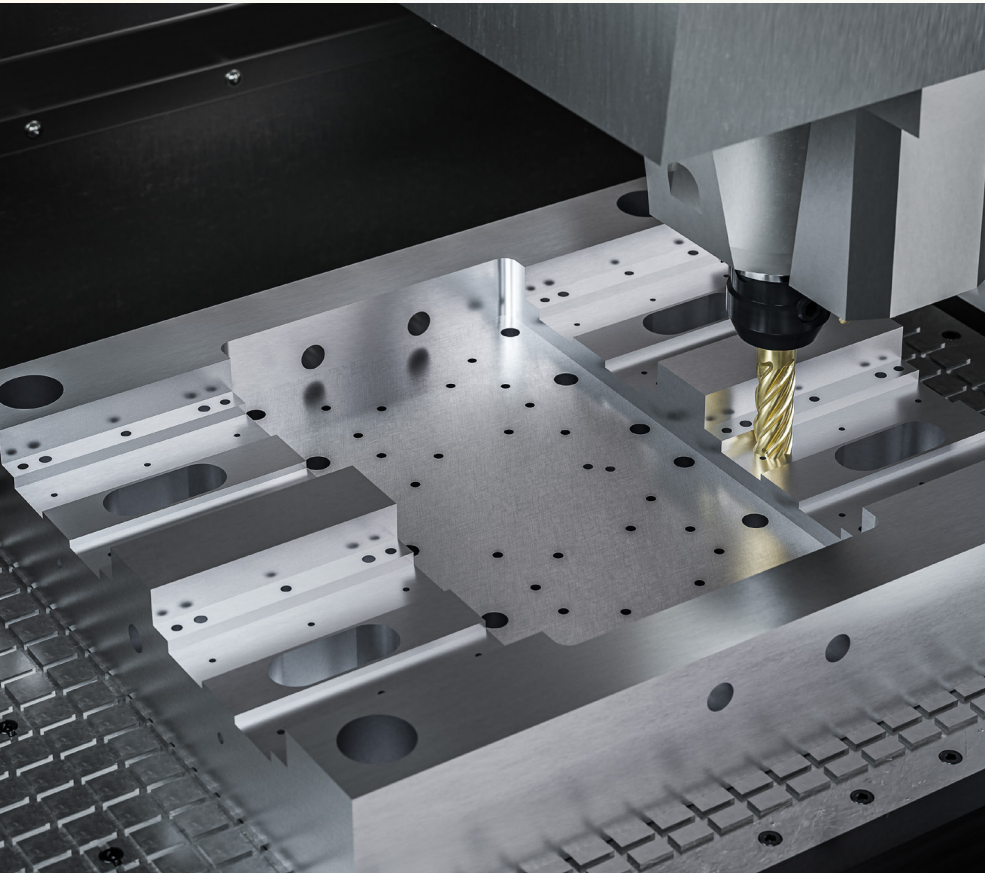




InventorCAM
The Solid Platform for Manufacturing

Training Course | InventorCAM 2023

2.5D Milling



iMachining 2D

iMachining 3D

2.5D Milling

Indexial Multi-Sided

HSS Machining

3D High Speed Milling

Simultaneous 5-Axis

Turning

Advanced Mill-Turn

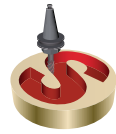
Swiss-Type

Solid Probe

The unique, revolutionary Milling technology
imachining®
patent by SolidCAM

MILLTURN + SWISS





InventorCAM
iMachining – The Revolution in CAM!

InventorCAM 2023

Milling Training Course

2.5D Milling

©1995-2023 SolidCAM

All Rights Reserved.

Contents

1. Introduction

1.1 About this Course	8
1.2 Basic Concepts	9
1.3 Process Overview	9

2. CAM-Part Definition

Exercise #1: CAM-Part Definition	14
--	----

3. InventorCAM 2.5D Operations

Exercise #2: Cover Machining	28
Exercise #3: Cover Machining	61
Exercise #4: Bracket Machining	104
Exercise #5: Electronic Box Machining	106
Exercise #6: Clamp Machining	109
Exercise #7: Basic Part Machining	114
Exercise #8: Cover Machining	117
Exercise #9: Mounting Base Machining	121
Exercise #10: Support Machining	127
Exercise #11: Engraving	136

4. Indexial 4-Axis Milling

Exercise #12: Frame Machining	146
Exercise #13: Mounting Machining	159

5. Indexial 5-Axis Milling

Exercise #14: Clamp Machining	166
-------------------------------------	-----

6. Toolbox

Exercise #15: Standard Cycles Machining	177
---	-----

7. Automatic Feature Recognition

Exercise #16: Pocket Recognition	191
Exercise #17: Mounting Box Machining	196
Exercise #18: Drill Recognition	197
Exercise #19: Electronic Box Machining	211
Exercise #20: Edge Deburring Recognition	213

Introduction

1

1.1 About this Course

The goal of this course is to teach you how to use InventorCAM to machine various parts applying 2.5D Milling technologies. This course covers the basic concepts of InventorCAM 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 InventorCAM 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.

Training materials



The file archive supplied 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. Unpack the complete **Exercises** folder on your computer. The Autodesk Inventor files used for the exercises were prepared with Autodesk Inventor Professional 2023.

Windows 10

The screenshots in this book were made using **InventorCAM 2023 SP2** integrated with Autodesk Inventor Professional 2023 running on Windows 10. 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 InventorCAM 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 InventorCAM functionality explanations embedded into the guided exercises. The lamp icon is also used to emphasize notes.

1.2 Basic Concepts

Every manufacturing project in InventorCAM 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 Autodesk Inventor model.

Operation – An Operation is a single machining step in InventorCAM. 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 InventorCAM 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, InventorCAM calculates how much material was actually removed from the CAM-Part and how much material remains unmachined (rest material). The rest material information enables InventorCAM to automatically optimize the tool path and avoid the air cutting.

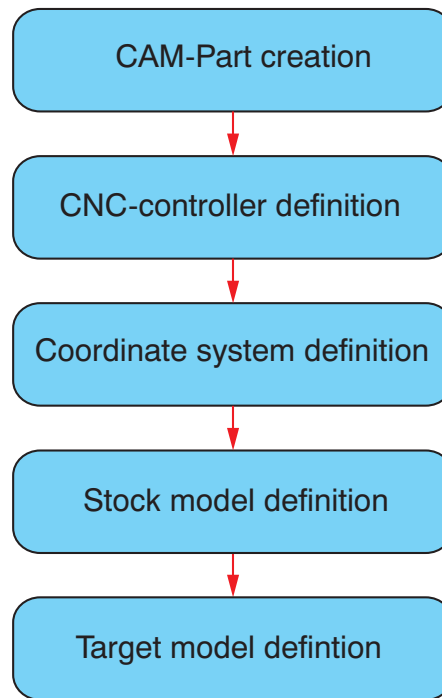
Operations definition

InventorCAM 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

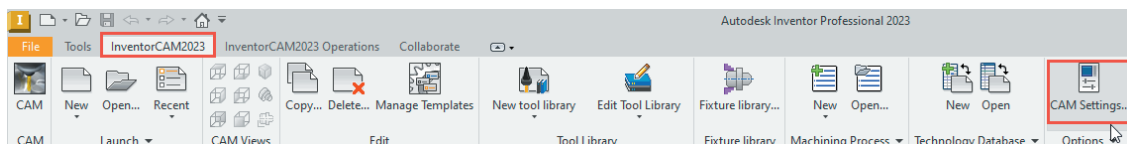
The CAM-Part definition process includes the following stages:



- **CAM-Part creation.** According to default settings, the CAM-Part name and location are defined automatically. However, you can define them manually, if you choose the manual option. At this stage, InventorCAM defines the necessary system files and a folder to allocate the place to store InventorCAM 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.** InventorCAM enables you to define the stock model that describes the initial state of the workpiece to be machined.
- **Target model definition.** InventorCAM 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 complete 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.

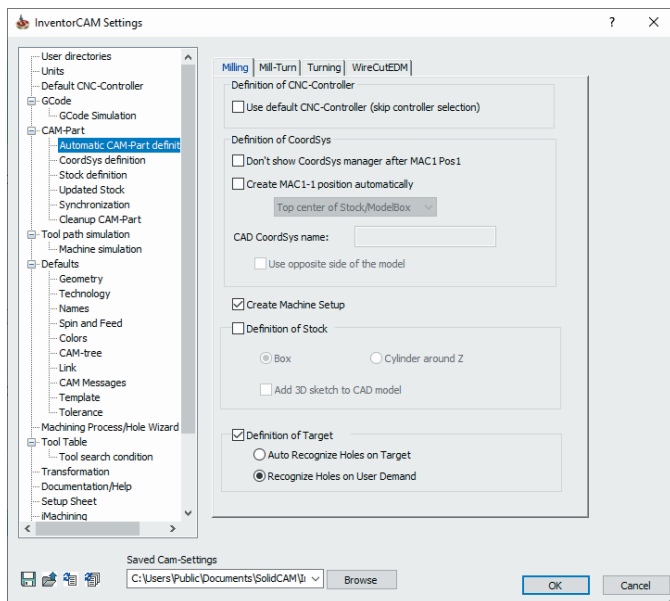
Before you start, select **CAM Settings** command from **InventorCAM 2023** ribbon.



The **InventorCAM Settings** dialog box is displayed.

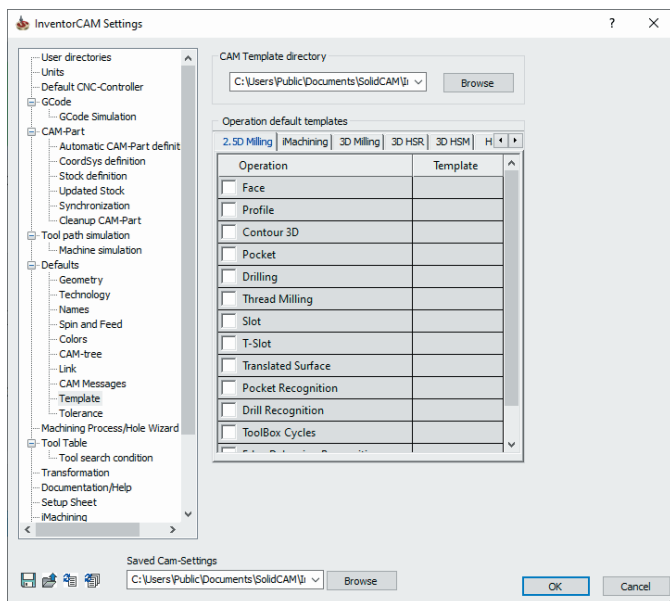
In the left pane, select **CAM-Part**. Select the **Auto select project location** check box. In the left pane, select **Automatic CAM-Part definition**.

In the right pane, click the **Milling** tab and clear check boxes under the following sections: **Definition of CNC-Controller**, **Definition of CoordSys**, **Definition of Stock**, and **Definition of Target**.



Then you have to turn off the default templates loading.

In the left pane, select **Defaults > Template**. In the right pane, under **2.5D Milling**, clear all the check boxes next to the template names.

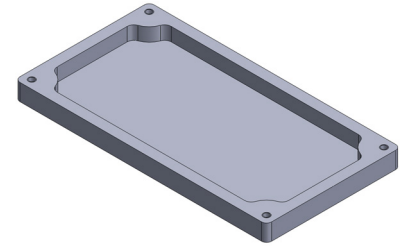


Click **OK** to confirm your choice.

These settings can be turned back on at any time.

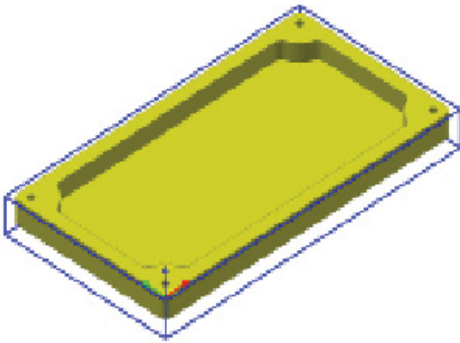
Exercise #1: CAM-Part Definition

This exercise illustrates the CAM-Part definition process in InventorCAM. 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.

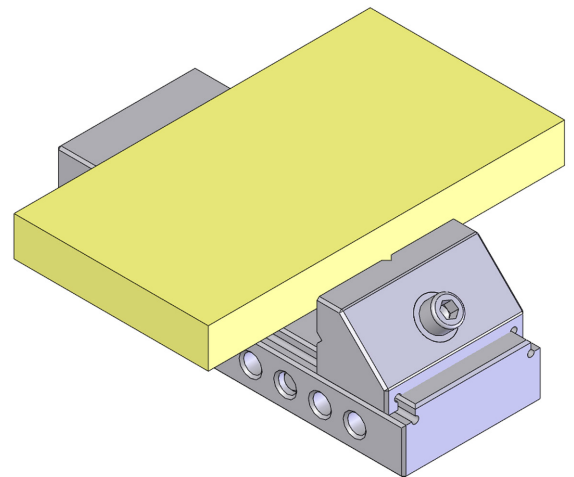


When you start to program a CAM-Part, you have to decide what workpiece you are going to use. This decision determines the number and the type of operations that are used to reach the final part shape.

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

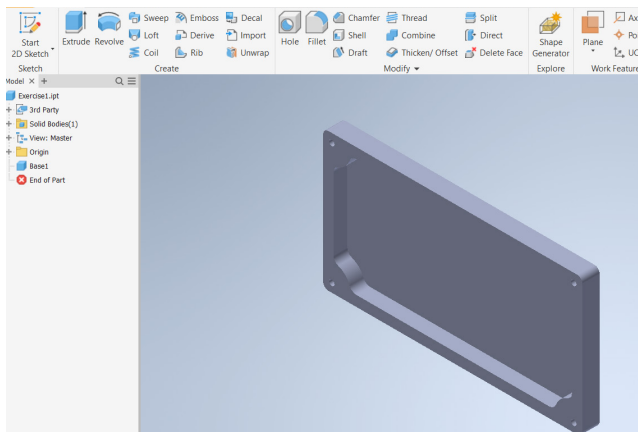


At the next stage, you have to decide on what type of CNC-machine you are going to use (3-, 4- or 5-axis). In this exercise, a 3-axis CNC-machine is chosen for the machining. With a CNC-machine of this type, all the required faces of the cover part can be machined using a single positioning.



1. Load the Autodesk Inventor model

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

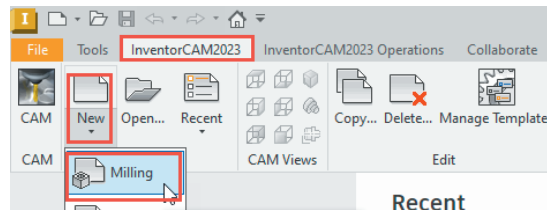


This model contains a number of features forming the solid body of the cover.

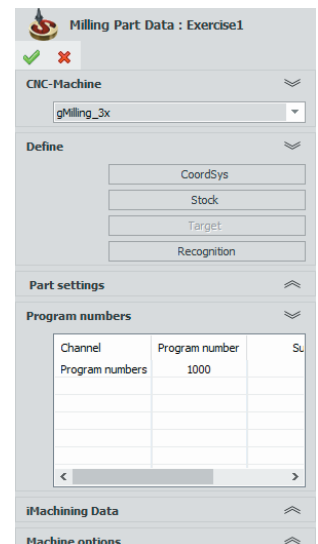


2. Start InventorCAM

To activate InventorCAM, click the **InventorCAM 2023 > New > Milling**.



InventorCAM is started, and the **Milling Part Data** dialog box is displayed.



3. Choose the CNC-Machine

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

In this exercise, use a 3-Axis CNC-machine. Choose the **gMilling_3x** from the **CNC-Machine** list.



4. Start the Coordinate System definition

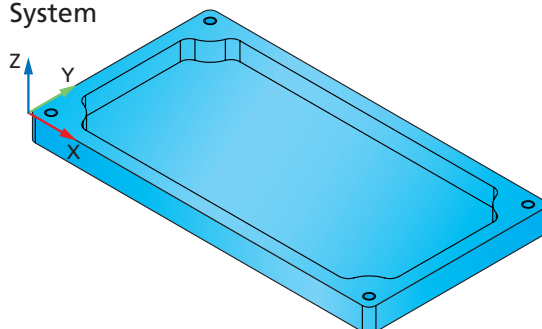
Click the **CoordSys** button in the **Define** area to define the Machine Coordinate System.



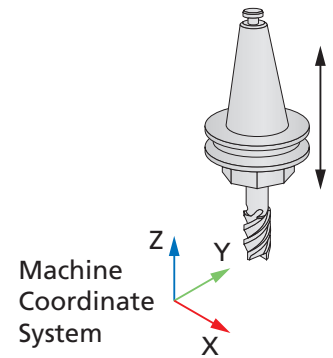
To complete the CAM-Part definition, you need to define the **Machine Coordinate System**. The **Machine Coordinate System** defines the origin for all machining operations of the CAM-Part. It corresponds with the built-in controller functions.

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

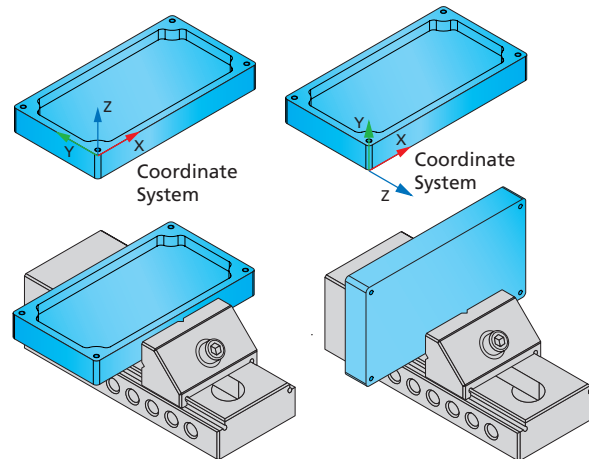
Coordinate System



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



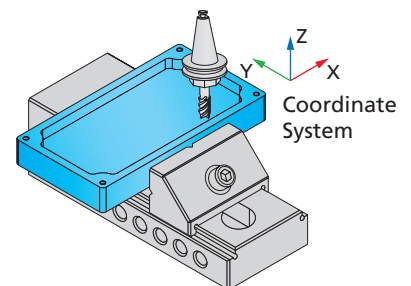
In InventorCAM, the tool approaches from the positive direction of the Z-axis (like on a vertical CNC-machine).



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.



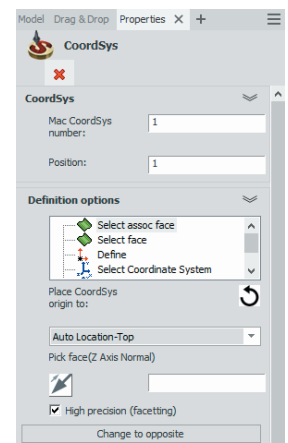
InventorCAM enables you to define the CoordSys using the following methods:

Select assoc face

This method enables you to define a coordinate system by selecting a face that remains associative to the model in case the model is later updated.

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, InventorCAM 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 Autodesk Inventor 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 Autodesk Inventor Coordinate System.

InventorCAM keeps the CAM coordinate system associative with the chosen CAD coordinate system. This means that any change made to the CAD coordinate system automatically updates the CAM 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 Autodesk Inventor Coordinate System, and the Z-axis will be directed normally to the chosen view of the model.

By 3 points (associative)

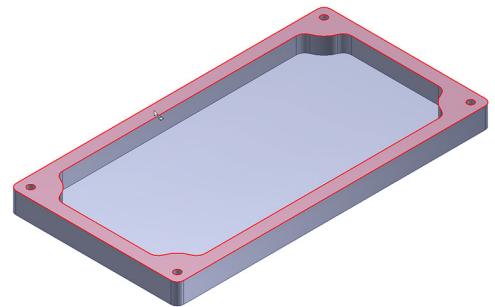
This option enables you to define the Coordinate System by selecting any three points.

**5. Select the model face**

Choose the **Top corner of model box** option from the **Place Coordsys origin to** list.

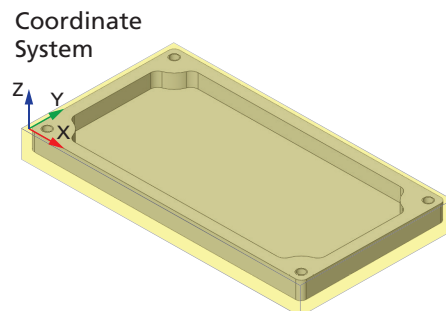
With the **Select face** method chosen, click on the model face as shown.

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

**Model box**

InventorCAM 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.



Confirm by clicking . The Coordinate System is defined.

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



6. 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 (MAC Number)** defines the number of the Coordinate System 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.

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.

The **Edit Coordsys** button enables you to change the CoordSys location or orientation of the axes with the **CoordSys** dialog box.

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 **Model** elements tree.

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

The **Levels: Planar** section displays levels for milling from the positive Z-direction.

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

The **Levels: Rear** section 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.

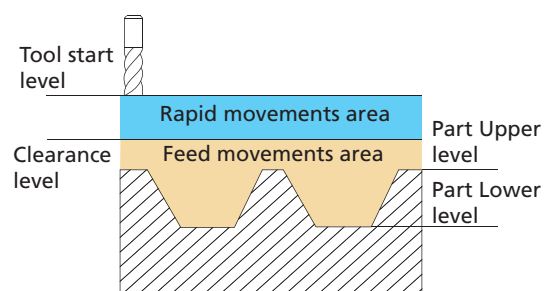
These sections contain the following parameters:

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.

In the delta fields of all the Planar Levels the associative values can be set for the face you select.

In the **Translation Data** section, **Shift** is the distance along each axis (X, Y, Z) 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 **Plane** box defines the default work plane for the operations using this CoordSys, as it is output to the GCode program. In the InventorCAM 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.

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

The **CoordSys Manager** dialog box is displayed. This dialog box displays the Machine CoordSys.

Confirm the **CoordSys Manager** dialog box with .

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

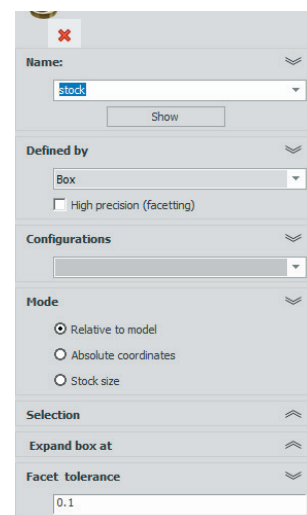


7. Define the Stock model

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 **Define** section of the **Milling Part Data** dialog box. The **Stock** 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 InventorCAM 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.

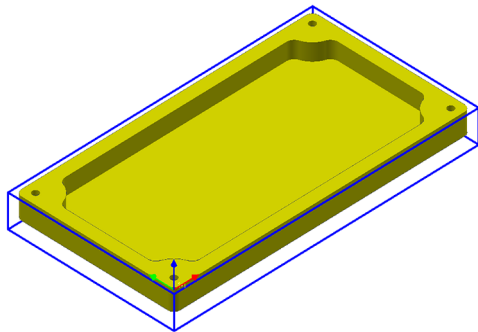
STL – this mode enables you to define the stock model based on a STL file that exists in your system. When you choose this mode, the **STL file** section becomes available. By clicking the **Browse** button, you can choose the STL file for the stock definition.

Choose the **Box** mode from the **Defined by** list. The appearing parameters enable 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 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. It is highlighted.

A screenshot of the 'Expand box at' dialog box. It has a title bar with a dropdown arrow. Below the title bar, there are six input fields for offsets: X+, X-, Y+, Y-, Z+, and Z-. The values are: X+ (2), X- (2), Y+ (2), Y- (2), Z+ (0.25), and Z- (5). Below these fields are two buttons: 'Add box to CAD model' and 'Create 3D model'. At the bottom of the dialog, there is a 'Facet tolerance' section with a dropdown arrow and an input field containing the value '0.1'.

InventorCAM automatically generates the surrounding box.

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




8. Define the Target model

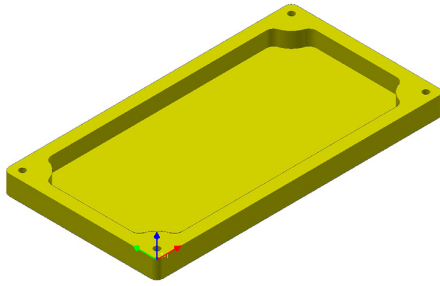
InventorCAM enables you to define the Target model, which is the final shape of the CAM-Part after the machining. The program uses the Target model for gouge checking in the **SolidVerify** simulation.

Click the **Target** button in the **Define** area of the **Milling Part Data** dialog box.

The **Target** dialog box is displayed.

A screenshot of the 'Target' dialog box. It has a title bar with a red 'X' icon and a green checkmark icon. Below the title bar, there is a 'Name:' section with a dropdown menu showing 'target' and a 'Show' button. Below that is a 'Configurations' section with a dropdown menu. Below that is a 'Selection' section with a list box containing 'Solid 1' and a 'CAD selection' button. At the bottom, there is a 'Facet tolerance' section with a dropdown arrow and an input field containing the value '0.01'.

Click on the solid body to choose the target. The solid body is highlighted, and the **Solid 1**  icon appears in the **Selection** section. The target model is defined.



Confirm the **Target** dialog box with .

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



9. Save the CAM-Part data

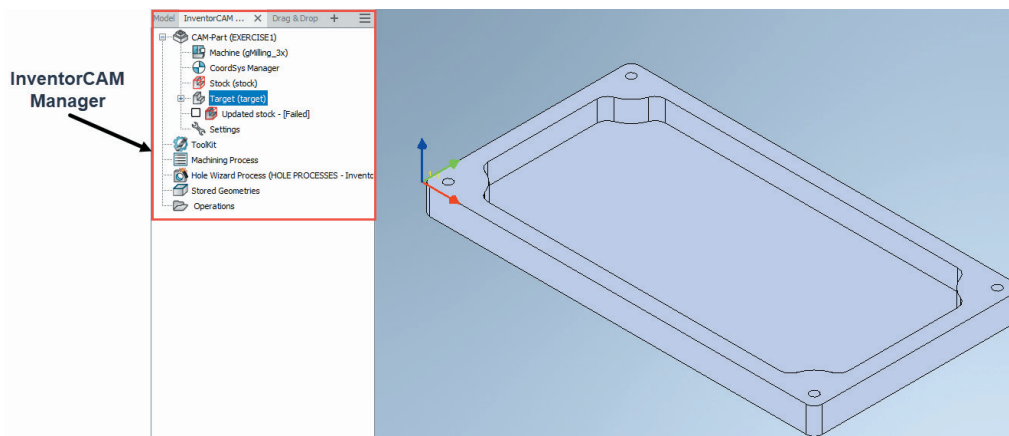
Confirm the **Milling Part Data** dialog box by clicking .

The **Milling Part Data** dialog box is closed, and the **InventorCAM 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 **InventorCAM Manager** tree is the main interface feature of InventorCAM that displays complete information about the CAM-Part.



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

- **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/Target** dialog boxes that enable you to change the definition of the Stock/Target models.

The **Updated stock** subheader is located under the **Target** header. Right-click this subheader to display the menu that enables you to calculate the updated stock and rest material.

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.

- **Hole Wizard Process header**

This header can recognise and process the holes created with Autodesk Hole Wizard feature and perform machining of the holes automatically.

- **Geometries header**

This header displays all InventorCAM 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 InventorCAM operations and setups defined for the current CAM-Part.

- **Setup header**

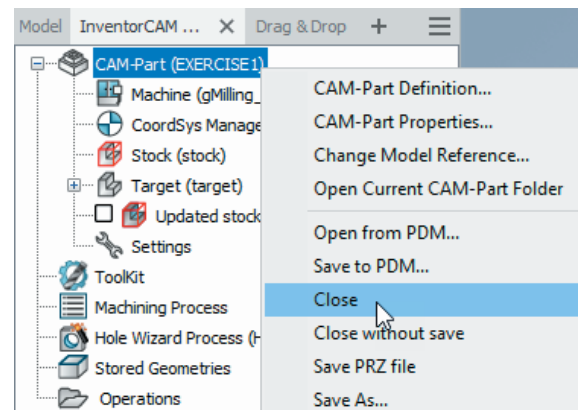
This header displays all InventorCAM operations defined for the current setup.



10. Close the CAM-Part

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

The CAM-Part is closed.



InventorCAM 2.5D Operations

3

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



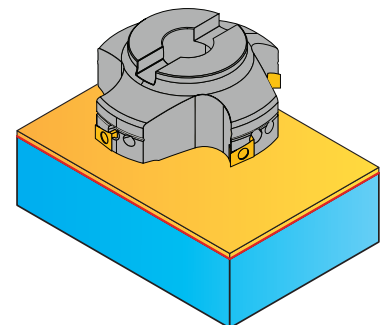
In InventorCAM, an operation is a single machining step. A workpiece is usually manufactured using several machining steps and technologies. For each of these steps you can define a separate operation. An operation can be very complex, but it always uses one tool, one major geometry and executes one 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 InventorCAM what and where you want to machine.

- A Geometry for Profile, Pocket, Contour 3D, Slot and T-Slot operations, Engraving 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, Pocket Recognition, and Edge Deburring Recognition operations is determined automatically by InventorCAM Automatic Feature Recognition functionality.

Face Milling Operation

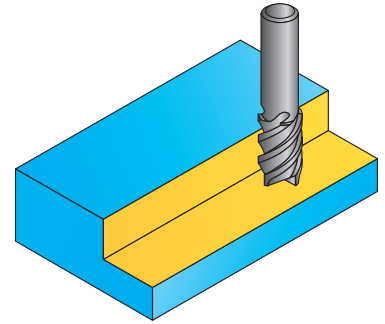
This operation enables you to machine large flat surfaces with face mill tools.



Profile Operation

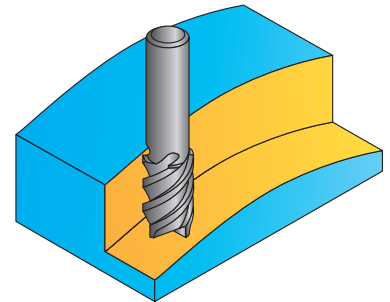
You can mill on or along a contour. The profile geometry can be open or closed. In profile milling you can optionally use tool radius compensation to the right or to the left side of the geometry. InventorCAM offers two types of profiling:

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



Contour 3D Operation

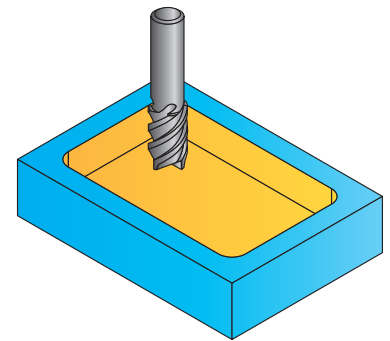
This operation enables you to utilize the power of the 3D Engraving technology for the 3D contour machining. In this operation, InventorCAM enables you to prevent the gouging between the tool and the 3D contour.



Pocket Operation

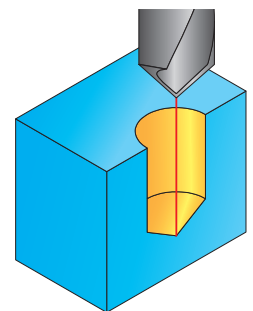
In pocket milling, you have to remove material from the interior of a closed geometry. InventorCAM offers two types of pocketing:

- 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.



Drilling Operation

This operation enables you to perform drills and other canned drill cycles. InventorCAM 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 VMID Editor. In the Technology list, you can choose the option of Thread milling to save and copy a drilling operation and then change the operation to a thread mill operation.



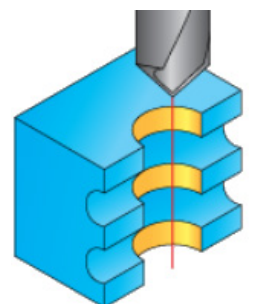
In the Technology list, select either a 2D Drill, 3D Drill, Thread Milling or Multi-depth Drilling option. The dialog box parameters are changed according to your selection.

When a Drilling Operation is opened and Thread Milling is selected under Technology, all the Drilling Operations are saved and copied into the new Thread Milling operation. However, you must re-select the tool for this new Thread Milling operation.

Multi-depth Drilling

Multi-depth drilling is very useful for deep hole drilling and drilling through cross holes. In this operation, you have full control of Drilling operations at every depth. The Multi-depth drilling operation enables you to define manually the areas where the feed rate changes. You can manually control the Spin rate/direction and coolant control during the retract and engage movements.

In the Technology list, select either a 2D Drill, 3D Drill, Thread Milling or Multi-depth Drilling option. The dialog box parameters are changed according to your selection.

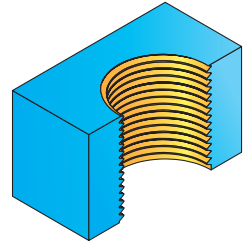


When a Drilling Operation is opened and Thread Milling is selected under Technology, all the Drilling Operations are saved and copied into the new Thread Milling operation. However, you must re-select the tool for this new Thread Milling operation.

Thread Milling Operation

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

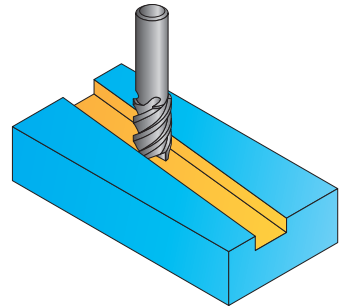
In the Technology list, select either a 2D Drill, 3D Drill, Thread Milling or Multi-depth Drilling option. The dialog box parameters are changed according to your selection.



When a Thread Milling Operation is opened and one of the Drilling operations is selected under Technology, all the Thread Milling Operations are saved and copied into the new Drilling operation. However, you must re-select the tool for this new Drilling operation.

Slot Operation

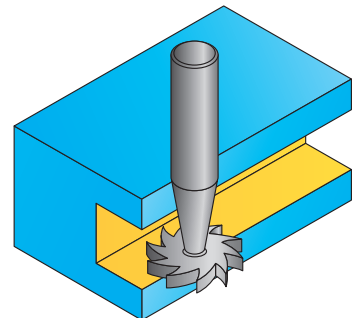
This operation generates a tool path along the centerline to the right or to the left of one or more profiles. Two types of slots can be defined: the Slot with constant depth operation machines the slot in several steps until the final depth is reached. In Slot with variable depth, the depth profile 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. All slot technologies in Toolbox Cycles operations are enabled to use Pre-Drill operations.

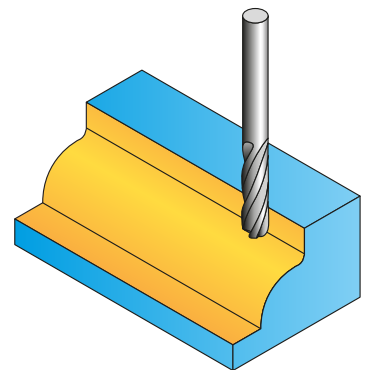
T-Slot Operation

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



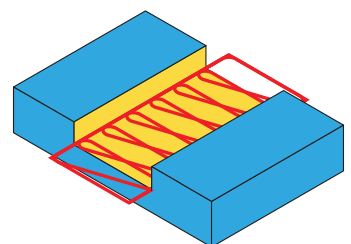
Translated Surface

This operation enables you to generate translated surface by moving a section along a profile geometry. Limit geometries can be projected on the translated surface. You can machine the resulting translated surface inside, outside or along the limit geometry.



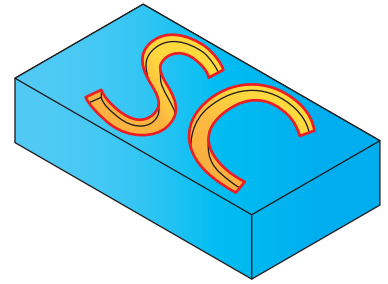
ToolBox Cycles

This operation enables you to choose a machining scenario intended for a specific machining case from a set of pre-defined 2.5D Milling operations.



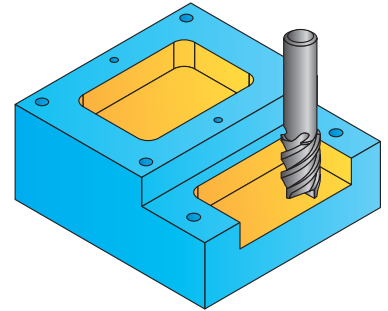
Engraving

This operation enables you to mill text or any other profile on a 2D or 3D geometry. Both 2D and 3D Engraving strategies are available.



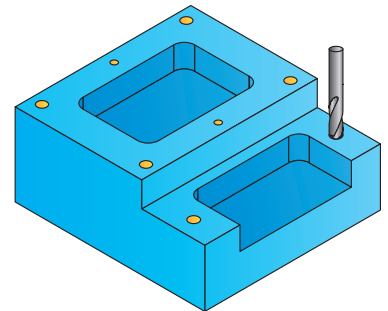
Pocket Recognition

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



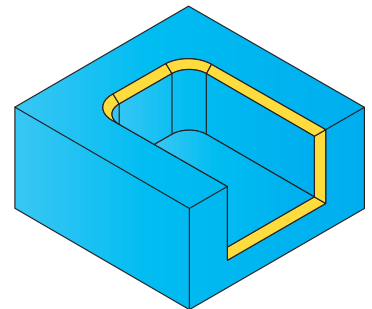
Drill Recognition

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.



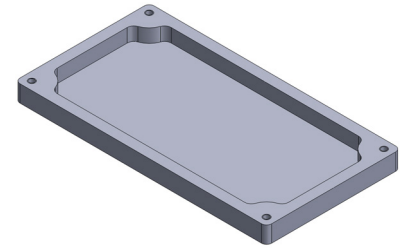
Edge Deburring Recognition

This operation recognizes automatically chamfer features at the target model and creates the necessary machining.



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.

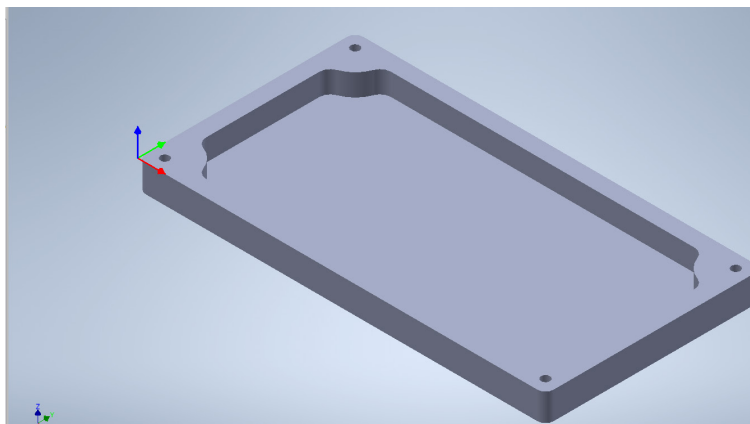


1. Open the CAM-Part

In the main menu, choose **InventorCAM 2023** and click **Open**.

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

The CAM-Part is loaded.

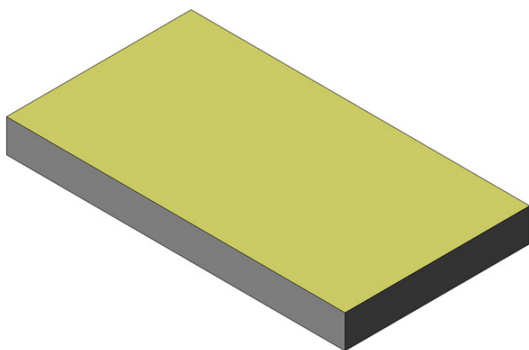


2. Add a Face Milling operation

In **InventorCAM 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

Face Milling Geometry can be defined in two ways:

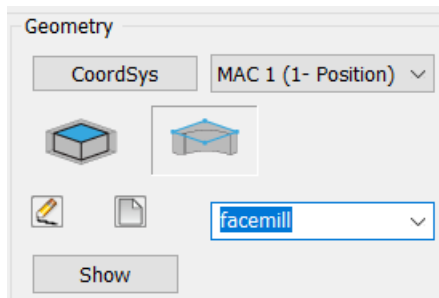
- Face Milling geometry can be based on the actual Updated Stock boundary thereby assuring full coverage of the stock face at any stage of machining.
- The traditional way of defining Geometry.

You can set the Geometry to Stock based boundary or Part based boundary as default in the InventorCAM settings.

Stock Based boundary – The Geometry is based on the existing updated stock. The Geometry field is disabled and named as Updated Stock by default.

Part based boundary – This section enables you to choose the appropriate Coordinate System and to define the Machining Geometry for the operation.

Click Part based boundary and then click .



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



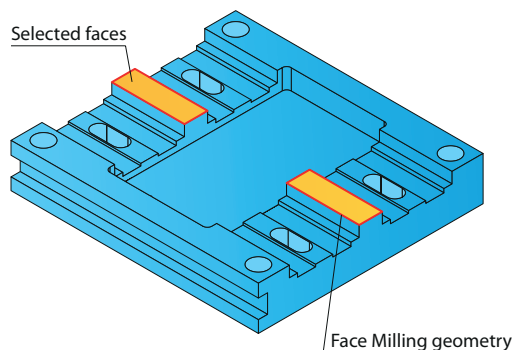
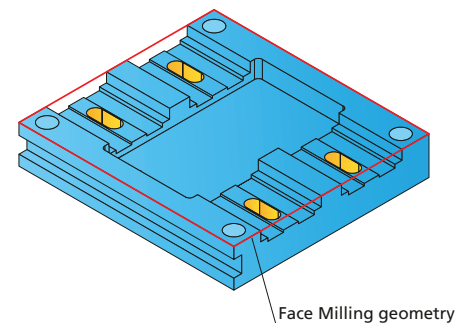
The **Type** section enables you to define the face milling geometry using the following methods:

- **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.

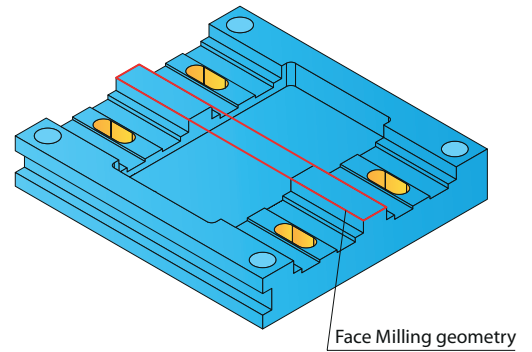
- **Faces**

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, InventorCAM 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 in **Base Geometry** section. The **3D Geometry** dialog box is displayed.




This dialog box enables you to define the 3D Model geometry:

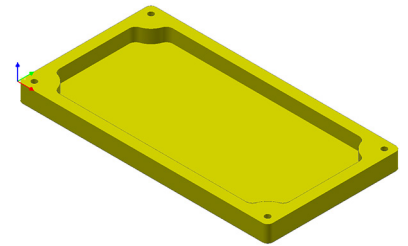
The **CAD selection** button enables you to select the 3D geometry with Autodesk Inventor tools.

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.

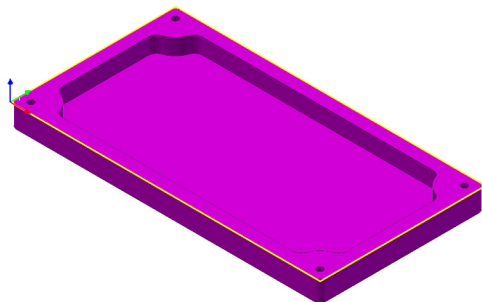
Confirm the **3D Geometry** dialog box by clicking . The **Face Milling Geometry** dialog box is displayed again.



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

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

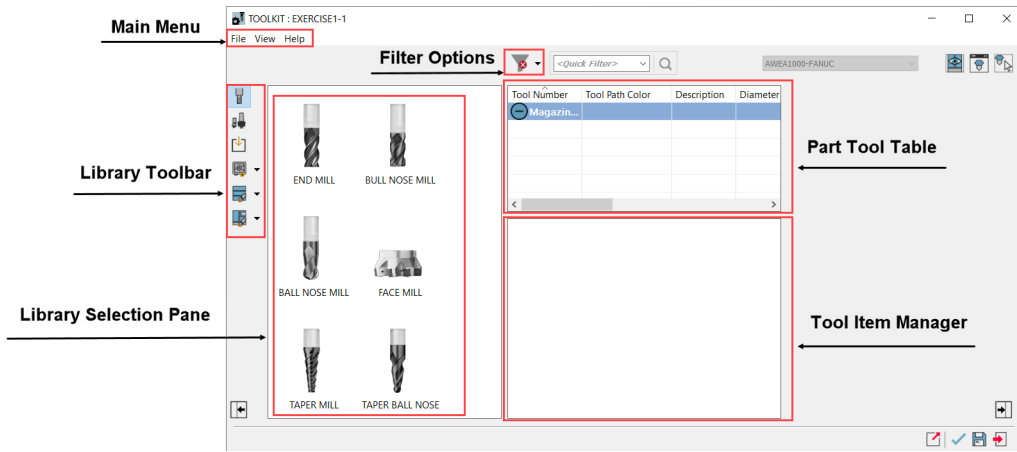
The geometry is defined for the operation.



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 **TOOLKIT** dialog box is displayed.



The Library Toolbar contains all tools available for use to machine a specific CAM-Part. This dialog box enables you to add, import, and manage the tool items to the Part Tool Library of your CAM-Part.

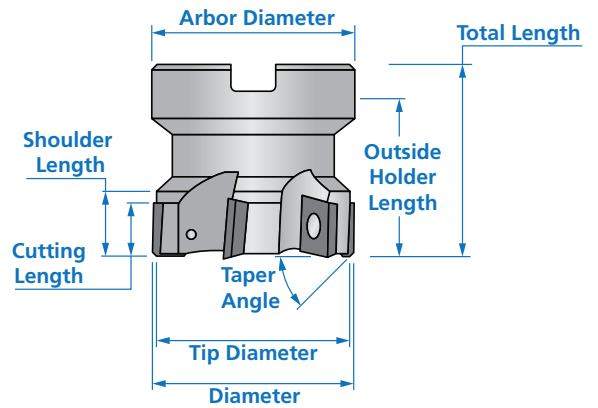
Currently, the **Part Tool Table** is empty. Define a new tool suitable for face milling. The available tools are displayed in the Library Selection Pane.

This pane 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 $\varnothing 100$ will be used.



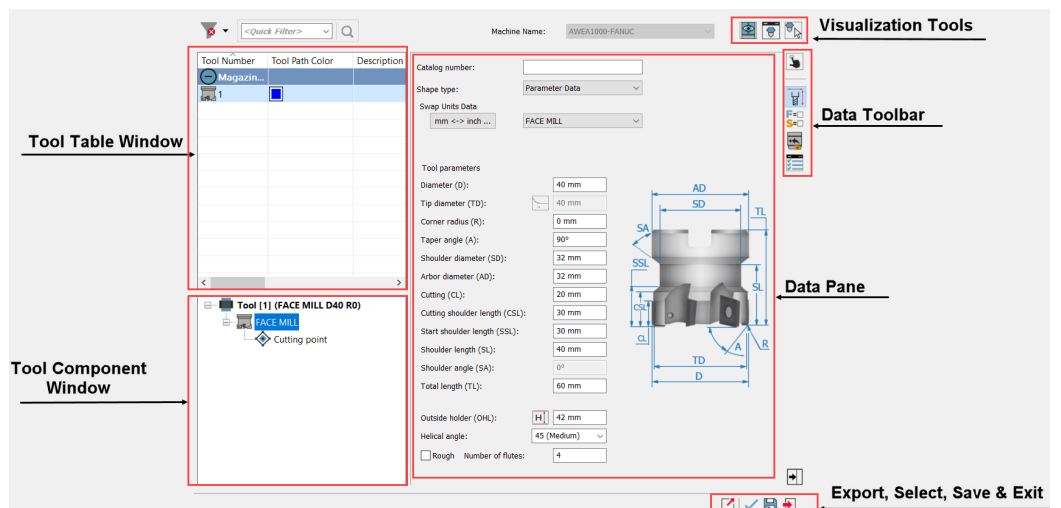
Face mill


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.



Double click on the **Face mill** tool icon. A new Face mill is added in the Data Pane.

When a new tool is added, the Data Pane is populated with the default parameters of the tool. The below image explains the right hand side of the **TOOLKIT** dialog box.



From the Data Toolbar, select the **Topology** data  icon.

In the **Tool parameters** section, edit the following parameters:

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

Ensure the other parameters are also the same as shown in the below image.

Click  to confirm the tool parameters and choose the tool for the operation.

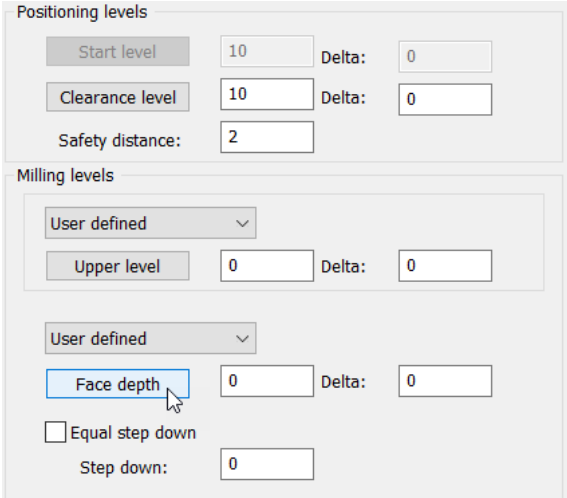
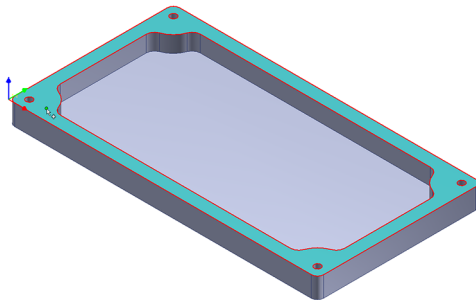


5. Define the Face depth

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.

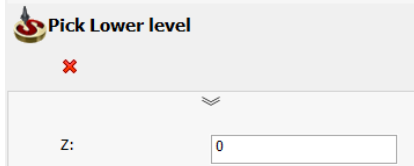


Positioning levels		
Start level	10	Delta: 0
Clearance level	10	Delta: 0
Safety distance:	2	

Milling levels		
User defined	▼	
Upper level	0	Delta: 0
User defined	▼	
Face depth	0	Delta: 0
<input type="checkbox"/> Equal step down		
Step down:	0	


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

Confirm this dialog box with 



Pick Lower level	
✗	
▼	
Z:	0

The **Face depth** value is displayed in the **Milling levels** area. The green 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; InventorCAM automatically updates the CAM data when the model is modified.

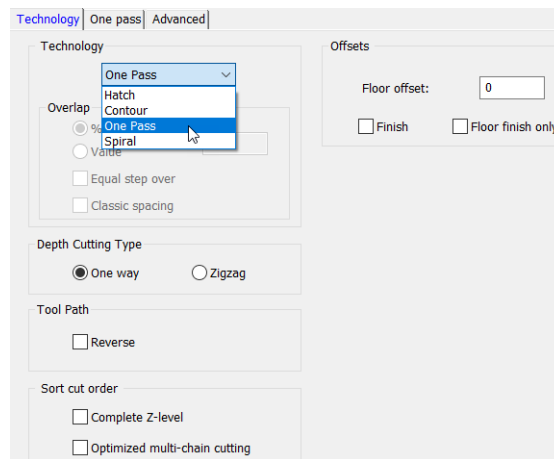


Face depth	0	Delta: 0
<input type="checkbox"/> Equal step down		
Step down:	0	



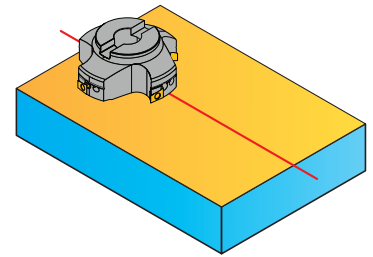
6. Define the technological parameters

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

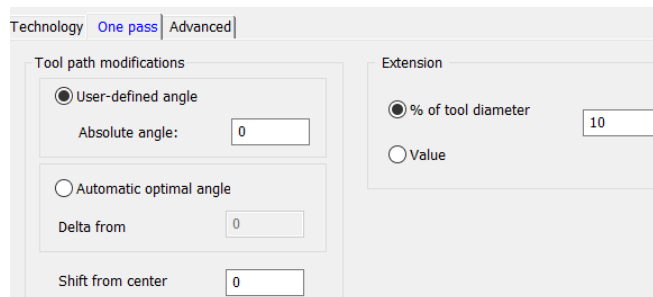


One Pass machining technology

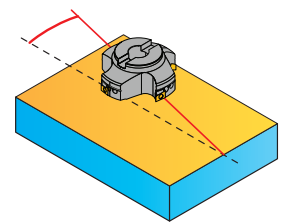
InventorCAM performs the face milling in one pass. The direction and location of the pass are calculated automatically according to the face geometry, in order to generate an optimal tool movement with the tool covering the whole geometry.



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



The **Tool path modifications** section enables you to define the angle between the tool path and the geometry, and the shift from the central line. The **User-defined angle** option enables you to determine the angle of the tool path. Enter the value of angle in the **Absolute angle** edit box.

