Pre-Competition Activity Pack

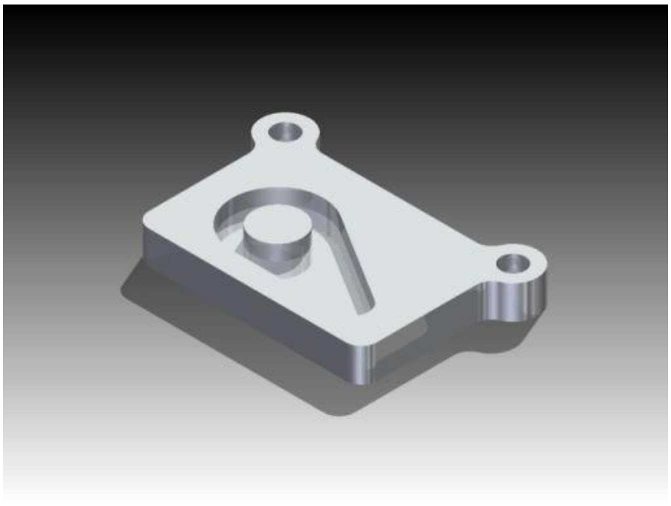
CNC Milling – Inventor HSM

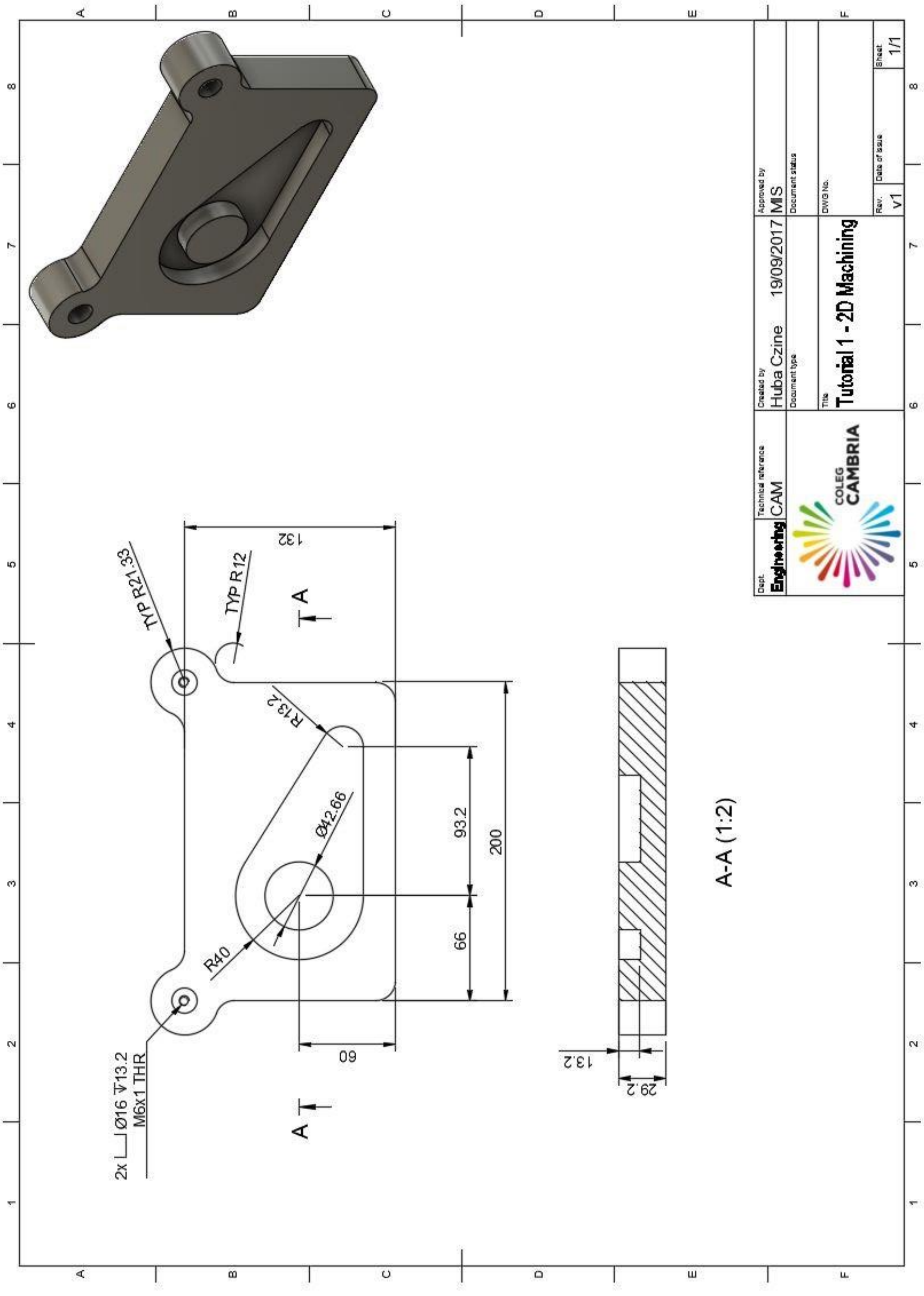
Tutorial 1 – 2D Machining

In this tutorial, you will learn how to machine the part shown above using the following procedures:

- Facing
- Contouring
- Pocket Machining
- Counterboring
- Drilling
- Tapping
- Post Processing

Before proceeding, please create a 3D model of the part on the picture below. Drawing can be found on the next page.




TYP R21.33
TYP R12
132
R13.2
Ø42.66
R40
93.2
200
66
60
2x ⌴ Ø16 ↧13.2
M6x1 THR
A
A
13.2
29.2
A-A (1:2)
Dept.
Engineering
Technical reference
CAM
Created by
Huba Czine
19/09/2017
Approved by
MIS
Document type
Document status
Title
Tutorial 1 - 2D Machining
DWG No.
Rev.
v1
Date of issue
Sheet
1/1
COLEG
CAMBRIA

To Perform Facing

The tutorial begins with a facing operation to clear the top face of the stock and ensure that it is completely flat.

1. On the ribbon, click CAM tab ➤ 2D Milling panel ➤ Face .

This creates a new facing operation and opens the **Operation** dialog box where you can edit the individual parameters controlling the toolpath, as well as select the actual geometry to machine.

Each tab on the **Operation** dialog box is divided into a number of groups. In this tutorial, the necessary settings are changed in each appropriate group as you go along.

 Tool tab

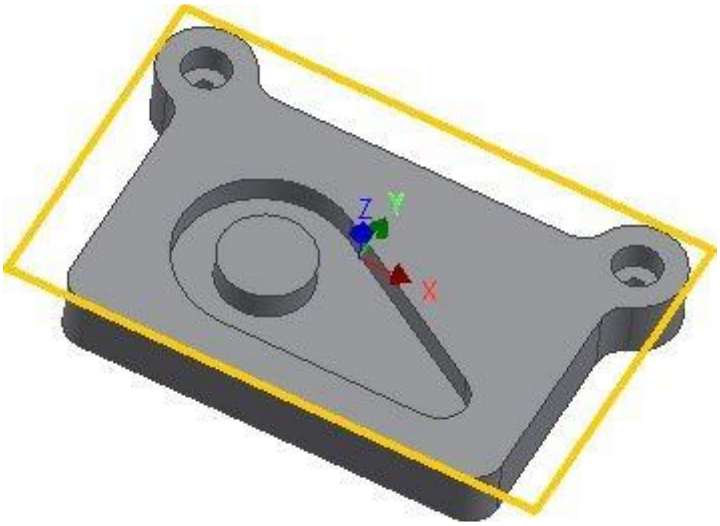
1. On the **Tool** tab, click .
2. This opens the Tool Library where you can select from existing tools in a library or define a new tool.
3. From the **Sample Libraries > Tutorial** tool library, select tool **#1 - Ø50 mm face**.
4. Click  to close the **Tool Library** dialog.

 Geometry tab

1. Click the **Geometry** tab.

The **Face** strategy automatically detects the size of the stock which is shown as an orange outline on the part.

In this example, the default stock and stock contours work fine and so the stock size need not be specified manually.



Automatically detected stock size

Passes tab

The parameters on the **Passes** tab control how the actual facing toolpath is laid out. The **Pass extension** setting specifies the distance to extend the passes beyond the machining boundary.

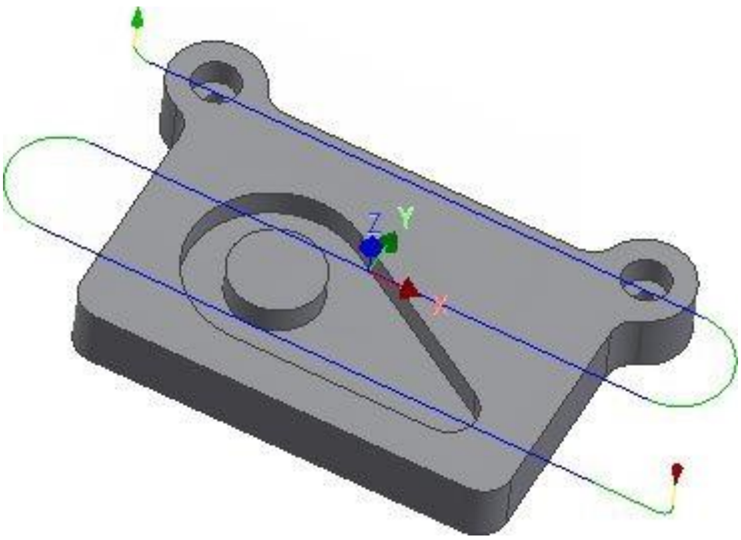
1. Click the **Passes** tab.
2. Change **Pass extension** to: 5 mm

Start the Calculation

1. Click 

at the bottom of the **Operation** dialog box, or right-click in the graphics window and select **OK** from the marking menu, to automatically start calculating the toolpath.

The toolpath is now calculated and a preview appears in the graphics window.



By default, the cutting parts of a toolpath are coloured in blue, lead moves in green, and rapid moves in yellow. The start and end of the toolpath are indicated by a red and a green triangle respectively.

To Contour the Part

Next, run a contouring toolpath along the outside of the part.

1. On the ribbon, click CAM tab ➤ 2D Milling panel ➤ 2D Contour.

Tool tab

1. On the **Tool** tab, click Tool.
2. Remember: Tool definitions can be saved in a library or just for the part you work on. In this example, we will save the tools in the part only. You can always copy the tools to a library at a later time, if you wish to re-use them.
3. Add a new tool with modified dimensions.
4. Click New Mill Tool.

5. You can use the default tool type and dimensions (an Ø8 mm flat mill) for this tutorial, but increase the flute length to be able to cut the entire height of the part (22 mm).
6. Click the **Cutter** tab.
7. Change **Flute length** to: 25mm
8. Attention: If you decide to execute this toolpath on a machine tool, ensure that the tool number corresponds with the tool position in your tool changer on the machine tool, i.e. that on position 6 you have an 8 mm flat end mill cutter.
9. Click the **OK** button to create your new tool.
10. Click the **Select** button to select the tool for your operation and close the **Tool Library** dialog.

Feed & Speed Group

Expand the **Feed & Speed** group and change the following parameters:

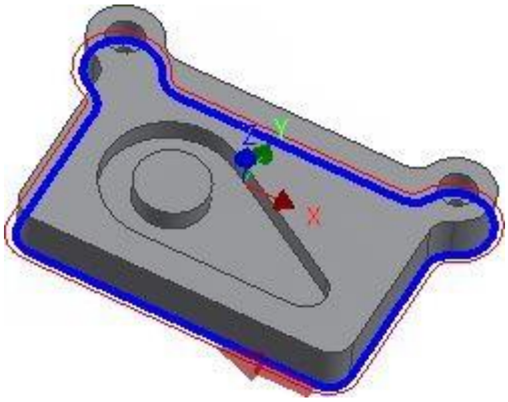
1. **Spindle speed** to: 3000 rpm
2. **Cutting feedrate** to: 800 mm/min

Attention: If you decide to execute this toolpath on a machine tool, you will also need to set the feed & speed parameters according to the material you are using, and the capabilities of your tool and machine.

Geometry tab

1. Click the **Geometry** tab. Ensure that the **Contour selections** button is active so that you can select the outside edge of the part geometry to run the tool around.

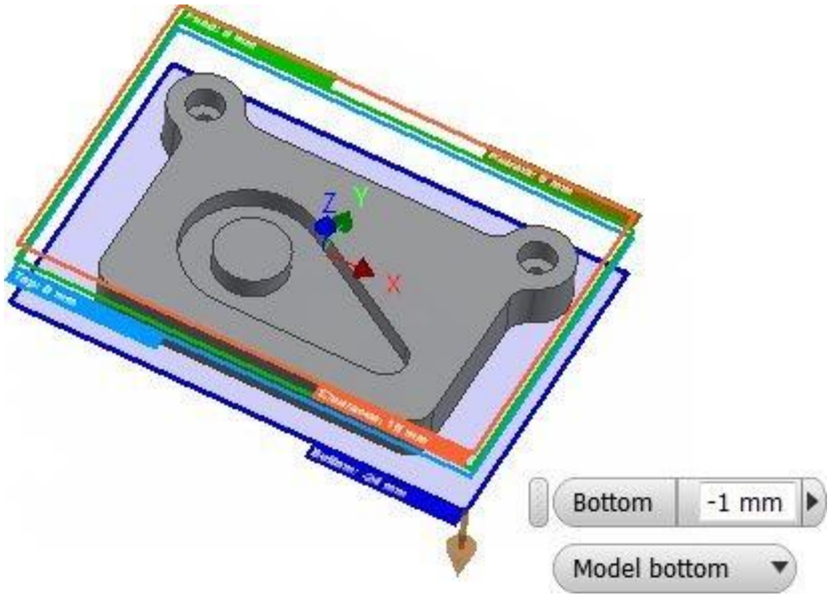
2. Move the mouse over the bottom front edge. When it highlights, click on it. The bottom contour of the part is chain selected automatically.
3. An arrow then appears near the chained contour indicating the direction of the toolpath. You can reverse the direction of a selected edge by simply clicking the arrow.



4. Do not close the **Operation** dialog box if you are satisfied with your selection, as that will end the **2D Contour** operation. Instead, simply proceed to the next step of this tutorial.

Heights tab

1. Click the **Heights** tab. A preview of the heights is shown.



*Preview of heights: The top orange plane represents the **Clearance Height**. The second olive green plane represents the **Retract Height**. The green plane represents the **Feed Height**. The light blue plane is the **Top Height**. The dark blue plane is the **Bottom Height**.*

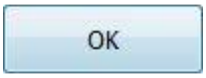
To make sure that the tool gets all the way through the stock, lower the bottom height by 1 mm.

2. From the **Bottom Height** drop-down menu, select *Model bottom*.
3. Change **Bottom offset** to: -1.0 mm

Observe that the preview plane moves in the graphics window.

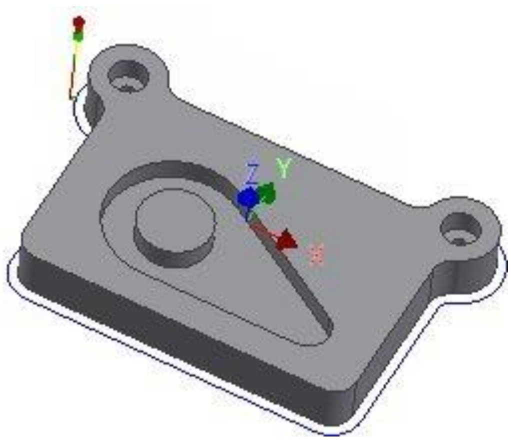
Tip: Observe that the various heights can also be adjusted using the mini-toolbar. Simply click the heights plane you wish to change and dynamically drag the arrow manipulator to the value you want. You can also enter the value directly in the mini-toolbar text field.

Start the Calculation

1. Click 

at the bottom of the **Operation** dialog box, or right-click in the graphics window and select **OK** from the marking menu, to automatically start calculating the toolpath.

The toolpath is calculated and shown in the graphics window.



Important: Notice that in the **CAM Browser**, a new Setup has been created automatically. This is automatically done when creating a new operation before a setup has been created. A setup defines a number of settings used in all the operations contained within the setup. For example, the WCS can be changed in the setup.

Machine the Pocket

The next operation is to machine the pocket with the central circular boss on the top surface of the part.

1. On the ribbon, click CAM tab ➤ 2D Milling panel ➤ 2D Pocket .

Tool tab

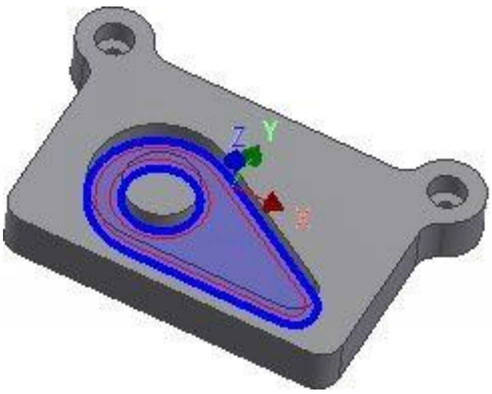
1. On the **Tool** tab, click the 

button and select the tool **#2 - Ø8 mm flat** from the library.
2. Click 

to close the **Tool Library** dialog.

Geometry tab

1. Click the **Geometry** tab. The contours of the pocket to be cleared are selected here.
2. Click anywhere on the face at the bottom of the pocket.



3.
4.
5. Note: Selecting faces for 2D geometry automatically uses all edges of the face for the contours. However, if two adjacent faces are selected, the edges they share are *not* included in the selection.

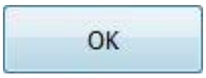
 Heights tab

1. By default, the **2D Pocket** operation machines from the top of the stock to the level of the selected contours. This is exactly what is needed in this operation, so there is no need to change any heights.

 Passes tab

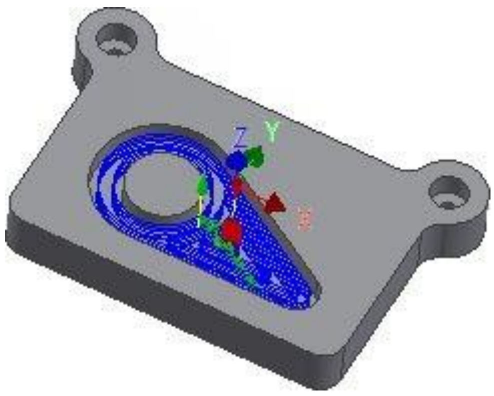
1. Click the **Passes** tab.
2. This group of settings controls how the 2D pocket toolpath is calculated. To clear out the pocket, the toolpath is generated in a number of Z levels, starting from the top of the stock and going down in steps of 2 mm to the bottom of the pocket.
3. Enable the **Multiple Depths** check box.
4. Change **Maximum roughing stepdown** to: 2.0 mm
5. Change **Finishing stepdowns** to: 1
6. We do not want to leave any stock in this operation, and since this is a roughing operation, the default is to leave stock.
7. Disable the **Stock to Leave** check box.

Start the Calculation

1. Click 

at the bottom of the **Operation** dialog box, or right-click in the graphics window and select **OK** from the marking menu, to automatically start calculating the toolpath.

The toolpath is now calculated and shown in the graphics window.



Machine the Counterbores

The next step is to machine the two counterbores at the top left and right corners of the part.

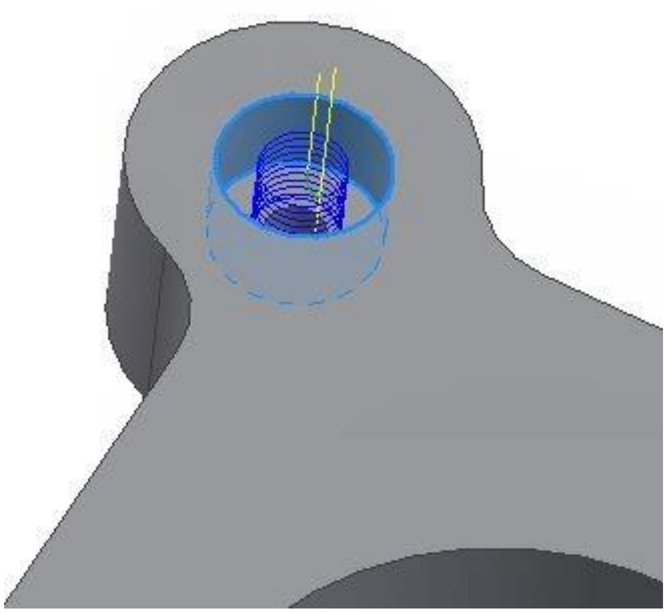
1. On the ribbon, click CAM tab ➤ 2D Milling panel ➤ Bore .

Tool tab

1. On the **Tool** tab, click the Tool button and select the tool **#2 - Ø8 mm flat** from the library.
2. Click Select to close the **Tool Library** dialog.

Geometry tab

1. Click the **Geometry** tab. The **Circular face selections** button should be active. Select the cylindrical faces of the two holes with the largest diameter at the top corners of the part.
2. If necessary, zoom in and click anywhere on the cylindrical surface of the upper large hole.

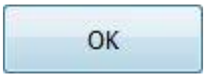


3. Do the same to select the other hole in the opposite corner.

 Passes tab

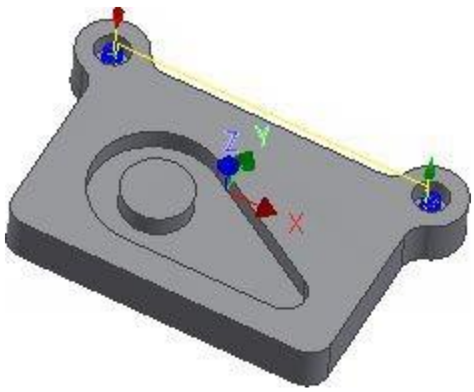
1. Click the **Passes** tab. Here we control how the helix toolpath is calculated.
2. Change **Pitch** to: 2.0 mm

Start the Calculation

1. Click 

at the bottom of the **Operation** dialog box, or right-click in the graphics window and select **OK** from the marking menu, to automatically start calculating the toolpath.

The toolpath is now calculated and shown in the graphics window.



Drill Holes

The next step is to drill the two small holes at the top left and right corners of the work piece.

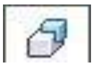
1. On the ribbon, click CAM tab ➤ Drilling panel ➤ Drill .

Tool tab

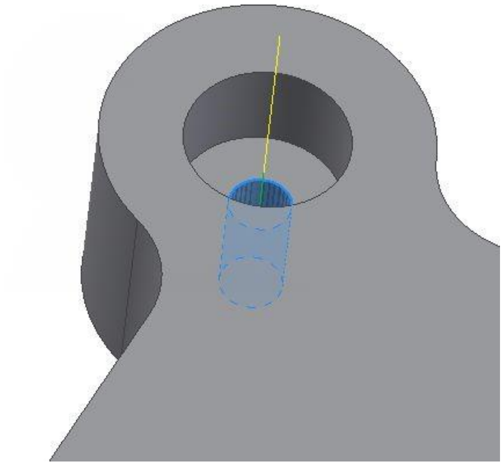
Change the default tool.

1. On the **Tool** tab, click the Tool button.
2. Click the New Mill Tool button.
3. This procedure creates a tool with default dimensions and cutting data. The default is a Ø10 mm drill. We will use most of the defaults, but change some of the cutting parameters a bit.
4. Click the **Cutter** tab.
5. Change **Type** to *Drill* by selecting it from the drop-down menu.
6. Change **Diameter** to 5 mm.

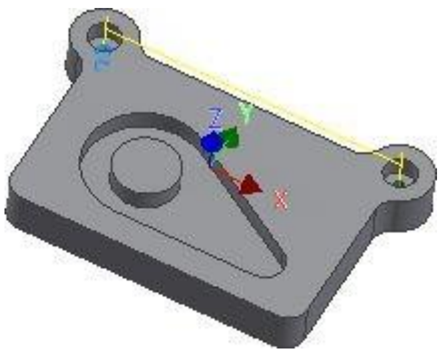
7. Click the **OK** button to create your new tool.
8. Click the **Select** button to select the tool for your operation and close the **Tool Library** dialog.

 Geometry tab

1. Click the **Geometry** tab. Ensure that *Selected faces* is selected from the **Hole mode:** drop-down menu and that the **Hole faces** selection button is active.
2. Zoom in, and select the cylindrical face of the smaller diameter hole at the top left corner of the part.



3. Enable the **Select same diameter** check box. Doing so automatically selects the identically sized small hole at the top right of the part.



Heights tab

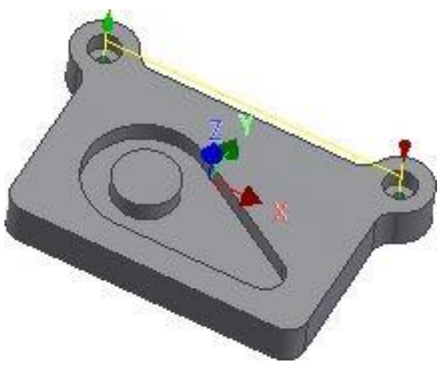
1. Click the **Heights** tab and enable the **Drill tip through bottom** check box.
2. Change **Break-through depth** to: 1 mm

Cycle tab

1. Click the **Cycle** tab.
2. Select *Chip breaking - partial retract* from the **Cycle type:** drop-down menu.

Start the Calculation

1. Click OK at the bottom of the **Operation** dialog box, or right-click in the graphics window and select **OK** from the marking menu, to automatically start calculating the toolpath.



To Tap Holes

A tapping operation differs only from a drilling operation in the type of cycle chosen from the **Cycle** tab. To simplify this step of the operation, we will just copy the existing drilling toolpath and edit it to create the new tapping toolpath.

1. In the **CAM Browser**, right-click on the **Drill1** node. (Do not be concerned if your Drill node appears with a number other than Drill1.)
2. Select **Duplicate** from the pop-up context menu. This creates a copy of the operation below the original one.
3. Left-click on the new operation (Copy of Drill1).
4. Enter a new name, such as *Tapping M6* and press Enter.
5. Next, you edit the tool and parameters.
6. Right-click on the operation **Tapping M6**.
7. Select **Edit** from the pop-up context menu.

 Tool tab

Create and select a new tapping tool.

1. On the **Tool** 

tab, click the button to open the **Tool Library**
2. Click the 

button.
3. On the **General** tab, change **Number** to 8.
4. On the **Cutter** tab, change **Type** to *Tap (Right Hand).*
5. Change **Diameter** to 6 mm.
6. Change **Flute length** to 15 mm.
7. On the **Feed & Speed** tab, change **Spindle speed** to 400 rpm.
8. Click the **OK** button to create your new tool.

9. Click the **Select** button to select the tool for your operation and close the **Tool Library** dialog.

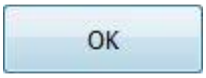
Important: If you want to run this toolpath on your machine tool, you may also need to set the **Pitch** parameter on the **Cutter** tab, as well as adjust the feed and speed parameters. The correct values can be found in the tool manufacturer's catalogue.

 Cycle tab

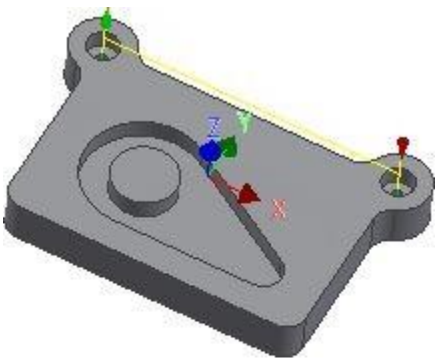
Now, make this a tapping cycle instead of a drilling cycle.

1. Click the **Cycle** tab and select *Tapping* from the **Cycle type:** drop-down menu.

Start the Calculation

1. Click 

at the bottom of the **Operation** dialog box, or right-click in the graphics window and select **OK** from the marking menu, to calculate the toolpath. The resulting toolpath should look like this:



You have now completed all the machining strategies for this part. Continue to Post Process the Toolpaths to finish the tutorial.

To Post Process the Toolpaths

In this final step of the tutorial, all toolpaths are post processed to produce the NC-code to be used by the machine tools. Before starting the post processing, it is good practice to regenerate all toolpaths and then simulate them. Doing so enables you to spot any errors in the toolpaths and rectify them.

1. Start by clicking **Setup1** at the top of the **CAM Browser**.

2. On the ribbon, click CAM tab ➤ Toolpath panel ➤ Generate

3. You may receive a dialog box message that the selected operation is already valid. This means that your toolpaths are good. You can click **Yes** to optionally regenerate them, or click **No** to leave them untouched and exit the dialog box.

4. Now, click CAM tab ➤ Toolpath panel ➤ Simulate .

5. Tip: As an alternative, you can also right-click on the **Setup** folder in the **CAM Browser** and select **Simulate (All)** from the pop-up context menu.

6. The **Simulation** player is displayed in the graphics window.

7. 

8.

9. Click the **Play** button on the **Simulation** player to playback the defined toolpaths.

10. When the simulation is complete, click the **Close** button in the **Simulation** dialog box, or right-click in the graphics window and select **Close** from the marking menu.

11. Next, click CAM tab ➤ Toolpath panel ➤ Post Process .

12. The **Post Process** dialog box is displayed.

13. Tip: As an alternative, you can also right-click on the **Setup** folder in the **CAM Browser** and select **Post Process (All)** from the pop-up context menu.

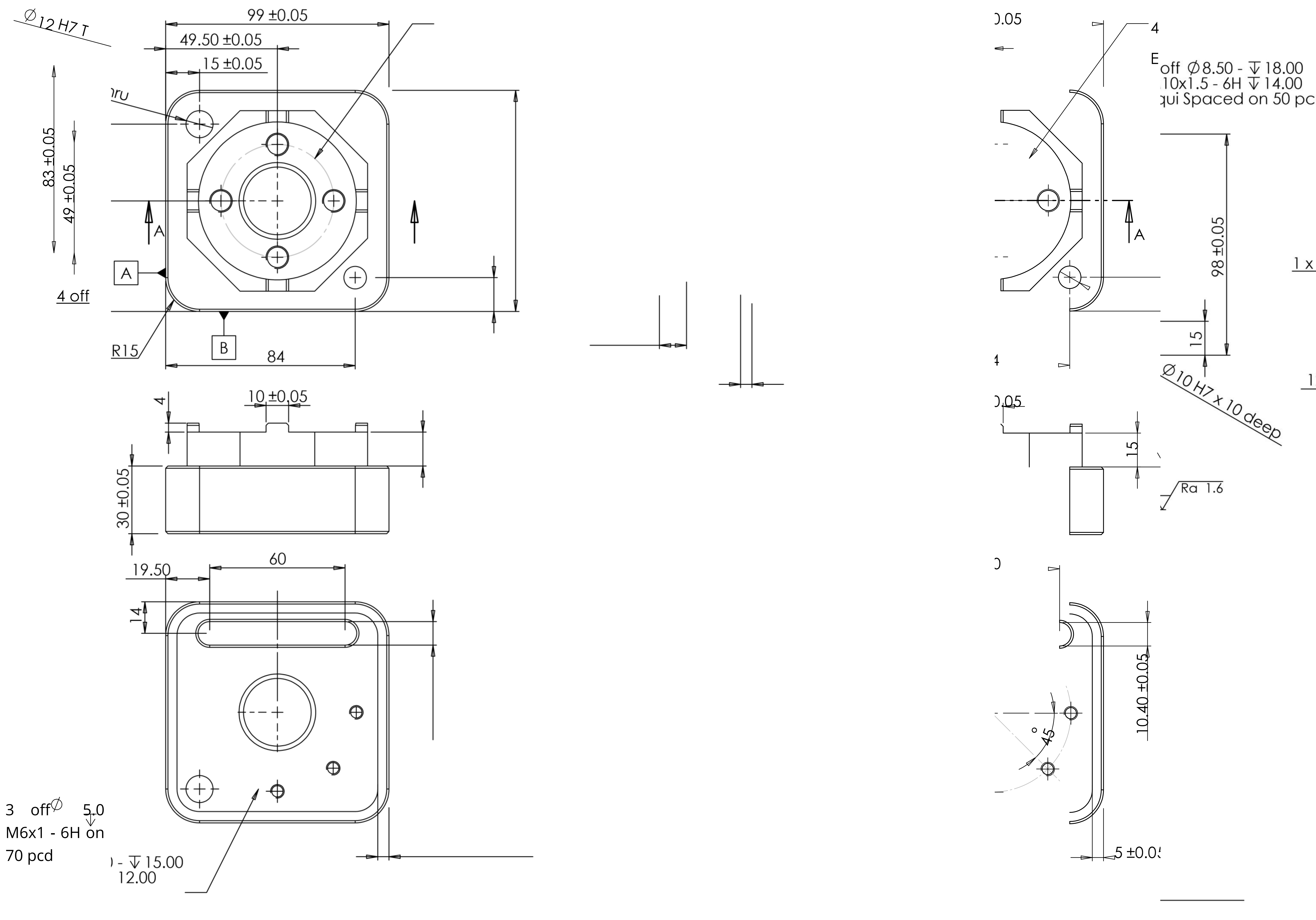
14. Select *heidenhain.cps - Generic Heidenhain* from the **Post Configuration** drop-down menu.
15. Accept the default output folder or choose another.
16. Accept the default program name/number or provide another.
17. Start the post processor by clicking the **Post** button.
18. Click the **Save** button.
19. Because the **Open NC file in editor** check box is enabled by default in the **Post Process** dialog box, the post processed file is automatically loaded into **Inventor HSM Edit**.

From the editor you can *edit*, *inspect*, and *transfer* your NC program to your CNC machine. The editor provides a number of CNC code-specific functions including line numbering/renumbering, XYZ range finder, and file comparison. The editor also features a DNC link for reliable RS-232 communications with a variety of CNC controls.

Remember: You can also post process *individual* operations by right-clicking the operation in the **CAM Browser**, and selecting **Post Process** from the pop-up context menu.

Congratulations! You have completed this tutorial.

1	2	3	4
A			
B			



99 ±0.05
49.50 ±0.05
15 ±0.05
Ø12 H7 T
hru
83 ±0.05
49 ±0.05
A
4 off
R15
B
84
10 ±0.05
4
30 ±0.05
19.50
60
14
3 off Ø 5.0
M6x1 - 6H on
70 pcd
- ↧15.00
12.00
.05
off Ø8.50 - ↧18.00
10x1.5 - 6H ↧14.00
qui Spaced on 50 pc
98 ±0.05
15
Ø10 H7 x 10 deep
1 x
.05
Ra 1.6
45°
10.40 ±0.05
5 ±0.05

D

E
E

chamfer All Round

1	2	3	4

F

WorldSkills UK, CNC Milling

Marking Scheme Overview

The marking scheme for the CNC Milling competition is broken down into 4 main areas of inspection and marking.

1. Primary Dimensions

- Measured Objectively (Yes, it's in tolerance / No, it's not in tolerance)
- Tight tolerance dimensions range between **±0.01mm** and **±0.05mm**
- Including H7 bore holes that are not standard reaming sizes
- There are between **13 – 17** of these dimensions on the model
- These dimensions account for about 50% of the project

2. Secondary Dimensions

- Measured Objectively (Yes, it's in tolerance / No, it's not in tolerance)
- These dimensions have a tolerance of **±0.1mm**
- Including H7 holes that are standard reaming sizes and tapped holes
- There are between **13 – 17** of these dimensions on the model
- These dimensions account for about 30% of the project

3. Surface Finish

- Measured Objectively (Yes, it's in tolerance / No, it's not in tolerance)
- Looking at a surface finish of 0.8Ra
- There is usually **2 or 3** surface finish markers on the drawing
- There is also a mark for only using **one** of piece of material
- These dimensions account for about 10% of the project

4. Conformity to Drawing

- Measured Subjectively (Industry Judges give a rating between **1 – 10** on how accurate it is to given criteria)
- Side 1 Complete – are all drawn features present and in correct position
- Side 2 Complete – are all drawn features present and in correct position
- Machined Chamfers – all edges that can be reached by a chamfer mill must be machined

- Hand Chamfers – all edge / corners that cannot be reached by a chamfer mill are blended in to the machined chamfer
- These dimensions account for about 10% of the project