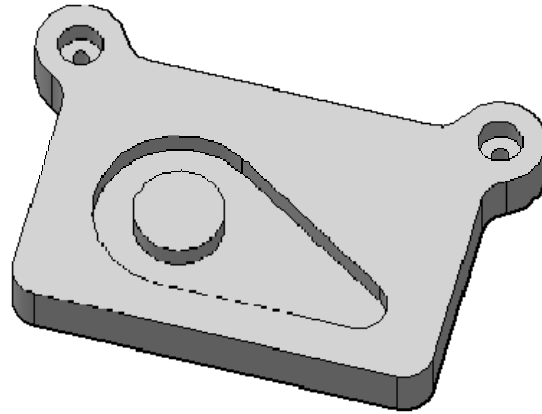




## Tutorial 1 - 2D Machining



In this tutorial you will learn how to machine the part above through the following steps:


- Contouring
- Pocketing
- Drilling
- Tapping
- Post processing

Before proceeding, please open the part **Tutorial1.SLDPRT** into HSMWorks, by choosing  **Open** from the **File** menu, and then browsing to the file.

 The files used in this manual can be found in the **examples** folder in the folder where HSMWorks is installed. Typically, this is something like **C:\Program Files\HSMWorks\examples\**

## Contouring

We want to run a contouring toolpath along the outer edges of the part.


- Click  **2D Contour** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

This creates a new operation, and opens the *property manager* where you can edit the individual parameters controlling the toolpath, as well as selecting the actual geometry to machine.

The property page is divided into a number of groups, and in this tutorial we will go through each one by one, changing the necessary settings in each group as we go along.

### Tool View

The Tool View group allows you to select a different orientation to machine from.

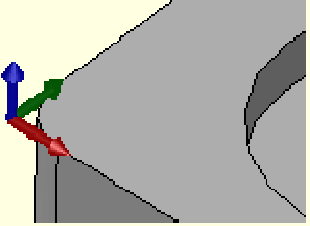
 The tool view of the active operation is always shown on the model like the coordinate system to the right.

Note the color of the axes:

- X-axis is *Red*.
- Y-axis is *Green*.
- Z-axis is *Blue*.

*You always need to have your Z-axis pointing upwards from the areas you want to machine since the tool rotates about the Z-axis on the machine tool with the tool tip pointing in negative Z.*

Most of the time, it is also an advantage to have the origin at the top of the model (or stock) such that positive Z-values are always in the air.




The default WCS (Working Coordinate System) uses the origin and orientation of the model, and since this model is designed with the Z-axis pointing upwards, we can use the default Tool View setting (*Use WCS*), and do not need to change the tool view in this example.

### Tool

- Press 

This opens the **Tool Library** where you can select from existing tools in a library or define a new tool.


 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.

To add a new tool with default dimensions:

- Press 

We can use the default tool type and dimensions (a 10mm flat mill) for this tutorial, but we will change the number, feed and speed:

- Click the **Cutter** tab
- Change **Flute Length** to 30mm.
- Click the **Feed & Speed** tab
- Change **Spindle Speed** to 3000rpm.
- Change **Cutting Feedrate** to 800mm/min.


 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, ie. that on position 6 you have a 10mm


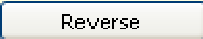
flat end mill cutter.

- Press the **OK** button to save your new tool.
- Press the **Select** button to select the tool for your operation and close the tool dialog.

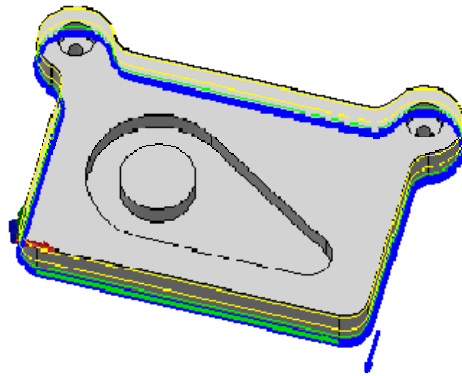
### Geometry

Here we select the geometry we want to machine. We want to run the tool around the outside edge of the part.

-  Select the edges to be machined. Move the mouse over the bottom front edge, it will be highlighted, then click on it. Depending on the side of the edge you click you can determine the direction of the contour. By clicking closer to the desired start of an edge you can determine the direction of the contour. For climb milling click close to the bottom right side of the edge.

 You can reverse the direction of a selected edge either by de-selecting it and clicking near the opposite end, or alternatively by pressing  in the property manager.

After you finished your geometry selections, the graphics window should look like this:

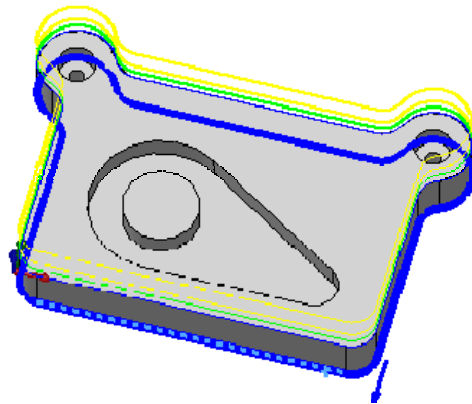


### Linking

Since we have the tool view origin at the top of the part, and since we selected the geometry around the bottom of the model, we can set some meaningful heights easily:

- For  **Clearance Height** press 
- For  **Retract Height** press 
- For  **Feed Height** press 
- For  **Top of Stock** press 

Notice that this changes the preview of the heights on the model.



Preview of depth and linking heights.

The top yellow boundary represents the **clearance height**.

The second yellow boundary represents the **retract height**.

The green boundary the **feed height**.

The thin blue boundary is the **top of stock**.

And finally, the thick blue boundary is drawn at the **depth**, or bottom, of the pocket.

### Leads & Transitions

This group allows you to control the shape of toolpath used to lead in and out on the actual cutting passes, as well as how the tool should move, or transition, between two passes.

We will leave all settings in this group unchanged.

### Filtering

This group allows you to apply a smoothing filter on the generated toolpath. Applying a smoothing filter will normally result in a simpler and smoother toolpath, but it requires a tolerance to work.

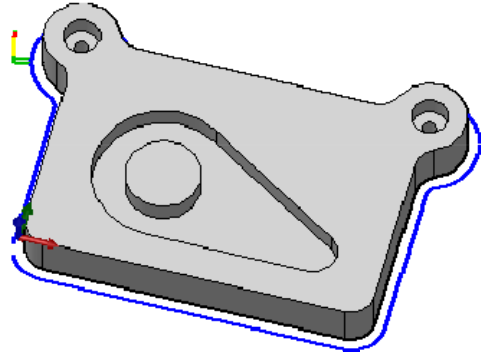
Additionally, this group allows you to control how rapid movements should be generated.

We will leave all settings in this group unchanged.

### Start calculation

- Click  at the top of the property manager.

The toolpath will now be calculated and shown in the graphic area. By default the cutting parts of a toolpath are colored 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.




**i** Notice that in the operation manager a new **job** has been created automatically. This is automatically done when creating a new operation before a job has been created. A job defines a number of settings used in all the operations contained within the job. For example the WCS can be changed in the Job.

Continue to **Pocketing...**

## Pocketing

We want to machine the pocket with the central circular boss on the top surface of the part.

- Click  **2D Pocket** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool view

The default (Use WCS) can be used again.

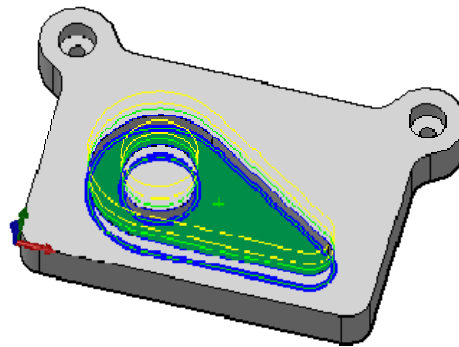
### Tool


By default the last tool used will be selected. Ensure that the tool **#6 - 10mm flat** is selected.

### Geometry

Here we select the contours of the pocket we want to clear.

-  Click anywhere on the face at the bottom of the pocket.

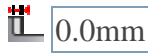


 Selecting faces for 2D geometry automatically uses all edges of the face for the contours. If two adjacent faces are selected, the edges they share are, however, *not* included in the selection.



We do not want to leave any stock in this operation, and since this is a roughing operation, the default is to leave stock.

- Change **Radial Stock to Leave**:



Changing the radial (Wall) stock to leave will also automatically update the axial (Floor) stock to leave, so you do not have to change this.

## 2D Pocket

This group controls how the 2D pocket toolpath will be calculated. We want to use this pocket toolpath to clear out the pocket. To do this we want to generate the toolpath in a number of z levels, starting from the top of the stock and going down in steps of 2mm to the bottom of the pocket.

- Enable **Multiple Depths**:



- Change **Maximum Roughing Stepdown**:









- Change **Finishing Stepdowns**:



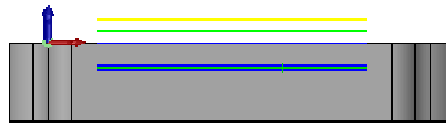
## Linking

As for the contouring operation, we will use set the linking to absolute:

- For  **Clearance Height** press 
- For  **Retract Height** press 
- For  **Feed Height** press 

- For **Top of Stock** press 

If you change the view to the side of the part (e.g. by pressing Space and then choosing Bottom), the heights should look like this:



### Leads & Transitions

We will leave all settings in this group unchanged.

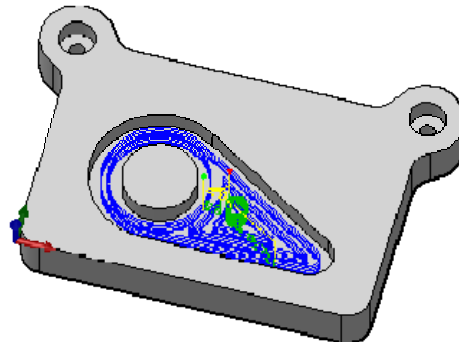
### Filtering

We will leave all settings in this group unchanged.

### Start calculation


- Click  at the top of the property manager.

The toolpath will now be calculated and shown in the graphic area. By default the cutting parts of a toolpath are colored 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.



## Drilling

Now we want to drill the two holes at the top left and right corners of our work piece.



- Click  **Drill** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool view

The default (Use WCS) can be used again.

### Tool


Change the default tool:

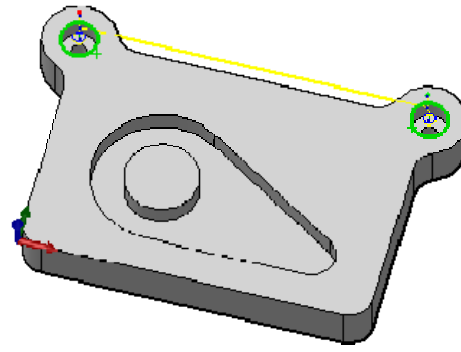
- Press  to open the *Tool Library*.
- Click 

This will create a tool with default dimensions and cutting data. The default is a 10mm flat mill. We will use most of the defaults, but change some of the cutting parameters a bit:

- Click the **Cutter** tab
- Change **Tool type** to *Drill* by selecting it from the drop down.
- Change **Diameter** to 5mm.
- Press the **OK** button to save your new tool.
- Press the **Select** button to select the tool for your operation and close the tool dialog.


### Geometry

-  Select the circular edges of the two holes at the top corners of the part.



### Hole


- Change **Depth**:

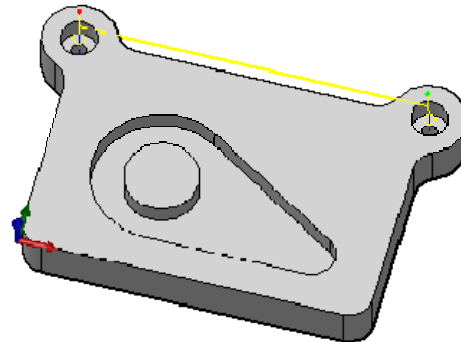
 -25mm

### Cycle

- Select *Chip breaking - partial retract* from the cycle type drop down.


### Start calculation

Click  at the top of the property manager.




## Tapping

This tapping operation only differs in the type of cycle from the previous drilling operation; tapping will be done for the same geometry. To save all the work of entering this data, we will just copy the existing drilling toolpath and create the tapping toolpath by editing this copy.

- Right-click on the operation  **Drill1**.
- Choose **Duplicate** from the context menu.
- Right-click on the new operation.
- Choose **Rename**.
- Enter a new name, such as "*Tapping M6*".

Next we have to edit the tool and parameters.

- Right-click on on the operation  **Tapping M6**.
- Choose **Edit** from the context menu.

To create and select a new tapping tool:

- Expand the group **Tool**.
- Press the **Select** button.
- Press the **New Tool** button.

On the **General** tab:


- Change **Number** to 8.

On the **Cutter** tab:

- Change **Tool type** to *Tap (Right Hand)*.
- Change **Diameter** to 6mm.

On the **Feed & Speed** tab:


- Change **Spindle speed** to 400rpm.
- Press the **OK** button to save your new tool.
- Press the **Select** button to select the tool for your operation and close the tool dialog.

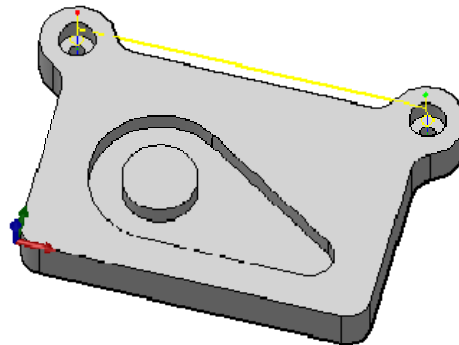
 If you want to run this toolpath on your machine tool, you may also need to set the **Pitch** parameter on the Cutter tab, as well as adjust the feed and speed parameters. You should always look up the correct values in your tool manufacturer catalog.

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

- In the group **Cycle** select *Tapping*.

### Start calculation


Click  at the top of the property manager. The resulting toolpath will look like this:





You have now finished all operations in this tutorial.


*As an exercise you can use the 2D Contour strategy to machine the two 16mm borings above the drill holes.*

## Post processing

 It is good practice before you start the post processing to regenerate all toolpaths and then verify or simulate the toolpaths. This way you can spot any errors in the toolpaths and rectify them.

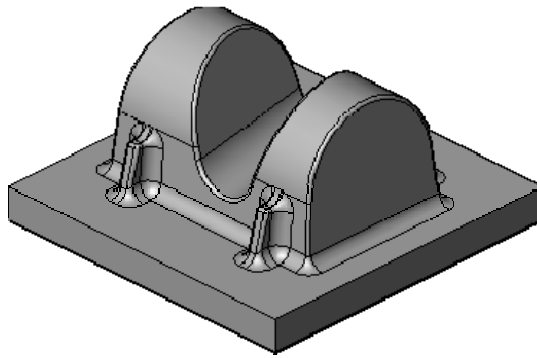
We are ready to postprocess all toolpaths in order to make the NC-code which can be used by the machine tool.

- Right-click on  **Job** in the operation manager.
- Select  **Post Process (All)**.
- From the pulldown **Post processor configuration** select *heidenhain.cps - Generic Heidenhain*
- Select an output folder of your choice.
- Start the post processor by clicking **Post**. By default the post processed file will be loaded into HSMWorks Edit.

 You can also post process individual operations by invoking the post processor through a right-click on the operation, and selecting **Post Process** from the context menu.

