Tutorial I

Getting started

In this tutorial, it is assumed that the user is conversant with the use of Mastercam, so ordinary actions in Mastercam are not explained in detail.

The file used for this tutorial is the MC9 file designated tutorial 1, which was included with this tutorial, or can be downloaded from the Mastercam HSM Performance Pack website (there is a link on the About tab in the Mastercam HSM Performance Pack).

The part in the file is a solid; if you do not have a solids license on your seat of Mastercam, you can still make this tutorial, as this tutorial is only concerned with toolpaths, but you cannot edit the part.

Tools used

The tools used in this tutorial are:

T1: 20 mm dia 1 mm CR bull mill, S16000, F9500
T2: 6 mm dia 1 mm CR bull mill, S30000, F6000
T3: 10 mm dia spherical mill, S24000, F7200
T4: 6 mm dia spherical mill, S30000, F6000

Plunge, normal and retract feed rates are all defined at the same value.

Roughing

Adaptive clearing

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.

To create the adaptive clearing, use Alt-C and select the HSM.DLL C-Hook, then select Adaptive for the machining strategy. Click on the
solid to select it, click Done, and Done to end the boundary selection (we do not need a boundary for the adaptive clearing strategy).

On the first tab of the dialog, you select or create the tool to be used for the adaptive clearing. The important data for the tool are T1: 20 mm dia 1 mm CR bull mill, S16000, F9500. You can see the tool tab below.
After setting up for the correct tool, go to the Surface tab, shown below.

The surface tab should be filled in as shown. There are not many things that can be set for an adaptive clearing toolpath, we will set the rest of the settings when we get to the next operation.
The adaptive clearing tab, shown below, contains the settings that have the greatest effect on the toolpath.

The *Tolerance* is set to 0.1 mm. A roughing operation does not need to be very accurate, and we also set a *Stock to leave* of 0.3 mm. *Minimum cutting radius* is set to 1 mm, as we do not want the tool to move through channels that are too narrow. *Maximum load* should normally be set to the side cutting load taken from the cutter manufacturer’s data sheet, with *Optimal load* set to a little less, but we are using higher values here to better illustrate how adaptive clearing works. *Maximum stepdown* is 20 mm, as an adaptive clearing uses side milling cutting data, with *Fine stepdown* set to 2 mm.

If the *Cutting depth Mode* was set to *No depth limit*, then Mastercam HSM Performance Pack would assume the dimensions of the stock was equal to the dimensions of the part. Setting the *Mode* to *Depth relative to*...
tool tip means we can set the Minimum depth and Maximum depth to our own values.

Ramping can be left with the default values, as that is only used in cavities, and there are no cavities on this part.

On the next tab, Rest material, Rest material source is set to Disabled, since everything should be machined.

The rest of the tabs should remain set to the default values. You can leaf through them and click the ‘Reset’ button on each, to ensure that they are set to the default values.

Clicking the Generate button finishes the definition of the toolpath and calculates it. Our adaptive clearing toolpath should look like this:
Pocket rest milling

There are some areas where our 20 mm mill did not fit; we will remove more material there with a pocket rest milling operation.

The Mastercam HSM Performance Pack is started and Pocket, the solid and the rectangular boundary are selected as before. The tool needed for this operation is T2: 6 mm dia 1 mm CR bull mill, S30000, F6000; nothing else needs to be changed on the tool tab.

The surface tab has to be set up correctly for the remaining operation. Fill it in as below:

![Pocket strategy - CIMCO Integration (cimco-software.com)](image)

There are two details of particular note here; these are the clearance plane settings and the retraction policy.
For the clearance plane, we have set 10 mm, *Absolute clearance* not checked. This means that the clearance plane is 10 mm above the highest part of the selected geometry, and that gives a correct clearance plane for this part. In 3D machining, the clearance plane is often set to an absolute value (i.e. *Absolute clearance* should be checked), known to be above the part, while in 5 axis positioning (also called 3+2 axis machining), it is most often set to a relative clearance plane as here, as that is easier than calculating an absolute clearance plane in the rotated coordinate system.

For this tutorial, the retraction policy is set to *Shortest path* (and on the toolpath tab, *Rapid filtering* is set to *Preserve rapid motion*). This gives the shortest possible rapid movement paths, but those settings can only be used on 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 those settings to suit your machine and control.
The pocket tab should be filled in as can be seen below.