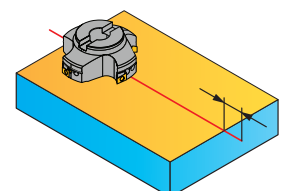


When the option of **Automatic optimal angle** is selected, InventorCAM automatically calculates the optimal angle of hatching to expedite the machining process. The tool path always follows the length of the face no matter what angle the machined surface is facing.

Using the **Automatic optimal angle** option, you can change the angle by entering a different value in the **Delta from** text field.

The **Shift from center** option shifts the generated tool path from the center of part by the specified distance.

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).



Use the **One pass** option with default parameters.

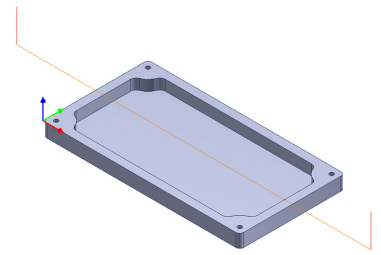


7. Save and Calculate

Click .

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

The preview of the calculated tool path is displayed on the solid model.






8. Simulate


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

Switch to the **SolidVerify** page and clear the **Single color** check box. This option enables you to display all tool paths in one color, regardless of the chosen tool color. By clearing it, you can display tool paths in colors according to the tool color. Start the simulation with **Play** icon.



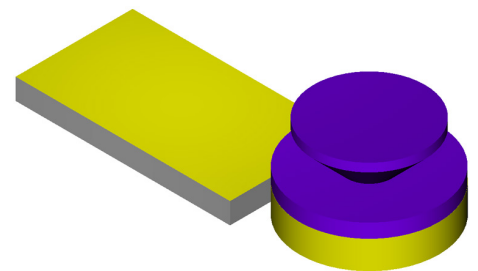
The solid stock model defined in **Exercise #1** is used in the **SolidVerify** simulation mode. During the machining simulation process, InventorCAM subtracts the tool movements (using solid Boolean operations) from the solid model of the stock. The remaining machined stock is a solid model that can 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  or by using the space bar on your keyboard.

Close the simulation. The **Face Milling Operation** dialog box is displayed.

Close this dialog box with .



Click the check box next to the defined Face Milling Operation to hide the preview of the tool path.



9. Add an operation

Right-click the **Operations** header in **InventorCAM 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. 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.



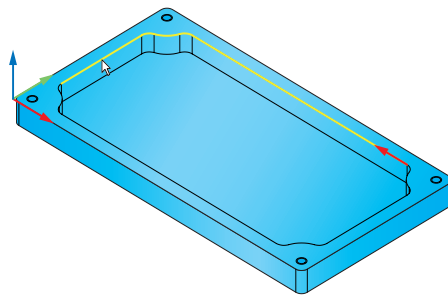
Chain Selection Options

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 continuous geometry chain by choosing its successive entities automatically or manually

If none of the additional options (**Up to Entity**, **Tangent propagation**, **Constant Z Propagation**) is selected, you can choose the chain entities manually one by one.



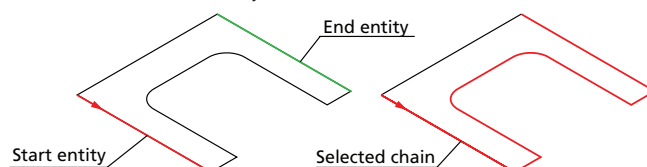
Associativity: InventorCAM keeps the associativity to any edge or sketch entity. Any change made to the model or sketch automatically updates the selected geometry.

Smart Face

When you select the **Smart Face** check box, it allows you to pick a geometry directly from the face that automatically recognizes all open edges, holes, bosses or semi-open pockets or a completely open pocket. The option of Smart face is not checked by default. When any of the option of Filter for circle/arc selection is used, Smart face remains unchecked and is disabled. When Smart face is checked, you can pick the face and sync works on that entire face. However, if a filter is used selecting Smart face, Smart face option automatically unchecks and disables. If Smart face is used and you want to delete a hole that was picked using Smart face, a message pops up saying “Uncheck Smart face to delete holes”. The moment you delete holes, Smart face is again disabled. You can enable Smart face again if all holes are removed.

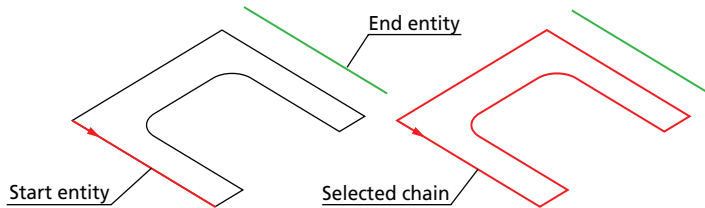
Up to Entity

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

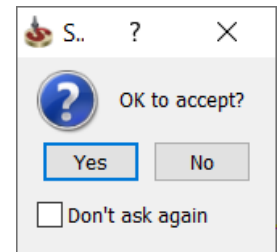


When the end item is chosen, InventorCAM determines the chain according to the rules of the chosen selection mode (**Tangent propagation** and/or **Constant Z Propagation**). The chain selection is terminated when the selected end item is reached.

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

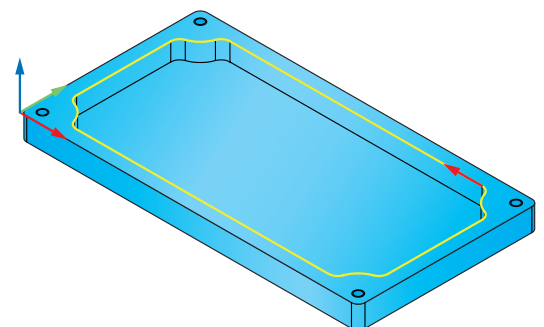
Tangent propagation

InventorCAM automatically selects the next entity tangent to the current one. If there is no any, it highlights all entities connected to the last chain entity. You have to select the entity along which you want the chain to continue.

Constant Z Propagation

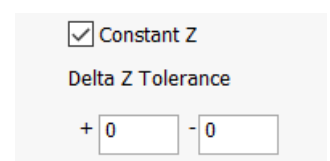
This option identifies only entities on the same XY-plane with the previously selected chain entity. You are only prompted to identify the next chain element when two entities on the same Z-level are connected to the chain.

The system tolerance for this option can be set in the InventorCAM Settings.



You are required to enter a positive and negative Z-deviation into the **Delta Z Tolerance** edit boxes. Only entities within this range are identified as the next possible entity of the chain.

1. Pick start curve - select the start entity of the chain.
2. Pick point in chain direction - indicate the chain direction by choosing a point on the selected start entity of the chain.
3. You are prompted to select one of the possible curves if the system detects that more than one entity can be the next element of the chain. Indicate your choice.



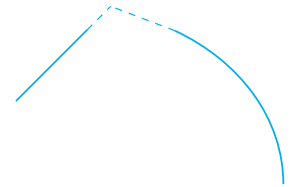
4. When the system can close the chain, the confirmation message is displayed.

Click **Yes** to accept. If you answer **No**, you can continue the chain selection: for example, undo the last step or cancel the chain selection.

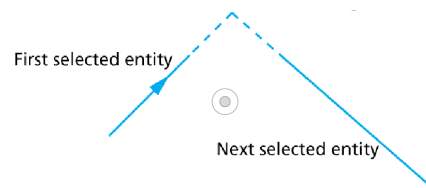
The combination of **Constant Z** and **Tangent Propagation** options enables you to choose entities which continue the chain with either tangent or having the same Z-level entities.

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 **Defaults > Geometry** section of **InventorCAM Settings**) by virtually extending the entities up to their intersection.



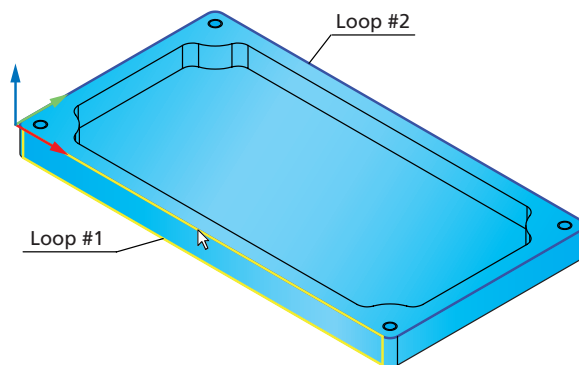
Splines and arcs are extended by lines tangential to the arc/spline at its end point.



Associativity: When the model used for the geometry definition is modified, InventorCAM enables you to synchronize the geometry with the updated model. During the synchronization, the program 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.

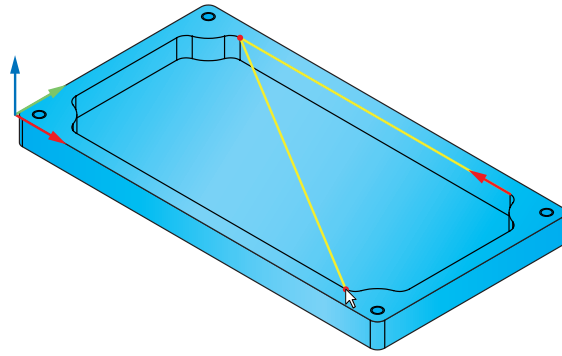


1. Pick an edge shared by two model faces. Two faces to which this edge belongs are determined, and their loops are highlighted. The first determined loop is considered to be the primary and is highlighted with yellow color. The second loop is considered to be the secondary and is highlighted with blue 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: InventorCAM 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 with a straight line.



Associativity: InventorCAM does not keep the associativity to any selected point. The program 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.



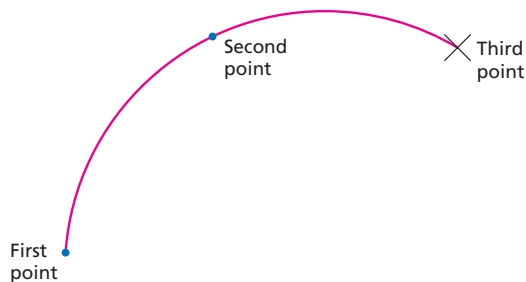
You cannot select a point that is not located on a Autodesk Inventor entity (if you need to select such a point, add a planar surface under the model and select the points on that surface).

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

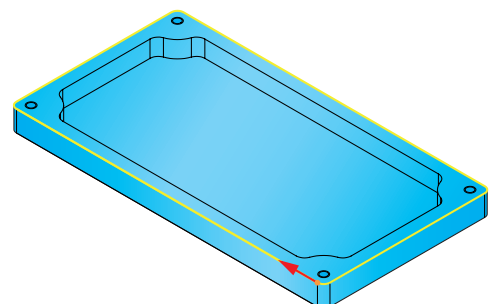
This option enables you to create a chain segment on an arc up to a specific point on the arc.



Associativity: InventorCAM does not keep the associativity to any selected arcs by points. InventorCAM 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.

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

The red arrow indicates the direction of the geometry.

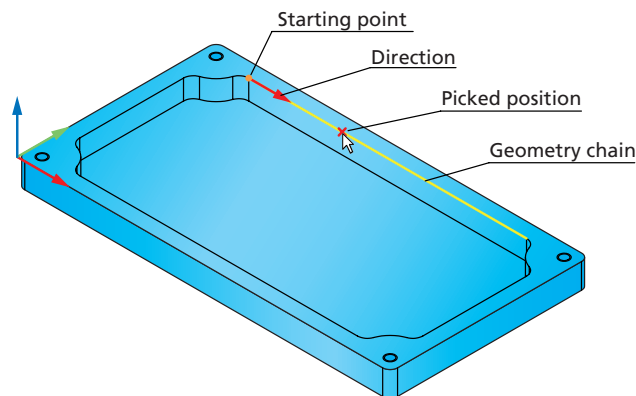
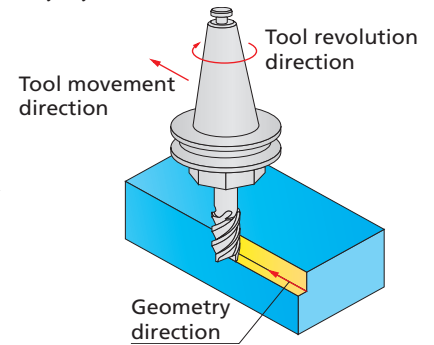




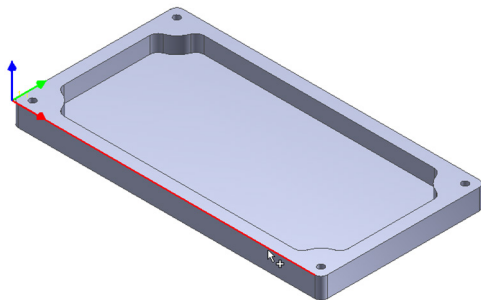
In InventorCAM operations, the direction of the chain geometry is used for the tool path calculation.

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 entity on the solid model, InventorCAM determines the start point of the picked entity closest to the picked position. The direction of the picked first chain entity is defined automatically from the start point to the picked position.

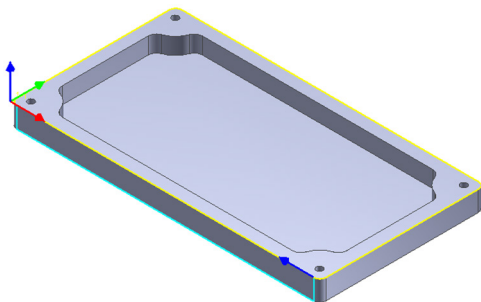


Choose the **Loop** option in the **Single entities** section and click on the model edge as shown.




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 arrow indicates the direction of the selected chain.



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

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.



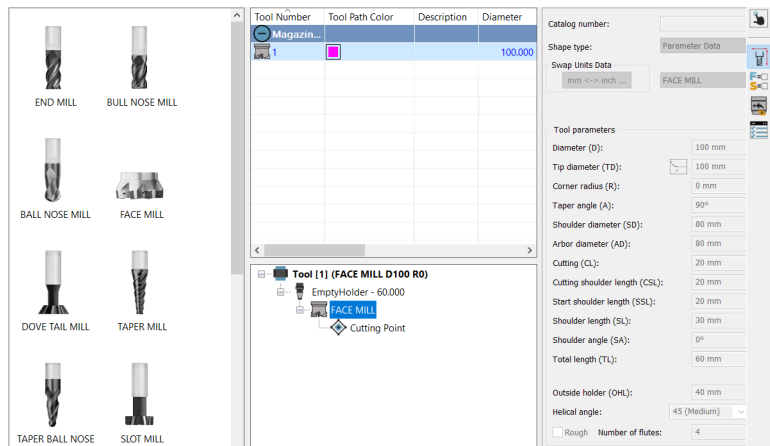
11. Define the Tool

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.

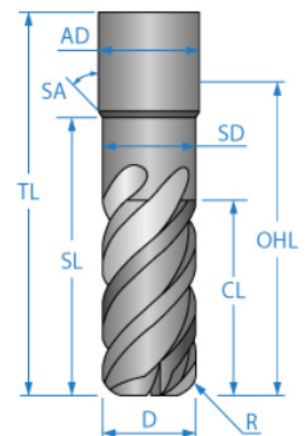
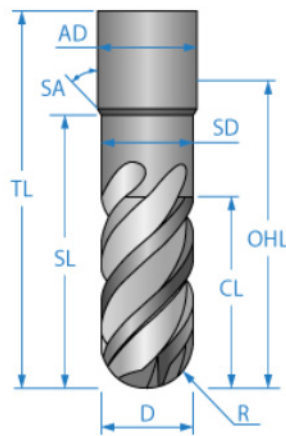
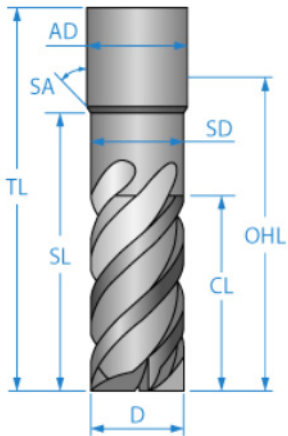
The **Part Tool Table** with the tool used in the previous operation is displayed.

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



End Mill/Bull Nose Mill/Ball Nose Mill

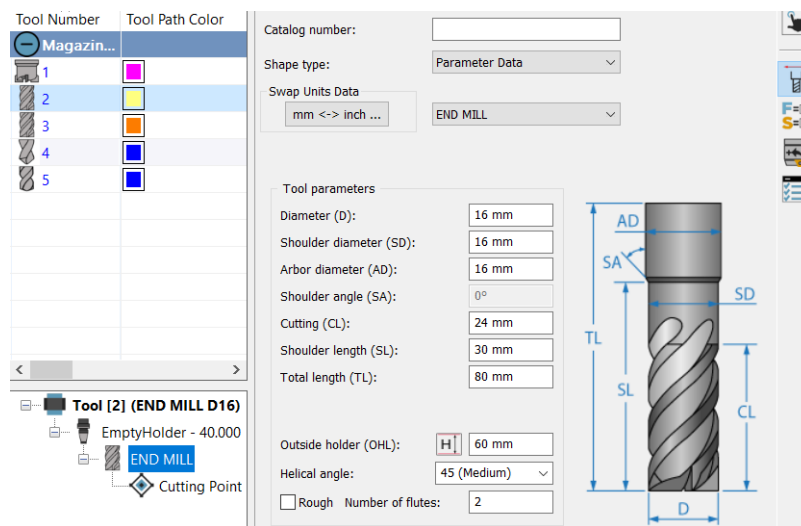
These tool types are used for the definition of rough/finish mills. The tool shapes and basic parameters are shown below:



Double click the **End Mill** tool to choose it for the operation.

From the Data Toolbar, select the **Topology** data  icon. In the **Tool parameters** section, set the **Diameter** value to **16**.

Set all the other parameters as shown in the image.



Click  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.

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

InventorCAM supports the machining of geometries with different depths in a single operation. This option is available for Profile, Contour 3D, Pocket, Slot and T-Slot operations.

Choosing the Constant icon in the Milling levels section, enables you to set the Upper level, Lower level and Step down values.

Choosing the Variable icon displays the Variable levels button. Clicking Variable levels displays the Levels dialog box.

The Variable levels option can help simplify the task of programming, especially when you want to use the same tool to perform the machining of many geometries having different depths. Such a task can now be achieved in just one operation.

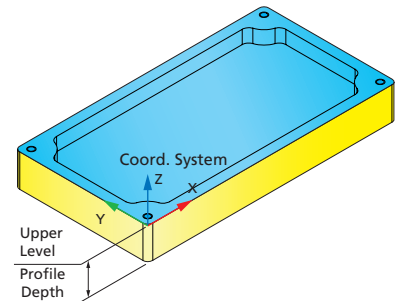


Upper level

This parameter defines the Z-level at which the machining starts. To define the upper level, you can select the options of By stock, By target or User defined from the list.

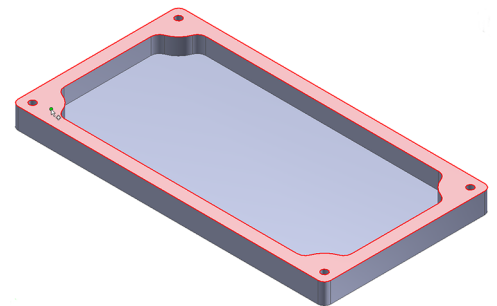
Profile depth


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.

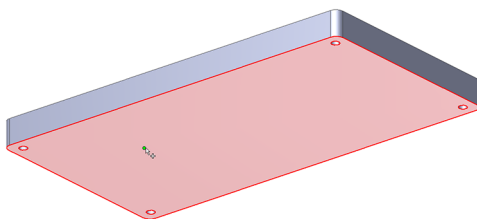
Select the model face as shown.



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** 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.

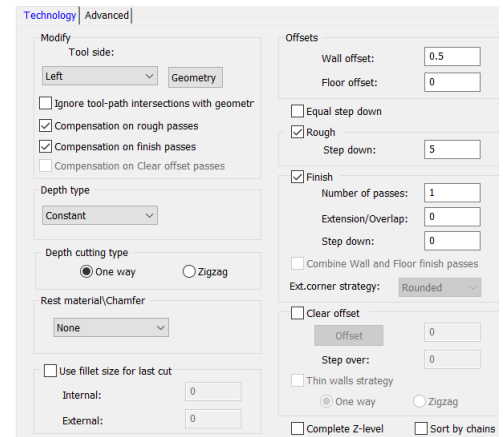
The milling levels are defined.



14. Define the technological parameters

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

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.



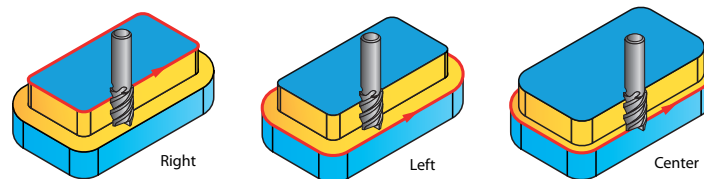
Modify

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.

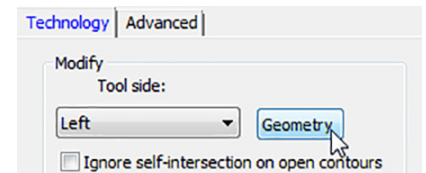
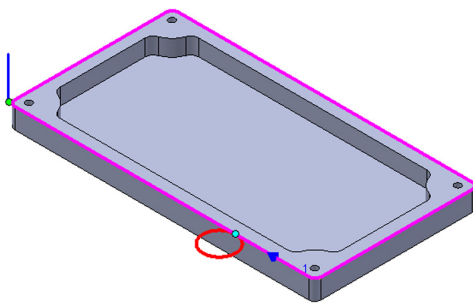
Left – the tool cuts on the left side of the profile geometry.

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  to close the **Modify Geometry** dialog box. The **Profile Operation** dialog box is displayed again.

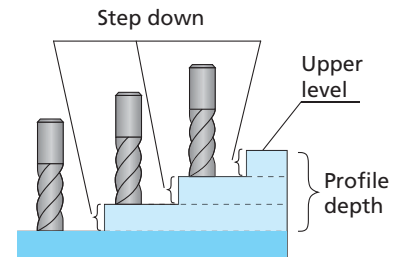
InventorCAM enables you to perform the rough and finish machining of the profile in a single Profile operation.

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.



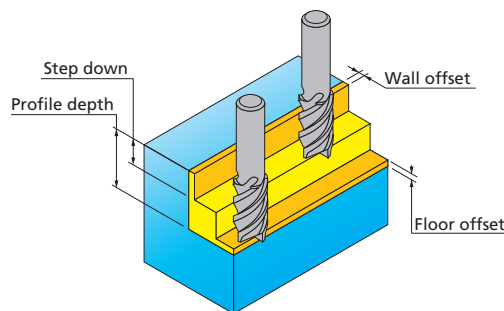
Set the **Step down** to **5**. With this value, InventorCAM performs two cuts at the following Z-levels: **-5, -10**; the last cut is performed at the Z-level defined by **Profile depth**.

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.

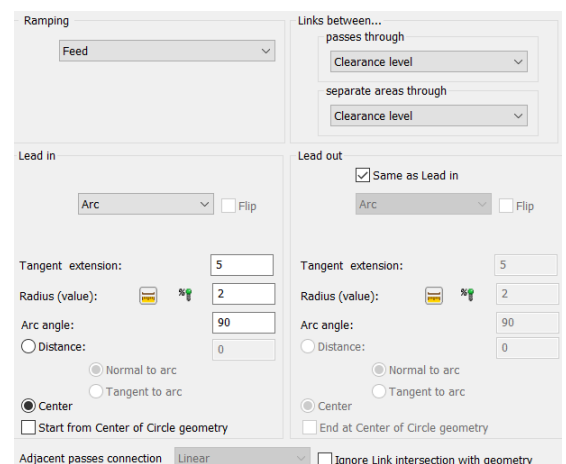
Select the **Finish** check box to perform the finishing of the profile.



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.





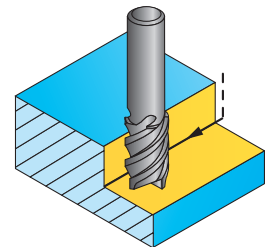
Profile Lead in and Lead out

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:

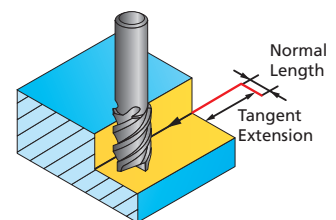
- **None**

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



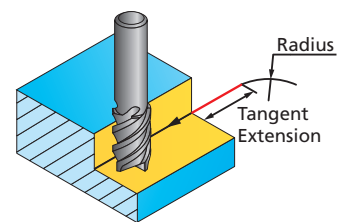
- **Normal**

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.



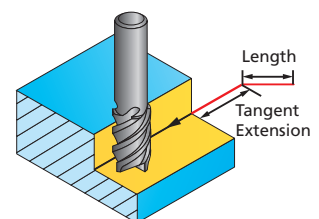
- **Arc**

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. The Flip check box is available for selection when Tool side is chosen as Center on the Technology page. When this check box is selected, the lead move is flipped to the opposite side from its initial orientation, as if it was mirrored relative to the tool path. This option is available only for Normal and Arc Lead in types.



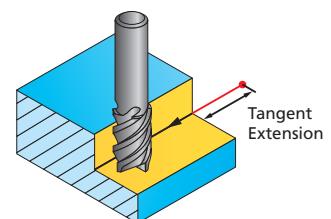
- **Tangent**

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.



The Lead In/Out with Points are fully associative. You can associate the point location to Vertex, Circular edge center, Sketch point, Sketch arc center, Work axis and Cylindrical face axis. The Points will synchronize and update the tool path if the location of the point on the solid model is changed.

- **User-defined**

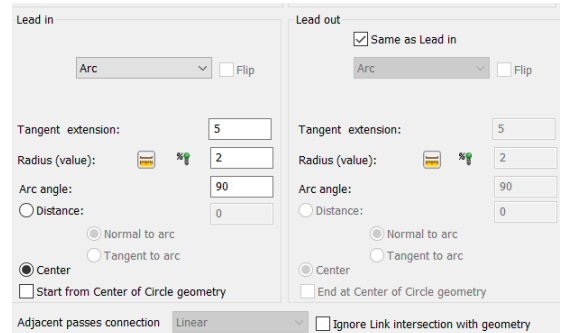
The tool leads in/out from a user-defined position. When you select this option, you can define a geometry of the tool approach to the material.

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

For **Lead in**, use the default **Arc** option. Set the **Tangent extension** value to **5** and the **Radius** value to **2**.

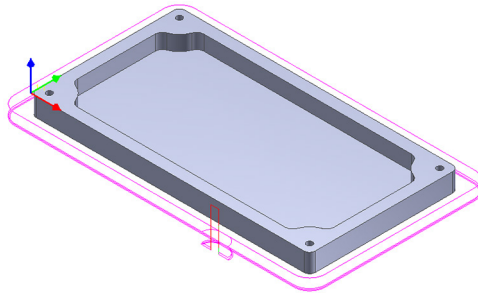
Under **Lead out**, select the **Same as Lead in** check box.

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



16. Calculate the Tool path

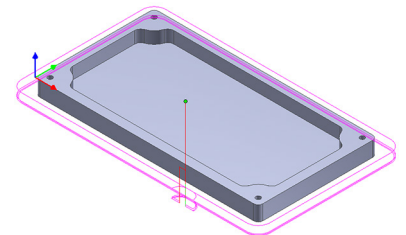
Click . The Profile operation data is saved, and the tool path is calculated. The preview of the calculated tool path is displayed on the solid model.



17. Simulate the operation

The preview of the tool path can also be activated using the **Host CAD** simulation mode in **Simulation** control panel.

Click the **Simulate** button in the **Profile Operation** dialog box. The **Simulation** control panel is displayed. Switch to the **Host CAD** simulation mode and click the **Play** icon.

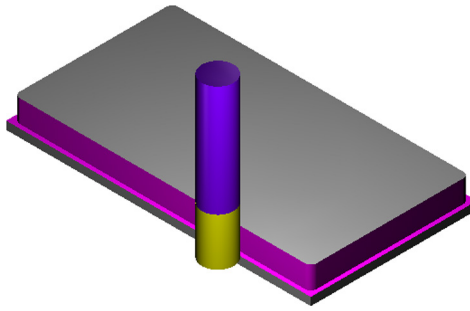


The **Host CAD** simulation mode enables you to display the tool path directly on the model in the Autodesk Inventor window. Since all the **View** options of InventorCAM are active during the simulation, you can see the tool path from different perspectives and zoom on a certain area of the model.

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

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

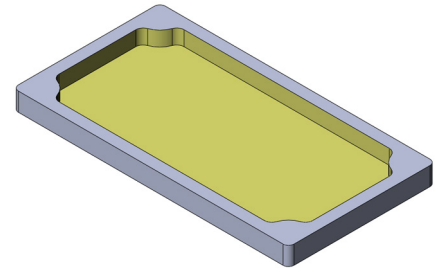
Close the simulation with the . The **Profile Operation** dialog box is displayed. Close the **Profile Operation** dialog box.



18. Add a Pocket operation

The Pocket operation is used for the internal pocket machining. Right-click the **Operations** header and choose **Pocket** from the **Add Milling Operation** submenu.

The **Pocket Operation** dialog box is displayed

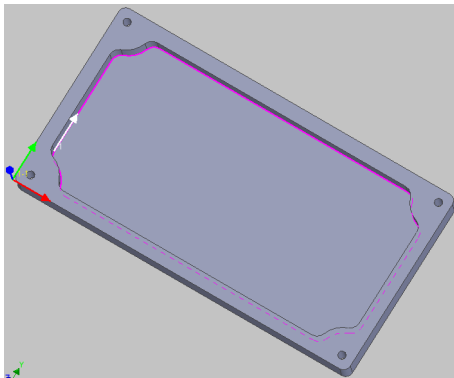


19. Define the Geometry

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

Using the **Tangent Propagation** 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.

In this Profile operation, an **End mill** of **Ø8** will be used.

Choose the **End mill** tool for the operation and set the **Diameter** value to **8**.

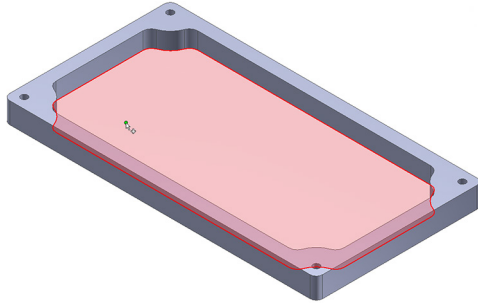
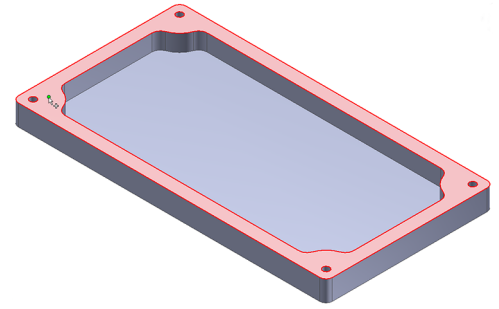
Click  to confirm the tool parameters and choose the tool for the operation.



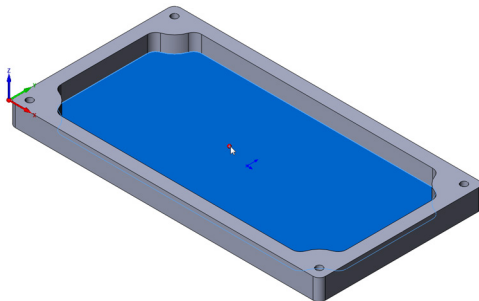
21. Define the Milling levels

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.

The default **Constant depth**  icon indicates that the pocket will be machined at a constant depth.



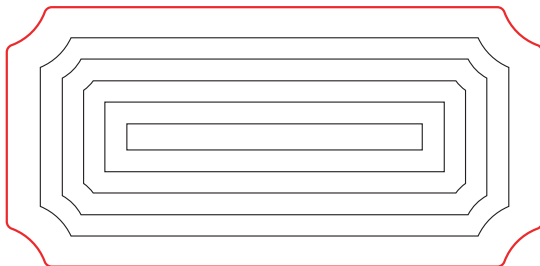
22. Define the technological parameters

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.

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.



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

Contour parameters

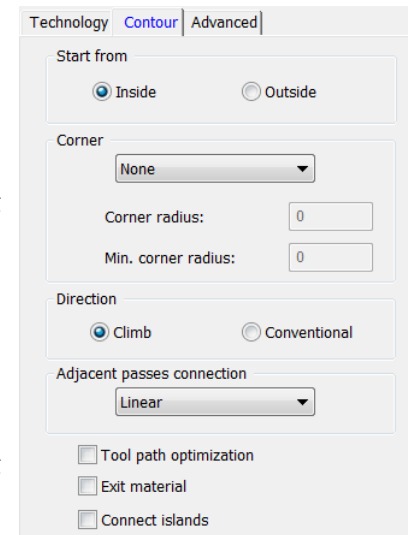
Start from

- **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**

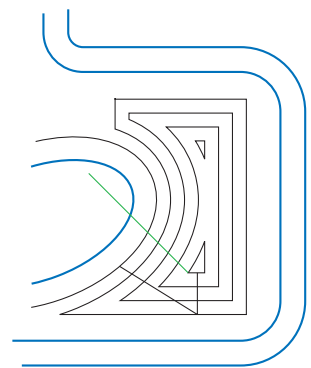
This option allows you to work in a pocket area starting from the outside border of the pocket and cutting towards the middle of the pocket.



Corner

- **None**

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



- **Fillet**

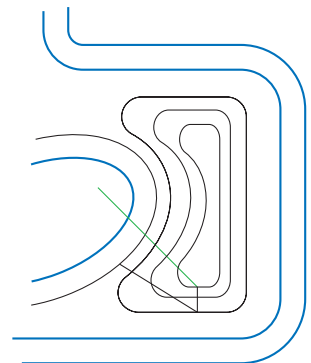
This option connects each direction with a given radius allowing for a smoother transition between path directions.

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



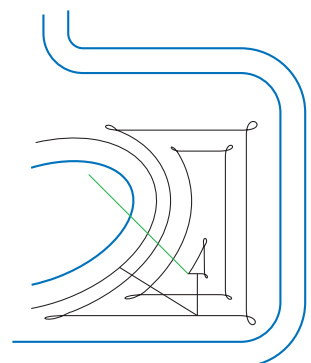
If the given radius is too large for a specific corner, it produces the largest possible radius at that point.

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.

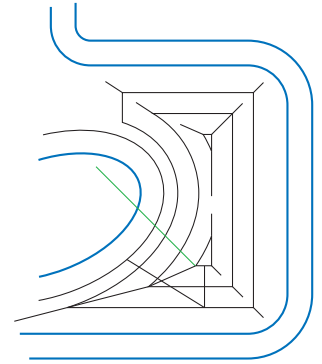


- **Sharp**

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.

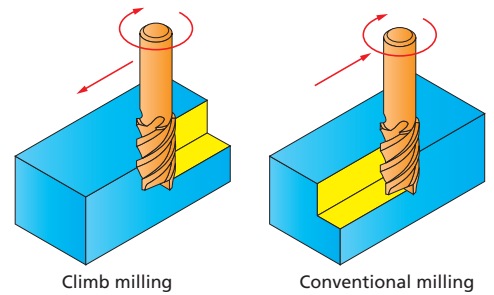


This option is not recommended for high-speed cutting.



Direction

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

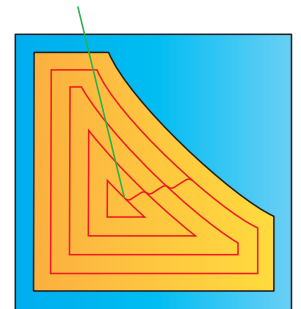


Adjacent passes connection

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

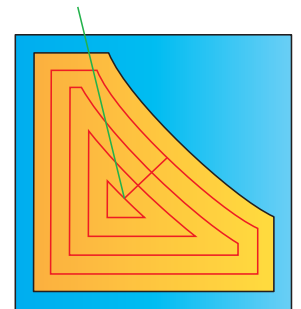
- **Smooth**

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



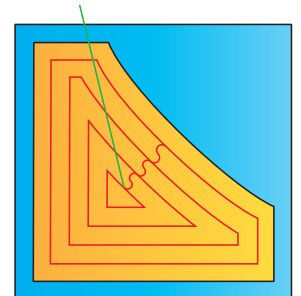
- **Linear**

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



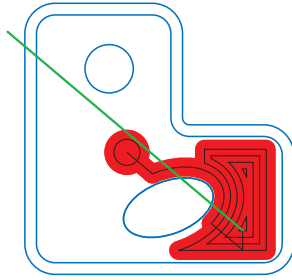
- **Rounded**

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.



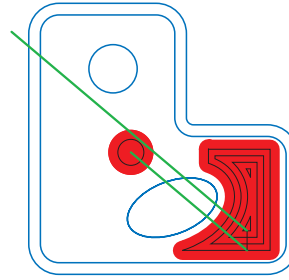
Exit material

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.

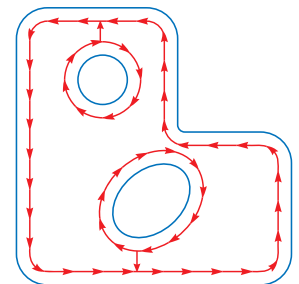


Exit material check box selected

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 InventorCAM 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 **Pick** 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. Enter the ramping angle value into the **Angle** edit box. The start point must be selected using the **Points** button.

InventorCAM 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 **Points** 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. This option 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:

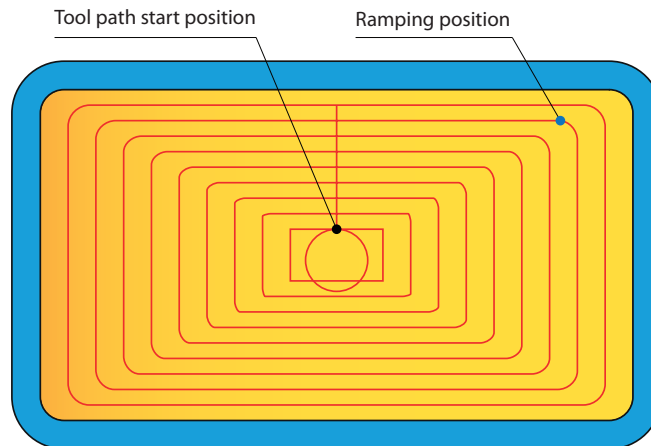
- The tool descends from the safety distance above the Upper level to the material in a circular motion up to the Tool Step down.
- At this point, the tool clears a circular path. Then it descends to the next Tool step down.
- This is repeated until the tool reaches the CAM-Part step down depth.
- When 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.

To define the ramping position, click the **Points** button. The **Ramping points** dialog box is displayed.

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, InventorCAM 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, InventorCAM 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.

Pre-Drill Operations

Upon selecting the **Pre-Drilling Operations** check box, the definition windows are activated. When a Drill operation is chosen from the list, the associated X- and Y-coordinates appear in the **Drill positions** list. The drill diameter and positions data are used for the approach point calculations.



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

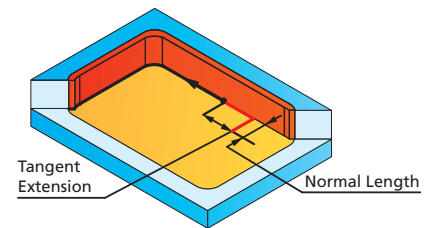
The following **Lead in** and **Lead out** strategies are available for pocket machining:

- **None**

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

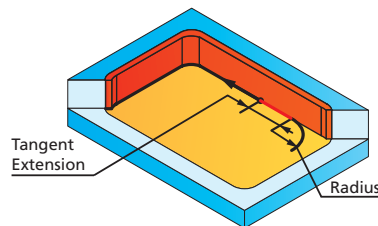
- **Normal**

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 **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.



- **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.



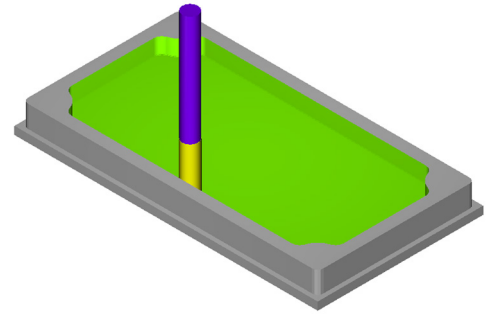
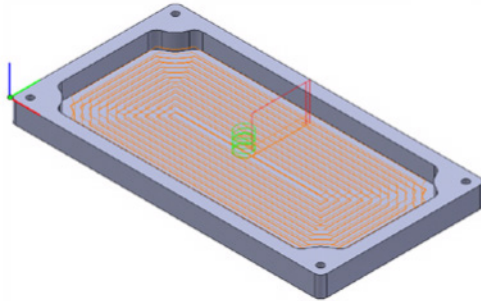
25. Calculate the Tool path

Click . The 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.

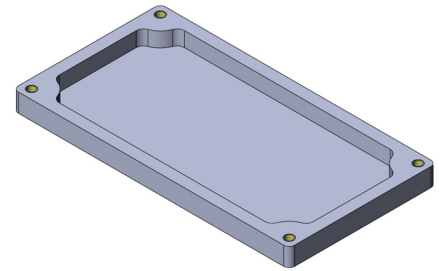


27. Add a Drilling operation

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 **InventorCAM Manager** and choose **Drilling** from the **Add Milling Operation** submenu.

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



28. Define the Drill geometry

Make sure that the **Technology** is set to the default **2D drill** option.

In the **Geometry** area, click .

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



InventorCAM 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. InventorCAM 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**

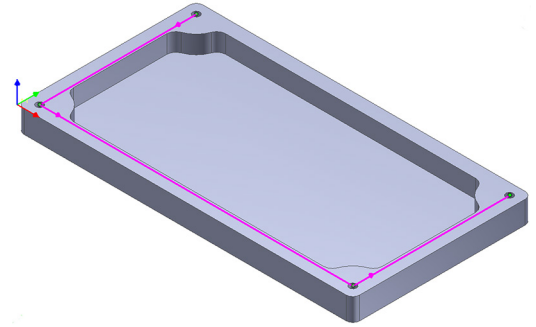
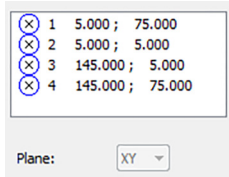
You can select the model face. InventorCAM automatically recognizes all arcs/circles located on the selected face and selects the center points as drill positions.

- **All circle/arc centers**

InventorCAM 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.

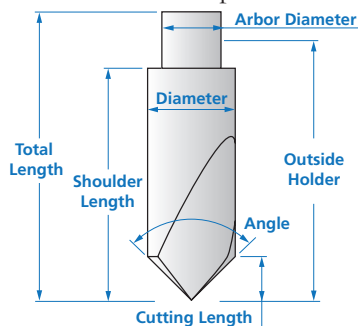
The **TOOLKIT** window is displayed.

Click the Drilling Tools icon in the Library Toolbar. Double click the **Spot drill** tool to add it in Tool Item Manager.



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.

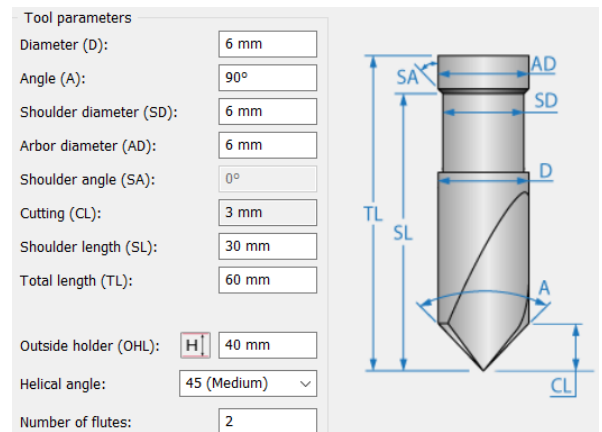


A **Spot drill** of $\text{Ø}6$ is used in this operation.

Click 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

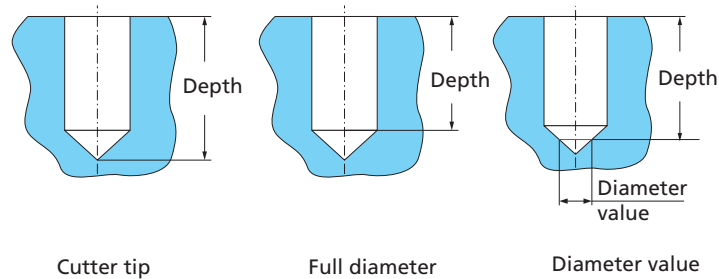
Switch to the **Levels** page of the **Drilling Operation** dialog box. Click the **Drill depth** button and select the upper face of the model. The **Drill depth** value (**0**) appears in the relevant edit box.

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.

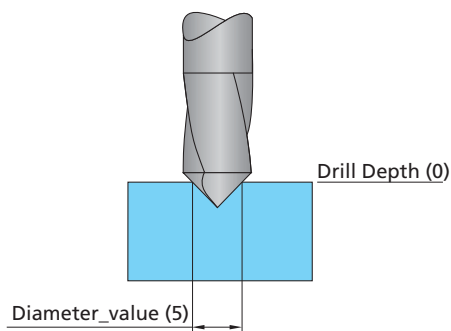
- **Diameter value**

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.

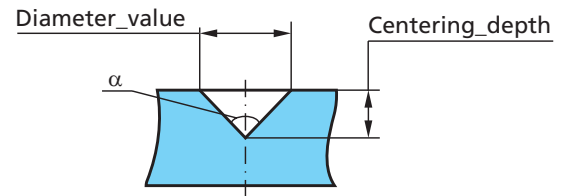


In accordance with these parameters, InventorCAM calculates the centering depth using the following formula:


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

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

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



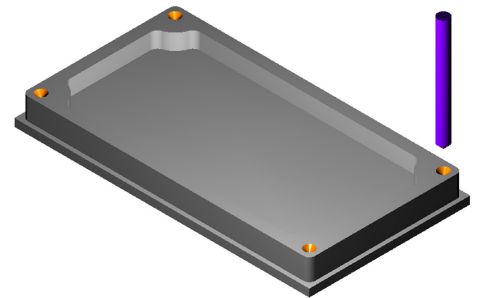
31. Calculate the tool path

Click . The 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.

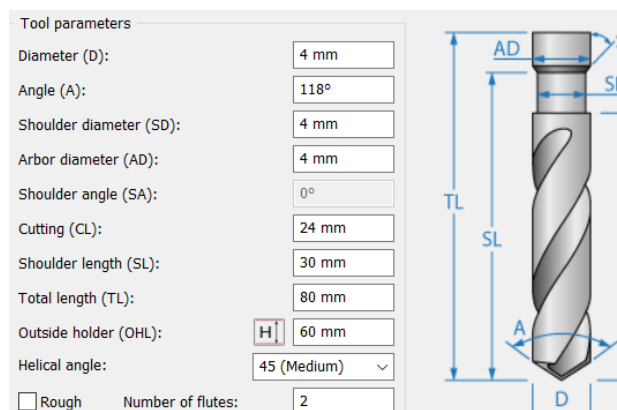


Each geometry defined in InventorCAM has a unique name. When the geometry is being defined, it is assigned a default name that can be changed. Using this name, you can choose the geometry for a specific operation.



35. Define the Tool

Define a new **Drill** tool of **Ø4** for this operation.



Click  to choose the tool for the operation.

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

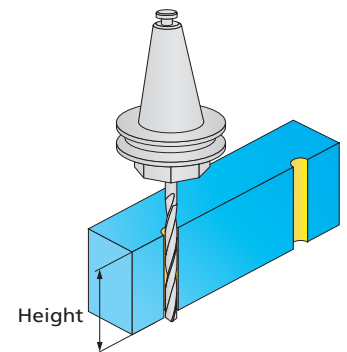
Define the spin and feed for the operation.

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



36. Define the Drilling depth

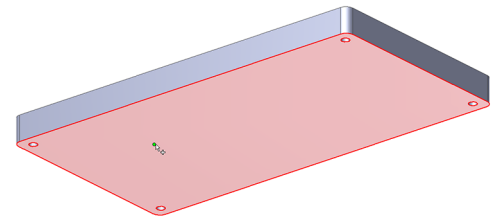
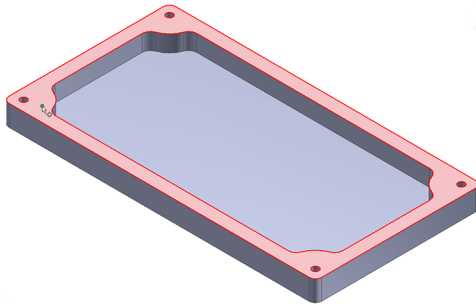
The overall height of the model is 10 mm plus the 5 mm bottom offset defined for the stock. The drilling has to be performed deeper than this depth in order to enable the tool to exit from the material and perform the through drilling.



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.

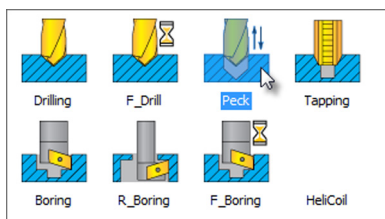


37. Define the Drilling type

InventorCAM 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.

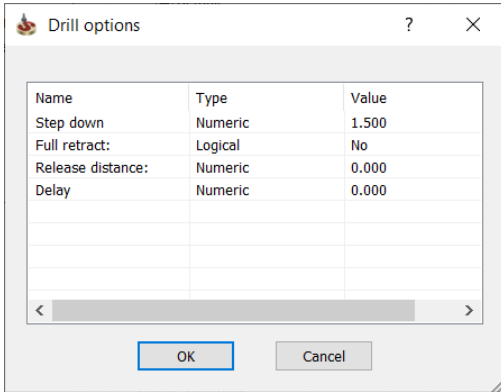
Switch to the **Technology** page and click the **Drill cycle type** button. Available drill cycles are displayed.

Select the **Peck** option. 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.




Name	Type	Value
Step down	Numeric	1.500
Full retract:	Logical	No
Release distance:	Numeric	0.000
Delay	Numeric	0.000

Confirm the data with the **OK** button.



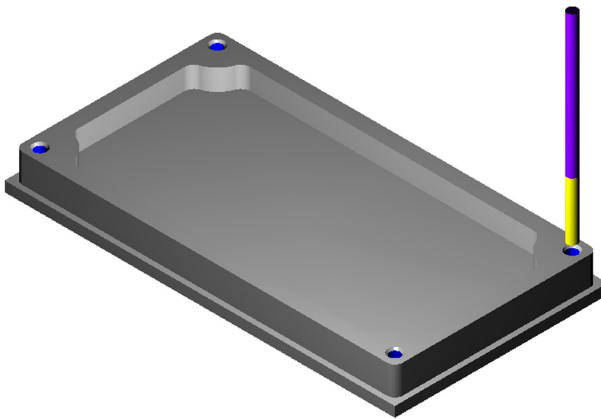
38. Calculate the tool path

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



39. Simulate the operation

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

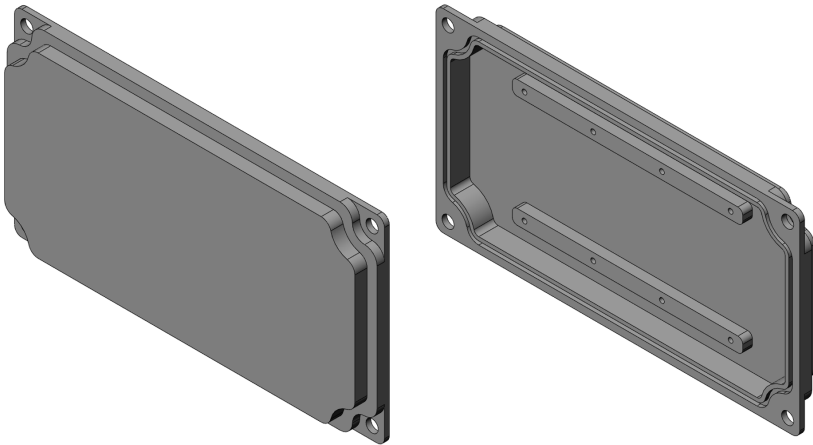


Since in the previous operation the drilling diameter was greater than that in this operation, the drilling results in a chamfer on the drilled holes.

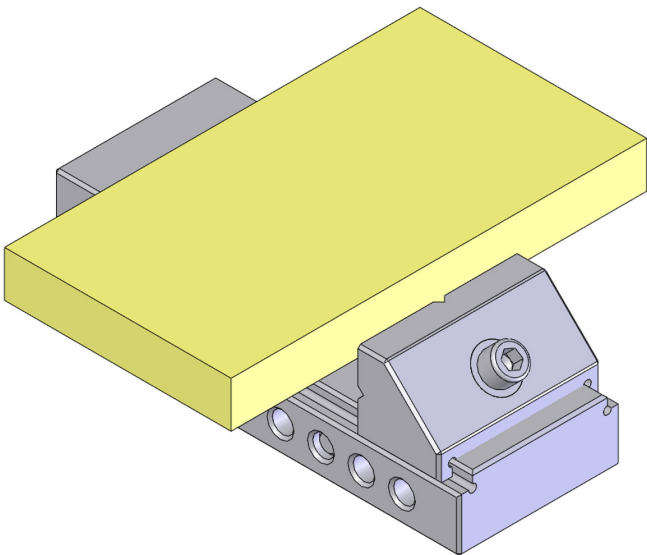
Now you have successfully finished the exercise.

