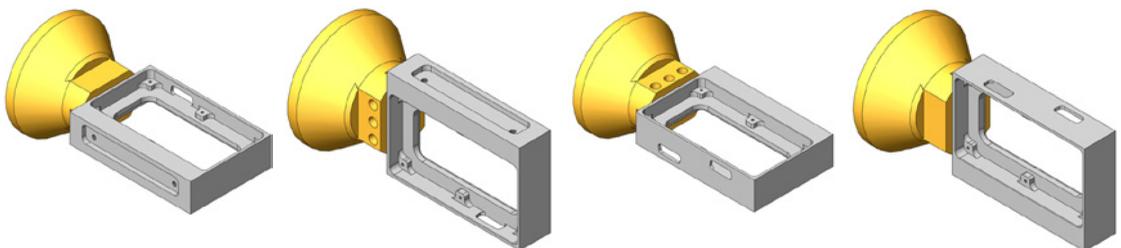
In this exercise, indexial 4-axis operations are used for the frame part machining.



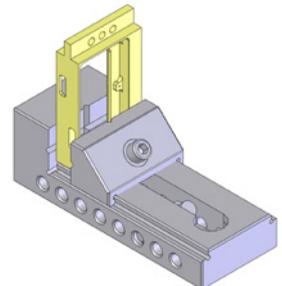
The frame is machined from the pre-machined stock on a vertical 4-axis milling CNC-machine using the special jig.



The jig with the clamped workpiece is placed on the rotary table of the machine. Such clamping enables you to machine four sides of the model using the rotation of the 4th axis.



The tail used for the part clamping is removed in a separate operation using a 3-axis CNC-machine.



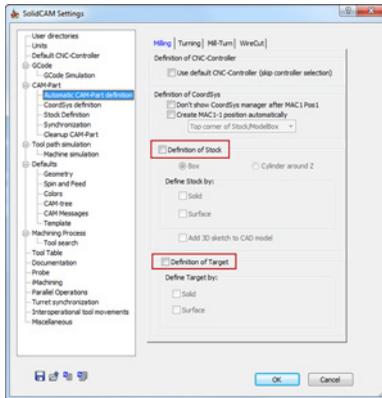
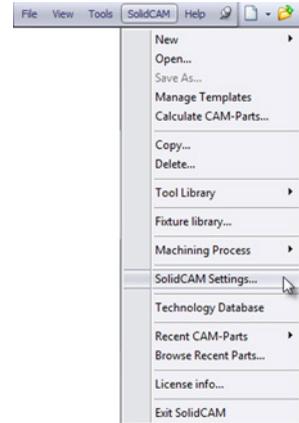


1. Change default settings

First, you have to cancel automatic definition of the stock and target models, which is set in the preferences by default.

In the SolidCAM menu, click the **SolidCAM Settings** command.

The **SolidCAM Settings** dialog box is displayed. From the left menu, choose **Automatic CAM-Part definition**. Clear the **Definition of Stock** and **Definition of Target** check boxes.



From now on, the stock and target models will not be defined automatically for each CAM part you create.



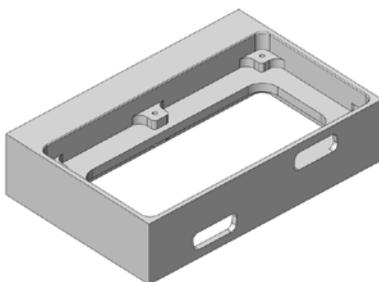
2. Load the SolidWorks model

Load the **Exercise11.sldprt** model located in the **Exercises** folder.

The model contains three configurations:

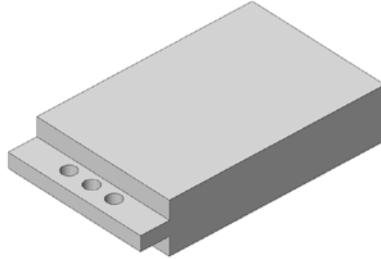
- **Default**

The original design model without changes.



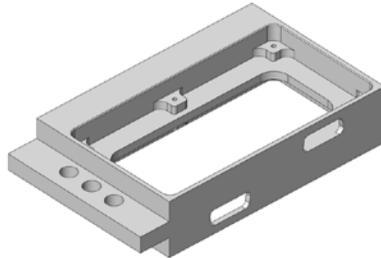
- **Stock**

The stock model that is used for the SolidCAM stock definition.



- **Machining**

The machining model with a tail that was not removed. This configuration is used for the SolidCAM geometries definition.



3. Start SolidCAM

To start SolidCAM, click the **SolidCAM** command in the main menu of SolidWorks and choose **Milling** from the **New** submenu. SolidCAM is started, and the **New Milling Part** dialog box is displayed.



4. Confirm the CAM-Part creation

Confirm the **New Milling Part** dialog box by clicking the **OK** button. The **Milling Part Data** dialog box is displayed.



5. Define the CNC-controller

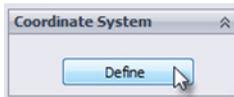
Choose a CNC-machine controller. Click the arrow in the **CNC-Machine** section to display the list of post-processors installed on your system.

In this exercise, use a vertical 4-axis CNC-machine with the **Fanuc** CNC-controller. Choose the **gMill_Haas_4x_eval** CNC-controller from the list.



6. Define the Machine Coordinate System

Click the **Define** button in the **Coordinate System** section of the **Milling Part Data** dialog box to start the Coordinate System definition.

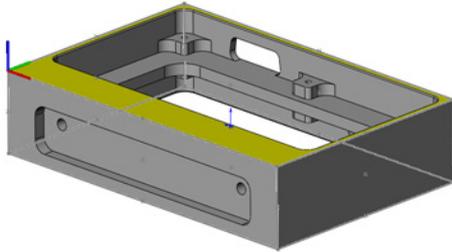


The **CoordSys** dialog box is displayed.

SolidCAM assigns the first **Position** number to the **Machine Coordinate System** by default.

Make sure that the default **Select Face** option is chosen.

Click on the model bottom face as shown.



The **Machine Coordinate System #1** is automatically defined in the corner of the model box. Click .

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



SolidCAM automatically determines the **Shift** and **Rotation** values. For the first Position related to a new Machine Coordinate System, the **Shift** and **Rotation** parameters values are always **0**.

Close this dialog box with the **OK** button. The **CoordSys Manager** dialog box is displayed.



7. Define the CoordSys positions

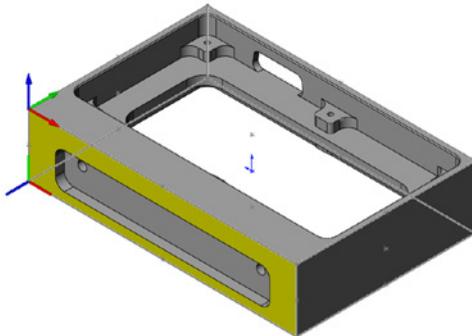
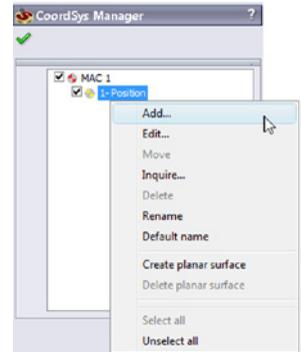
Now you have to define several additional **Positions** around the **Machine Coordinate System #1**.

In **CoordSys Manager**, right-click the **Position #1** item and choose the **Add** command from the menu.

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

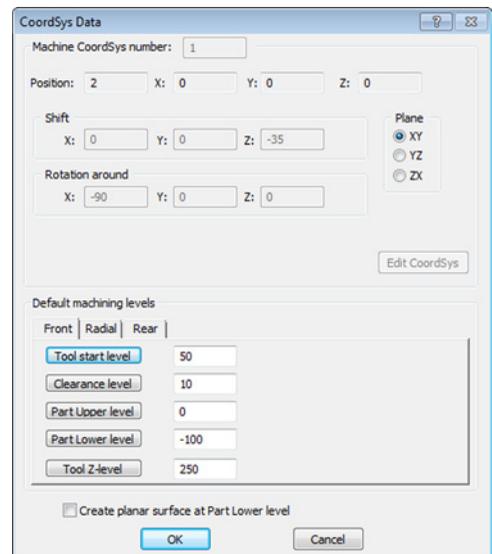
SolidCAM assigns the next sequential number to the position.

Click on the model face as shown.



The origin is automatically placed in the corner of the model. The Z-axis is directed outwards normally to the selected face. The X-axis direction is the same as the direction of the X-axis of the Machine Coordinate System (**Position #1**).

Click  to confirm the position location. The **CoordSys Data** dialog box is displayed.

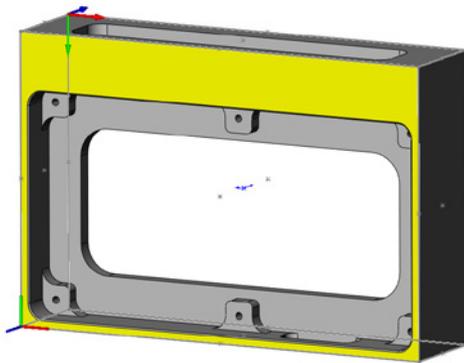
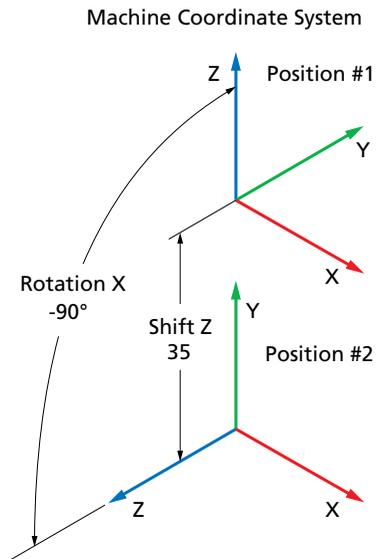


The **Part upper level (0)** and the **Part lower level (-100)** are automatically determined on the model.

The **Shift** and **Rotation** parameters are defined automatically for the **Position #2**. The **Shift Z** is **-35**; the **Shift X** and **Shift Y** are **0**. The **Rotation around X** is **-90**; other **Rotation** values are **0**.

Confirm the dialog box with the **OK** button.

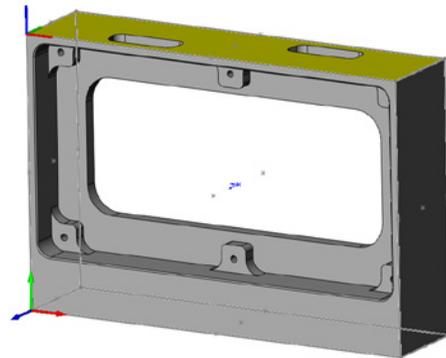
In the same manner, define **Position #3** and **Position #4** using the model faces as shown.



Position #3

Shift (0,100,-35)

Rotation (180,0,0)



Position #4

Shift (0,100,0)

Rotation (90,0,0)

Confirm the **CoordSys Manager** dialog box with



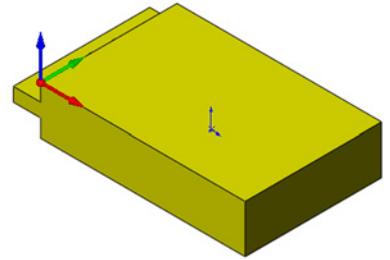
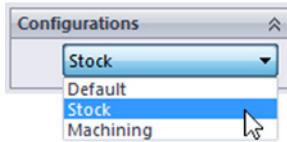
8. Define the Stock model

Click the **Stock** button in the **Stock & Target model** area. The **Model** dialog box is displayed.

Select the **3D Model** option in the **Defined by** section. This option enables you to define the stock model by picking solid bodies.

In the **Configurations** section, switch to the **Stock** configuration.





The model is displayed in the **Stock** configuration.

Click on the model to select it for the stock model.

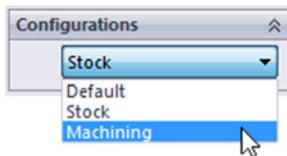
Close the **Model** dialog box with .



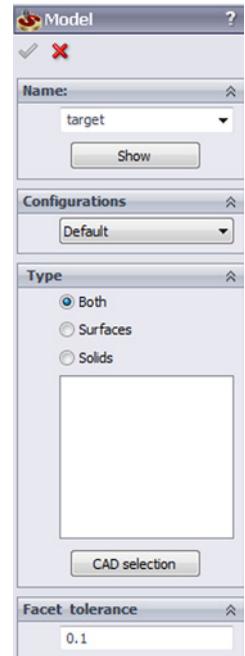
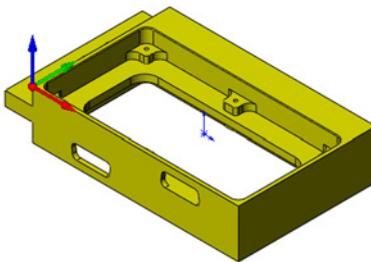
9. Define the Target model

In the **Milling Part Data** dialog box, click the **Target** button. The **Model** dialog box is displayed.

In the **Configuration** section, choose the **Machining** configuration that describes the model after the machining.



The configuration is changed. Click on the model as shown.



Close the **Model** dialog box with .



10. Save the CAM-Part

Click  to save the CAM-Part data and close the **Milling Part Data** dialog box.

The **SolidCAM Manager** dialog box is displayed.



11. Define the Fixture

SolidCAM enables you to define the **Part fixtures** such as clamps, vises, jig plates, etc. This feature enables you to get a more realistic picture during the simulation and check possible collisions between the cutting tools and fixtures.

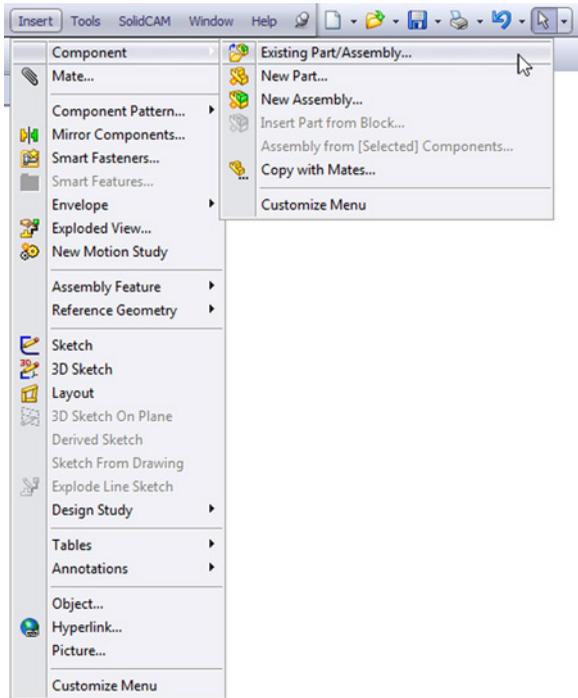
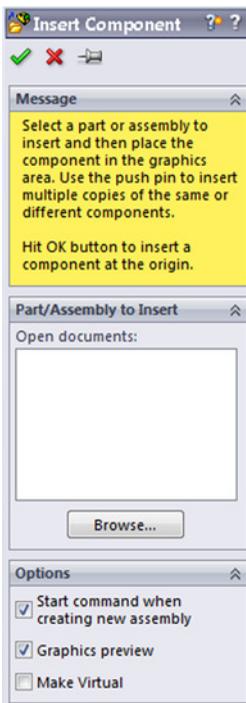
At first, you have to add the model of the clamping device to the CAM-Part assembly.



It is recommended to copy your clamp model files into the CAM-Part folder before the insertion into the CAM-Part assembly.

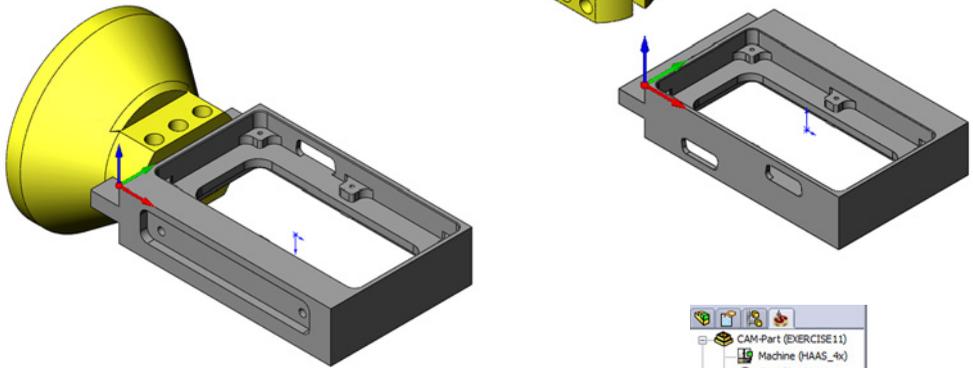
Click the **Insert** command in the SolidWorks menu and choose the **Existing Part/Assembly** option from the **Component** submenu. The **Insert Component** dialog box is displayed.

Click the **Browse** button and choose the **Clamp.sldprt** file. This file contains the model of the clamping device.

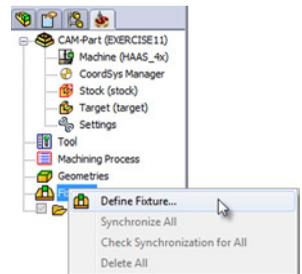


Confirm the component insertion with . The model is added.

Align the model as shown below using the **Mate** or the **SmartMates** option.

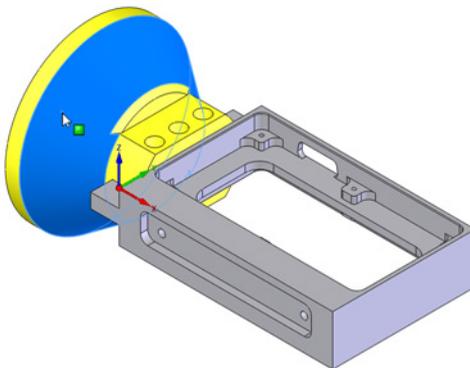


In **SolidCAM Manager**, right-click the **Fixtures** header and choose the **Define Fixture** command from the menu.



The **Model** dialog box is displayed. This dialog box enables you to define the fixture by selecting solid bodies of the CAM-Part assembly components.

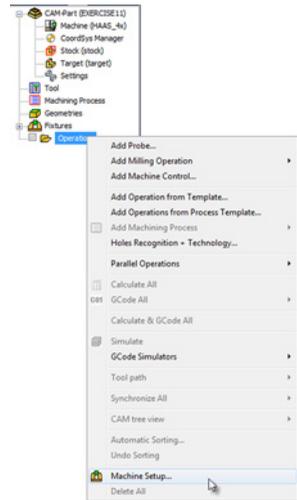
Click on the clamping device model as shown.



Confirm the **Model** dialog box with .

To choose this new fixture for future operations, right-click the **Operations** header and choose the **Machine Setup** command from the menu.

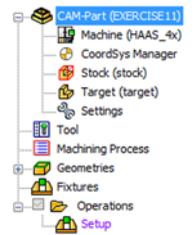
The **Machine setup** dialog box is displayed.



Until now, no fixture has been chosen for operation, therefore the **Fixture** cell in the table contains the **None** field. Double-click this field and choose the **clamping_ fixture** option for the fixture that you have defined earlier.

In the **Z** cell, type in **35** to raise the model 35 mm above the machine table. Click **OK**.

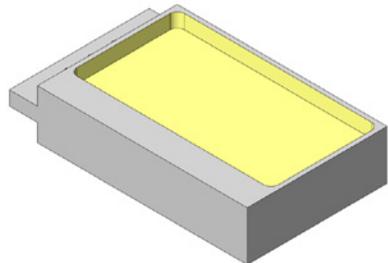
The **Setup** sub-header is now displayed under the **Operations** header.



12. Define a Pocket operation

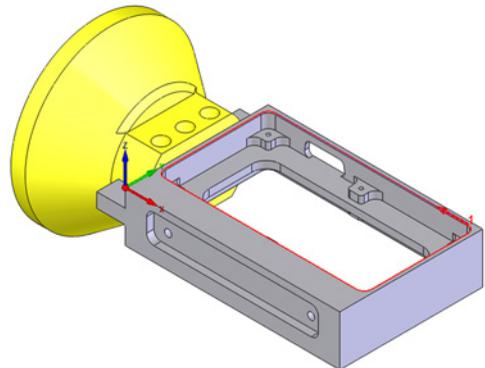
Define a Pocket operation to perform the rough machining of the highlighted faces.

Choose the **Coordinate System Position #1** in the **Geometry** page of the **Pocket Operation** dialog box.



Define the Geometry as shown.