The *Maximum stepdown* is set to 1 mm, and stock to leave is 0.3 mm as in the prior operation. The *Corner deviation* of 0.4 mm allows the tool to move in smooth arcs at the corner in the toolpath, instead of trying to make sharp corners.

By not activating *Set stepover* under *Stepover*, the stepover will be set optimally by the Mastercam HSM Performance Pack, based on the tool shape.

In the *Cutting depths* we set a *Minimum depth* of 40 mm to avoid unnecessary air cuts above the part, and a *Maximum depth* of 0.3 mm (our base surface plus stock to leave).

We do not need the *Shallow* option, as there are no large flat areas to be machined by this operation.
We also need to fill in the next tab, Rest material, as we only want to remove the material the previous operation did not remove. The Rest material tab is shown here:

![Rest material tab screenshot](image)

We set the Rest material source to Rest material operation, and select the previous operation. The Stock model resolution is the size of the steps along the surfaces used to calculate the stock model. The Adjustment setting is used to ignore rest material areas less than the specified size, so the tool does not needlessly machine areas that can be handled by a later operation. We are choosing Rest material operation as the rest material source here, as that gives the most correct result. For an in depth description of the differences between Rest material tool and Rest material operation, see the manual for the Mastercam HSM Performance Pack.
Click Generate to calculate the toolpath. It should look like this:

![Toolpath Diagram]

**Semi-finish**

We now have a minimum of 0.2 mm material left on the part (stock to leave minus tolerance), and more in some areas. We need to do several semi-finish operations to reduce that.

Rename the first operation group in the operations manager to “Rough”, and create a new operation group named “Semi-finish”.

**Contour machining the steep areas**

We will first make a contour operation to handle the steep areas of the part. Start the Mastercam HSM Performance Pack, select *Contour* as machining strategy, and select the solid and the rectangular boundary.

The tool to be used for this is T3: 10 mm dia spherical mill, S24000, F7200.
On the contour tab we will need these settings:

We set a *Tolerance* of 0.02 mm, and a *Stock to leave* of 0.1 mm. *Maximum stepdown* and *Corner deviation* are both set to 1 mm.

*Order by depth* is selected to ensure that all cuts are done from highest Z to lowest Z.

Since we *only* want to machine the steep areas with this operation, we limit the *Slope range* to areas between 60 degrees from horizontal (that is 30 degrees from vertical) to 90 degrees from horizontal (that is vertical). *Contact only* is selected to filter out contour segments where the tool is not in contact with the surfaces. If we did not select *Contact only*, the Mastercam HSM Performance Pack would also create contour passes on the outer boundary, which would be cutting air on this part.
On the rest material tab, Rest material source should be set to Disabled. Click Generate to calculate the toolpath. It should look like this:

![Scallop machining the shallow areas](image)

Scallop machining the shallow areas

Next we will make a scallop operation on the boss area.

Start the Mastercam HSM Performance Pack, select Scallop as machining strategy, and select the solid and the inner boundary.

The tool is the same as in the contour operation we just made, T3: 10 mm dia spherical mill, S24000, F7200.
The scallop tab looks like this:

![Scallop Tab Screenshot](image)

The **Tolerance**, **Stepover** and **Stock to leave** are the same as in the contour operation, but we are setting the **Slope range** differently, machining from 0 degrees (horizontal) to 60 degrees from horizontal. We also selected **Machine using boundaries**, and set a **Boundary overlap** of 2 mm. This ensures that there is no excess material in the transition zone between the two operations, and that the tool starts in areas that are already machined, as we are linking the toolpath from the outside to inside (since we have not selected **Link from inside to outside**). Extending the boundary like this will also smoothen the boundary of the machining area, which smoothenes the toolpath.

We are using depth limits here, as we do not want the tool to go below Z3 (which is 3 mm above the base surface). We will be machining the base surfaces and the fillet from the base surfaces to the part later. The
Minimum depth is set to 40 mm, which is a depth we know is now above our part.

On the rest material tab, Rest material source should be set to Disabled.

Click Generate to calculate the toolpath. It should look like this:

![Toolpath Image]

**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.

Create a new operation group in the operations manager named “Finish”.

---

CIMCO Integration
Scallop machining the entire boss

We will first reduce the leftover material to 0.05 mm on the entire boss with a scallop operation. Select the machining strategy Scallop, the solid, and the inner boundary.

The tool to be used for this operation is T4: 6 mm dia spherical mill, S30000, F6000.