Exercise #3: Cover Machining

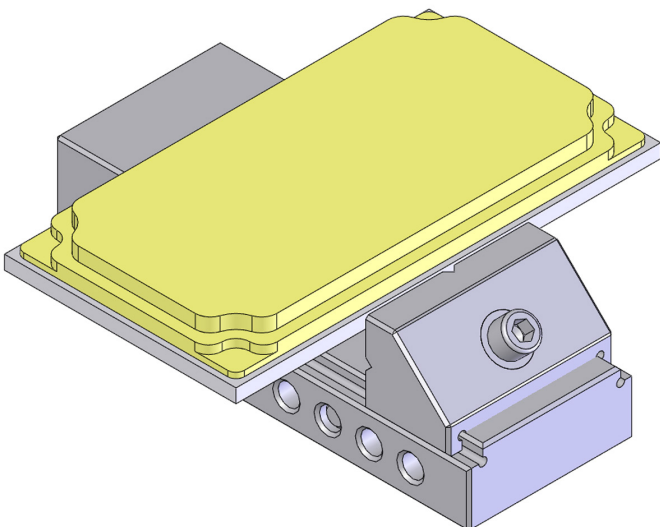
In this exercise, a number of InventorCAM 2.5D operations are used for the cover part machining.



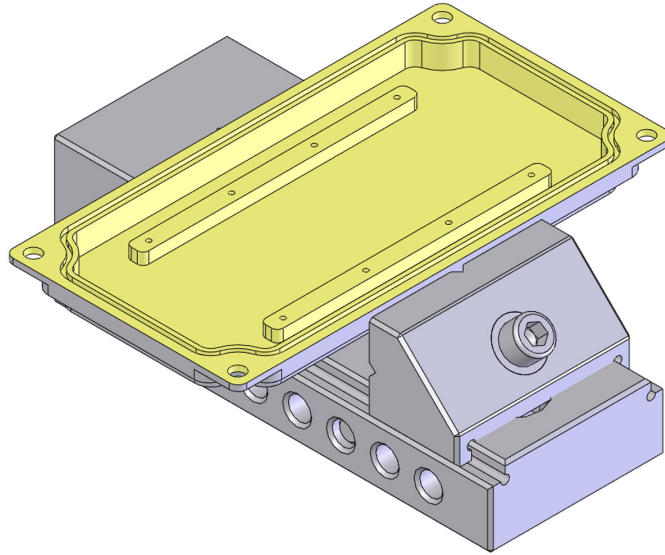
The cover is machined on the 3-Axis milling CNC-machine using the machining vice. The part is machined using two setups.



At the first stage, the workpiece is positioned in the vice as shown below. The highlighted faces are machined.

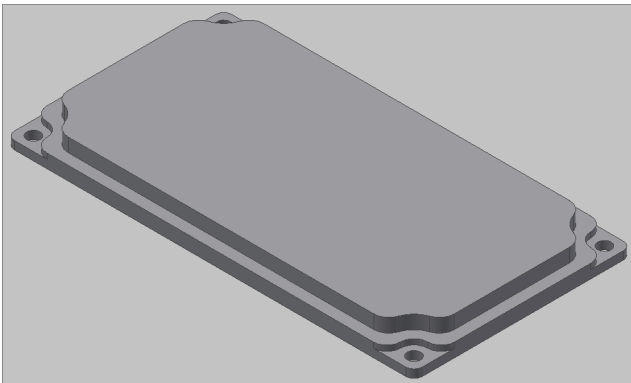


At the next stage, the rest of the cover faces are machined using the second positioning.



1. Load the Autodesk Inventor model

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



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



2. Start InventorCAM

To activate InventorCAM, click **InventorCAM 2023** and choose **Milling** from the **New** submenu.

InventorCAM is started, and the **Milling Part Data** dialog box is displayed.



3. Choose the CNC-controller

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

In this exercise, you use a 3-Axis CNC-machine. Choose the **gMilling_3x** CNC-controller from the list.



4. 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 **Define** section of the **Milling Part Data** dialog box.

The **Stock** dialog box is displayed.

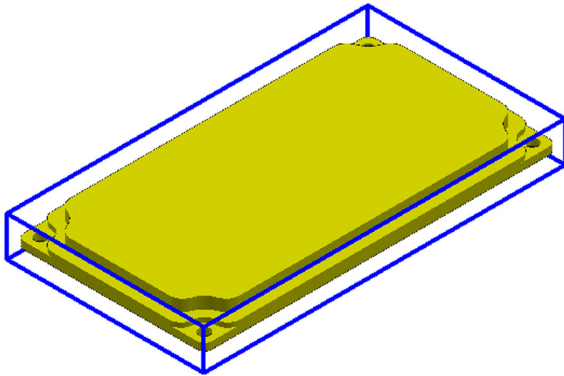


When the Coordinate System is not defined, InventorCAM enables you to use only the **Box** method of the Stock model definition.

InventorCAM 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 **Stock** dialog box with . The **Milling Part Data** dialog box is displayed.



5. Define the Coordinate System

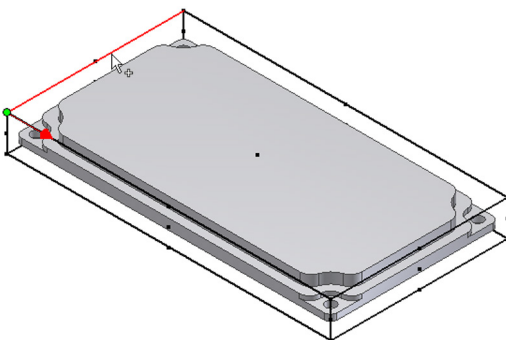
Click the **CoordSys** button in the **Define** area to start the Coordinate System definition.

The **CoordSys** dialog box is displayed.

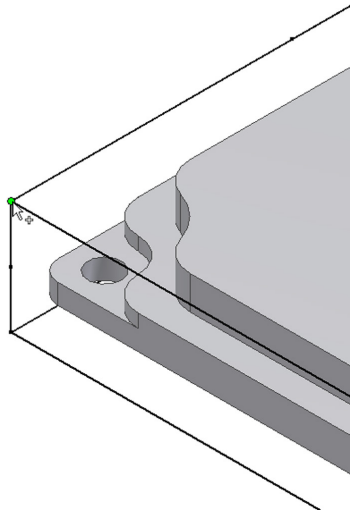
In the **Definition options** list, choose the **Define** option.

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.

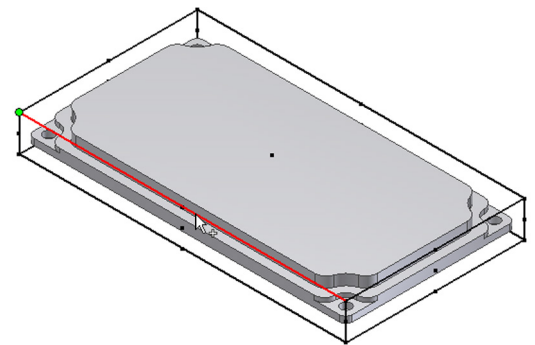



Click on the stock model edge as shown to define the X-axis of the Coordinate System.

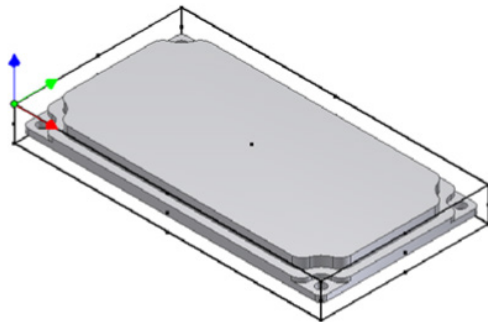
Click on the stock model edge as shown to define the Y-axis of the Coordinate System.



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 Coordinate System is displayed.



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

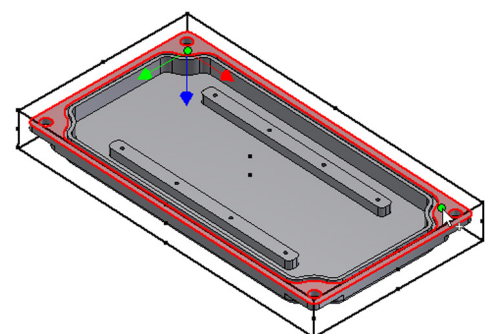
Define the Part Lower level directly on the solid model. This parameter defines the lower surface level of the part to be milled.

In the **Levels: Planar** section, 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 by clicking .

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.

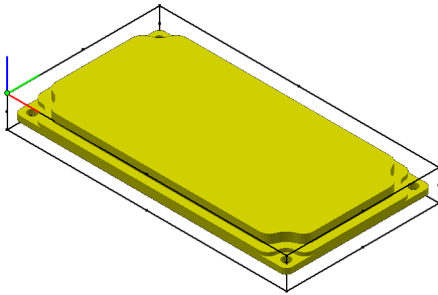


6. Define the Target model

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

The **Target** dialog box is displayed.

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



Click on the solid body to select a target model. The solid body is highlighted.

Confirm the **Target** dialog box with .

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



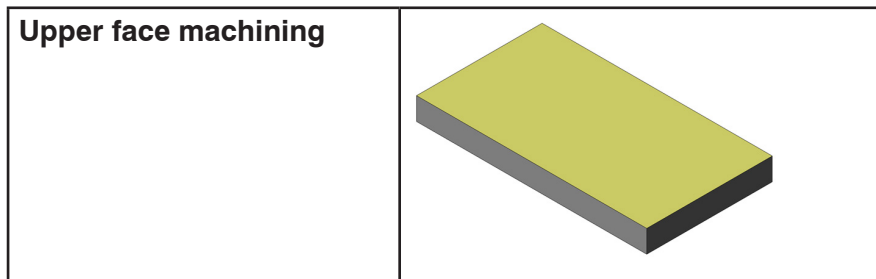
7. Save the CAM-Part data

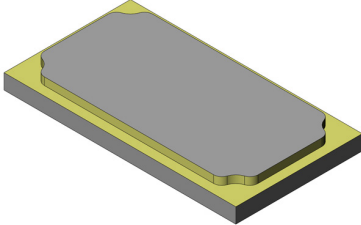
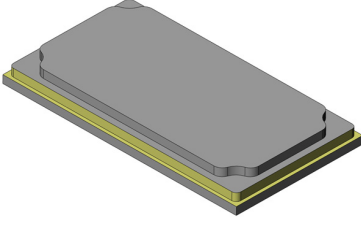
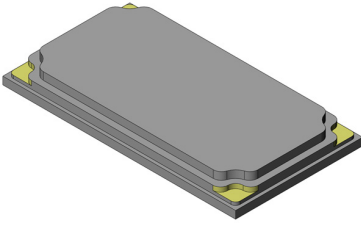
In the **Milling Part Data** dialog box, click .

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

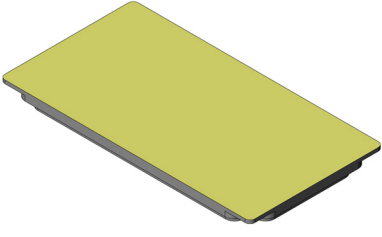
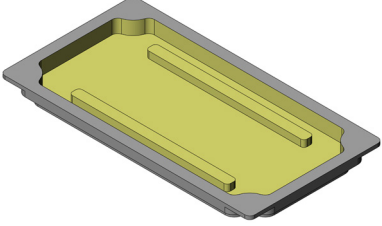
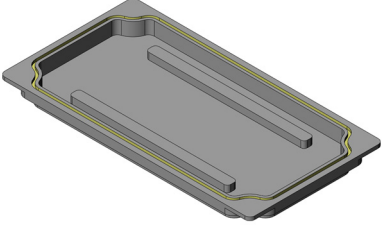
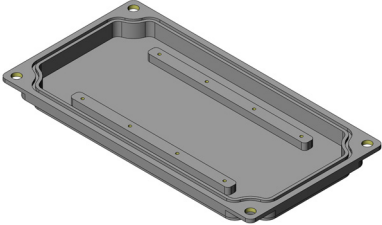
At this stage, the definition of the CAM-Part is finished.

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



Upper profile machining	
Lower profile machining	
Hole pads machining	

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

Upper face machining	
Pocket machining	
Slot machining	
Holes machining	



8. Add a Face Milling operation

In **InventorCAM Manager**, right-click the **Operations** header and choose **Face** from the **Add Milling Operation** submenu. The **Face Milling** operation is used for the upper face machining.

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



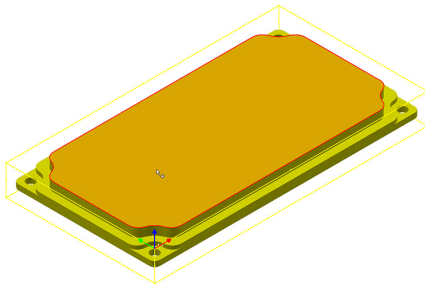
9. Define the Face Milling geometry

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

Using the default **Model** option, click the **Define** button. The **3D Geometry** dialog box is displayed. Click on the solid model to select a face.

Confirm the **3D Geometry** dialog box with .

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**.



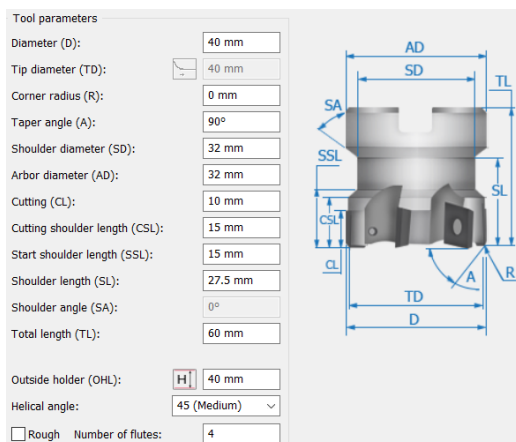
Confirm the **Face Milling Geometry** dialog box by clicking . The geometry is defined for the operation.




10. Define the Tool

Define a new **Face mill** of **Ø40** for this operation and edit the following parameters:

- Set the **Shoulder length** to **27.5**
- Set the **Cutting length** to **10**

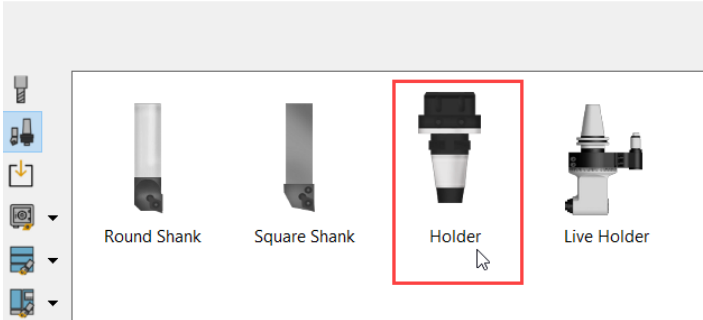


InventorCAM enables you to define a variety of tool holders to help you check and prevent all possible collisions between the tool holding system and the workpiece. This feature also enables you to see a more realistic simulation in the **SolidVerify** simulation.

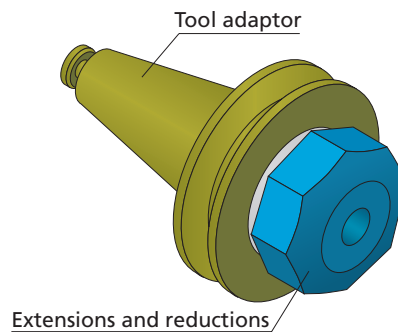
Select the **Holder**  icon from the Library Toolbar. Double click the Holder icon to display the available holders.

TOOLKIT : EXERCISE3

File View Help



The InventorCAM 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.

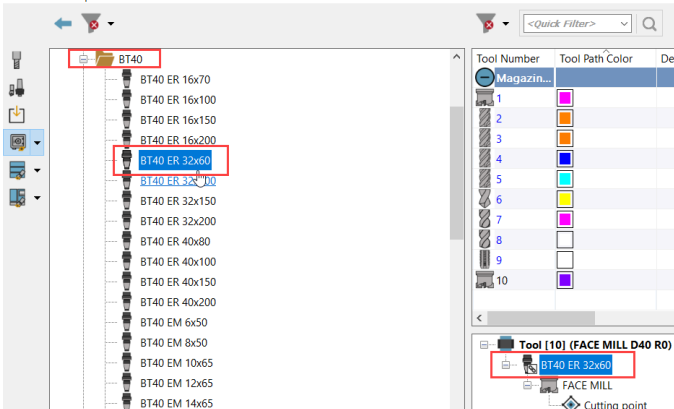


InventorCAM enables you to define the Tool adaptor in the *.vmid file under **Machine Definition > Devices > Spindle > Stations**.

Expand **BT40** section and double click the **BT40 ER32x60** collet chuck by clicking its check box. This collet chuck is suitable for the chosen tool diameter (40 mm).

TOOLKIT : EXERCISE3

File View Help

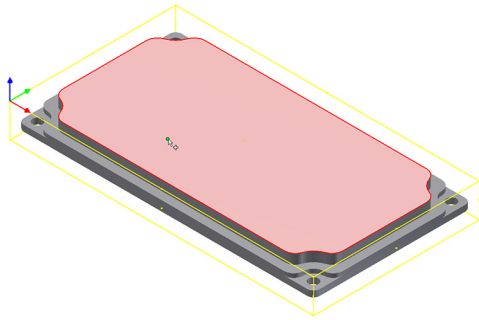


Choose the defined tool for the operation by clicking . The **Face Milling Operation** dialog box is displayed.



11. 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.



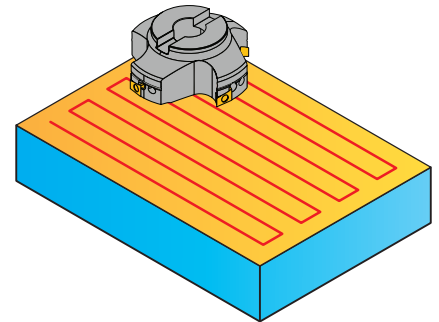
12. Define the technological parameters

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



Hatch Machining Technology

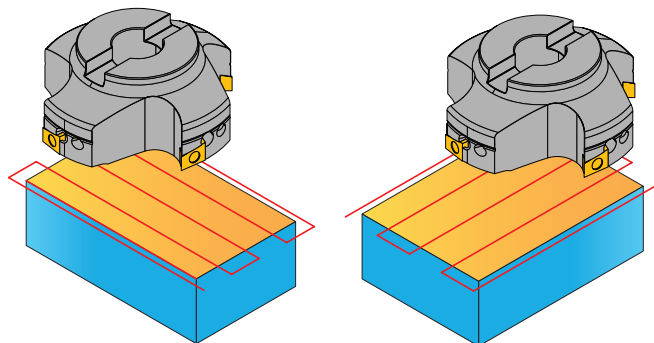
The machining is performed in a linear pattern.



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

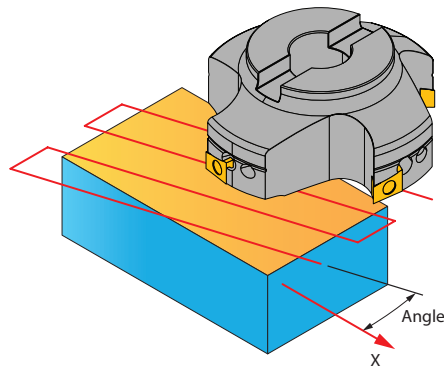
- **Hatch angle**

When the **Automatic optional angle** option is chosen, 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** parameter enables you to change the hatching angle.

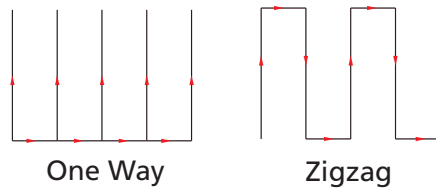
When the **User-defined angle** option is chosen, you can determine the angle of the tool path.



- **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.



- **Cutting order**

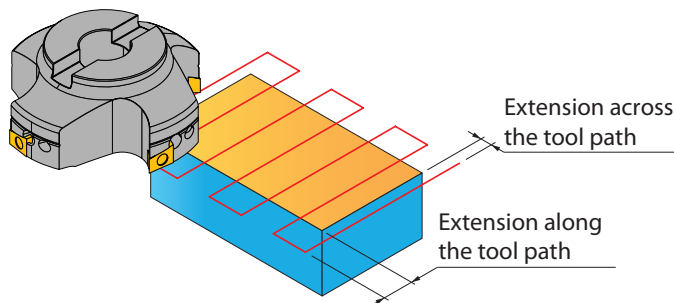
The **Side to side** option creates a tool path that starts the cutting on one side of the geometry, machines the face according to the Cutting direction and then performs its last cutting pass on the side opposite of the starting side. Selecting the **Side to side** option unchecks and enables the **Classic spacing** option in the **Technology** page.

The **Side to middle** option creates a tool path that starts the cutting on one side of the face geometry and then moves to the opposite side before making the next cut. Each successive series of opposing parallel passes step inward until the last cutting pass is performed in the middle.

- **Extension**

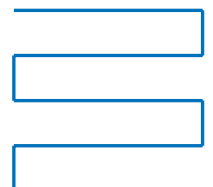
During face milling, the tool path is extended over the edges of the machined face. The **Extension** section enables you to define the extension both along the tool path (the **Along** section) and across the tool path (the **Across** section).

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.

- **Approach**

When the Roll in option is selected, the tool approaches the material while rolling into it with an arc movement. The Approach distance parameter enables you to define the length of the arc's linear extension.



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

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

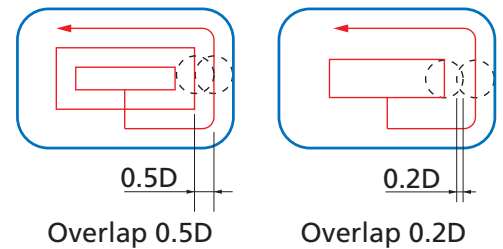
Switch back to the **Technology** tab.

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** and **Classic spacing** check boxes are selected.



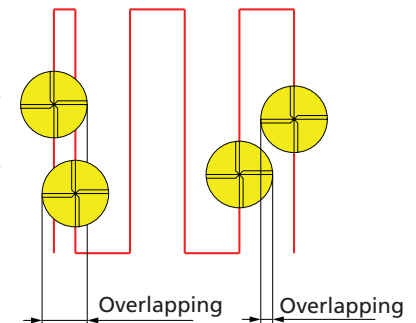
Overlap

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**.



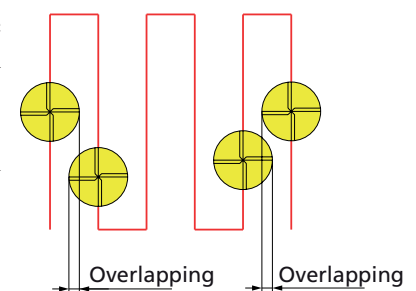
Equal Step Over

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**. Make sure that the **Finish** check box is selected to perform finishing of the face in this operation. This option enables you to remove the remaining offset with the last cutting pass.

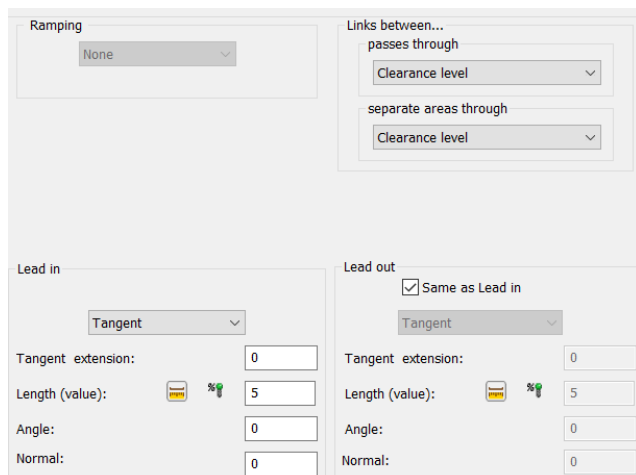


13. Define the Lead in and Lead out

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, use the default **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.



14. Save and Calculate

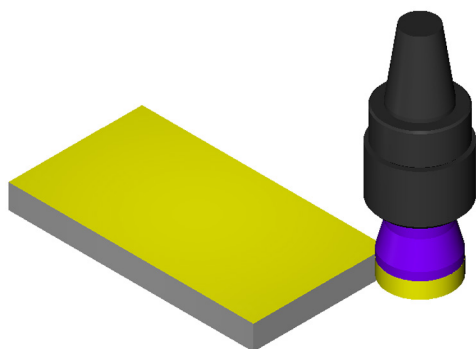
Click . The Face Milling operation data is saved, and the tool path is calculated.



15. 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.



Close the simulation. The **Face Milling Operation** dialog box is displayed.

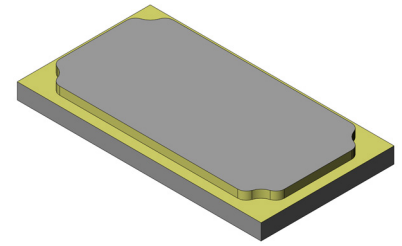
Close this dialog box with .




16. 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 **InventorCAM Manager**, add a new Profile operation.



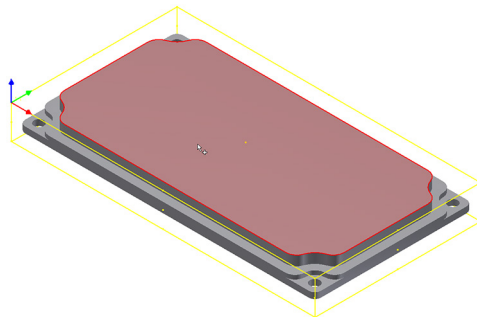
17. Define the Geometry

Click  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. InventorCAM 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.

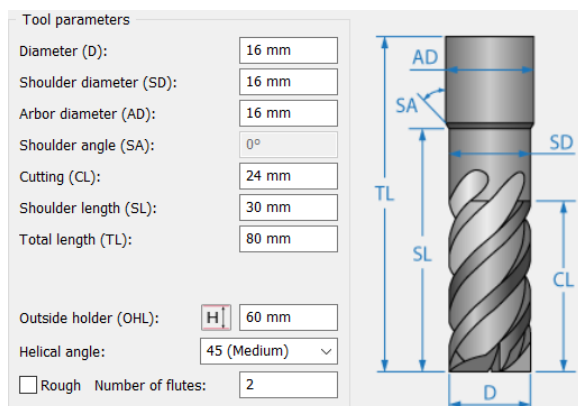
Click  to choose the selected chain as the geometry. The **Geometry Edit** dialog box is displayed.

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



18. Define the Tool

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



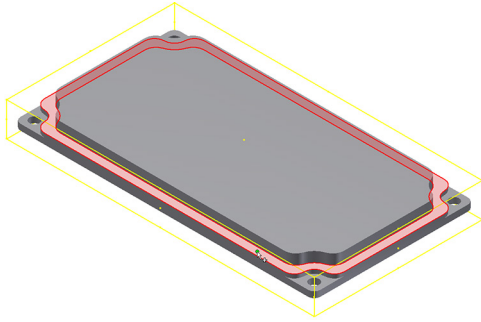
Select the **BT40 ER32x60** collet chuck as the holder.

Confirm the tool selection by clicking .



19. 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.



20. 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.

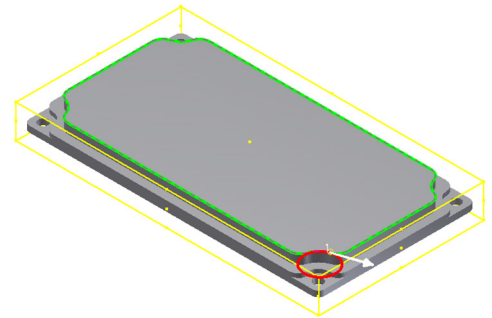
Close the **Modify Geometry** dialog box with the .

Now you have to define the roughing and finishing parameters. InventorCAM 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**.

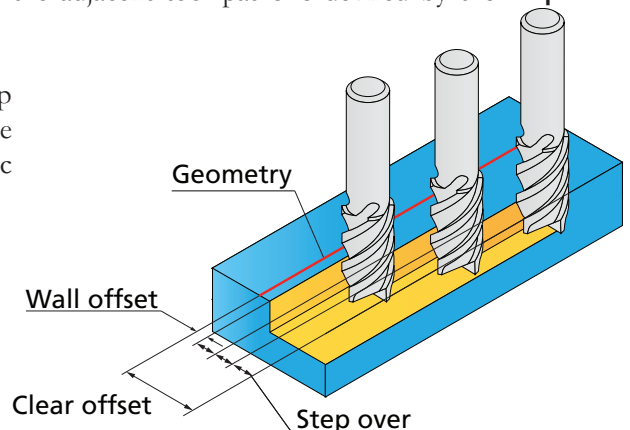


Clear Offset

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 will be removed in one step.




21. 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.



22. Save and Calculate

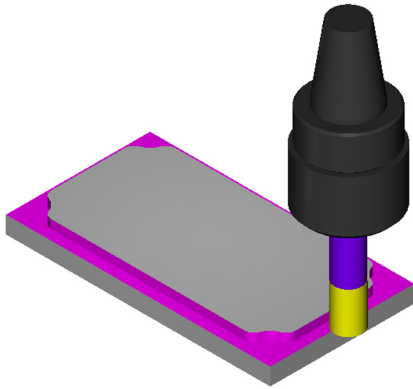
Click . The Profile operation data is saved, and the tool path is calculated.



23. Simulate

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

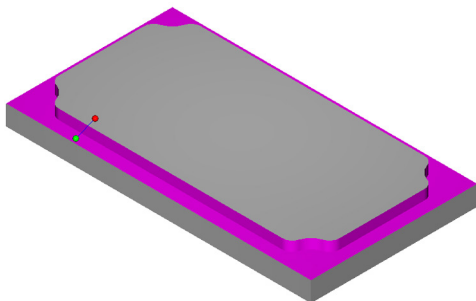
Switch to the **SolidVerify** page and start the simulation.



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.

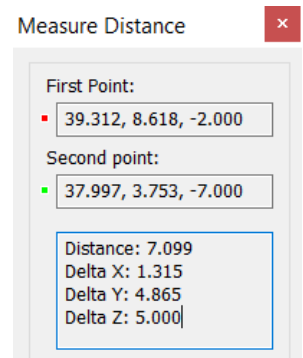
Click  on the toolbar. The **Measure Distance** dialog box is displayed.

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.

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

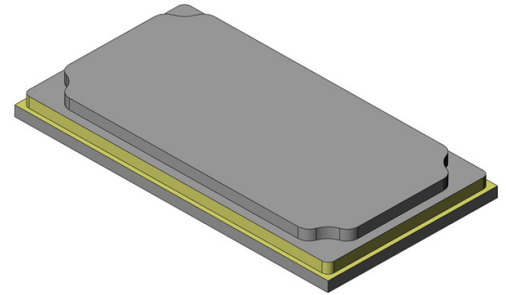
Close the **Profile Operation** dialog box with .



24. Add a Profile operation

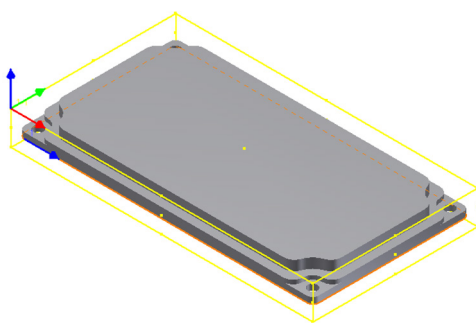
At this stage, you have to define a Profile operation in order to machine the lower profile of the cover.

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



25. Define the Geometry

Click  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 **Constant Z Propagation** option. The closed chain is automatically selected.

The confirmation message is displayed. Confirm it by clicking **Yes**.

Confirm the geometry with .

26. Define the Tool

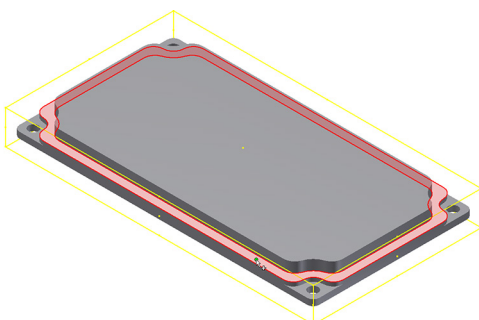
In this operation, use the same tool as in the previous operation (**Ø16 End mill**).




27. Define the Profile depth

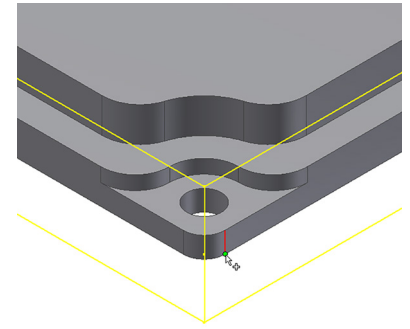
You have to define a new Upper level for the operation taking into account the already machined faces. Switch to the **Levels** page and click the **Upper level** button in the **Milling levels** area.

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



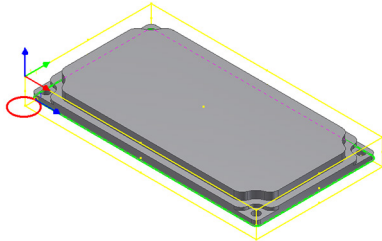
Confirm the definition of the **Upper level** with .

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



28. Define the technological parameter

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 adjust the tool position relative to the geometry.



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

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.




29. Define the Lead in and Lead out

Switch to the **Link** page of the **Profile Operation** dialog box. In the **Lead in** section, use the default **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.



30. Save and Calculate

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

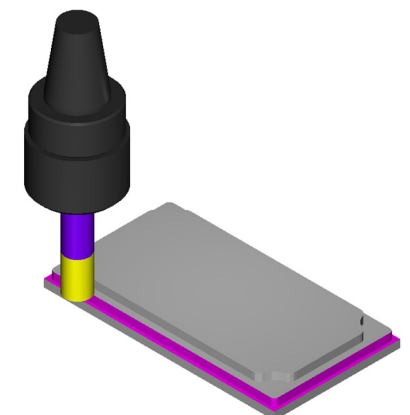


31. Simulate

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.

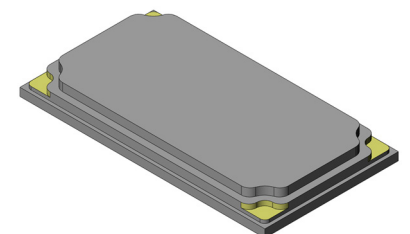
Close the simulation. The **Profile Operation** dialog box is displayed. Close the **Profile Operation** dialog box with .



32. Add a Profile operation

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

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



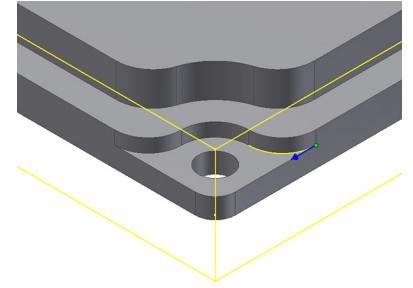
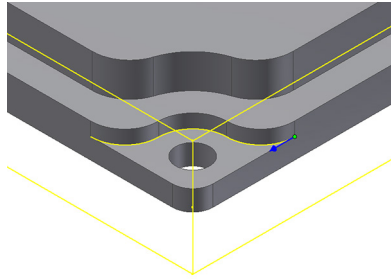


33. Define the Geometry

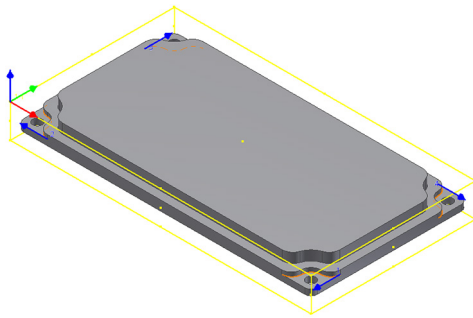
In the **Geometry** page, click . The **Geometry Edit** dialog box is displayed.

Click on the model edge as shown.

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



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 .



34. Define the Tool

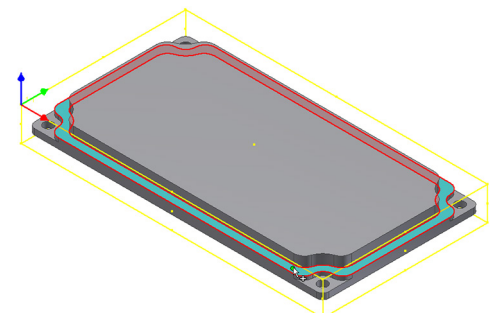
Define a new tool for the operation. Use an **End mill** of **Ø8** with the **BT40 ER32x60** tool holder.

Tool parameters	
Diameter (D):	8 mm
Shoulder diameter (SD):	8 mm
Arbor diameter (AD):	8 mm
Shoulder angle (SA):	0°
Cutting (CL):	24 mm
Shoulder length (SL):	30 mm
Total length (TL):	80 mm
Outside holder (OHL):	H1 60 mm
Helical angle:	45 (Medium)
<input type="checkbox"/> Rough	Number of flutes: 2

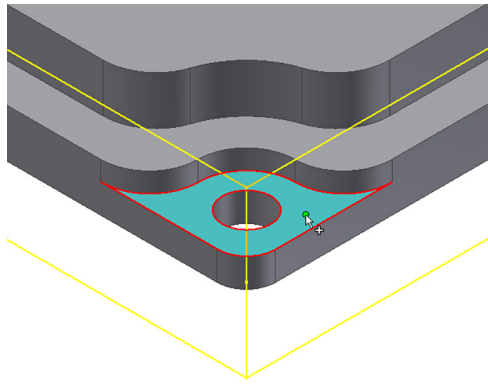


35. Define the Milling levels

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.



36. 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 will be removed in one step.




37. 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.



38. Save and Calculate

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



39. Simulate

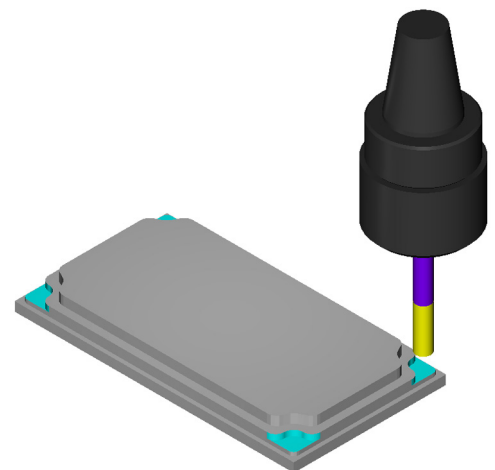
Click the **Simulate** button in the **Profile Operation** dialog box. The **Simulation** control panel is displayed.

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

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

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

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





40. 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.

InventorCAM enables you to define the second Coordinate System.

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

Click the **Add** button. The **CoordSys** dialog box is displayed.

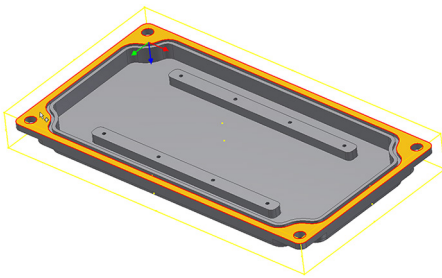
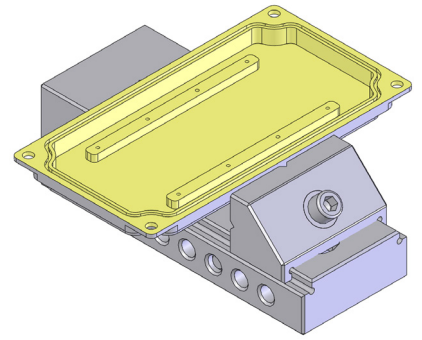
In the **Mac CoordSys number** field, set the value to **2**. Changing of the **Mac CoordSys number** means that a new clamping is used.

Make sure that the **Select face** mode is chosen.

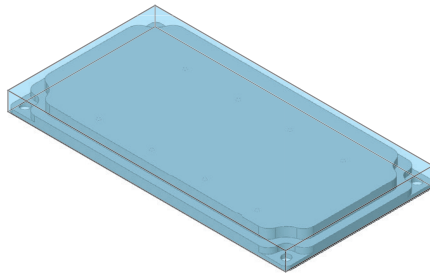
When this mode is chosen, InventorCAM enables you to define a new Coordinate System by selecting a planar face. In this case, the Z-axis of the Coordinate System is normal to the selected face.

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

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

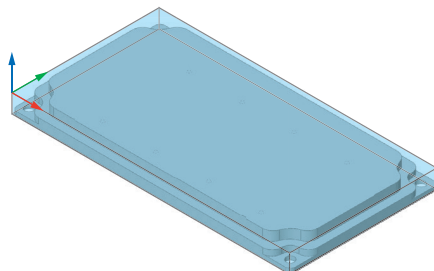


In this case, the box surrounding the model is calculated.



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 **Modify by 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.

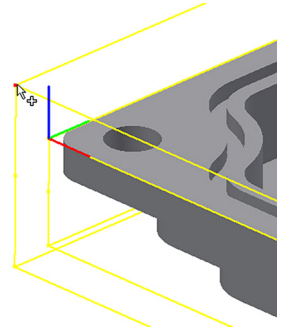
InventorCAM automatically determines the **Part Upper level** and the **Part Lower level** parameters relative to the Coordinate System location.

Confirm the dialog box with .

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

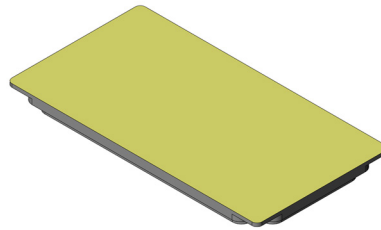
Two Machine Coordinate Systems are listed in the **CoordSys Manager** dialog box. Using them you can program operations for different positions (clamping). The **Machine Coordinate System #2** is used for the machining of the back face and the internal faces.

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



41. Add a new Face Milling operation

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



42. Define the Geometry

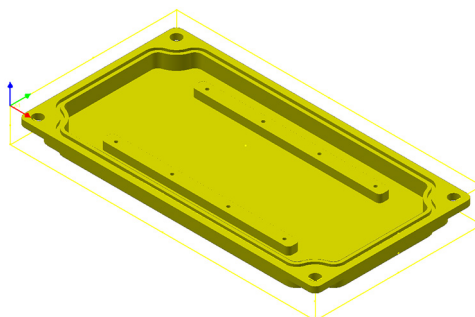
Since this operation is performed with the second Coordinate System position, choose the **Machine Coordinate System #2**.

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

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

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.

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.



43. Define the Tool

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

Click the **Select** button in the **Tool** page. Choose the Face mill tool that was defined for the first face milling operation.

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

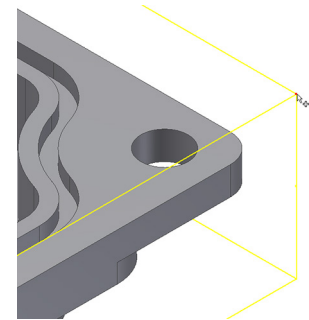
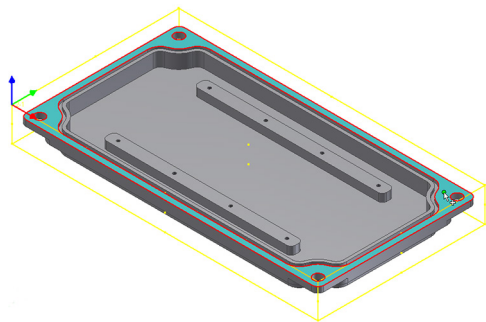


44. Define the Milling levels

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**.



45. Define the technological parameters

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



46. 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, use 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**.

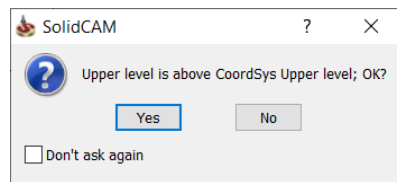
In the **Lead out** section, select the **Same as Lead in** check box.



47. Save and Calculate

Click  .

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



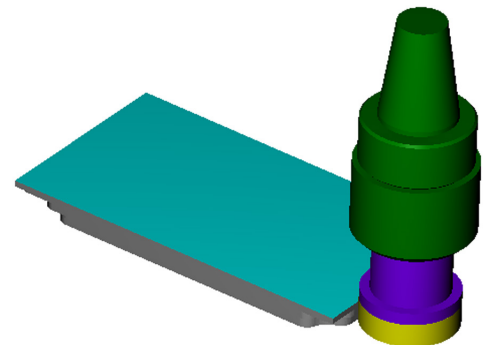
Confirm this message with the **Yes** button. The operation data is saved, and the tool path is calculated.



48. Simulate

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

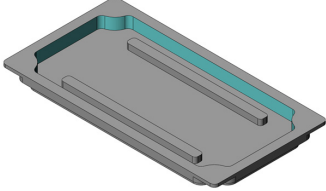
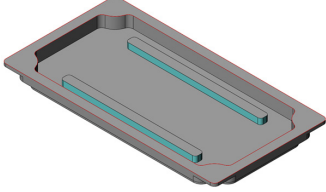
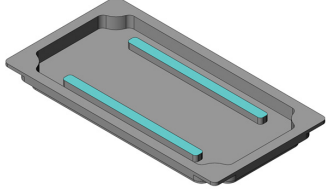
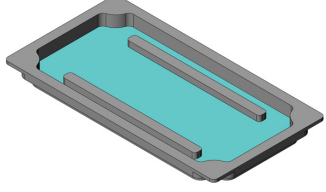
The bottom face machining is finished. Now you have to perform the pocket machining.



Pocket machining overview

The pocket is machined in several technological steps:

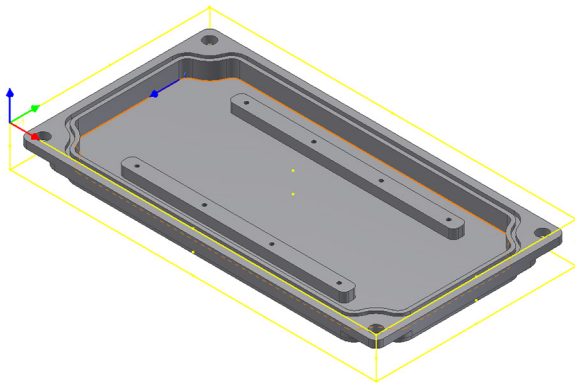
	<p>The rough machining of the upper part of the pocket. The machining is performed until the Z-level of the pads is reached.</p>
	<p>The rough machining of the pocket with islands (pads). The machining is performed from the upper face of the pads till the pocket floor.</p> <p>At this stage, two operations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

	The finish machining of the outside wall of the pocket.
	The finish machining of the island walls.
	The finish machining of the island top face.
	The finish machining of the pocket floor.



49. 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.



50. Define the Geometry

Choose the **Machine Coordinate System #2** and define the geometry as shown below using the **Constant Z Propagation** option.



51. Define the Tool

Choose the already defined **Ø16 End mill** for the pocket rough machining operation.

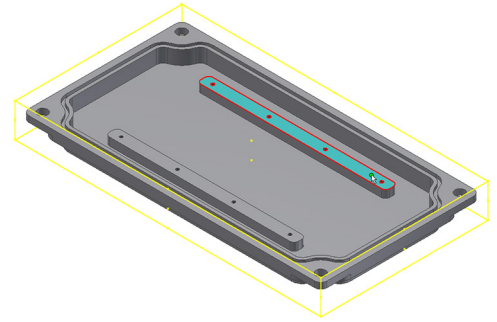


52. Define the Pocket depth

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

InventorCAM 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.



53. 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.



54. Define the Ramping strategy

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

Set the **Radius** of the descent helix to **3**.

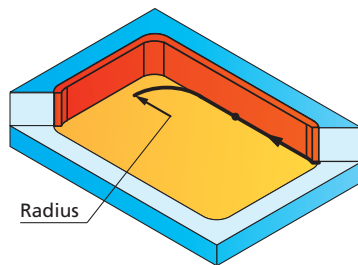


55. Define the Lead out


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.



56. Save and Calculate

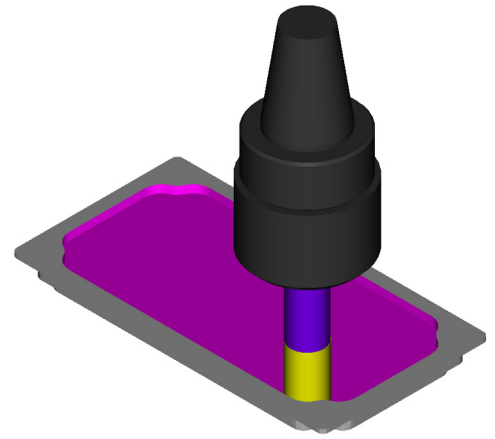
Click . The Pocket operation data is saved, and the tool path is calculated.



57. Simulate

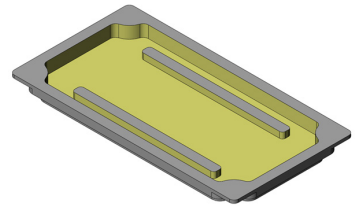
Perform the simulation of the Pocket operation in the **SolidVerify** mode.

During the simulation, notice the helical lead in movement.



58. 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).

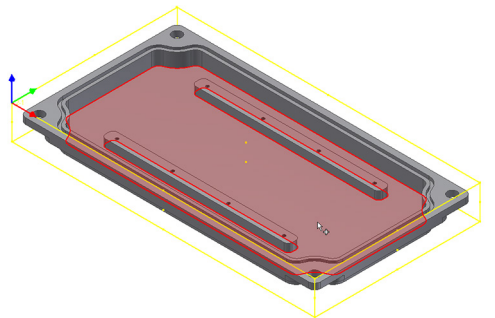


59. Define the Geometry

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

In the same manner as explained in the **Step #17** of this exercise, define the multi-chain geometry by clicking on the pocket bottom face as shown below.

InventorCAM 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.



60. Define the Tool

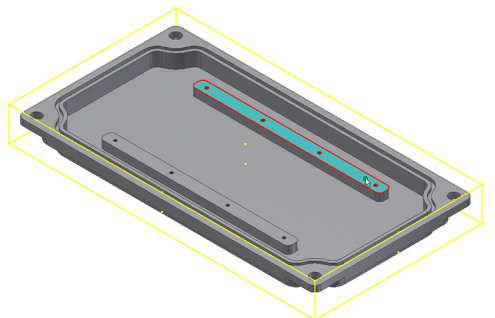
Choose the **Ø16 End mill** for the operation.



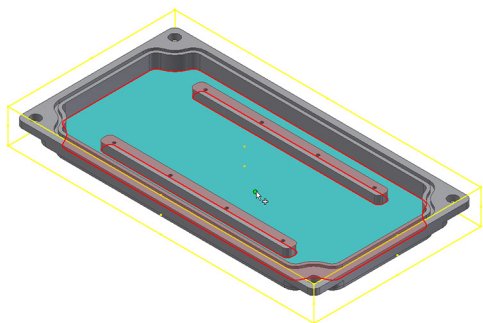
61. Define the Milling levels

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**.



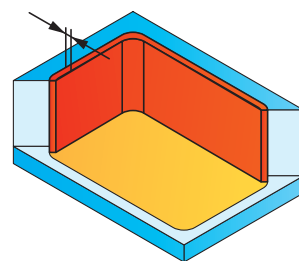
62. Define the technological parameters

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

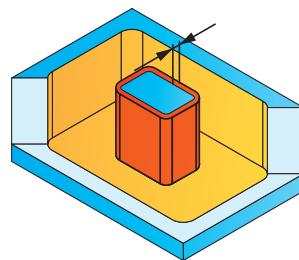
In the **Offsets** section of the **Technology** page, set the **Wall offset**, **Island offset** and **Floor offset** values to **0.2**. These offsets remain unmachined during roughing and are removed in the next finishing operations.



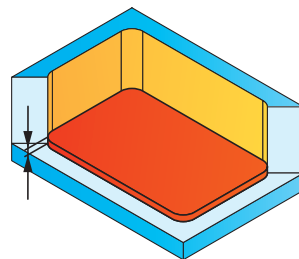
Wall offset – the roughing offset remaining on the wall of the pocket.



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



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



63. Define the Ramping strategy

In the **Ramping** section of the **Link** page, choose the **Helical** option. Set the **Radius** of the descent helix to **3**.




64. Define the Lead out

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



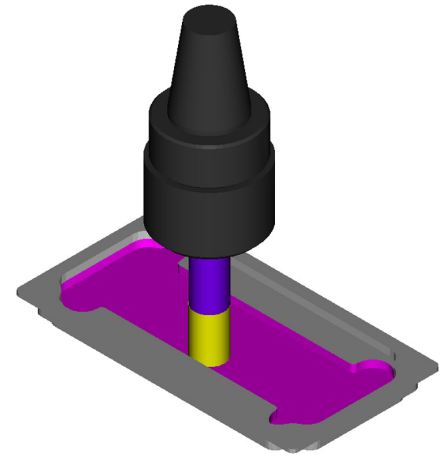
65. Save and Calculate

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



66. 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.



67. Add a new Pocket operation

Add a new Pocket operation. InventorCAM 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.