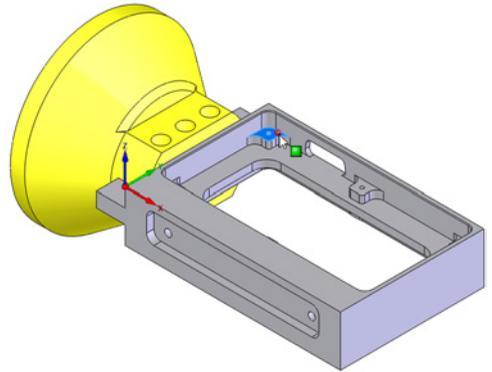
Define a new **End mill** tool of **Ø10** and choose it for the operation. Choose the appropriate tool holder.



Define the **Pocket depth** on the top face of the pad as shown.

Set the **Wall offset** and the **Floor offset** values to **0.2** to leave allowance for the next finishing operation.

Use the **Helical Ramping** strategy. Set the **Radius** value to **5**.



13. Define a Pocket operation

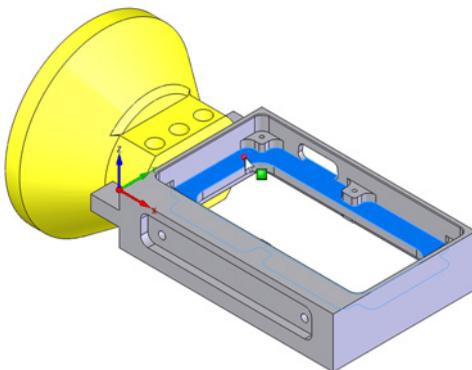
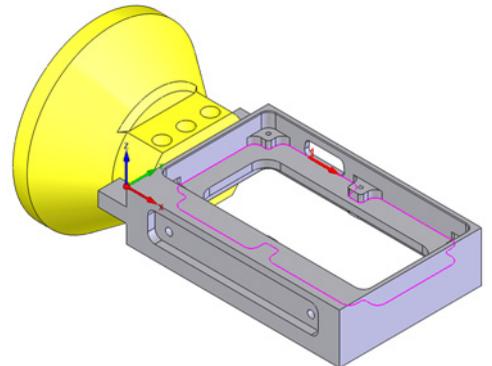
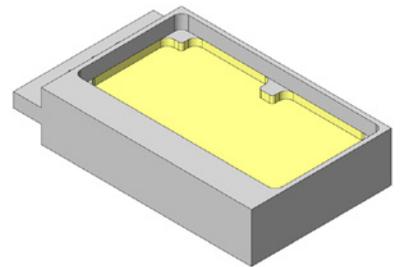
Define a Pocket operation to perform the rough machining of the highlighted faces.

The **Coordinate System Position #1** has to be chosen in the **Geometry** page of the **Pocket Operation** dialog box.

Define the Geometry as shown.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Define the **Upper level** for the operation on the top face of the pad (used in the previous operation for the **Pocket depth** definition). Define the **Pocket depth** on the bottom face as shown.



Set the **Wall offset** and the **Floor offset** values to **0.2** to leave allowance for the further finishing.

Use the **Helical Ramping** strategy. Set the **Radius** value to **5**.



14. Define a Pocket operation

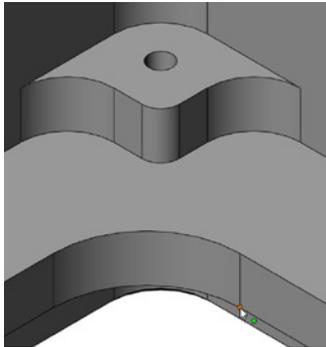
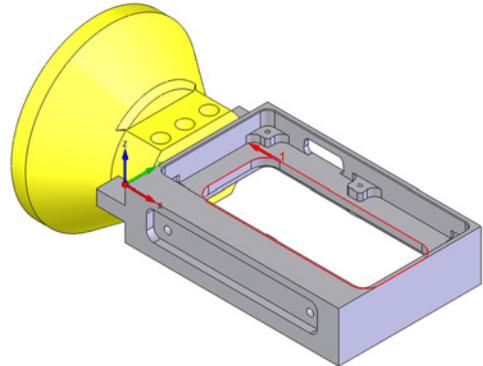
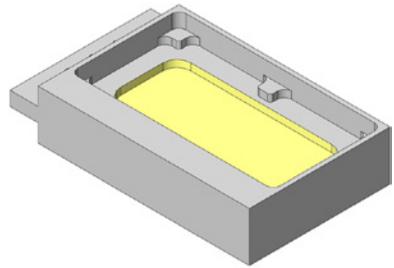
Define a Pocket operation to perform the rough machining of the highlighted faces.

Choose the **Coordinate System Position #1** in the **Geometry** page of the **Pocket Operation** dialog box.

Define the Geometry as shown.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Define the **Upper level** for the operation using the bottom face of the previously machined pocket. Pick the **Pocket depth** as shown.



Set the **Delta** value for the Pocket Depth to **-1** to perform the through pocket machining.

Set the **Wall offset** value to **0.2** to leave allowance for the further finishing.

Use the **Helical Ramping** strategy. Set the **Radius** value to **5**.



15. Define a Profile operation

Define a Profile operation to perform the finish machining of the highlighted faces.

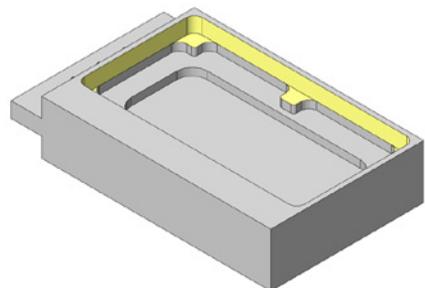
Choose the **Coordinate System Position #1** in the **Geometry** page of the **Profile Operation** dialog box.

Use the geometry defined in **Step #12**.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Use the same milling levels that were defined in **Step #12**.

Choose the **Arc** strategy for **Lead in** and select the **Same as Lead in** check box under **Lead out**.





16. Define a Profile operation

Define a Profile operation to perform the finish machining of the highlighted faces.

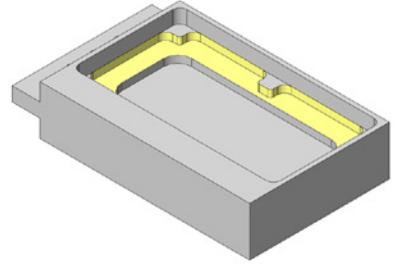
Choose the **Coordinate System Position #1** in the **Geometry** page of the **Profile Operation** dialog box.

Use the geometry defined in **Step #13**.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Use the same milling levels used in **Step #13**. Set the **Clear offset** and the **Step over** parameters to perform the machining of the bottom face in several concentric profiles.

Choose the **Arc** strategy for **Lead in** and select the **Same as lead in** check box under **Lead out**.



17. Define a Profile operation

Define a Profile operation to perform the finish machining of the highlighted faces.

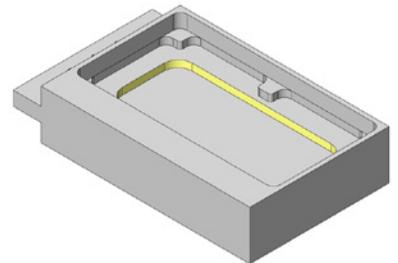
Choose the **Coordinate System Position #1** in the **Geometry** page of the **Profile Operation** dialog box.

Use the geometry defined in **Step #14**.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Use the same milling levels used in **Step #14**.

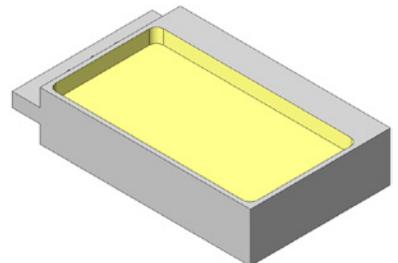
Choose the **Arc** strategy for **Lead in** and select the **Same as lead in** check box under **Lead out**.



18. Define a Pocket operation

In the same manner as explained in **Step #12**, define a Pocket operation to perform the rough machining of the highlighted faces.

Use the **Coordinate System Position #3** for the operation.

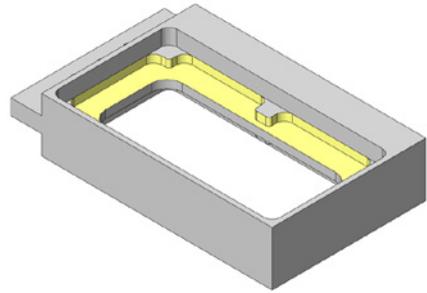




19. Define a Pocket operation

In the same manner as explained in **Step #13**, define a Pocket operation to perform the rough machining of the highlighted faces.

Use the **Coordinate System Position #3** for the operation.

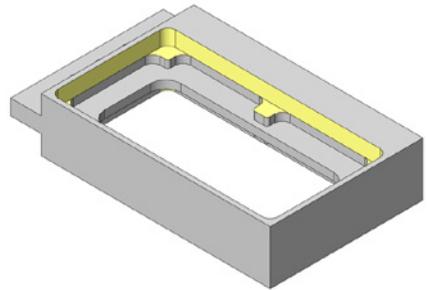


20. Define a Profile operation

In the same manner as explained in **Step #15**, define a Profile operation to perform the finish machining of the highlighted faces.

Use the geometry defined in **Step #18**.

Use the **Coordinate System Position #3** for the operation.



21. Define a Profile operation

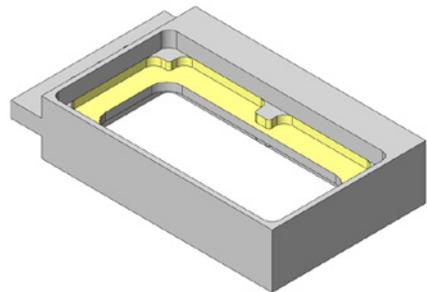
In the same manner as explained in **Step #16**, define a Profile operation to perform the finish machining of the highlighted faces.

Use the **Coordinate System Position #3** for the operation.

Use the geometry defined in **Step #19**.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Set the **Clear offset** and the **Step over** values to perform the machining of the bottom face in several concentric profiles.

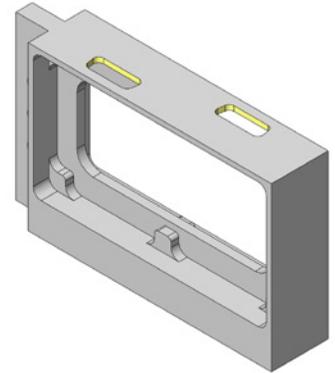
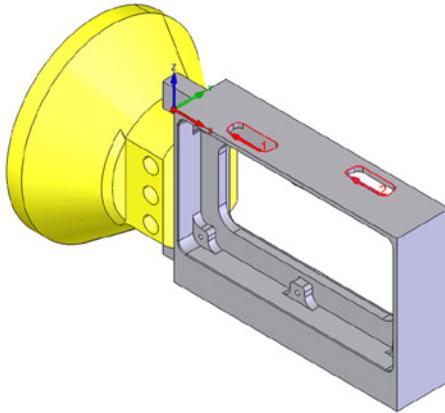




22. Define a Pocket operation

Define a Pocket operation to perform the rough and finish machining of two connector openings.

Use the **Coordinate System Position #4** for the operation. Define the geometry as shown.



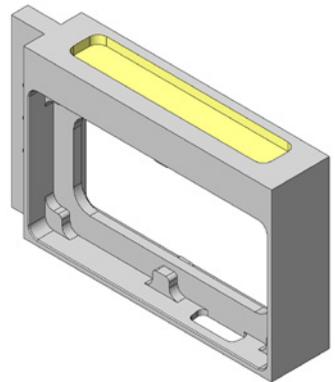
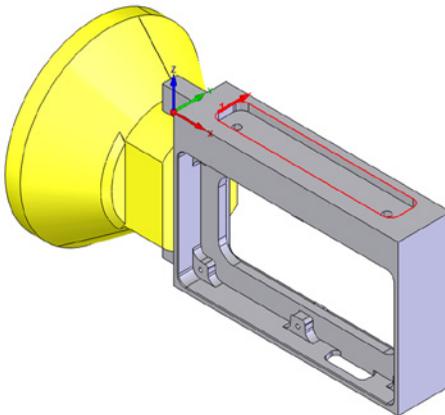
Define a new **End mill** tool of $\text{Ø}6$ for the operation. Choose the appropriate tool holder.



23. Define a Pocket operation

Define a Pocket operation to perform the rough and finish machining of the highlighted pocket faces.

Use the **Coordinate System Position #2** for the operation. Define the geometry for the operation as shown.



Choose the **Tool #1 (Ø10 End mill)** for the operation.

In the **Finish** section of the **Technology** page, select the **Wall** and the **Floor** check boxes to perform finishing of the walls and the floor of the pocket.

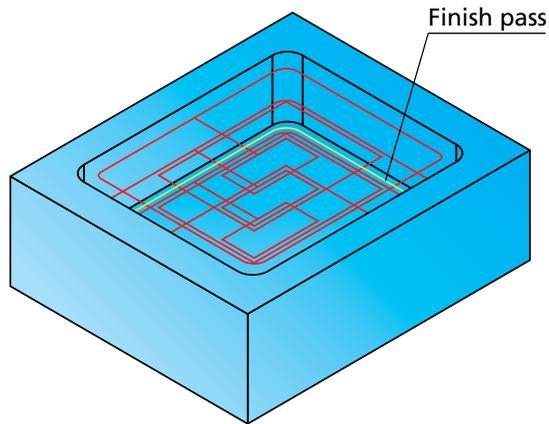


Pocket Wall Finishing

Wall finishing can be performed in a single cut at the whole depth or in a number of cuts at each step down.

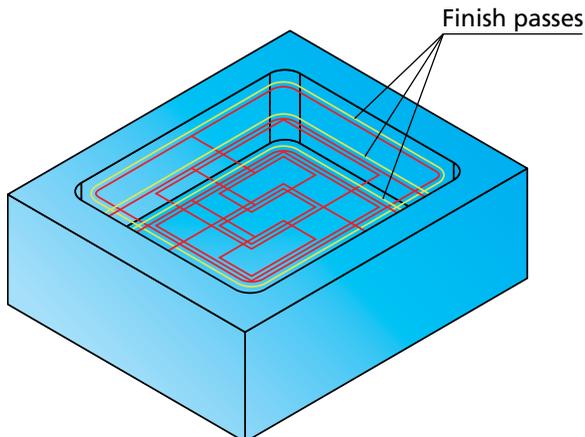
- **Total depth**

When this option is chosen, the wall finishing is performed in a single cut at the whole depth.



- **Each step down**

When this option is chosen, the wall finishing is performed in several successive cuts at each step down.





The options of the **Depth** section are available only when the wall finishing is performed in the operation (the **Wall** check box is selected).

Under **Depth**, choose the **Each step down** option.



24. Perform the holes machining

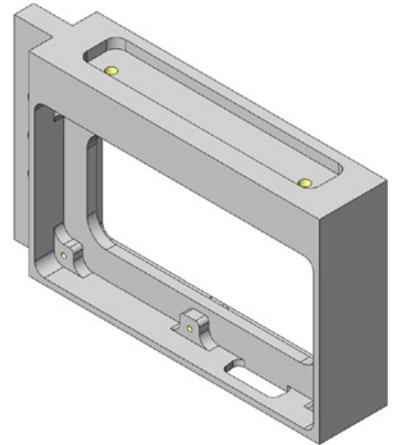
Define several Drilling operations to perform center drilling, drilling and threading of the holes.

Using the **Coordinate System Position #1**, perform the machining of the **M3** holes.

Use a **Ø6** spot drill (with **90°** tool tip angle), a **Ø2.5** drill and a **Ø3** tap tool. Choose the appropriate tool holders.

Using the **Coordinate System Position #2**, perform the machining of the **M6** holes.

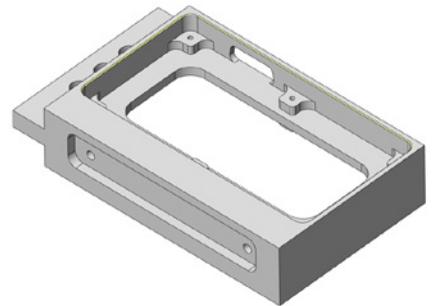
Use the **Ø6** spot drill (with **90°** tool tip angle); a **Ø5** drill and a **Ø6** tap tool. Choose the appropriate tool holders.



25. Perform the chamfer machining

At this stage, you have to define a Profile operation for the 0.5 mm chamfer machining.

Choose the geometry defined in **Step #12** for the operation. This geometry is defined using the **Coordinate System Position #1**.

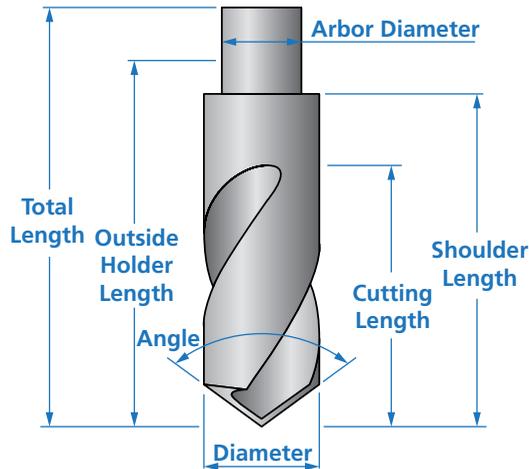


Define a **Chamfer drill** tool of $\text{Ø}6$ for the operation.



Chamfer drill

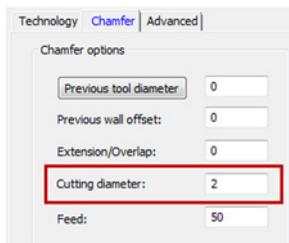
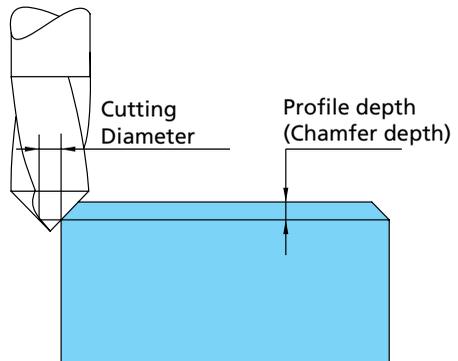
This tool is used for chamfering. A tool of this type is defined with the parameters shown in the image.



For the chamfer machining, set the value of the chamfer depth in the **Profile depth** field. In this case, set the value of **0.5**.

In the **Rest material/Chamfer** section, choose the **Chamfer** option from the list. A new page under the **Chamfer** tab is displayed.

Define the **Cutting diameter**, which is the diameter of the tool coincident with the geometry at the chamfer depth.



Set the **Cutting diameter** value to **2**.

In the **Link** section of the **Profile Operation** dialog box, choose the **Arc** strategy for **Lead in** and select the **Same as Lead in** check box under **Lead out**.

In the same manner, define a new Profile operation to machine another chamfer of 0.5 mm using **Coordinate System Position #3**.

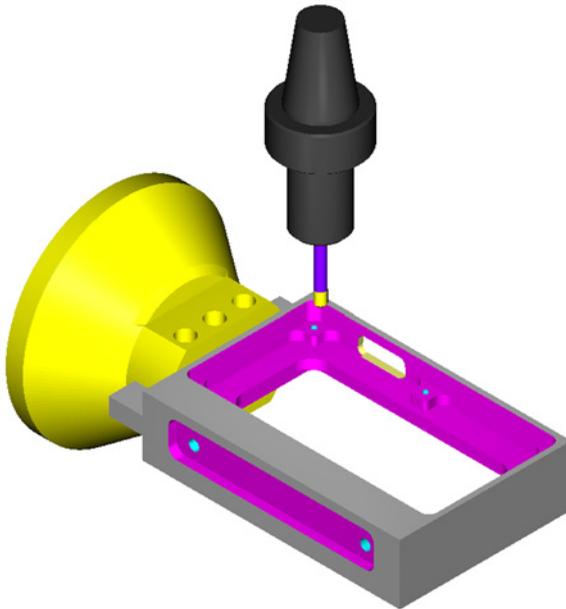


26. Simulate the tool path

Simulate the CAM-Part in the **SolidVerify** mode.

SolidCAM displays the fixture and checks the possible gouges during the simulation.

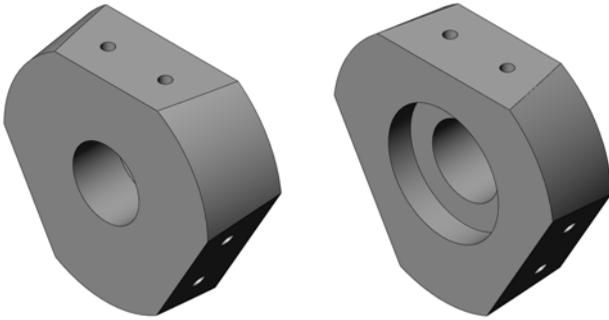
In the **SolidVerify** simulation, the tool is shown going around the part (rather than rotating the part as it actually occurs in the CNC-machine). Therefore, for all CNC-machine types, the simulation looks the same.