**Congratulations!** You have completed this tutorial.

## Tutorial 2 - High-Speed 3D Machining



In this tutorial you will learn how to machine the part above on a high speed machine tool through the following steps:

- **Creating a Tool Library**
- **Roughing**
- **Semi-finish**
- **Finish**

Before proceeding, open the part **Tutorial2.SLDPRT** into HSMWorks.


**i** The files used in this manual can be found in the **examples** folder in the folder where HSMWorks is installed. Typically, this is something like  
**C:\Program Files\HSMWorks\examples\**




## Creating a Tool Library


In this tutorial, we will start of by creating a tool library containing all the tools we will be using first.



First, open the tool library:


- Choose  **Tool Library** from the HSMWorks toolbar or the HSMWorks menu.

 The tool library allows you to select from existing tools in a library as well as define a new tools.  
Tool definitions can be saved in a library or just for the part you work on.

Next, create a new library called Tutorial 2:

- In the tool library dialog, select **My Libraries**
- Click 
- Name the new library **Tutorial2**. *Press Enter to accept the new name*

 This creates a new library file in **My Libraries** subfolder of your **My Documents** folder.  
You can locate the library file by right-clicking on the library and pressing  **Open in Windows Explorer**.

Now, add each of the following tools by pressing , and then changing the type and the parameters specified below. All other parameters should be left at their defaults.

Number	Tool Type	Diameter (mm)	Corner Radius (mm)	Flute length (mm)	Spindle Speed (rpm)	Cutting feedrate (mm/min)
10	Bull-nose Mill	20	1	25	16000	9500
13	Bull-nose Mill	6	1	default	30000	6000
20	Ball Mill	10	n/a	default	30000	1800
21	Ball Mill	6	n/a	default	30000	6000


When you are done setting up the tools, press **OK** to return to HSMWorks. You will be asked to save the library.

## Roughing

For roughing the part, we will first use an adaptive clearing strategy to remove the majority of the excess material, then a pocket rest milling strategy with a smaller tool to remove additional material.

- **Adaptive clearing**
- **Pocket (Rest Machining)**


## Adaptive clearing

- Click the strategy **Adaptive Clearing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

We will leave all settings in this group unchanged.

### Tool

- Press 
- From the library *Tutorial2* that we created before, select tool **#10 - 20mmx1mm bullnose**
- Press **Select**

You will be presented with a dialog message informing you that a copy of the tool will be created in the current library (the document). Press **OK**.

### Geometry

- Set **Radial Stock to Leave:**



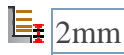
*Note that the axial stock to leave is automatically updated to reflect the radial stock to leave.*

### Adaptive Clearing

- Change **Maximum Roughing Stepdown:**



- Change **Fine Stepdown:**



*(automatically updated)*

- Enable **Flat Area Detection**:

Flat Area Detection

### **Confinement**

We will leave all settings in this group unchanged.

### **Linking**

We will leave all settings in this group unchanged.

### **Leads & Transitions**

We will leave all settings in this group unchanged.

### **Fillets**

We will leave all settings in this group unchanged.


### **Filtering**


We will leave all settings in this group unchanged.

### **Rest Machining**

We will leave all settings in this group unchanged.

### **Start calculation**


Click  at the top of the property manager. This will start generation of the toolpath.

 Whenever a toolpath is being generated, you can continue to work by pressing the **Hide** button on the Task Manager dialog.

When the Task Manager is hidden, toolpath calculation will continue in the background, and you can see progress directly in the operation

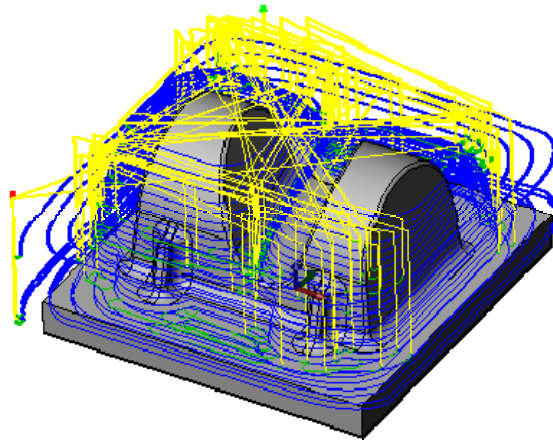
manager.

Note also that for most of the strategies, it is possible to see a preview of the toolpath as it's being generated. This can be valuable to help you see early on if something is wrong so you can abort the task and go back and change the parameters.


You can bring the dialog back by selecting  **Task Manager** from the HSMWorks toolbar or menu.

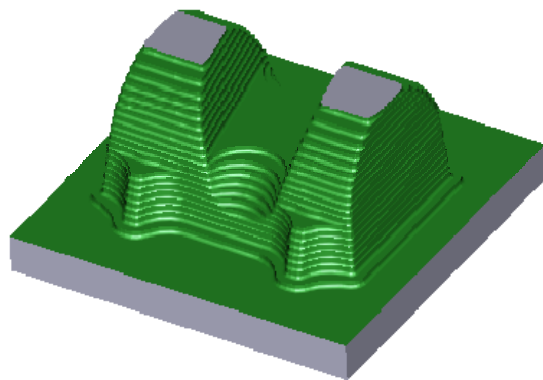
You can either abort a task by pressing the **Abort** button in the task manager, or by choosing **Abort Generation** when right-clicking on the operation in the operation manager.

When the toolpath generation is complete, the resulting toolpath should look like this:



### Verification


To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification

## Pocket (Rest Machining)

There are some areas where the 20mm tool did not fit. We will remove more of the remaining material there with a pocket rest milling operation.

- Click the strategy **Pocket Clearing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

We will leave all settings in this group unchanged.

### Tool

- Select tool **#13 - 6mmx1mm bullnose** from the *Tutorial2* library.

### Geometry

We use the same settings as in the previous operation *Adaptive1*.

- Set **Radial Stock to Leave:**



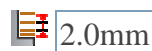
### Pocket Clearing

Use these settings:

- Set **Smoothing Deviation:**



- Set **Maximum Roughing Stepdown:**




### Confinement



We will leave all settings in this group unchanged.

### Linking

The default linking heights are automatically calculated from the geometry such that the entire depth of the part is machined.

 For this tutorial, we use the **Shortest Path** retraction policy. This gives the shortest possible rapid movement paths, but can only be used on high speed enabled machines where rapid motion is interpolated as linear moves. Should you wish to machine the tutorial part on your machine, you may have to change this setting to suit your machine and control.

### Leads & Transitions

We will leave all settings in this group unchanged.

### Filtering

We will leave all settings in this group unchanged.

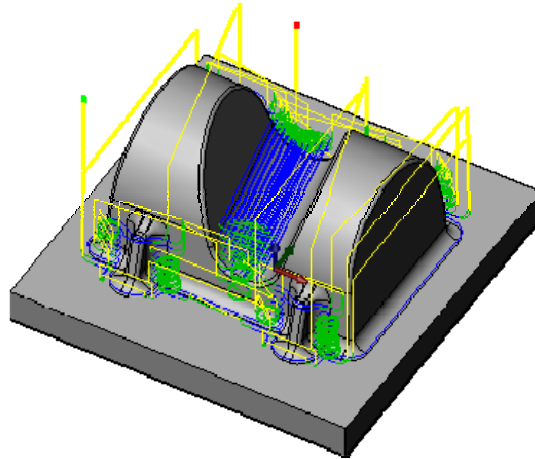
### Rest Machining

We only want to remove the material the previous operation did not remove.


- Select *From Previous Operation(s)* from the **Rest material source** drop down

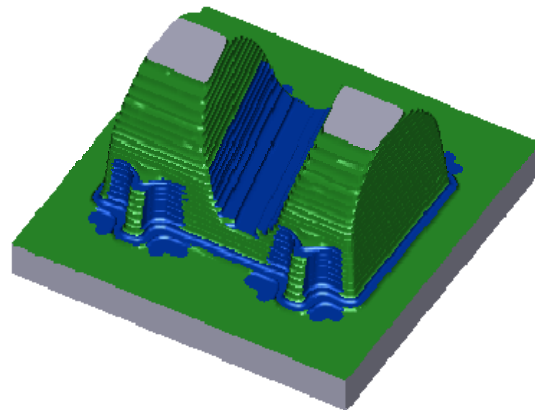
### Start calculation

Click  at the top of the property manager. The resulting toolpath should look like this:



### Verification

To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification


## Semi-finish

We now have between 0.2mm and 0.4mm material left on the part (0.3mm **stock to leave** plus/minus 0.1mm **tolerance**). We will use several semi-finish operations to reduce this material. In the following part of the tutorial you will learn how to set up these toolpaths:

- **Contour machining**
- **Scallop Machining**

## Contour machining

We will create a contour operation to handle the steep areas of the part.

- Click the strategy **Contour Finishing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

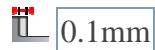
We will leave all settings in this group unchanged.

### Tool

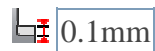
- Select tool **#20 - 10mm ball** from the *Tutorial2* library.

### Geometry

- Set **Radial Stock to Leave:**



- Set **Axial Stock to Leave:**



### Contour Finishing

Use these settings:

- Set **Tolerance:**



- Set **Maximum Stepdown:**



- Enable **Order by Depth**:

Order by Depth

### Confinement

Under *Machining boundary*:

- Select *Tool Center on Boundary* from the **Tool Confinement** drop down.

We want to confine the machining to steep areas with this operation. This can be achieved by limiting the **Slope range**.

- Change **From Slope Angle**:

 60deg

- Change **To Slope Angle**:

 90deg

All other settings remain default.

### Linking

- Select *Minimum Retraction* as the **Retraction Policy**
- Change **Maximum Stay-Down Distance**:

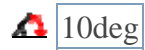
 10mm

All other settings remain default.

### Leads & Transitions

Use these settings:

- Change **Ramping Angle (deg)**:




### Filtering

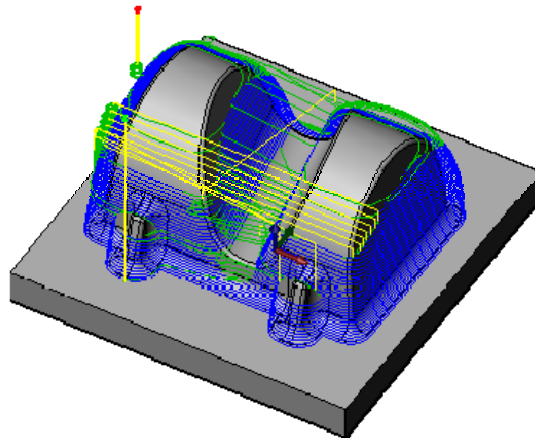
We will leave all settings in this group unchanged.


### Rest Machining

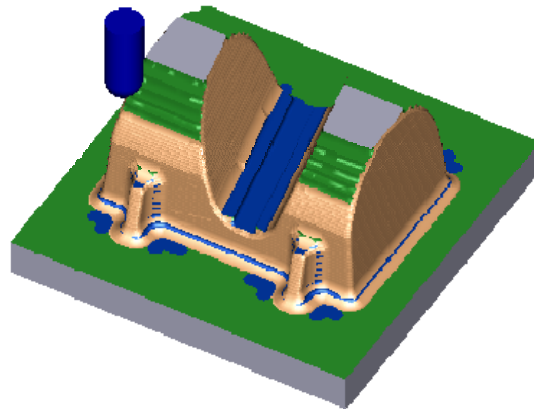
We will leave all settings in this group unchanged.

### Start calculation

- Click  at the top of the property manager. The resulting toolpath should look like this:




To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification

## Scallop Machining

Next we will create a constant scallop toolpath on the shallow areas of the boss.

- Click the strategy **Scallop Finishing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

We will leave all settings in this group unchanged.

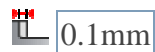
### Tool

The previous tool **#20 - 10mm ball** should still be selected. We will leave all settings in this group unchanged.

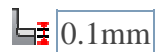
### Geometry

Enter the same **Stock to leave** as we used in the previous contour semi finishing operation.

- Set **Radial Stock to Leave**:



- Set **Axial Stock to Leave**:



### Scallop Finishing

Use these settings:

- Set **Tolerance**:





- Set **Stepover**:



## Confinement

Under *Machining boundary*:

- Select *Tool Center on Boundary* from the **Tool Confinement** drop down.

We want to confine the machining to steep areas with this operation. This can be achieved by limiting the **Slope range**.


- Change **From Slope Angle**:



- Change **To Slope Angle**:



All other settings remain default.

 Machining from 0 degrees (horizontal) to 65 degrees from horizontal ensures that there is no excess material in the transition zone between this operation and the previous operation on steep areas, and that the tool starts in areas that are already machined.

## Linking

- Change **Depth**:



**i** We are using depth limits, because we do not want the tool to go below  $Z = 10.3\text{mm}$  (which is above the base surface). We will be machining the base surfaces and the fillet between the base surfaces and the part later.

### Leads & Transitions

We will leave all settings in this group unchanged.

### Filtering

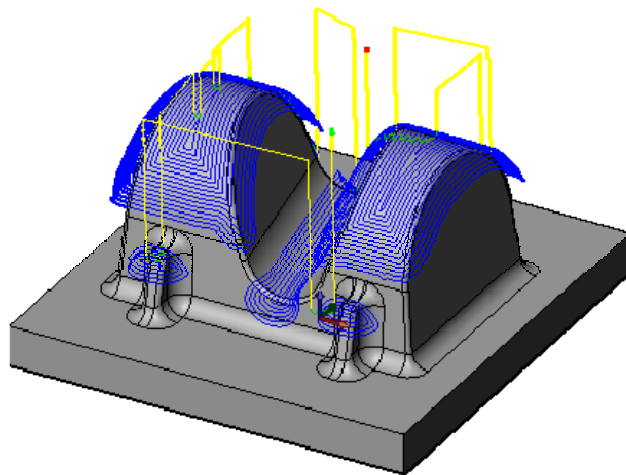
We will leave all settings in this group unchanged.


### Rest Machining

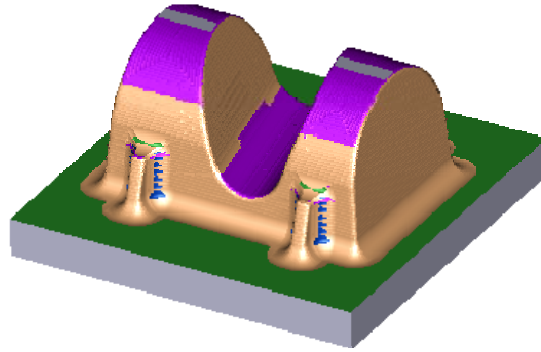
We will leave all settings in this group unchanged.

### Start calculation

- Click  at the top of the property manager. The resulting toolpath will look like this:



To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.




Solid Verification

## Finish

We now have 0.1 mm left on most of the part. There is a little more on top of the base surfaces, inside the 4 mm fillets and on the lower part of the fillet at the base surfaces. We will now finish the part in these steps:

- Scallop
- Horizontal Finishing
- Pencil Milling
- Collapsed Pencil
- Postprocessing

## Scallop

- Click the strategy **Scallop Finishing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

We will leave all settings in this group unchanged.

### Tool

Select tool **#21 - 6mm ball** from the library *Tutorial2*.

### Geometry

Use these settings:

- Set **Radial Stock to Leave:**

 0.05mm

- Set **Axial Stock to Leave:**

 0.05mm

### Scallop Finishing

Use these settings:

- Set **Tolerance:**

 0.02mm

- Set **Stepover:**

 1mm

- Enable **Link from inside to outside**:

Link from inside to outside

### Confinement

Use these settings:

Under *Machining boundary*:

- Select *Tool Center on Boundary* from the **Tool Confinement** drop down.