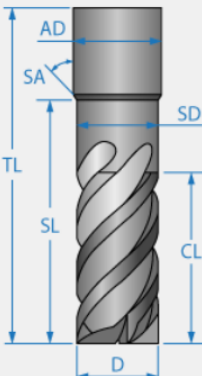
68. Define the Tool

Define a new tool for the operation.


Click the **Select** button in the **Tool** page.

Define a new **End mill** tool with the following parameters.

Tool parameters	
Diameter (D):	6 mm
Shoulder diameter (SD):	6 mm
Arbor diameter (AD):	6 mm
Shoulder angle (SA):	0°
Cutting (CL):	24 mm
Shoulder length (SL):	30 mm
Total length (TL):	80 mm
Outside holder (OHL):	<input type="text" value="H1"/> 60 mm
Helical angle:	45 (Medium) ▾
<input type="checkbox"/> Rough	Number of flutes: 2



Select **BT40 ER32x60** collet chuck as the holder.

Click  to choose the defined tool.

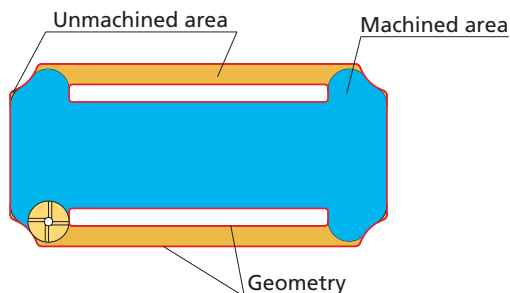


69. Define the Rest material parameters

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.

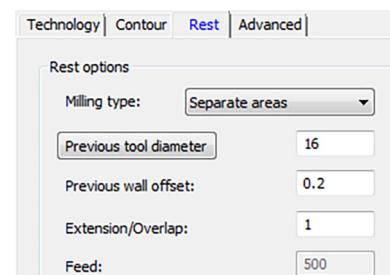


The **Rest material** option enables you to remove the material from these areas without defining a new geometry.

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, InventorCAM 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 **TOOLKIT** window 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 .

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**.



70. Save and Calculate

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

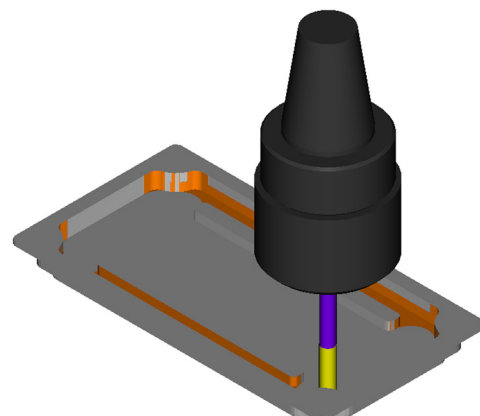


71. Simulate

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.

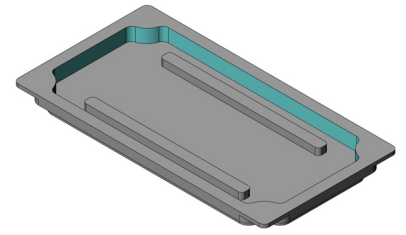
At this stage, the rough machining of the pocket is completed, and you have to program the finishing operations.





72. Add a Profile operation

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

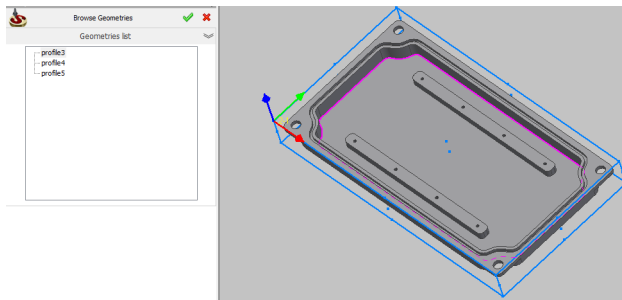


73. Define the Geometry

In the **Geometry** page, click 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 **profile3** geometry from the list.



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



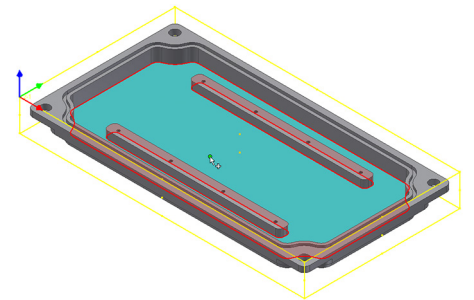
74. Define the Tool

Choose the already existing **Ø6 End mill**.



75. Define the Profile depth

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



76. Define the technological parameters

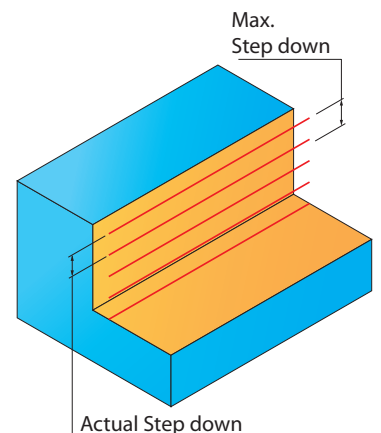
In this operation, use the **Equal step down** option to keep an equal distance between all Z-levels.



Equal Step down

This option enables you to perform all cuts at an equal Z-level distance one from the other. InventorCAM 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**.

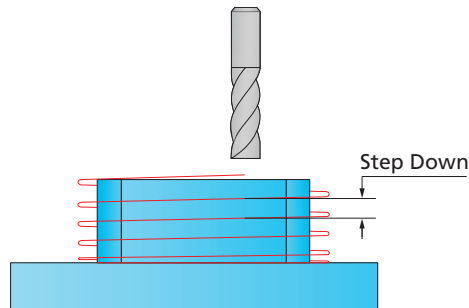
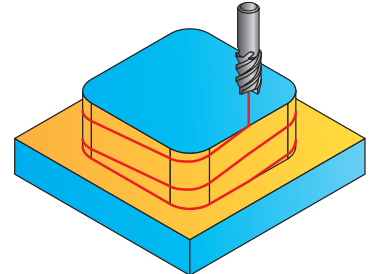
In the **Depth type** area, choose the **Helical** option.



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, InventorCAM performs the last cut with the constant-Z movement at the Profile depth.




77. 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.



78. Save and Calculate

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



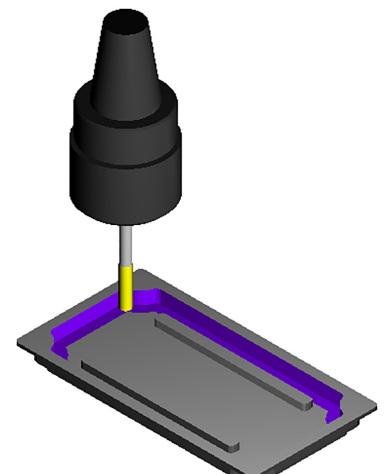
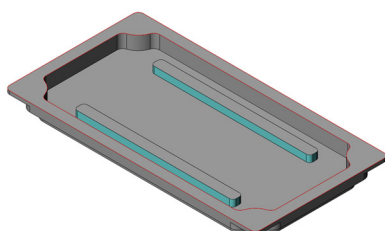
79. Simulate

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



80. Add a Profile operation

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



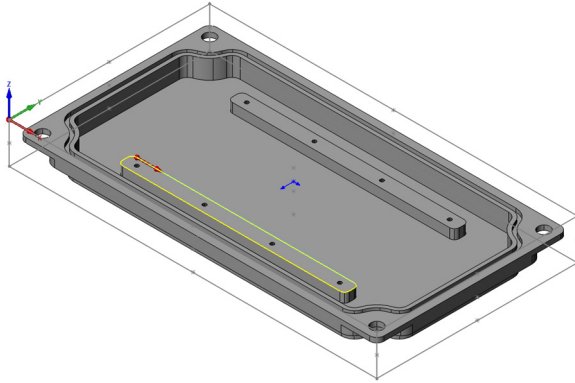


81. Define the Geometry

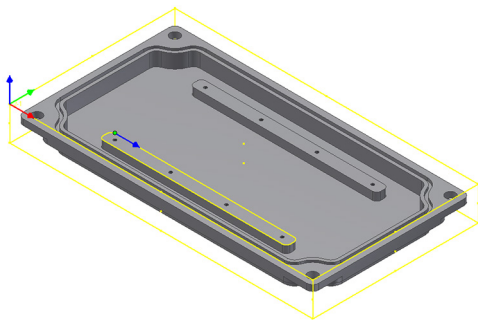
Choose the **Machine Coordinate System #2** for the operation and click  in the **Geometry** page. The **Geometry Edit** dialog box is displayed.

Select the edge of the island as shown.

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

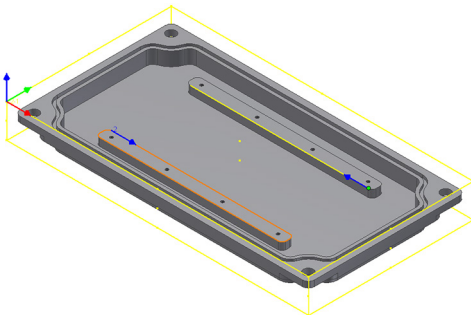


In the **Single entities** section, choose the **Constant Z Propagation** option. The chain is automatically completed.



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 **Constant Z Propagation** 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 .



Define the Tool

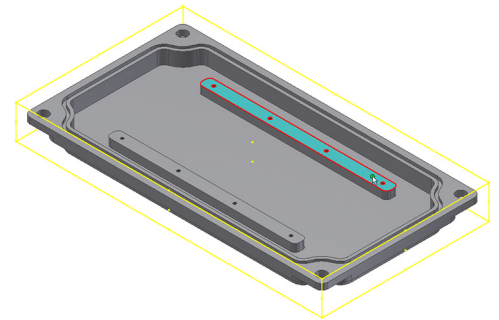
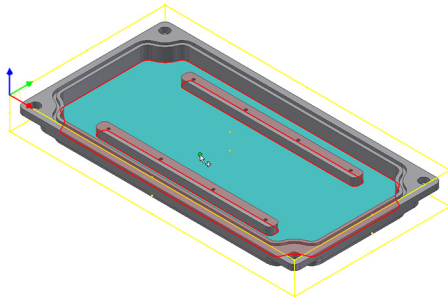
Choose the **Ø6 End mill** for the operation.



82. 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.

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



83. 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.




84. 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.



85. Save and Calculate

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



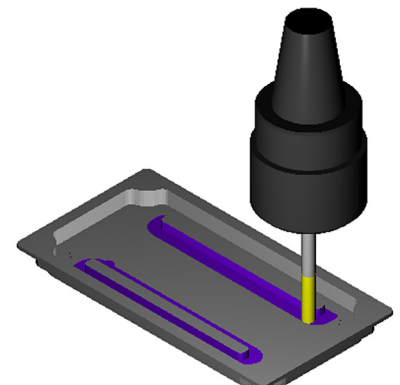
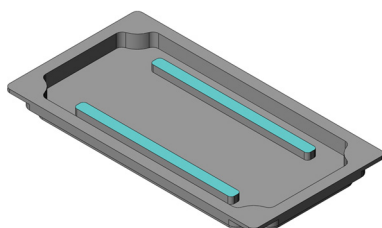
86. Simulate

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



87. Add a Face Milling operation

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





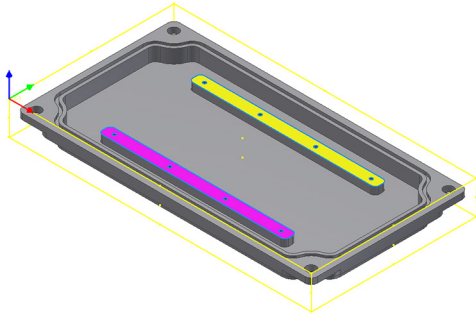
88. Define the Geometry

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.

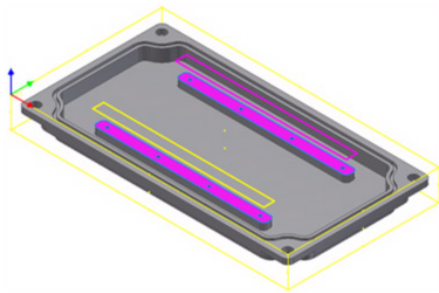
The **Select Faces** dialog box is displayed.

Click on the surfaces of both pads and confirm your selection by clicking .



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

Two separate chains are created.



Confirm the geometry definition by clicking .



89. Define the Tool

Choose **Ø8 End mill** for the operation.



90. 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.




91. Define the technological parameters

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



92. Save and Calculate

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



93. Simulate

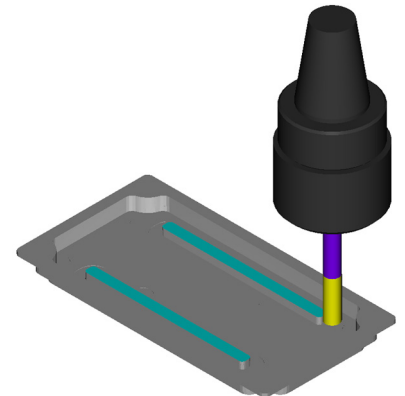
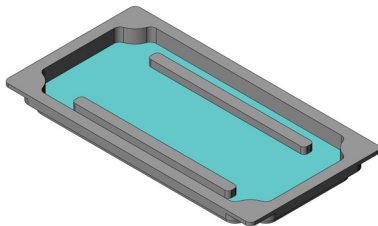
Play the simulation in the **SolidVerify** mode.

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



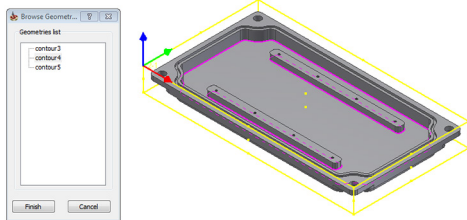
94. Add a Pocket operation

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



95. Define the Geometry

Use the already defined geometry **contour4**.



96. Define the Tool

Choose the **Ø6 End mill** for the operation.



97. 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.




98. 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.



99. Save and Calculate

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



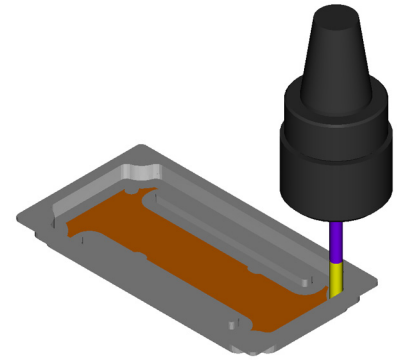
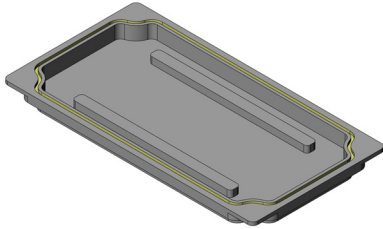
100. Simulate

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



101. Add a Slot operation

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



In **InventorCAM Manager**, right-click the **Operations** header and choose the **Slot** item from the **Add Milling Operation** submenu.

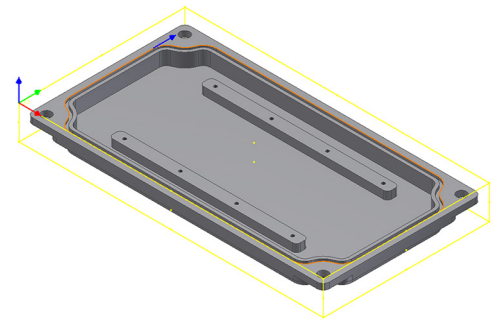
The **Slot Operation** dialog box is displayed.



102. 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.



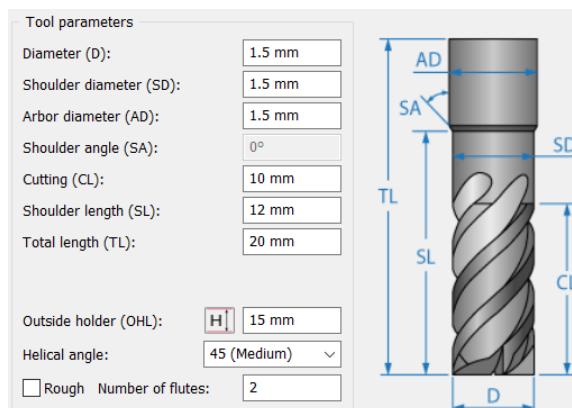
103. Define the Tool

Define a new **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 as the holder.





104. Define the technological parameters

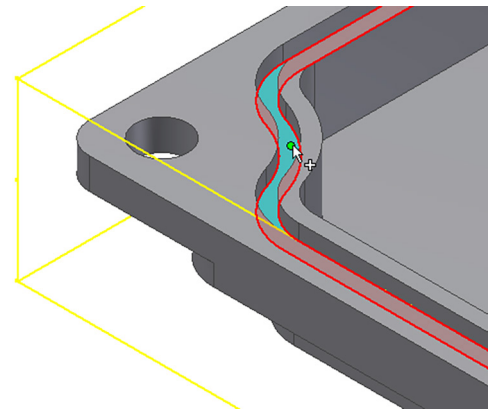
In the **Tool side** area, choose the **Right** option to machine the slot at the right side of the selected geometry.

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**.



105. Save and Calculate

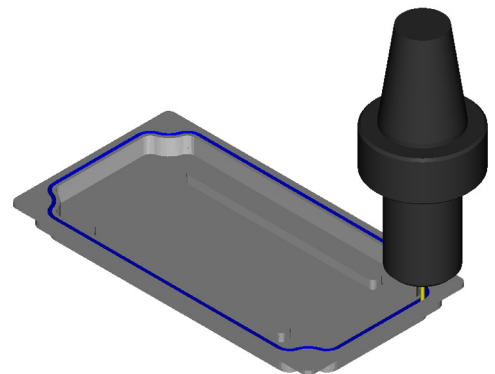
Click . The Slot operation data is saved, and the tool path is calculated.



106. 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.



107. Define a Drilling operation

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



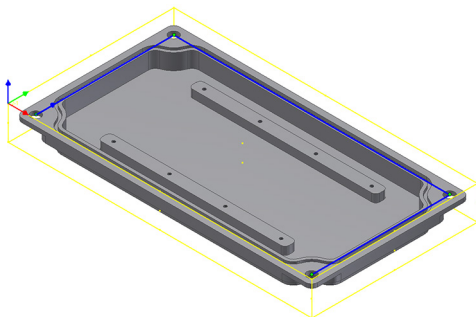
108. Define the Geometry

In the **Geometry** page, choose the **Machine Coordinate System #2** for the operation and click 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.



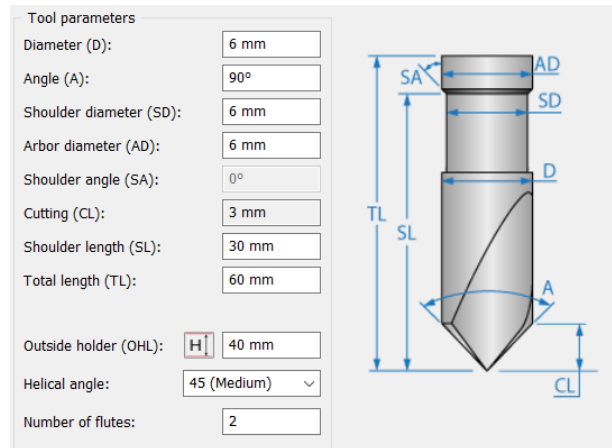
All the circular edges located on the selected face are collected, and the drill positions are determined. Confirm the geometry selection with .




109. Define the Tool

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

Define a new SPOT DRILL tool with the following parameters.



Select **BT40 ER16x70** as the holder.

Click  to confirm the tool for operation.



110. 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.



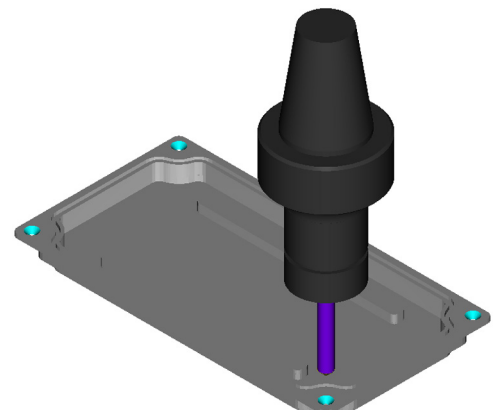
111. Save and Calculate

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



112. Simulate

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





113. Add a Drilling operation

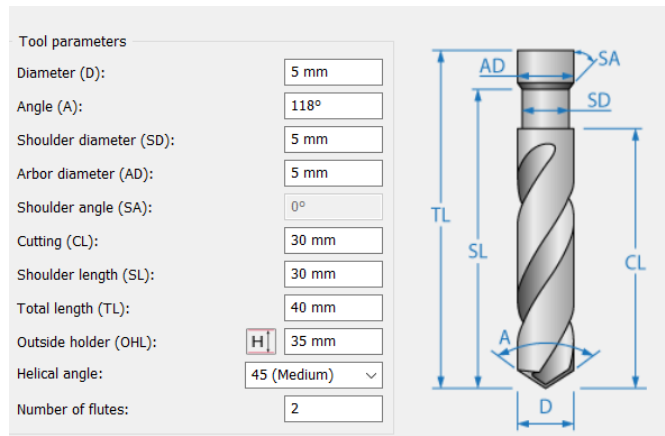
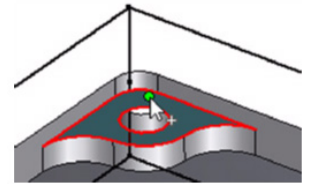
In **InventorCAM Manager**, add a new Drilling operation to drill the holes center-drilled at the previous stage.

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



114. Define the Tool

Define a **Ø5** Drill tool with following parameters for the operation.



Select **BT40 ER 16x70** as the holder.



115. 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** 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.

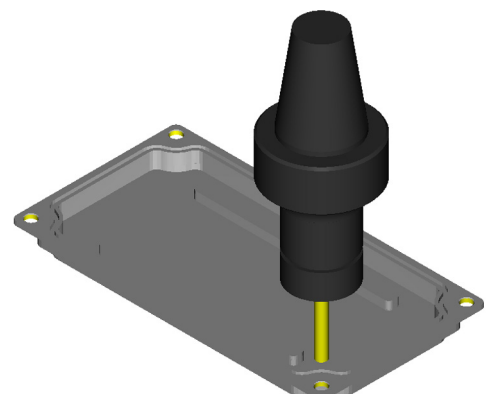


116. Save and Calculate

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

117. 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.



118. Add a Drilling operation

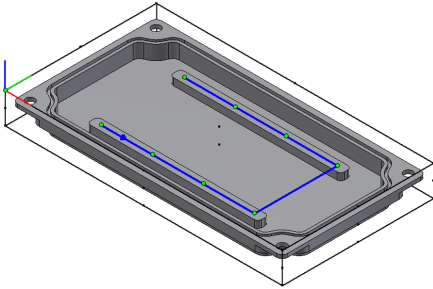
Add a new Drilling operation to perform the center drilling of the pad holes.



119. Define the Geometry

In the same manner as explained in **Step # 108**, define the geometry for the drilling.

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



120. Define the Tool

Use the earlier defined **Ø6 Spot Drill** tool.




121. 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**.



122. Save and Calculate

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



123. Add a Drilling operation

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

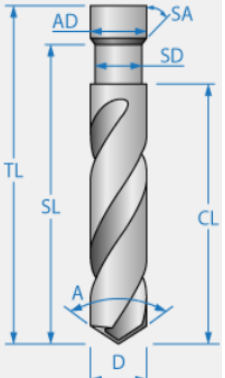
In the **Geometry** area, choose the **drill1** geometry defined in **Step #119**.



124. Define the Tool

Define a **Ø1.6** Drill tool for the operation with following parameters. This tool enables you to perform preliminary drilling for the M2 tapping.

Tool parameters	
Diameter (D):	1.6 mm
Angle (A):	118°
Shoulder diameter (SD):	1.6 mm
Arbor diameter (AD):	1.6 mm
Shoulder angle (SA):	0°
Cutting (CL):	15 mm
Shoulder length (SL):	15 mm
Total length (TL):	25 mm
Outside holder (OHL):	<input type="checkbox"/> 20 mm
Helical angle:	45 (Medium) ▾
Number of flutes:	2



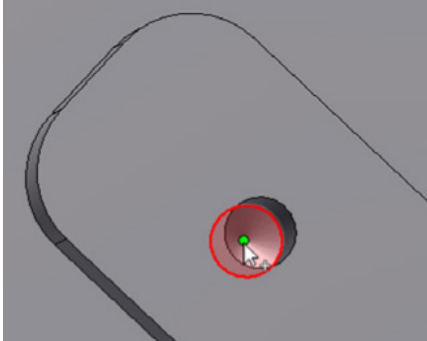
Select the holder as **BT40 ER 16x70**.



125. Define the Milling levels

In the **Levels** page, define the **Upper level** in the same manner as explained in **Step #121**.

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



depth.



126. Define the Drill cycle type

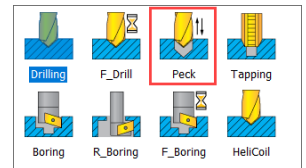
Click the **Drill cycle type** button in the **Technology** page. The Drill Cycles panel is displayed.

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**.



127. Save and Calculate

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

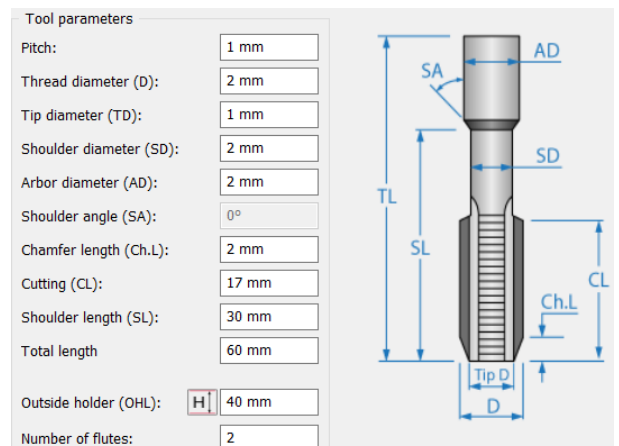


128. Add a Drilling operation

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

In the **Geometry** area, choose the **drill1** geometry defined in **Step #119**.

Add a new **Ø2 Tap** tool for the operation. Select the holder as **BT40 ER 16x70**.

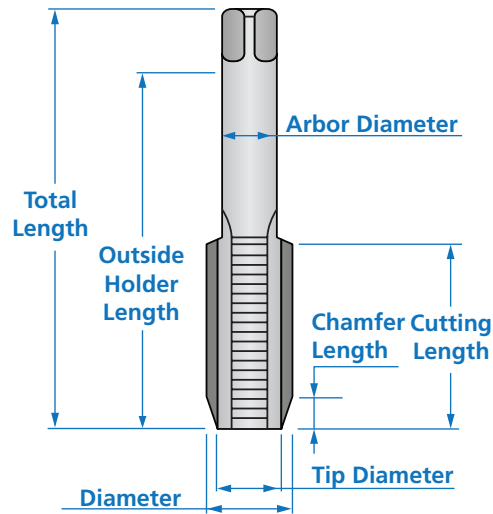




Tap Tool

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.




129. Define the Drilling depth

In the **Levels** page, define the **Upper level** in the same manner as explained in **Step #121**.

Set the **Drill depth** to **4**.



130. Save and Calculate

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

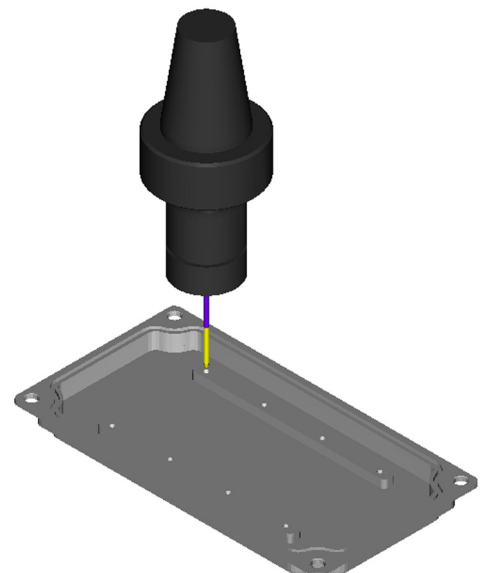


131. Simulate

Click . The **Simulation** control panel is displayed.

InventorCAM enables you to simulate several operations together. Holding the **Shift** key, select the last three operations in **InventorCAM 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.

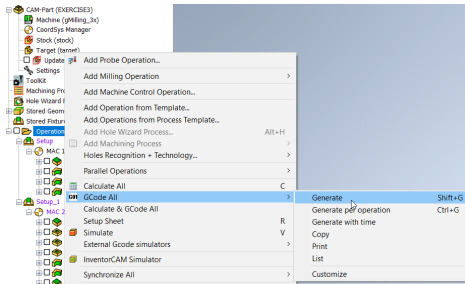




132. Generate the GCode

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

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



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

```

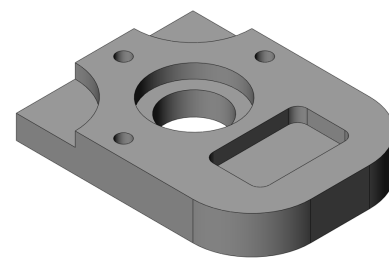
EXERCISE3.TAP - Notepad
File Edit Format View Help
%
:5000 (EXERCISE3.TAP)
(AWEA AV-1000)
G90 G10 L2 P1 X0 Y0 Z0
G90 G17
G54
N10 T1 M6
T2
M01
(*TOOL 1 - DIA 40.0*)
G90 G0 M3 S5000 X180. Y7.
M8
G43 H1 Z50.
M98 P2 (FM-face mill)
N19 M98 P1 (-CHANGE TOOL-)
N20 T2 M6
T3
M01

```

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 Autodesk Inventor model of the Bracket (**Exercise4.ipt**) 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

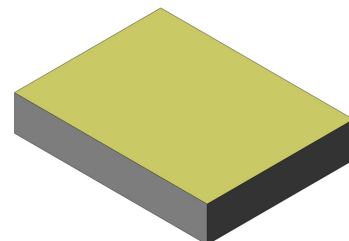
At this stage, you have to define the CAM-Part, CNC-controller, Machine Coordinate System, the Stock and Target models.

The **gMilling_3x** CNC-controller has to be chosen for this exercise.



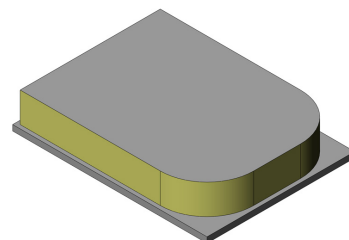
2. Machine the top face

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



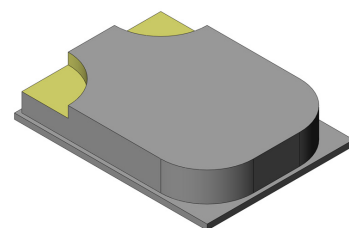
3. Machine the side faces

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



4. Machine the open pockets

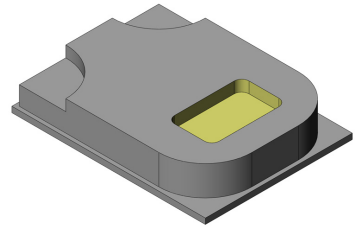
Define a Profile operation to perform the machining of the open areas as shown.





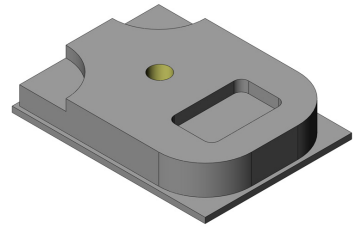
5. Machine the pocket

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

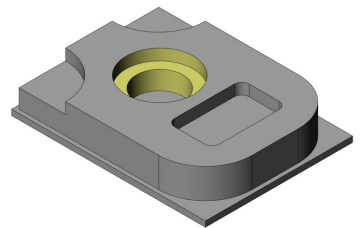
Define two Drilling operations to perform the center drilling and preliminary drilling for the further pocket operation.



7. Machine the cylindrical hole

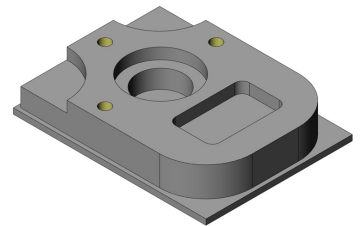
Perform two Pocket operations to machine the faces of the through hole.

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



8. Machine the holes

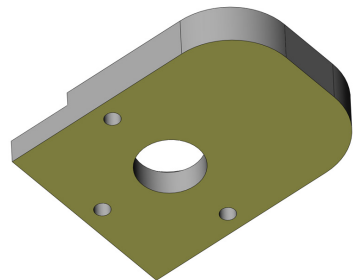
Perform center drilling and drilling of the holes.



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.

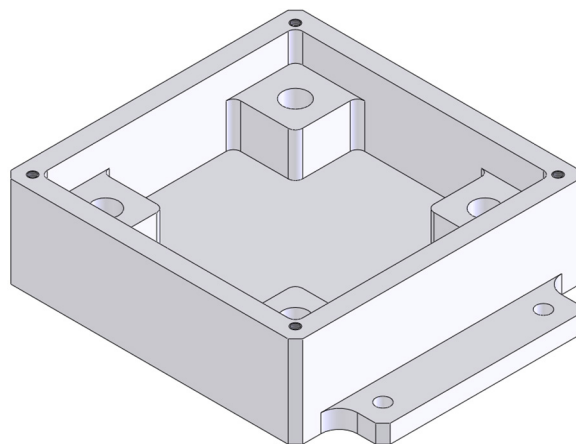


Exercise #5: Electronic Box Machining

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 Autodesk Inventor model of the Electronic Box (**Exercise5.ipt**) is located in the **Exercises** folder.

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



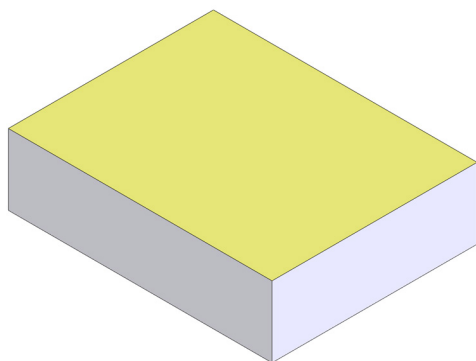
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**gMilling_3x**), Machine Coordinate System, the Stock and Target models.



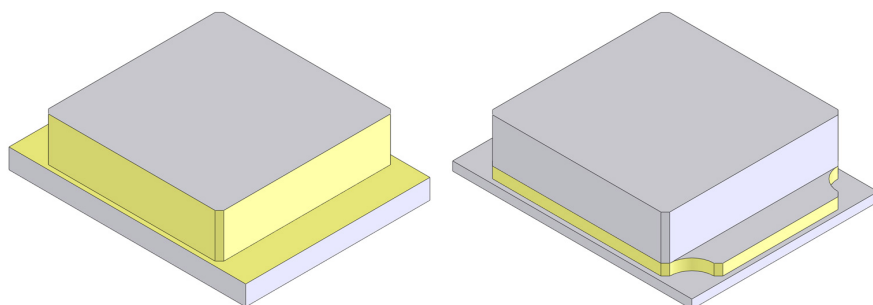
2. Machine the top face

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



3. Machine the side faces

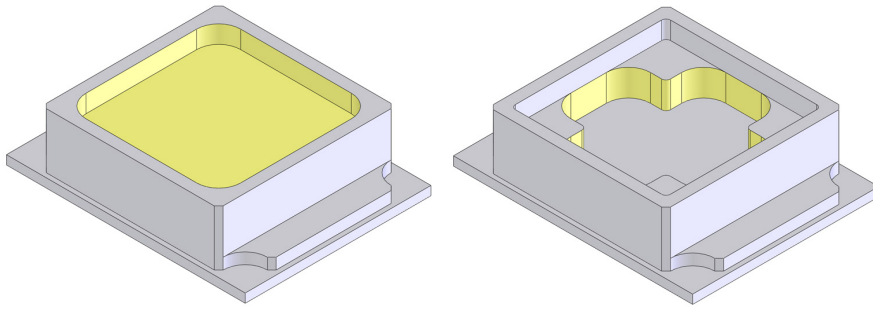
Define two Profile operations to perform rough machining of the external faces.





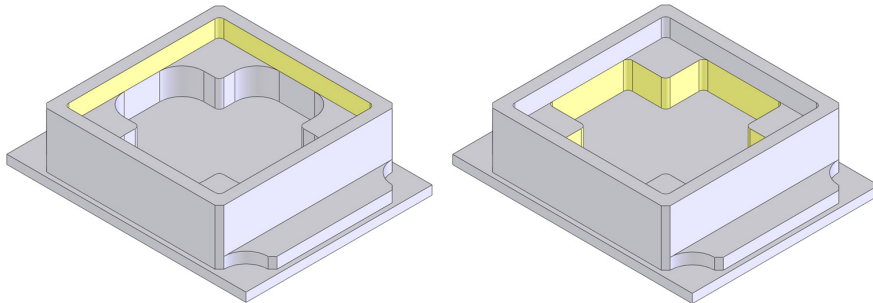
4. Machine the pocket

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

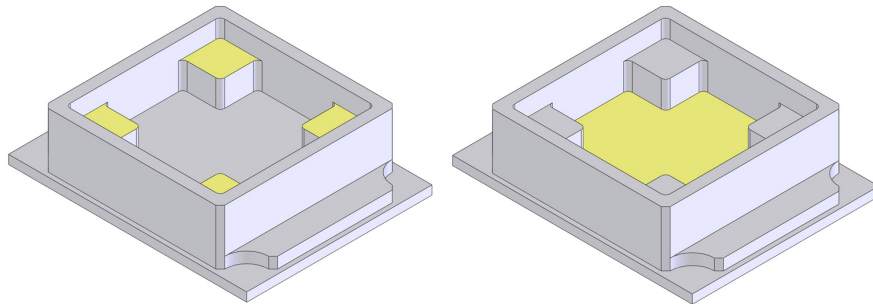


5. Perform the pocket finish machining

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.

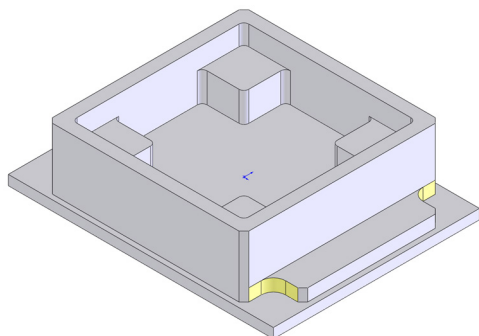


During the wall finish machining, use the **Rest Material** strategy to remove the bulk of material in the corners.

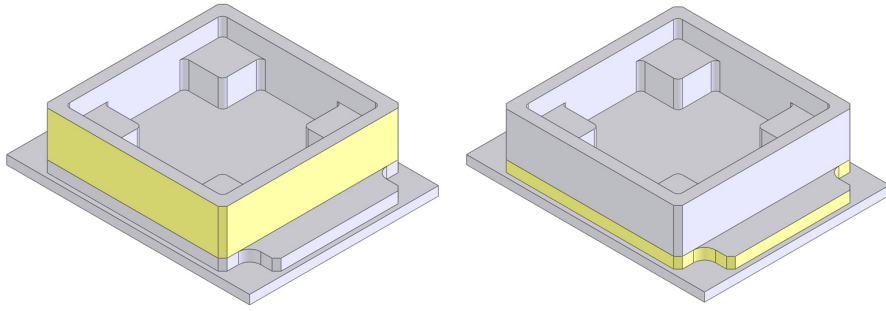


6. Perform finish machining of the external faces

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

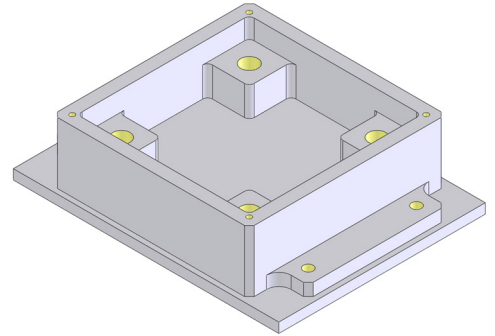


Define a number of Profile operations to machine the external faces.



7. Machine the holes

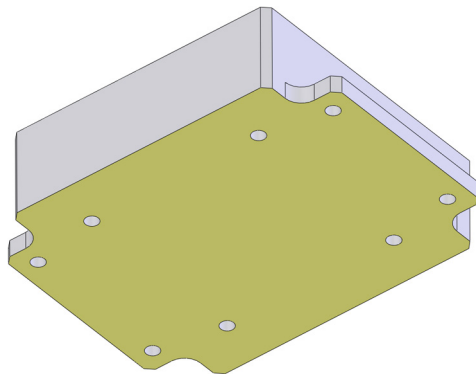
Define a number of Drilling operations to perform the center drilling, drilling, and tapping of the holes. Use a Pocket operation for counterboring of the holes.



8. Machine the bottom face

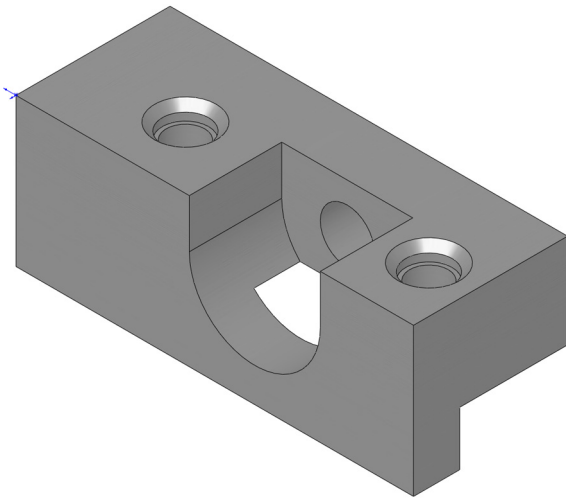
Define a Face Milling operation to machine the bottom face of the electronic box part using the **Hatch** strategy.

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 Autodesk Inventor model of the Clamp (**Exercise6.ipt**) is located in the **Exercises** folder.

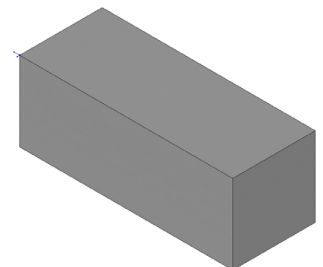
The following steps have to be implemented in order to obtain the final CAM-Part:



1. Define the CAM-Part

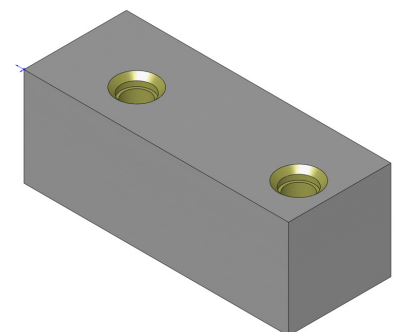
At this stage you have to define the CAM-Part, CNC-controller (**gMilling_3x**), Machine Coordinate System, the Stock and Target models.

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

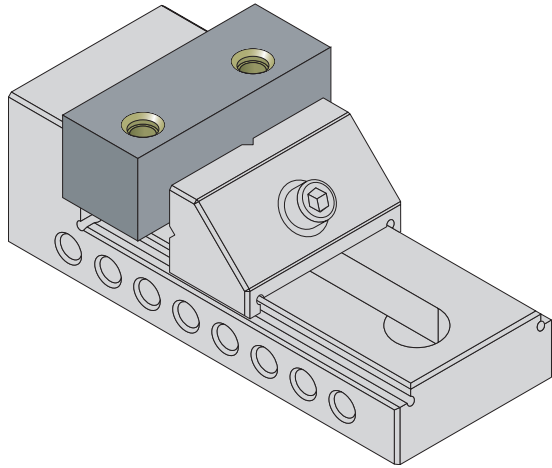


2. Holes machining

Perform a number of Drilling operations to machine the holes.

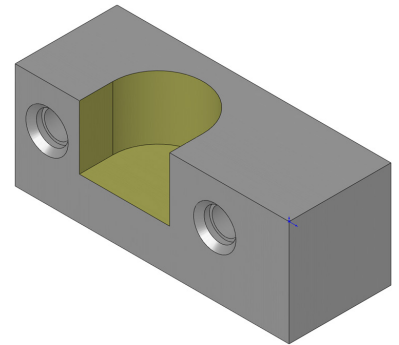
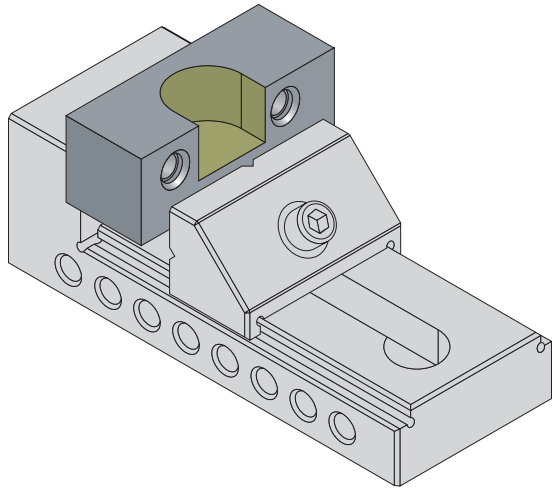


Use the appropriate Machine Coordinate System defined for the positioning shown below.



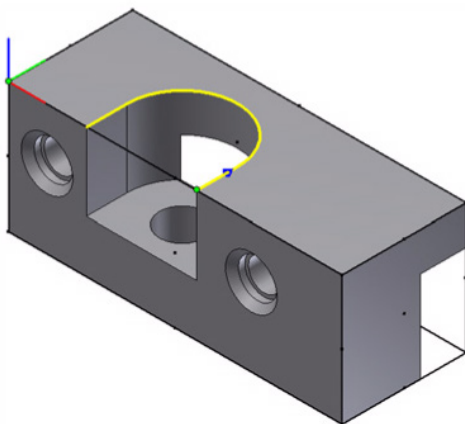
3. Open pocket machining

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, InventorCAM returns to the geometry definition and enables you to close the chain manually by the model element selection.

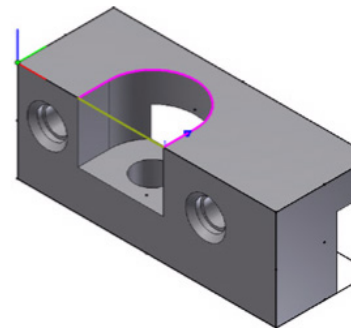
Select the **Mark line as open edge** check box and confirm the dialog box with **OK**.

The open edge is marked.

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

Choose the **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, choose the **Contour** machining strategy. Switch to the **Open pocket** tab to define the parameters of the open pocket machining. The open pocket extension can also be user-defined as tangent angle from the wall. Define the extension of the tool path beyond the open edge.

Select the **Use profile strategy** check box.

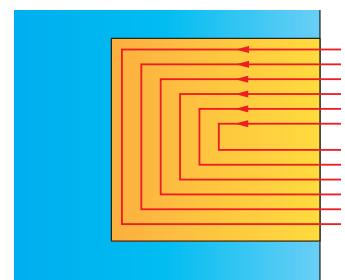
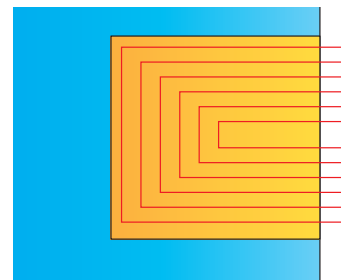


Use Profile Strategy

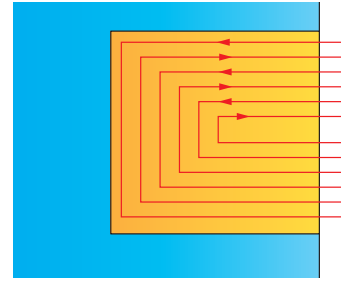
This option enables you to perform the open pocket machining in a Profile manner. The tool path at a specific Z-level consists of several equidistant profiles starting from outside the model at the distance defined by the **Extension** parameter. The tool moves on offsets parallel to the pocket geometry.

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 (**G0**) 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.



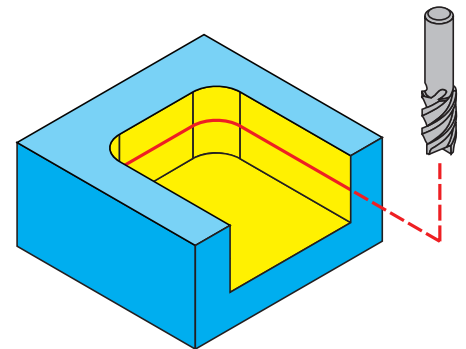
Use the default **One way** option.

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.

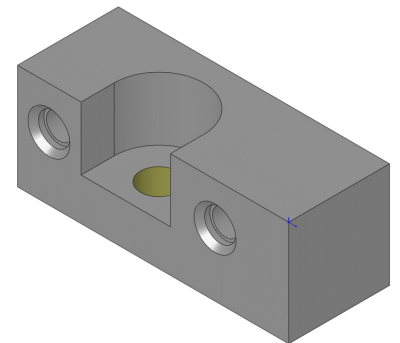


Switch back to the **Technology** tab. In the **Offsets** section, set the **Wall offset** and the **Floor offset** to **0.2**. To remove these offsets with the finishing cut, select the **Wall** and **Floor** check boxes in the **Finish** section.



4. Hole machining

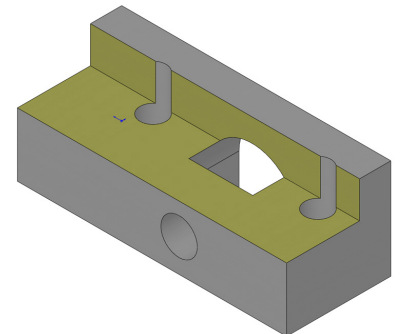
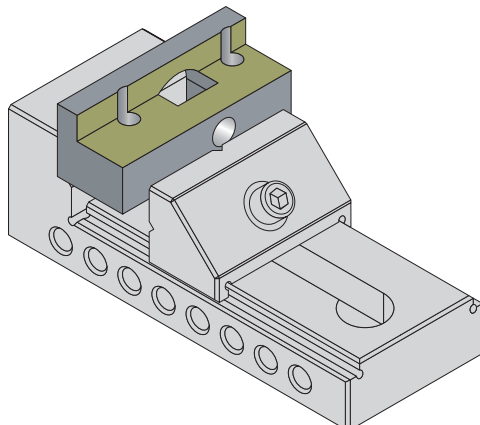
Define two Drilling operations to perform center drilling and drilling of the hole located at the bottom of the open pocket. Use the same Machine Coordinate System (positioning) as in the previous operation.



5. Step machining

Define a Profile operation to machine the step.

Define the Machine Coordinate System for the positioning shown below.



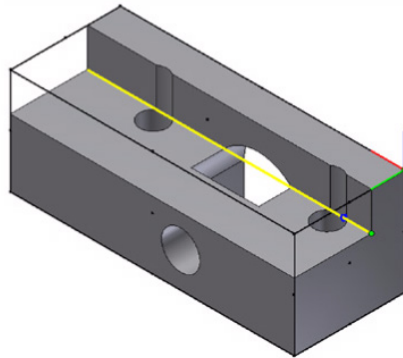
Use the **Point to point** option in the **Geometry Edit** dialog box to define the geometry.



Point to Point mod

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.

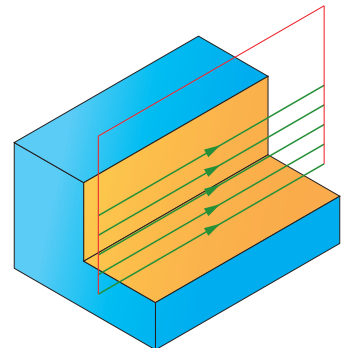


Use the **Zigzag** method for **Depth cutting type** to connect the successive Z-level passes directly from the end of a pass to the beginning of the next pass.

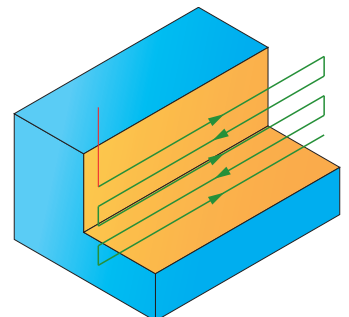


Depth cutting type

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

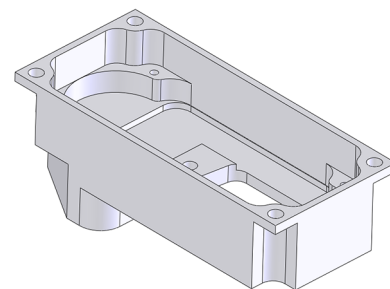
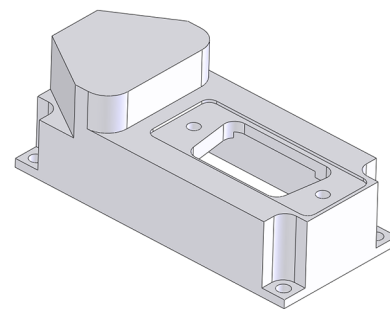
Define the CAM-Part and operations for the machining of the basic part presented on the illustration on a Milling CNC-machine.