The scallop tab should be filled in as follows for this operation:

*Stock to leave* is set to 0.05 mm now, of course. We are using *Link from inside to outside* because that will cause the tool to start at the center (which is also the top of the part) and work towards the boundary (which is the bottom of the part, where the fillet meets the base surfaces). *Slope range* is set to 0 degrees (horizontal) to 90 degrees (vertical), to machine all areas, and we are not using any depth limits either.
On the rest material tab, *Rest material source* should be set to *Disabled* as in the prior operation.

Click *Generate* to calculate the toolpath. It should look like this:

![Horizontal clearing](image)

**Horizontal clearing**

The base surfaces are completely horizontal, so they should be machines with a flat bottomed mill. They also go into some corners, so it needs to be a fairly small mill, and we do not want a sharp mark at the transition to the fillet, so a bull mill will be best. We will use T2: 6 mm dia 1 mm CR bull mill, S30000, F6000, which we also used in the pocket rest milling.

Select the machining strategy *Horizontal*, the solid, and the outer rectangular boundary.
No depth limits are needed, and there is no reason to set a limiting stepover, the Mastercam HSM Performance Pack calculates the largest stepover that will remove all material that has to be removed. While this operation will machine both the base surfaces and the flat tops of the flanges, there is no reason to specify a specific ordering of cuts.
Click *Generate* to calculate the toolpath. It should look like this:

**Pencil milling the fillets**

The tool we have for finishing the part is a 6 mm ball mill, and those fillets are radius 4, so we will need some close passes to make them look good. We will use the pencil milling strategy, with the overthickness and limited pencil parallel passes functions to do that.

Select the machining strategy *Pencil*, the solid, and the outer rectangular boundary.

We will be using T4: 6 mm dia spherical mill, S30000, F6000 for this operation.
Fill in the pencil milling tab like this:

![Pencil milling strategy - CIMCO Integration (cimco-software.com)](image)

The **Stepover** is set to 0.3 mm to ensure a good finish in the fillets. The **Bitangent angle** of 20 degrees is the default setting, and we do not need to change that for this part.

In this example we are setting the **Overthickness** to 1.2 mm. The overthickness setting is needed by the Mastercam HSM Performance Pack to generate pencil passes where they would not otherwise exist because the tool radius (3 mm) is smaller than the fillet radius (4 mm). With the overthickness set to 1.2 mm, the Mastercam HSM Performance Pack will generate the pencil passes for a larger tool radius, and uses them to guide the smaller tool we are using into the fillets.

We are linking from the outside to inside to minimize the amount of material to be removed by the final pass along the centerline of the fillets.
We are limiting our toolpath to 5 pencil parallel passes by selecting *Limit* and setting the limit to 5, as we only want the passes that close in the fillets.

Click *Generate* to calculate the toolpath. It should look like this:

**Pencil parallel collapse**

For the last finishing operation, we will make a pencil parallel collapse toolpath.

Select the machining strategy *Pencil*, the solid, and the inner boundary (we do not need to machine the base surfaces, they are already finished).
We are still using T4, the pencil tab should be changed to look like this:

The changes from the prior operation are:

The *Stepover* is increased to 0.7 mm.

The *Limit* is deselected to create pencil parallel collapse toolpaths over the entire part.

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.
Click *Generate* to calculate the toolpath. It should look like this:
That finishes this tutorial; you should now have these operations in the operations manager:
You can try verifying the toolpaths in the verification software of your choice. Using Powercut from CIMCO Integration, with tool colors switched off, it looks like this:
Tutorial II

Getting Started

In this tutorial, it is assumed that the user is conversant with the use of Mastercam, so ordinary actions in Mastercam are not explained in detail.

The file used for this tutorial is the MC9 file designated tutorial 2, which was included with this tutorial, or can be downloaded from the Mastercam HSM Performance Pack website (there is a link on the About tab in the Mastercam HSM Performance Pack).

The part in the file is a solid; If you do not have a solids license on your seat of Mastercam, you can still make this tutorial, as this tutorial is only concerned with toolpaths, but you cannot edit the part.