At this stage, you have completed the exercise.

Exercise #12: Mounting Machining

Define the CAM-Part and operations for machining of the mounting part shown below on the 4-axis vertical milling CNC-machine.



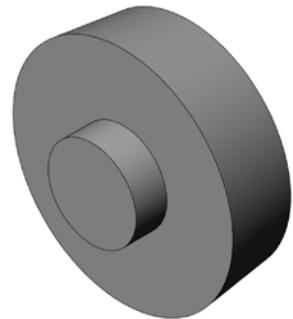
This exercise reinforces the following skills:

- CAM-Part definition
- Face milling
- Pocket machining
- Drilling

The SolidWorks model of the Mounting part (**Exercise12.sldprt**) is located in the **Exercises** folder.

The mounting is machined from the stock pre-machined by turning.

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



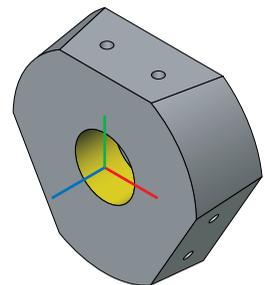
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**gMill_Haas_4x_eval**), Machine Coordinate System, the Stock and Target models.



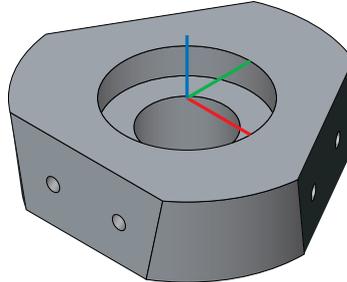
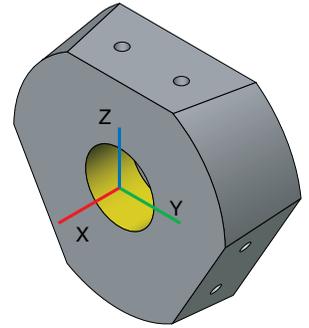
Machine Coordinate System definition

In this exercise, the Machine Coordinate System has to be defined in the center of the revolution face as shown.



Using the **Flip** options in the **CoordSys** dialog box, rotate the Coordinate System so that the X-axis is collinear with the revolution axis and the Z-axis is directed upwards as shown.

Define an additional **Machine Coordinate System #2** to machine the back faces of the mounting part.

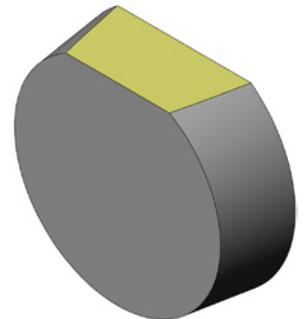


Define the Stock and the Target models using the configurations of the original SolidWorks model.

2. Face milling

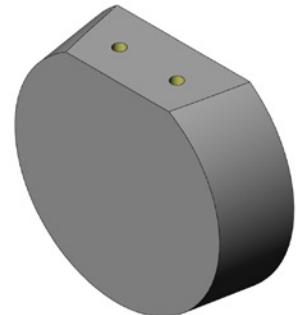
Define a Face Milling operation with the **Hatch** strategy to machine the shown face.

Use the **Machine Coordinate System #1 (Position #1)** for the machining.



3. Holes machining

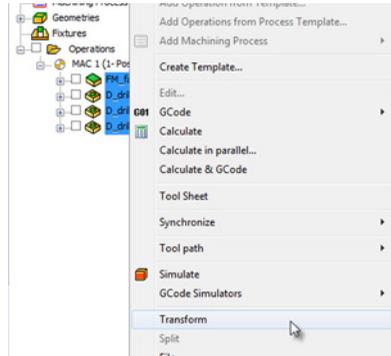
Define three Drilling operations to perform center drilling, drilling and threading of the **M6** holes.



4. Define the operation pattern

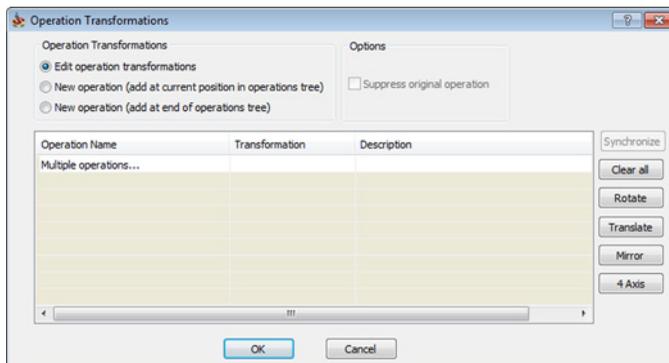
Now you have to define a circular pattern of the operations around the revolution axis (4th axis) of the CAM-Part in order to machine all the sides.

In **SolidCAM Manager**, select all the defined operations holding the **Shift** or **Ctrl** keys. Right-click one of the selected operations and choose the **Transform** option 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.

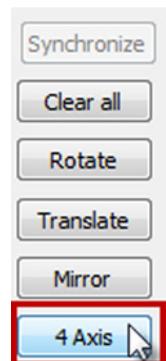
- **New operation (add at end of operations tree)**

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.

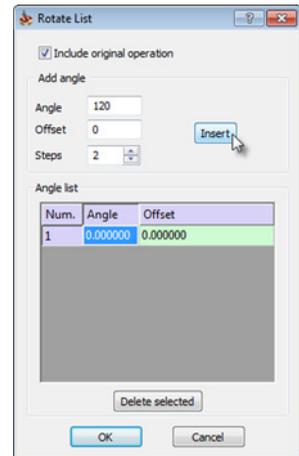
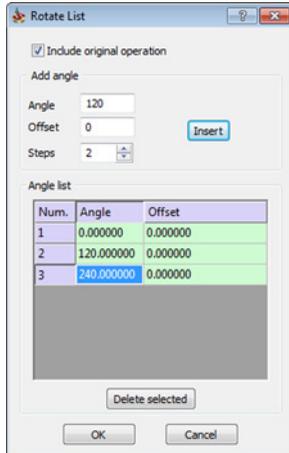
The **Rotate List** dialog box is displayed.



Select the **Include original operation** check box. When this check box is selected, the original operation tool path is included in the transforming action.

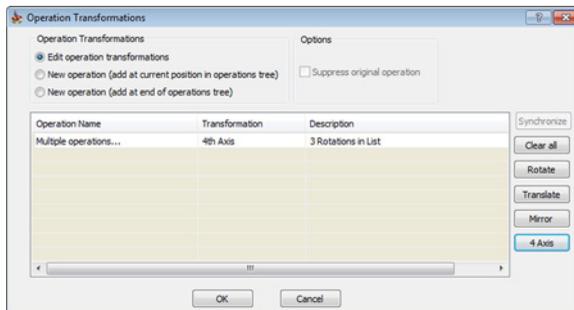
In the **Angle** field, type in the value of **120°**. This parameter defines the rotation angle for each pattern instance. In the **Steps** field, enter **2** for the number of instances.

Click the **Insert** button. The specific value of the rotation angle is calculated for each instance of the pattern.



Confirm the **Rotate List** dialog box with the **OK** button.

The operations to be transformed appear in the **Operation Transformations** dialog box.

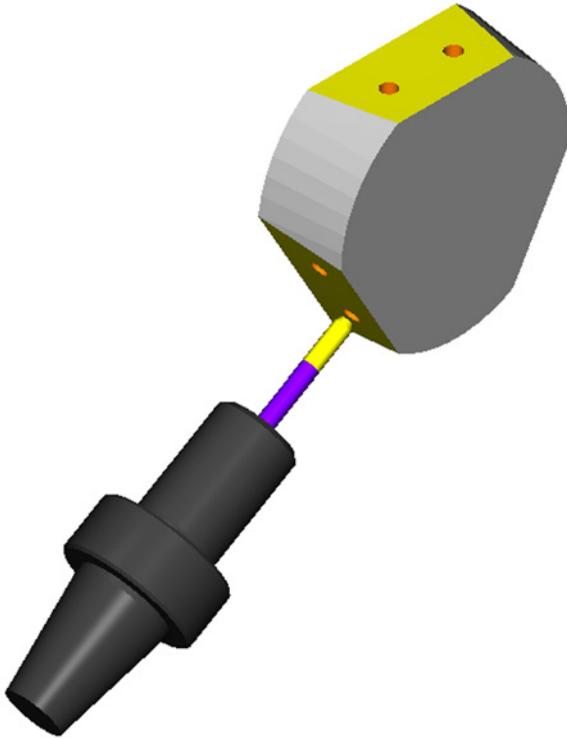


Confirm this dialog box with the **OK** button.

Note that the icons of the transformed operations are marked with the  icon. This icon means that the operations were patterned using the 4th axis.

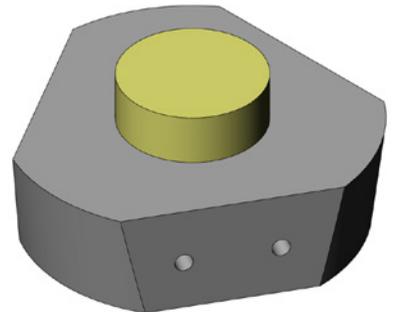
5. Simulate the transformed operations

Play the simulation of the transformed operations in the **SolidVerify** mode.



6. Remove the clamping tail

Define a Face Milling operation with the **Hatch** strategy using the **Machine Coordinate System #2** to remove the tail that was necessary for the part clamping at the 4th axis operations.

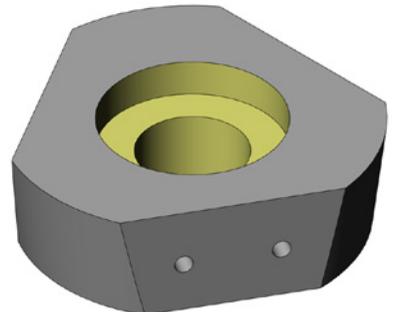


7. Machine the back side

Machine the pocket faces on the back side of the mounting.

Define several Pocket operations to perform rough and finish machining of the pockets.

Use the **Machine Coordinate System #2** for the operations.



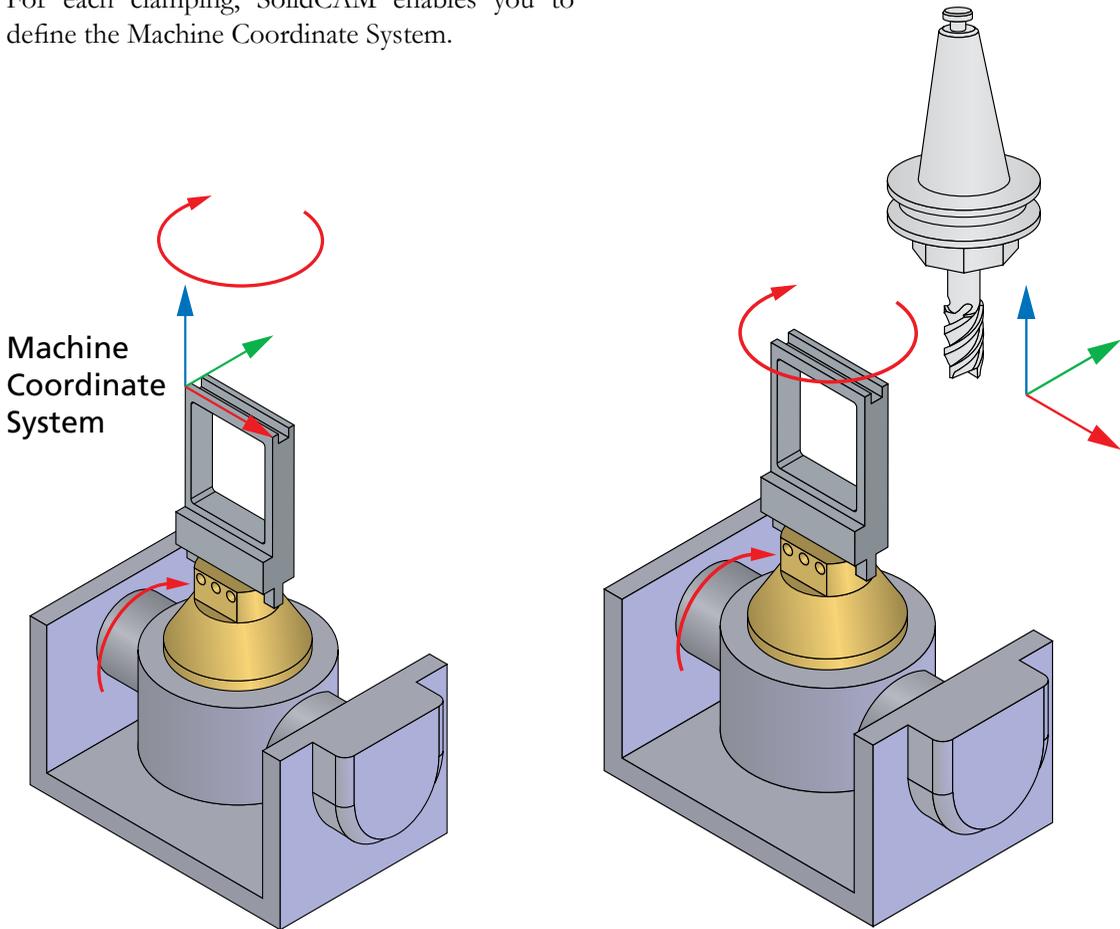
Indexial
5-Axis Milling

5

SolidCAM enables you to perform indexial 5-axis milling using either vertical or horizontal 5-axis CNC-machines.

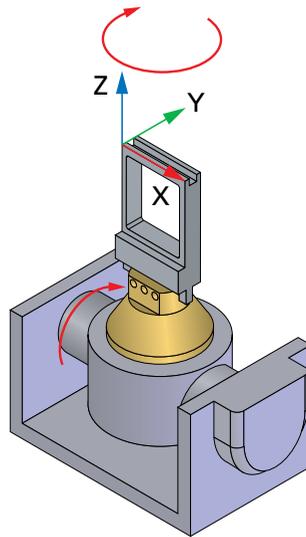
The 4th and 5th axis rotation is used for positioning of a part. When the part is positioned to the necessary orientation, the 4th and the 5th axes are locked and machining is performed with 3-axis movements.

For each clamping, SolidCAM enables you to define the Machine Coordinate System.

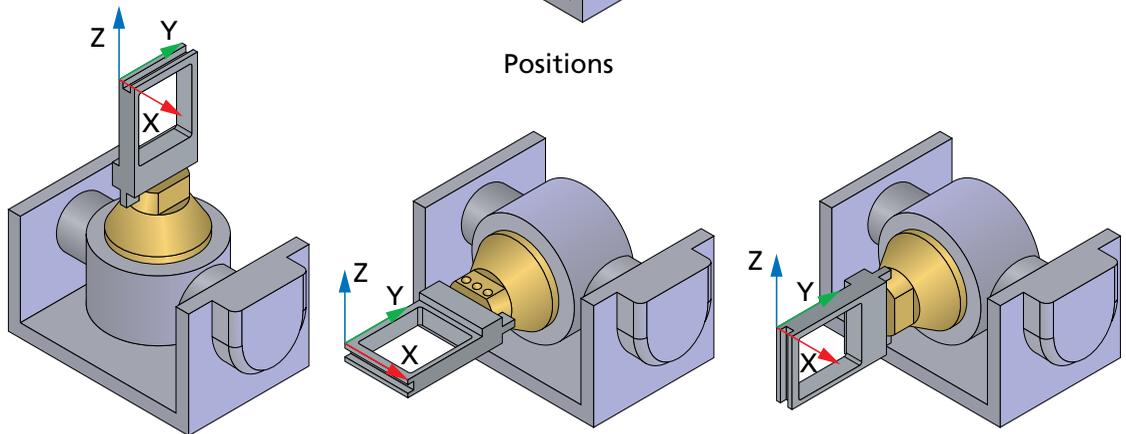


For each clamping (Machine Coordinate System), SolidCAM enables you to define a number of related positions around the Machine CoordSys. Take care of the machine limitations (angles or undercuts).

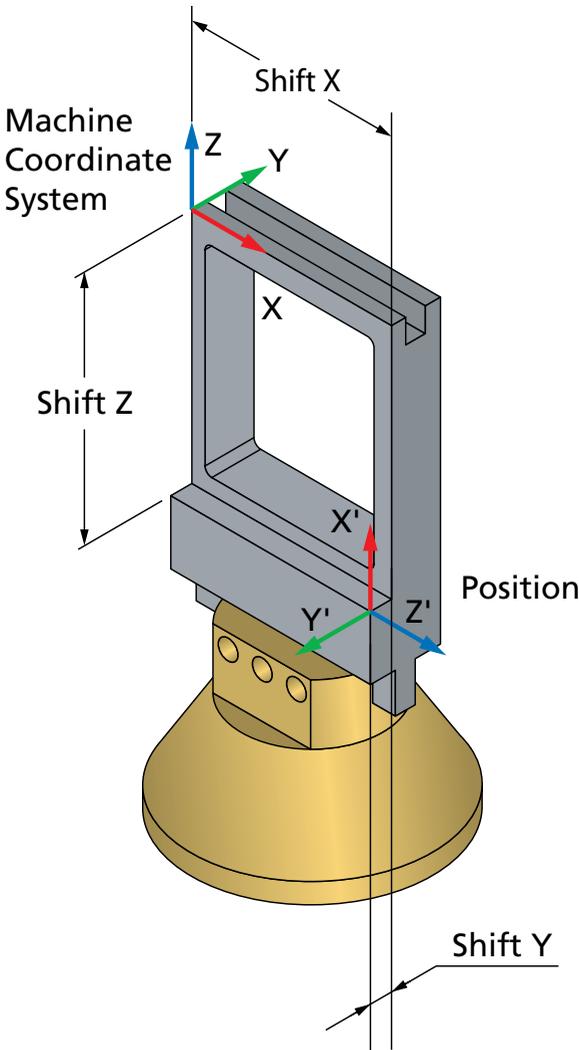
Machine Coordinate System



Positions

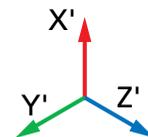
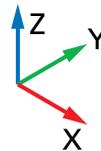


For each position, SolidCAM automatically determines the **Shift** and **Rotation** parameters relative to the Machine Coordinate System.



Machine Coordinate Systems and their positions are used for the geometry definition. Before the geometry definition you have to choose the Coordinate System position. The geometry has to be located in the plane parallel to the XY-plane of the chosen CoordSys.

Machine
Coordinate
System



Position

Rotation:

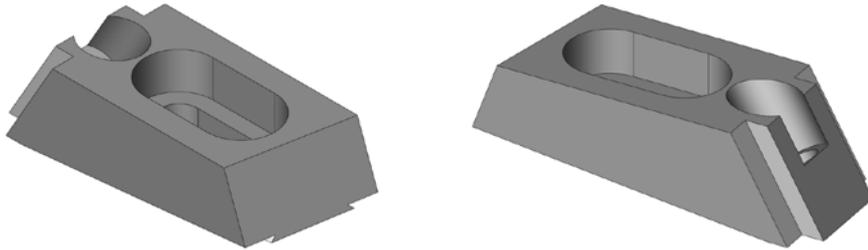
X= 0

Y= -90

Z= 180

Exercise #13: Clamp Machining

Define the CAM-Part and operations for the machining of the clamp shown below on the 5-axis vertical milling CNC-machine.



This exercise reinforces the following skills:

- CAM-Part definition
- Face milling
- Pocket machining
- Profile machining
- Drilling
- Indexial 5-axis machining

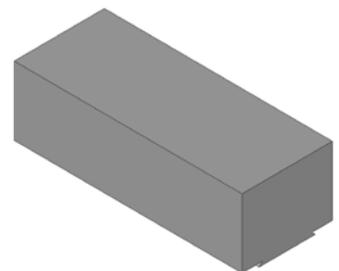
The SolidWorks model of the Clamp (**Exercise13.sldprt**) is located in the **Exercises** folder. The model contains two SolidWorks configurations:

- **Machining**

This configuration represents the model after the machining.

- **Stock**

This configuration represents the pre-machined workpiece before machining. The dovetail is pre-machined to perform simple clamping of the part during the machining.



1. Define the CAM-Part

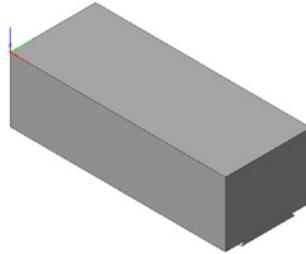
At this stage, you have to define the CAM-Part, CNC-controller (**TTC_Hermle_C20_full**), Machine Coordinate System and its Positions, the Stock and Target models.