In the previous operations we limited the slope range to confine machining to steep areas. In this case, set the slope range to machine the entire part.

- Change **From Slope Angle**:



- Change **To Slope Angle**:



All other settings remain default.

### Linking

We will leave all settings in this group unchanged.

### Leads & Transitions

We will leave all settings in this group unchanged.


### Filtering

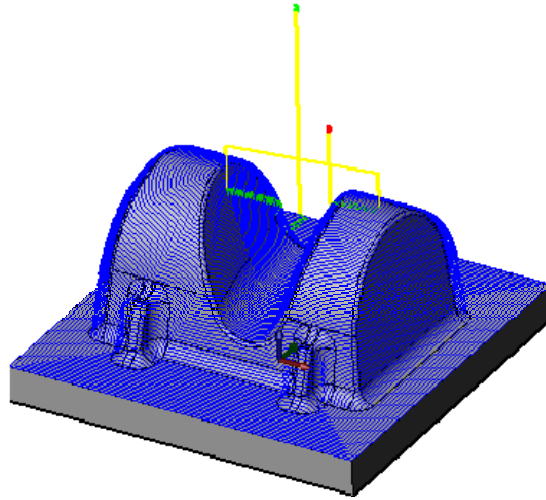
We will leave all settings in this group unchanged.


### Rest Machining

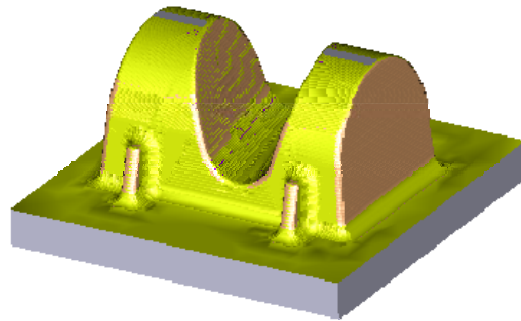
We will leave all settings in this group unchanged.

**Start calculation**

- Click  at the top of the property manager. The resulting toolpath will look like this:




To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification

## Horizontal Finishing

We will now clear the base surfaces where they are perfectly horizontal.

- Click the strategy **Horizontal Finishing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View


We will leave all settings in this group unchanged.

### Tool

Select tool **#13 - 6mmx1mm bullnose** from the document tool table.

### Geometry

We will leave all settings in this group unchanged.

 By default the amount of stock to leave is 0. In this operation we will machine down to the final shape of the part, without any stock allowance.

## Horizontal Finishing

Use these settings:

- Set **Tolerance**:



- Set **Smoothing Deviation**:



## Confinement



We will leave all settings in this group unchanged.

### Linking

We will leave all settings in this group unchanged.

### Leads & Transitions

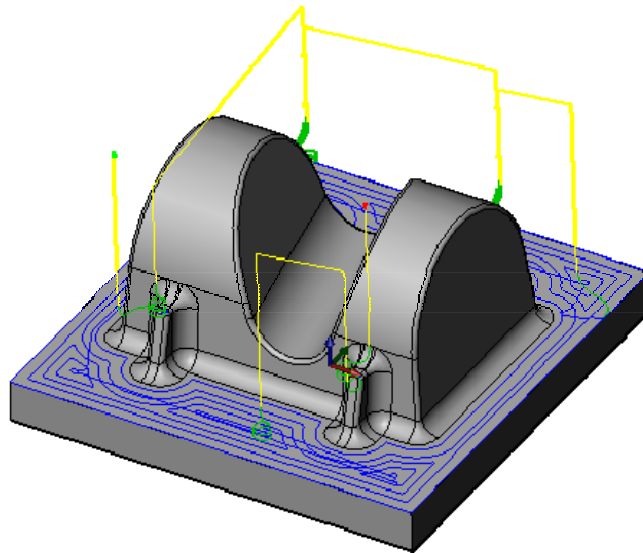
We will leave all settings in this group unchanged.


### Filtering

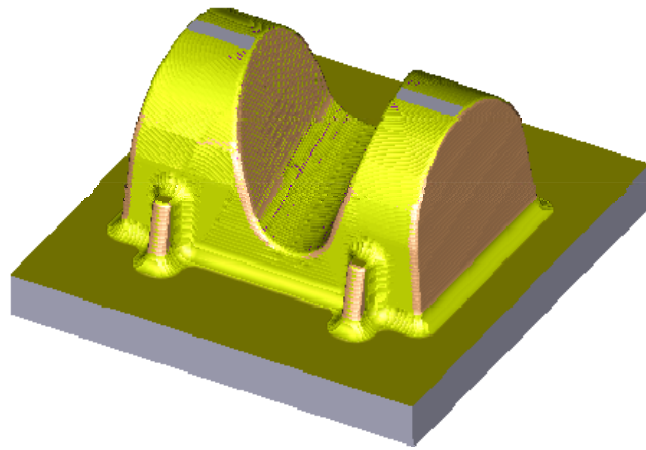
We will leave all settings in this group unchanged.

### Start calculation

- Click  at the top of the property manager. The resulting toolpath will look like this:



To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification

## Pencil Milling

We are using a 6 mm ball mill for finishing toolpaths. The fillets between base and boss are 4mm radius. We will use parallel pencil passes to machine the fillets.

- Click the strategy  **Pencil Finishing** on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

We will leave all settings in this group unchanged.

### Tool

Select tool **#21 - 6mm ball** from the document tool table.

### Geometry

Use these settings:

- Set **Radial Stock to Leave:**



- Set **Axial Stock to Leave:**



### Pencil Finishing

Use these settings:

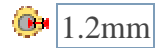
- Set **Tolerance:**



- Set **Bitangency Angle**:



- Set **Overthickness**:



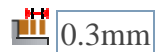
- Enable **Limit number of stepovers**:



- Set **Number of Stepovers**:



- Set **Stepover**:



- Disable **Link from inside to outside**:



**i** **Overthickness** is used by HSMWorks to generate pencil passes where they would not exist for the specified tool. Here the tool radius of 3mm is smaller than the fillet radius of 4mm. With **Overthickness** set to 1.2mm, HSMWorks will generate the pencil passes for a larger tool radius, and uses them to guide the smaller tool into the fillets.

### Confinement

We will leave all settings in this group unchanged.

### Linking

We will leave all settings in this group unchanged.

### Leads & Transitions

We will leave all settings in this group unchanged.

### Filtering

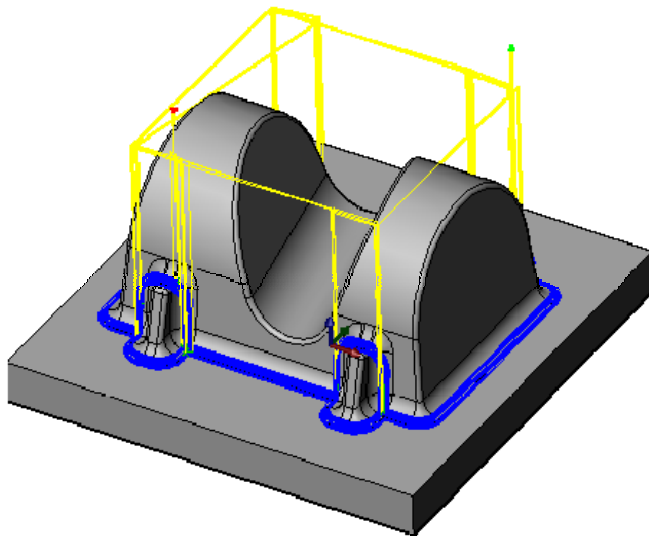
We will leave all settings in this group unchanged.


### Rest Machining

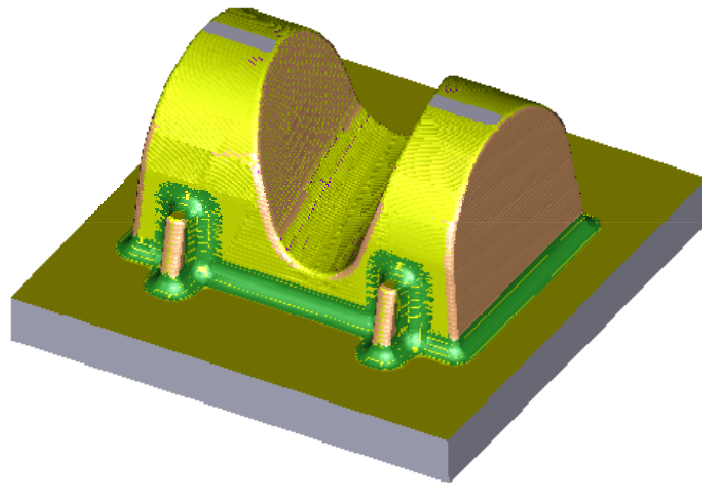
We will leave all settings in this group unchanged.

### Start calculation

- Click  at the top of the property manager. The resulting toolpath will look like this:



To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification

## Collapsed Pencil

For the last finishing operation, we will use a *collapsed pencil* finishing toolpath over the entire part except the base surface. We can copy the previous operation, and then edit the parameters.

- Right-click on the last operation, **Pencil1**.
- Select **Copy** from the context menu.
- Right-click again.
- Select **Paste** from the context menu. A copied operation **Copy of Pencil1** is added to the bottom of the toolpath list.
- Rename the copied toolpath: select and press **F2**. Type the new name **Parallel pencil**.
- To open the **Property manager** right-click **Parallel pencil** and select **Edit** from the context menu. You will see the following groups:

### Tool View

We will leave all settings in this group unchanged.

### Tool

The selected tool should still be **#21 - 6mm ball**.

### Geometry

We will leave all settings in this group unchanged.

### Pencil Finishing

Use these settings:


- Disable **Limit number of stepovers**:  
 Limit number of stepovers

- Set **Stepover**:



- Enable **Link from inside to outside**:

Link from inside to outside

 We are linking from inside to outside now, to make the tool enter the already machined areas along the centerline of the fillets and work outwards from there.

### Confinement

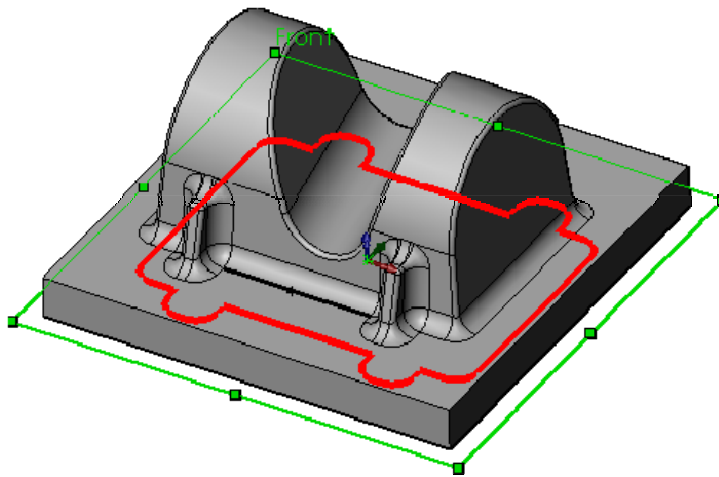
We will confine the toolpath to the area of the boss.

- In the pulldown menu **Machining Boundary** switch to **Selection**.
- Disable **Tangent Propagation**:

Tangent Propagation

- Select an edge on the the inner boundary as shown here:





Boundary selection

- Set **Additional Offset**:



### Linking

We will leave all settings in this group unchanged.

### Leads & Transitions

We will leave all settings in this group unchanged.

### Filtering

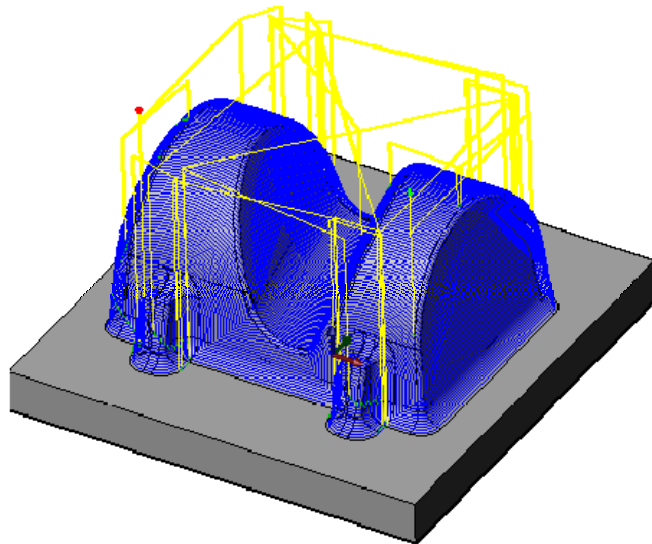
We will leave all settings in this group unchanged.


### Rest Machining

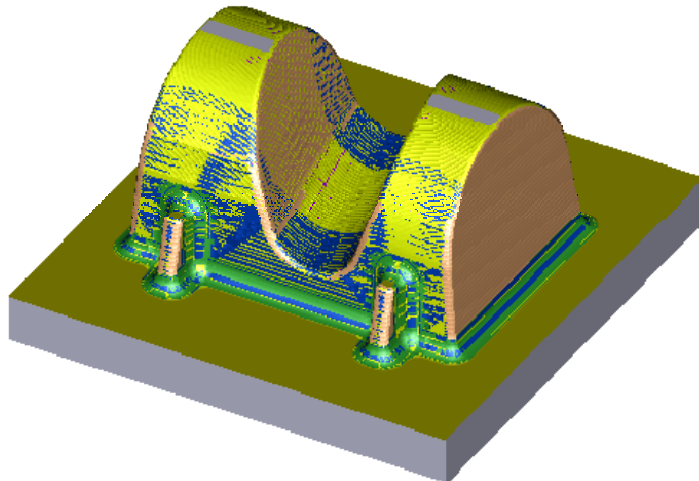
We will leave all settings in this group unchanged.

### Start calculation

- Click  at the top of the property manager. The resulting toolpath will look like this:

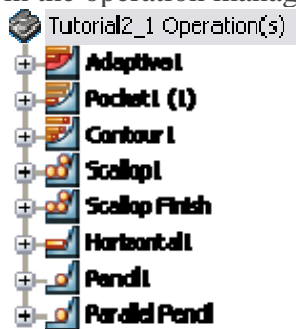


To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.