Tools used

The tools used in this tutorial are:

T1: 10 mm dia 1 mm CR bull mill, S9550, Plunge feed F1900, Cutting and retract feed F3820
T2: 3 mm dia 0.5 mm CR bull mill, S30000, Plunge feed F1800, Cutting and retract feed F3600
T3: 2 mm dia spherical mill, S30000, Plunge feed F900, Cutting and retract feed F1800
T4: 3 mm dia spherical mill, S30000, Plunge feed F1800, Cutting and retract feed F3600
T5: 4 mm dia 0.5 mm CR bull mill, S23800, Plunge feed F1500, Cutting and retract feed F2860
T6: 1 mm dia spherical mill, S30000, Plunge feed F400, Cutting and retract feed F900
Roughing

Pocket roughing

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

To create the pocket toolpath, use Alt-C and select the HSM.DLL C-Hook, then select Pocket for the machining strategy. Click on the solid to select it, click Done, then you select the two outer boundaries (check that the chaining method is “Full”), and Done to end the boundary selection.

On the first tab of the dialog, you select or create the tool to be used for the pocket operation. The important data for the tool are T1: 10 mm dia 1 mm CR bull mill, S9550, Plunge feed F1900, Cutting and retract feed F3820. You can see the tool tab on next page.
After setting up for the correct tool, go to the Surface tab. The default values will do nicely, so you click the Reset button to set it to the default values.
The Pocket tab should then be filled in as shown below.

The *Tolerance* is set to 0.1 mm. A roughing operation does not need to be very accurate, and we also set a *Stock to leave* of 0.3 mm. *Maximum stepdown* is 0.7 mm, as we will be engaging material with the full diameter of the tool in some areas, and we are using a high feed rate. The *Corner deviation* of 0.5 mm will smoothen the tool movements a little at the corners in the toolpath, while still not leaving too much material for the later operations.

By not activating *Set stepover* under *Stepover*, the stepover will be set optimally by the Mastercam HSM Performance Pack, based on the tool shape.

We do not need to limit the cutting depths, as the top of stock is equal to the highest point on the part.
We are activating the *Shallow* option to avoid too large steps where the walls are becoming shallow. We set a *Stepover* of 0.7 mm (same as our *Maximum stepdown*) to make the steps similar, but we also set a *Minimum stepdown* of 0.2 mm to avoid having too many Z layers in this roughing operation.

On the next tab, Rest material, *Rest material source* is set to *Disabled*, since everything should be machined.

The rest of the tabs should remain set to the default values. You can leaf through them and click the *Reset* button on each to ensure that they are set to the default values.

Clicking the *Generate* button finishes the definition of the toolpath and calculates it. Our pocket toolpath should look like this:
Pocket rest milling

There are some areas where our 10 mm mill did not fit; we will remove more material there with a pocket rest milling operation.

Start the Mastercam HSM Performance Pack, select Pocket, the solid and the two outer boundaries as before. The tool needed for this operation is T2: 3 mm dia 0.5 mm CR bull mill, S30000, Plunge feed F1800, Cutting and retract feed F3600; nothing else needs to be changed on the tool or surface tabs.

The pocket tab needs a few changes for this operation, as can be seen below.

![Pocket strategy - CIMCO Integration (cimco-software.com)](image)

The Maximum stepdown is reduced to 0.2 mm because of the smaller tool, and the Corner deviation is reduced to 0.15 mm.
In the Cutting depths we set a Minimum depth of 0 mm to avoid unnecessary air cuts above the part, and a Maximum depth of –40 mm (which is well below the lowest depth of our cavities).

We are not using the Shallow option, as there are no areas with rest material where that is relevant.

We also need to fill in the next tab, Rest material, as we only want to remove the material the previous operation did not remove. The Rest material tab is shown here:

We set the Rest material source to Rest material operation, and select the previous operation. The Stock model resolution is the size of the steps along the surfaces used to calculate the stock model. The Adjustment setting is used to ignore rest material areas less than the specified size, so the tool does not needlessly machine areas that can
be handled by a later operation. We are choosing Rest material operation as the rest material source here, as that gives the most correct result. For an in depth description of the differences between Rest material tool and Rest material operation, see the manual for the Mastercam HSM Performance Pack.

Click Generate to calculate the toolpath. It should look like this:

**Semi-finish**

We now have a minimum of 0.2 mm material left on the part (stock to leave minus tolerance), and more in some areas. We need to do several semi-finish operations to reduce that.
Rename the first operation group in the operations manager to “Rough”, and create a new operation group named “Semi-finish”.

Contour machining the steep areas

We will first make a contour operation to handle the steep areas of the part. Start the Mastercam HSM Performance Pack, select Contour as machining strategy, and select the solid and the two outer boundaries.