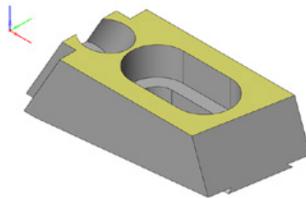
Coordinate System definition

Define **Machine Coordinate Systems #1** and **#2** and related positions using the previously defined Stock model.

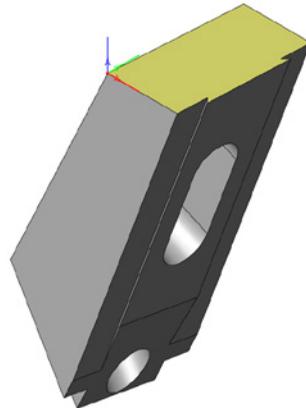
**Machine Coordinate System
#1
Position #1**



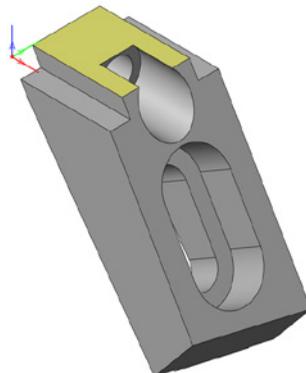
**Machine Coordinate System
#1
Position #2**



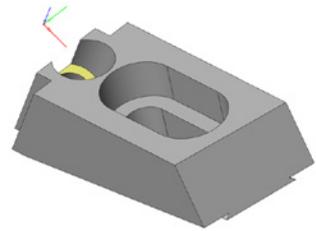
**Machine Coordinate System
#1
Position #3**



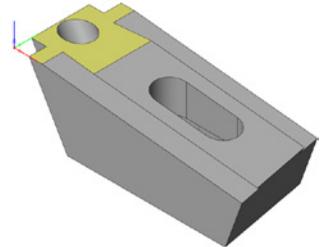
**Machine Coordinate System
#1
Position #4**



**Machine Coordinate System
#1
Position #5**



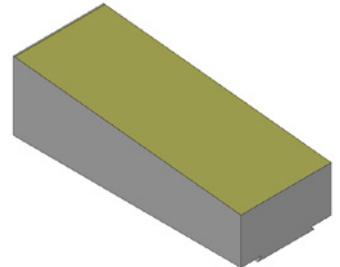
**Machine Coordinate System
#2
Position #1**



Define the Stock model using the **Stock** configuration of the CAM-Part assembly.
Define the Target model using the **Machining** configuration of the CAM-Part assembly.

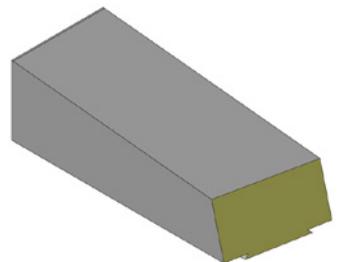
2. Top face machining

Define a Face Milling operation with the **Hatch** strategy to machine the top inclined face of the clamp. Use the **Machine Coordinate System #1 (Position #2)** for the operation.



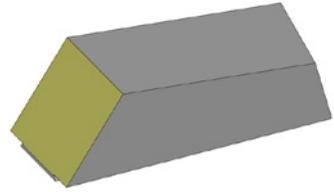
3. Back face machining

Define a Face Milling operation with the **Hatch** strategy to machine the back inclined face of the clamp. Use the **Machine Coordinate System #1 (Position #3)** for the operation.



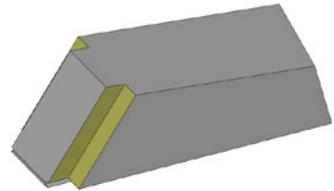
4. Front face machining

Define a Face Milling operation with the **Hatch** strategy to machine the front inclined face of the clamp. Use the **Machine Coordinate System #1 (Position #4)** for the operation.



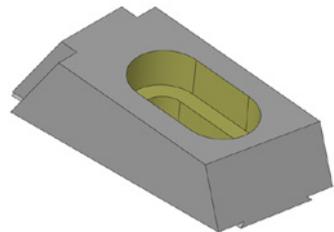
5. Openings machining

Define a Profile operation to machine two openings located on the front inclined face of the clamp. Use the **Machine Coordinate System #1 (Position #4)** for the operation.



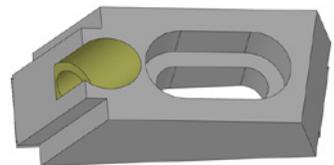
6. Slot machining

Define a number of operations to machine the slot faces located on the top inclined face of the clamp. Use the **Machine Coordinate System #1 (Position #2)** for the operation.



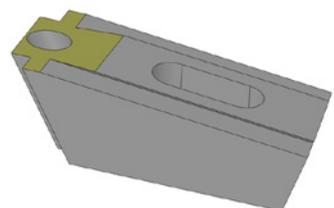
7. Hole machining

Define a number of operations to machine the inclined counterbore hole located on the top inclined face of the clamp. Use the **Machine Coordinate System #1 (Position #5)** for the operation.



8. Bottom face machining

Define a Face Milling operation with the **Hatch** strategy to machine the bottom inclined face of the clamp. Use the **Machine Coordinate System #2 (Position #1)** for the operation.



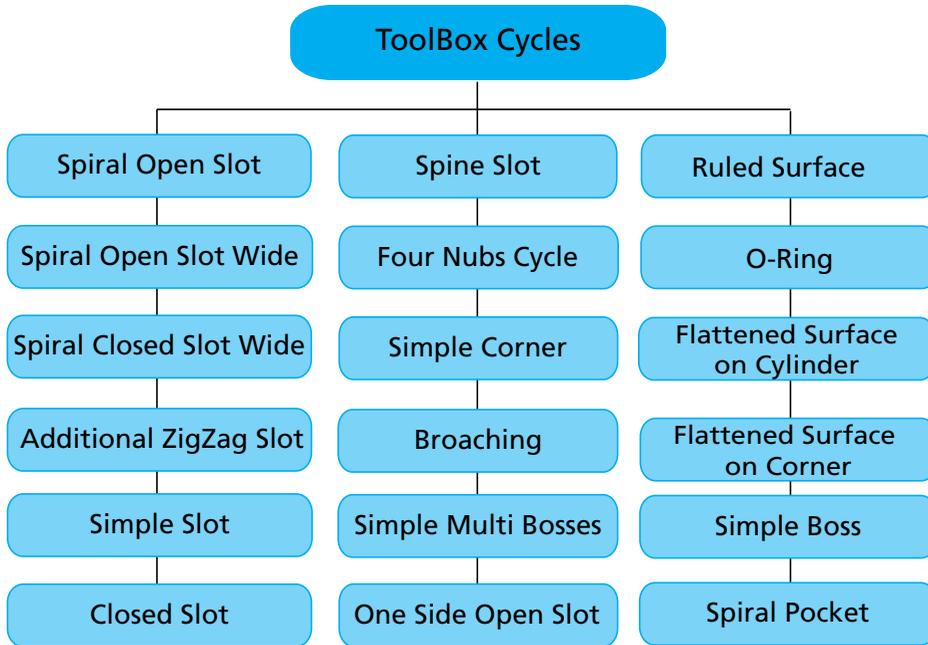
ToolBox Cycles

6

Toolbox sub-operations are a set of 2.5D Milling operations, each intended for a specific machining case. For example, some operations cover different strategies of the counterbore machining. Each sub-operation provides you with a specific machining strategy optimal for a particular machining case.

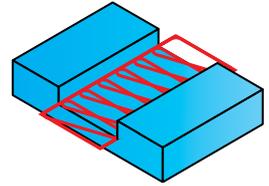
These operations are combined in a single dialog box as sub-operations.

SolidCAM offers you the following types of ToolBox Cycles:



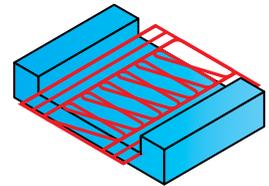
Spiral Open Slot

This strategy enables you to perform machining of open slots. The geometry consists of two chains. The machining is performed in a spiral manner.



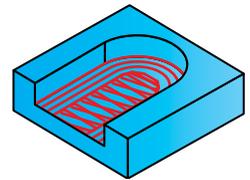
Spiral Open Slot Wide

This strategy enables you to perform machining of wide open slots. The first cutting pass is performed in the spiral manner along the determined spine of the slot, and then the machining of the slot walls is performed by a number of profiles equidistant to the slot geometry.



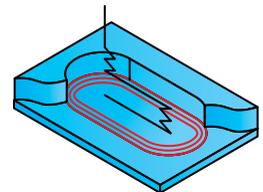
Spiral Closed Slot Wide

This strategy enables you to perform the machining of wide closed slots. The first cutting pass is performed in the spiral manner along the determined spine of the slot, and then the machining of the slot walls is performed along the slot geometry.



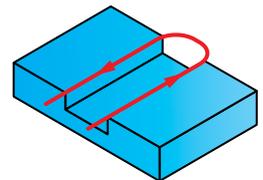
Additional Zigzag Slot

This strategy enables you to machine long slots. The tool path starts from the slot center approaching it in a zigzag movement and goes toward the walls.



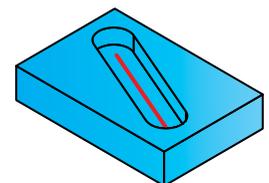
Simple Slot

This strategy enables you to perform one-pass machining of the slots with the width equal to or smaller than double diameter of the tool.



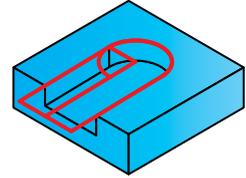
Closed Slot

This strategy enables you perform machining of a closed slot with the width equal to the tool diameter. The machining is performed in the zigzag manner; the last cut is horizontal to clean the slot floor.



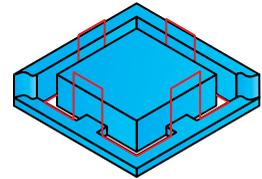
Spine Slot

This strategy enables you to perform machining of a slot opened from one side. SolidCAM automatically determines the spine of the slot and performs the machining according to the spine's shape. In this sub-operation, the tool approaches the material along the spine and then machines the walls of the slot.



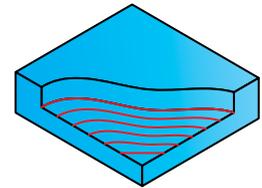
Four Nubs Cycle

This strategy enables you to machine the profile of a pocket while preserving the rest of the material fixed to the wall using small thin bridges. Inner material will be hammered for shorter machining time.



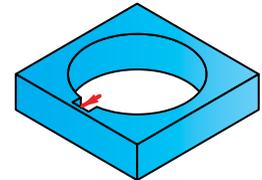
Simple Corner

This strategy enables you to perform machining of an open corner area in a number of equidistant cutting passes.



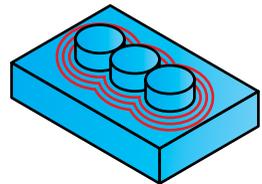
Broaching

This strategy enables you to perform broaching of holes.



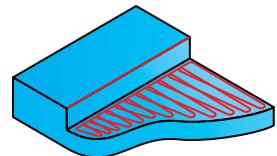
Simple Multi Bosses

This strategy enables you to perform the machining of several bosses starting from one face. The machining is performed in a number of equidistant **Clear Offset** passes parallel to the selected bosses geometry. The cutting passes are generated for the whole geometry combined from the initial bosses.



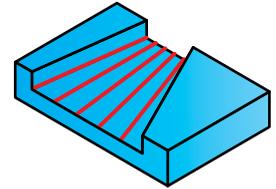
One Side Open Slot

This strategy enables you to machine an one-sided slot using two chains: a main chain and a drive chain. A spiral tool path is generated during this sub-operation.



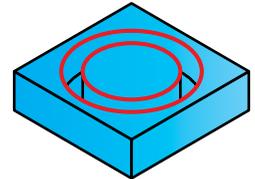
Ruled

This strategy enables you to perform the machining of an analytical ruled surface defined by two open contours. The cutting passes are evenly distributed between the defined contours.



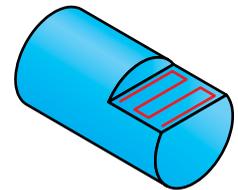
O-Ring

This strategy enables you to machine O-Ring slots.



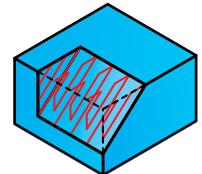
Flattened Surface on Cylinder

This strategy enables you to perform machining of the flattened surface on a cylinder.



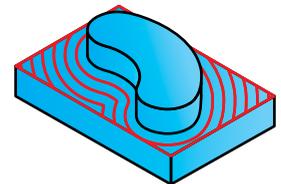
Flattened Surface on Corner

This strategy enables you to machine an angled plane using a defined working area in order to prevent air cuts.



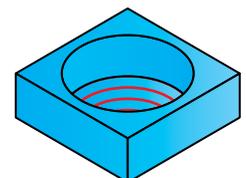
Simple Boss

This strategy enables you to perform machining of a boss located on a model face. SolidCAM enables you to define the closed geometry chain for the boss and closed chain for the initial model face from which the boss starts. The machining is performed in a number of equidistant cutting passes parallel to the boss geometry. The passes are cut with the outer contour of the initial face.



Spiral Pocket

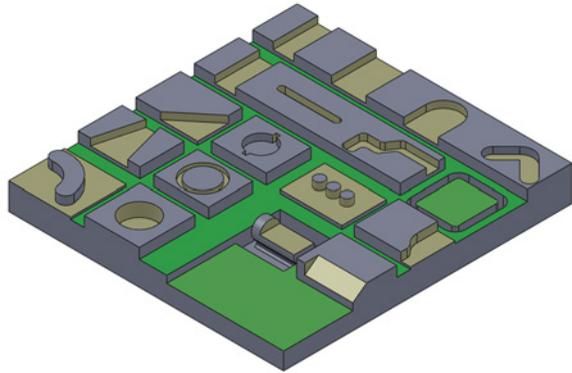
This strategy enables you to perform machining of the closed pocket areas using spiral techniques. With this strategy, the tool penetrates into the material in the automatically calculated point inside the pocket with a helix (with the automatically calculated radius and angle). Then the tool performs the spiral cutting pass morphing from the penetration point to the closed area walls.



Exercise #14: Standard Cycles Machining

In this exercise, you can use the Standard Cycles Plate to machine eighteen Standard Toolbox operations.

The part file (**Exercise14.prz**) is located in the **Exercises** folder.



1. Open the CAM-Part

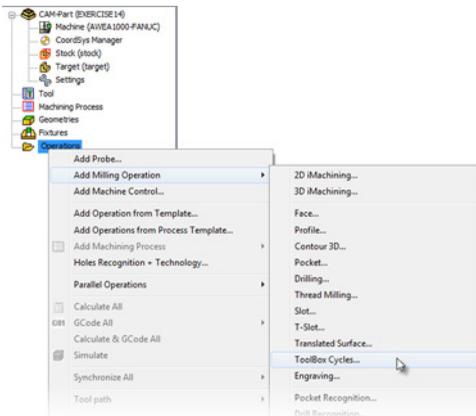
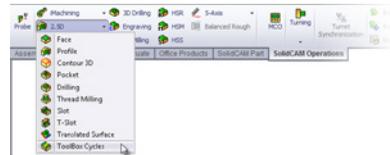
Open the **Exercise14.prz** file located in the **Exercises** folder.



2. Spiral open slot machining

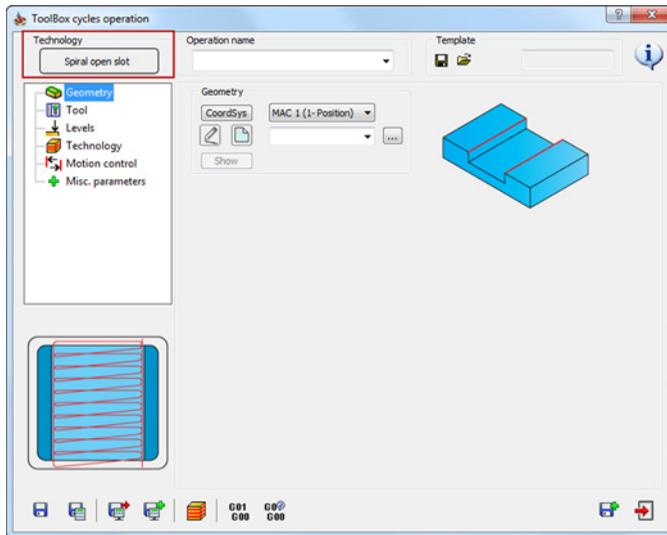
In the **SolidCAM Operations** toolbar, choose **2.5D > Toolbox Cycles**.

Alternatively, you can right-click the **Operations** header in **SolidCAM Manager** and select **Add Milling Operation > Toolbox Cycles**.



The **ToolBox Cycles Operation** dialog box is displayed.

Make sure the default **Spiral Open Slot** strategy is chosen in the top left corner under **Technology**.



Click  to define a new geometry.

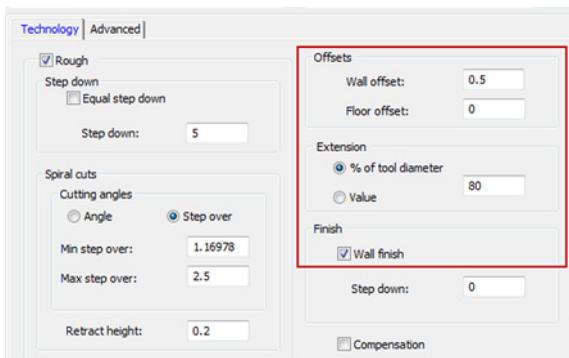
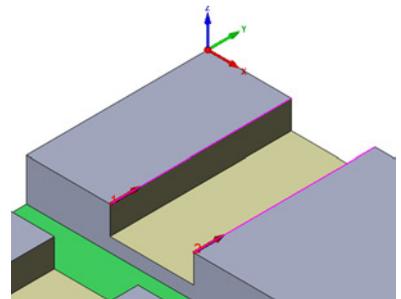
Choose a pair of chains as shown.

Confirm the geometry by clicking .

Add a new **End mill** of **Ø10**.

Select the **Milling levels** directly on the model.

In the **Technology page**, under **Offsets**, set **Wall Offset** to **0.5**. Under **Extension**, set **% of tool diameter** to **80**. Under **Finish**, select the **Wall finish** option.

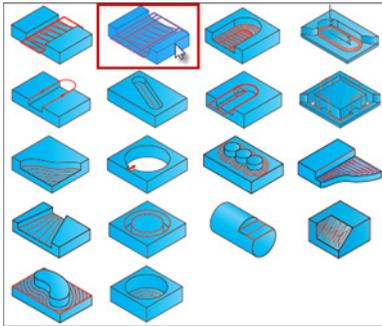




3. Wide Spiral Open Slot machining

In the manner as described in the previous step, define a new **ToolBox Cycles** operation.

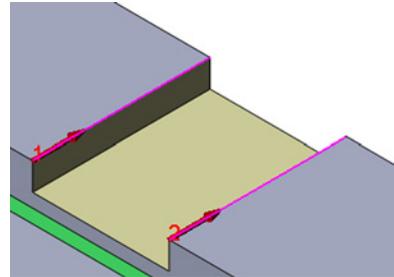
Under **Technology**, select the **Spiral Open Slot Wide** strategy.



Define a new geometry, selecting a pair of chains on the wide slot edges.

Use the same tool as in the previous operation.

In the **Technology page**, under **Offsets**, set **Wall Offset** to **1**. Under **Finish**, select the **Wall Finish** option.

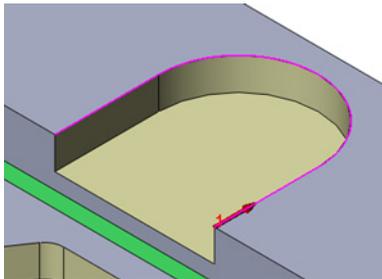


4. Wide Spiral Closed Slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Spiral Closed Slot Wide**  strategy.

Define a new geometry, selecting the open chain as shown.



Use the same tool as in the previous operation.

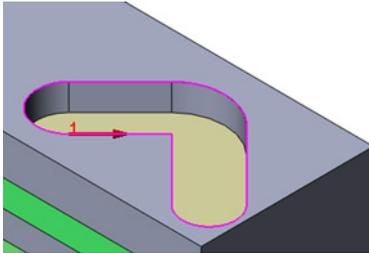


5. Additional Zigzag slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Additional Zigzag Slot**  strategy.

Define a new geometry, selecting the closed chain as shown.



Use the same tool as in the previous operation.