Solid Verification

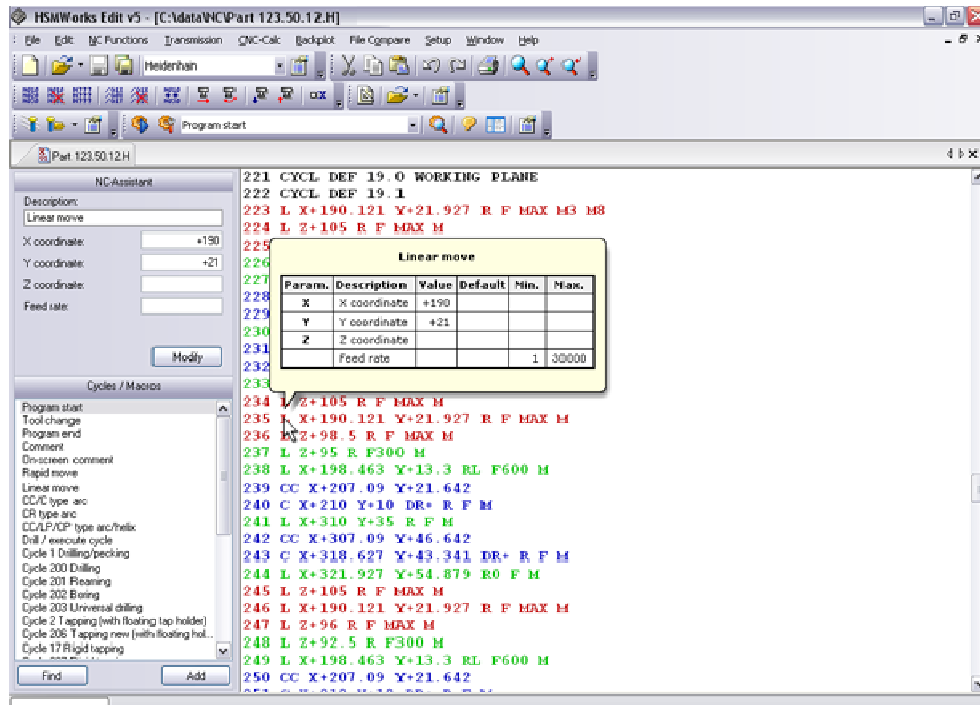
That finishes this tutorial's toolpaths; you should now have these operations in the operation manager:



## Postprocessing

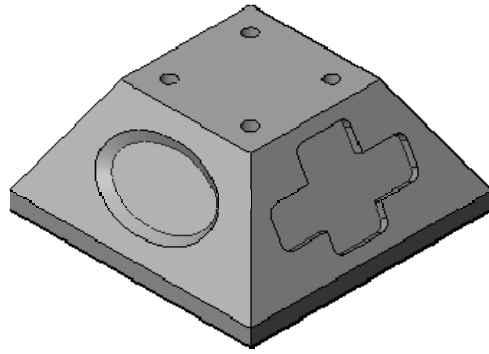
We are ready to postproces all toolpaths in order to make the NC-code which can be used by the machine tool.

- Right-click on  **Job** in the operation manager.
- Select  **Post Process (All)**.
- Choose the desired post processor configuration, program file name and location.
- When you have finished editing your settings click on **Post** to generate the CNC program file. By default HSMWorks Edit will open and you can view and simulate the program code, and transfer it to the machine control.



**Congratulations!** You have completed this tutorial.


## Tutorial 3 - 3+2 Machining



In this tutorial you will learn how to set up **3+2 Machining** toolpaths.

- **Job Setup**
- **Adaptive Machining, Pocket**
- **Pocket Clearing**



Before proceeding, open the part **Tutorial3.SLDASM** into HSMWorks.

 The files used in this manual can be found in the **examples** folder in the folder where HSMWorks is installed. Typically, this is something like **C:\Program Files\HSMWorks\examples\**

We will assume that the stock we start from is already roughed to a shape that only requires the machining of the two pockets using 3+2 toolpaths. The file for this tutorial is an assembly and contains the final component as well as the stock shape we start from.

## Job Setup



We begin with setting up common job parameters.

- Open the operation manager by clicking 
- Right-mouse click on **Tutorial3 Operation(s)** and select  **New Job**.
- In the group **Stock**, change the Stock Mode from **Automatic** to **From Solid**.

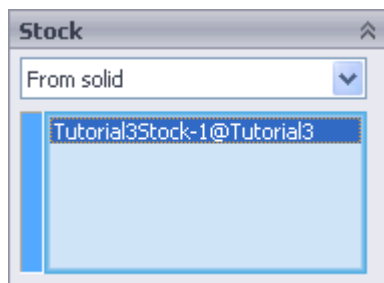
Next, select the component **Tutorial3Stock** as the stock by *either*:

- In the model window, right-click on the model.
- Choose **Select Other**.
- From the list, select **Tutorial3Stock**.

*or*:

- In the model window click the  in front of  Tutorial3 to expand the Assembly tree.
- Select the **Tutorial3Stock** component feature.

Your selection should look like this:






- In the group **Surfaces**, click the selection field so that the field is highlighted as the active selection field.

Next, select the component **Tutorial3** as the surfaces by *either*:


- In the model window, click on the model.

*or*:

- In the model window click the  in front of  Tutorial3 to expand the Assembly tree.
- Select the **Tutorial3** component feature.
- Click  at the top of the property manager.

## Adaptive Machining, Pocket


We will use an **Adaptive Clearing** toolpath to clear the oval shaped pocket:

- Click the strategy **Adaptive Clearing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

- Select **Use Coordinate System**
- From the tree view in the model window, expand the node **Tutorial3**.
- Select **Coordinate System1**.

### Tool

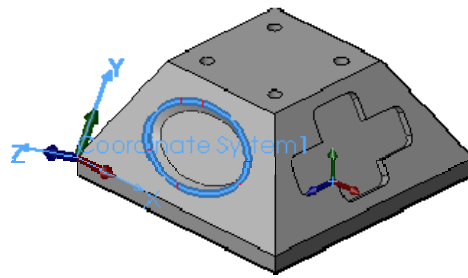
- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#12 - 8mmx1mm bullnose**
- Press **Select**

You will be presented with a dialog message informing you that a copy of the tool will be created in the current library (the document). Press **OK**.


### Confinement

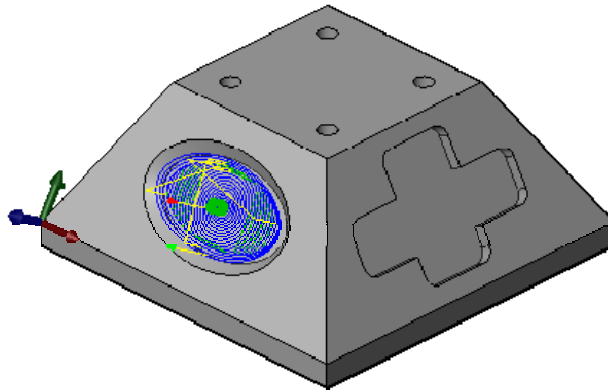
- For the machining boundary, select **Selection**.
- In the graphics window, click on the outer oval edge of the pocket to select it.
- Select **Tool center on boundary**.




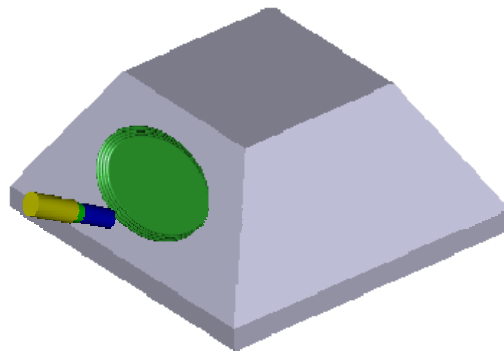


### Start calculation

- Click  at the top of the property manager. The resulting toolpath will look like this:




To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification

## Pocket Clearing


We will use a **Pocket Clearing** toolpath to clear the cross shaped pocket on the adjacent face to the oval pocket.

- Click the strategy **Pocket Clearing**  on the HSMWorks toolbar. This opens the **Property Manager** and you will see the following groups.

### Tool View

- Select **Use Coordinate System**
- From the tree view in the model window, expand the node **Tutorial3**.
- Select **Coordinate System2**.


### Tool

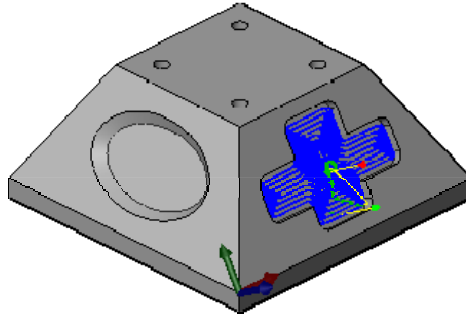
- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#4 - 5mm flat**
- Press **Select**


### Confinement

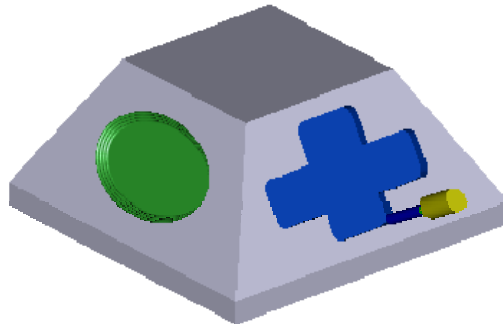
- For the machining boundary, select **Selection**.
- In the graphics window, click on the outer edge of the cross shaped pocket to select the circumference of the pocket automatically.

### Start calculation

- Click  at the top of the property manager. The resulting toolpath will look like this:



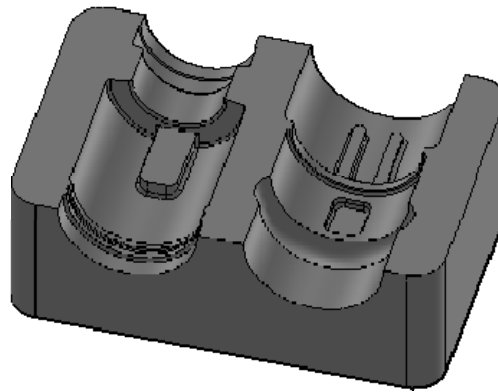
To verify the toolpath use the solid verification, right click on the **Job** node in the **Operation Manager** and select **Solid Verification(All)**. Press the **Start** button  and watch the stock simulation up to the final stock shape.



Solid Verification

**Congratulations!** You have completed this tutorial.

## Tutorial 4 - Advanced 3D Machining




In this tutorial we will learn how to machine a 3D part using the following steps:

- Create a Machining Assembly
- Create a Coordinate System
- Job settings
- Facing
- Contouring
- Adaptive Clearing
- Adaptive Clearing Rest Machining
- Horizontal
- Contour
- Parallel
- Pencil 1
- Pencil 2
- Pencil 3



- **Post Processing**

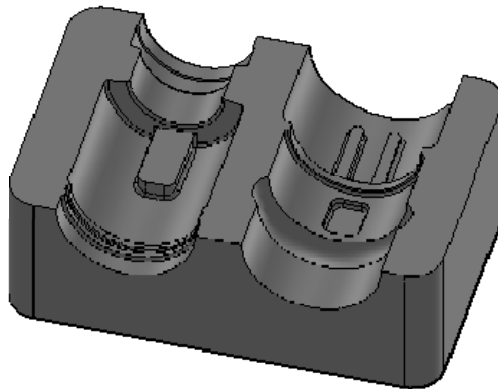
Before proceeding, open the part **Tutorial4.SLDPRT** into HSMWorks.


 The files used in this manual can be found in the **examples** folder in the folder where HSMWorks is installed. Typically, this is something like **C:\Program Files\HSMWorks\examples\**

## Create a Machining Assembly

Before proceeding we will make a new *machining assembly* to keep the machining operations and toolpaths in.

- From the File menu choose  **Make Assembly from Part**. This will create a new assembly file, and preselect the already opened part document as the component to insert.
- Click  at the top of the property manager.
- Rotate the part with the Z-axis (blue) upwards so it corresponds better to the placement on the machine:




-  **Save** the new assembly as *Tutorial4.SLDASM* in the examples folder, next to the part file, or choose a name and location of your own choice.

When saving the assembly you may receive a message that referenced models have been modified:

*Some of the models referenced in this document have been modified.  
They must be saved when the document is saved.*

*Save the document and the referenced models now?*

You receive this message if using a newer version than the one the tutorial part was created with and you can safely answer *Yes*.





 It is almost always better to save machining operations and toolpaths as an assembly. This helps maintain associativity with the model by alerting you of any changes when the assembly is opened.

For more information see [Machining Assemblies](#).




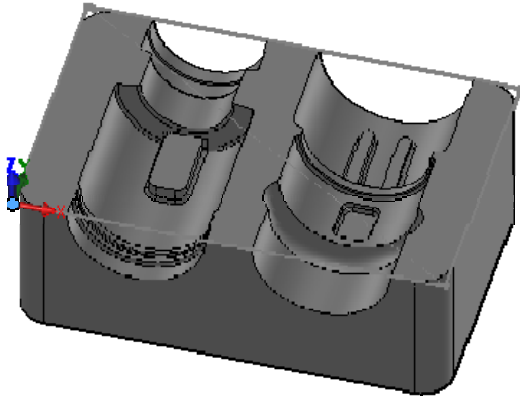
## Create a Coordinate System

First you have to create the coordinate system specific for the following operations.

- From the **Insert** menu select  **Reference Geometry** and  **Coordinate System**.
- In the model window click the  in front of  **Tutorial4** to unfold the feature tree.
- In the feature tree, locate **Sketch1** on the **Extrude1** feature in the **Tutorial4** component.



- Right click **Sketch1** and choose  **Show**.
- Select the point on the sketch as is shown below. This will make it the **Origin** of your coordinate system.






Notice that the coordinate system is now drawn at this position and the point is selected in the Origin field on the property page.


**i** Depending on your machine set up, you can change the location of the origin and the direction of the axes by selecting different entities here. For example, if the X and Y axes don't have the correct direction you can select a line of the sketch for **X axis** and another for the **Y axis**. It is also possible to select any plane or planar face which gives the axis perpendicular to that face or plane.

- Click  at the top of the property manager.

## Job settings




- Go to the HSMWorks operations manager by clicking on the *Machining Operations* tab: 
- Right click on  Tutorial4 Operation(s) and select  **New Job**.

### Set the Working Coordinate System (WCS)

 When creating a new job, the default WCS can often be used.

The default WCS depends on the the design of the assembly or part. In this case HSMWorks detects that a single coordinate system exists and uses that for the default WCS.

In the following section, we show how to manually select a coordinate system for the WCS even though the default could be used. This allows you to change the WCS if you are not happy with the default selection, or if you have created more than one coordinate system feature.

- Scroll down to the group **Working Coordinate System (WCS)**.
- In the drop down menu **Tool view selection**, make sure that **Use Coordinate System** is chosen.
- Make sure that the  **WCS origin** field is active by clicking on it.
- In the model window click the  in front of  Tutorial4 to expand the Assembly tree.
- Make sure that the **Coordinate System1** feature is selected:



## Define the stock

**i** Defining the stock on a simple example like this is not strictly necessary since by default HSMWorks uses an *Automatic* stock as the bounding box of the model.

Defining the stock to match that actually used on the machine does, however, make the solid verification more accurate.

On machining assemblies with multiple parts and/or fixtures in the assembly, this exercise will prove important.

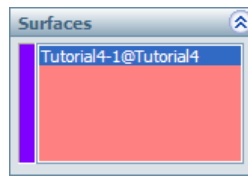
- Scroll up to the **Stock**.
- From the drop down menu select **Automatic**.
- Set the values **Offset X** and **Offset Y** to 0mm and **Offset Z** to 1mm.

## Define the surfaces

**i** Again, the defaults can be used, but we perform the selection manually to show how to correctly define surfaces in an assembly with more than one component.

- Scroll down to the group **Surface**.
- Click the selection field to set the selection focus.


- Click anywhere on the model in graphics window to select the part as the machining surfaces.



- Click  at the top of the property manager.

## Facing

To clear the top face of our stock and ensure that it is completely horizontal, we will begin with a facing operation.

- Click  **Face** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

This creates a new operation and opens the *property page* where you can edit the individual parameters controlling the toolpath, as well as selecting the actual geometry to machine.


The property page is divided into number of groups, and in this tutorial we will go through each one by one changing the necessary settings in each group as we go along.

In the following, we will go through each group on the property page explaining what to change. However, any group or parameters not mentioned should keep their default values.

### Tool View

By default, all new operations default to *Use WCS* as the Tool View. Since we have already configured the WCS in the Job, we don't need to change the Tool View in this operation - or any of the following.


### Tool

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#1 - 50mm flat**
- Press **Select**

You will be presented with a dialog message informing you that a copy of the tool will be created in the current library (the document). Press **OK**.

### Geometry

For the facing operation we can use the Sketch1 feature that we made visible when creating the coordinate system.



- Make sure that the  **Geometry Selection** field is focused.
- From the feature tree, locate **Sketch1** like we did when creating the coordinate system feature.
- Click **Sketch1** to select it as the facing geometry.

We are now done using the sketch, and can hide it to avoid unnecessary items shown on our model.

- Right click **Sketch1** and choose  **Hide**.

## Face

The parameters in this group controls how the actual facing toolpath is laid out.

When you selected the 50mm tool, the  **Stepover** and  **Pass Extension** parameter were automatically updated to reflect the new tool diameter. We will leave the parameters at their defaults, except the pass extension, which we can lower.

- Change **Pass Extension**:



## Linking

The parameters in the linking group control the tool motion between cutting passes.

We can use the default values in this example.

## Leads

The parameters in the leads group control the tool motion when transitioning from a cutting pass to a linking pass.


We can use the default values in this example.

## Filtering

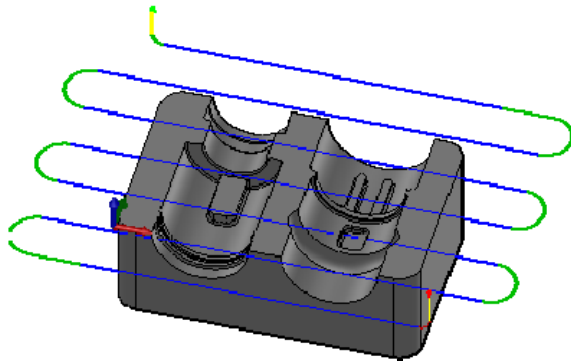
The parameters in the filtering group control how the output is processed when it is post processed.

We can use the default values in this example.

## Start Calculation

- Click  at the top of the property manager. This will automatically start calculation of the toolpath.

The toolpath will now be calculated and a preview will be shown in the model view:




By default the cutting parts of a toolpath are colored 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.





## Contouring

We want to run a contouring toolpath along the outer edges of the part to create the rounded corners and to finish the walls properly.

- Click  **2D Contour** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool


 By default, newly created operations use the same tool as the previous operation. In this case, we get the tool from the from the facing operation selected. This is not useful for this operation, so we will select a different tool instead.

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#2 - 16mm flat**
- Press **Select**

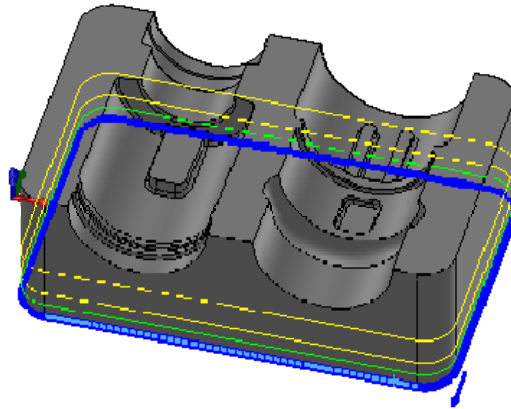
You will be presented with a dialog message informing you that a copy of the tool will be created in the current library (the document). Press **OK**.

### Geometry

Here we select the geometry we want to machine. We want to run the tool around the outside edge of the part.

-  Select the edges to be machined. Move the mouse over the bottom front edge, it will be highlighted, then click on it. Depending on the side of the edge you click you can determine the direction of the contour. By clicking closer to the desired start of an edge you can determine the direction of the contour. For climb milling click

close to the bottom right side of the edge. After you finished your geometry selections, the graphics window should look like this:



## 2D Contour

This group is specific to the selected machining operation, we want to generate a **2D Contour** toolpath. We will leave all settings in this group unchanged.

To machine the contour in steps of 10mm, set these parameters:

- Enable **Multiple Depths**:

Multiple Depths

- Change **Maximum Roughing Stepdown**:

 10.0mm

To avoid leaving marks at every step, we can do a single finishing pass at the final depth by setting these parameters:

- Enable **Finish Only at Final Depth**:

Finish Only at Final Depth

- Enable **Multiple Finishing Passes**:

Multiple Finishing Passes

- Change **Stepover**:









 1.0mm

- Change **Number of Finishing Passes**:

 1

## Linking

Since we have the tool view origin at the top of the part, and since we selected the geometry around the bottom of the model, we can set some meaningful heights easily:

- For  **Clearance Height** press 
- For  **Retract Height** press 
- For  **Feed Height** press 
- For  **Top of Stock** press 

Notice that this changes the preview of the heights on the model. If you change the view to the side of the part (e.g. by pressing Space and then choosing Bottom), the heights should look like this:

Since our job stock is set to have 1mm Z offset, we need the contouring toolpath to go below the height of the selected geometry.

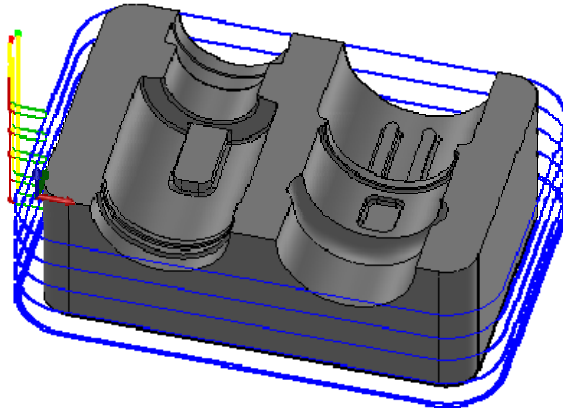
- Change **Depth**:

 -2.0mm

## Start calculation

- Click  at the top of the property manager.

The toolpath will now be calculated and shown in the graphic area.




By default the cutting parts of a toolpath are colored 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.

Continue to [Adaptive Clearing...](#)


## Adaptive Clearing

In this tutorial, we will use the Adaptive Clearing strategy to rough out the bulk of material.


 Adaptive Clearing is a modern HSM (High Speed Machining) strategy designed for roughing on modern machines capable of running complex NC files at high speeds.

The defining parameters in Adaptive Clearing is the *Optimal Load* and the *Stepdown*, but whereas traditional roughing strategies requires you to set the load (or stepover) and stepdown for the worst case scenario, Adaptive Clearing allows you to use the recommended maximum values provided by your tool vendor. This is possible since the specified *Maximum Load* is guaranteed not to be exceeded.

The Maximum Load is the Optimal Load + the Load Deviation.

- Click  **Adaptive Clearing** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#11 - 10mmx1mm bullnose**
- Press **Select**

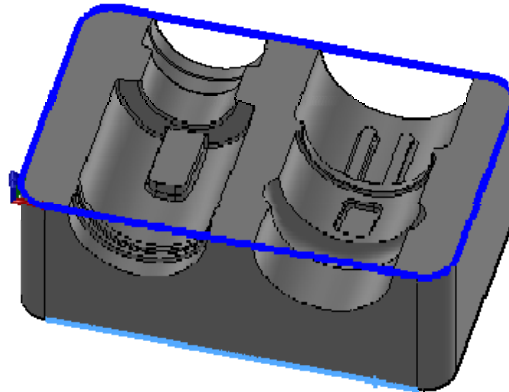
### Geometry

The parameters in this group control which areas to machine, as well as the amount of material to leave.

Since we have already cleared the outside of the part, we can define the stock from the silhouette of the part.

- Click the **Stock contours** selection field.

- Select the one of the edges along the bottom of the part. This will automatically create a chain around the entire part since the **Tangent propagation** and **Propagate along Z** options are enabled by default.



The **Stock to leave** parameter control the amount of material to leave in the radial (in the XY-plane) and axial (along the Z-axis) directions. The default values of 0.5mm are suitable in this example, so we will leave them unchanged.

Other settings in this group we will leave unchanged.


### Adaptive Clearing

The parameters in this group control the for Adaptive Clearing passes.

In this tutorial we will use the default parameters. Generally, however, you will have to look up the values for the stepdown and load parameters in your tool catalog.

### Confinement

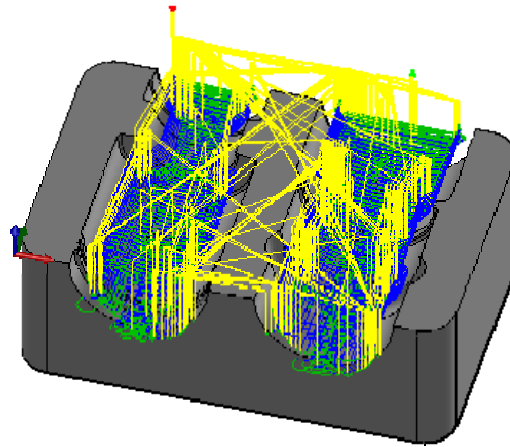
With Adaptive Clearing, the best way to machine this part is to allow it to start outside the part and work its way into each of the two slots.

To allow the tool to start outside the part, make sure that  **Machining Boundary** is *None*.

### Start calculation

Click  at the top of the property manager. This will start generation of the toolpaths.


When the toolpath generation is complete, the resulting toolpath should look like this:

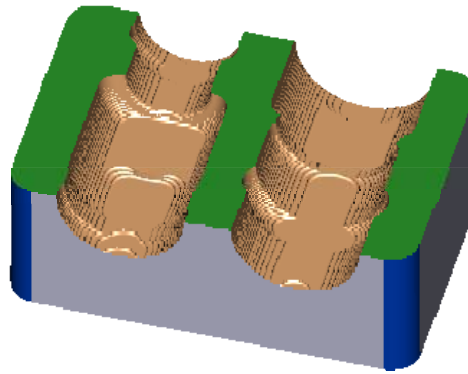


By default the cutting parts of a toolpath are colored in blue, lead moves in green and rapid moves in yellow. The start and end of the toolpath are indicated by a red and a green triangle respectively.

### Verification

To verify the toolpath use the solid verification.

- Right click on the top of the tree **Operation Manger**
- Select **Solid Verification (All)**
- Press the Start button  and watch the stock simulation up to the final stock shape.



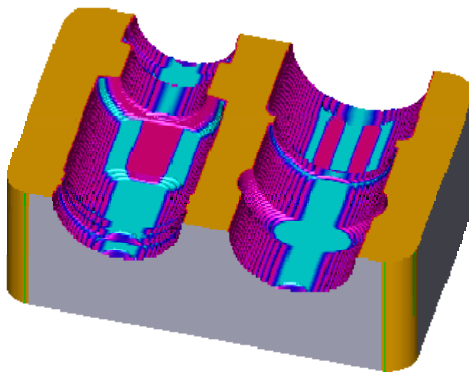
**i** If you want to see the entire model during the solid verification right click on the screen and select **Fit to window** in the bottom of menu. You can also switch to various views to see the final part from exactly directions.

### Verification Comparison

Once the verification is complete, you can do a *comparison* with the stock by using the stock compare feature.

- Press  to compare


This will color the verified part depending on the amount of material left:



The color in comparison depends on the amount of material left. Red is used



where there is little stock left while light blue is used where there is a lot of stock left.

 You can measure the exact amount of stock left by moving your mouse over the model and watching the *Distance* display update in the *Stock* group on the property page.


Additionally, the *Comparison* group lets you control the number of steps in calculated in the comparison, as well of the size of the steps. The default is 100 steps of 0.01 mm. This means that the coloring is done between 0.0 and 1.0 mm stock in 100 steps. Any area with 1.0 mm stock left or more will be colored light blue.


The comparison clearly shows that there are a number of areas that our 10mm tool cannot reach. In the next step we will rough these areas using *rest machining*.

## Adaptive Clearing Rest Machining

To rough out the remaining stock where the 16 mm tool could not fit, we will remove the rest material using the Adaptive Clearing with a smaller tool.


We will start by making a copy of the previously created operation.

- Right click on the  *Adaptive1* operation
- Choose **Duplicate**
- Right click on the newly created operation and select **Rename** from the context menu.


 Alternatively, you can use F2 key to rename.

- Change the name to **Adaptive2-Rest**.
- Right click on the new operation
- Choose **Edit**

This will open the property page for the new operation. Notice that all parameter values are the same as the Adaptive Clearing operation created before.

 In many cases creating a copy (using Cut and Paste, or the Duplicate option) is faster than creating a new operation since you often use similar parameter settings in successive operations.

### Tool

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#14 - 5mmx0.5mm bullnose**

- Press **Select**

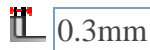
You will be presented with a dialog message informing you that a copy of the tool will be created in the current library (the document). Press **OK**.

### Geometry

In this group only change stock to leave.

Both values (radial and axial stock to leave) set to value **0.3mm**.

- Change **Radial Stock to Leave**:



- Change **Axial Stock to Leave**:



### Adaptive Clearing

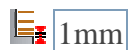
- Change **Optimal Load**:



- Change **Maximum Roughing Stepdown**:




- Change **Fine Stepdown**:



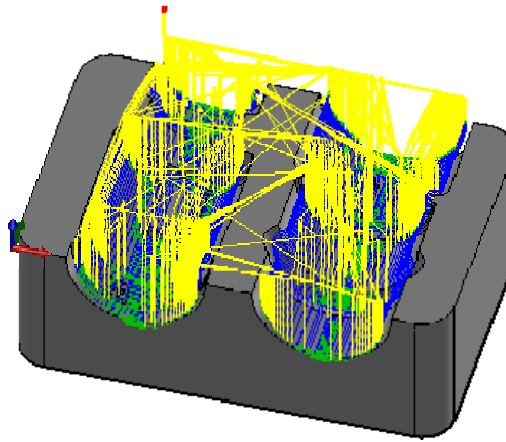
### Rest Machining

- Set **Rest machining source** to *From previous Operation(s)*

### Start Calculation

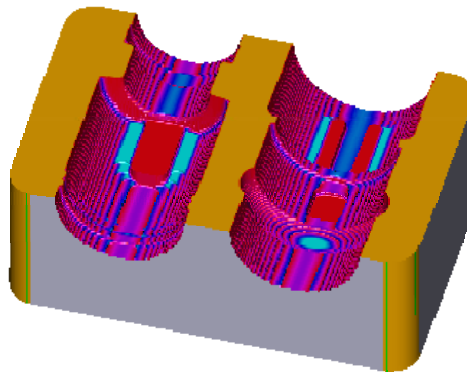
Click  at the top of the property manager. This will start generation of the toolpaths.

When the toolpath generation is complete, the resulting toolpath should look like this:



### Verification


As in the previous step, you can run the solid verification to visualize the remaining material:



There is now primarily material left in the narrow slots on the part.

## Horizontal

In next operation we will use *Horizontal strategy* to machine the horizontal parts of the part.

- Click  **Horizontal** on the HSMWorks toolbar or select it from the **HSMWorks Toolpaths** menu.

### Tool

The 5mm bull nose tool used in the adaptive rest operation can be used for clearing the small horizontal areas left on the part, and using this will save a tool change.

### Horizontal Finishing

The Adaptive Clearing operations have left a maximum of 0.3mm (Stock to leave) + 1.0mm (fine stepdown) + 0.1mm (tolerance) = 1.4mm. This is a little too much for the tool to remove in one pass, so we will do it in 3 x 0.4 mm passes instead.