This exercise reinforces the following skills:

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

The Autodesk Inventor model of the Basic part (**Exercise7.ipt**) is located in the **Exercises** folder.

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



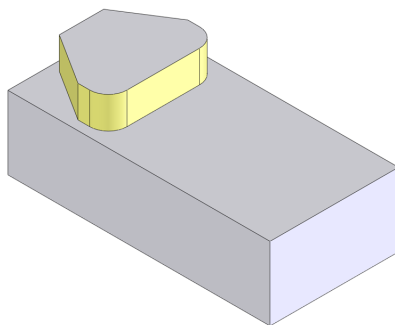
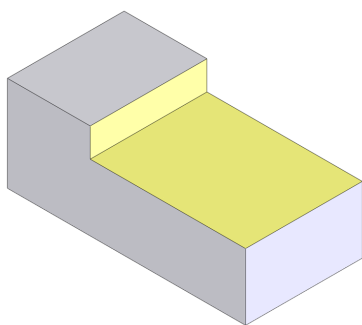
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**gMilling_3x**), Machine Coordinate System, the Stock and Target models.



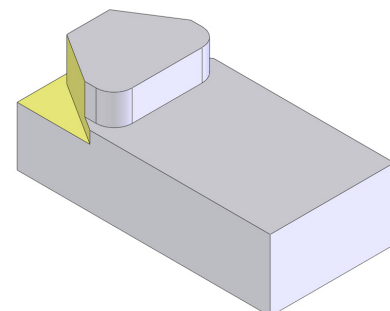
2. Upper cusp machining

Define two Profile operations to remove the bulk of material and perform the rough and the finish machining of the cusp.



3. Step machining

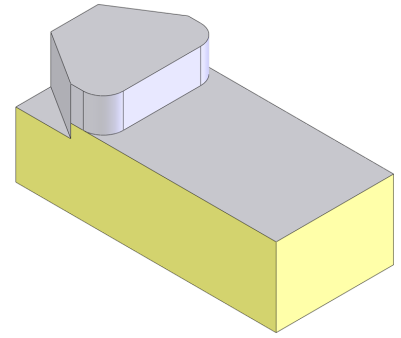
Perform the rough and finish machining of the step faces using the Profile operation.





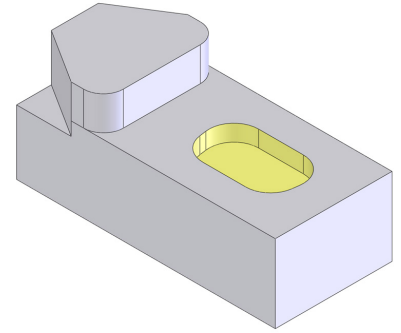
4. External contour machining

Define a Profile operation to perform the rough and finish machining of the external model faces.



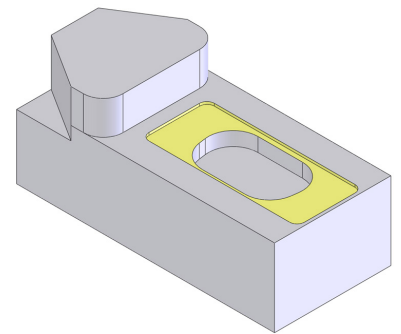
5. Connector opening machining

Define a Pocket operation to perform the rough machining of the connector opening faces.



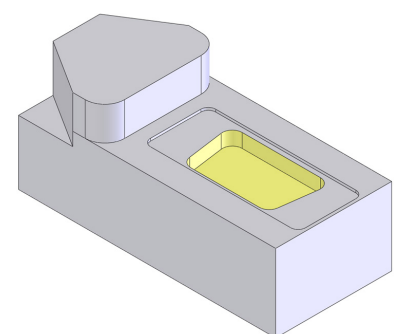
6. Connector pocket machining

Define a number of Pocket and Profile operations to perform the roughing and finishing of the connector pocket.



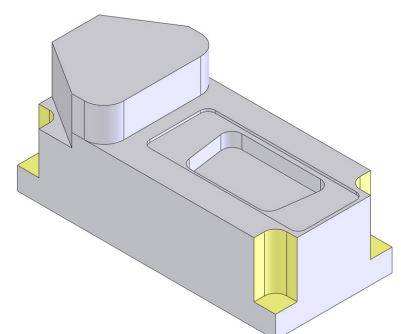
7. Connector opening finish machining

Use the Pocket rest material strategy to perform the finish machining of the connector opening faces.



8. Machine screw head areas

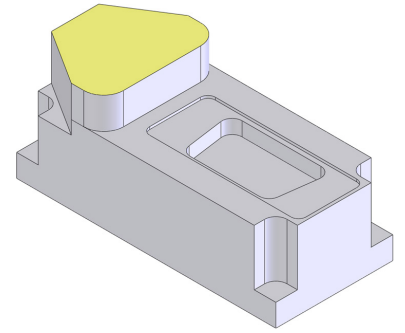
Define a Profile operation to perform the rough and finish machining of the screw head areas.





9. Top face machining

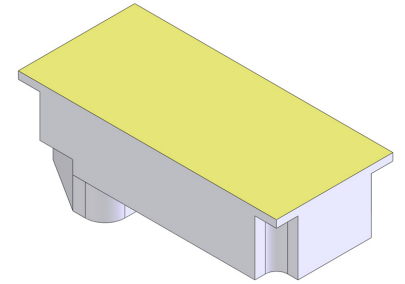
Use the Face milling **Hatch** strategy to perform the rough and finish machining of the top face.



10. Bottom face machining

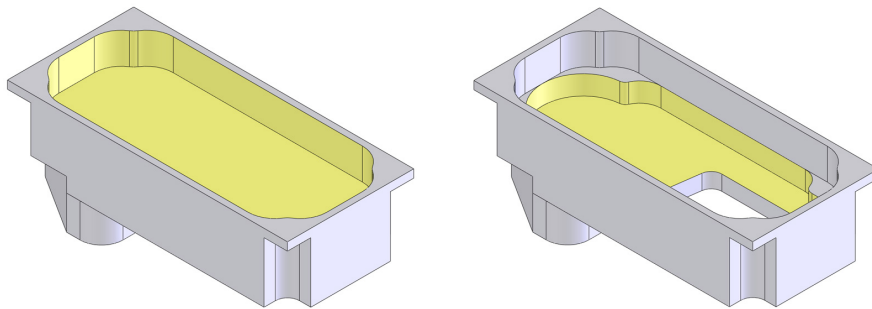
Use the Face milling **Hatch** strategy to perform the rough and finish machining of the bottom face of the Cover part.

Note that this operation and the following ones use a new positioning and therefore you have to define a new Machine Coordinate System.



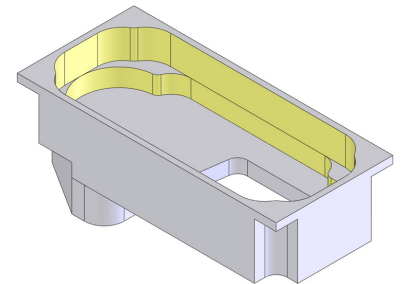
11. Pocket rough machining

Use a Pocket operation to perform the rough machining of the highlighted faces.



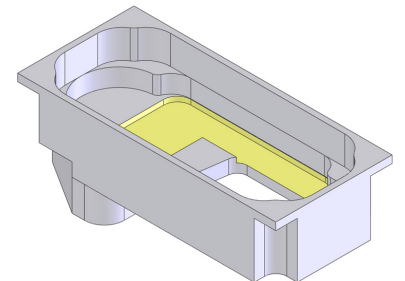
12. Wall finish machining

Define two Profile operations to perform the finish machining of the wall faces of the pockets.



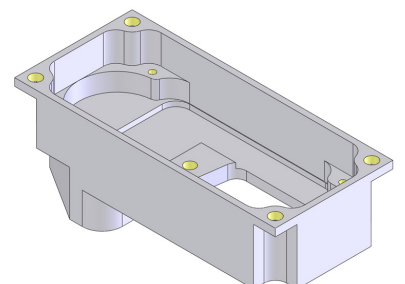
13. Bottom pocket machining

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.

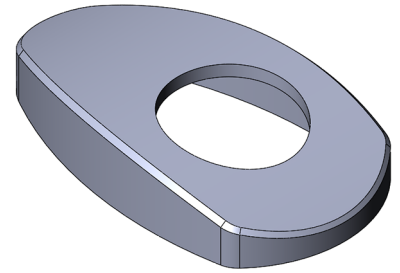


Exercise #8: Cover Machining

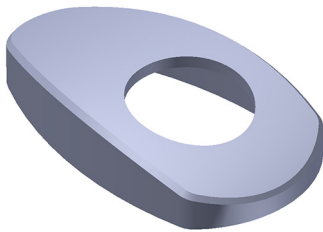
Machine the hole and chamfer on the pre-machined cover part presented on the illustration on a Milling CNC-machine.

This exercise reinforces the following skills:

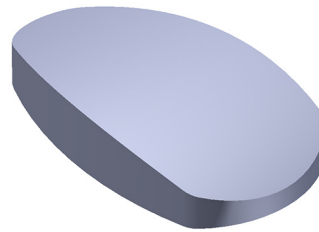
- 3D contour machining



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 iPart members: **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.



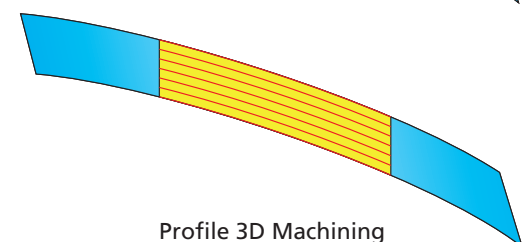
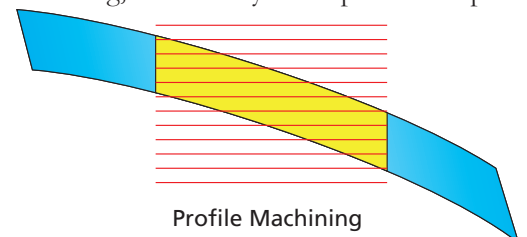
2. Add a Contour 3D operation

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.



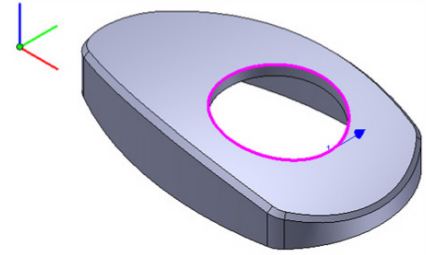
A through hole located on a convex surface can be machined in several methods, such as Profile machining and Contour 3D machining. Profile milling used for machining of such holes results in a great number of tool path passes with unavoidable air cutting, because it yields a planar tool path parallel 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.





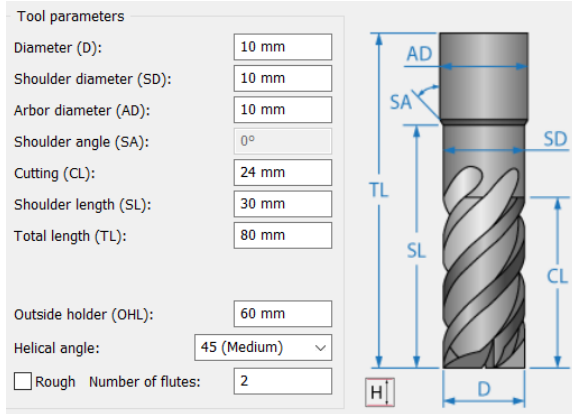
3. Define the Geometry

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

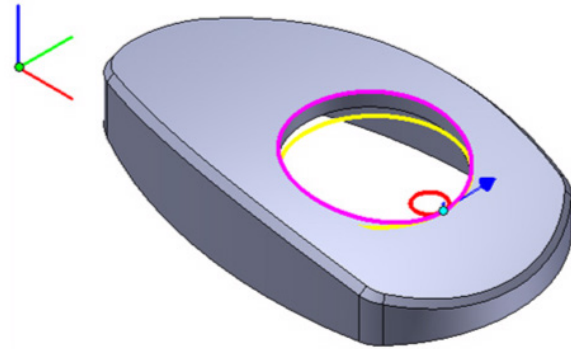
In the **Modify** section, click the **Geometry** button to check the position of the tool relative to the geometry.

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 .

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.

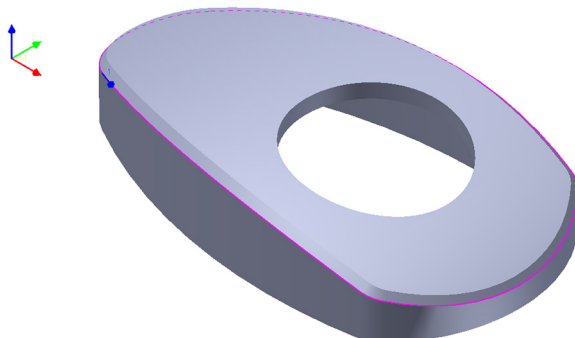


6. Define the Lead in and Lead out

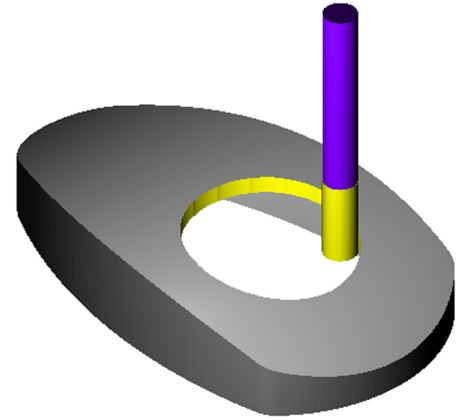
For **Lead in**, choose the **Arc** option and use the default **Radius** value of **2**. For **Lead out**, select the **Same as Lead in** check box.



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.

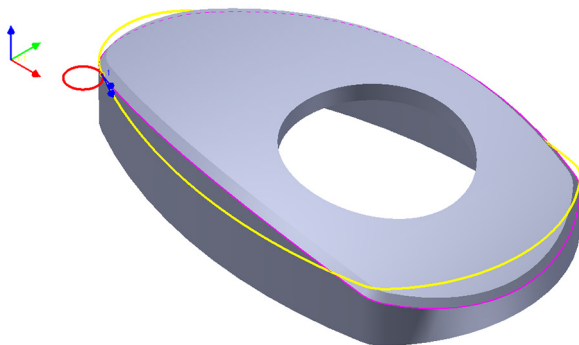


11. Define the Milling levels

In the **Levels** page, set **Depth** to **0**.



12. Define the technological parameters



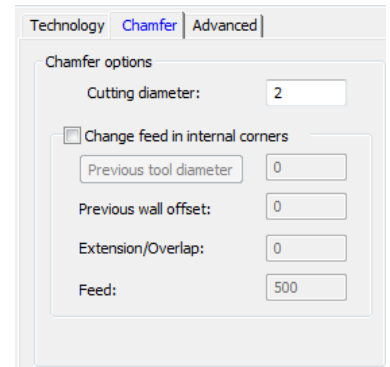
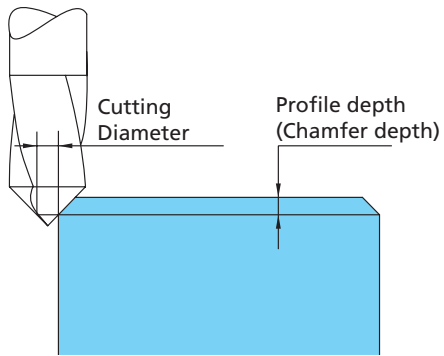
In the **Modify** section of the **Technology** page, set the **Tool side** to **Right**. Click the **Geometry** button to check the tool position relative to the geometry.

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**.



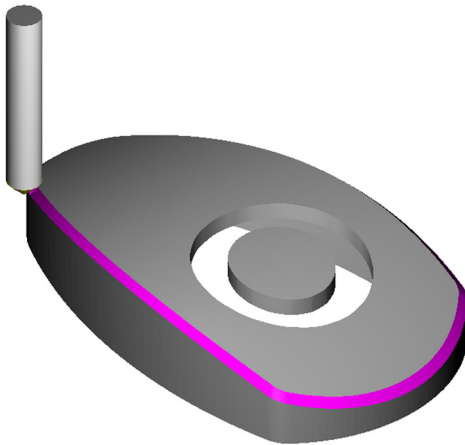
13. Define the Lead in and Lead out

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.

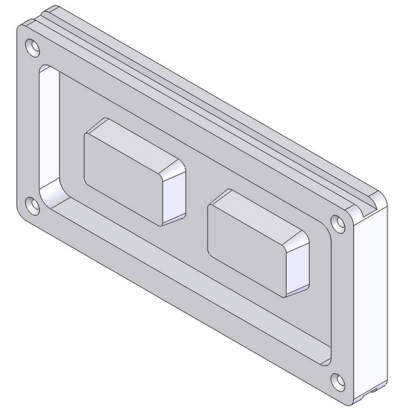


Exercise #9: Mounting Base Machining

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 Autodesk Inventor model of the Mounting base (**Exercise9.ipt**) is located in the **Exercises** folder.

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



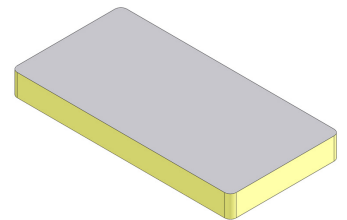
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**gMilling_3x**), Machine Coordinate System, the Stock and Target models.



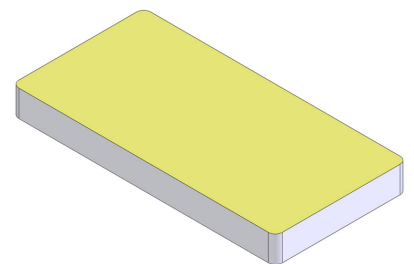
2. Contour machining

Define a Profile operation to machine the mounting base external faces.



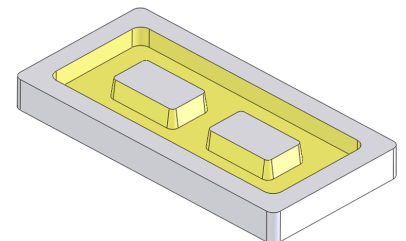
3. Top face machining

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 **Ø8** with **Corner radius** of **0.2** mm for machining of the pocket.



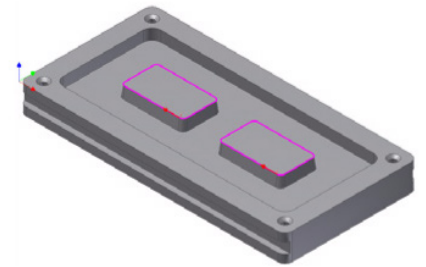
5. Profile machining for islands

Define a Profile operation for two draft angled walls of the islands. Use the **Bull nose tool** of $\varnothing 8$ already defined in **Step #4**.

Define the geometry as shown.

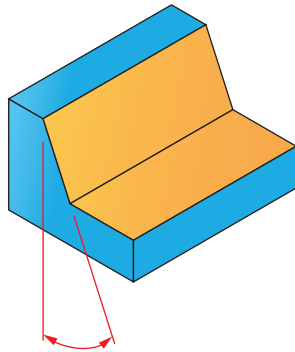
Select the **Finish** option in the **Technology** page and set the **Step down** value to **0.1**.

Switch to the **Advanced** tab and click the **Wall draft angle** button.



Wall Draft Angle

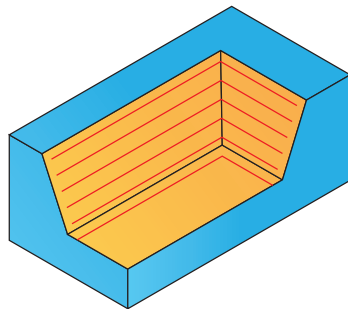
When this check box is selected, the inclined wall machining is performed.



- **External wall angle**

This parameter defines the draft angle of the wall measured from the Z-axis direction as shown.

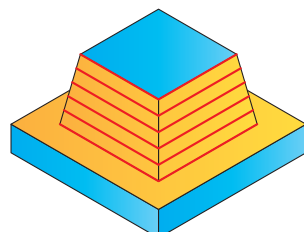
For the inclined wall machining, each cutting pass located at a specific Z-level is generated according to the specified **External wall angle** value.



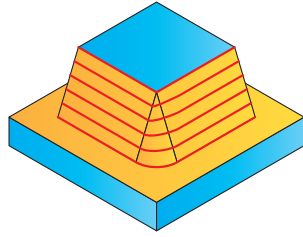
- **External corner type**

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:

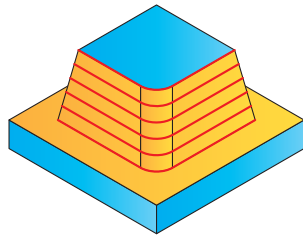
Sharp corner: the tool path is calculated to perform the machining of a sharp corner. This option affects only the machining of geometry sharp corners.



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.



In the new dialog box, check the **Wall draft angle** option.

Set the **External wall angle** to **8**.

In the **Cut from** area, select the **Bottom to top** option.

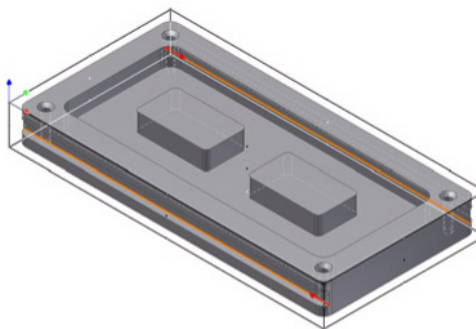
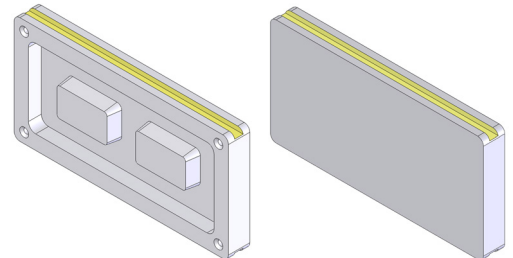


6. T-Slot machining

In **InventorCAM Manager**, right-click the **Operations** header and choose **T-Slot Operation** from the **Add Milling Operation** submenu.

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.



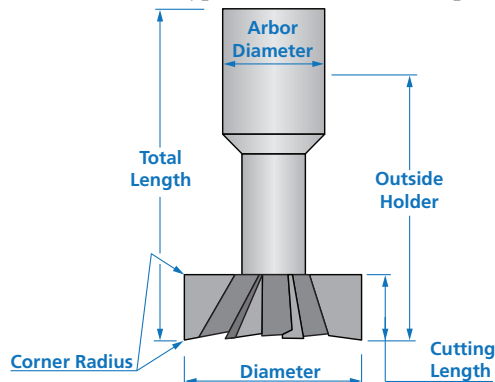
Add a new **Slot mill** tool.



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.

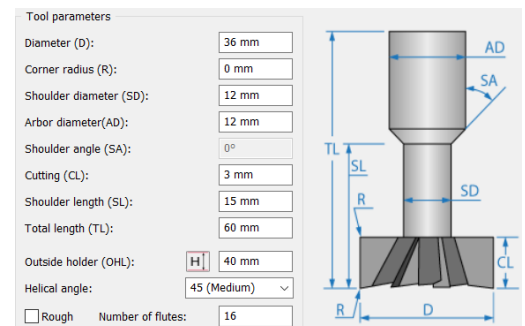
A tool of this type is defined with the parameters shown in the image.




Edit the tool parameters as follows:

- Set **Diameter** to **36**
- Set **Corner radius** to **0**
- Set **Cutting length** to **3**

Add the **BT40 ER32x80** tool holder.



Confirm the tool definition by clicking .

In the **Tool** page, click the **Data** tab.

Select the **Offsets** option and the **Top tool offset number** check box.

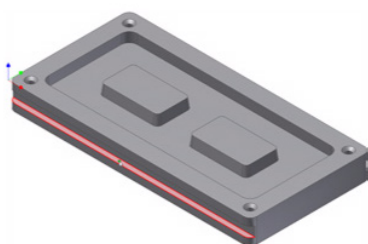


The **Length offset number** defines the length offset register of the current tool in the Offset table of the CNC-machine. The **Top tool offset number** defines the register number of the upper cutting edge of the T-slot mill. This option enables InventorCAM to automatically take into account the minor size differences between the defined tool and the one actually used for cutting the workpiece, if there are any.



7. Define the Milling levels

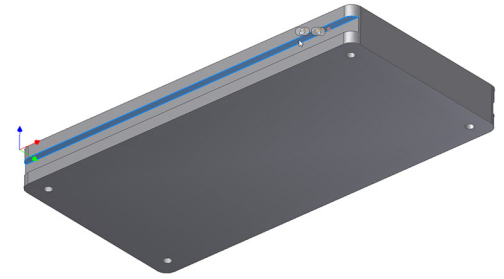
Define the **Ceiling (Upper level)** for slot machining by clicking on the model face as shown.



Confirm the value (**-4.5**) in the **Pick Upper level** dialog box.

Define the **Depth** of the T-slot by clicking on the model as shown.

The depth value calculated in the **Depth** area is **6**.



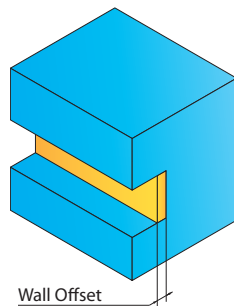
8. Define the technological parameters

Define the rough parameters in the **Technology** page. To perform roughing of the slot, select the **Rough** check box.

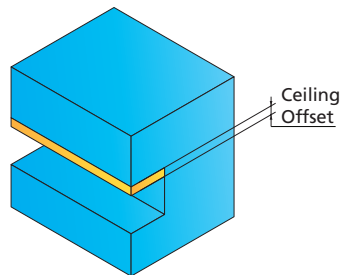
To perform the rough machining of the slot, you have to specify offsets on the slot geometry. These offsets are removed with the finishing cut.



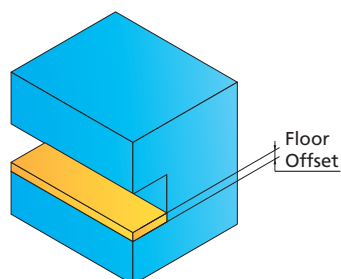
The **Wall offset** option defines the offset that remains on the slot wall after roughing.



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 offsets to **0.2**

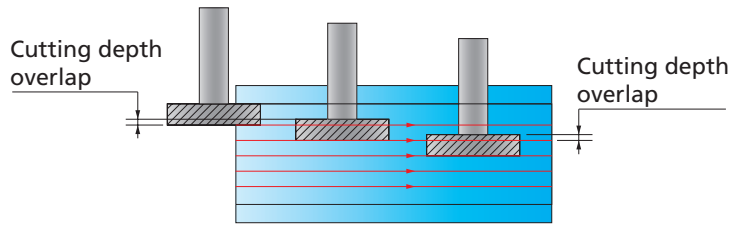
Define the finishing parameters.

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**.



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.



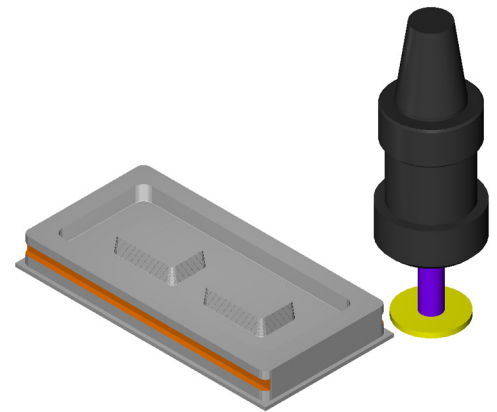
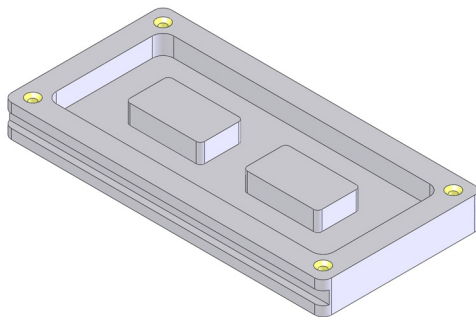
10. Simulate the operation

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

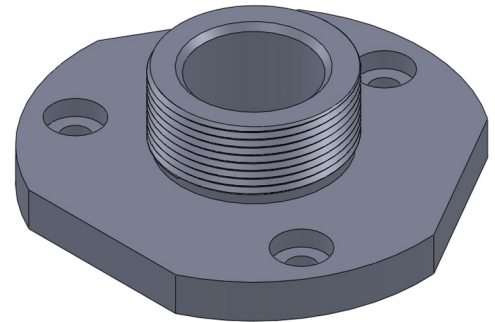
Machine the threads on the pre-machined support part presented on the illustration on a Milling CNC-machine.

This exercise reinforces the following skills:

- Threading
- Creating shaped tools

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.

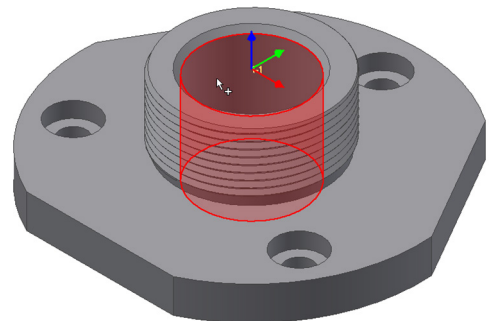


3. Define the Geometry

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.

⊗ 1 0.000; 0.000



4. Define the Tool

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

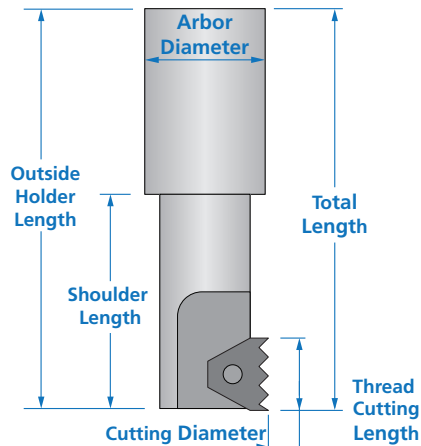
The available tools are displayed in the TOOLKIT.

Choose the **Thread Mill** tool for the operation.

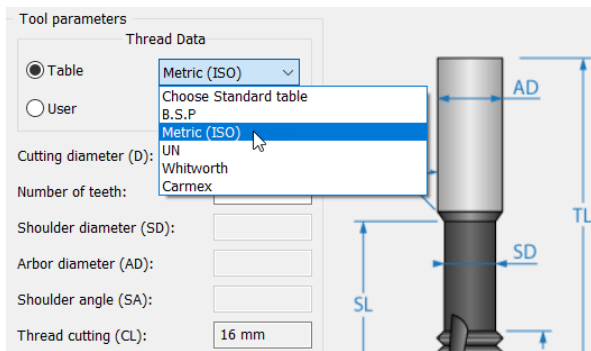


Thread Mill

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.

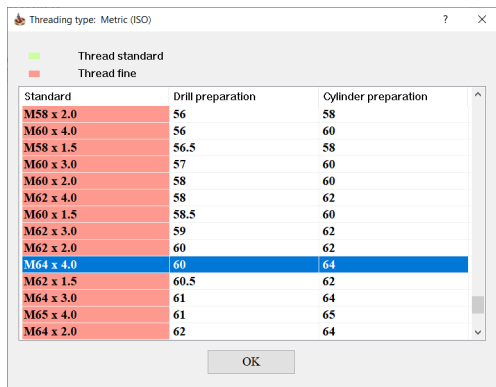


In the **Tool Parameters** section, select the **Metric (ISO)** option from the list.




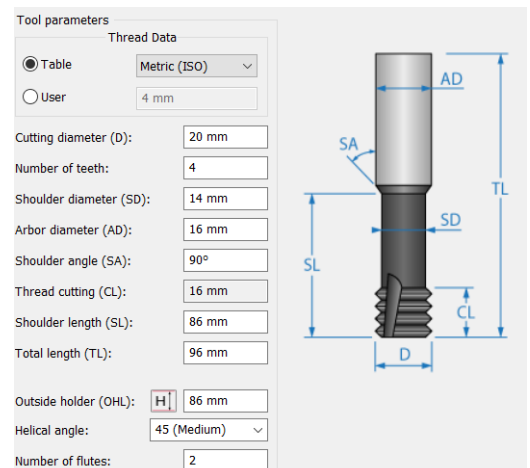
The **Threading Type: Metric (ISO)** dialog box is displayed.

Select the **M64 x 4.0** standard from the list and click **OK**.



Define the tool parameters as shown in the image.

Click  to select the tool for the operation.



The tool is displayed in the illustration area. This picture is generic; it does not reflect the specified parameters.

You can edit its appearance by adding a shaped tool to the Tool Library.

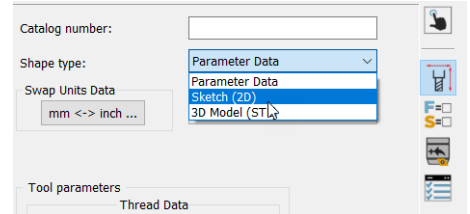
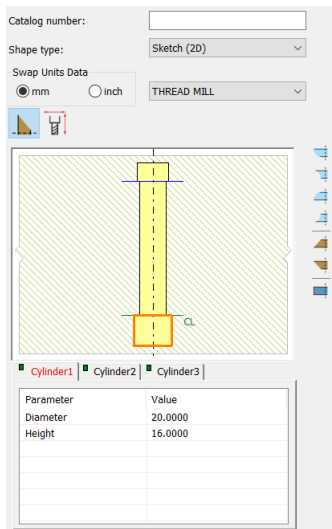
Click the **Select** button.



5. Add a Shaped tool

In the **Shape Type** list, select **Sketch (2D)**.

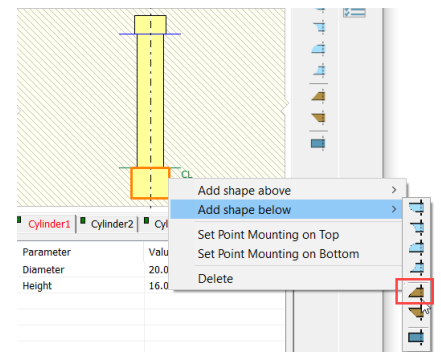
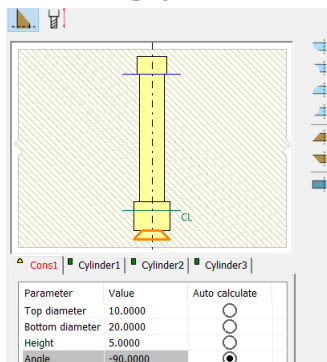
The editing box for shaped tools is displayed.



You can see in the above image that three cylinders are already defined by default. You must add some cone shapes below cylinder 1.

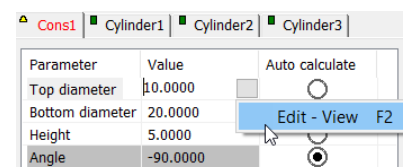
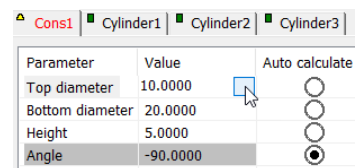
Right click **Cylinder 1** > **Add shape below** and select the cone shape as shown in the image.

The **Cons1** page is added.

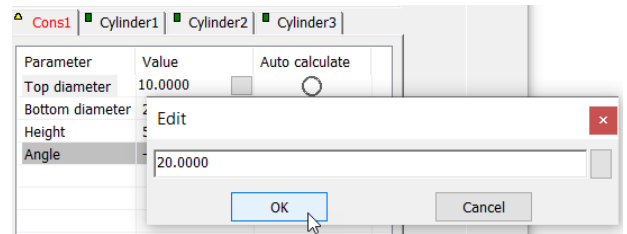


Edit the cone dimensions as follows:

- Select **Top diameter** and click the small box that highlights
- Click **Edit**.

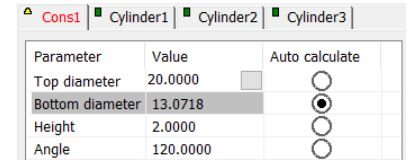


- Set the **Top diameter** to **20** and click **OK**.
- Choose the **Auto-calculate** option for the **Bottom diameter**
- Set the **Height** value to **2** in the same manner you edited the top diameter value.
- Set the **Angle** to **120** in the same manner you edited the top diameter value.



The **Bottom diameter** is calculated automatically and set to **13.0718**.

Right-click **Cons1** > **Add shape above** and select the same cone shape you used for adding the first cone. Edit its dimensions as follows:



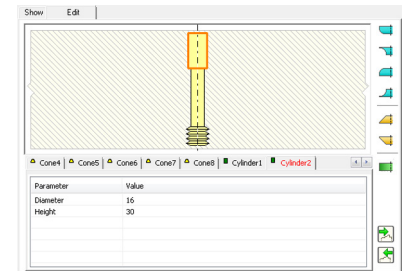
- Choose the **Auto-calculate** option for the **Top diameter**
- Set the **Bottom diameter** to **20**
- Set the **Height** value to **2**
- Set the **Angle** to **-120**

Repeat these steps six more times until you have defined total eight cones. For cone 3, 5, and 7 use the same parameters as were defined for cone 1. for cone 4, 6, and 8 use the parameters that were defined for cone 2.

In the same manner as described above, edit the values of Cylinder1 and Cylinder2 with the following parameters:

- For Cylinder1, set the **Diameter** to **12** and the **Height** to **50**
- For Cylinder2, set the **Diameter** to **16** and the **Height** to **30**
- Right click Cylinder3 and delete it as it is not required

The final tool shape is as shown on the picture.



Confirm the new shaped tool by clicking 

The new tool shape is now displayed in the tool picture section and visible during simulation.

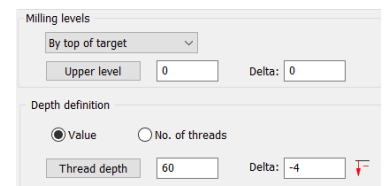


6. Define the Milling levels

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.

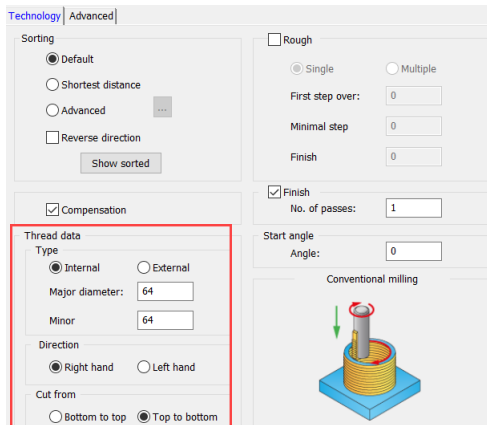
In the **Depth definition** area, set the **Delta** value to **-4**.





7. 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**. When the **Compensation** check box is selected, the tool radius compensation options G4x of the CNC-controller are used in the GCode.



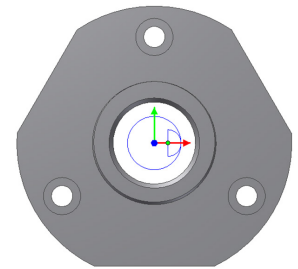
8. Define the Lead in

Switch to the **Link** page. Under **Lead in/out arc**, clear the **Start from the center** check box.

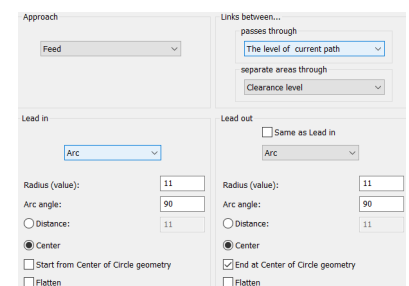


Start from the center

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.



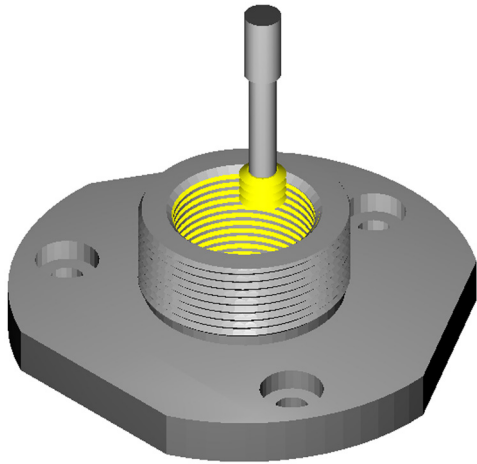
Set other parameters as shown in the image.



9. Simulate the operation

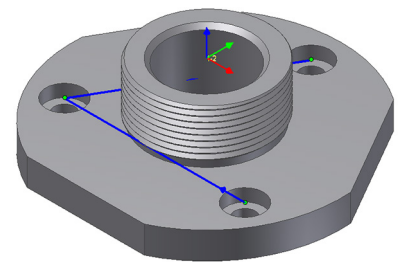
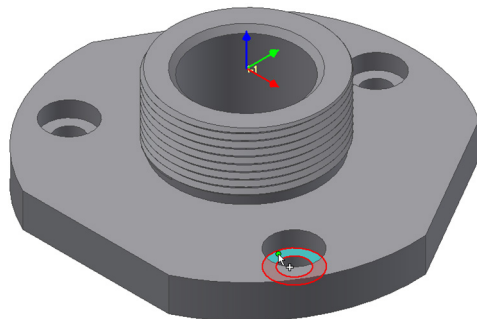
Perform the simulation of the operation in the **Host CAD** mode. Set the CAM-Part in the **Top** view to observe that the tool approaches the material not in the center.

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



10. Add a Thread Milling Operation

Add a new Thread Milling operation to perform small internal threads.



11. Define the Geometry

Define the geometry by clicking on the upper surface of the model containing the holes.



12. Define the Tool

Define a new **Thread mill** selecting the **M14x1.5** standard from the **Metric (ISO)** table.



13. Define the Milling levels

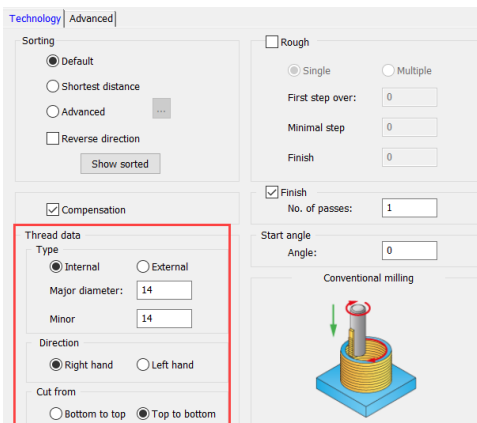
Define the operation **Upper level** directly on the solid model by clicking on the small pocket surface as shown.

Define the **Thread depth** by clicking on the model bottom face.



14. Define the technological parameters

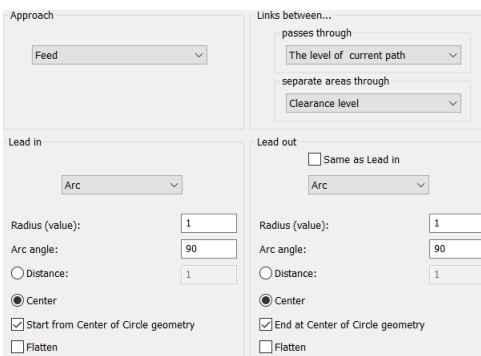
In the **Technology** page, make sure the default **Internal** type is chosen in the **Thread data** area. Set the **Major thread diameter** value to **14**. In the **Cut from** section, select **Top to bottom**.



15. Define the Lead in

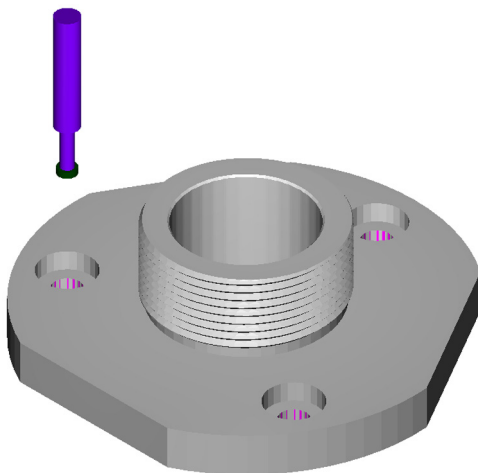
Switch to the **Link** page. Under **Lead in/out arc**, make sure that the **Start from the center** check box is selected.

Set the other parameters as shown in the image.



16. Simulate the operation

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



17. Add a Thread Milling operation

Add a new Thread Milling operation to perform threading of the external surface of the part.



18. Define the Geometry

Use the **drill** geometry defined in the first operation.



19. Define the Tool

Define a new **Thread mill**.

In the **Tool parameters** area, select **User** and set **4** as the value.

- Set the **Cutting diameter** to **20**
- Set the **Number of teeth** to **4**
- Set the **Arbor diameter** to **16**
- Set the **Total length**, **Outside holder**, and **Shoulder length** to **86**
- Set the **Thread cutting length** to **16**

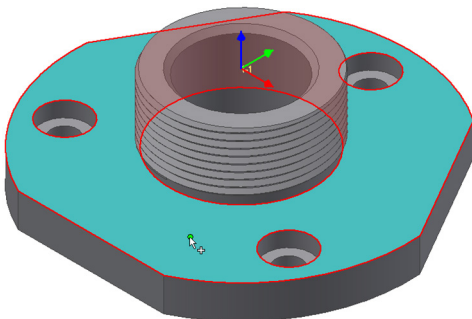
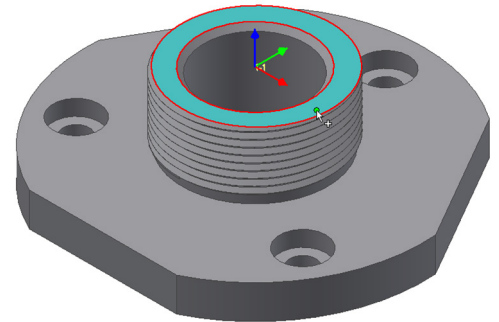
Add a new shaped tool with a shorter shaft, defining the **Cylinder1** height as **40 mm**.



20. Define the Milling levels

Define the operation **Uppr level** directly on the solid model by clicking on the top face of the part as shown.

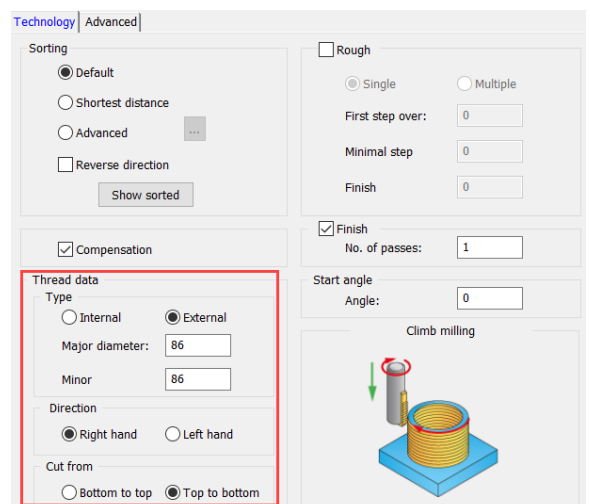
Define the **Thrad depth** by clicking on the model surface as shown below.




21. Define the technological parameters


In the **Technology** page, choose the **External** type in the **Thread data** area. Set the **Nominal** value to **86**.


In the **Cut from** area, choose the **Top to bottom** option.




 **22. Define the Lead in**

Switch to the **Link** page. Set the parameters as shown in the image. Under **Lead in/out arc**, set the **Arc radius** value to **2**.

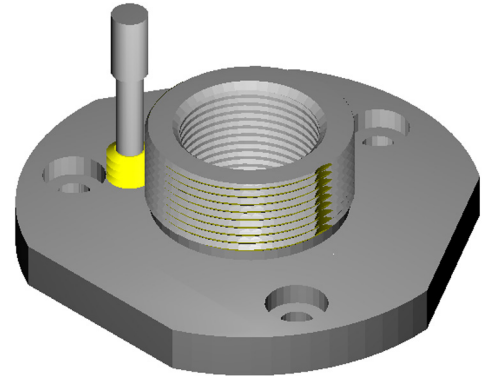
 **23. Save and Calculate**

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

 **24. Simulate the operation**

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

At this stage, the exercise is completed.



Exercise #11: Engraving

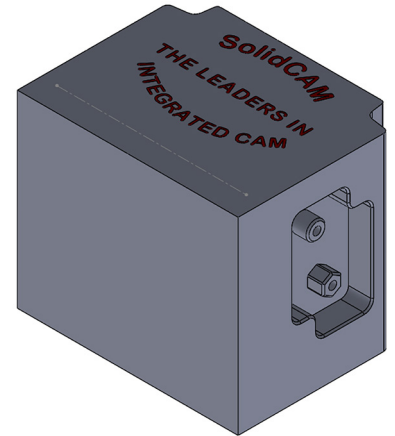
This exercise demonstrates the use of engraving with the text carved on the solid model or typed in using a computer keyboard.

This exercise reinforces the following skill:

- Engraving

The part file (**Exercise11.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 text engraving:



1. Open the CAM-Part

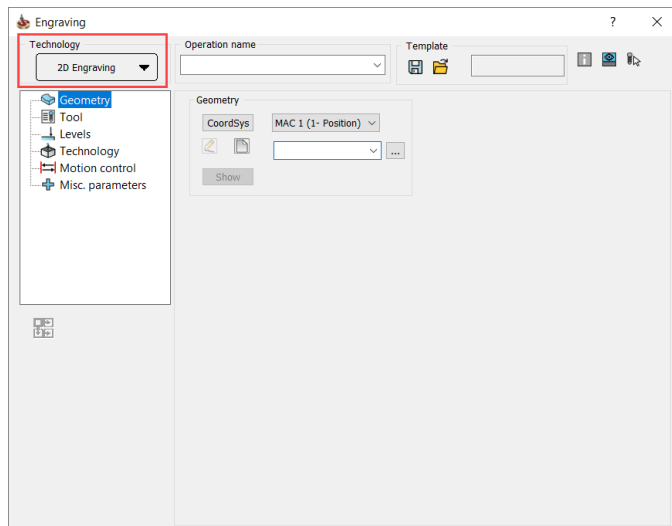
Open the **Exercise11.prz** file located in the **Exercises** folder.



2. Add an Engraving Operation

Add a new Engraving operation to perform machining of the inscription on the upper face of the part.

The **Engraving Operation** dialog box is displayed. Make sure that the default **2D Engraving** technology is selected.



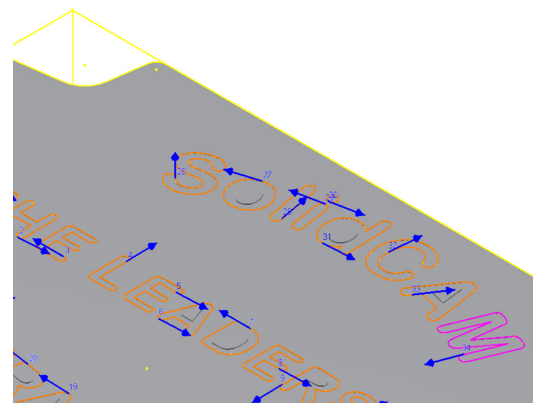
3. Define the Geometry

The geometry for this operation is a set of closed chains comprising the letters.

Click  in the **Geometry** page. In the **Geometry Edit** dialog box, ensure that **104** is selected from the **Configurations** list. Click the **Add** button in the **Multi-chain** section. The **Chains Selection** dialog box is displayed.

In the **Type** section, choose the **Faces** option. All chains found on the selected face will be added to the geometry. To exclude from the geometry the external chain formed by the face edges, apply a filter. Choose the **Loops filter** option. Clear the check box near the **External loops** option. Now only letters will be included in the geometry.

Click on the face that contains the letter. The face is highlighted.

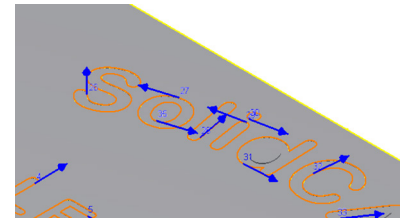


Click  to choose the selected chains as the geometry. The **Geometry Edit** dialog box is displayed.

Zoom in the selected face. Some letters consist of two chains, internal and external, and the internal chains are not selected automatically.

In the **Geometry Edit** dialog box, select the **Constant Z Propagation** option from the **Chain** section. Click on the internal chain of the letter “o”. Accept closing the open chain. The internal chain is added to the geometry.

Select again the **Constant Z Propagation** option from the **Chain** section. Click on the internal chain of the letter “d”. Accept closing the open chain. Repeat the same steps with all letters that include internal chains: “o,” “d,” “A,” “D,” and “R.”



Confirm the geometry definition with .



4. Define the Tool

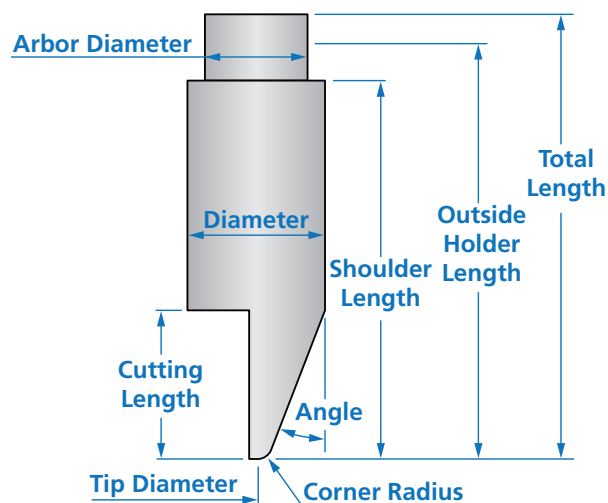
Switch to the **Tool** page of the **Engraving Operation** dialog box and click the **Select** button.

Choose the **Engraving** tool for the operation.



Engraving tool

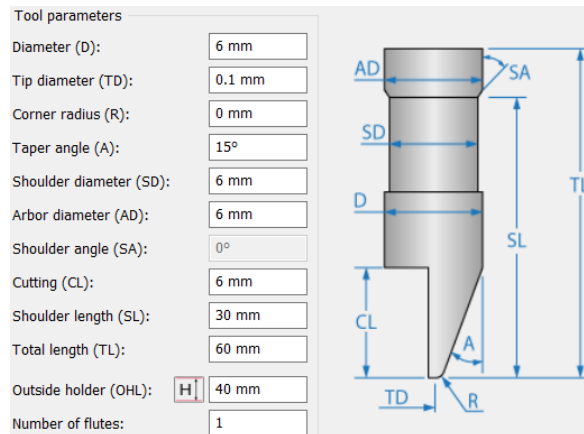
This tool type is used for engraving. A tool of this type is defined with the parameters shown in the image.



An **Engraving tool** of $\text{Ø}6$ will be used.

Edit the following tool parameters:

- Set the **Tip diameter** to **0.1**
- Set the **Taper angle** to **15**



Click  to edit the tool data:

- Set the **Feed XY** to **1000**
- Set the **Feed Z** to **300**
- Set the **Spin rate** to **5000**



5. Define the Milling levels

Define the **Upper level** directly on the part surface. Define the **Engraving depth** on the floor of the engraved text.



6. Define the technological parameters

Switch to the **Technology** page.

In the **Modify** section, choose **Left** in the **Tool side** list.

Select the **Fill area** option and choose **Contour** from the list.



Fill area

Hatch

When this option is chosen, the tool moves inside the closed contour in linear movements.

Contour

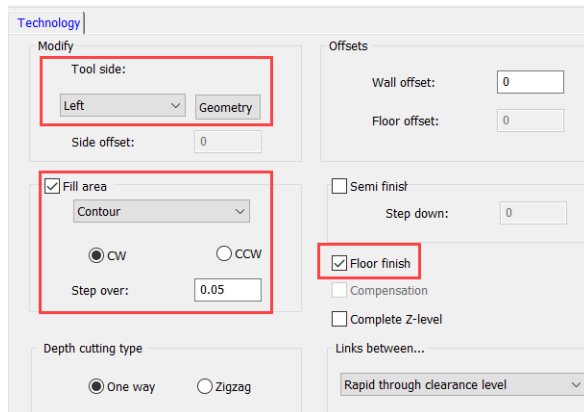
When this option is chosen, the tool moves on offsets parallel to the geometry contour. The distance between two consecutive cuts is defined by the **Step over** value.

Middle line only


When this option is chosen, the tool moves along the center line of the closed contour. When this option is selected, the **Tool side** is always set as **Center**. Selecting the **V-Carving** check box, creates a 3D tool path from 2D fonts. This also sets the **Min. contact angle** to 60. Set the **Min. contact angle** to 60 or less for improved results. When **V-Carving** check box is enabled, the **Engraving depth** on the **Levels** page is set to 0 and locked from editing because the depth varies. The varying depth, which is the result of the operation maintaining the font appearance on the engraving surface, is determined automatically by the tool tip geometry and width of the font.

Set the **Step over** value to **0.05**.

The default **Floor finish** option enables you to perform finishing on the bottom of the engraved text.



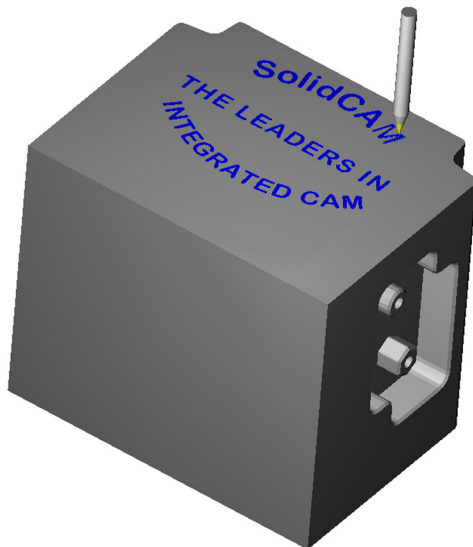
7. Save and Calculate

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



8. Simulate the operation

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



9. Add an Engraving Operation


Add another Engraving operation.

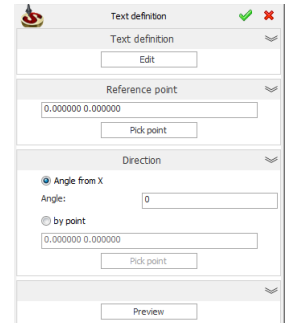
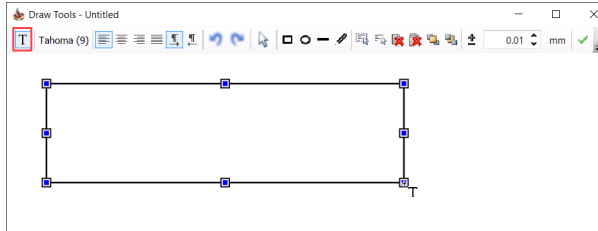


10. Define the Geometry

The geometry for this operation is the text typed in directly from your keyboard.

Click  in the **Geometry** page. In the **Geometry Edit** dialog box, click the **Add** button in the **Text** section.


In the **Text definition** dialog box, click the **Edit** button in the **Text definition** section. The **Draw Tools** dialog box is displayed. Click the text symbol  and draw a rectangle to define a text frame.

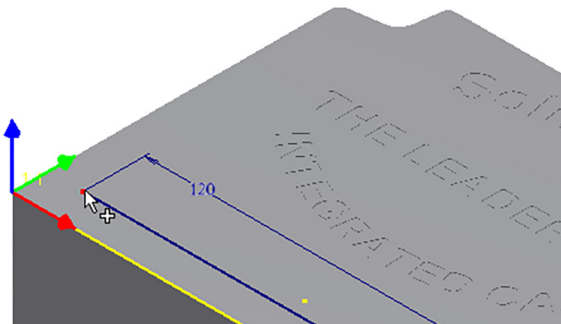


Type in the text that you want to engrave, for example “InventorCAM 2023.” Use the toolbar icons to change the text properties: font, size, alignment, and so on.

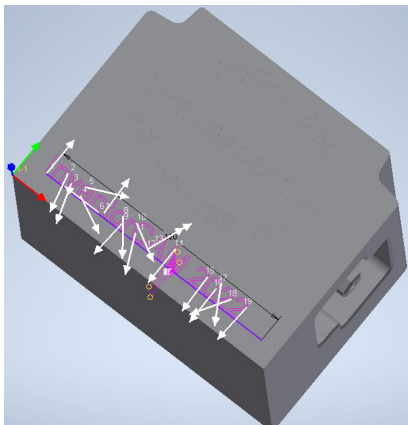
Click  to confirm the text.

In the **Reference point** section, click the **Pick point** button. Select the reference point directly on the model, clicking the point in the beginning of the sketch line.

The point coordinates are displayed in the **Pick reference point** dialog box. Confirm the point selection by clicking . The coordinates of left bottom corner of the engraved text frame will be in this point.



Click the **Preview** button to see how the text fits the available space. InventorCAM creates a number of chains on the model surface.



Click **Resume** and accept the chains by clicking . The **Geometry Edit** dialog box is displayed.

The **Chain List** section contains the **Text** item which includes all selected chains. This item can be edited and modified every time you need to make a change.

Click to choose the selected geometry for the operation.



11. Define the Tool

Use the same **Engraving tool** of $\varnothing 6$ as in the previous operation.



12. Define the Milling levels

Set the **Engraving depth** value to **0.2**.



13. Define the technological parameters

Switch to the **Technology** page.

In the **Modify** section, choose **Center** in the **Tool side** list. The tool tip will be positioned exactly on the geometry.



14. Save and Calculate

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



15. Simulate the operation

Perform the simulation of both the operations in the **SolidVerify** mode.



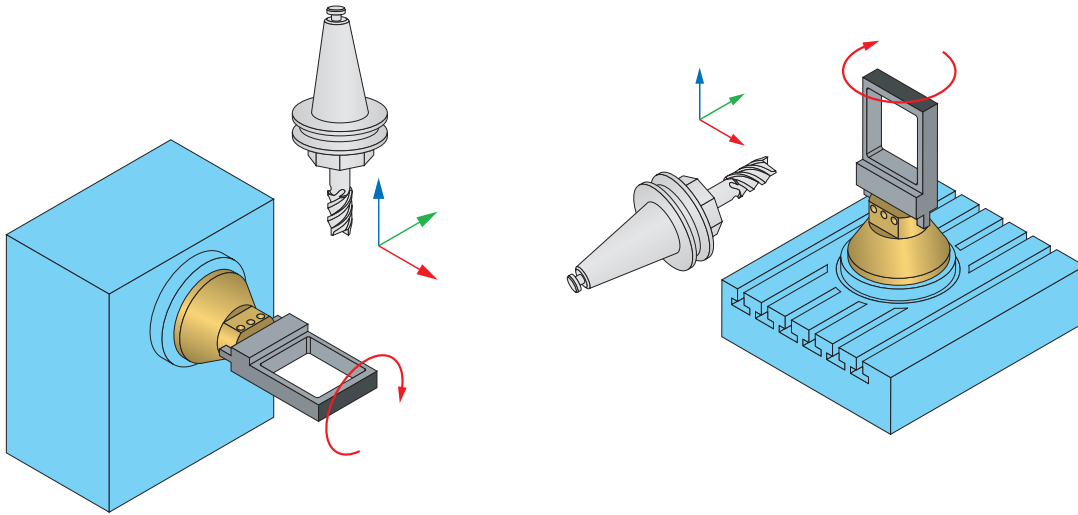
At this stage, the exercise is completed.

Indexial 4-Axis Milling

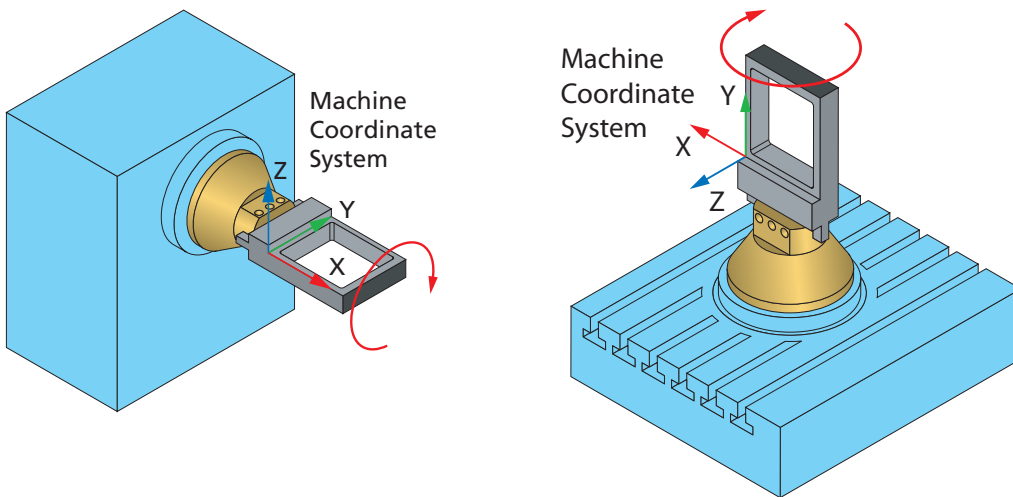
4

InventorCAM 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.

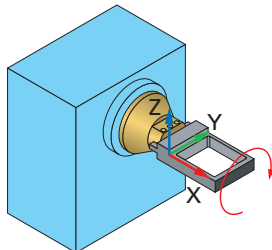


For each clamping, InventorCAM enables you to define the Machine Coordinate System with X-axis (for vertical machines) or Y-axis (for horizontal machines) parallel to the revolution axis of the rotary table.

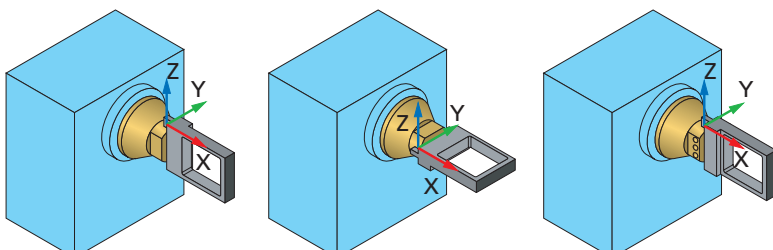


Within one clamping (Machine Coordinate System), InventorCAM enables you to define a number of positions around the revolution axis. For vertical machines the X-axis of these positions is parallel to the X-axis of the Machine Coordinate System.

Machine Coordinate System

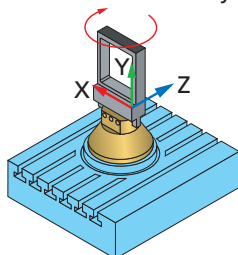


Positions

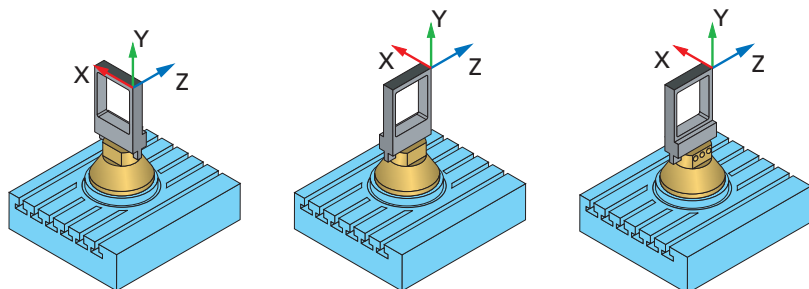


For horizontal machines, the Y-axis of these positions is parallel to the Y-axis of the Machine Coordinate System.

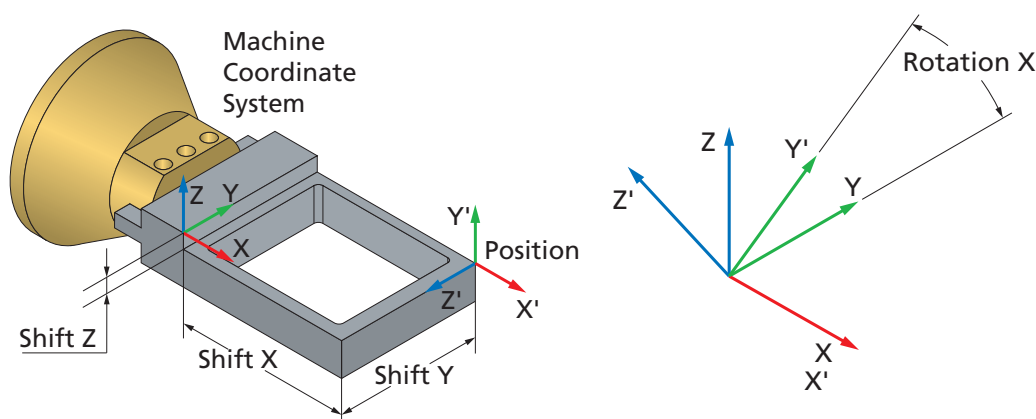
Machine Coordinate System



Positions



For each position, InventorCAM automatically determines the **Shift** and the **Rotation** parameters relative to the Machine Coordinate System.



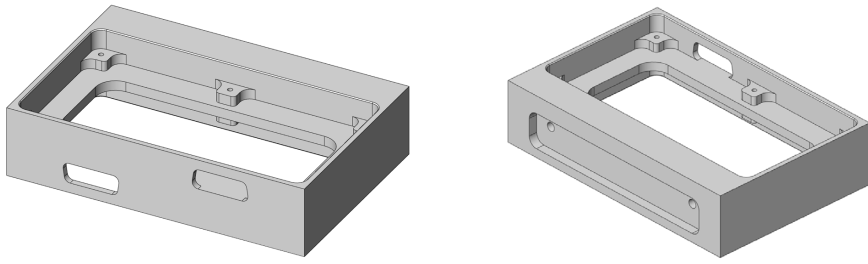
The **Shift** components (**Shift X**, **Shift Y** and **Shift Z**) describe the shifting of the position relative to the Machine Coordinate System.

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**.

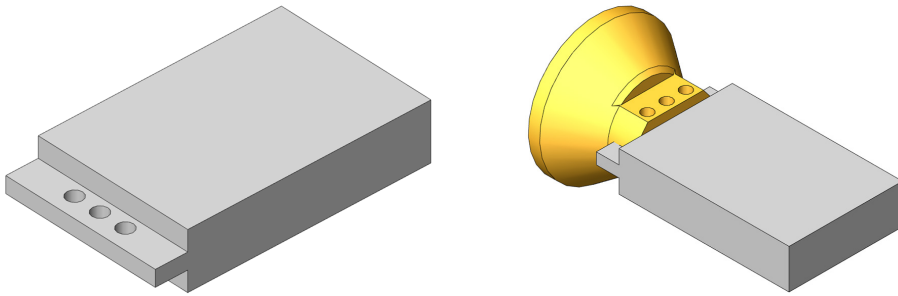
Machine Coordinate Systems and their positions are used for the geometry definition. Before the geometry definition you have to choose the Coordinate System position. The geometry has to be located in the plane parallel to the XY-plane of the chosen CoordSys.

Exercise #12: Frame Machining

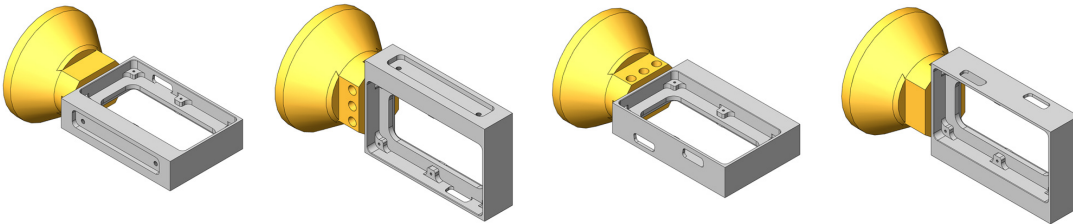


In this exercise, indexial 4-axis operations are used for the frame part machining.

The frame is machined from the pre-machined stock on a vertical 4-axis milling CNC-machine using a 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.



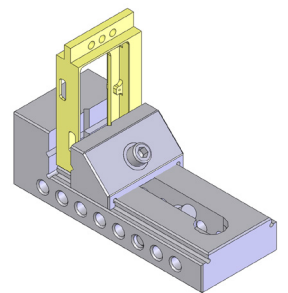
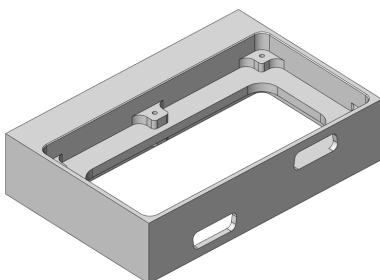
1. Load the Autodesk Inventor model

Load the **Exercise12.ipt** model located in the **Exercises** folder.

The model contains three iPart members:

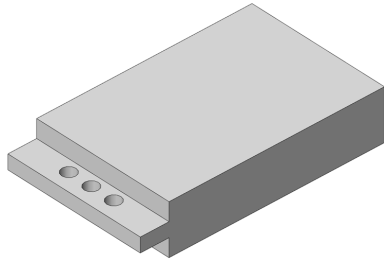
Default

The original design model without changes.



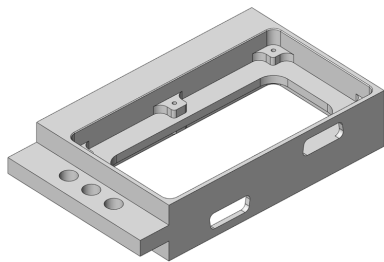
Stock

The stock model that is used for the InventorCAM stock definition.



Machining

The machining model with a tail that was not removed. This configuration is used for the InventorCAM geometries definition. You can switch between these iPart members by double-clicking their subheaders



under the **Table** header of the **Model** elements tree.



2. Start a new project

To start a new project, click the **InventorCAM 2023** in the main menu of Autodesk Inventor and choose **Milling** from the **New** submenu. The **Milling Part Data** dialog box is displayed.



3. 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, choose the **gMilling_3x** CNC-controller from the list.



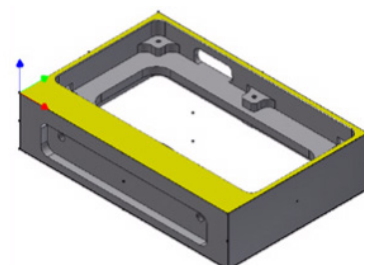
4. Define the Machine Coordinate System

Click the **CoordSys** button in the **Define** section of the **Milling Part Data** dialog box to start the Coordinate System definition.

The **CoordSys** dialog box is displayed.

InventorCAM assigns the first **Position** number to the **Machine Coordinate System** by default.

Make sure that the **Select face** option is chosen. Select the **Top corner of model box** option in the **Place CoordSys origin to** list.



Click on the model bottom face as shown.

The **Machine Coordinate System #1** is automatically defined in the top corner of the model box. Click .

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

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



5. Define the CoordSys positions

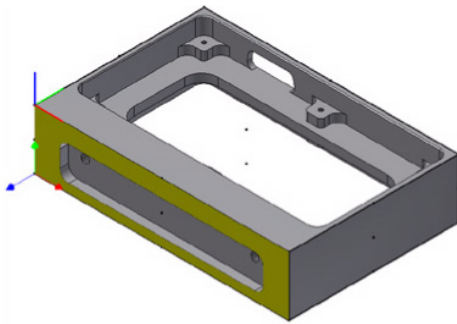
Now you have to define several additional **Positions** around the **Machine Coordinate System #1**.

In **CoordSys Manager**, click the **Add** button.


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

InventorCAM assigns the next sequential number to the position.

Click on the model face as shown.



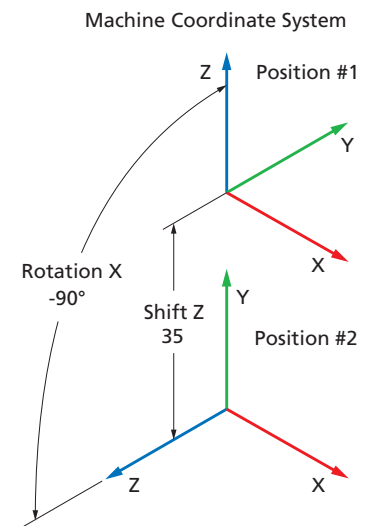
The origin is automatically placed in the top corner of the model. The Z-axis is directed outwards normally to the selected face. The X-axis direction is the same as the direction of the X-axis of the Machine Coordinate System (**Position #1**).

Click  to confirm the position location. The **CoordSys Data** dialog box is displayed.

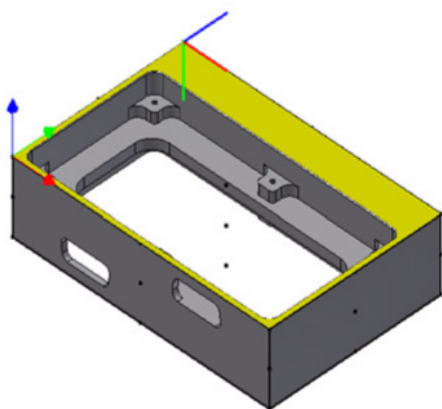
The **Part Upper level (0)** and the **Part Lower level (-100)** are automatically determined on the model.

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 .



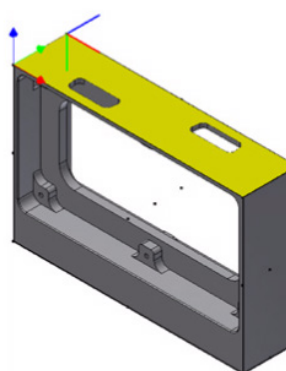
In the same manner, define **Position #3** and **Position #4** using the model faces as shown.



Position #3

Shift (0,100,-35)

Rotation (180,0,0)



Position #4

Shift (0,100,0)

Rotation (90,0,0)

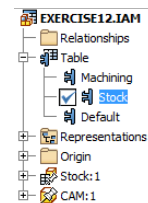
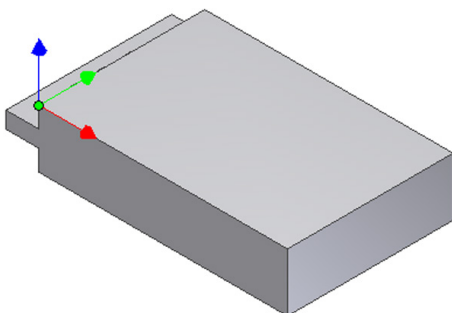
Confirm the **CoordSys Manager** dialog box with .

Confirm the **Milling Part Data** dialog box with .



6. Define the Stock model

Switch from **InventorCAM Manager** to the **Model** tab. Switch to the **Stock** iPart member. Expand the **Table** component and double-click the **Stock** item in the list of the iPart members. The model is displayed in the configuration ready for the stock definition.



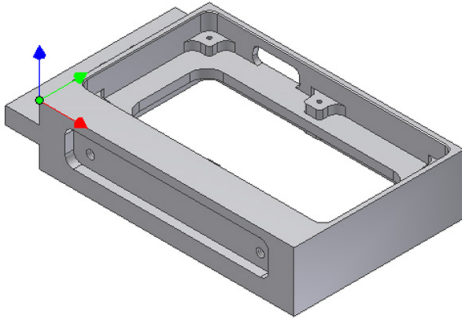
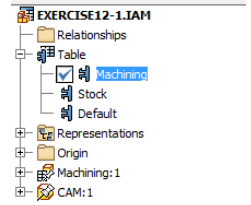
Right click the **Stock > Define**. The **Stock** dialog box is displayed.

In the **Defined by** list, select the **3D Model** option. This option enables you to define the stock model by picking solid bodies. Click on the model to select it for the stock model.



7. Close the Model dialog box with Define the Target model

In the **Model** elements tree, double-click on the **Machining** iPart member that describes the model after the machining. The model is changed.



Right click the **Target > Define**. The **Target** dialog box is displayed.

Click on the model to select it. Set the **Facet tolerance** value to **0.01**.

Close the **Target** dialog box with .

Click the **InventorCAM Manager** tab.



8. Define the Fixture

InventorCAM enables you to define the **Part fixtures** such as clamps, vises, jig plates, etc. This feature allows you to get a more realistic picture during the simulation and check possible collisions between the cutting tools and fixtures.

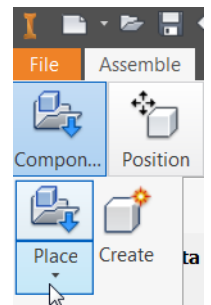
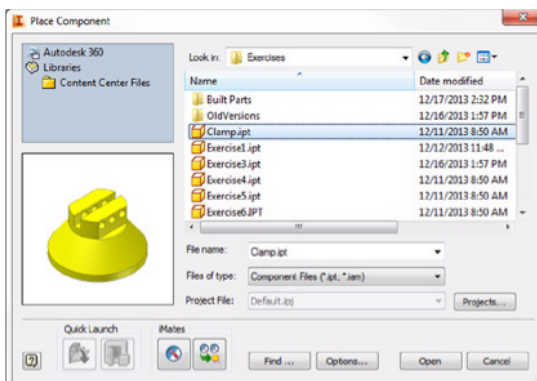
At first, you have to add the model of the clamping device to the CAM-Part assembly.



It is recommended to copy your clamp model files into the CAM-Part folder before insertion into the CAM-Part assembly.

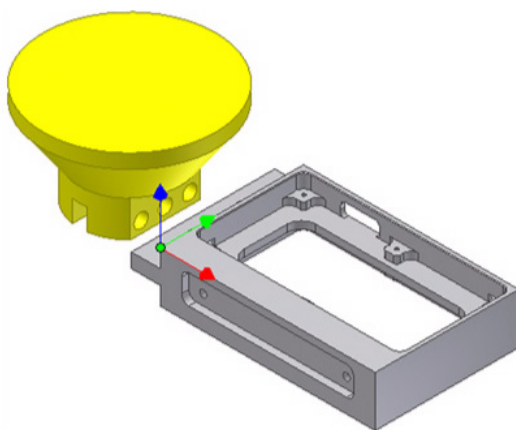
In the **Assemble** menu of Autodesk Inventor, click **Place**.

The **Place Component** dialog box is displayed.



Choose the **Clamp.ipt** file. This file contains the model of the clamping device.

Confirm the component insertion with the **Open** button. The model is added.



Align the model as shown.

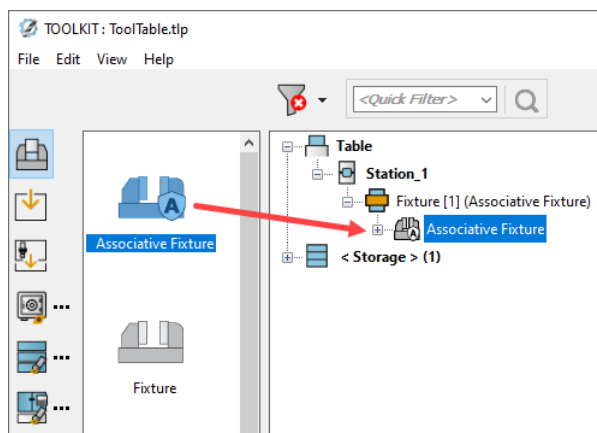
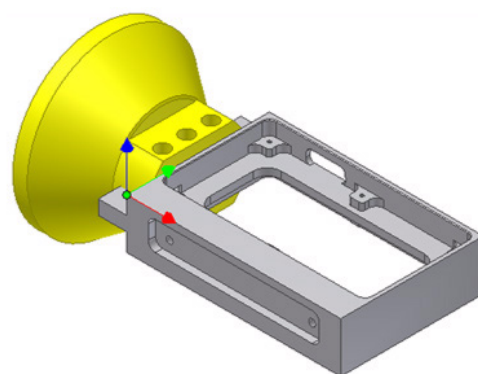
In **InventorCAM Manager**, right-click the **Setup > Edit** menu.

The **Machine setup** window is displayed.

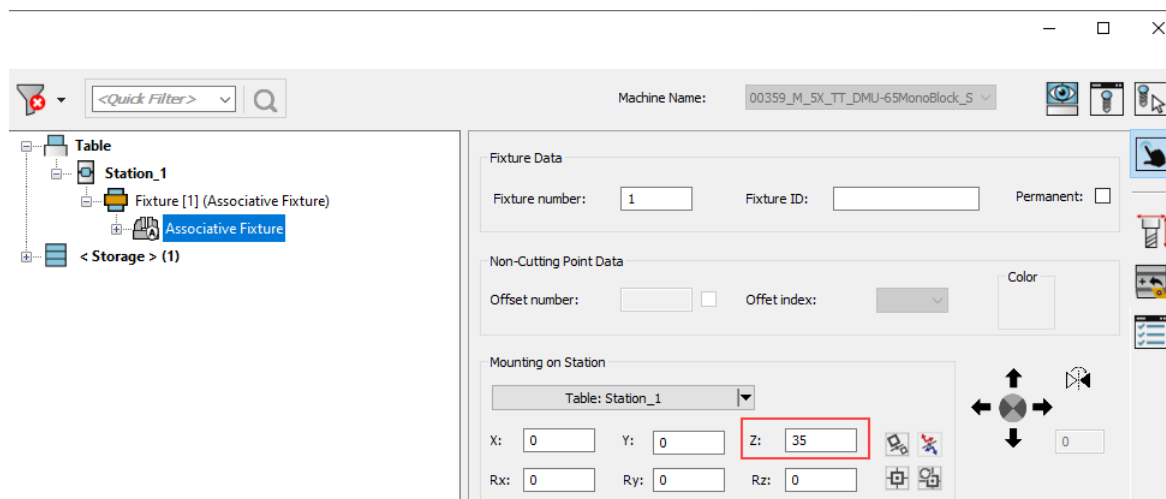
Click the **Open ToolKit** icon.

The **ToolKit** window is displayed.

Double click the **Associative Fixture** icon to add it below **Station_1**.

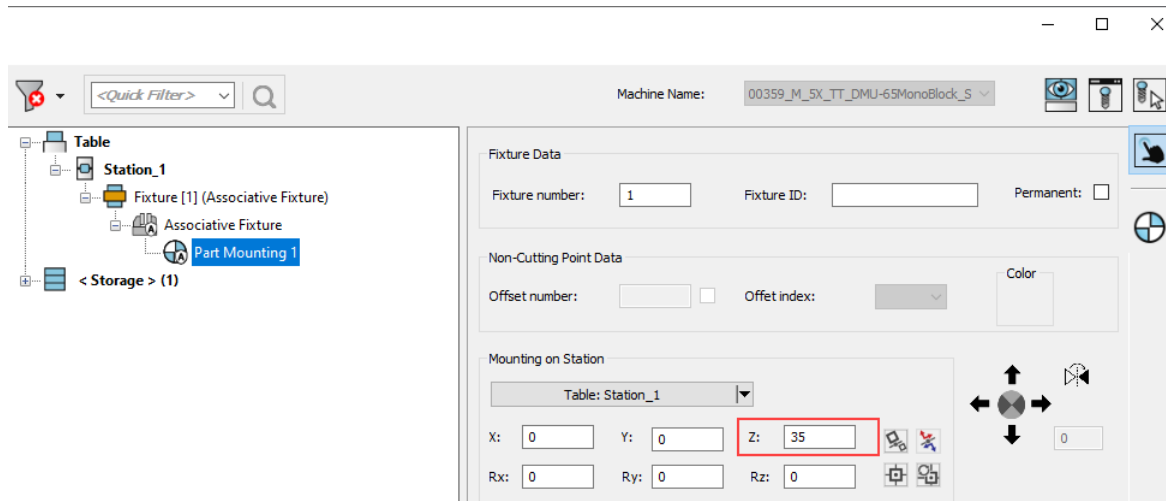


Click  and set the **Z** value as **35**.



Expand **Associative Fixture** and click **Part Mounting**.

Ensure the **Z** value is set as **35**.



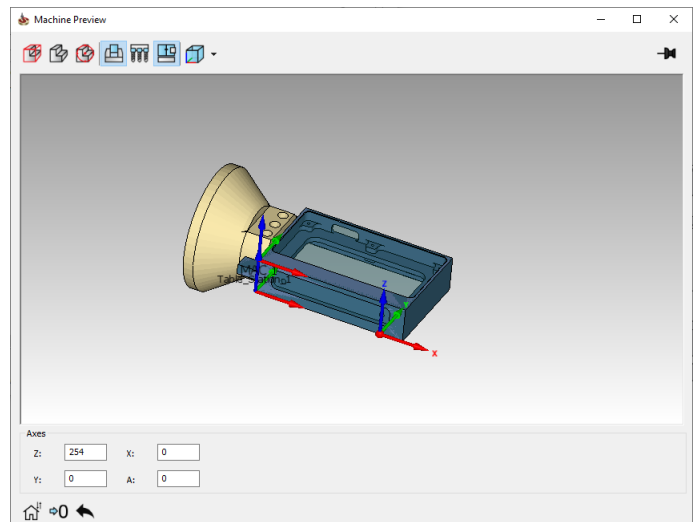
Click to select the fixture.

Click on the **Machine Preview** icon in the **Machine setup** window to visualise the fixture mounted on the part.

Close the **Machine Preview** window.

Click **Save &Exit** in the **Machine setup** window.

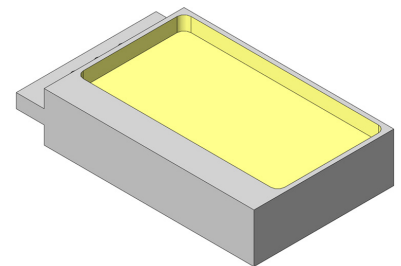
The fixture is now defined.



9. 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.



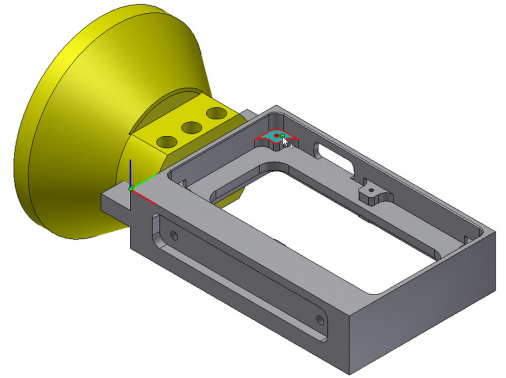
Define the Geometry as shown.

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.

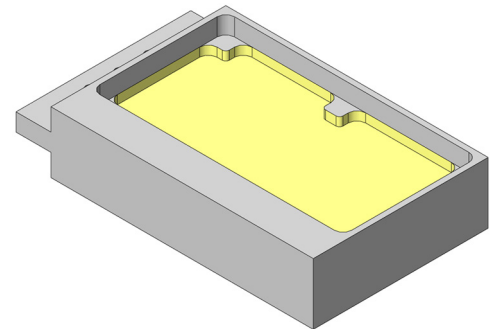
Use the **Helical Ramping** strategy. Set the **Radius** value to **5**.



10. 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.

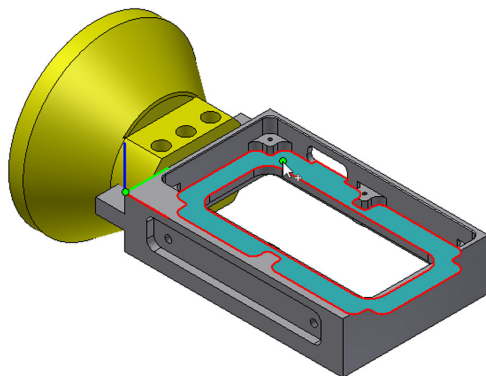
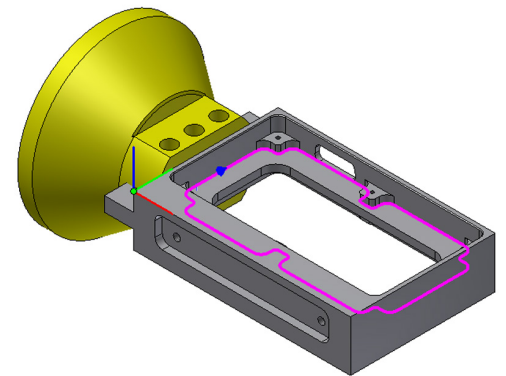


Define the Geometry as shown.

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.

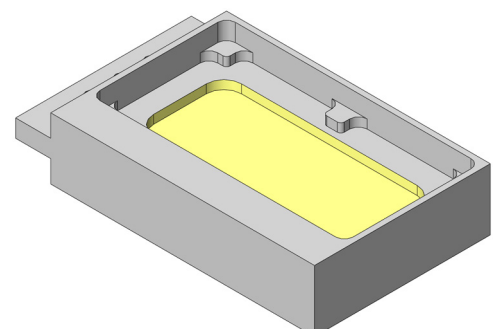
Use the **Helical Ramping** strategy. Set the **Radius** value to **5**.



11. Define a Pocket operation

Define a Pocket operation to perform the rough machining of the highlighted faces.

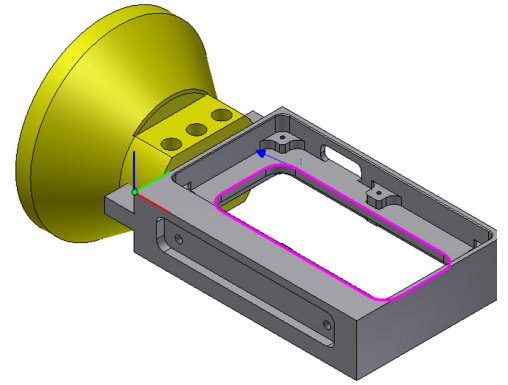
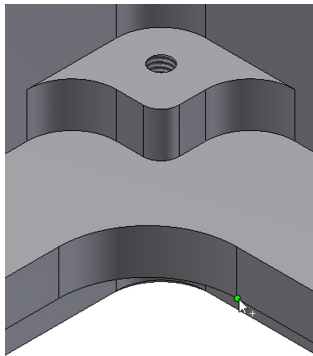
Choose the **Coordinate System Position #1** in the **Geometry** page of the **Pocket Operation** dialog box.



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**.



12. Define a Profile operation

Define a Profile operation to perform the finish machining of the highlighted faces.

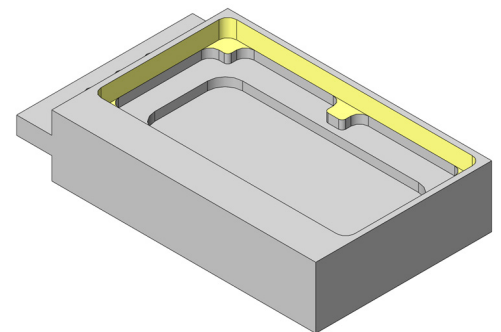
Choose the **Coordinate System Position #1** in the **Geometry** page of the **Profile Operation** dialog box.

Use the geometry defined in **Step #9**.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Use the same milling levels that were defined in **Step #9**.

Choose the **Arc** strategy for **Lead in** and select the **Same as Lead in** check box under **Lead out**.



13. Define a Profile operation

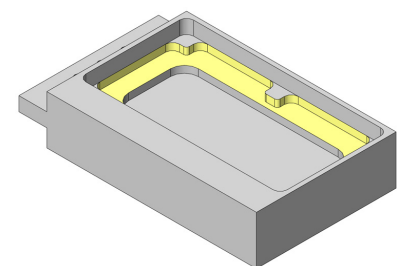
Define a Profile operation to perform the finish machining of the highlighted faces.

Choose the **Coordinate System Position #1** in the **Geometry** page of the **Profile Operation** dialog box.

Use the geometry defined in **Step #10**

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Use the same milling levels used in **Step #10**. 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**.



14. Define a Profile operation

Define a Profile operation to perform the finish machining of the highlighted faces.

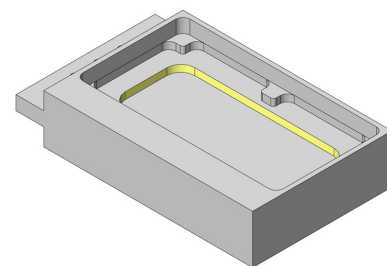
Choose the **Coordinate System Position #1** in the **Geometry** page of the **Profile Operation** dialog box.

Use the geometry defined in **Step #11**.

Choose the **Tool #1 (Ø10 End mill)** for the operation.

Use the same milling levels used in **Step #11**.

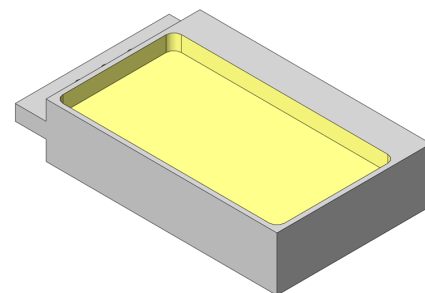
Choose the **Arc** strategy for **Lead in** and select the **Same as lead in** check box under **Lead out**.



15. Define a Pocket operation

In the same manner as explained in **Step #9**, define a Pocket operation to perform the rough machining of the highlighted faces.

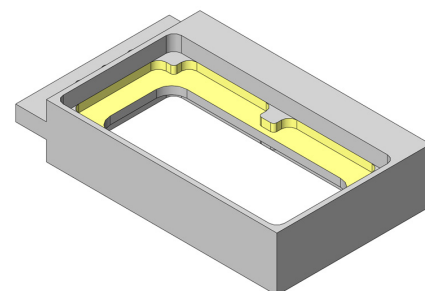
Use the **Coordinate System Position #3** for the operation.



16. Define a Pocket operation

In the same manner as explained in **Step #10**, define a Pocket operation to perform the rough machining of the highlighted faces.

Use the **Coordinate System Position #3** for the operation.

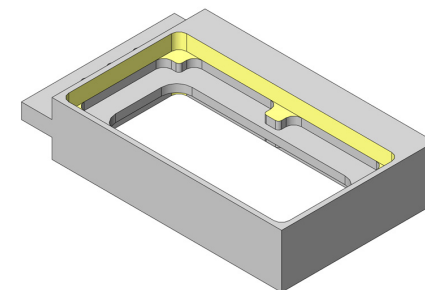


17. Define a Profile operation

In the same manner as explained in **Step #12**, define a Profile operation to perform the finish machining of the highlighted faces.

Use the geometry defined in **Step #15**.

Use the **Coordinate System Position #3** for the operation.

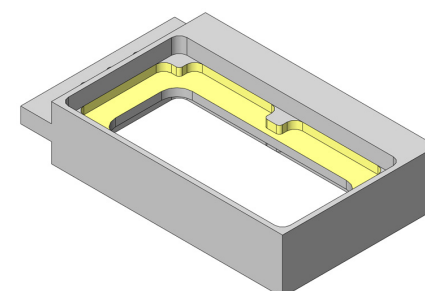


18. Define a Profile operation

In the same manner as explained in **Step #13**, 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 #16**.

Choose the **Tool #1 (Ø10 End mill)** for the operation.



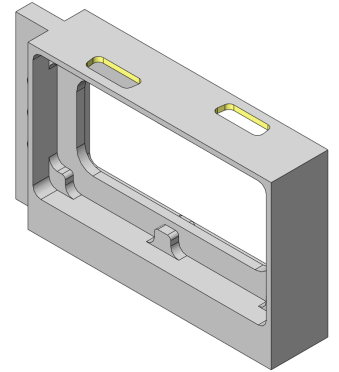
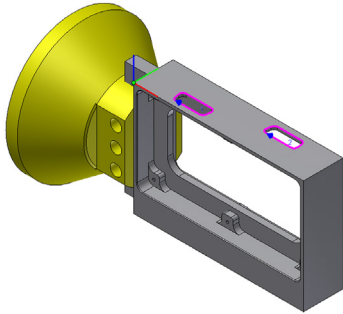
Set the **Clear offset** and the **Step over** values to perform the machining of the bottom face in several concentric profiles.



19. 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.



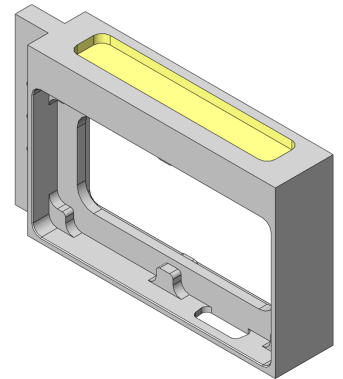
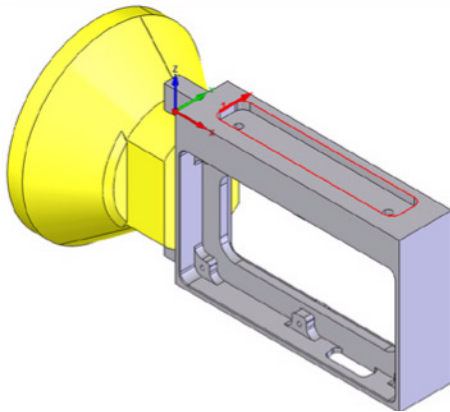
Define a new **End mill** tool of $\varnothing 6$ for the operation. Choose the appropriate tool holder.



20. 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 ($\varnothing 10$ 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

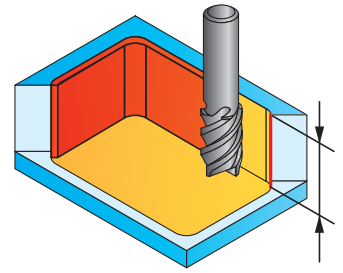
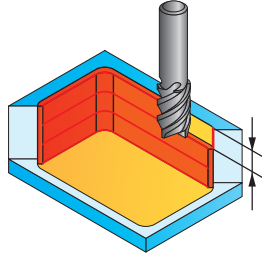
Wall finishing can be performed in a single cut at the whole depth or in a number of cuts at each step down.

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.

**21. Perform the holes machining**

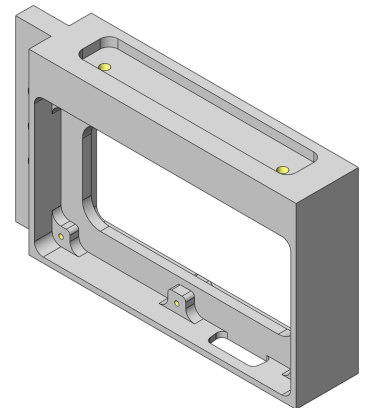
Define several Drilling operations to perform center drilling, drilling and threading of the holes.

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.

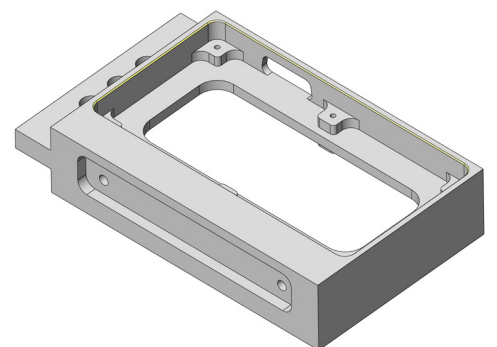
Use the **Ø6** spot drill (with **90°** tool tip angle); a **Ø5** drill and a **Ø6** tap tool. Choose the appropriate tool holders.

**22. 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 #9** for the operation. This geometry is defined using the **Coordinate System Position #1**.

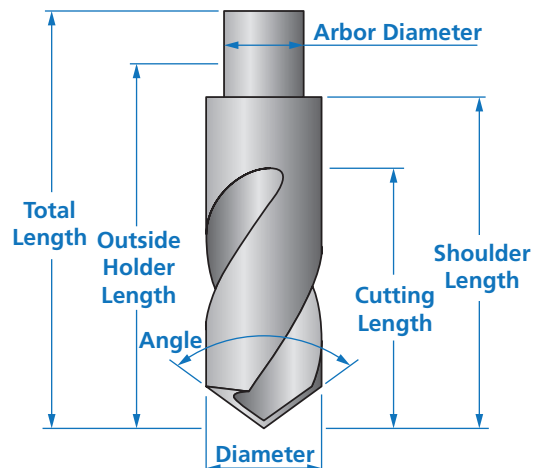
Define a **Chamfer mill** tool of **Ø6** for the operation.





Chamfer mill

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**.

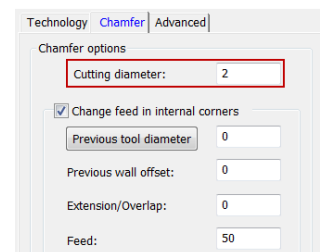
In the **Rest material/Chamfer** section, choose the **Chamfer** option from the list. A new page under the **Chamfer** tab is displayed.

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**.

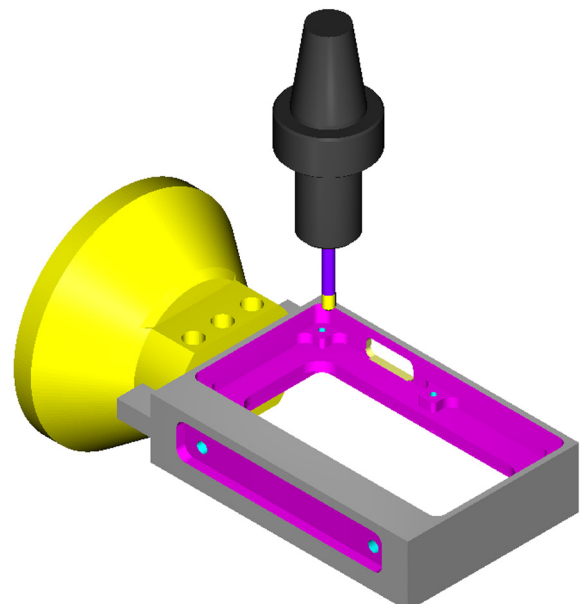


23. Simulate the tool path

Simulate the CAM-Part in the **SolidVerify** mode.

InventorCAM displays the fixture and checks the possible gouges during the simulation.

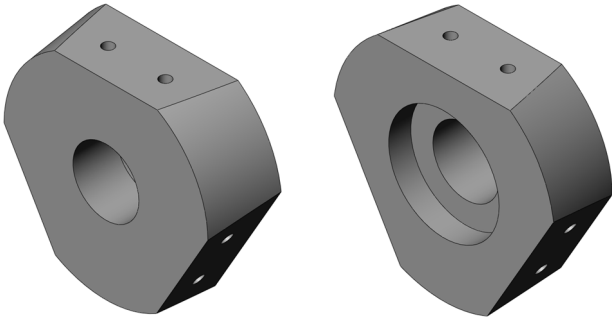
In the **SolidVerify** simulation, the tool is shown going around the part (rather than rotating the part as it actually occurs in the CNC-machine). Therefore, for all CNC-machine types, the simulation looks the same.



At this stage, you have completed the exercise.

Exercise #13: Mounting Machining

Define the CAM-Part and operations for machining of the mounting part shown below on the 4-axis vertical milling CNC-machine.