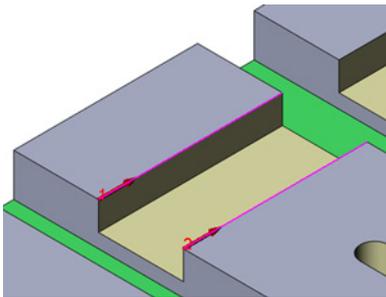


6. Simple Slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Simple Slot**  strategy.

Define a new geometry, selecting a pair of chains on the slot edges.



Add a new **End mill** of **Ø16**.

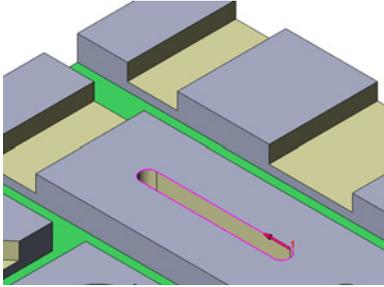


7. Closed Slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Closed Slot**  strategy.

Define a new geometry, selecting the closed chain as show.



Use the previously defined **Tool #1** of $\text{Ø}10$.



In the **Closed Slot** cycle, the tool diameter is equal to the slot width.

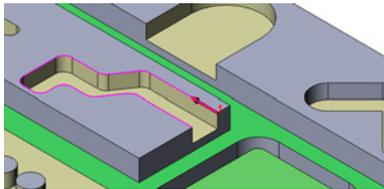


8. Spine Slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Spine Slot**  strategy.

Define a new geometry, selecting the open chain as shown.



Use the same tool as in the previous operation.

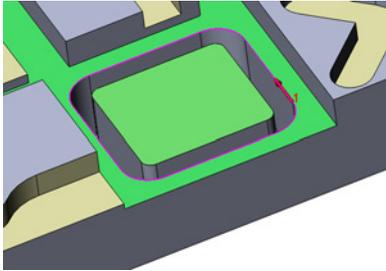


9. Four Nubs Cycle machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Four Nubs Cycle**  strategy.

Define a new geometry, selecting the closed chain as shown.



Define a new **End mill** of $\text{Ø}10$ for this operation and edit the following parameters:

- Set the **Shoulder length** to **60**
- Set the **Cutting length** to **60**

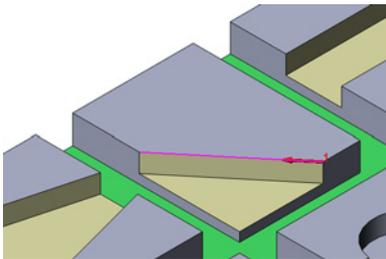


10. Simple corner machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Simple Corner**  strategy.

Define a new geometry, selecting the open chain as shown.



Use the previously defined **Tool #1** of $\text{Ø}10$.

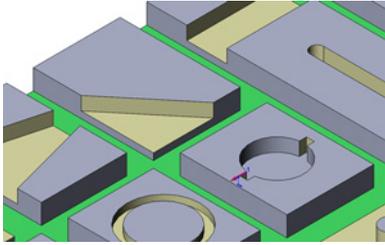


11. Broaching

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Broaching**  strategy.

Define a new geometry, selecting a linear chain as shown.

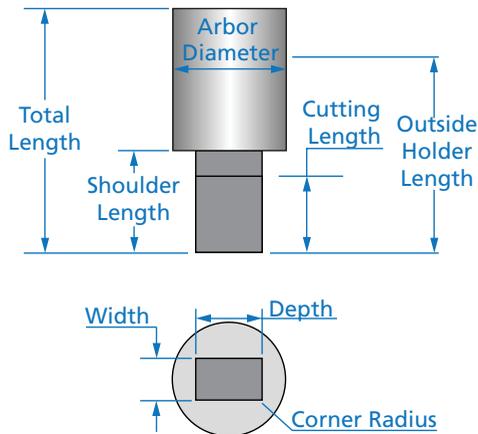


With this operation, only a Broaching tool is available.

Broaching Tool



The tool of this type can only be used within the Broaching sub-operation. It is not available for defining or choosing in any other operation.



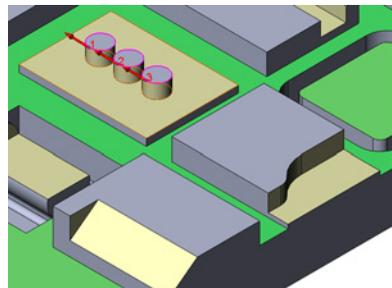
12. Simple Multi Bosses machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Simple Multi Bosses** strategy.

Define a new geometry, selecting the closed chains as shown.

Use the previously defined **Tool #1** of **Ø10**.



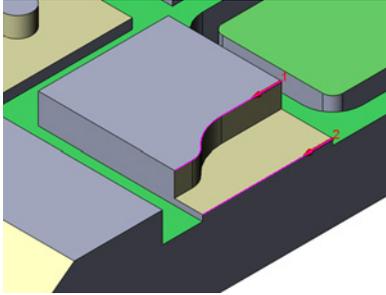


13. One Side Open Slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **One Side Open Slot**  strategy.

Define a new geometry, selecting a pair of open chains: the first one is considered main, and the second one is the drive chain.



Use the same tool as in the previous operations.

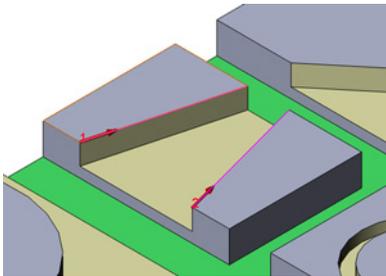


14. Ruled Surface machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Ruled**  strategy.

Define a new geometry, selecting a pair of open chains on the slot edges.



Use the same tool as in the previous operations.

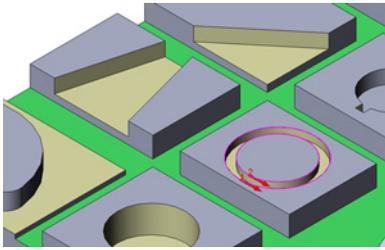


15. O-ring machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **O-Ring**  strategy.

Define a new geometry, selecting a pair of closed chains as shown. The first chain is considered external and the second one – internal.



Add a new **End mill** of $\text{Ø}4$.



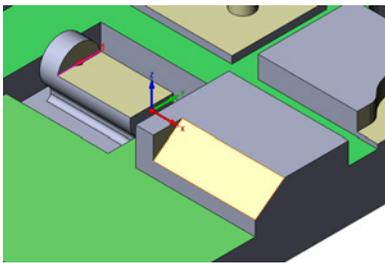
16. Surface Flattening on Cylinder

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Flattened Surface on Cylinder**  strategy.

For this operation, define a new Coordinate System position with the origin located at the cylinder revolution axis.

Define a new geometry, selecting the linear open chain as shown.



Use the previously defined **Tool #1** of $\text{Ø}10$.



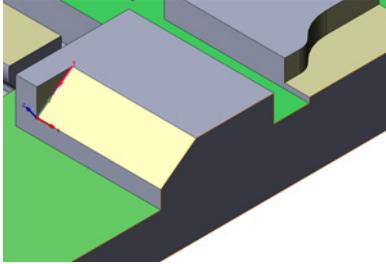
17. Surface Flattening on Corner

Define a new **ToolBox Cycles** operation.

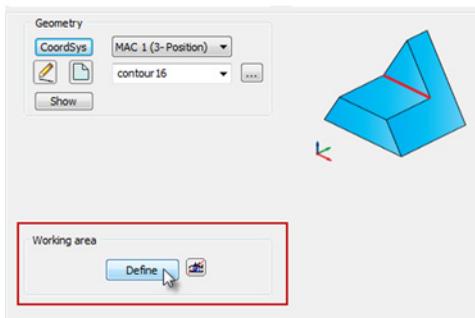
Under **Technology**, select the **Flattened Surface on Corner**  strategy.

For this operation, define a new Coordinate System position with the origin located on the flattened surface plane.

Define a new geometry, selecting a straight line chain.



In the **Working area** section, click the **Define** button to assign a triangle working area to the selected line.



The **Working area** dialog box is displayed. Make sure the **Start angle** and **End angle** are **45°**.

Use the same tool as in the previous operation.

Define the **Upper level** by clicking on the upper edge of the flattened plane. Define the **Depth** by clicking on the lower edge of the flattened plane.

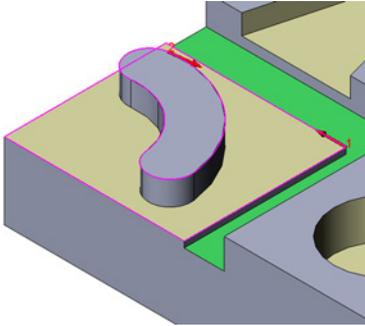


18. Simple Boss Machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Simple Boss**  strategy.

In the **Coordinate System #1**, define a new geometry using the **Multi-chain** option. A pair of closed chains is selected automatically, where the first chain is considered external and the second one is internal.



Use the same tool as in the previous operation.

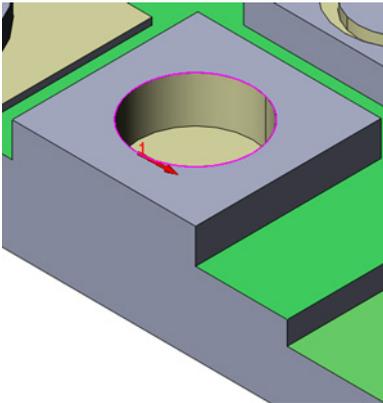


19. Spiral Pocket Machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Spiral Pocket**  strategy.

Define a new geometry, selecting the closed chain as shown.

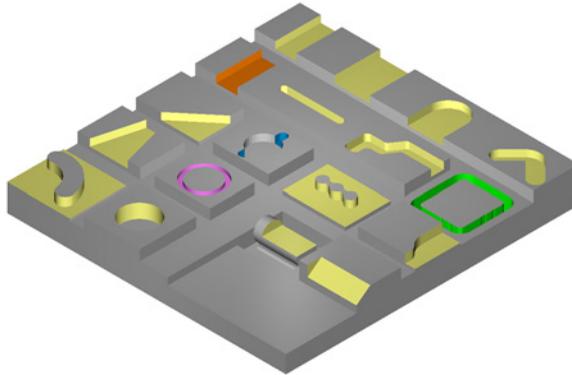


Use the same tool as in the previous operation.



20. Simulate the operation

Right-click the **Operations** header and choose **Simulate** to play simulation for all ToolBox operations.



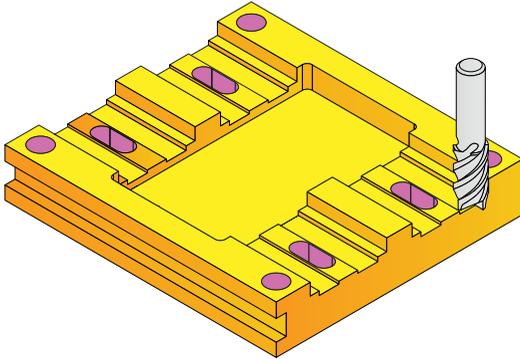
Automatic Feature Recognition

7

This chapter covers the following types of 2.5D Milling operations performed using the Automatic Feature Recognition module (AFRM) functionality:

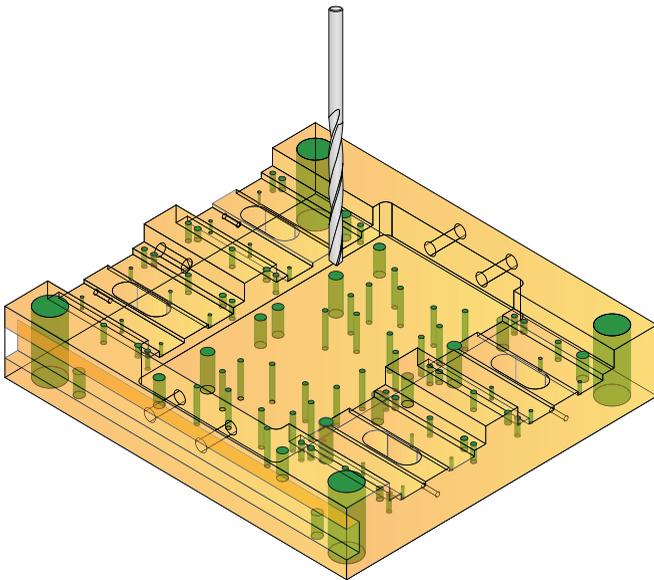
Pocket Recognition Operation

This operation enables you to recognize pocket features on a solid model and perform machining of these features.



Drill Recognition Operation

This operation performs powerful drill feature recognition and automatic Drill geometry creation. 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.



A Geometry for Drill Recognition and Pocket Recognition operations is determined automatically by SolidCAM AFRM functionality.

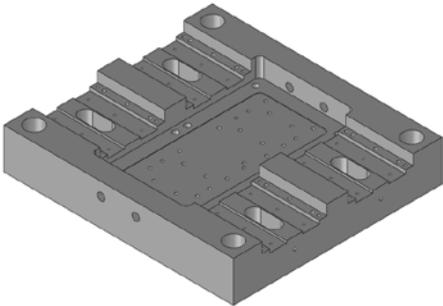
Exercise #15: Pocket Recognition

This exercise illustrates the SolidCAM functionality of automatic pocket recognition and machining.

This exercise reinforces the following skills:

- Automatic pocket recognition and machining

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



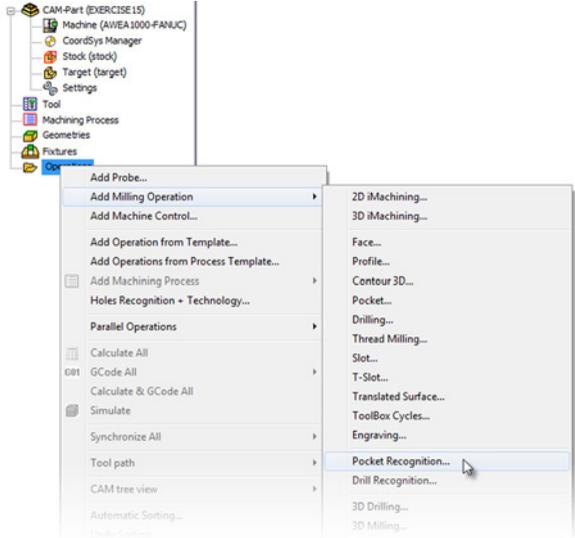
1. Open the CAM-Part

Open the **Exercise15.prz** file located in the **Exercises** folder. This file contains the pre-defined CAM-Part.

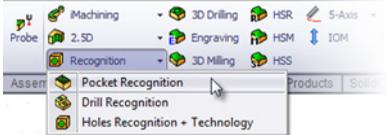


2. Add a Pocket Recognition operation

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



Alternatively, you can choose the **Pocket Recognition** icon from the **Recognition** menu on the **SolidCAM Operations** toolbar.



The **Pocket Recognition Operation** dialog box is displayed.



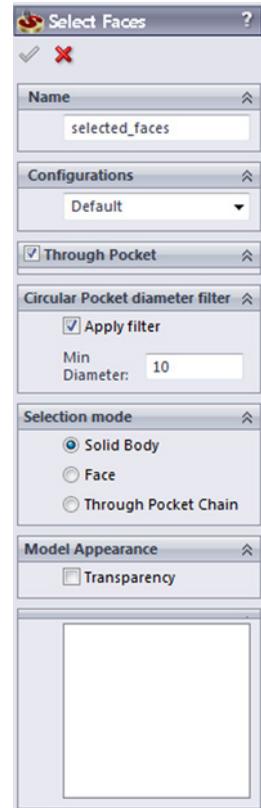
3. Define the Geometry

In the **Geometry** page, click the  button.

The **Select Faces** dialog box is displayed. This dialog box enables you to define the parameters of the pocket feature recognition geometry.

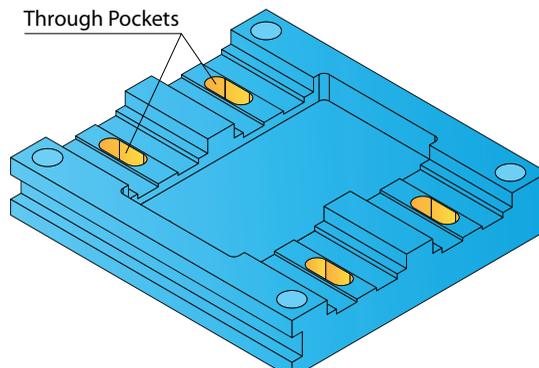
First, you need to set the pocket recognition filters and then the mode of geometry selection.

Make sure that the **Through Pocket** check box is selected.

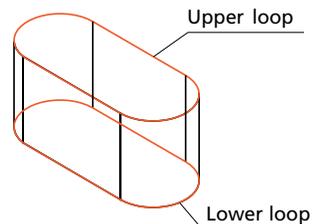


Through Pocket

When this check box is selected with the **Selection Mode** set to **Solid Body**, the recognition of through pocket features is also performed.



Only the through pocket features where the upper loop is identical to the lower loop are recognized. The through pocket features that cannot be machined with the current CoordSys position cannot be selected.



In the **Circular Pocket diameter filter** section, make sure that the **Apply filter** check box is selected.



Circular Pocket Diameter Filter

When this check box is selected with the **Selection mode** set to **Solid Body**, circular pocket features can be selected according to their diameter. All circular pocket features with the diameter greater than the specified **Min. Diameter** value will be selected. Using this option, you can avoid machining of drill features that are supposed to be machined in other operations (**Drilling, Drill recognition**, etc.).

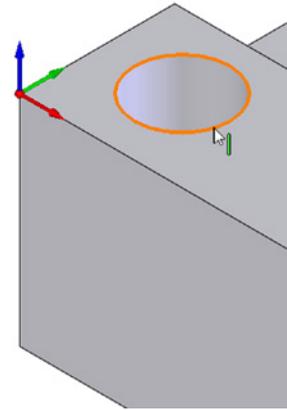
The **Min. Diameter** parameter defines the minimum diameter of the pocket features. You can enter the value in the edit box or pick a cylindrical surface or a circular edge on the solid model when the cursor is placed in the **Min. Diameter** edit box.

When the **Circular Pocket** check box is not selected, all recognized circular pocket areas suitable for the current Coordinate System are selected.

Generally, all the through holes of this part are recognized as circular pockets. In this operation, not all of these holes will be machined, only four column holes located in the corners of the top face.

Place the cursor in the **Min. Diameter** edit box and then pick the edge of the column hole on the solid model. The circular edge diameter of 42 mm is displayed in the edit box. Edit the diameter by setting the value to **40**. All of the holes with the diameter smaller than the specified value will not be machined.

Define the mode of geometry selection.

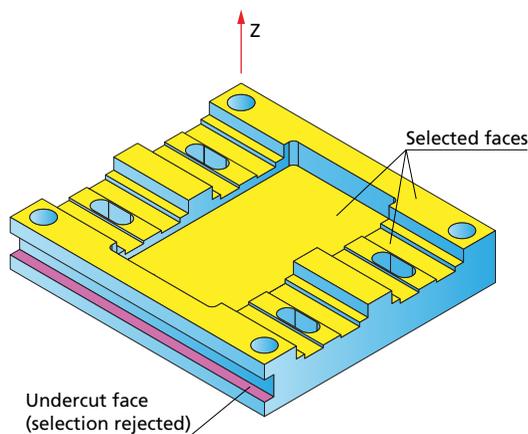


Selection Mode

This section enables you to set the mode of the geometry selection. The following selection modes are available:

- **Solid Body**

This mode enables you to perform automatic recognition of pocket features on the picked solid body. All planar faces are recognized with the surface normal vector oriented parallel to the Z-axis of the current CoordSys. The planar faces that cannot be machined with the current CoordSys position (undercuts) cannot be selected (the selection is rejected). All the through pocket areas are recognized according to the **Through pocket** option.



- **Face**

This mode enables you to pick single planar faces. When a face is picked, SolidCAM checks the parallel relation between the surface normal vector of the picked face and the positive direction of the Z-axis of the current CoordSys. If they are not parallel, the face is not selected. The undercut faces that cannot be machined with the current CoordSys position are also not selected. The selection is working in the toggling mode: the first click on a face selects it, the next click on the selected face removes the selection.

- **Through Pocket Chain**

This mode enables you to select the through pockets by picking one of the edges of the lower loop of the pocket area. When the edge is picked, the complete geometry of the through pocket is automatically selected. The lower loop is highlighted on the model. The selection is working in the toggling mode: the first click on an edge selects the loop, the next click on an edge of the already selected loop removes the selection.



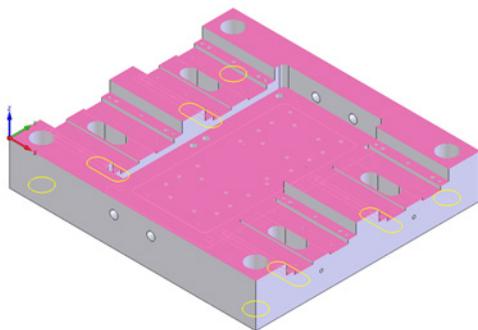
Under **Selection mode**, use the default **Solid Body** option.