The tool to be used for this is T4: 3 mm dia spherical mill, S30000, Plunge feed F1800, Cutting and retract feed F3600

On the contour tab we will need these settings:

![Contour strategy settings](image)

We set a Tolerance of 0.02 mm, and a Stock to leave of 0.1 mm. Maximum stepdown is set to 0.3 mm and Corner deviation to 0.15 mm.
Order by depth is not selected, as that would cause far too many retracts between the different steep areas.

Since we only want to machine the steep areas with this operation, we limit the Slope range to areas between 50 degrees from horizontal (that is 40 degrees from vertical) to 90 degrees from horizontal (that is vertical). We are changing the angle limiting our semi-finish contour from the 60 degrees used in tutorial I, because 50 degrees is a better choice for this part; The choice of limiting angle depends both on the geometry of the part and on the type of toolpath used to machine the shallow areas. Contact only is selected to filter out contour segments where the tool is not in contact with the surfaces.

On the rest material tab, Rest material source should be set to Disabled.
Click *Generate* to calculate the toolpath. It should look like this:

*Parallel machining the shallow areas*

For the shallow areas, we will use the parallel machining strategy. Start the Mastercam HSM Performance Pack, select *Parallel* as machining strategy, and select the solid and the two outer boundaries.

The tool to be used for this is T4, as in the prior operation.
On the parallel tab we will need these settings:

The *Tolerance* and *Stock to leave* are the same as in the contour operation, and the *Stepover* is the same as the *Maximum stepdown* in our contour operation. We are setting the *Direction* to 0 degrees (i.e. parallel to the X axis), as that will give the best result for this part. The *Slope range* is set to 0 degrees to 50 degrees, to cover the areas not machined by the contour operation. We do not need to overlap the two areas, as most of the transition areas are on inner radii, and the rest are at nearly vertical surfaces.

In the *Cutting depths* we set a *Minimum depth* of –3 mm to avoid unnecessary toolpaths at the top edges of the cavities, and a *Maximum depth* of –40 mm (which is well below the lowest depth of our cavities).
Click *Generate* to calculate the toolpath. It should look like this:

---

**Pencil parallel collapse machining**

For our third semi-finish operation we will use a pencil parallel collapse. This is a series of constant steps along the surfaces starting from a set of pencil passes running along the internal corners of the part. In certain cases this is the best way to get a smooth finish.

Start the Mastercam HSM Performance Pack, select *Pencil* as machining strategy, and select the solid. We will use additional boundaries for this toolpath: after selecting the two outer boundaries, select the three boundaries around the internal projections, but not the fourth internal boundary which is around an internal pocket.
The tool to be used for this is T4, as in the prior operation.

On the pencil tab we will need these settings:

![Pencil settings](image)

The *Stock to leave* is reduced to 0.05 mm. The *Stepover* is still 0.3 mm as in the earlier operations with this tool, but here it is measured along the surface instead of in the Z direction as in the contour toolpath or in the XY direction as in the parallel toolpath. We are not using any *Overthickness*, as there are no fillets larger than the tool radius that we need to track, and the default *Bitangency angle* of 20 degrees works fine.

Since we want to machine the entire part within the boundaries, we do not set a limit on the number of pencil parallel passes. Also, we want the passes to be linked from the outside in.
In the *Cutting depths* we set a *Minimum depth* of $-1.5$ mm to keep the tool from "rolling" over the top edges of the cavities, and a *Maximum depth* of $-40$ mm.

Click *Generate* to calculate the toolpath. It should look like this:

![Toolpath Diagram](image)

**Scallop machining with rest material limit**

We have some areas at internal corners and fillets, where there is more than the desired $0.05$ mm stock left, which we need to remove with a smaller tool.

Start the Mastercam HSM Performance Pack, select *Scallop* as machining strategy, and select the solid and the two outer boundaries.
The tool we will use for this is T3: 2 mm dia spherical mill, S30000, Plunge feed F900, Cutting and retract feed F1800.

On the scallop tab we will need these settings:

![Scaletop Settings](image)

We reduce the Stepover to 0.1 mm because of the smaller tool, and because we will be machining at internal corners. There is no need for depth limits for this operation.