- Enable **Axial Offset Passes**:
- Change **Maximum Roughing Stepdown**:

Axial Offset Passes

 0.4mm


- Change **Number of Stepdowns**:

 3

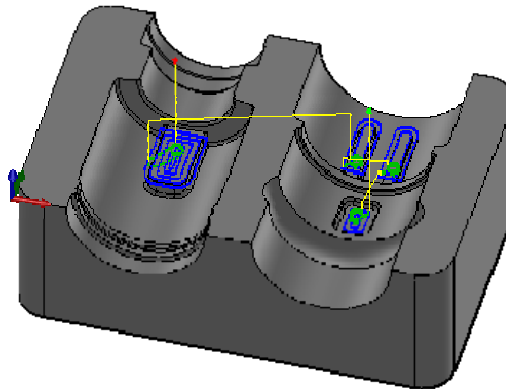
### Linking

- Change **Retraction policy** to *Minimum Retraction*.

### Start Calculation


Click  at the top of the property manager. This will start generation of the toolpaths.

When the toolpath generation is complete, the resulting toolpaths should look like this:



## Contour


In this operation we will use contour finishing strategy to finish the steep areas of the part.

- Click  **Contour** on the HSMWorks toolbar or selected it from the **HSMWorks Toolpaths menu**.

This creates a new operation and opens the Property manager where you can edit the individual parameters controlling the toolpaths, as well as selecting the actual geometry to machine.

### Tool

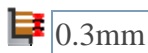
For this operation we will need a 6mm ball end mill. Again, we select this from the **Tutorial** sample library.

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#21 - 6mm ball**
- Press **Select**

### Contour Finishing

In this group change only following settings:

- Change **Maximum Stepdown**:



- Disable **Flat Area Detection**:

Flat Area Detection

### Confinement

- Change **Tool Containment** to *Tool Center on Boundary*.

- Change **Additional Offset**:



- Change **From Slope Angle**:

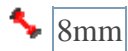


### Linking

- Change **Retraction policy** to *Minimum Retraction*.
- Change **Safe Distance**:



- Change **Maximum Stay-Down Distance**:

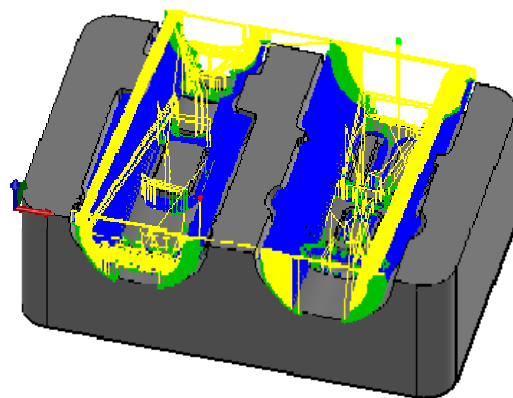


### Start Calculation

Click  at the top of the property manager. This will start generation of the toolpaths.

When the toolpath generation is complete, the resulting toolpath should look like this:





## Parallel

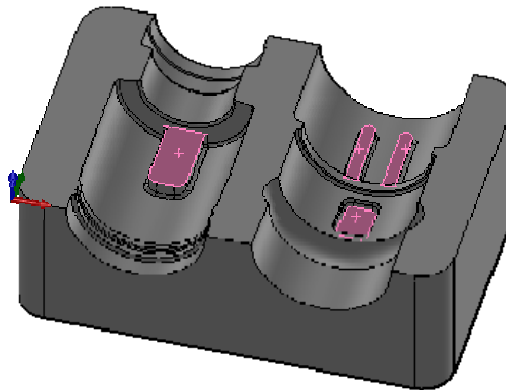
In this operation we will use the parallel finishing strategy to machine the part .

- Click  **Parallel** on the HSMWorks toolbar or select it from the **HSMWorks Toolpaths** menu.

### Geometry

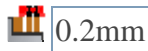
We want to finish the shallow areas of the part. Specifically those under 30 degrees not reached by the contour finishing operation. But since we have already finished the flat areas with the horizontal operation, we want to prevent the tool from touching these flat surfaces. For this purpose we can use check surfaces.

- Enable **Use check surface**:
  - Use check surface
- In the **Check surface selection** field, select the flat faces machined in the horizontal strategy:



### Parallel Finishing

- Change **Stepover**:



- Change **Pass Direction**:



### Confinement


The contour operation have covered the slope range from 30 to 90 degrees. Thus, if we specify that this parallel operation should machine from 0 to 35 degrees this gives us a 5 degree overlap between the two operation.

- Change **To Slope Angle**:



### Linking

- Set  **Top of Stock** to -1mm

 When you set the Top of Stock value to 1mm under the top plane of the part, the tool will machine only inside cavities of the part and not the on the upper surface.

Alternatively we could have included the top surfaces in the check surface selection.


- Change **Retraction policy** to *Minimum Retraction*.
- Change **Safe Distance**:



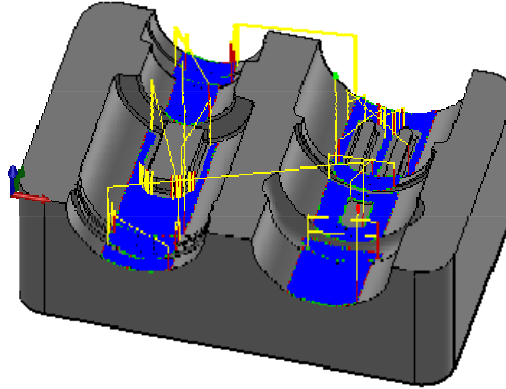
- Change **Maximum Stay-Down Distance**:



### Start Calculation

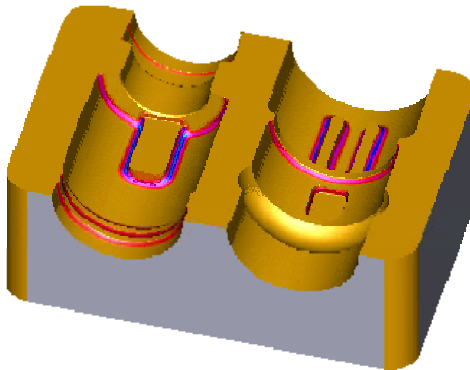
Click  at the top of the property manager. This will start generation of the toolpaths.

When the toolpath generation is complete, the resulting toolpath should look like this:



### Verification

By using the Comparison function in Solid Verification, we get the following illustration of the material remaining.




The orange colored areas have 0.0 mm stock remaining, while the red to blue areas have more material left.

Moving your mouse over the surface in the Solid Verification window shows the amount of stock left at that position. The distance is shown in the Stock group of the property manager page.

From the comparison we can see that we have now finished most areas of the part, but that we are still missing material in the smaller fillets.

## Pencil 1

To remove the material in the smaller fillets we can use the Pencil finishing strategy.

 The Pencil strategy is a special finishing strategy which detects inner corner edges on the surface and creates a toolpath here. This method is particularly effective for finishing small radius fillets.

- Click  **Pencil** on the HSMWorks Toolbar or select it from the **HSMWorks Toolpaths** menu.

This creates a new operation and opens the Property manager where you can edit the individual parameters controlling the toolpath, as well as selecting the actual geometry to machine.


The property page is divided into number of groups, and in this tutorial we will go through each one by one changing the necessary settings in each group as we go along.

### Tool

By looking at the comparison in the previous step, we could see that the maximum amount of material left is about 1.4mm.

Also, by investigating the model, we find that we have some fillets with a radius of 1.0mm and some smaller ones of 0.5mm.

In this pencil operation we will remove most of the material not reachable by the previous tools by using a 3mm ball mill.

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#22 - 3mm ball**
- Press **Select**

### Pencil Finishing

- Change Number of Steppers:



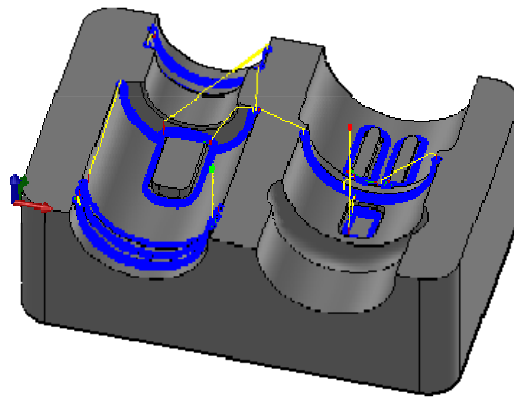
- Change Stepper:



### Start Calculation

Click  at the top of the property manager. This will start generation of the toolpaths.

When the toolpath generation is complete, the resulting toolpath should look like this:




## Pencil 2

Since the 3mm tool does not remove all the material in the R1.0mm and R0.5mm fillets we will create a second pencil operation with a 2mm ball mill to handle the 1mm fillets.

- Click on operation *Pencil1* and select **Duplicate**. This will copy the operation Pencil1 and create a copy of this operation called *Copy of Pencil1*.
- Click on newly created operation and select **Rename**.
- Change name to **Pencil2**.
- Right-click on the new operation and choose **Edit** to open the parameter page.

### Tool

- Press  **Select**
- From the library **Tutorial** under **Sample Libraries**, select tool #23 - **2mm ball**
- Press **Select**

### Pencil Finishing

- Change **Number of Stepovers**:



- Change **Stepover**:

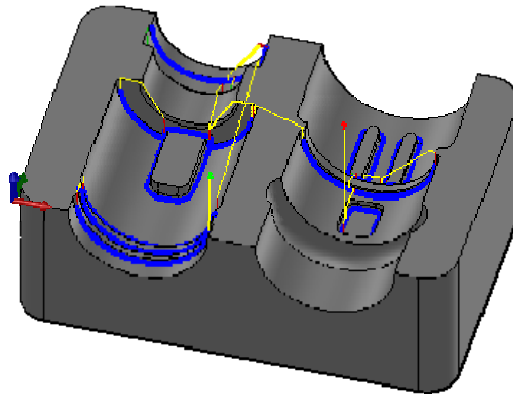


### Start calculation



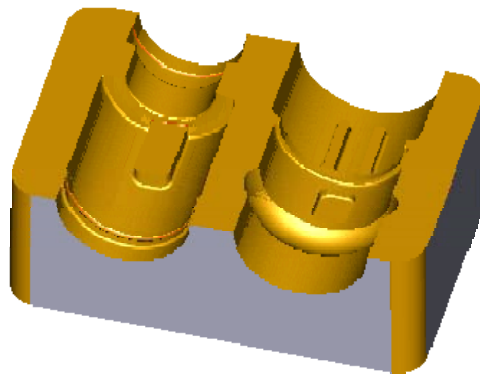
Click  at the top of the property manager. This will start generation of the toolpaths.

When the toolpath generation is complete, the resulting toolpath should look like this:



### Verification

By using the Comparison function in Solid Verification, we get the following illustration of the material remaining.




By zooming in on the comparison we see that we have are still missing a bit of material in the fillets with radius 0.5mm.

## Pencil 3

Since the 2mm tool does not remove all the material in the R0.5mm fillets we will create a third pencil operation with a 1mm ball mill to handle the 0.5mm fillets.

- Click on operation *Pencil2* and select **Duplicate**. This will copy the operation Pencil2 and create a copy of this operation called *Copy of Pencil2*.
- Click on newly created operation and select **Rename**.
- Change name to **Pencil3**.
- Right-click on the new operation and choose **Edit** to open the parameter page.

### Tool

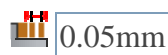
- Press  **Select**
- From the library **Tutorial** under **Sample Libraries**, select tool **#24 - 1mm ball**
- Press **Select**

### Pencil Finishing

- Change **Number of Stepovers**:



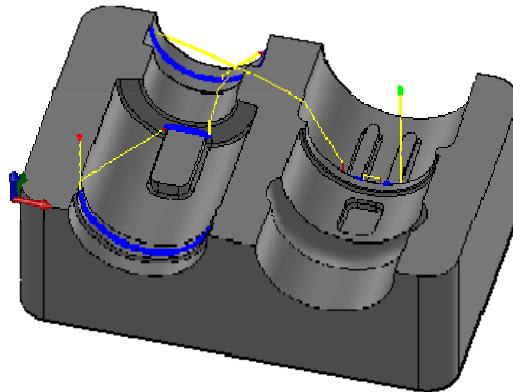
- Change **Stepover**:



### Start calculation

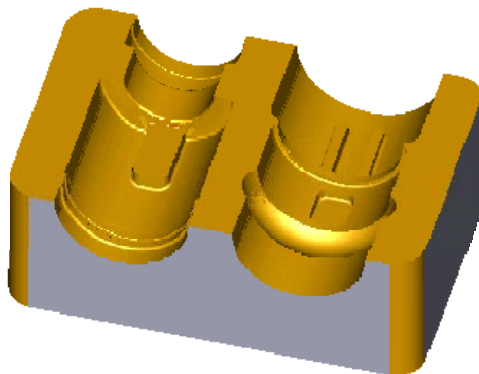
Click  at the top of the property manager. This will start generation of the toolpaths.

When the toolpath generation is complete, the resulting toolpath should look like this:



### Verification

By using the Comparison function in Solid Verification, we get the following illustration of the material remaining.






We have essentially finished our part to within the tolerance specified in the various operations, and the only stock that is remaining is that between the steppers of the various toolpaths.

If we want an even better finish, you can edit the operations and use a higher tolerance and smaller steps in the toolpath.

## Post Processing

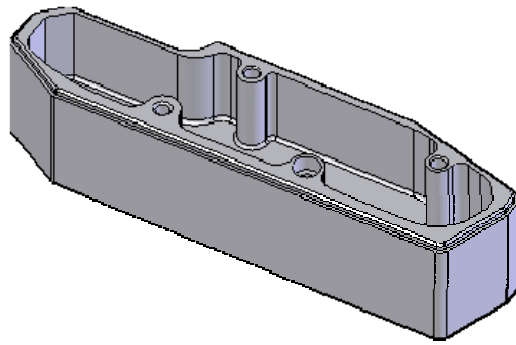
We are now ready to post process the toolpaths in order to make the NC-code which can be used by the machine tool.

- Right click on  **Job** in the operation manager.
- Select  **Post process (All)**.
- From the pulldown Post processor configuration select **heidenhain.cps Generic Heidenhein**.
- Select an output folder of your choice.
- Start the post processor by clicking  button.

By default the post processed file will be opened in **HSMWorks Edit** which allows you to inspect the generated NC-code as well as transferring it to your machine.

**Congratulations!** You have completed this tutorial.


## Tutorial 5 - More 2D Machining



In this tutorial we will learn how to machine a 2.5D part using the following steps:




- Create a Machining Assembly
- Sketch the Stock
- Create a Coordinate System
- Job Settings
- 2D Contour 1
- Pocketing
- Drilling
- Tapping
- Drilling 2
- Circular Pocketing
- Counter Sinking
- Chamfering
- Post Processing

Before proceeding, open the part **Tutorial5.SLDPRT** into HSMWorks.

 The files used in this manual can be found in the **examples** folder in the folder where HSMWorks is installed. Typically, this is something like  
**C:\Program Files\HSMWorks\examples\**

### Create a Machining Assembly

Before proceeding we will make a new *machining assembly* to keep the machining operations and toolpaths in.


- From the File menu choose  **Make Assembly from Part**. This will create a new assembly file, and preselect the already opened part document as the component to insert.
- Click  at the top of the property manager.
-  **Save** the new assembly as *Tutorial5.SLDASM* in the examples folder, next to the part file, or choose a name and location of your own choice.

When saving the assembly you may receive a message that referenced models have been modified:

*Some of the models referenced in this document have been modified.  
They must be saved when the document is saved.*

*Save the document and the referenced models now?*

You receive this message if using a newer version than the one the tutorial part was created with and you can safely answer *Yes*.

 It is almost always better to save machining operations and toolpaths as an assembly. This helps maintain associativity with the model by alerting you of any changes when the assembly is opened.  
For more information see [Machining Assemblies](#).

## Sketch the Stock

Since we need an entity to place our origin on, and because having the stock defined makes the solid verification more accurate, we will set up the stock for this part.

**i** When you have a part without any geometry at the machining origin, or if there are no planes in the machining direction, you may need to construct helper geometry to be used from HSMWorks.

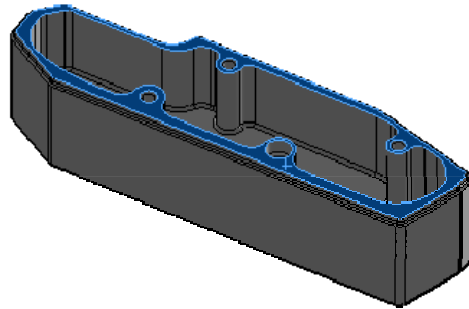
This part, for example, we want to machine from a rectangular block of aluminum which is slightly larger than the part, and for easier setup we want keep the tool view origin at the corner of the stock piece.





**i** There are two ways to set up a stock in an assembly:

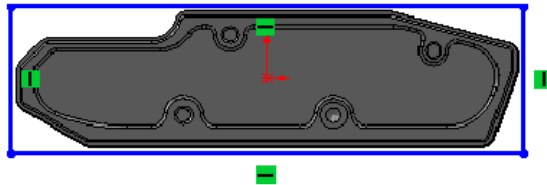
- Create a separate model representing your stock. *See [Tutorial 3 - 3+2 Machining](#) for an example of this.*
- Create a 2D sketch of the stock silhouette. *This works well if your stock is simple - like a block or a cylinder.*


Create a rectangular sketch, representing a block of stock measuring 154mm by 44mm (ie. slightly larger than the extends of the part):

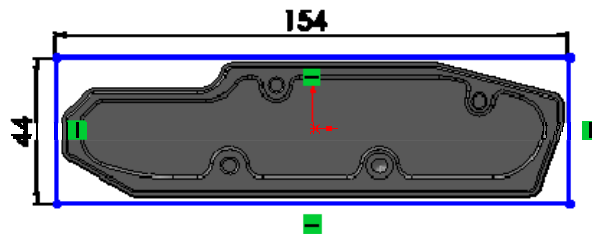
- Select the flat face on the top of the model:



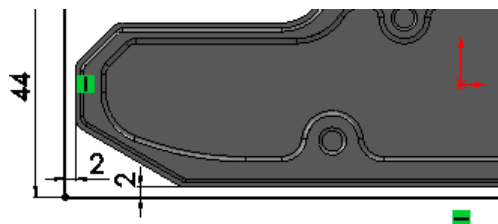
- Start a new Sketch on this face by clicking , available from the *Insert menu* or the *Sketch toolbar*
- Change the view to *Normal To*.  
Press *Ctrl+8* or choose from the *View toolbar*:  
- Click  to start sketching a rectangle.
- Sketch a rectangle enclosing the entire model, like this:



- Using the  Smart Dimension tool, add dimensions to the stock sketch, like this:





- *Optional*: Add extra dimensions between the sketch and the part, to fixate it:

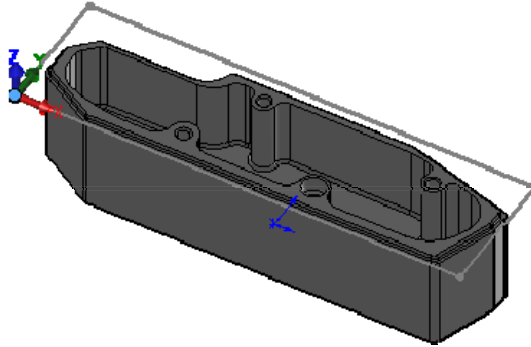





- Now, finish the sketch by clicking  in the upper right corner of the graphics window.




## Create a Coordinate System

- From the **Insert** menu select  **Reference Geometry** and  **Coordinate System**.
- Select the point in the lower left corner of your sketch. This will make it the **Origin** of your coordinate system. *Notice that the the coordinate system is now drawn at this position, and that the point is selected in the Origin field on the property page.*






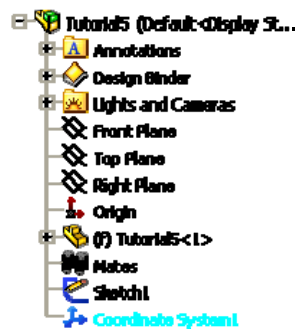
- *Optional:* Select the lower line in the sketch. This will make it the **X axis**.
- *Optional:* Select the left line in the sketch as. This will make it the **Y axis**.
- Click  at the top of the property manager.

## Job Settings


- Go to the HSMWorks operations manager by clicking on the *Machining Operations* tab: 
- Right click on  **Tutorial5 Operation(s)** of the operation tree and select  **New Job**.

### Set the Working Coordinate System (WCS)

- Scroll down to the group **Working Coordinate System (WCS)**.
- Make sure that **Use Coordinate System** is chosen in the Tool View Selection box.
- Make sure the  WCS selection field is highlighted.
- In the model window click the  in front of  Tutorial5 to unfold the Assembly tree.
- Select **Coordinate System1**:






### Define the Stock


 Defining the stock on a simple example like this is not strictly necessary, but it does make the solid verification more accurate.

On machining assemblies with multiple parts and/or fixtures in the

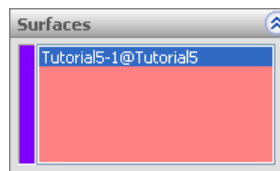
assembly, this exercise will prove important.


- Scroll up to the group **Stock**.
- From the drop down, change **Automatic** to **From extruded sketch**.
- Select  **Sketch1** from the feature tree.  
*Alternatively select the sketch directly on the model by clicking on one of the lines.*
- In the  **Offset Z** field, enter 1mm.
- In the  **Height** field, enter 36mm.

### Define the Surfaces

 As for the stock, defining the machining surfaces are not strictly necessary.  
Again, we do it here as an exercise for more complicated setups.


- Scroll down to the group **Surfaces**.
- Click the selection field to set the selection focus.
- Click the model in the model view window, to select the part as the machining surfaces:



Click  at the top of the property manager.

## 2D Contour 1

First of all we want to machine the outer planes of the part.  
We want to run the tool along the outer edges of the part.

- Click  **2D Contour** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

This creates a new operation, and opens the *property page* where you can edit the individual parameters controlling the toolpath, as well as selecting the actual geometry to machine.


The property page is divided into a number of groups, and in this tutorial we will go through each one by one, changing the necessary settings in each group as we go along.

### Tool View

The Tool View group allows you to select a different orientation to machine from.

In this example, the default (Use WCS) can be used since we have already defined the WCS in the Job.


### Tool

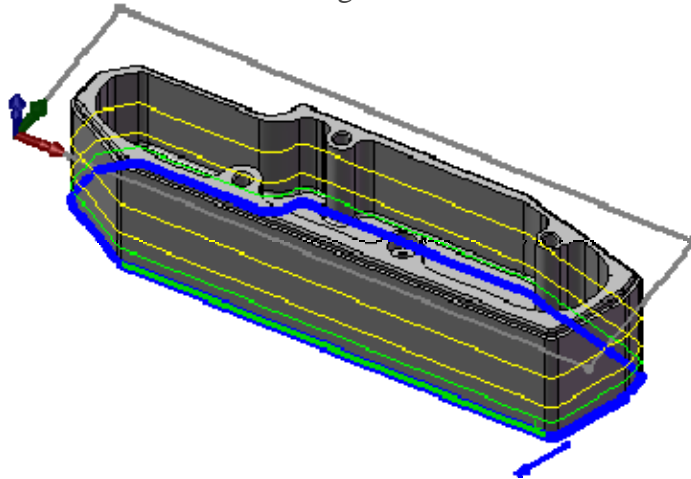
- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#3 - 10mm flat**
- Press **Select**

You will be presented with a dialog message informing you that a copy of the tool will be created in the current library (the document). Press **OK**.

### Geometry


We want to machine around the outside outline of the part. To do this, select a chain of edges from the model:

- Click the  Geometry selection field
- Select the bottom front edge on the model:



- If the direction arrow does not appear as shown, change the direction by clicking 

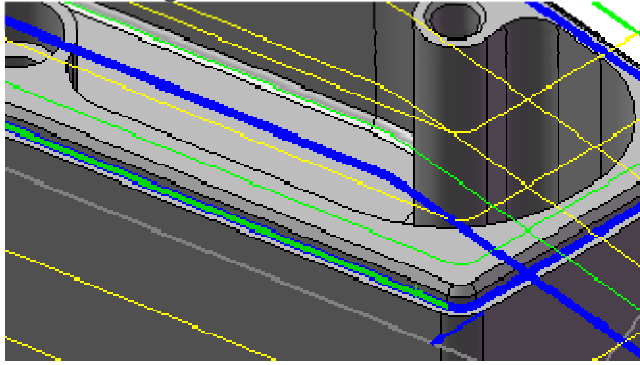
Notice that HSMWorks automatically creates a chain around the part. This is because **Tangent propagation** and **Propagate along Z** are enabled by default.

 Depending on which end of the edge you click you can determine the direction of the contour.  
By clicking closer to the desired start of an edge you can determine the direction of the contour. For climb milling click close to the bottom right side of the edge.  
For more information see [Selecting Chains](#).

We can machine the indent around the top of the part in the same operation:

- Zoom in on the top right corner of the part

- Select the edge inside the indent:




## 2D Contour

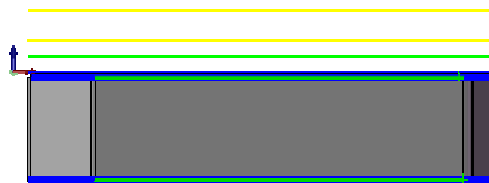
*We will leave all settings in this group unchanged.*

## Linking

Since we have the tool view origin at the top of the part, and since we selected the geometry around the bottom of the model, we can set some meaningful heights easily:

- For  **Clearance Height** press 
- For  **Retract Height** press 
- For  **Feed Height** press 
- For  **Top of Stock** press 


Notice how the preview of the heights on the model is changed. If you look from the side of the part (press Ctrl+6), it should look like this:



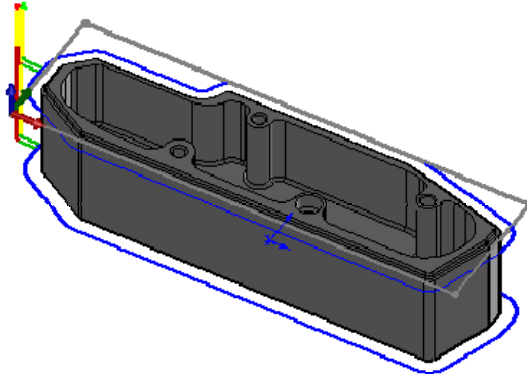
## Leads & Transitions

*We will leave all settings in this group unchanged.*

### Start Calculation

- Click  at the top of the property manager. This will automatically start calculation of the toolpath.

The toolpath will now be calculated and a preview will be shown in the model view:




By default the cutting parts of a toolpath are colored 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.



## Pocketing

As the next operation we want to machine the internal pocket of the part. In this operation we will use the 2D Pocket strategy with a 10mm bull nosed tool with a 1mm corner radius.


- Click  **2D Pocket** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool View

Again, we can use the default (Use WCS) as we have already defined the WCS in the Job.


### Tool

In this operation we will use a tool with a corner radius of 1mm to match the fillet radius at the bottom of the pocket.

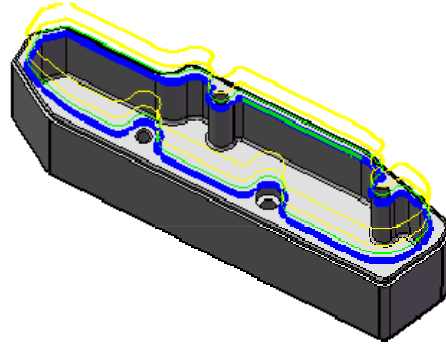
- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#11 - 10x1mm flat**
- Press **Select**

### Geometry

In this group we select the contour of pocket we want to clear.

- Make sure that both **Tangent propagation** and **Propagate along Z** are enabled for simpler chain selection
-  Select one of the edges at the top of the pocket

The selection should now look like this:



We want to leave some stock on the sides, as we will need to finish the fillets with a smaller tool later on anyway.

The bottom, however can be finished by the pocket, and we should remove the vertical stock to leave:

- Set **Axial Stock to Leave**:



## 2D Pocket

This group controls how the 2D pocket toolpath will be calculated. We want to use this pocket toolpath to clear out the pocket. To do this we want to generate the toolpath in a number of z levels, starting from the top of the stock and going down in steps of 2mm to the bottom of the pocket. The depth of pocket is 25mm.

Change the following parameter values, and leave all others at their default:

- Set **Maximum Stepper**:



- Enable **Multiple Depths**:



- Set **Maximum Roughing Stepdown**:



- Set **Finishing Stepdowns**:

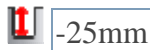



## Linking

The parameters in this group control the different height where toolpath is generated, as well as when to retract.

In this example, we will change only the depth:

- Change **Depth**:



 If you don't know the exact depth of your geometry, you can measure it directly on the model:

- Right click in the field
- Choose **Measure**
- Select any point on the model  
*In this example, you could select any vertex on the face on the bottom of the pocket*
- Click OK to bring the value into the field

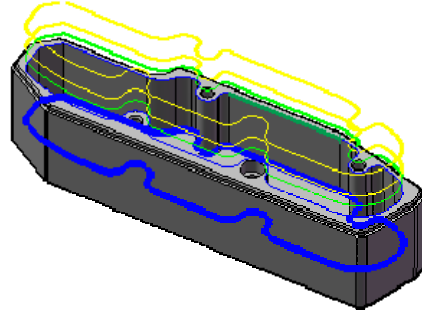
Assuming that this part is made of a soft material, where we can do full width cuts, we can avoid some ramping by allowing the tool to stay down inside the pocket:

- Enable **Keep Tool Down**:

Keep Tool Down

- Change **Maximum Stay-Down Distance**:

 250mm



Having changed the depth, the preview should now look like this.

### Leads & Transitions

We will leave all settings in this group unchanged. We will use a helix ramp for plunge to material with the default settings from the program.

### Filtering

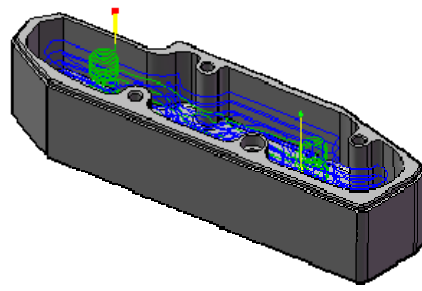
We leave the setting Preserve rapid movement and make no change.

### Start Calculation

- Click  at the top of the property manager.

The toolpath will now be calculated and shown in the graphic area. By default the cutting parts of a toolpath are colored 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.


The toolpath should look like in the picture below:



## Drilling

### Drilling

There are four holes in the part. One is clear and three are with thread. First, we will drill the three holes with a 4.3mm drill.

- Click  **Drill** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.


This creates a new operation, and opens the *property page* where you can edit the individual parameters controlling the toolpath, as well as selecting the actual geometry to machine.

The property page is divided into a number of groups, and in this tutorial we will go through each one by one, changing the necessary settings in each group as we go along.

### Tool View

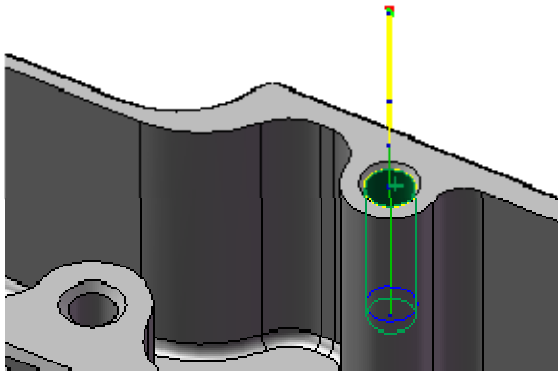
Again, we can use the default (Use WCS) as we have already defined the WCS in the Job.

### Tool

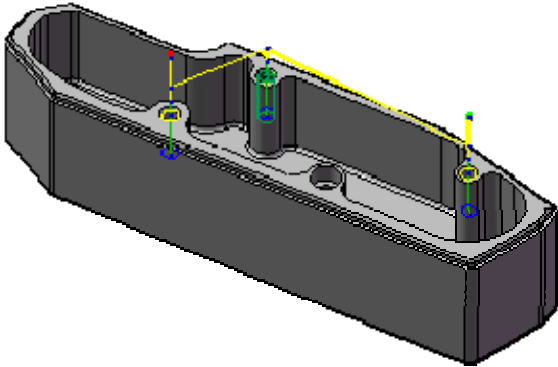
- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#33 - 4.3mm drill**
- Press **Select**

### Geometry

Select the cylinder of one of the three 4mm holes



- Check **Select same diameter**



*Notice that the two other 4mm holes are selected.*

**i** If **Select same diameter** is disabled (grayed out), you have most likely selected the edge of the cylinder instead of the face. You need to select cylindrical faces to use this feature.

When selecting faces instead of edges, it has the added benefit that the depth of each hole is automatically determined from the height of the cylinder.

Depending on the selection, you may want to enable **Optimize order**. The optimization reorders the holes in order to make the linking distance as short as possible.

If you check **Order by depth**, this will keep holes with the same depth and plane together. This may reduce the number of cycles in the output, but may not result in the shortest toolpath, even if Optimize order is enabled.

## Hole

This group controls the heights of each hole.

*Having selected cylinders for our holes, all heights are relative to the top of the cylinder, and the depth is relative to the bottom.*

*This allows us to avoid entering the depths of the holes manually. If we had selected edges (which do not have a height), we would have to enter the depth manually.*

Because of the chamfer on the holes, the cylinders start a bit lower, and we will adjust for this by making the top of stock absolute:

- For  **Top of Stock** press 


To ensure that the feed height follows the top of stock:

- For  **Feed Height** press 


## Cycle

In this group select type of the cycle **Chip breaking - partial retract**.

- Change **Pecking Depth**:

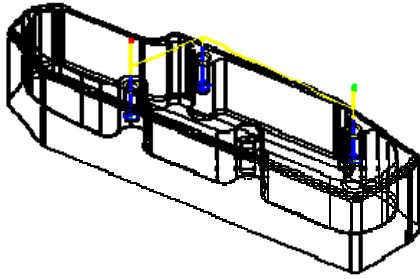


## Start calculation

Click  at the top of the property manager.

This will calculate the toolpath, which should look like this:





Drilling toolpaths are often best seen on a wireframe model.

Select  **Wireframe** from the **View, Display** menu or the View toolbar to change to wireframe mode.

## Tapping


This tapping operation only differs in the type of cycle from the previous drilling operation; tapping will be done for the same geometry. To save all the work of entering this data, we will just copy the existing drilling toolpath and create the tapping toolpath by editing this copy.

- Right-click on the toolpath Drill1, select **Duplicate**  
*This creates a copy of the operation below the original one.*
- Right-click on the new operation (Copy of Drill1)
- Choose **Rename**
- Type the new name, e.g. *Tapping M5*

Now we have to change the parameters and settings of the operation:

- Right-click on Tapping M5
- Select **Edit** from the context menu

### Tool

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#43 - 5mm right tap**
- Press **Select**

### Hole

Now, to avoid breaking the tap, we need to reduce the depth so it is less than the drilled depth:

- Change **Depth**:

 2.0mm


This will reduce the depth by 2mm.

**Cycle**

Now, to change the drilling cycle to a tapping cycle:


- Go to the **Cycle** group
- From the **Cycle type** dropdown, to **Tapping**

All other setting in the groups can be left unchanged.

Click  at the top of the property manager.

## Drilling 2

Next, we need to drill the single 4.5mm hole.

- Click  **Drill** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool View

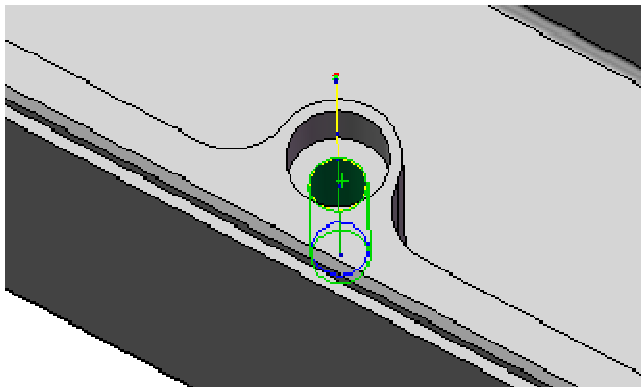
Again, we can use the default (Use WCS) as we have already defined the WCS in the Job.

### Tool

- Press
- From the library **Tutorial** under **Sample Libraries**, select tool **#32 - 4.5mm drill**
- Press **Select**

### Geometry

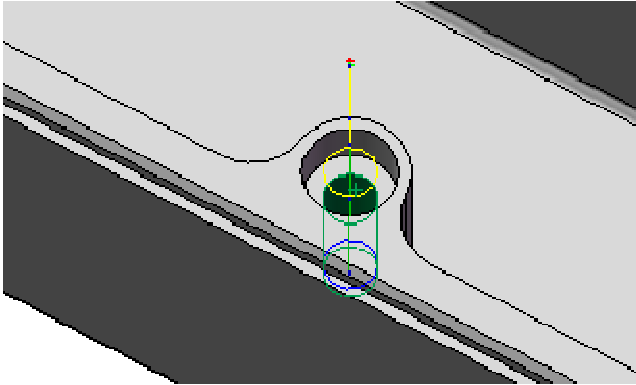
Select the lower cylinder of the last hole



### Hole


Because of the counter sink on this hole, the cylinder we have selected does not start at the top where we want to start our drilling. We adjust for this by making the top of stock absolute:

- For  **Top of Stock** press 
- For  **Feed Height** press 



### Cycle


In this group select type of the cycle **Chip breaking - partial retract**.

Click  at the top of the property manager.

Now you have finished all the operations in this tutorial.

## Circular Pocketing

To machine the counter boring on the single 4.5mm hole, we will use a the 2D pocket strategy with a 4mm flat end tool.


- Click  **2D Pocket** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool View

Again, we can use the default (Use WCS) as we have already defined the WCS in the Job.

### Tool

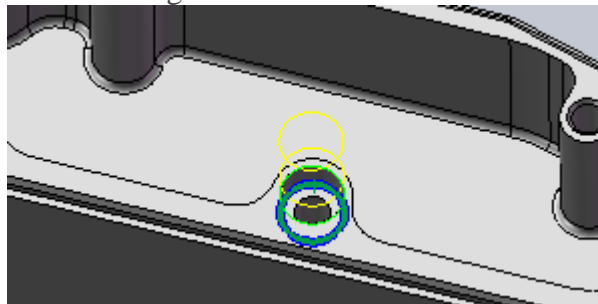
For this pocket we need a flat tool with a diameter less than 4.5mm so we can plunge into the pre-drilled hole.

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#5 - 4mm flat**
- Press **Select**

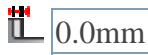
### Geometry

In this group we select the contour of pocket we want to clear.

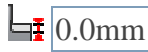
- Select the edge at the bottom of the counter-sink:



- Set **Radial Stock to Leave:**



- Set **Axial Stock to Leave**:

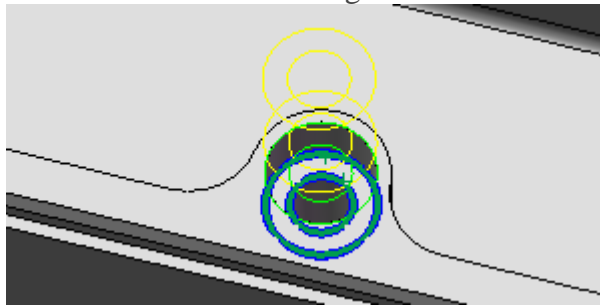


Since we have already drilled at this position, we can make the tool plunge into the drill hole, by enabling *Use pre-drilled hole(s)*:

- Enable **Use pre-drilled hole(s)**:



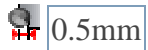
- Select the inner circular edge of the hole:



## 2D Pocket

Change the following parameter values, and leave all others at their default:

- Set **Maximum Stepover**:



## Linking

Since we have the tool view origin at the top of the part, and since we selected the geometry around the bottom of the model, we can set some meaningful heights easily:



- For  **Clearance Height** press 

- For  **Retract Height** press 
- For  **Feed Height** press 
- For  **Top of Stock** press 

### Leads & Transitions

Because we have set the pre-drill position, the lead-in and ramping parameters will not be used in this pocket.

The lead-out parameters will, however, and we can adjust this a bit to give a smoother exit:

- Disable **Lead-In (Entry)**:  
 Lead-In (Entry)
- Disable **Ramp**:  
 Ramp
- Set **Horizontal Lead-Out Radius**:  

- Set **Linear Lead-Out Distance**:  


### Filtering

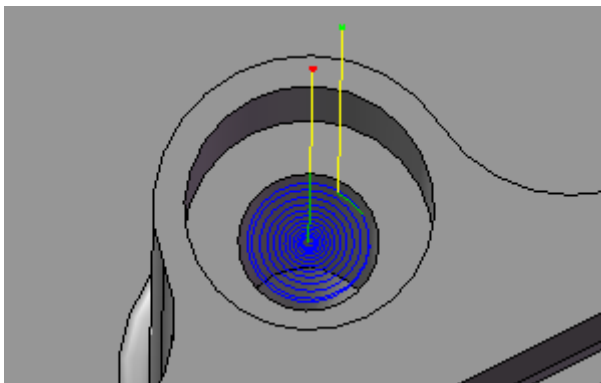
We leave the setting Preserve rapid movement and make no change.

### Start Calculation

- Click  at the top of the property manager.




The toolpath should look like in the picture below:



## Counter Sinking


Three of the drill holes have small counter sinks. We can machine all three in a single Drill operation by using a Counter Sink tool.

- Click  **Drill** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

### Tool View

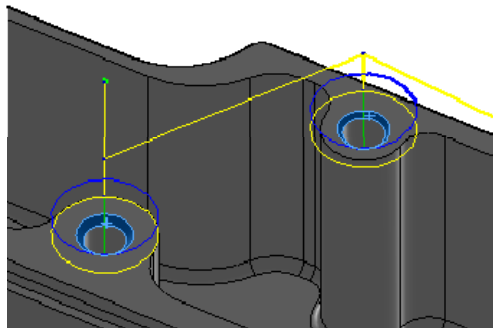
Again, we can use the default (Use WCS) as we have already defined the WCS in the Job.

### Tool

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#60 - 10mm counter sink**
- Press **Select**


### Geometry

- Check **Select same diameter**
- Select the counter sink faces of the two first holes:

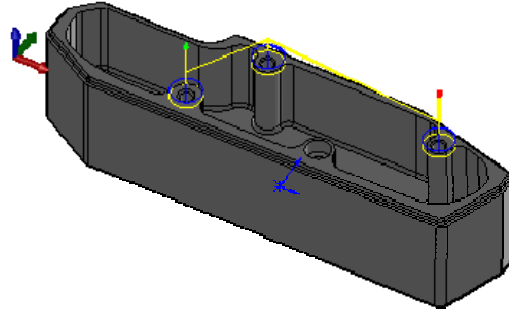


The counter sunk face of the third hole should be automatically selected since it is identical to the second.

Start Calculation


- Click  at the top of the property manager.

The toolpath should look like in the picture below:




## Chamfering

To machine the chamfer along the outside edge we can use the 2D Contour strategy with a chamfer mill.

- Click  **2D Contour** on the HSMWorks toolbar or select it from the **HSMWorks, Toolpaths** menu.

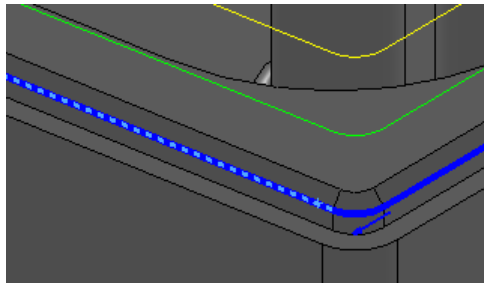
### Tool

- Press 
- From the library **Tutorial** under **Sample Libraries**, select tool **#50 - 10mm chamfer**
- Press **Select**

### Geometry

In this model, the chamfer already exists as a part of the model, which means that we have two edges to choose from as the geometry. One at the top of the chamfer and one of the bottom. HSMWorks allows either to be used, but in most cases choosing the bottom one allows for the easiest setting of parameters. In both cases, however, HSMWorks will automatically calculate the correct horizontal offset, and you only have to enter the additional vertical tip offset.

- Select the edge at the bottom of the counter-sink:



## 2D Contour

When choosing a Chamfer Mill as the tool, the **Chamfer** option was automatically enabled, thereby showing the chamfering parameters **Chamfer Width** and **Chamfer Tip Offset**.

Since we have chosen an edge on an actual chamfer feature in the model we don't have to set the Chamfer Width, but we do need to set the Chamfer Tip Offset so that the tool tip is not coincident with the lower edge of the chamfer:

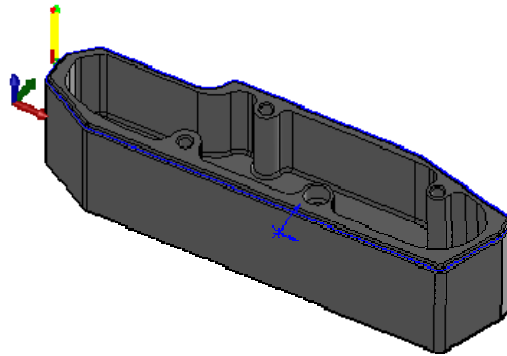
- Set **Chamfer Tip Offset**:



## Start Calculation

- Click  at the top of the property manager.



The toolpath should look like in the picture below:



This completes the toolpaths for this part and you can now simulate and post-process the result.

## Post Processing

We are ready to postprocess all toolpaths in order to make the NC-code which can be used by the machine tool.

- Right-click on  **Job** in the operation manager.
- Select  **Post Process (All)**.
- From the pulldown **Post processor configuration** select *heidenhain.cps - Generic Heidenhain*
- Select an output folder of your choice.
- Start the post processor by clicking **Post**. By default the post processed file will be loaded into HSMWorks Edit.

By default the post processed file will be opened in **HSMWorks Edit** which allows you to inspect the generated NC-code as well as transferring it to your machine.

**Congratulations!** You have completed this tutorial.

---

Generated for HSMWorks 2009 R0.16904.

Copyright © 2003-2009 by HSMWorks ApS

Email: [hsmworks@hsmworks.com](mailto:hsmworks@hsmworks.com)