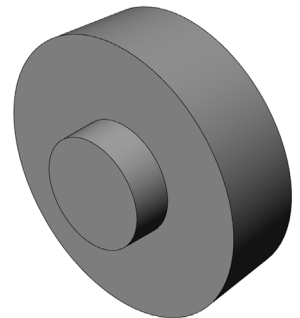
This exercise reinforces the following skills:

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

The Autodesk Inventor model of the Mounting part (**Exercise13.ipt**) 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:



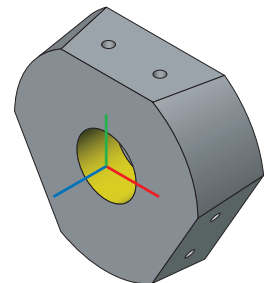
1. Define the CAM-Part

At this stage, you have to define the CAM-Part, CNC-controller (**gMill_Haas_4x_eval**), Machine Coordinate System, the Stock and Target models.

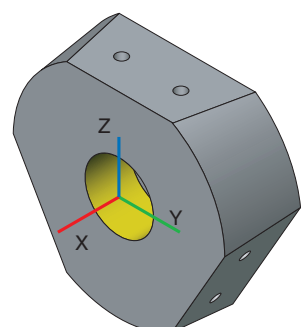


Machine Coordinate System definition

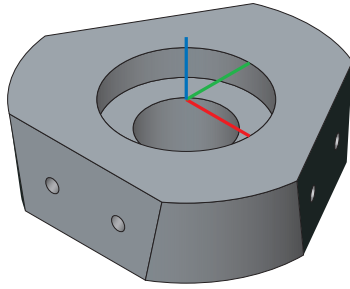
In this exercise, the Machine Coordinate System has to be defined in the center of the revolution face as shown.



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.



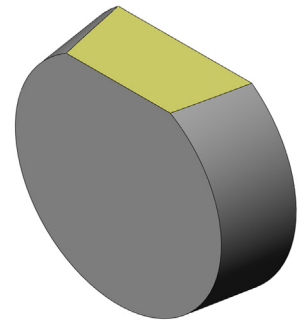
Define the Stock and the Target models using the configurations of the original Autodesk Inventor model.



2. Face milling

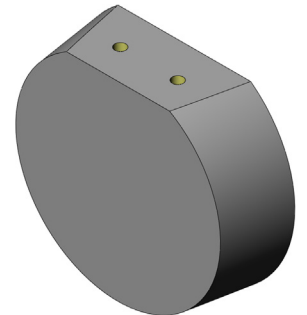
Define a Face Milling operation with the **Hatch** strategy to machine the shown face.

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.



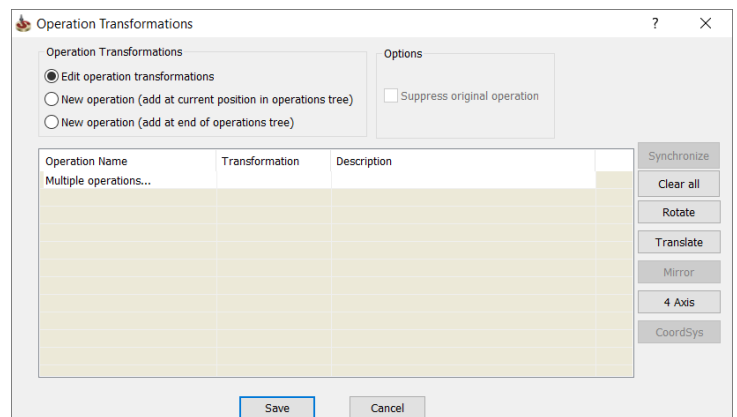
4. Define the operation pattern

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 **InventorCAM 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 **InventorCAM 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 **InventorCAM 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)

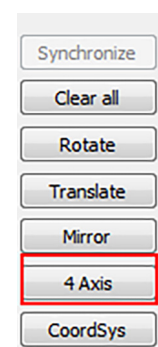
This option also enables you to add a copy of the selected operation and apply the transformation to the copied operation, without effect on the original operation. The transformed copy is inserted at the end of the operation list in the **InventorCAM Manager** tree. The original operation can be suppressed 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.

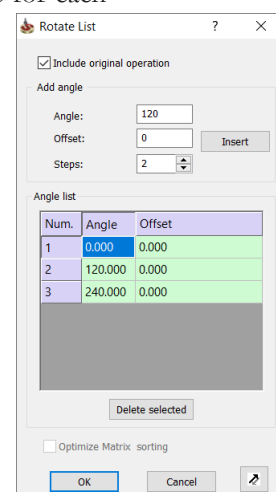
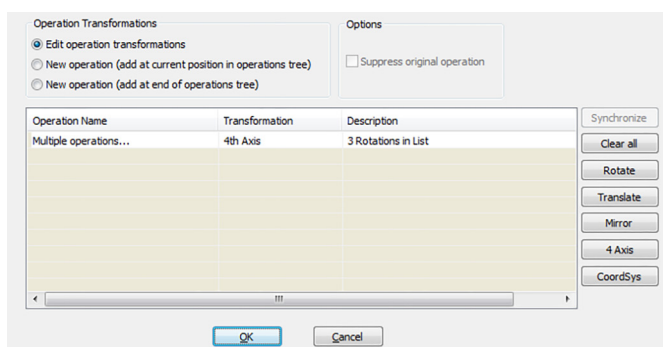


In the **Angle** field, type in the value of **120°**. This parameter defines the rotation angle for each pattern instance. In the **Steps** field, enter **2** for the number of instances.


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.



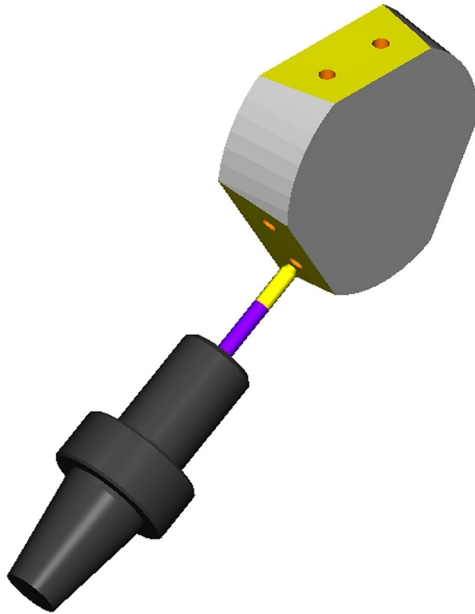
Confirm this dialog box with the **OK** button.

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.



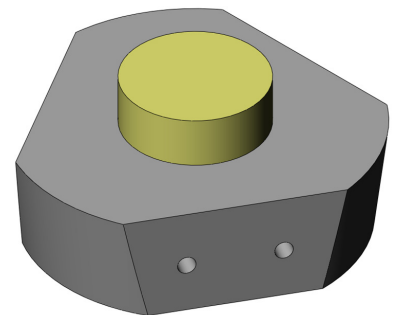
5. Simulate the transformed operations

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.

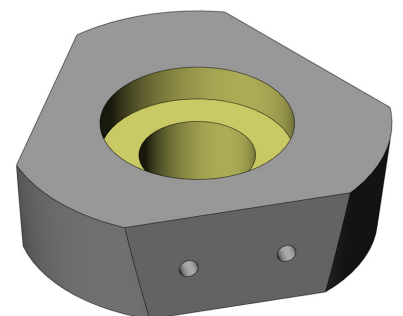


7. Machine the back side

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.



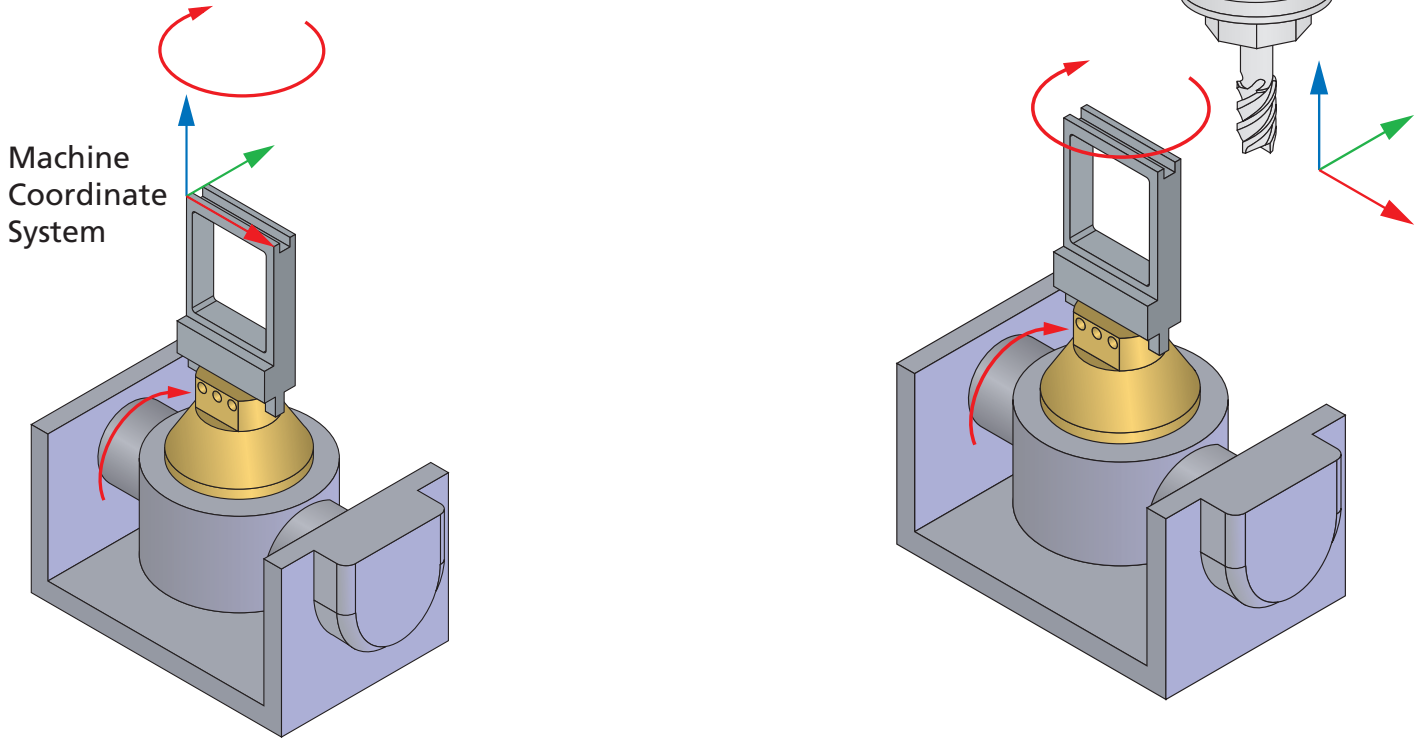
Indexial 5-Axis Milling

5

InventorCAM enables you to perform indexial 5-axis milling using either vertical or horizontal 5-axis CNC-machines.

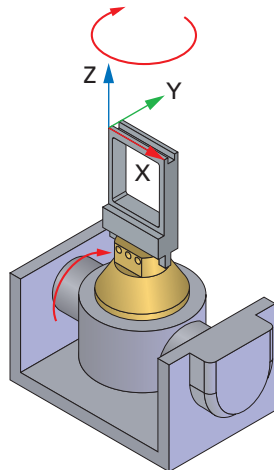
The 4th and 5th axis rotation is used for positioning of a part. When the part is positioned to the necessary orientation, the 4th and the 5th axes are locked and machining is performed with 3-axis movements.

For each clamping, InventorCAM enables you to define the Machine Coordinate System.

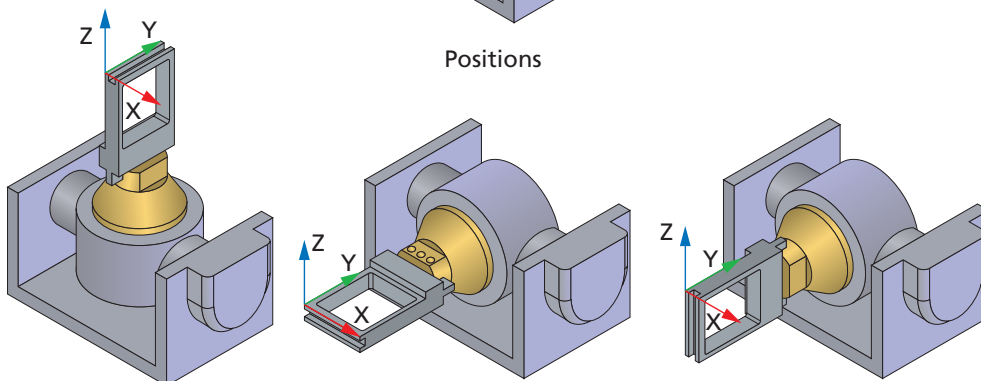


InventorCAM enables you to perform indexial 5-axis milling using either vertical or horizontal 5-axis CNC-machines.

Machine Coordinate System



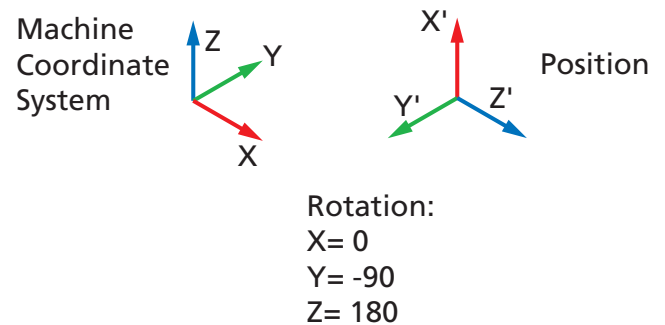
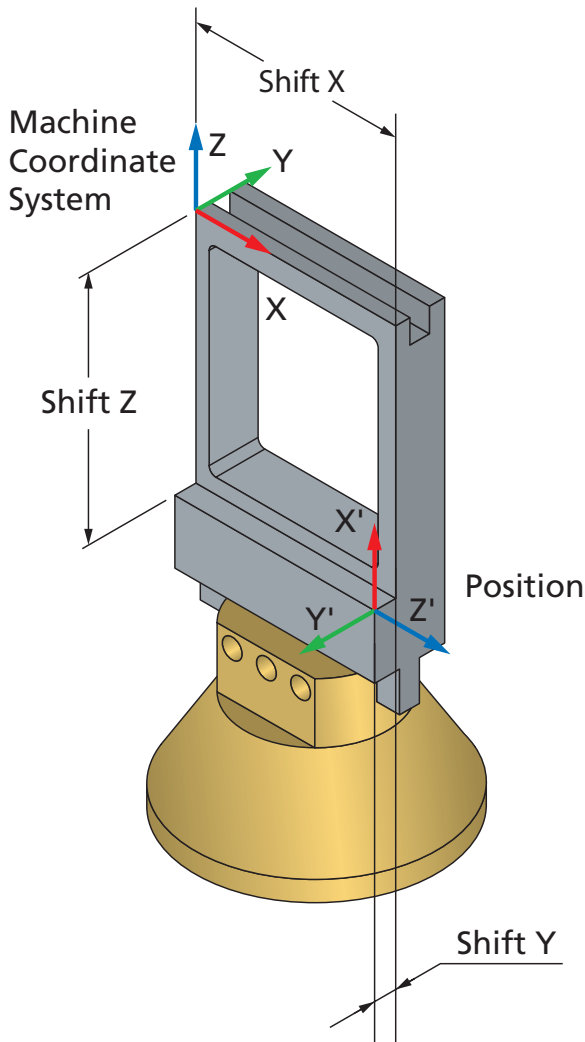
Positions



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, InventorCAM enables you to define the Machine Coordinate System.

For each clamping (Machine Coordinate System), InventorCAM enables you to define a number of related positions around the Machine CoordSys. Take care of the machine limitations (angles or undercuts).

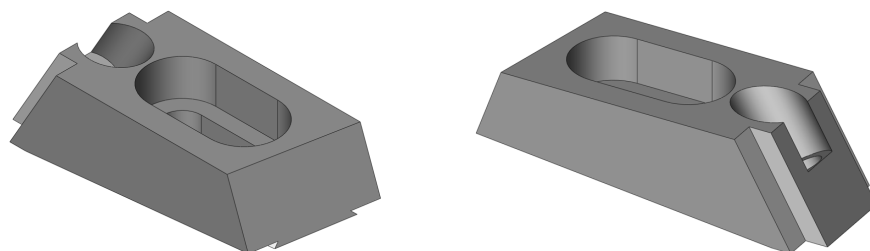


For each position, InventorCAM automatically determines the **Shift** and **Rotation** parameters relative to the Machine Coordinate System.

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.

Exercise #14: 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

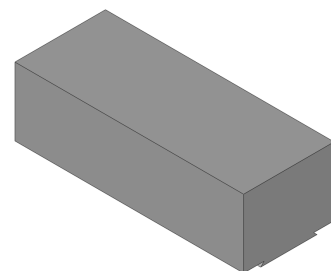
The Autodesk Inventor model of the Clamp (**Exercise14.ipt**) is located in the **Exercises** folder. The model contains two Autodesk Inventor iPart members:

- **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

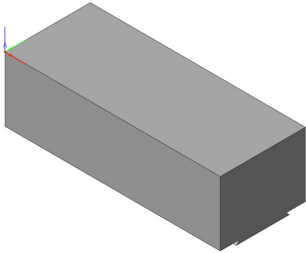
At this stage, you have to define the CAM-Part, CNC-controller (**TTC_Hermle_C20_full**), Machine Coordinate System and its Positions, the Stock and Target models.



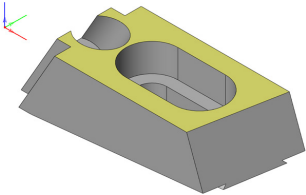
Coordinate System definition

Define **Machine Coordinate Systems #1** and **#2** and related positions using the previously defined Stock model.

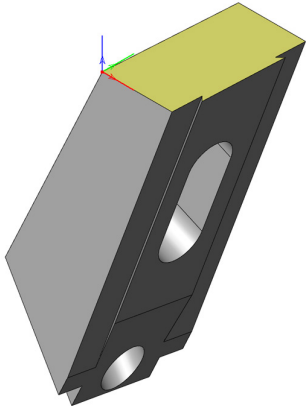
Machine Coordinate System #1 Position #1



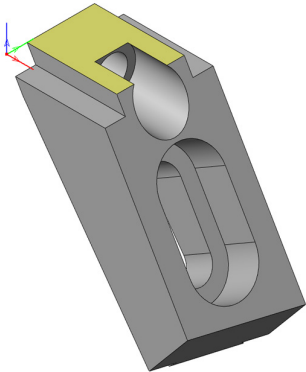
Machine Coordinate System #1 Position #2



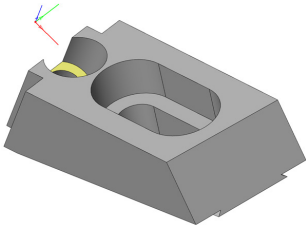
Machine Coordinate System #1 Position #3



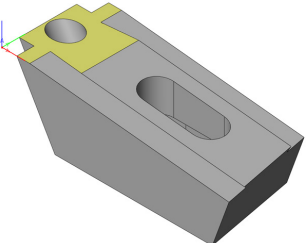
Machine Coordinate System #1 Position #4



Machine Coordinate System #1 Position #5



Machine Coordinate System #2 Position #1



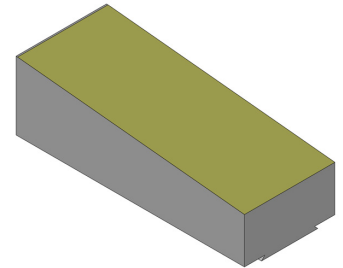
Define the Stock model using the **Stock** configuration of the CAM-Part assembly.

Define the Target model using the **Machining** configuration of the CAM-Part assembly.



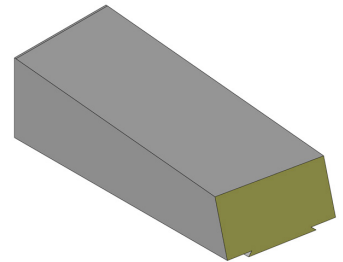
2. Top face machining

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.



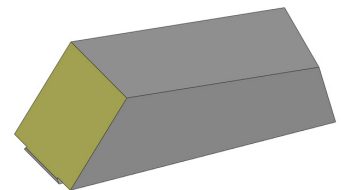
3. Back face machining

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.



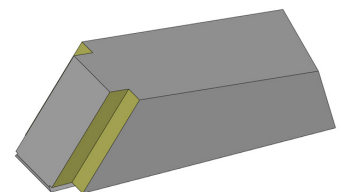
4. Front face machining

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.



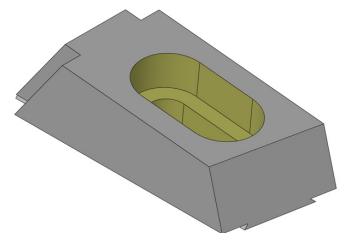
5. Openings machining

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

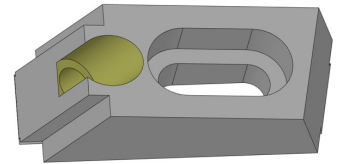
Define a number of operations to machine the slot faces located on the top inclined face of the clamp. Use the **Machine Coordinate System #1 (Position #2)** for the operation.





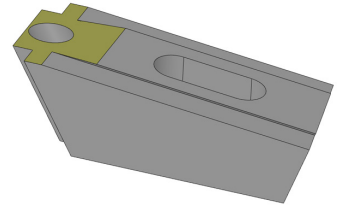
7. Hole machining

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.



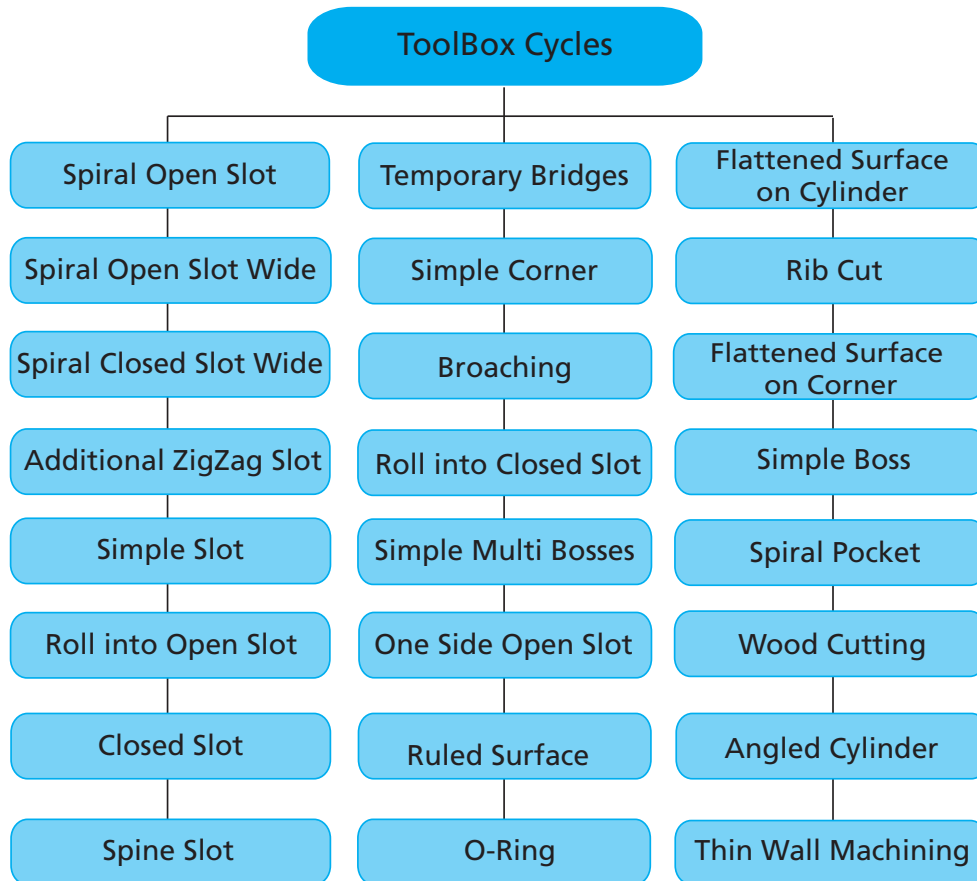
Toolbox

6

ToolBox sub-operations are a set of 2.5D Milling operations, each intended for a specific machining case. Some operations cover different strategies of slots machining. Each sub-operation provides you with a specific machining strategy optimal for a particular machining case.

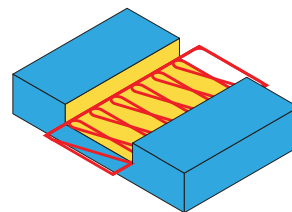
These operations are combined in a single dialog box as sub-operations.

InventorCAM offers you the following types of ToolBox Cycles:



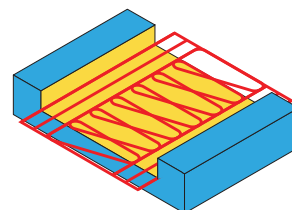
Spiral Open Slot

This strategy enables you to perform machining of open slots. The geometry consists of two chains. The machining is performed in a spiral manner.



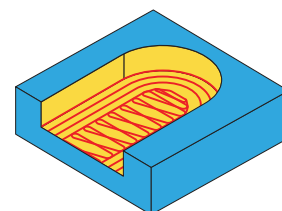
Spiral Open Slot Wide

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.



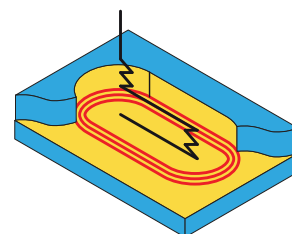
Spiral Closed Slot Wide

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.



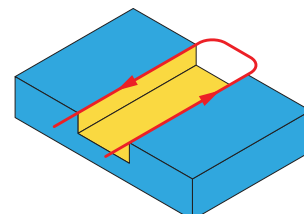
Additional Zigzag Slot

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.



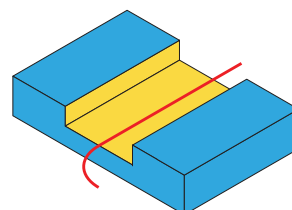
Simple Slot

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.



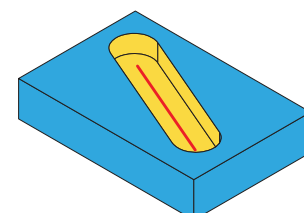
Roll into open slot

This strategy enables you to machine an open slot starting from the slot center to the walls, while rolling in the material with an arc movement.



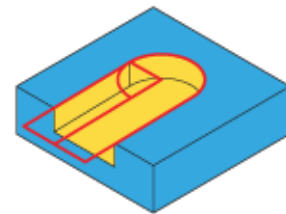
Closed Slot

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.



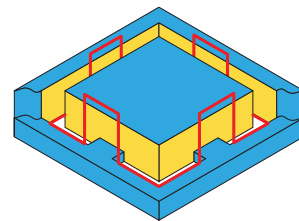
Spine Slot

This strategy enables you to perform machining of a slot opened from one side. InventorCAM 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.



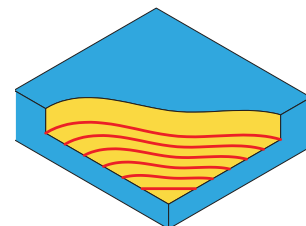
Temporary Bridges

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.



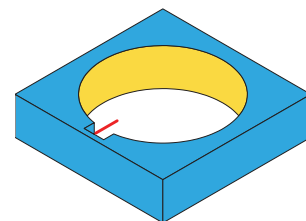
Simple Corner

This strategy enables you to perform machining of an open corner area in a number of equidistant cutting passes..



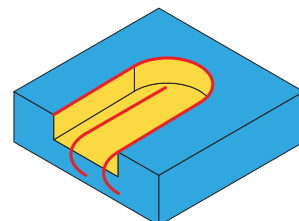
Broaching

This strategy enables you to perform broaching of holes.



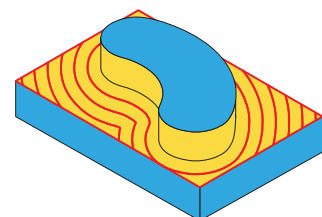
Roll into closed slot

This strategy enables you to machine a closed slot starting from the slot center to the walls, while rolling in the material with an arc movement.



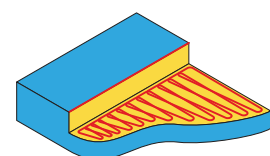
Simple Multi Bosses

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.



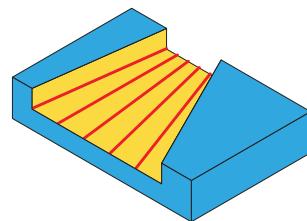
One Side Open Slot

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.



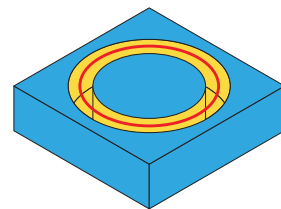
Ruled

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.



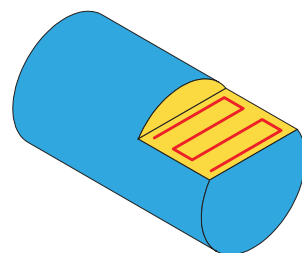
O-Ring

This strategy enables you to machine O-Ring slots.



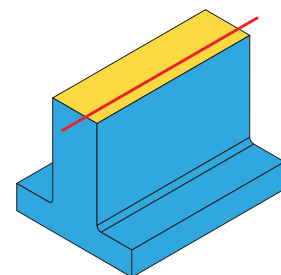
Flattened Surface on Cylinder

This strategy enables you to perform machining of the flattened surface on a cylinder.



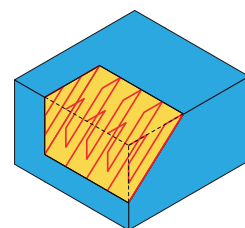
Rib Cut

This strategy enables you to machine a rib upper surface in a single pass shifted from the center line by a specified value. This is a safe method of cutting the upper face of a thin wall.



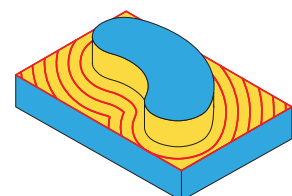
Flattened Surface on Corner

This strategy enables you to machine an angled plane using a defined working area in order to prevent air cuts.



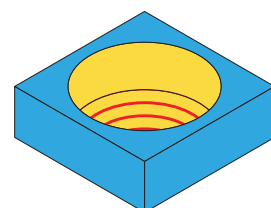
Simple Boss

This strategy enables you to perform machining of a boss located on a model face. 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.



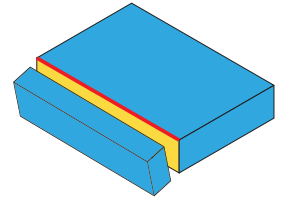
Spiral Pocket

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. Then the tool performs the spiral cutting pass morphing from the penetration point to the closed area walls.



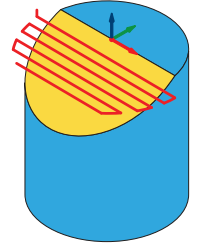
Wood Cutting

This strategy uses a special tool called “saw” to cut off a block of wood in a single pass. The First Step down parameter defines the depth of cutting.



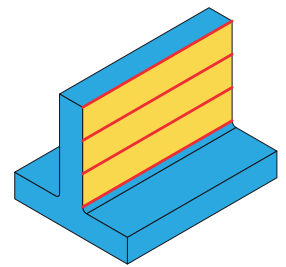
Angled Cylinder

In this cycle, the cylinder top is cut to create an angled surface. The cutting is performed by parallel cuts with a defined Step over.



Thin Wall Machining

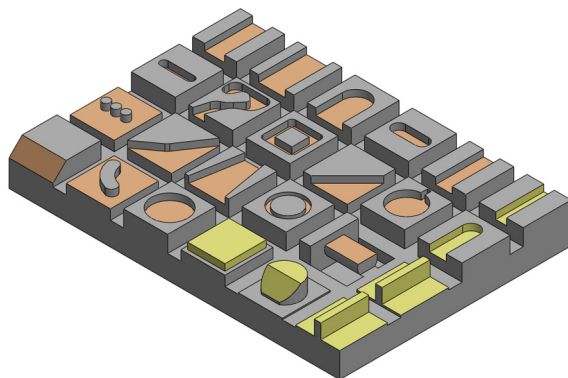
This strategy enables you to machine thin walls while removing gradually the excess material from either side of the wall. The method can be chosen from two options: Alternating steps or Supporting steps.



Exercise #15: Standard Cycles Machining

This exercise illustrates the usage of Standard Cycles to machine twenty four sample blocks located on a plate. Each block represents a specific machining case machined in a cycle with customized parameters.

The part file (**Exercise15.prz**) is located in the **Exercises** folder. This file contains the pre-defined CAM-Part.



1. Open the CAM-Part

Open the **Exercise15.prz** file located in the **Exercises** folder.



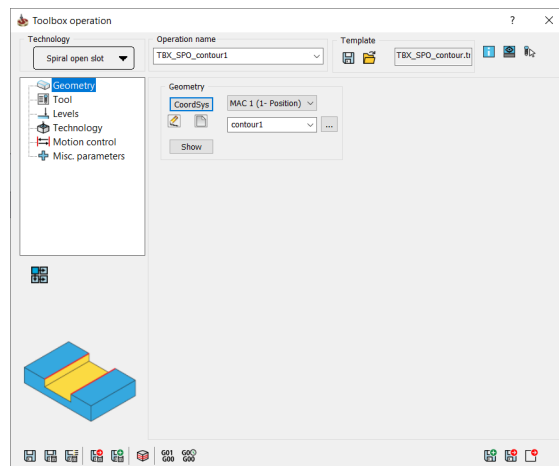
2. Spiral open slot machining

In **InventorCAM Manager**, right-click the **Operations** header 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.

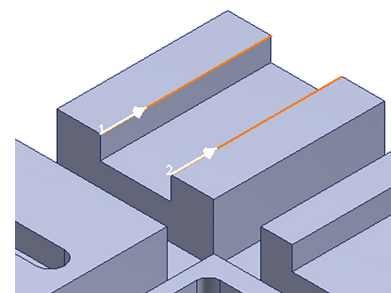
Confirm the geometry by clicking .

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.

Click the **Save** icon.

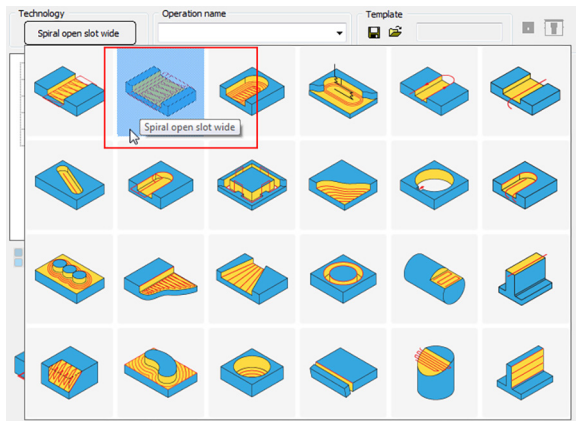




3. Wide spiral open slot machining

In the same manner as described in the previous step, define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Spiral open slot wide** strategy.

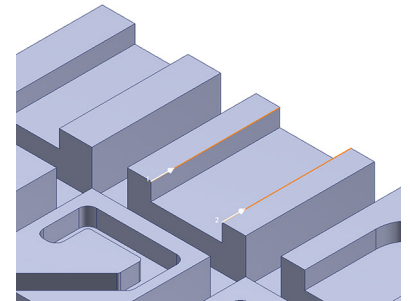


Define a new geometry, selecting a pair of chains on the wide slot edges.

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.

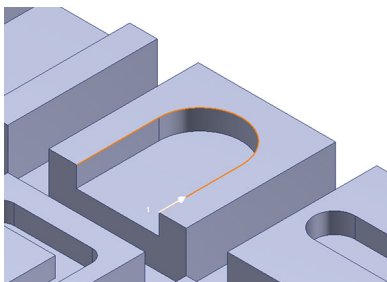
Click the **Save** icon.



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. Click the **Save** icon.

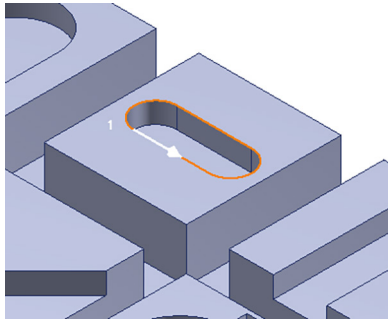


5. Additional Zigzag slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Additional Zigzag Slot** strategy.

Define a new geometry, selecting the closed chain as shown.

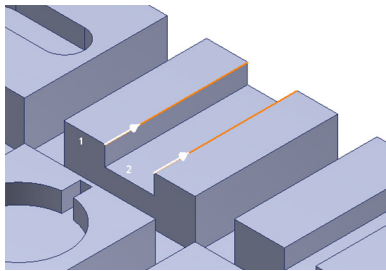


Use the same tool as in the previous operation.

.Click the **Save** icon



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**.

Click the **Save** icon.

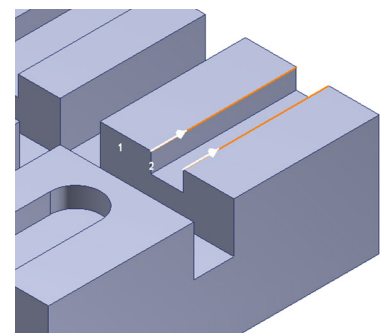


7. Open slot machined by rolling

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Roll into open slot** strategy.

Define a new geometry selecting a pair of chains as shown.



Add a new **End mill** of $\text{Ø}12$.

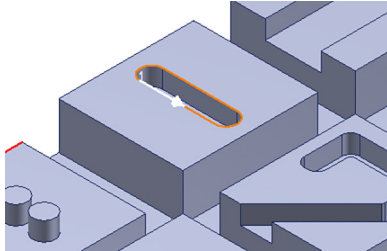
In the **Technology** page, set the **Step down** to **10**. This slot will be machined in one pass. Set the **Approach distance** to **1.25**. Under **Extension**, set **% of tool diameter** to **60**.

Click the **Save** icon.



8. 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 **End mill** of $\text{Ø}12$. Click the **Save** icon.



In the **Closed Slot** cycle, the tool diameter should be equal to the slot width.

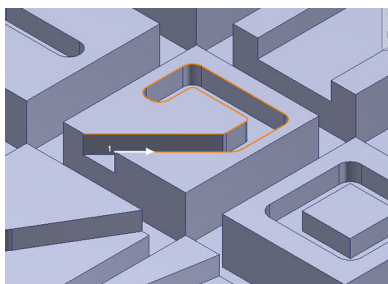


9. Spine Slot machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Spine Slot** strategy.

Define a new geometry, selecting the open chain as shown.



Define a new **End mill** tool of $\text{Ø}8$.

In the **Technology** page, set the **Step down** to **10**. This slot will be machined in one pass. Set the **Tolerance** to **1**. Under **Extension**, set **% of tool diameter** to **10**.

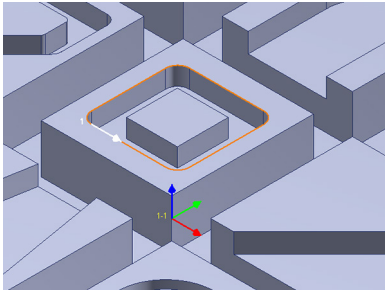


10. Temporary Bridges machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Temporary Bridges** 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**

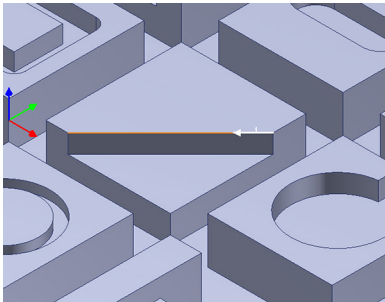


11. Simple corner machining

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 $\text{Ø}10$.

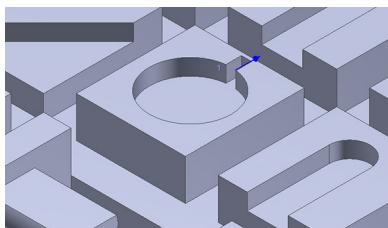


12. Broaching

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Broaching** strategy.

Define a new geometry, selecting a linear chain as shown.

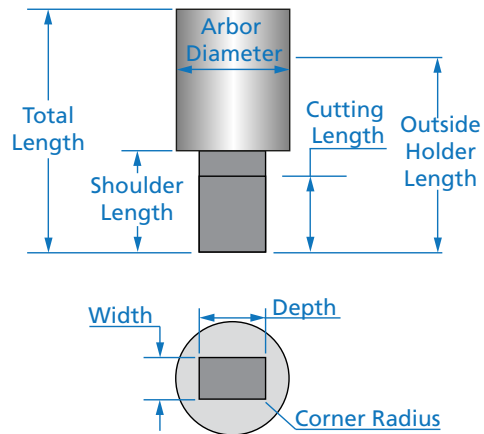


With this operation, only a Broaching tool is available.



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.



Add a new **Broaching** tool and set its **Width** and **Depth** to **6**.

In the **Technology** page, set the **Tool side** to **Left**.

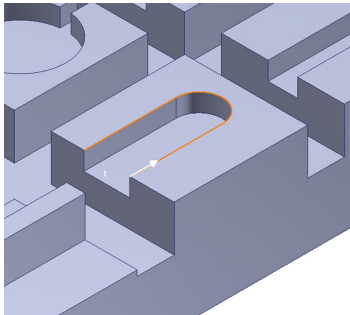


13. Closed slot machined by rolling

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Roll into closed slot** strategy.

Define a new geometry, selecting the open chain as shown.



Use the previously defined **Tool #1** of **Ø10**.

In the **Technology** page, set the **Approach distance** value to **1.25**. Under **Extension**, set **% of tool diameter** to **60**.

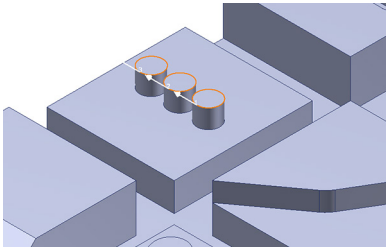


14. Simple Multi Bosses machining

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 $\varnothing 10$.

In the **Technology** page, set the **Step down** value to **10**. Under **Clear offset**, set **Offset** to **30**.

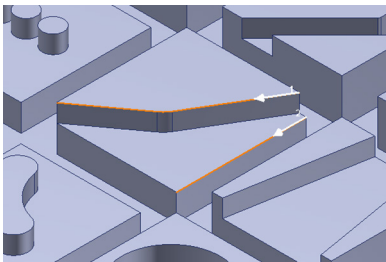


15. One Side Open Slot machining

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.

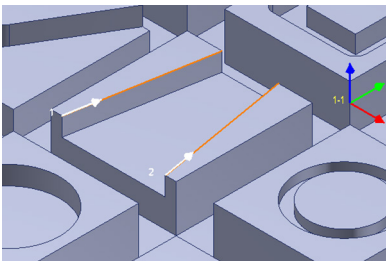


16. Ruled Surface machining

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.

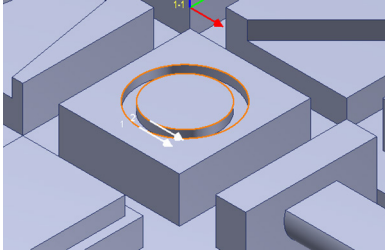


17. O-ring machining

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 $\text{Ø}4$.



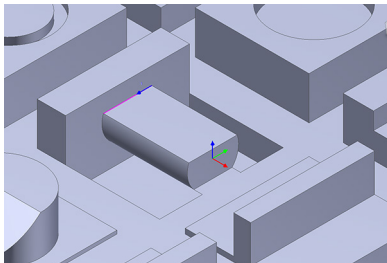
18. Surface flattening on cylinder

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 on the cylinder revolution axis.

Define a new geometry, selecting the linear open chain as shown.



Use the previously defined **Tool #1** of $\text{Ø}10$.

On the **Technology** page, in the **Clear offset** section, click the **Offset** button and pick the farther edge on the solid model. Set the **Step over** value to **3**.

Place the mouse cursor in the **Cylinder diameter** edit box and pick the external cylinder surface. The edit box is updated and set to **30**.

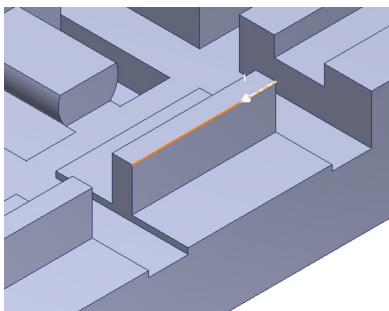


19. Rib cutting

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Rib cut** strategy.

Define a new geometry, selecting the edge as shown.



Use the same tool as in the previous operation.

In the **Technology** page, use the default parameters. The tool is shifted from the center line by 10% of its radius.



20. Surface flattening on corner

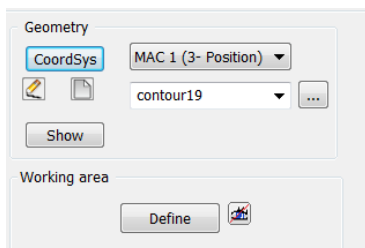
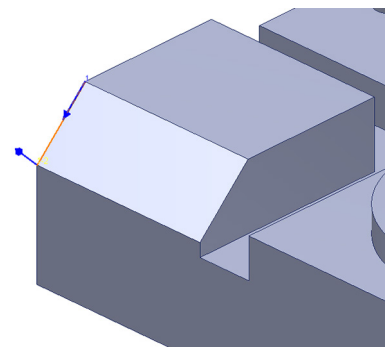
Define a new **ToolBox Cycles** operation.

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.

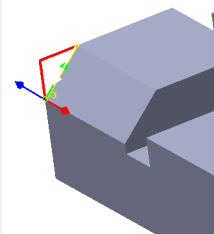
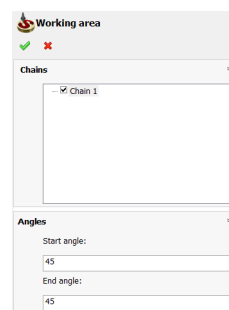


The **Working area** dialog box is displayed. Make sure the **Start angle** and **End angle** are **45°**.

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.

On the **Technology** page, in the **Clear offset** section, click the **Offset** button and pick the farther edge on the solid model. Set the **Step over** value to **2**.



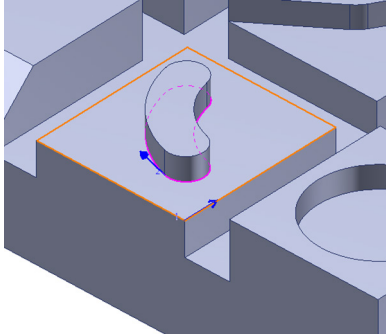


21. Simple Boss Machining

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.

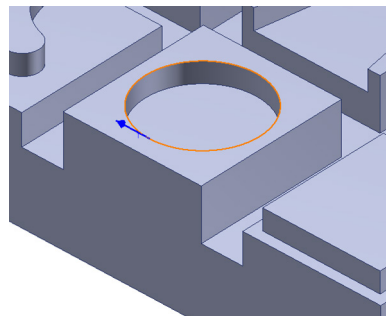


22. Spiral pocket machining

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Spiral pocket** strategy.

Define a new geometry, selecting the closed chain as shown.



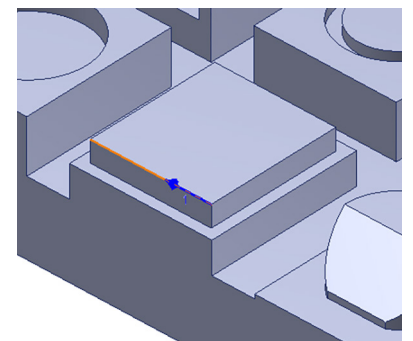
Use the same tool as in the previous operation.



23. Wood cutting

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Wood cutting** strategy.



Define a new geometry, selecting the open chain.

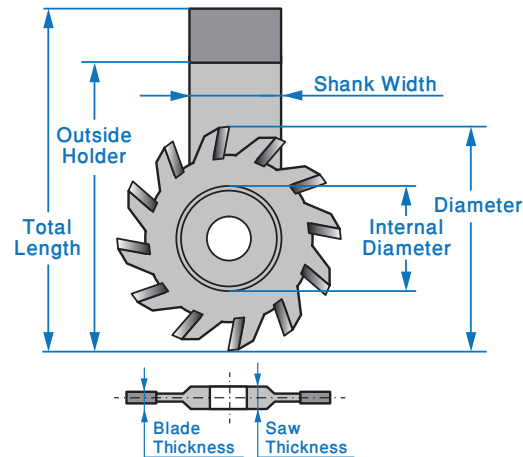
Add a new **Saw** tool of $\varnothing 40$. Set the **Shank width** to **20**.



Saw Tool

This tool type can only be used with the **Wood cutting** sub-operation. It is not available for defining or choosing in any other operation.

A tool of this type is defined with the parameters shown in the image.



24. Cylinder top cutting

Define a new **ToolBox Cycles** operation.

Under **Technology**, select the **Angled cylinder** strategy.

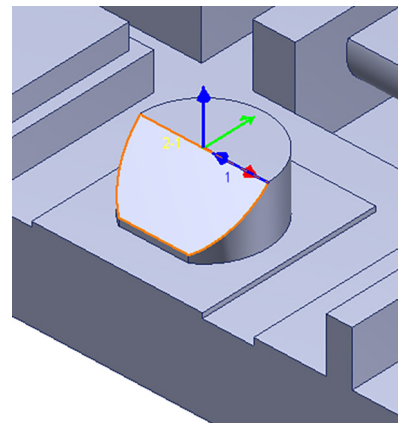
Define a new Coordinate System position with the origin located at the cylinder revolution axis.

Define a new geometry, selecting the closed chain as shown.

Use the previously defined **Tool #1** of $\varnothing 10$.

In the **Levels** page, click the **Depth** button and select the depth of the cut directly on the solid model.

In the **Technology** page, set the **Max step down** to **4**. Choose the **Zigzag** direction of cuts. Under **Extension**, set **% of tool diameter** to **60**.

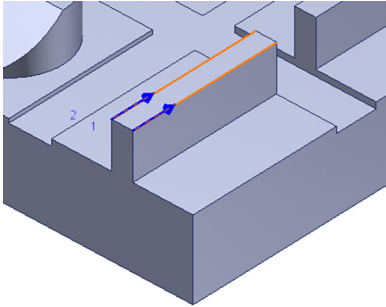




25. Thin Wall Machining

Define a new **ToolBox Cycles** operation and select the **Thin Wall Machining** strategy.

Define a new geometry, selecting two edges as shown.

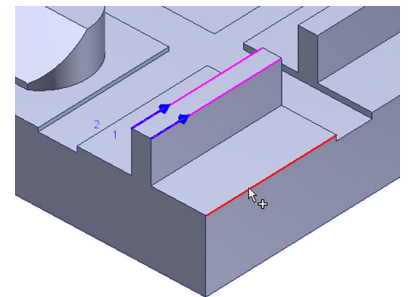


Use the same tool as in the previous operation.

In the **Technology** page, use the default **Alternating steps** strategy for roughing passes. Set the **Step down** to **6**. Click the **Offset** button to clear the offset up to the block edge.

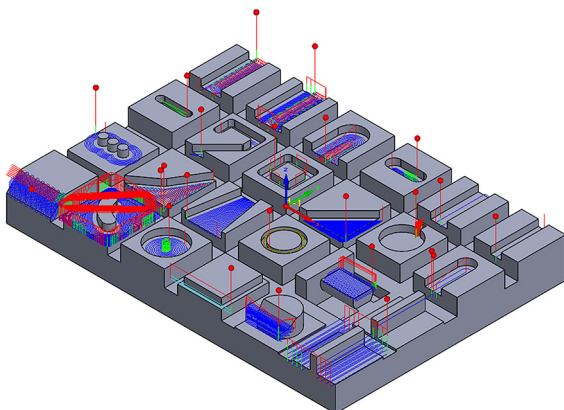
Pick the edge as shown.

Edit manually the **Offset** value to **28**.



26. Simulate the operations

Right-click the **Operations** header and choose **Simulate** to play simulation for all ToolBox operations.



Now you have successfully finished the exercise.

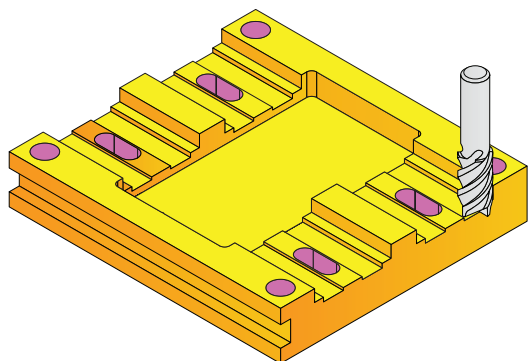
Automatic Feature Recognition

7

This chapter covers the following types of 2.5D Milling operations performed using the Automatic Feature Recognition module (AFRM) functionality:

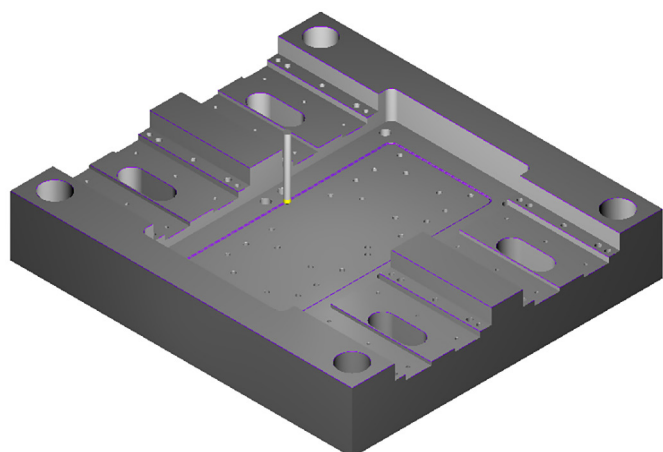
Pocket Recognition Operation

This operation enables you to recognize pocket features on a solid model and perform machining of these features.



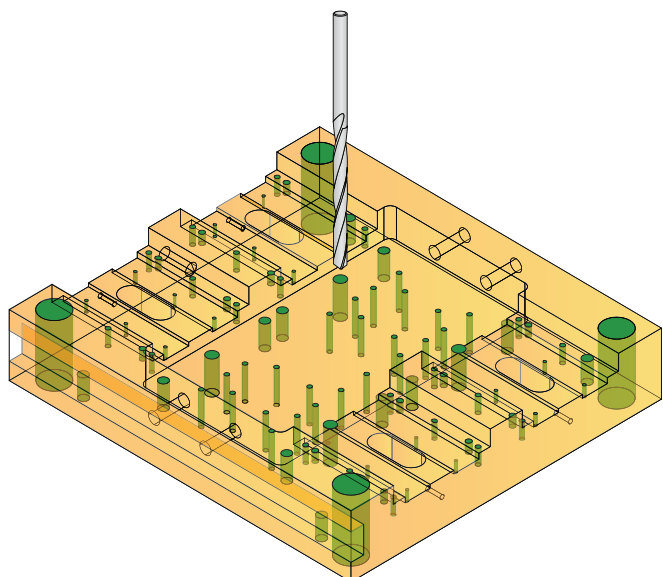
Chamfer Recognition Operation

This operation enables you to recognize the chamfer features on the solid model and perform machining of these features. The geometry creation process is similar to that of Pocket Recognition operation. The chamfer mill tool works on all edges maintaining the user-defined safety offset from the corners.



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 Pocket, Chamfer, and Drill Recognition operations is determined automatically by InventorCAM AFRM functionality.



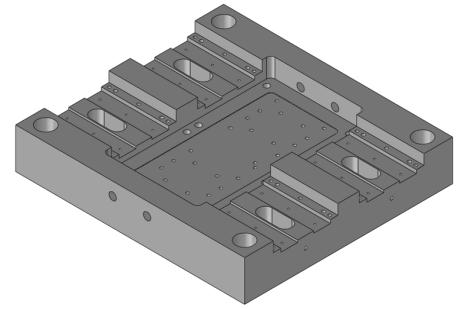
Exercise #16: Pocket Recognition

This exercise illustrates the InventorCAM 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 **Exercise16.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 **InventorCAM Manager** and choose **Pocket Recognition** from the **Add Milling Operation** submenu.

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



3. Define the Geometry

In the **Geometry** page, click  .

The **Select Faces** dialog box is displayed. This dialog box enables you to define the parameters of the pocket feature recognition geometry.

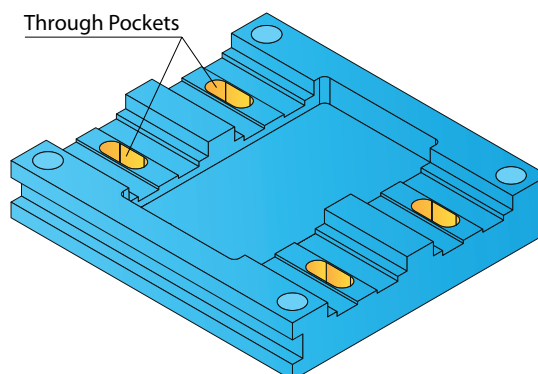
First, you need to set the pocket recognition filters and then the mode of geometry selection.

Make sure that the **Through Pocket** check box is selected.

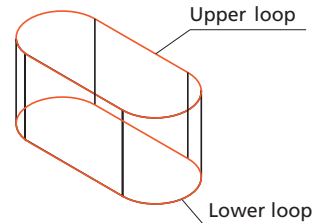


Through Pocket

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 the **Apply filter** 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.

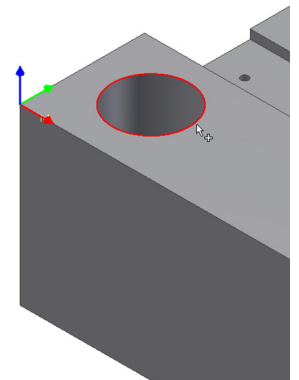
The **Min. Face area** parameter enables you to set a value to avoid picking up very small areas or faces that cannot be machined.

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.



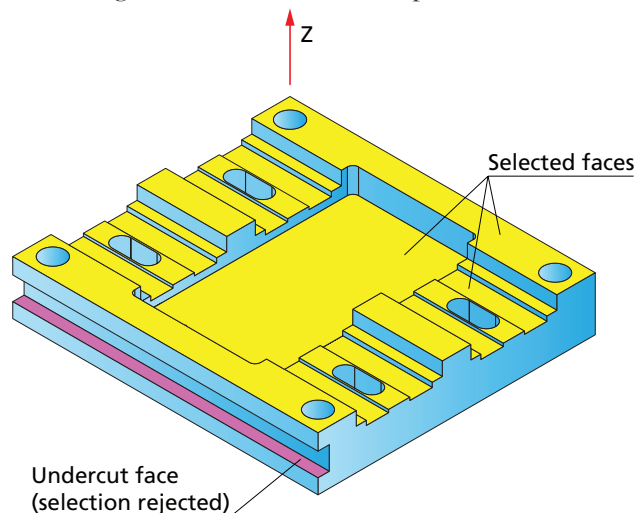
Selection Mode

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, InventorCAM 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.

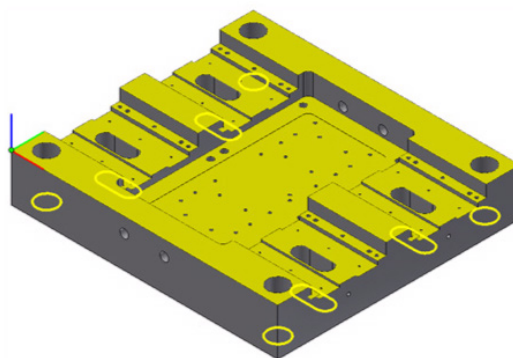
Through Pocket Chain

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. InventorCAM 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**

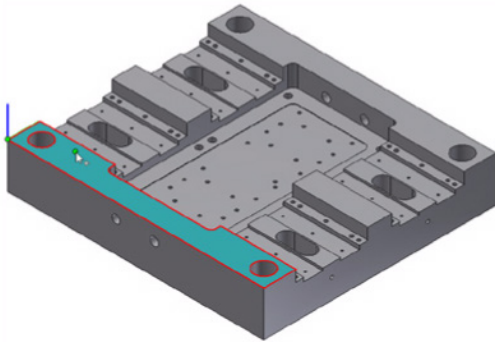
Set the **Cutting length** to **100**

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 **Max step down** value to **5**.

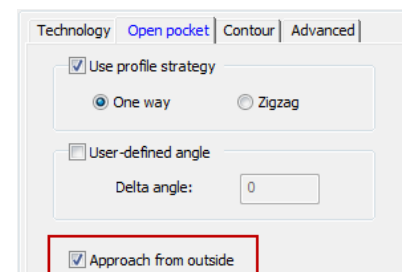


6. Define the technological parameters

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 areas 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.



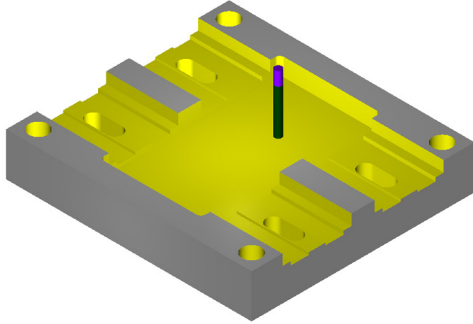
8. Calculate the tool path

Click  .



9. Simulate the operation

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



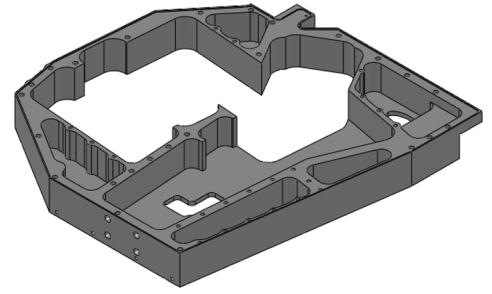
Now you have successfully finished the exercise.

Exercise #17: Mounting Box Machining

In this exercise, the InventorCAM 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 Autodesk Inventor model of the mounting box part (**Exercise17.ipt**) is located in the **Exercises** folder.

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



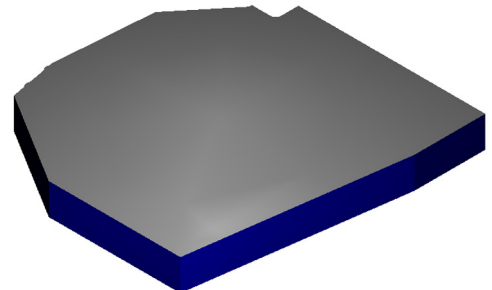
1. Define the CAM-Part

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



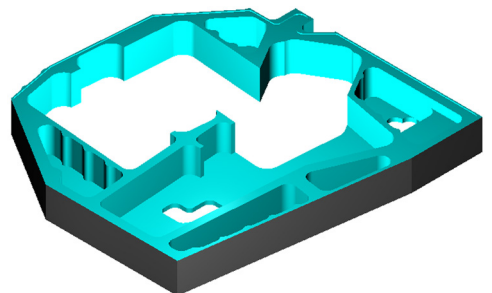
2. External contour machining

Define a Profile operation to perform rough machining of the external contour of the part. Use an **End mill** tool of **Ø20**.



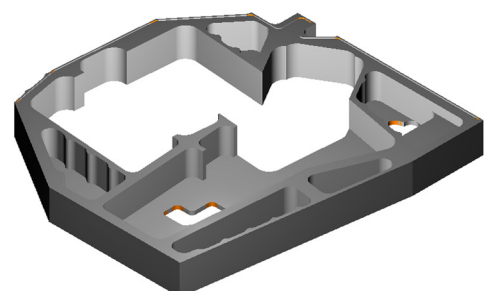
3. Pocket roughing

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**.



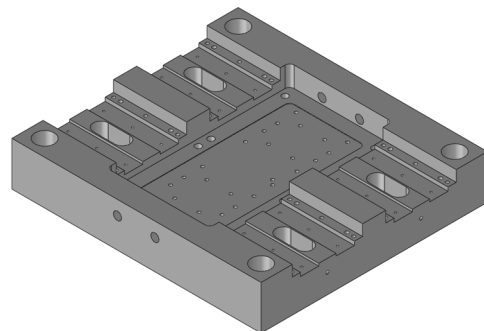
Exercise #18: Drill Recognition

This exercise illustrates the InventorCAM 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

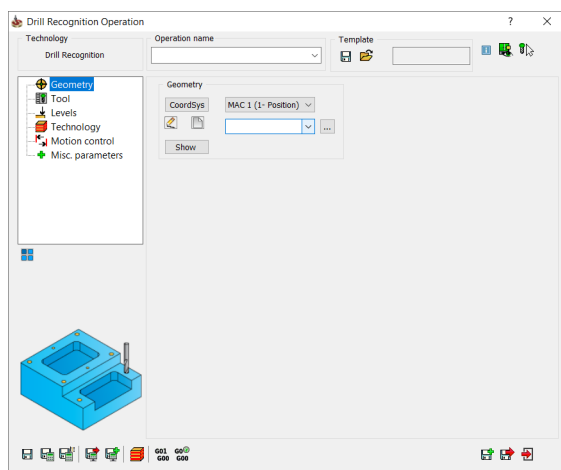
Open the **Exercise18.prz** file located in the **Exercises** folder. This file contains the CAM-Part machined in the **Pocket Recognition** exercise.



2. Add a Drill Recognition operation

Add a new Drill Recognition operation to perform center drilling of holes.

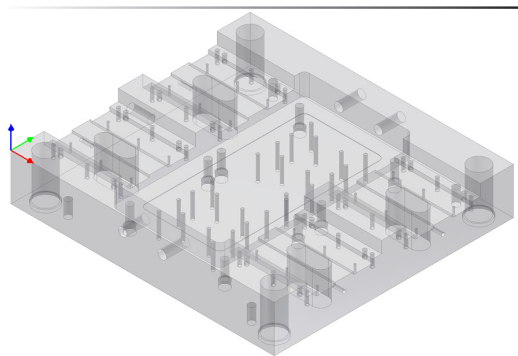
The **Drill Recognition Operation** dialog box is displayed.



3. Define the Geometry

In the **Geometry** page, click  .

InventorCAM parses the solid model and recognizes all of its hole features. The model becomes transparent so that all of its hole features are visible



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.

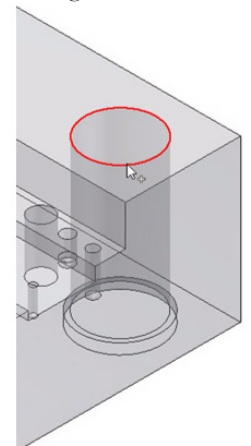
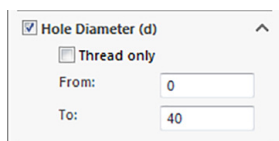
When a cylindrical surface or a circular edge is picked, its diameter is calculated and inserted into the corresponding edit box replacing the previous value.

The edit box changes the background color to indicate the associativity field (by default, pink). When you remove the automatically determined value, the edit box background turns white again.

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 changes the background color to indicate the associativity field (by default, pink).

When you remove the automatically determined value, the edit box background turns white again.

In this operation, the holes of all heights will be machined. Make sure that the **Hole height** check box is not selected.



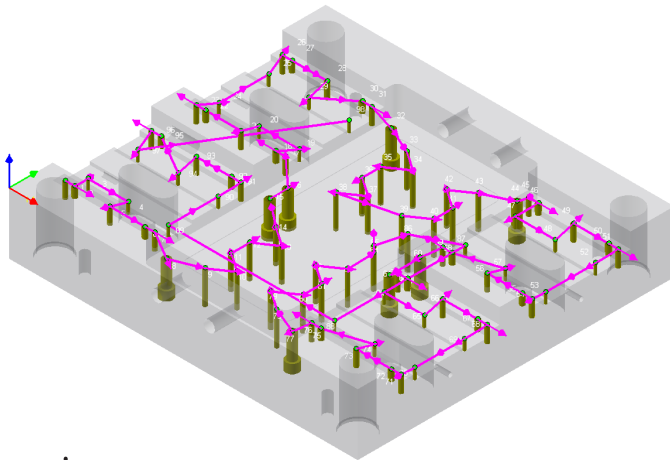
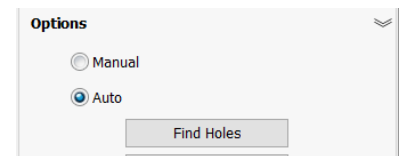
Hole Height (h)

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 changes the background color to indicate the associativity field (by default, pink). When you remove the automatically determined value, the edit box background turns white again.

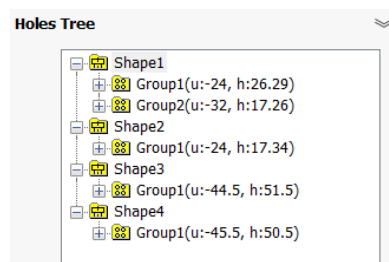
Click the **Find Holes** button. The hole features that match the specified filter criteria are determined. These features are listed in the **Holes Tree** section.

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**

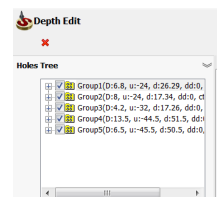
Choose the tool for the operation.



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

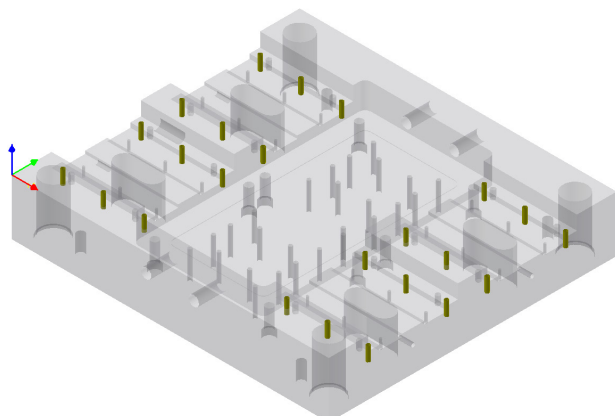
This section displays the list of all drill instances chosen for the geometry. All the drill instances in the list are structured into **Groups** with the same **Upper Level**, **Drill Depth**, **Delta Depth** and **Depth Type**. For each **Group** in the list, the drill instances data are displayed 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/Chamfer 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). InventorCAM 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, InventorCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters. InventorCAM 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, the program 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 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.

Auto Chamfer/Spot drill

When this option is selected, the **Drill depth** section changes to **Chamfer depth**. The drill tip reaches the drilling depth, which is calculated automatically, according to the value set in the **Chamfer depth** section. To apply the **Spot drill** that only touches the hole edges, set this value to **0**. To apply the **Auto Chamfer**, set a defined value in the **Chamfer depth** field.

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, InventorCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters. The program checks all groups to find a group with identical **Upper Level**, **Drill Depth**, **Delta Depth** and **Depth Type** parameters. If such group is found, the program 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 **Auto Chamfer** option for **Depth Type**. The **Drill depth** section name changes to **Chamfer depth**. This type should be applied to all hole features in the operation. Select the **All** checkbox 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 of the chamfering.



Drill/Chamfer Depth (d)

This value can be applied to a single hole, a group or all holes.

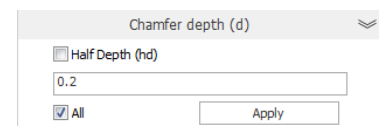
The edit box displays the **Drill/Chamfer depth** value of the selected item (a group or a hole). The edit box enables you enter the value or pick the value directly from the model (when the cursor is placed in the edit box). You can pick drill faces or edges. The program determines the depth of the 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, InventorCAM checks the **Holes Tree** items and reorganizes them into groups according to the changed parameters. The program checks all groups to find a group with identical **Upper Level**, **Drill Depth**, **Delta Depth** and **Depth Type** parameters. If such group is found, the program 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 **Chamfer depth** value to **0.2**, select the **All** check box and click the **Apply** button.

Apply the changes and confirm the **Depth Edit** dialog box with .

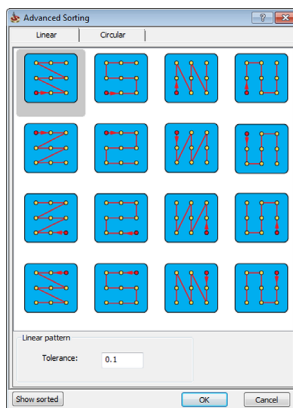


6. Define the sorting of drill positions

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.

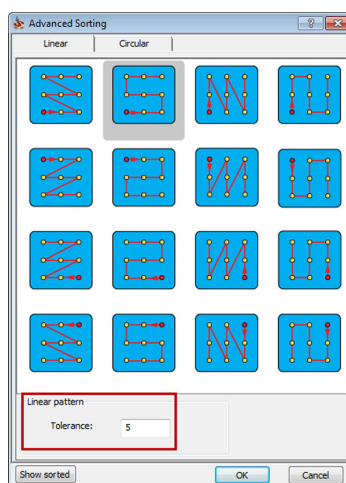
Linear pattern

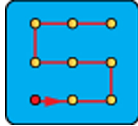
According to linear sorting methods, the machining is performed either by rows of drill positions with the same Y-coordinate or by columns of drill positions with the same X-coordinate. The **Tolerance** parameter enables you to define the tolerance that is used to determine if a drill position belongs to the current row/column or to a different row/column.

When the machining is performed by rows, InventorCAM 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, the program 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, the program 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.

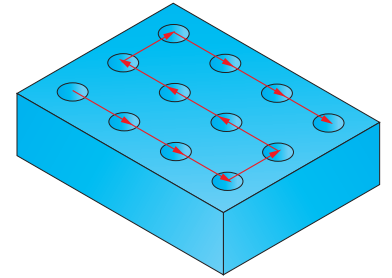
In the similar manner, when the machining is performed by columns, the specified **Tolerance** value is used to determine whether a specific drill position belongs to the current column. In this case, the deviation of X-coordinates between the first drill position in the column and the current drill 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, InventorCAM determines the position with the **minimal X- and Y-coordinates** and accepts it as the start position of the drill sequence. From this position, the program continues the machining of the drills with the same Y-coordinate; the machining is performed in the direction of increasing X-coordinates.

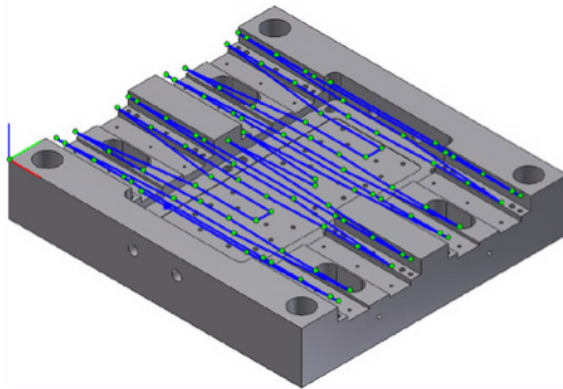


When the last position of the current Y-coordinate is machined, the program passes to the next Y-coordinate (in the increasing order) and continues the machining of the next row from the position with the maximal X-coordinate to the position with the minimal X-coordinate (decreasing order), and so on. 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 .



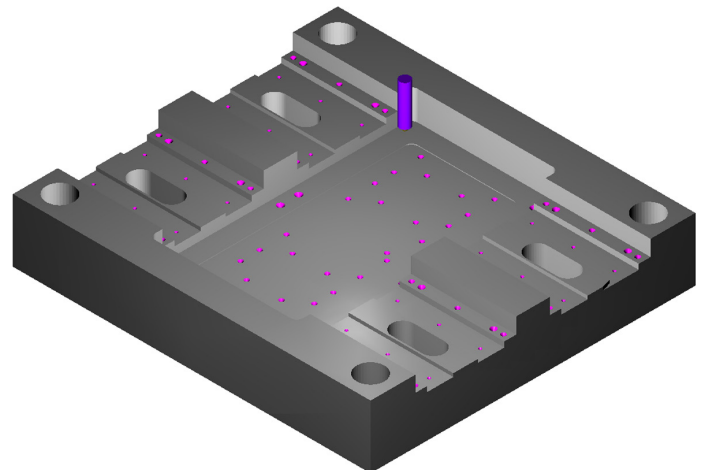
7. Calculate the tool path

Click .



8. Simulate the operation

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





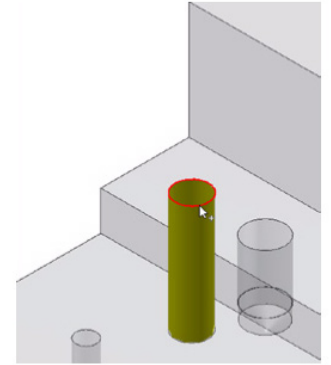
9. Machine the Ø6.8 holes

Add a new Drill Recognition operation to perform drilling of the **Ø6.8** holes.

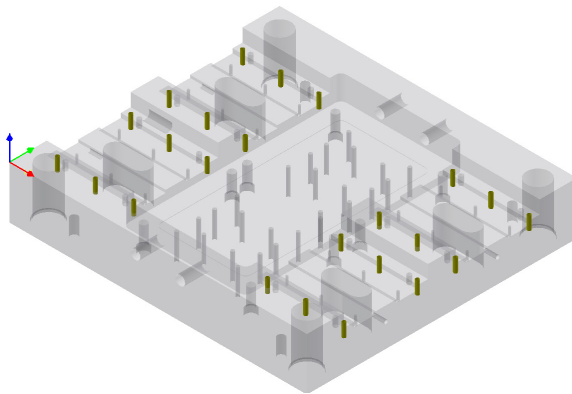
In the **HR Drill Geometry Selection** dialog box, activate the **Hole Diameter** section. Place the cursor in the **From** edit box and pick the hole edge as shown.

The diameter value (**6.8**) is displayed in both **From** and **To** edit boxes defining the range for the **Hole Diameter** filter.

<input checked="" type="checkbox"/>	Hole Diameter (d)	
<input type="checkbox"/>	Thread only	
From:	<input type="text" value="6.8"/>	
To:	<input type="text" value="6.8"/>	

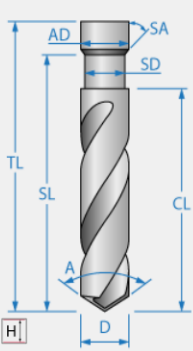


Click the **Find Holes** button. The holes of the specified diameter are highlighted on the model.



Confirm the **HR Drill Geometry Selection** dialog box by clicking .

Add a new **Ø6.8 Drill** tool.

Tool parameters		
Diameter (D):	<input type="text" value="6.8 mm"/>	
Angle (A):	<input type="text" value="118°"/>	
Shoulder diameter (SD):	<input type="text" value="6.8 mm"/>	
Arbor diameter (AD):	<input type="text" value="6.8 mm"/>	
Shoulder angle (SA):	<input type="text" value="0°"/>	
Cutting (CL):	<input type="text" value="50 mm"/>	
Shoulder length (SL):	<input type="text" value="60 mm"/>	
Total length (TL):	<input type="text" value="80 mm"/>	
Outside holder (OHL):	<input type="text" value="60 mm"/>	
Helical angle:	<input type="text" value="45 (Medium)"/>	
Number of flutes:	<input type="text" value="2"/>	

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 on the model.

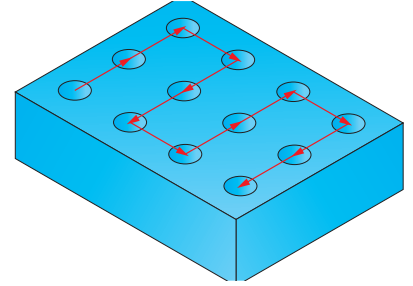
Confirm the **Depth Edit** dialog box dialog box by clicking .

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.



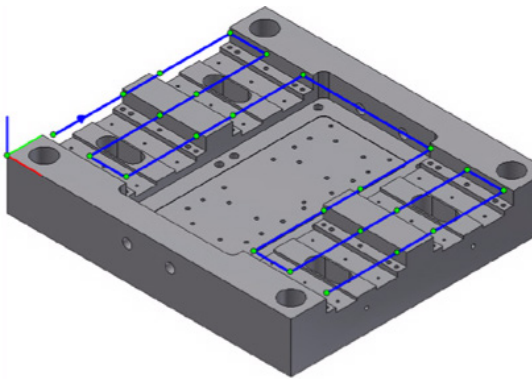
When this option is chosen, InventorCAM determines the position with the **minimal X- and Y-coordinates** and accepts it as the start position of the drill sequence. From this position, the program continues the machining of the drills with the same X-coordinate; the machining is performed in the direction of increasing Y-coordinates.



When the last position of the current X-coordinate is machined, the program 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.

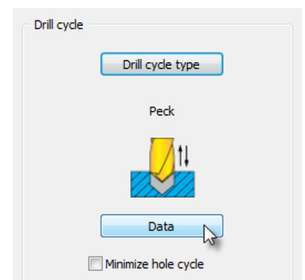
Click the **Show Sorted** button to preview the sorted drill positions on the solid model.



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.

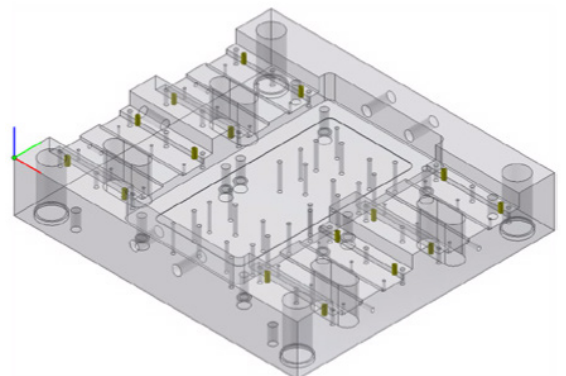


10. Machine the Ø8 holes

Add a new Drill Recognition operation to perform drilling of the **Ø8** holes.

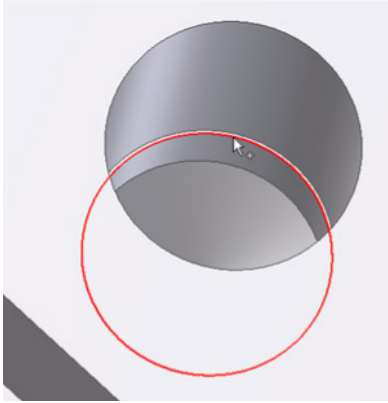
In the **HR Drill Geometry Selection** dialog box, activate the **Hole Diameter** section. Define the range for the **Hole Diameter** filter as explained in the previous step. The values of **8 mm** are displayed in the **Hole Diameter** section.

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.



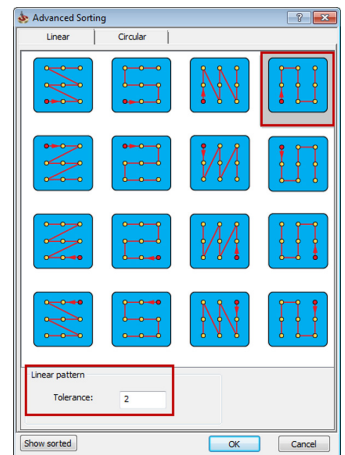
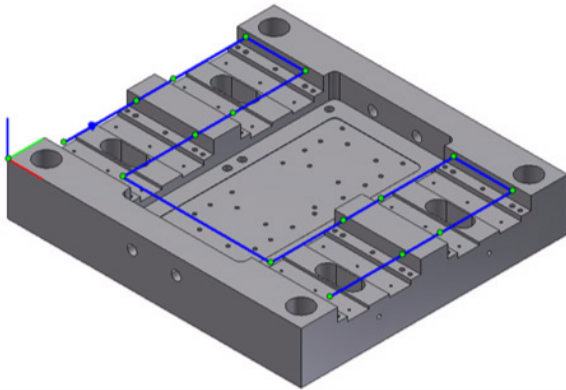
The **Diameter** value (**7.8**) is displayed in the edit box.

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.

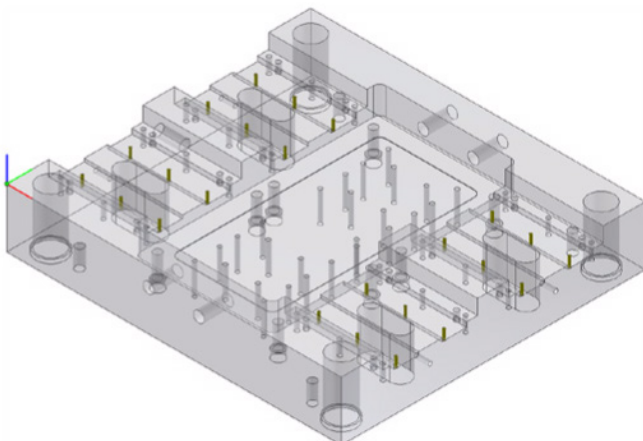
Under **Drill cycle**, choose the **Peck** cycle type and set the **Step down** value to **2**.

Save and calculate the operation.



11. Machine the $\varnothing 4.2$ holes


Add a new Drill Recognition operation to perform drilling of the $\varnothing 4.2$ holes.

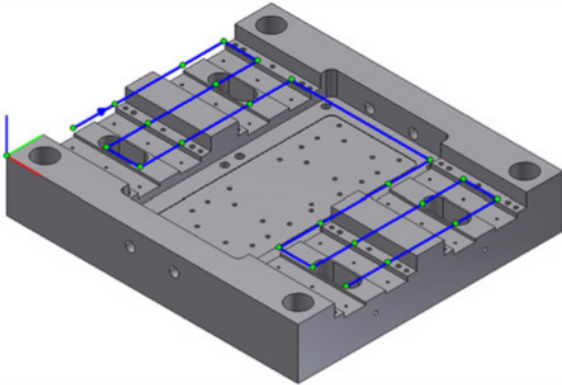


In the **HR Drill Geometry Selection** dialog box, define the range for the **Hole Diameter** filter as explained in the previous steps. The values of **4.2 mm** will be displayed in the **Hole Diameter** section.

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.



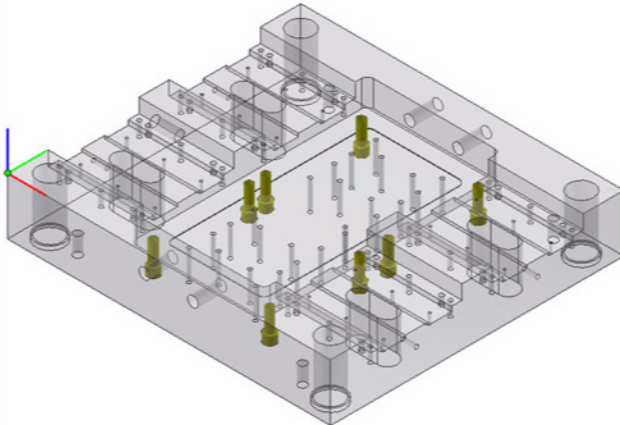
Under **Drill cycle**, choose the **Peck** cycle type and set the **Step down** value to **2**.

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.

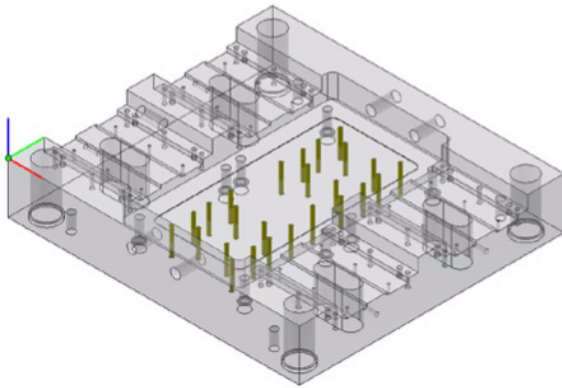


Use the **Default** drill sorting option.

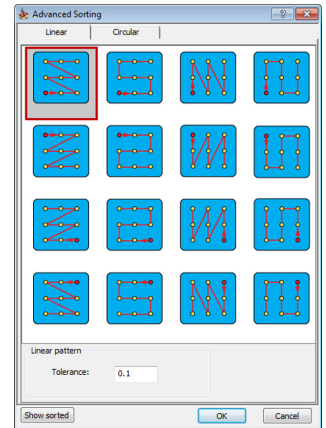


13. Machine the $\varnothing 6.5$ holes

In the same manner as explained in the previous steps, perform drilling of the set of $\varnothing 6.5$ holes.



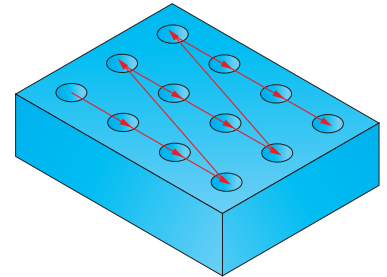
Use the **Advanced** option of **Advanced** drill sorting.



When this option is chosen, InventorCAM determines the position with the **minimal X- and Y-coordinates** and accepts it as the start position of the drill sequence. From this position, the program continues the machining of the drills with the same Y-coordinate; the machining is performed in the direction of increasing X-coordinates.

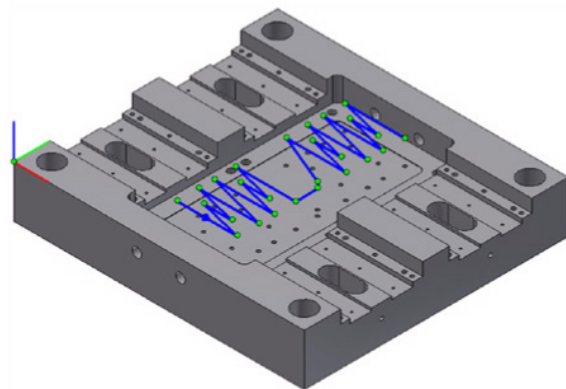


When the last position of the current Y-coordinate is machined, the program passes to the next Y-coordinate (in the increasing order) and continues the machining of the next row from the position with the minimal X-coordinate to the position with the maximal X-coordinate (increasing order) and so on. 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.

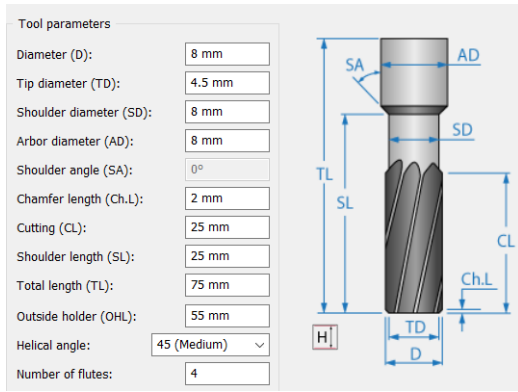


14. Perform reaming of the $\varnothing 8$ holes

Add a new Drill Recognition operation to perform reaming of the $\varnothing 8$ holes.

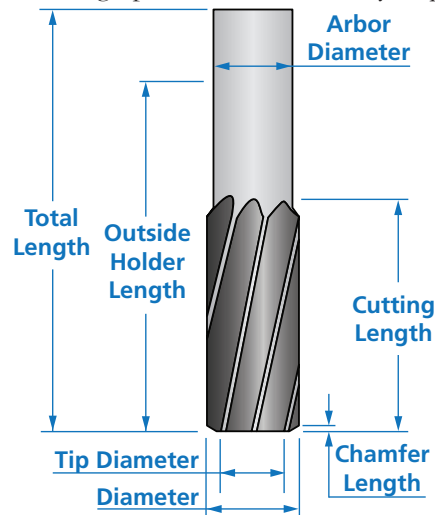
Use the **Drill_r2** geometry defined in **Step #**.

Add a new **Ø8 Reamer** tool.



Reamer

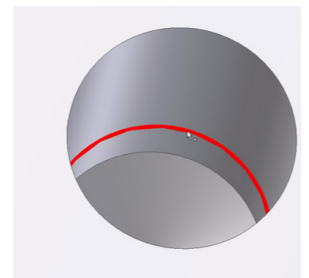
This tool is used to machine precise holes in Drilling operations. A tool of this type is defined with the parameters shown in the image. Selecting a reamer tool in a drilling operation, automatically displays the correct default cycle for reamers.



To define the depth, pick the model face as shown.

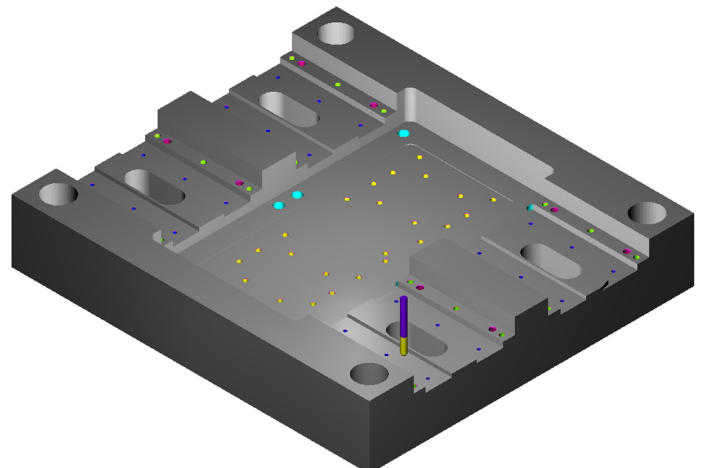
The **Drill depth** value (**12**) is displayed. Click the **Apply** button.

Define the sorting of drill positions as explained in **Step #**. Click .



15. Simulate the operations

Simulate the operations defined in **Steps #9 – #13** in the **SolidVerify** mode.



Now you have successfully finished the exercise.

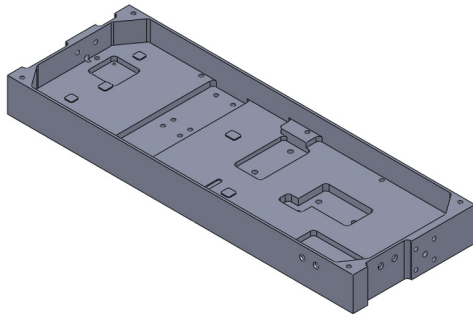
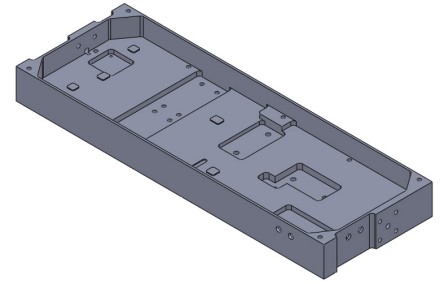
Exercise #19: Electronic Box Machining

In this exercise, the InventorCAM functionality of automatic drill recognition and machining is used for rough milling of the electronic box hole features.

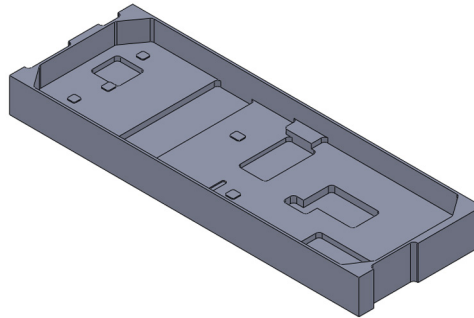
This exercise reinforces the following skills:

- Automatic drill recognition and machining

The electronic box part file (**Exercise19.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



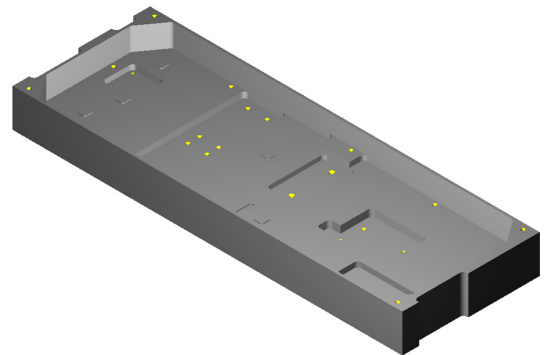
Stock

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



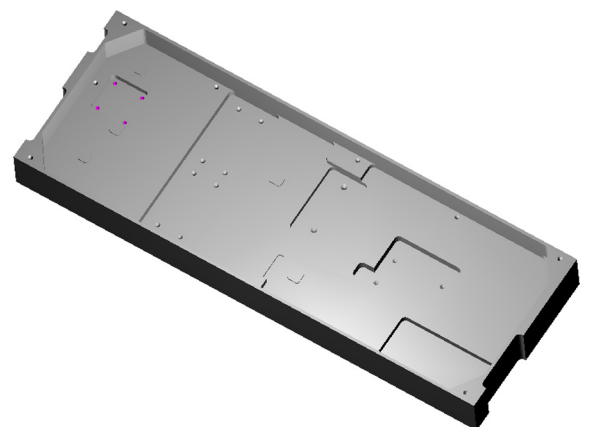
1. Center drilling

Define a Drill Recognition operation to perform center drilling of all hole features. Use a **Spot drill** tool of $\text{Ø}2.5$.



2. Drilling

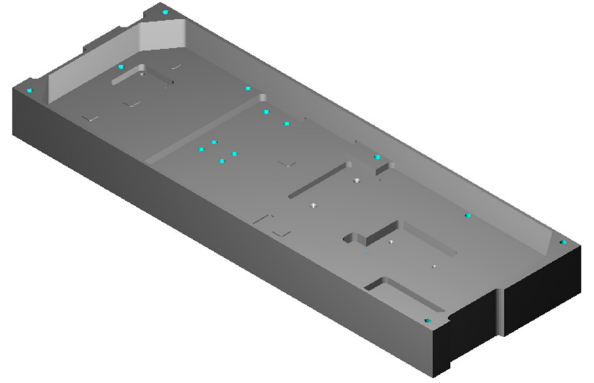
Define a Drill Recognition operation to perform drilling of the $\text{Ø}1.2$ holes. Use a **Drill** tool of $\text{Ø}1.2$.





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 $\varnothing 2.2$.



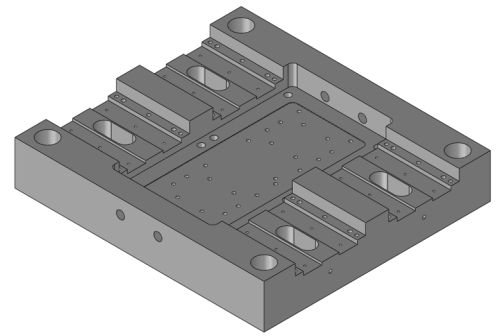
Exercise #20: Edge Deburring Recognition

This exercise illustrates the InventorCAM functionality of automatic chamfer recognition and machining.

This exercise reinforces the following skills:

- Automatic chamfer 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 **Exercise20.prz** file located in the **Exercises** folder. This file contains the CAM-Part machined in the **Pocket Recognition** and **Drill Recognition** exercises.



2. Add a Chamfer Recognition operation

Right-click the **Operations** header in **InventorCAM Manager** and choose **Edge Deburring Recognition** from the **Add Milling Operation** submenu.

The **Edge Deburring 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 chamfer feature recognition geometry.

First, you need to set the chamfer recognition filters and then the mode of geometry selection.

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 for chamfering.

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 **Ignore edges followed by fillet** check box is selected, chamfering will not be performed on any solid model edge featuring a fillet.

When the **Ignore floor-wall angle less than** check box is selected, chamfering will not be performed in the already existing chamfers at the edges of floors and/or the edges of angled walls.

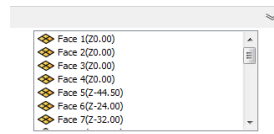
When the **Apply filter** check box is not selected, all recognized circular pocket areas suitable for the current Coordinate System are selected.

Set the **Min. Diameter** value to **20**. All of the holes with the diameter smaller than the specified value will not be chamfered.

Define the mode of geometry selection. Under **Selection mode**, use the default **Solid Body** option.

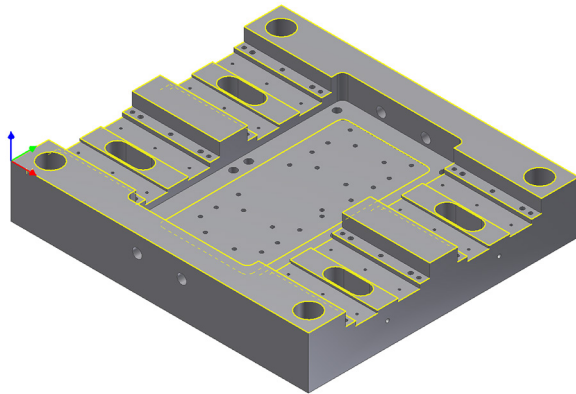
Click on the solid model. InventorCAM generates the list of elements that suit your filter settings.

Click the **Solid1** element in the list. All the recognized part in this geometry are highlighted. The geometry through pockets of slots, and circular through pockets of element is accompanied by its Z-level value enclosed in



model features that take includes all the planar faces, column holes. Each model parentheses.

Confirm the **Select Faces** dialog box by clicking .



4. Define the Tool

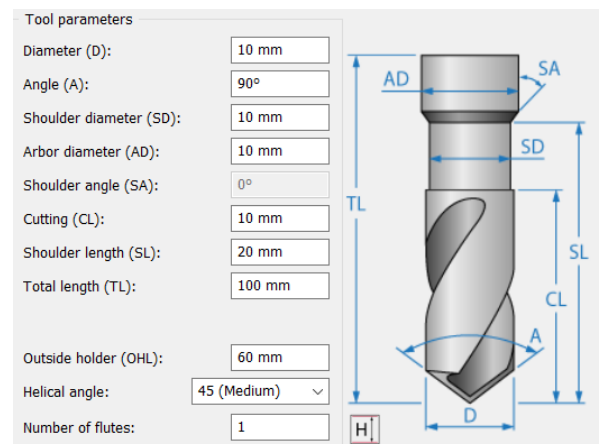
Add a **Chamfer mill** of $\varnothing 10$ for the operation. Define the tool parameters as follows:

Set the **Total length** to **100**

Set the **Outside holder length** to **60**

Choose the **HSK A 63 ER 40x80** collect chuck.

Select the tool for the operation.



5. Define the machining levels

Define the **Chamfer depth** by setting the value to **0.5**.



6. Define the technological parameters

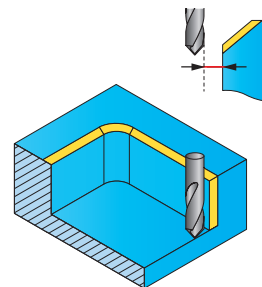
Switch to the **Technology** page. Set the **Cutting diameter** value to **2**. The chamfer depth will be reached with the tool diameter of 2 mm.



Safety offset

For geometries that consist of open chains, **Safety offset** is the offset that prevents the tool gouging with the vertical wall.

InventorCAM automatically calculates the optimal value of the **Safety offset**.



Set the **Safety offset** value to **2**.



7. Define the Link in and Link out

In the **Link** page, select the **Normal** option from **Lead in** list and set the **Normal length** to **3**. Select the **Same as Lead in** check box in the **Lead out** section.



8. Calculate the tool path

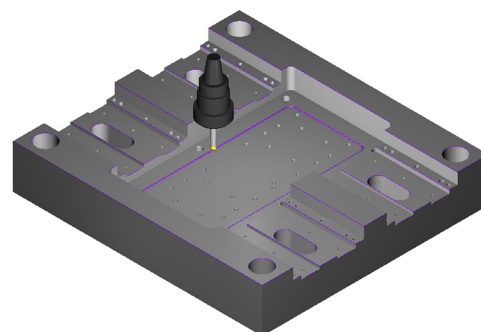
Click  .



9. Simulate the operation

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

Now you have successfully finished the exercise.



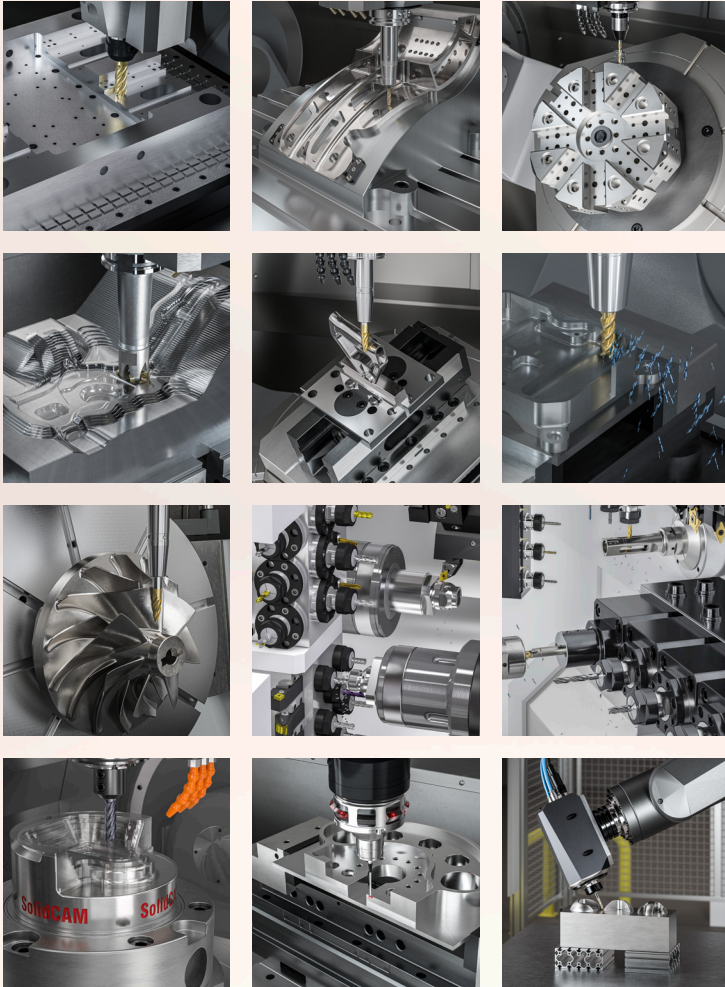
Congratulations!

You have successfully finished the **2.5D Milling** part of the **InventorCAM 2023 Milling Training Course**.



InventorCAM

The Solid Platform for Manufacturing



The Complete CAM Solution, with Revolutionary iMachining, best support for Multi-Channel Mill-Turn and Swiss CNCs, seamlessly Integrated in and associative to SOLIDWORKS®, Inventor® and Solid Edge®.

- iMachining 2D
- iMachining 3D
- 2.5D Milling
- Indexial Multi-Sided
- HSS Machining
- 3D High Speed Milling
- Simultaneous 5-Axis Turning
- Advanced Mill-Turn
- Swiss-Type
- Solid Probe

The unique, revolutionary Milling technology
imachining®
patent by SolidCAM

MILLTURN + SWISS

solidcam.com