Click on the solid model. SolidCAM generates the list of elements that suit your filter settings.

Click the **Solid1** element in the list. All the recognized model features that take part in this geometry are highlighted. The geometry includes all the planar faces, through pockets of slots, and circular through pockets of column holes. Each model element is accompanied by its Z-level value enclosed in parentheses.



Confirm the **Select Faces** dialog box by clicking .



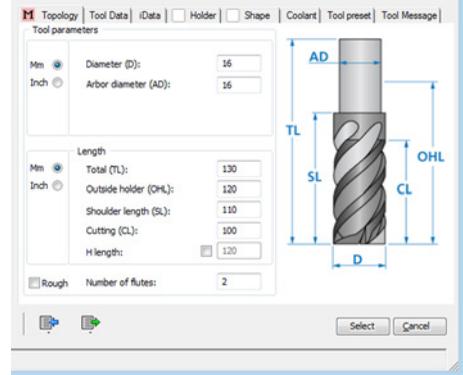


4. Define the Tool

Add an **End mill** of $\text{Ø}16$ for the operation. Define the tool parameters as follows:

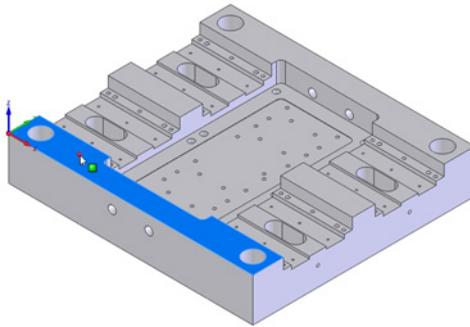
- Set the **Total length** to **130**
- Set the **Outside holder length** to **120**
- Set the **Shoulder length** to **110**
- Set the **Cutting length** to **100**

Choose the tool for the operation.



5. Define the machining levels

Define the **Upper level** by clicking on the upper model face as shown.



The cutting depth is calculated automatically for each recognized feature according to the solid model.

Select the **Equal step down** option and set the **Step down** value to **5**.



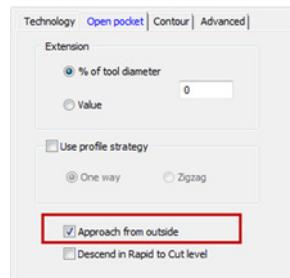


6. Define the technological parameters

Switch to the **Technology** page. All the parameters in this page are identical to those used in the Pocket operation.

Use the default parameters of pocket machining.

Since the recognized geometry contains open pocket areas, the **Open pocket** tab is made available. Switch to the **Open pocket** page and select the **Approach from outside** check box. With this option, the approach to the open pocket area is performed from the outside of the material.



7. Define the Ramping strategy

In the **Link** page, choose the **Helical** ramping strategy and set the **Radius** to **5**. The chosen ramping strategy affects the close pocket areas where the horizontal approach from outside (defined in the previous step) is impossible.



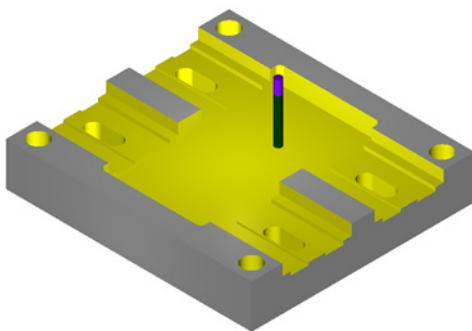
8. Calculate the tool path

Click the **Save & Calculate** button.



9. Simulate the operation

Simulate the operation in the **SolidVerify** mode.



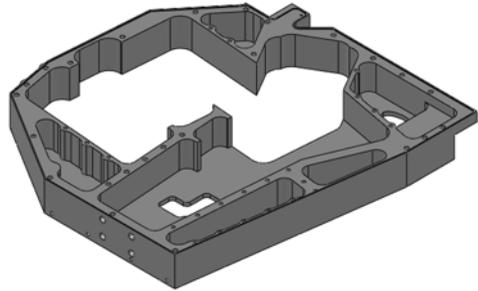
Now you have successfully finished the exercise.

Exercise #16: Mounting Box Machining

In this exercise, the SolidCAM functionality of automatic pocket recognition and machining is used for rough milling of the mounting box inner surfaces.

This exercise reinforces the following skills:

- CAM-Part definition
- Profile machining
- Automatic pocket recognition and machining
- Rest material machining



The SolidWorks model of the mounting box part (**Exercise16.sldprt**) is located in the **Exercises** folder.

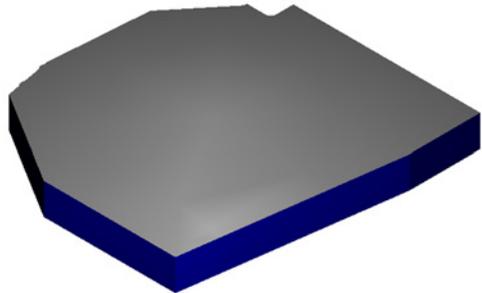
The following steps have to be implemented in order to reach the final CAM-Part:

1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**AWEA1000-Fanuc**), Machine Coordinate System, the Stock and Target models.

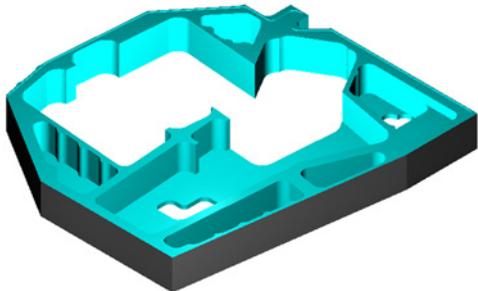
2. External contour machining

Define a Profile operation to perform rough machining of the external contour of the part. Use an **End mill** tool of **Ø20**.



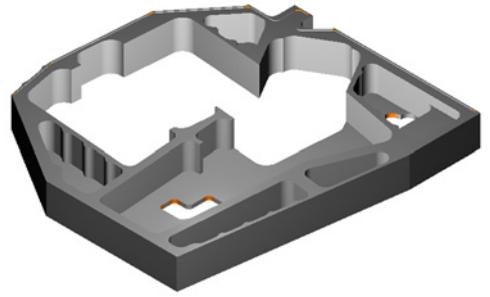
3. Pocket roughing

Define a Pocket Recognition operation to remove the bulk of material and perform the rough machining of the part inner surfaces. Use the **End mill** tool of **Ø20** used in the previous operation.



4. Pocket rest material machining

Define a Pocket Recognition operation to perform the rest material machining of the part inner surfaces. Use an **End mill** tool of **Ø10**.



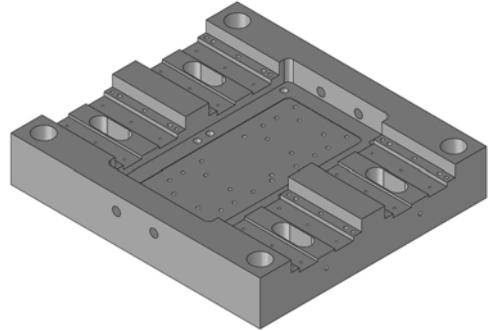
Exercise #17: Drill Recognition

This exercise illustrates the SolidCAM functionality of automatic drill recognition and machining.

This exercise reinforces the following skills:

- Automatic drill recognition and machining

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



1. Open the CAM-Part

Open the **Exercise17.prz** file located in the **Exercises** folder. This file contains the CAM-Part machined in the **Pocket Recognition Exercise**.



2. Add a Drill Recognition operation

Add a new Drill Recognition operation to perform center drilling of holes.



3. Define the Geometry

In the **Geometry** page, click the  button.

SolidCAM parses the solid model and recognizes all of its hole features.

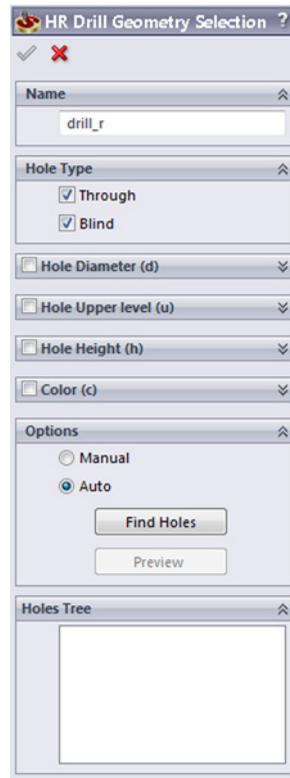
The **HR Drill Geometry Selection** dialog box is displayed. This dialog box enables you to specify the holes to be included in the operation geometry.

In this operation, you will perform the center drilling of all the holes that can be machined using the defined CoordSys position, except for the column holes that were machined in the previous operation as through pocket features.

In the **Hole Type** section, make sure that both the **Through** and **Blind** check boxes are selected. In this operation, both through and blind holes will be drilled.

The **HR Drill Geometry Selection** dialog box enables you to filter the hole features to be machined by diameter, upper level and height.

Select the **Hole Diameter** check box.



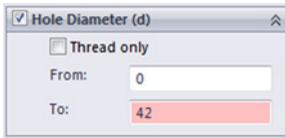
Hole Diameter (d)

When this section is activated, the hole features are filtered according to the diameter of the hole. With this filter, only the hole features with the **Hole Diameter** value within the specified range are recognized.

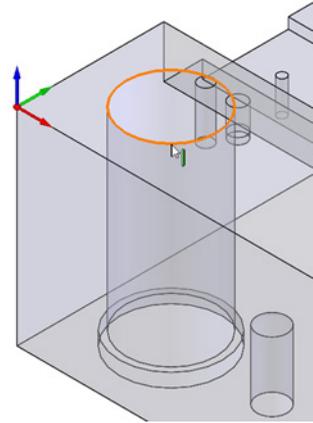
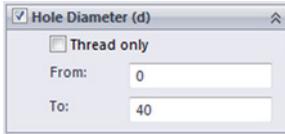
The **From** and **To** values enable you to define the **Diameter range** by entering the values or by picking on the solid model. When the cursor is placed in the **From/To** edit box, you can specify the diameter value by picking a specific cylindrical surface or a circular edge on the solid model. When a cylindrical surface or a circular edge is picked, its diameter is calculated and inserted into the corresponding edit box replacing the previous value. The edit box becomes pink. When you remove the automatically determined value, the edit box becomes white.



Define the upper limit for the diameter range. Place the cursor in the **To** edit box and pick the edge of the column hole as shown. The diameter of the hole (42 mm) is displayed in the edit box.



Edit the diameter by setting the value to **40**.



In this operation, the holes will be machined irrespective of their upper levels. Make sure that the **Hole Upper level** check box is not selected.



Hole Upper level (u)

When this section is activated, the hole features are filtered according to the **Upper level**. With this filter, only the hole features with the **Upper level** value within the specified range are recognized.

The **From** and **To** values enable you to define the **Upper level range** by entering the values or by picking on the solid model. When the cursor is placed in the **From/To** edit box, you can specify the **Upper level** value by picking on the solid model. When a model point is picked, the Z-value of the picked position is calculated and inserted into the corresponding edit box replacing the previous value. The edit box becomes pink. When you remove the automatically determined value, the edit box becomes white.



In this operation, the holes of all heights will be machined. Make sure that the **Hole height** check box is not selected.



Hole Height (h)

When this section is activated, the hole features are filtered according to the **Hole height**. With this filter, only the hole features with the **Hole height** value within the specified range are recognized.

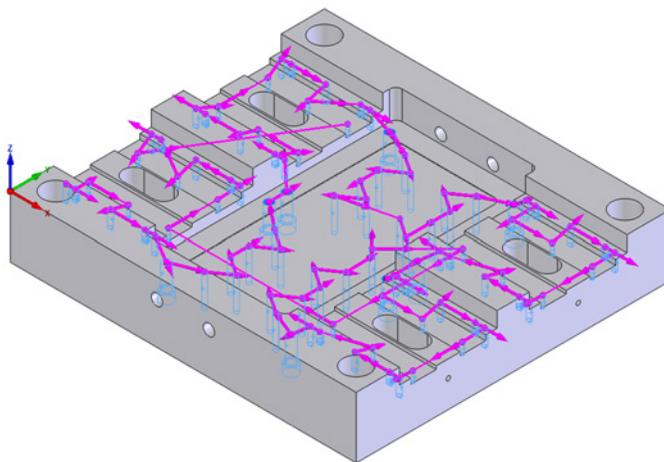
The **From** and **To** values enable you to define the **Hole height range** by entering the values or by picking on the solid model. When the cursor is placed in the **From/To** edit box, you can specify the **Hole height** value by

picking on the solid model. When a model point is picked, the Z-value of the picked position is calculated and inserted into the corresponding edit box replacing the previous value. The edit box becomes pink. When you remove the automatically determined value, the edit box becomes white.



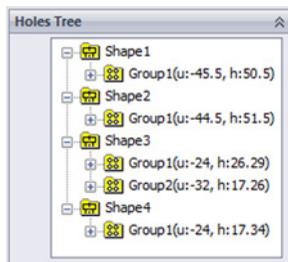
Click the **Find Holes** button. The hole features that match the specified filter criteria are determined. These features are listed in the **Holes Tree** section.

Click the **Preview** button. The hole features are highlighted on the model.



Holes Tree

In this list, the recognized hole features are classified by **Shapes** and **Groups**. A **Shape** contains all the hole features with the same topology; within a **Shape**, the **Groups** include all the hole features with the same parameter values (**Upper Level**, **Height** and **Diameter**). For each **Group**, the **Upper level (u)** and **Height (h)** are displayed in parentheses. When you select a **Shape** or a **Group** item in the list, the hole features that belong to this **Shape** or **Group** are highlighted on the model.



Confirm the **HR Drill Geometry Selection** dialog box by clicking .

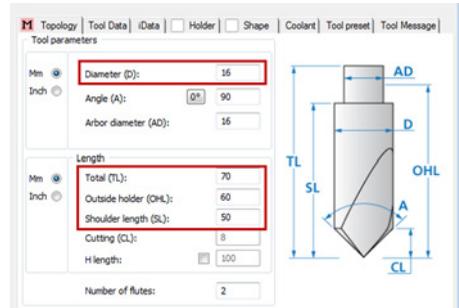


4. Define the Tool

Add a new **Spot drill** tool of $\varnothing 16$. Define the tool parameters as follows:

- Set the **Total length** to **70**
- Set the **Outside holder length** to **60**
- Set the **Shoulder length** to **50**

Choose the tool for the operation.

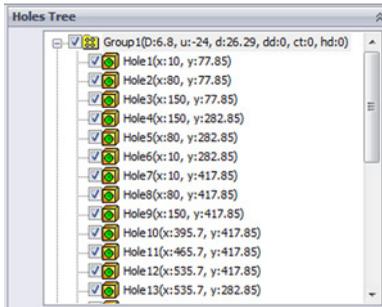


5. Define the Levels

Click the **Depth Edit** button in the **Levels** page.

The **Depth Edit** dialog box is displayed. This dialog box enables you to edit the **Milling Levels** parameters (**Upper Level**, **Drill Depth**, **Delta Depth**) for each drill separately.

Notice the list of drill instances displayed in the **Holes Tree** section.





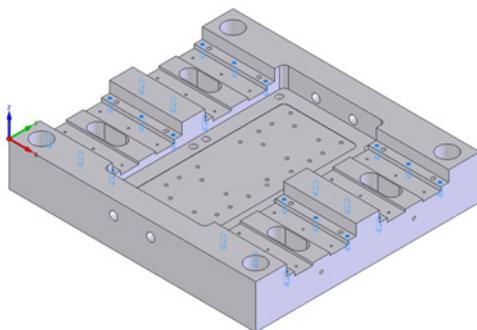
Holes Tree

This section displays the list of all drill instances chosen for the geometry. All the drill instances in the list are structured into **Groups** with the same **Upper Level**, **Drill Depth**, **Delta Depth** and **Depth Type**. For each **Group** in the list, the drill instances data are displayed in parentheses. Each drill instance in the list is accompanied by the X- and Y-coordinates of the drilling position. Selecting the check boxes next to each element in the list enables you to include the hole features, Shapes or Groups into the geometry.



Do not confuse the **Group** classification in this dialog box with that taking place at the stage of geometry definition.

By default, the **Depth Edit** dialog box displays the parameter values recognized from the solid model. Now you need to define the **Upper Level**, **Drill Depth** and **Depth Type** parameters for each group of drill instances separately.



Select the **Group1** item in the **Holes Tree** section. The drill instances that belong to this group are highlighted on the model.

First, define the upper level for drilling.



Upper Level (u)

This value can be applied to a single hole, a group or all holes.

The edit box displays the **Upper Level** value of the selected item (a group or a hole). You can enter the value or pick it directly from the model (when the cursor is placed in the edit box). SolidCAM enables you to pick planar faces parallel to the XY-plane of the Coordinate System, model vertices and sketch points. When a model is picked, the Z-value of the picked position is displayed in the edit box. The edit box has pink background, which means that the parameter is defined associatively to the solid model. When the edit box value is typed in, it is not associative to the model; in this case the edit box has white background.

The **Apply** button enables you to apply the defined **Upper Level** value to the selected list item (a group or a hole). When a new **Upper Level** value is applied, SolidCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters. SolidCAM checks all the groups to find a group with identical **Upper Level**, **Drill Depth**, **Delta Depth** and **Depth Type** parameters. If such group is found, SolidCAM adds the updated drill to this group. In case of absence of a suitable group, a new group is created; the updated drill instance is added to this group.

Leave the **Upper Level** value unchanged (-24).

Define the depth of the center drilling.



Drill Depth (d)

This value can be applied to a single hole, a group or all holes.

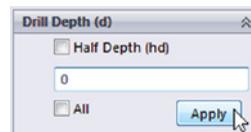
The edit box displays the **Drill Depth** value of the selected item (a group or a hole). The edit box enables you enter the value or pick the value directly from the model (when the cursor is placed in the edit box). You can pick drill faces or edges. SolidCAM determines the depth of the picked drill according to the defined **Upper Level** and displays the value in the edit box. The edit box has pink background, which means that the parameter is defined associatively to the solid model. When the edit box value is typed in, it is not associative to the model; in this case the edit box has white background.

The **Apply** button enables you to apply the defined **Drill Depth** value to the selected list item (a group or a hole). When a new **Drill Depth** value is applied, SolidCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters. SolidCAM checks all groups to find a group with identical **Upper Level**, **Drill Depth**, **Delta Depth** and **Depth Type** parameters. If such group is found, SolidCAM adds the updated drill to this group. In case of absence of a suitable group, a new group is created; the updated drill instance is added to this group.

Set the **Drill Depth** value to **0** and click the **Apply** button.



Notice that if you do not click the **Apply** button after editing values, the change does not take effect.

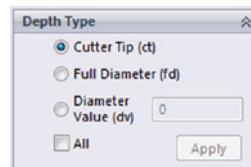


Define the **Depth Type**.



Depth Type

This parameter can be defined for a single hole, a group or all holes. You can define the diameter on the conical part of the drilling tool that will reach the specified drilling depth during the machining. You can also deepen a drilled hole in order to obtain a given diameter at the specified drill depth.



The following options are available:

- **Cutter Tip**

The drill tip reaches the defined drilling depth.

- **Full Diameter**

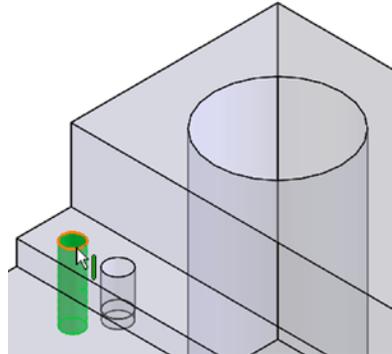
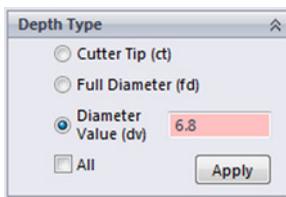
The drill reaches the defined drilling depth with the full diameter.

- **Diameter Value**

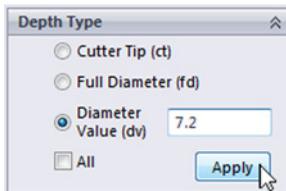
The drill reaches the defined drilling depth with the drill cone diameter specified by the **Diameter Value** parameter.

The **Apply** button enables you to apply the defined **Depth Type** to the selected list item (a group or a hole). When a new **Depth Type** is applied, SolidCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters. SolidCAM checks all groups to find a group with identical **Upper Level**, **Drill Depth**, **Delta Depth** and **Depth Type** parameters. If such group is found, SolidCAM adds the updated drill to this group. In case of absence of a suitable group, a new group is created; the updated drill instance is added to this group.

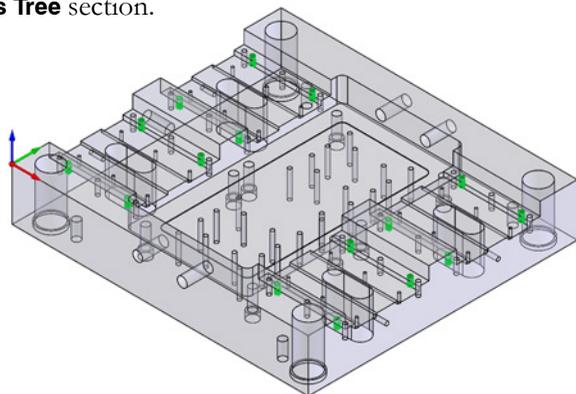
Choose the **Diameter Value** option for **Depth Type**. Place the cursor in the edit box and pick the upper edge of one of the **Group1** holes on the model. The diameter of the hole (6.8 mm) is displayed in the edit box.



Edit this value to add a 0.2 mm chamfer to the holes. Set the **Diameter Value** to **7.2** and click the **Apply** button.



Select the **Group2** item in the **Holes Tree** section.

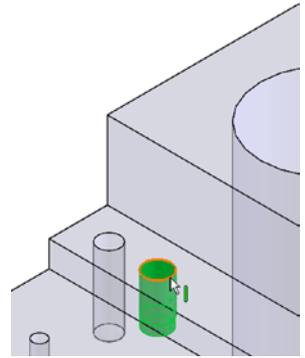


Leave the **Upper Level** value unchanged (-24).

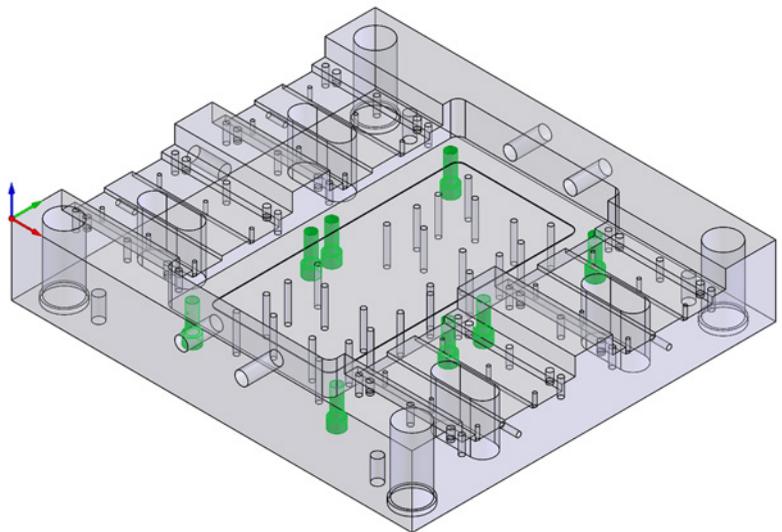
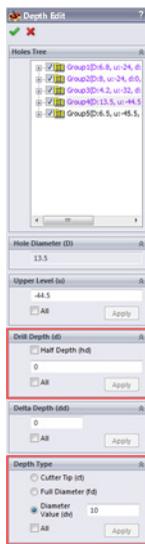
Set the **Drill Depth** value to **0** and click the **Apply** button.

Under **Depth Type**, choose the **Diameter Value** option, place the cursor in the edit box and pick the upper edge of the hole. The value displayed in the edit box is **8**. To add a 0.5 mm chamfer to the holes, set the **Diameter Value** to **9** and click the **Apply** button.

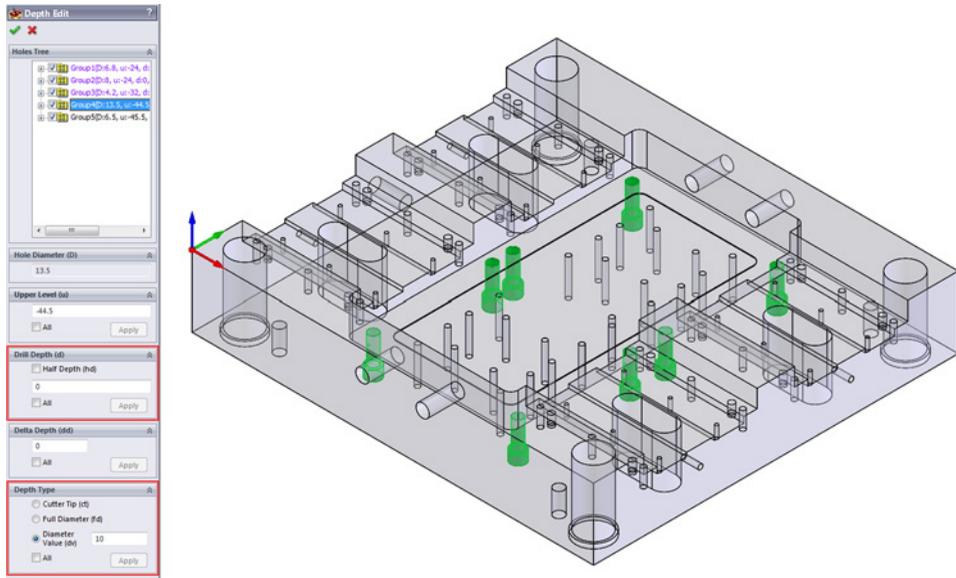
In the same manner, define the levels for the rest of the groups.



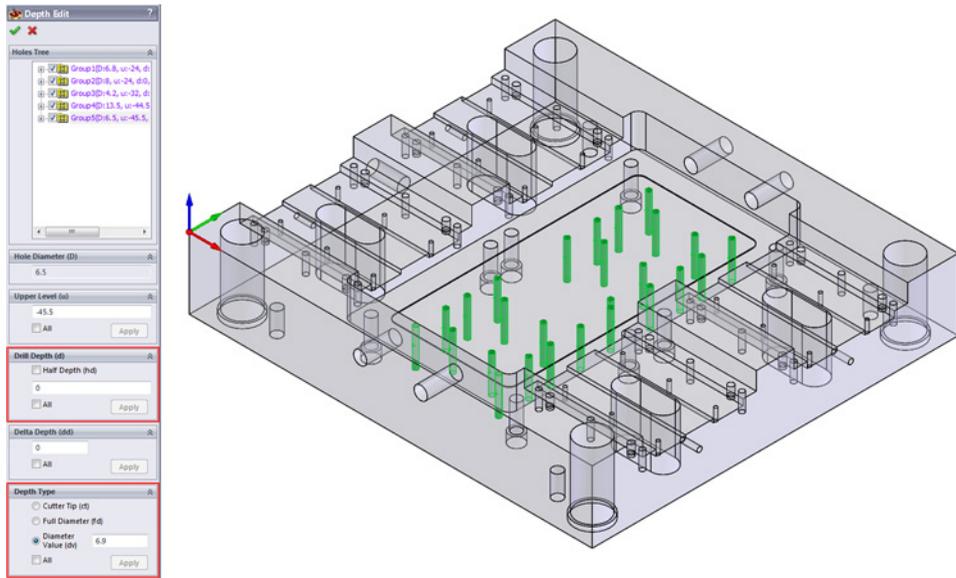
For **Group3**, set the **Drill Depth** to **0** and the **Diameter Value** to **4.6** to add a 0.2 mm chamfer. Click the **Apply** button.



For **Group4**, set the **Drill Depth** to **0** and the **Diameter Value** to **10**. This value will not result in a chamfer, because the diameter of the holes is greater, therefore only center drilling will be performed. Click the **Apply** button.



For **Group5**, set the **Drill Depth** to **0** and the **Diameter Value** to **6.9** to add a 0.2 mm chamfer. Click the **Apply** button.



Apply the changes and confirm the **Depth Edit** dialog box with .

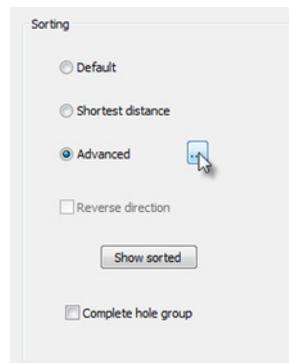
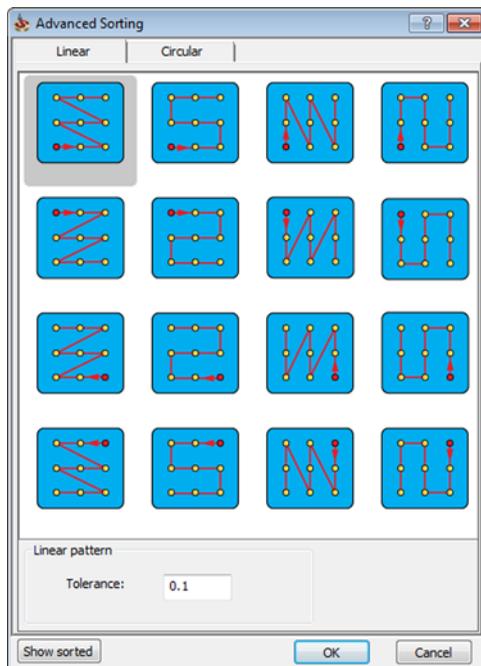


6. Define the sorting of drill positions

Switch to the **Technology** page.

Use the advanced sorting option to sort the drill positions of the geometry. Under **Sorting**, choose the **Advanced** option and click .

The **Advanced Sorting** dialog box is displayed. This dialog box enables you to define the parameters of sorting of drill positions for machining of linear and circular drill patterns.



Linear sorting of drill positions

The **Linear** tab contains sorting methods appropriate for machining of linear drill patterns.

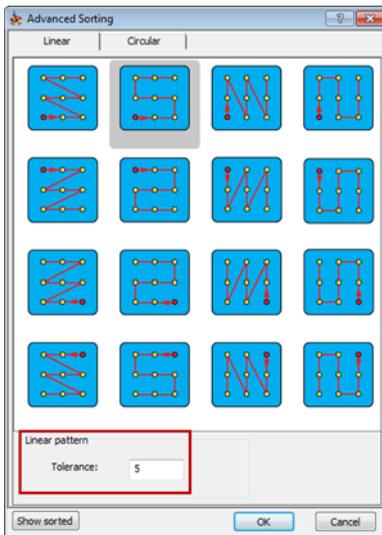
Linear pattern

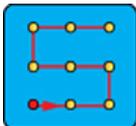
According to linear sorting methods, the machining is performed either by rows of drill positions with the same Y-coordinate or by columns of drill positions with the same X-coordinate. The **Tolerance** parameter enables you to define the tolerance that is used to determine if a drill position belongs to the current row/column or to a different row/column.

When the machining is performed by rows, SolidCAM determines the first drill position for each row (the start position of the first row is the start position of the whole pattern). For each next drill position, SolidCAM determines the Y-coordinate and compares it with the Y-coordinate of the first drill position of the current row. If the linear deviation is within the specified **Tolerance** limit, SolidCAM considers this position to belong to the current row. If the deviation exceeds the **Tolerance** value, the drill position is rejected from the current row; later this position will be checked again for belonging to next rows.

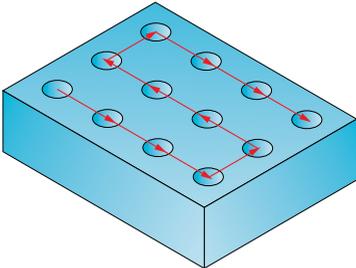
In the similar manner, when the machining is performed by columns, the specified **Tolerance** value is used to determine whether a specific drill position belongs to the current column. In this case, the deviation of X-coordinates between the first drill position in the column and the current drill position is compared with the **Tolerance** value.

Choose the  option in the **Linear** tab of the **Advanced Sorting** dialog box, and set the **Tolerance** value to **5**.





When this option is chosen, SolidCAM determines the position with the **minimal X- and Y-coordinates** and accepts it as the start position of the drill sequence. From this position, SolidCAM continues the machining of the drills with the same Y-coordinate; the machining is performed in the direction of increasing X-coordinates.



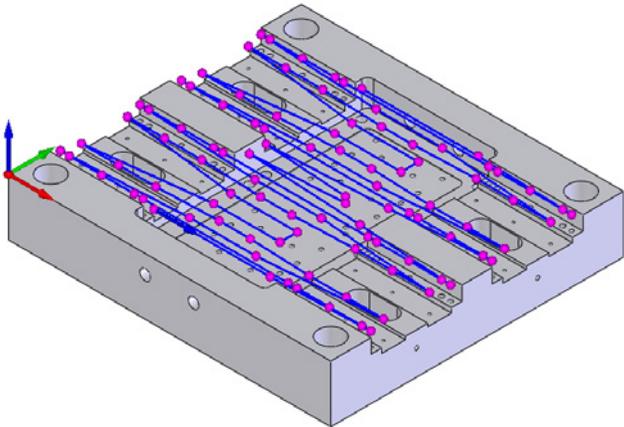
When the last position of the current Y-coordinate is machined, SolidCAM passes to the next Y-coordinate (in the increasing order) and continues the machining of the next row from the position with the maximal X-coordinate to the position with the minimal X-coordinate (decreasing order), and so on. In such manner, the machining of two successive rows is performed in the opposite order: each odd row is machined in the direction of increasing X-coordinates, each even row is machined in the direction of decreasing X-coordinates.

The rows are machined in the direction of increasing Y-coordinates.



In the **Advanced Sorting** dialog box, click the **OK** button to confirm the definition of sorting parameters.

Click the **Show Sorted** button to preview the sorted drill positions on the solid model.



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



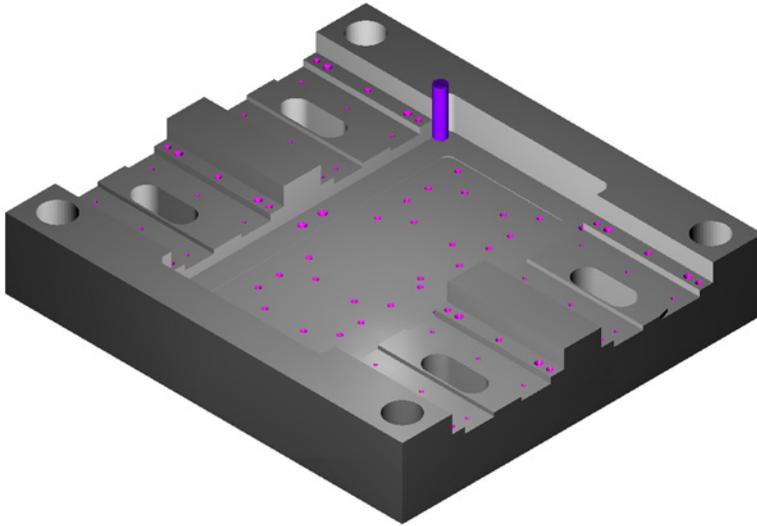
7. Calculate the tool path

Click the **Save & Calculate** button.



8. Simulate the operation

Simulate the operation in the **SolidVerify** mode.

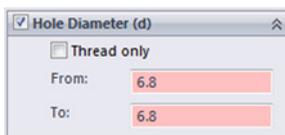
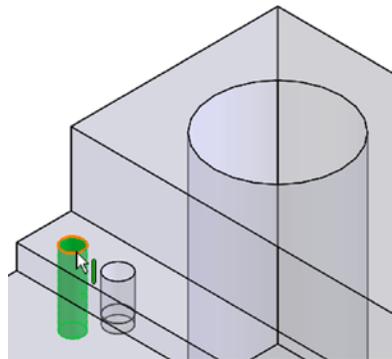


9. Machine the Ø6.8 holes

Add a new Drill Recognition operation to perform drilling of the **Ø6.8** holes.

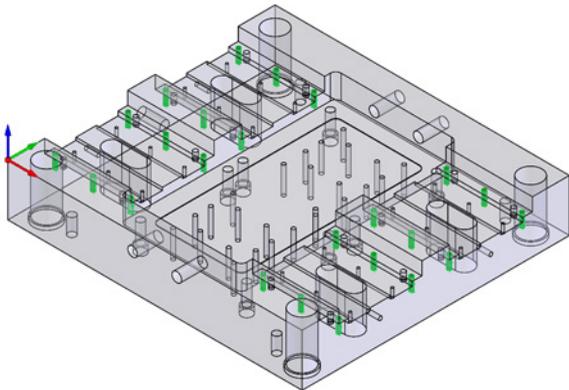
In the **HR Drill Geometry Selection** dialog box, activate the **Hole Diameter** section. Place the cursor in the **From** edit box and pick the hole edge as shown.

The diameter value (**6.8**) is displayed in both **From** and **To** edit boxes defining the range for the **Hole Diameter** filter.

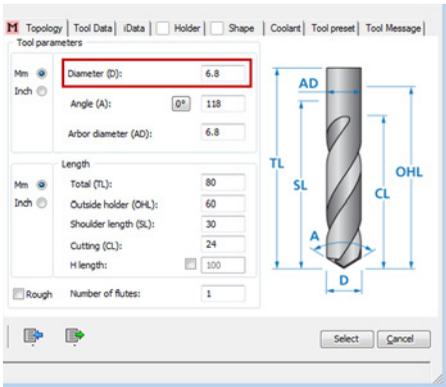


Click the **Find Holes** button. The holes of the specified diameter are highlighted on the model.

Confirm the **HR Drill Geometry Selection** dialog box by clicking



Add a new **Ø6.8 Drill** tool.



Click the **Depth Edit** button in the **Levels** page. The **Depth Edit** dialog box is displayed. Note that the values of the **Upper Level (-24)** and **Drill Depth (26.29)** parameters are offered you as defaults. These values are recognized automatically to the model.

Confirm the **Depth Edit** dialog box dialog box by clicking

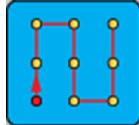
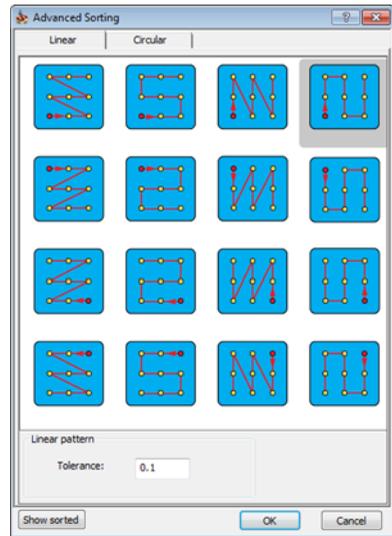


In the **Technology** page, choose the **Advanced** sort option under **Sorting** and click the  button.

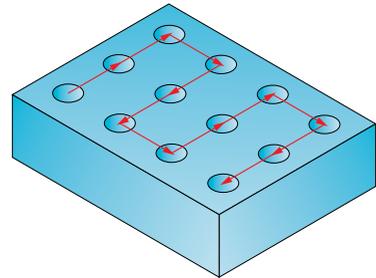


In the **Linear** tab of the **Advanced Sorting** dialog

box, choose the  option.



When this option is chosen, SolidCAM determines the position with the **minimal X- and Y-coordinates** and accepts it as the start position of the drill sequence. From this position, SolidCAM continues the machining of the drills with the same X-coordinate; the machining is performed in the direction of increasing Y-coordinates.

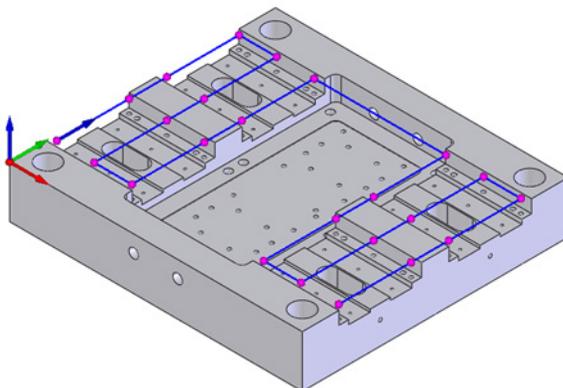


When the last position of the current X-coordinate is machined, SolidCAM passes to the next X-coordinate (in the increasing order) and continues the machining of the next column from the position with the maximal Y-coordinate to the position with the minimal Y-coordinate (decreasing order), and so on. In such manner, the machining of two successive columns is performed in the opposite order: each odd column is machined in the direction of increasing Y-coordinates, each even column is machined in the direction of decreasing Y-coordinates.

The columns are ordered in the direction of increasing X-coordinates.

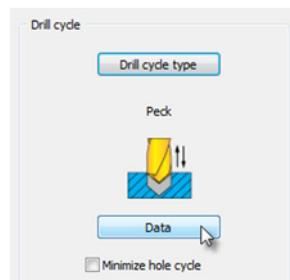
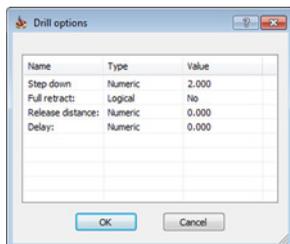


Click the **Show Sorted** button to preview the sorted drill positions on the solid model.



Under **Drill cycle**, click the **Drill cycle type** button and choose the **Peck** type from the list. Click the **Data** button to define the pecking parameters.

In the displayed **Drill Options** dialog box, set the **Step down** value to **2** and confirm the dialog box with the **OK** button.



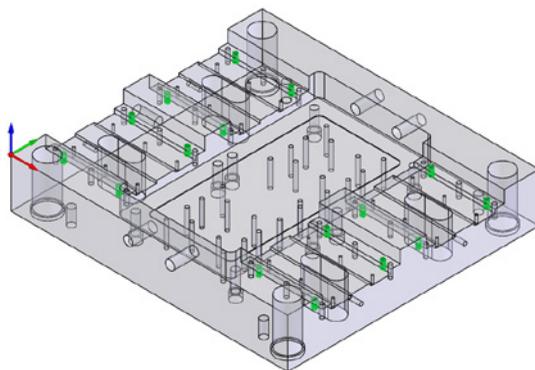
Save and calculate the operation.



10. Machine the Ø8 holes

Add a new Drill Recognition operation to perform drilling of the **Ø8** holes.

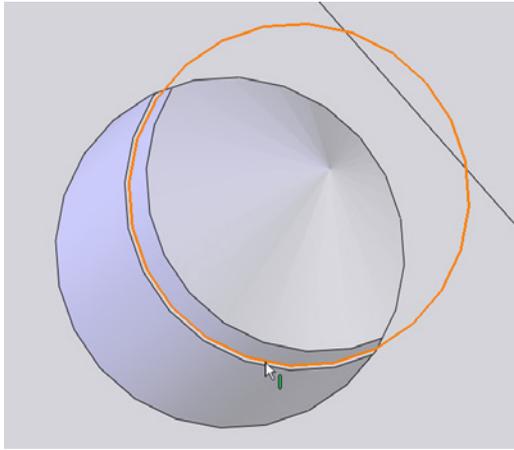
In the **HR Drill Geometry Selection** dialog box, activate the **Hole Diameter** section. Define the range for the **Hole Diameter** filter as explained in the previous step. The values of **8 mm** are displayed in the **Hole Diameter** section.



Add a new **Drill** tool. The diameter of the drilling tool must be smaller than the diameter of the holes, because these holes will be reamed at the next stage.



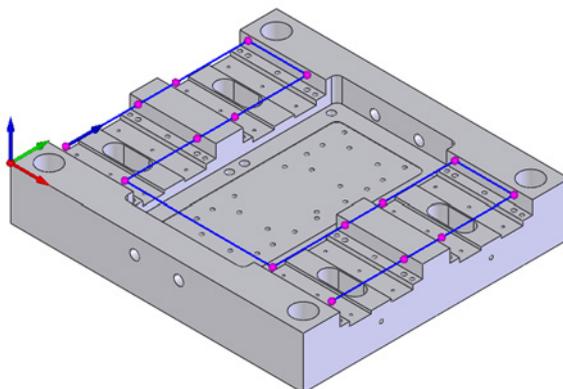
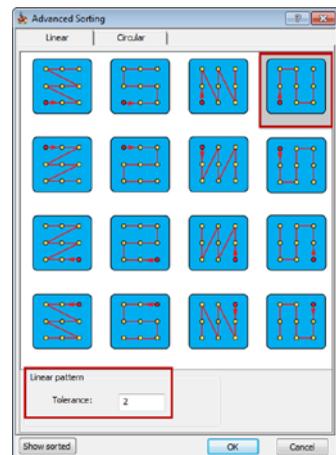
You can define the diameter of the tool by clicking on the entity to be machined with this tool directly on the solid model. Place the cursor in the **Diameter** edit box in the **Topology** page of the **Part Tool Table** and pick the edge of the hole as shown.



The **Diameter** value (**7.8**) is displayed in the edit box.

In the **Technology** page, choose the **Advanced** option under **Sorting** and click the button.

In the **Linear** tab of the **Advanced Sorting** dialog box, choose the option, set the **Tolerance** value to **2** and click the **Show sorted** button.



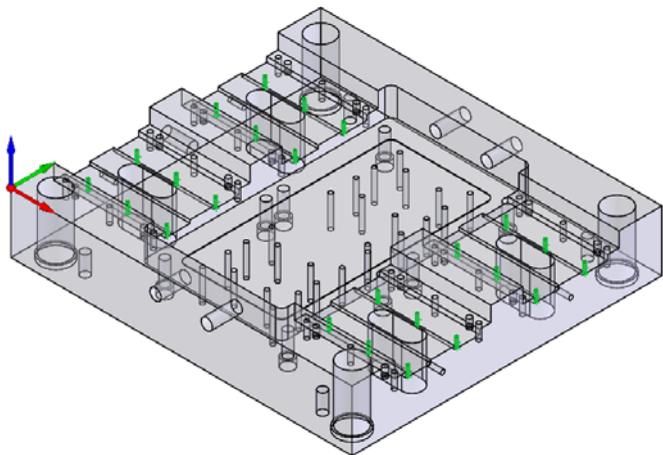
Under **Drill cycle**, choose the **Peck** cycle type and set the **Step down** value to **2**.

Save and calculate the operation.



11. Machine the Ø4.2 holes

Add a new Drill Recognition operation to perform drilling of the **Ø4.2** holes.



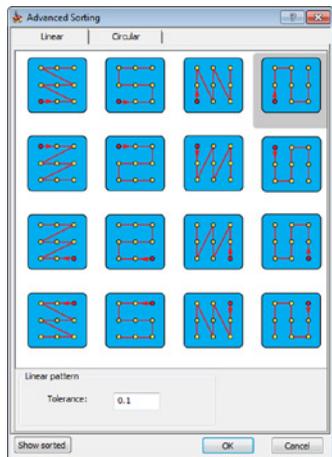
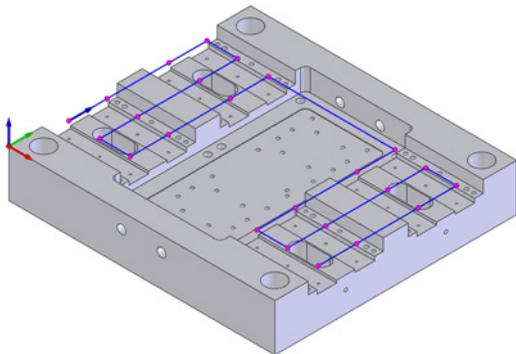
In the **HR Drill Geometry Selection** dialog box, define the range for the **Hole Diameter** filter as explained in the previous steps. The values of **4.2 mm** will be displayed in the **Hole Diameter** section.

Add a new **Ø4.2 Drill** tool.

In the **Technology** page, choose the **Advanced** option under **Sorting** and click the  button.

In the **Linear** tab of the **Advanced Sorting** dialog box,

choose the  option and click the **Show Sorted** button.



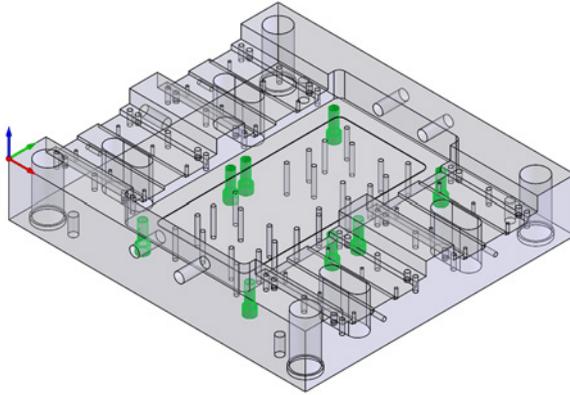
Under **Drill cycle**, choose the **Peck** cycle type and set the **Step down** value to **2**.

Save and calculate the operation.



12. Machine the Ø13.5 holes

In the same manner as explained in the previous steps, perform drilling of the set of **Ø13.5** holes.

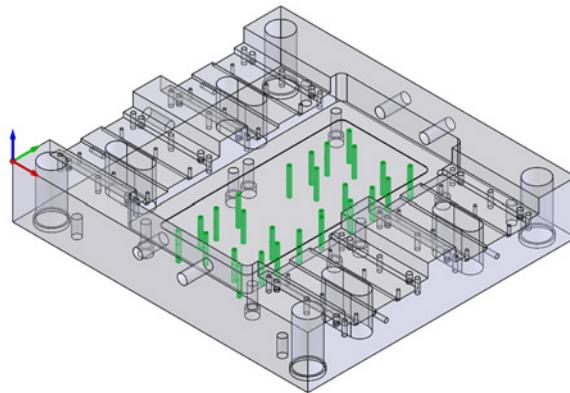


Use the **Default** drill sorting option.

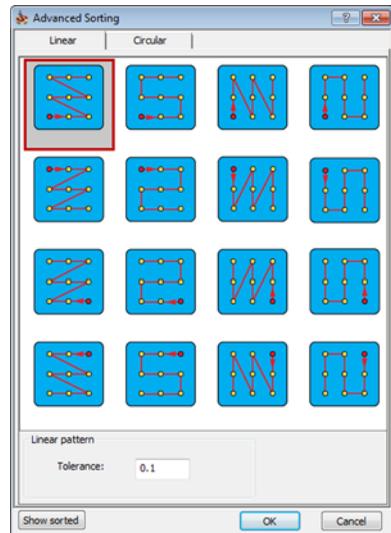


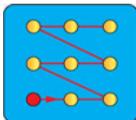
13. Machine the Ø6.5 holes

In the same manner as explained in the previous steps, perform drilling of the set of **Ø6.5** holes.

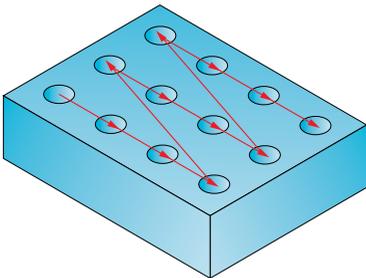


Use the **Advanced** drill sorting option.





When this option is chosen, SolidCAM determines the position with the **minimal X- and Y-coordinates** and accepts it as the start position of the drill sequence. From this position, SolidCAM continues the machining of the drills with the same Y-coordinate; the machining is performed in the direction of increasing X-coordinates.

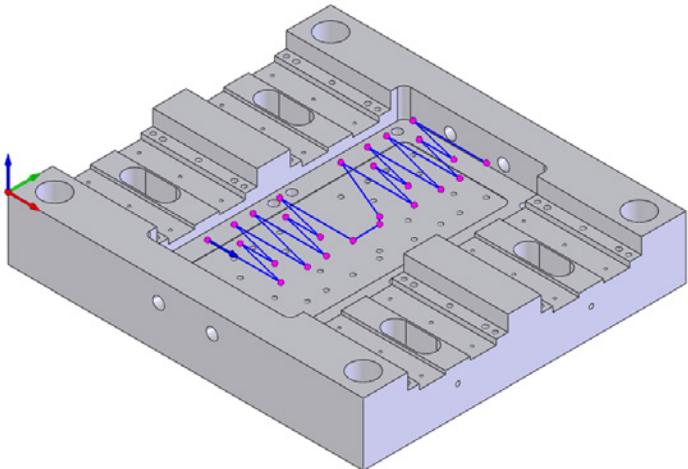


When the last position of the current Y-coordinate is machined, SolidCAM passes to the next Y-coordinate (in the increasing order) and continues the machining of the next row from the position with the minimal X-coordinate to the position with the maximal X-coordinate (increasing order) and so on. In such manner, the machining of the rows is performed in the same direction of increasing X-coordinates.

The ordering of rows is performed in the direction of increasing Y-coordinates.



Click the **Show Sorted** button to preview the sorted drill positions on the solid model.



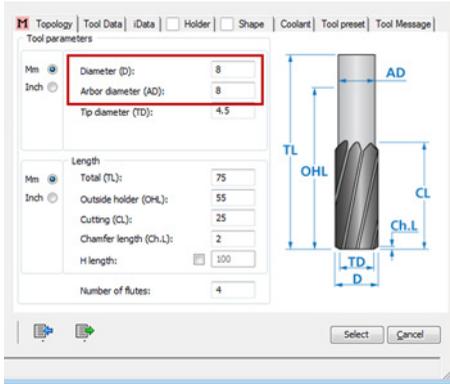


14. Perform reaming of the Ø8 holes

Add a new Drill Recognition operation to perform reaming of the Ø8 holes.

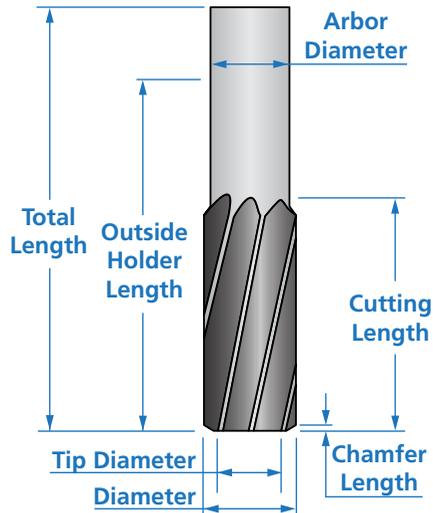
Use the **Drill_r2** geometry defined in **Step #10**.

Add a new **Ø8 Reamer** tool.



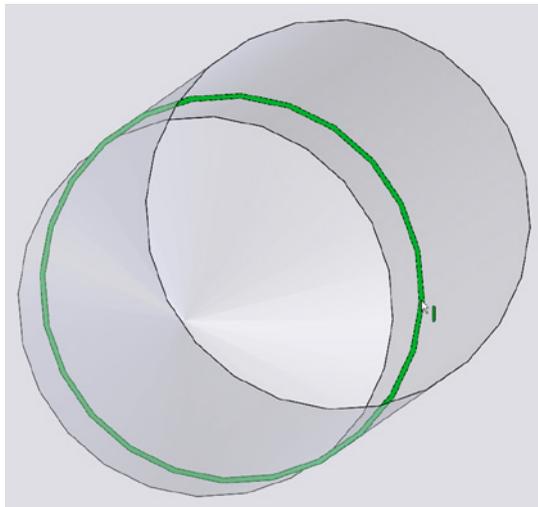
Reamer

This tool is used to machine precise holes in Drilling operations. A tool of this type is defined with the parameters shown in the image.





To define the depth, pick the model face as shown.



The **Drill Depth** value (12) is displayed. Click the **Apply** button.

Define the sorting of drill positions as explained in **Step #10**.

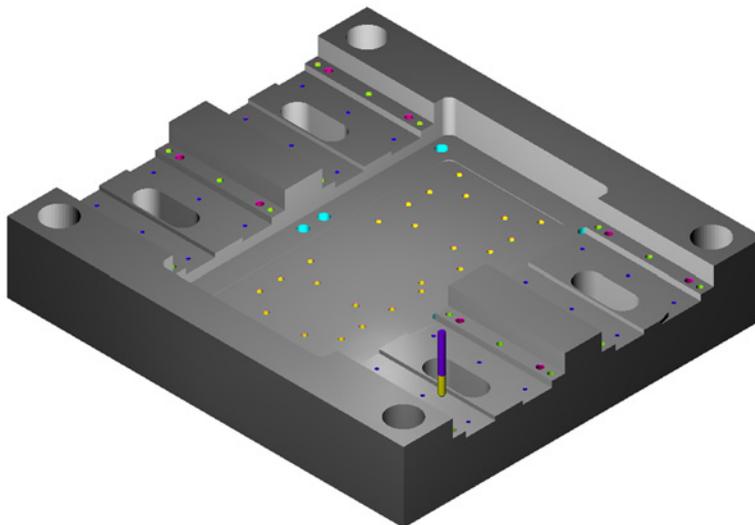
Click the **Save & Calculate** button.



15. Simulate the operations

Simulate the operations defined in **Steps #9 – #14** in the **SolidVerify** mode.

Now you have successfully finished the exercise.



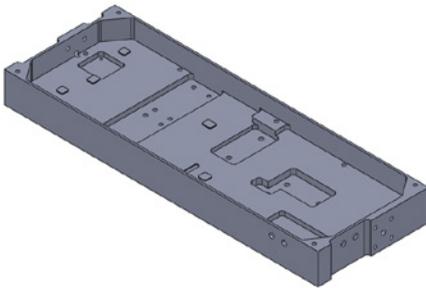
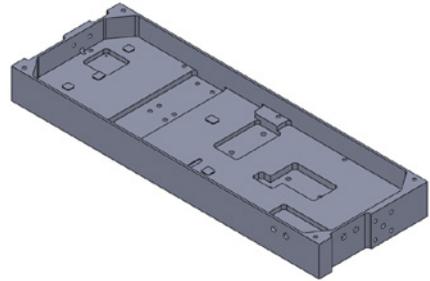
Exercise #18: Electronic Box Machining

In this exercise, the SolidCAM functionality of automatic drill recognition and machining is used for rough milling of the electronic box hole features.

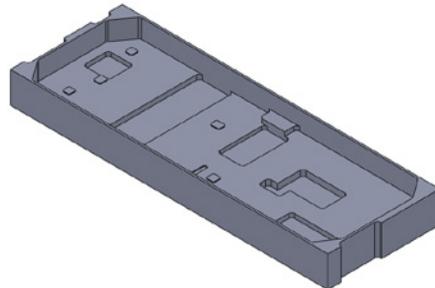
This exercise reinforces the following skills:

- Automatic drill recognition and machining

The electronic box part file (**Exercise18.prz**) is located in the **Exercises** folder. This file contains the defined CAM-Part. The model used for the part definition contains two configurations: **Default** (model with hole features) and **Stock** (hole features suppressed).



Default



Stock

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

1. Center drilling

Define a Drill Recognition operation to perform center drilling of all hole features. Use a **Spot drill** tool of **Ø2.5**.



2. Drilling

Define a Drill Recognition operation to perform drilling of the $\varnothing 1.2$ holes. Use a **Drill** tool of $\varnothing 1.2$.



3. Drilling

Define a Drill Recognition operation to perform drilling of the $\varnothing 1.8$ holes. Use a **Drill** tool of $\varnothing 1.8$.



4. Drilling

Define a Drill Recognition operation to perform drilling of the $\varnothing 2.2$ holes. Use a **Drill** tool of $\varnothing 2.2$.



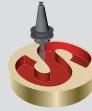
Congratulations!

You have successfully finished the **2.5D Milling** part of the **SolidCAM 2013 Milling Training Course**.



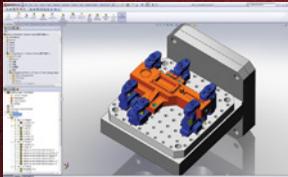
SolidCAM + SolidWorks

The complete integrated Manufacturing Solution

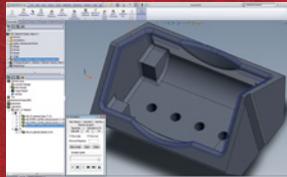


SolidCAM

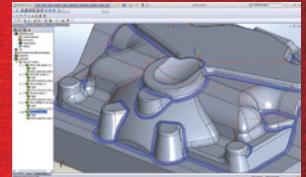
iMaching – The Revolution in CAM!



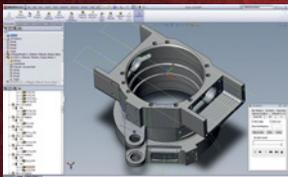
2.5D Milling



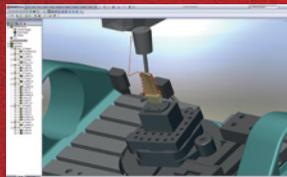
HSS (High-Speed Surface Machining)



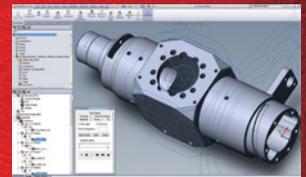
HSM (High-Speed Machining)



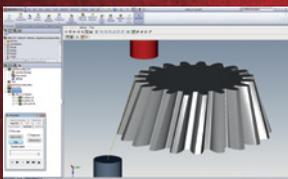
Indexed Multi-Sided Machining



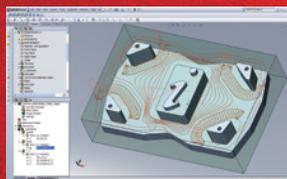
Simultaneous 5-Axis Machining



Turning and Mill-Turn up to 5-Axis



Wire EDM



iMaching



Service and Support



www.youtube.com/SolidCAMProfessor
www.youtube.com/SolidCAMiMaching



www.facebook.com/SolidCAM