We then go to the rest material tab, shown below.
We set *Rest material source* to *Rest material tool*. *Diameter*, *Stock to leave* and *Corner radius* are set to match the prior operation, of course, and we are setting an *Overlap* of 0.2 mm to ensure a smooth transition at the edges of the areas we are machining with this operation. The *Stock model resolution* is the size of the steps along the surfaces used to calculate the stock model. We are choosing *Rest material tool* as the rest material source this time, because that gives an accurate definition of the areas we need to machine, and using *Rest material operation* with the necessary previous operations selected would take longer to calculate.
Click *Generate* to calculate the toolpath. It should look like this:

![Toolpath Diagram]

**Finish**

We now have 0.05 mm left on nearly all of the part.

Create a new operation group in the operations manager named "Finish".

**Scallop machining the entire cavities**

First we will use a scallop toolpath to finish the majority of the part, leaving only the horizontal areas on top of the three projections, and the
internal corners and fillets that we cannot reach with our first finish toolpath.

Start the Mastercam HSM Performance Pack, select Scallop as machining strategy, and select the solid. We will need the same boundaries as for the pencil parallel collapse toolpath, so after selecting the two outer boundaries, you should select the three boundaries around the internal projections, but the boundary around the internal cavity in the lower right area of the part should not be selected.

The tool to be used for this is T4.

On the scallop tab we will need these settings:

Stock to leave is (of course) 0, and we are using a Stepover of 0.15 mm to get a good surface finish on the part. We are machining the entire
part, so the only limiting setting is the *Minimum depth* of –1.5 mm to prevent the tool from “rolling” over the top edges of the cavities.

On the rest material tab, *Rest material source* should be set to *Disabled*.

Click *Generate* to calculate the toolpath. It should look like this:

**Horizontal clearing**

The three projections in the cavities, the area around the two projections in the right cavity and the bottom of the small internal cavity are all horizontal areas. We have already machined the bottom of the small internal cavity to an acceptable surface finish, but the other three horizontal areas should be machined with a flat bottomed mill. We will
use this tool: T5: 4 mm dia 0.5 mm CR bull mill, S23800, Plunge feed F1500, Cutting and retract feed F2860.

Start the Mastercam HSM Performance Pack, select *Horizontal* as machining strategy, and select the solid and the two outer boundaries, plus the boundary around the internal cavity to exclude that area.

Remember to select or create the correct tool, then fill in the horizontal tab like this:

![Horizontal tab](image)

We are using a *Corner deviation* of only 0.1 mm, as we need to get out in the corners of the machined areas. The *Minimum depth* of –1 mm prevents the operation from creating toolpaths at the top edges of the cavities.

We are selecting *Order by depth* in order to machine the tops of the two projections to the right before the area between them.
Click *Generate* to calculate the toolpath. It should look like this:

Scallop machining with rest material limit (final finish of internal corners and fillets)

Now we only need to remove the last remaining material from the internal corners and small internal fillets. For that, we will use this tool:

T6: 1 mm dia spherical mill, S30000, Plunge feed F400, Cutting and retract feed F900.

Start the Mastercam HSM Performance Pack, select *Scallop* as machining strategy, and select the solid and the two outer boundaries.
On the scallop tab, we need these settings:

![Scallop Tab Settings](image)

The Stepover is set to 0.05 mm for this small tool.
On the rest material tab shown below, we set the parameters to match our finish scallop operation with T4, with an **Overlap** of 0.2 mm and a **Resolution** of 0.1 mm.
Click *Generate* to calculate the toolpath. It should look like this:
That finishes this tutorial; you should now have these operations in the operations manager:

![Operations Manager screenshot](image)

<table>
<thead>
<tr>
<th>Rough</th>
<th>Semi-finish</th>
<th>Finish</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Rough Strategy</td>
<td>3. Conour strategy</td>
<td>7. Scallop strategy</td>
</tr>
<tr>
<td>[3x1.5, 0.0]</td>
<td>[3x1.5, 0.1]</td>
<td>[3x1.5, 0.0]</td>
</tr>
<tr>
<td>[3x0.5, 0.0]</td>
<td>strategy [3x1.5, 0.1]</td>
<td>[4x0.5, 0.0]</td>
</tr>
<tr>
<td>5. Pencil milling strategy [Collapse]</td>
<td>6. Scallop</td>
<td>9. Scallop strategy [3x1.5, 0.0]: [1x0.5, 0.0]</td>
</tr>
<tr>
<td>[3x1.5, 0.05]</td>
<td>[3x1.5, 0.05]</td>
<td>[2x1.0, 0.05]</td>
</tr>
</tbody>
</table>
You can try verifying the toolpaths in the verification software of your choice. Using Powercut from CIMCO Integration, with tool colors switched off, it looks like this: