

TABLE OF CONTENTS

Introduction	1
Machining Methods	2
Area Oriented Options	2
Operation Oriented Machining Operations	3
Multi-Axis Machining Workbench	7
Specifications Tree	7
Multi-Axis Machining Workbench	11
Review	25
Roughing Operations	41
Sweep Roughing	41
Geometry Tab	44
Strategy Parameters	56
Macros	61
Roughing	71
Geometry Tab	73
Strategy Parameters	83
Macros Tab	97
Roughing Rework Operation	101
Sweeping	107
Geometry Tab	109
Strategy Parameters	113
Surface Quality	131
ZLevel Machining	137
Strategy Parameters	140
Macros	146
Spiral Milling	149
Strategy Parameters	151
Pencil Operations	159
Strategy Parameters	161
Contour Driven Milling	165
Strategy Parameters	167
Advanced Finishing	189
Isoparametric Machining	194
Geometry Definition	195
Strategy Parameters	197

Profile Contouring	205
Between Two Curves	206
Between Curve and Surfaces	207
Machining Surfaces and Solids	217
Multi-Axis Machining	233
Rotary Axis Machining	233
Five Axis Machining	265
Multi-Axis Sweeping	265
Strategy Parameters	268
Tool Axis Parameters	273
Macros	287
Multi-Axis Contour Driven	291
Strategy Parameters	293
Multi-Axis Isoparametric Machining	303
Multi-Axis Flank Contouring	319
Strategy Parameters	326
Multi-Axis Curve Machining	363
Multi-Axis Helix Machining	375
Multi-Axis Tube Machining	386
Practice Problems	397
SMAC - Problem1	397
SMAC - Problem2	398
SMAC - Problem3	399
SMAC - Problem4	400
SMAC - Problem5	401
SMAC - Problem6	402
SMAC - Problem7	403
SMAC - Problem8	404
SMAC - Problem9	405
SMAC - Problem10	406
Appendix A	407
App Preferences - Simulation - Machining - General	407
App Preferences - Simulation - Machining - Resources	410
App Preferences - Simulation - Machining - Operation	413
App Preferences - Simulation - Machining - Output	414
App Preferences - Simulation - Machining - Program	415
App Preferences - Simulation - Machining - Material Removal	416
App Preferences - Simulation - Machining - Simulation	418

Introduction

DELMIA Version 6 Surface Machining

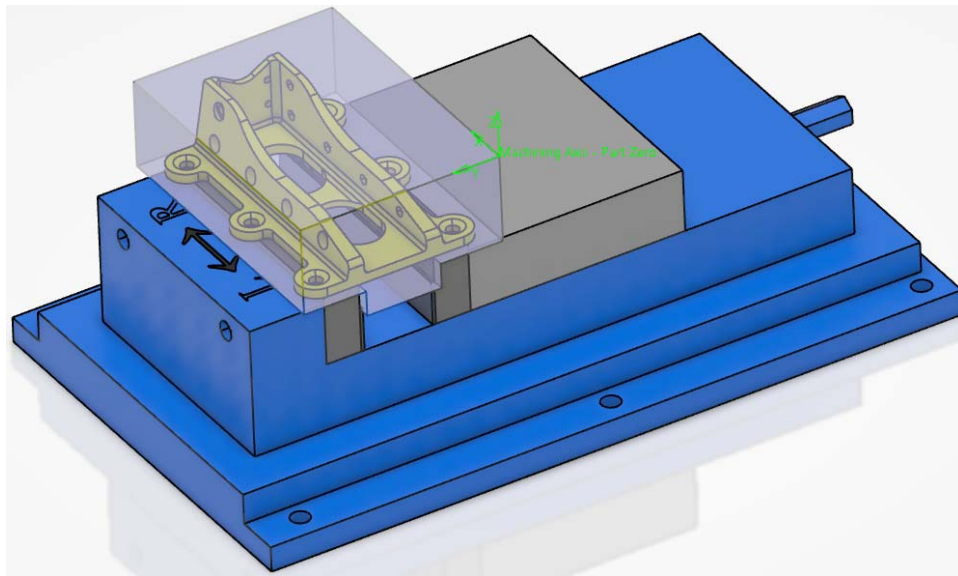
Upon completion of this course, you should have a full understanding of the following topics.


- Use the Multi-Axis Machining workbench to machine complex contours and part surfaces
- Machine through the use of various roughing, sweeping, and reworking operations
- Integrate Prismatic Machining and Multi-Axis Machining workbenches together to get a full range of manufacturing tools
- Machine through the use of multiple part operations and stock material at various stages
- Machine surfaces as well as part bodies
- Machine complex parts utilizing multi-axis machining operation

Roughing

Sweep roughing works well when a series of surfaces need to be machined. Unfortunately, not all parts are simple surfaces. Many times pockets, channels, or various other features are present, which the sweep roughing operation does not work well with. This is where the Roughing Operation comes into play.

Open the SMAC030 - Roughing process document. This machining process already has the Part Operation geometry and parameters defined.




Select the **Roughing** icon and select *Manufacturing Program.1*.  The *Tool Assembly Search* window appears. A tool assembly must be defined in order to create the operation.

The 'Tool Assembly Search' dialog box is shown. It has a title bar with a gear icon and a close button. Inside, there are two radio buttons: 'Tool Assembly Criteria' (selected) and 'Tool Criteria'. Below them is a search bar containing the word 'Milling'. Under the search bar are three icons representing different tool types. Below the icons is a section titled 'Parameters' with three rows of input fields: 'Nominal Diameter' (Min: 0in, Max: 0in), 'Cutting Length' (Min: 0in, Max: 0in), and 'Corner Radius' (Min: 0in, Max: 0in). Below the parameters is a section titled 'Feeds and Speeds Configuration' with three dropdown menus, all set to 'Undefined'. Below that is a section titled 'Attachment' with a 'Type' dropdown set to 'Undefined' and a 'Characteristic' dropdown. At the bottom of the dialog are three icons: a mouse cursor, a magnifying glass over a document, and a stack of documents.

Select the *Tool Assembly Criteria* option and select the **End Mill** icon.



Select the **Select in Session** icon.  The view should automatically switch to the *Resource Configuration View* and filter the tools by end mills.

Select the *SMAC030 - T1 EM Tool Assembly*. The *Tool Axis* window will appear.


Select **OK**. The part definition appears.

Select the **Esc** key twice. The *Roughing* window appears.








The basic geometry definition of the roughing operation is the same as the sweep roughing, with a few extra options. Take a quick look at the sensitive area of the geometry definition tab.


Note: Many of the options found here are the same as what you investigated in the Sweep Roughing operation. If you find an option that you would like more information on, please refer to the Sweep Roughing section.


Geometry Tab




New Feature 






▼ Mandatory






 Part  0.039in     




 Tool Axis






▼ Optional 




☐ Rough stock   






☐ Check 0.039in     






☐ Limiting Contour 0in     






☐ Start point(s)   



☐ Imposed 0in     

☐ Zone Order   


☐ Top 0in     


☐ Bottom 0in     



☐ Safety plane 0in     



Minimum thickness to machine 0.012in  

▼ Limit definition

Side to machine Inside 

Stop position On 

Ignores holes ☐ Diameter 0.394in  

Compute with tool holder ☐ Offset on assembly 0in  


No Offset Group

Optional

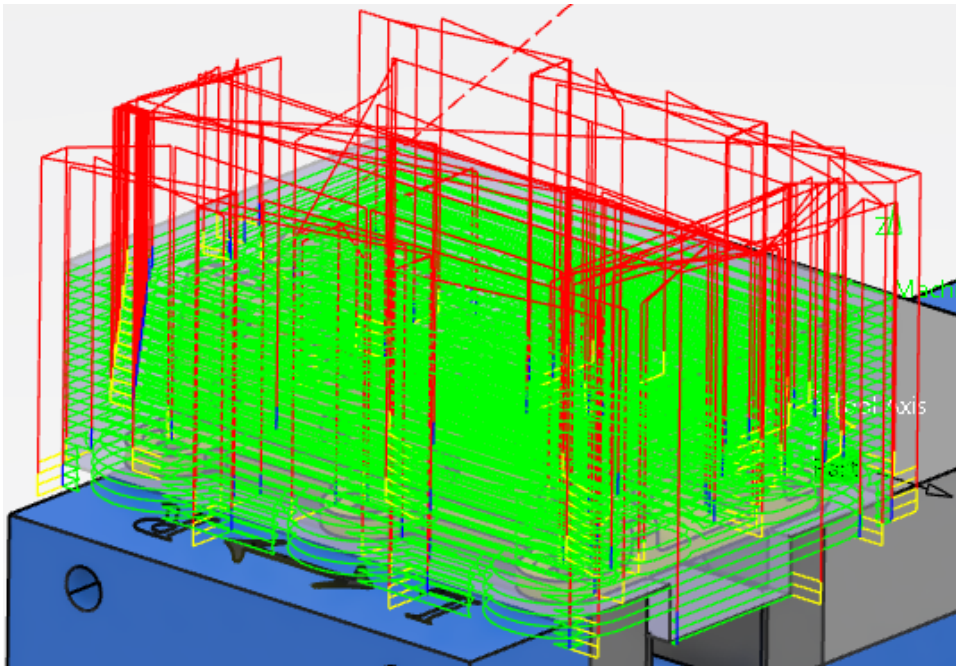
- Rough stock* Defines the rough stock around the part. If a rough stock is defined in the part operation setup, then it is not necessary here.
- Start point(s)* Defines a point or multiple points that the roughing operation can start from.
- Imposed* Defines an imposed plane position. This will require the tool to make a pass at that Zlevel. This option is useful when machining with flat regions.

<i>Zone order</i>	Defines what areas are to be machined in what order. With the contextual menu, you can also require that only ordered zones get machined.
<i>Minimum thickness...</i>	Defines the minimum amount of material that will be removed when performing a reworking operation
<i>Ignore holes on...</i>	Ignores holes in the rough stock
<i>Diameter</i>	Defines the size of holes to ignore
<i>Compute with tool...</i>	Computes the tool path to avoid collisions with the tool holder
<i>Offset on assembly</i>	Defines the offset on the tool holder assembly

Now to actually define geometry to machine the part.

Select the **Define Part on PO** icon for the **Part** definition.  This will automatically select the part that was defined in the part operation.

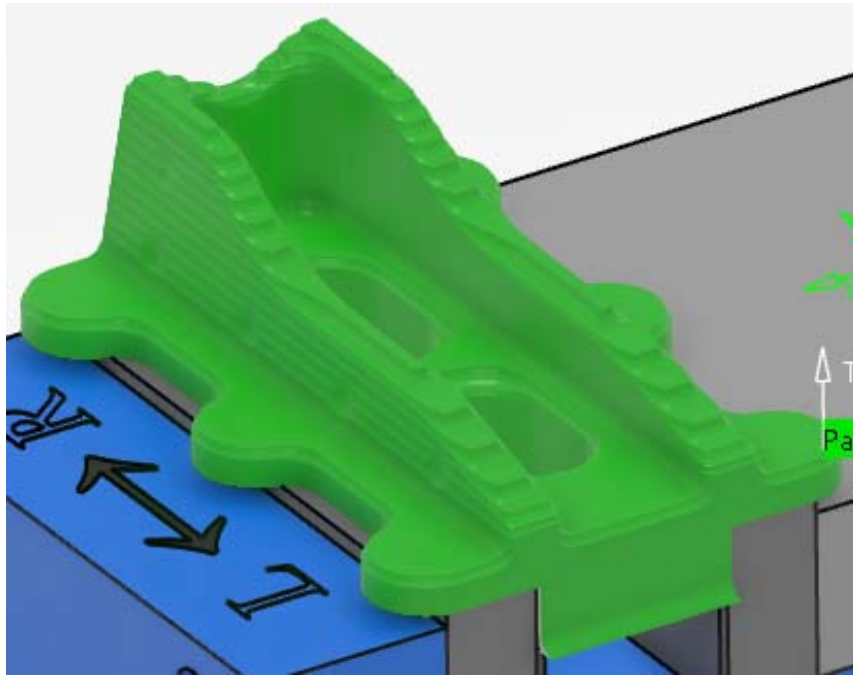
Simulate the operation. Notice that by simply defining the part definition, you have given the roughing operation enough information to machine the part. As you can see, the roughing operation is designed to take a piece of stock material and convert the entire solid into chips until it reaches the design part. The remaining parameters help control how it machines the part.



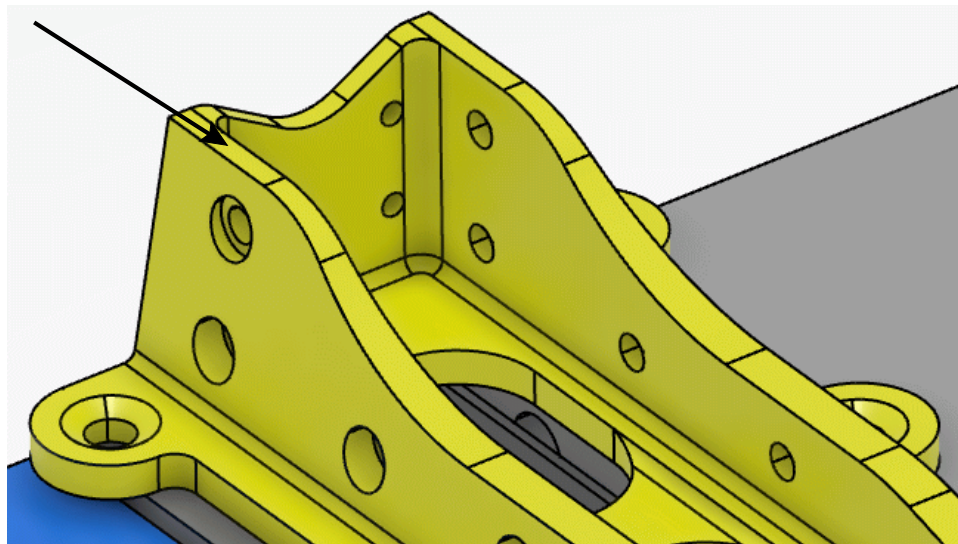
Change offset value for the *Part* definition to be 0.125in. This will tell the roughing operation to leave an 1/8th inch on the part surface. The surfaces will be finished with other operations.

Since the rough stock is defined in the Part Operation, it is unnecessary to define it here. If you did not define the stock material, or are using a separate CGR file as the stock, then you would want to select the *Rough Stock* sensitive area, and select the appropriate stock material.

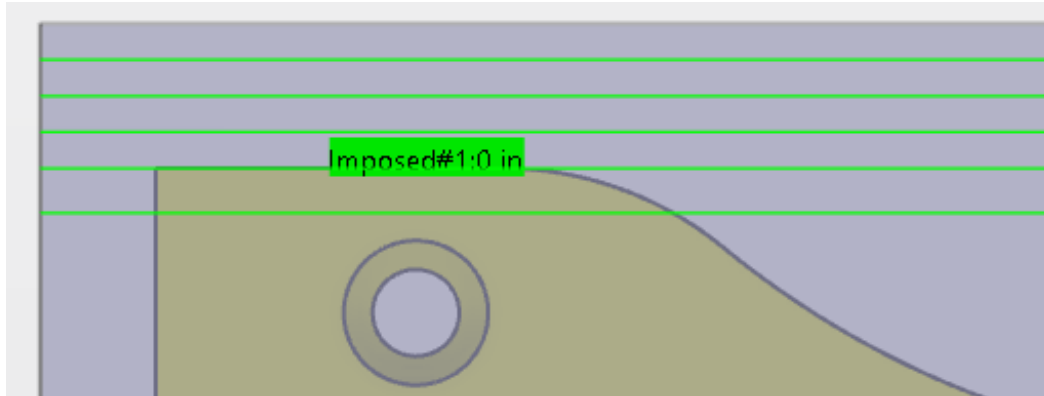
Simulate the material removal. Notice that the entire stock material within reach of the tool is removed. Also the side of the part has divots where the holes exist. This is common to the roughing operation. If the holes in the flanges were large enough or the tool small enough, they would be roughed out as well.



Select the *Imposed* definition and the planar face as shown. The imposed plane allows you to define flat planar areas that the tool will make a pass on. This is also helpful to define points that must be cut by the tool.



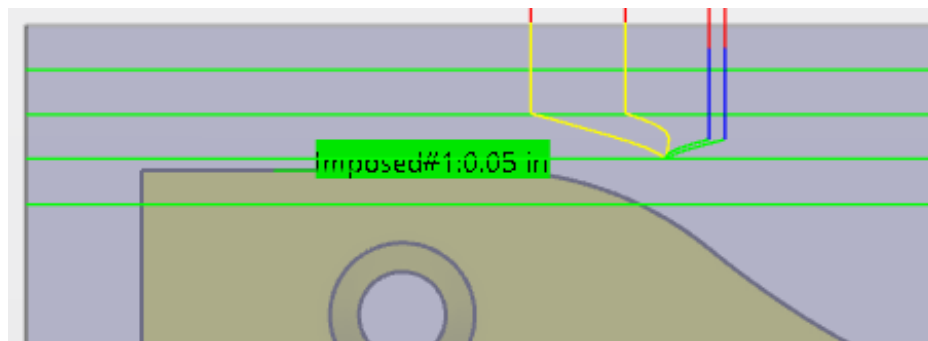
Simulate the operation. There is now a tool path at the same level as the imposed plane. If you look close, you should notice that the tool does not actually cut the top of the part.



Even though you imposed the plane, the offset on part is always obeyed. This is why the tool does not cut across the top of the part.

With the third mouse button, select on the *Imposed* plane in the display. This will bring up the contextual menu for the imposed plane.

Select *Offset* and set it to 0.050in. This will make the tool pass 0.050in above the imposed plane.



Even though you have added a 0.050in offset, the tool still does not cut at the imposed plane location. Until you make the imposed plane offset equal the *Offset on part* the tool will not cut at the imposed plane.

Change the offset on the *Imposed* plane to 0.125in and replay. This will set the imposed plane offset equal to offset on the part. Now the tool cuts at the imposed plane location. No matter what you define the *Offset on part* will never be violated.

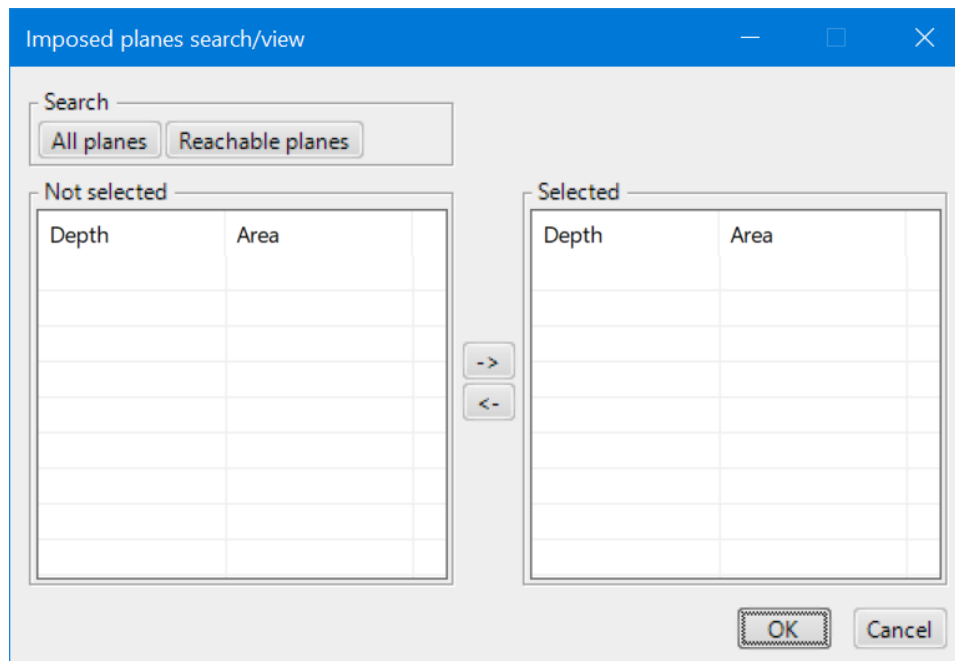
This method of defining imposed planes works, but what happens when a lot of planes need to be selected? There is another option to define imposed planes.

Select the **Remove icon for the *Imposed* option.**



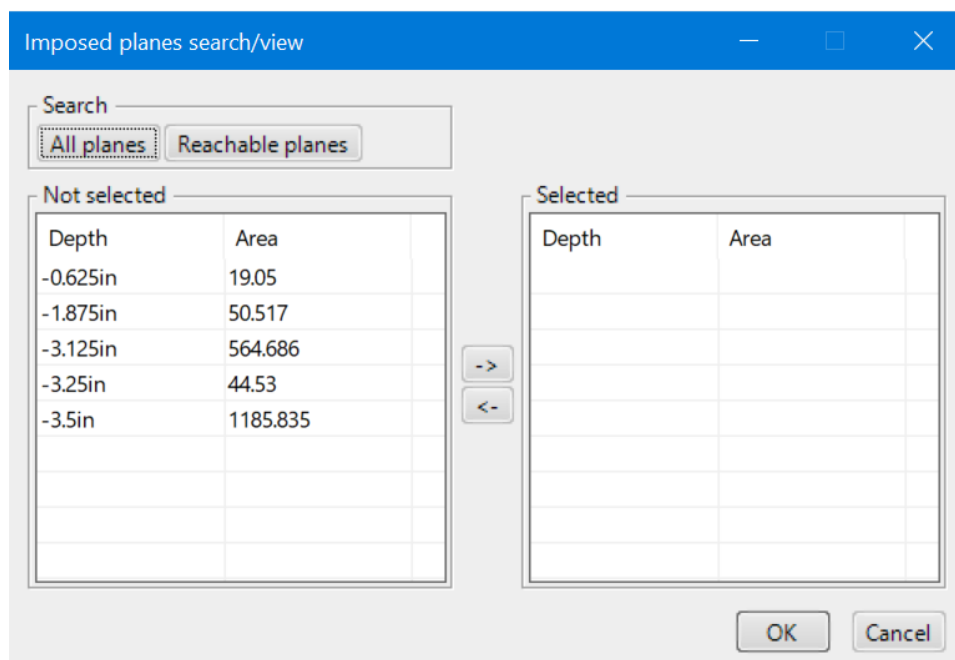
This will remove all the imposed planes.

Select the **More** icon for the **Imposed** option and select **Search/View...** This will display the *Imposed planes search/view* window.

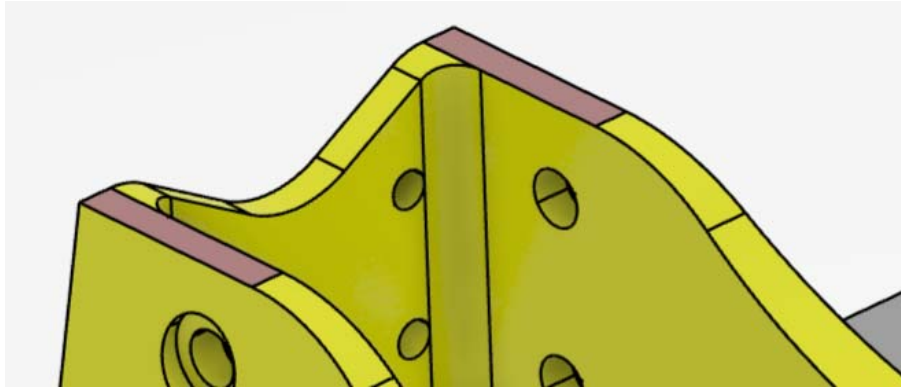


The *Imposed planes search/view* window will seek out all horizontal planes on the defined finished part and display them in the *Not Selected* frame. You can then decide which planes need to be imposed and which do not.

Select **All planes** from the *Imposed planes search/view* window. This will display all of the horizontal planes on the finished part.



Select on the **-0.625in depth**. Notice areas highlight in red. This allows you to visually inspect the area that will be imposed before making your decision.

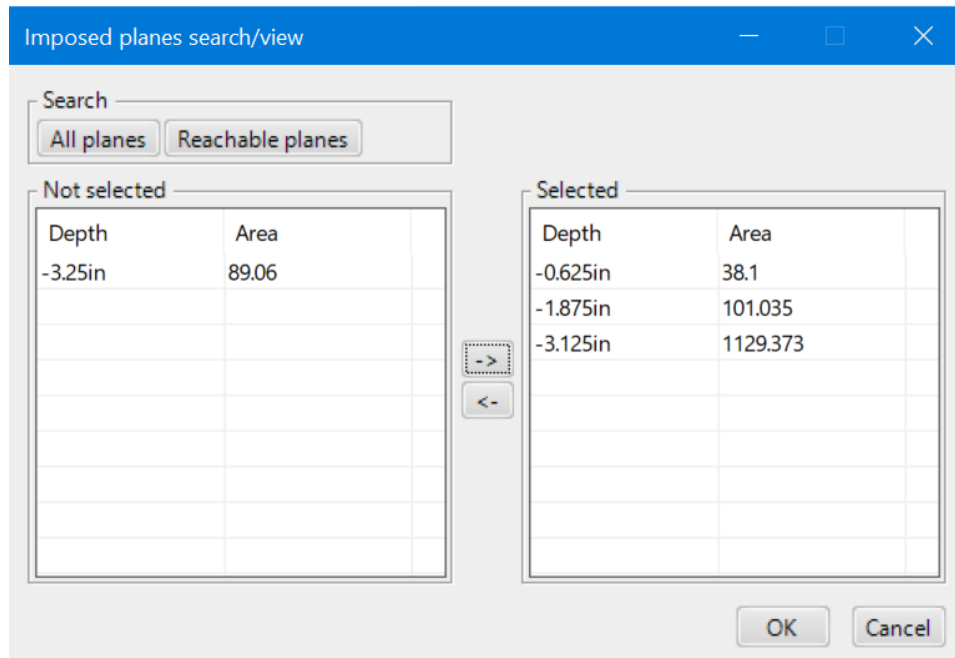


Select the **Reachable planes** button in the **Search** frame of the **Imposed planes search/view window**. This time, the list is reduced to the horizontal areas that can be reached by the tool. In this case, the 0.5in plane was removed. If other non-reachable planes existed, they would have been removed as well.

Search	
All planes	Reachable planes
Not selected	
Depth	Area
-0.625in	38.1
-1.875in	101.035
-3.125in	1129.373
-3.25in	89.06


Select the **-0.625in depth**, then select the **Right Arrow** icon.  This will move the -0.625in into the *Selected* field.

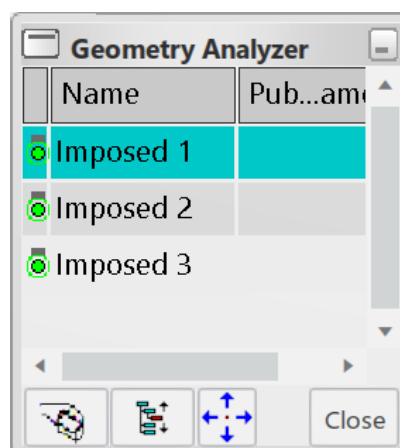
Move the -1.875in and -3.125in depths into the selected field as well. This will cause the tool to machine all three horizontal planes.



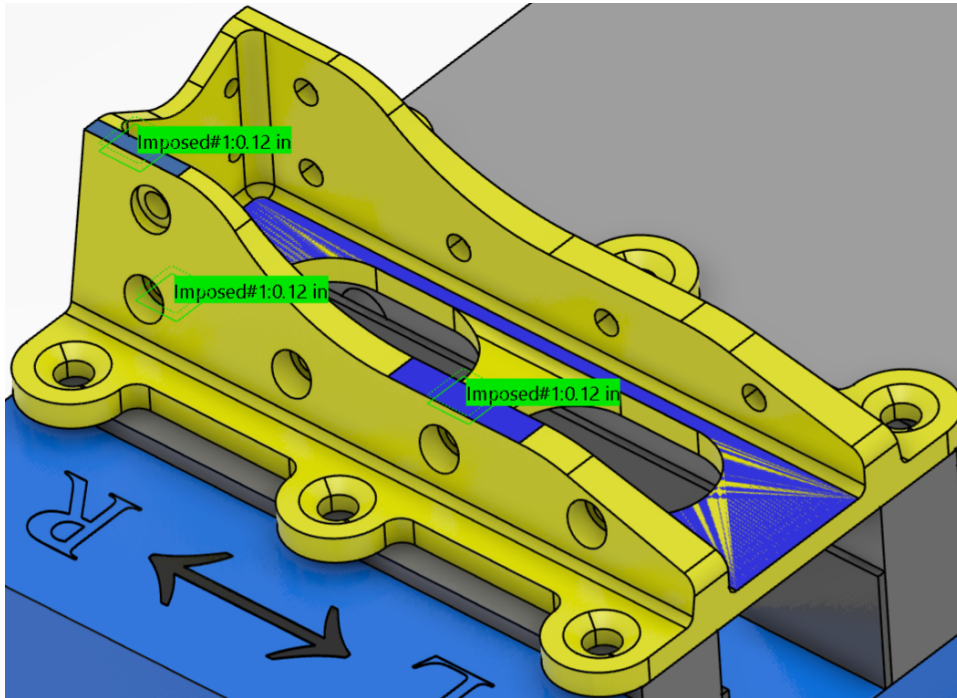
Select *OK*. This will make all the planes imposed. They should be called out in the display.

You can always go back and analyze the geometry that you have attached various areas to by using the *Geometry Analyzer*.

Select the *Analyze* icon for the *Imposed* option.  This will display the *Geometry Analyzer* window.



Select the **Imposed 1** item from the list. The top face representing that plane will turn a teal blue, indicating what plane is currently selected. The other faces representing planes 2 & 3, are royal blue.



Select the **Imposed 3** item from the list. This will change the teal emphasis from the upper plane, to the plane down inside the slot.

The *Geometry Analyzer* works with any geometry specification area. Any time you wish to view what has been selected, or what another user used in the definition of a geometry item, the *Geometry Analyzer* is exceptionally useful.

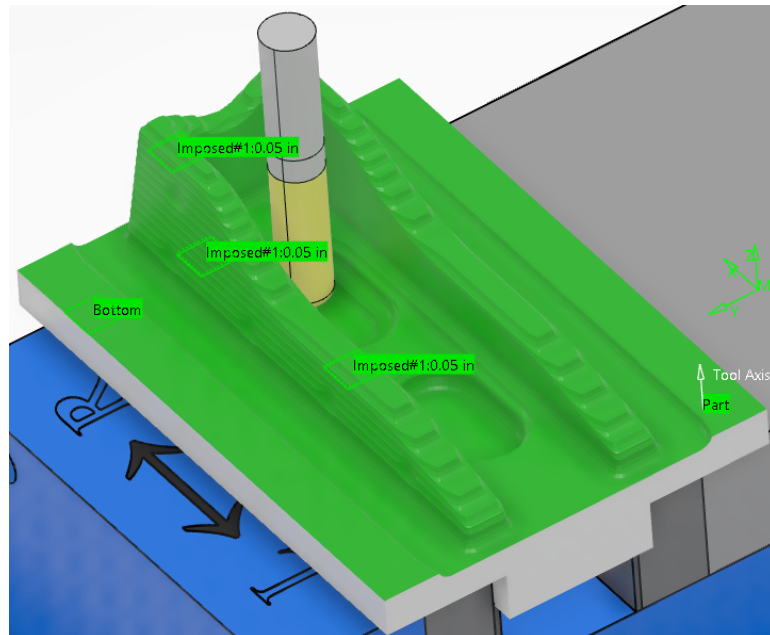
Select Close. It is not always necessary to ensure the selection of the surfaces, but sometimes, it is nice to know what was defined.

This now has the tool passing 0.125in above each of the three selected horizontal areas. Before moving on, there is one last option that needs to be set.

Select the **Bottom** definition, then select the bottom of the part. This will keep the tool from cutting into the vise below the part.

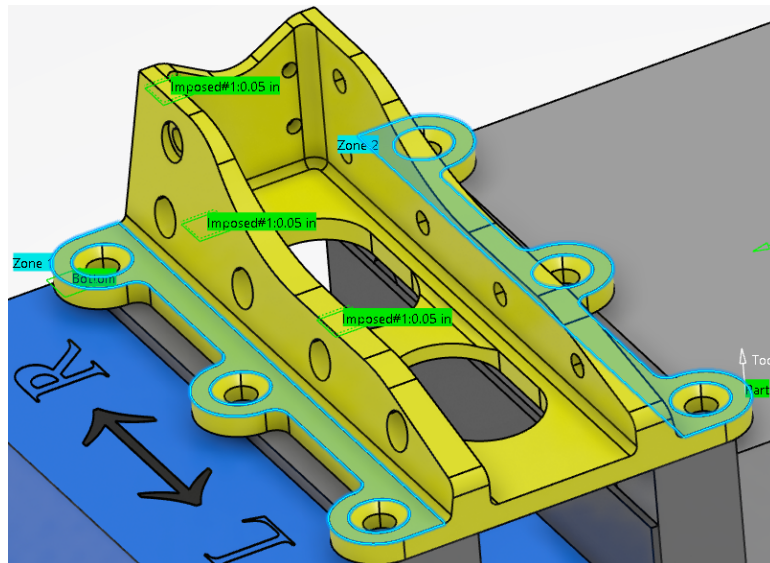
Change the offset value for the **Imposed** definition to 0.050in.

Watch the simulation with material removal. Notice the overall progress of the machining. You will see that the center holes in the part are machined prior to the exterior profile.



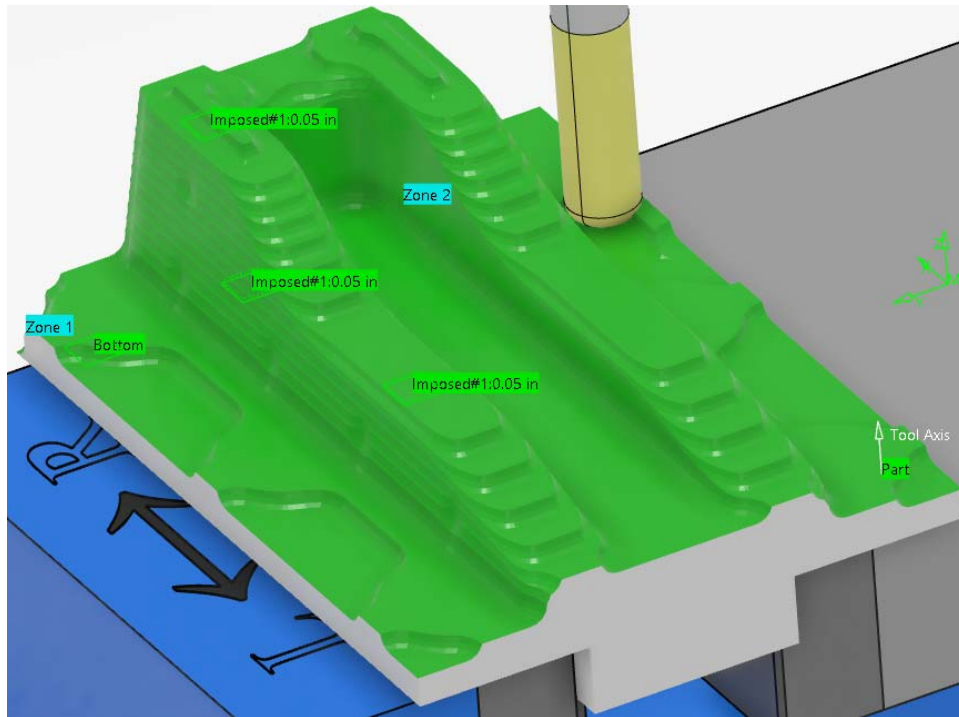
Select the *Zone order* option. The zone order allows you to define pockets or areas in the order you want them machined.

Select both of the outer boundary surfaces around the bottom of the part. This will identify the first two zones to be machined.



Double select in space when done. This will indicate that you are done selecting zones.

Simulation the operation. The exterior profile is now the first to be machined.



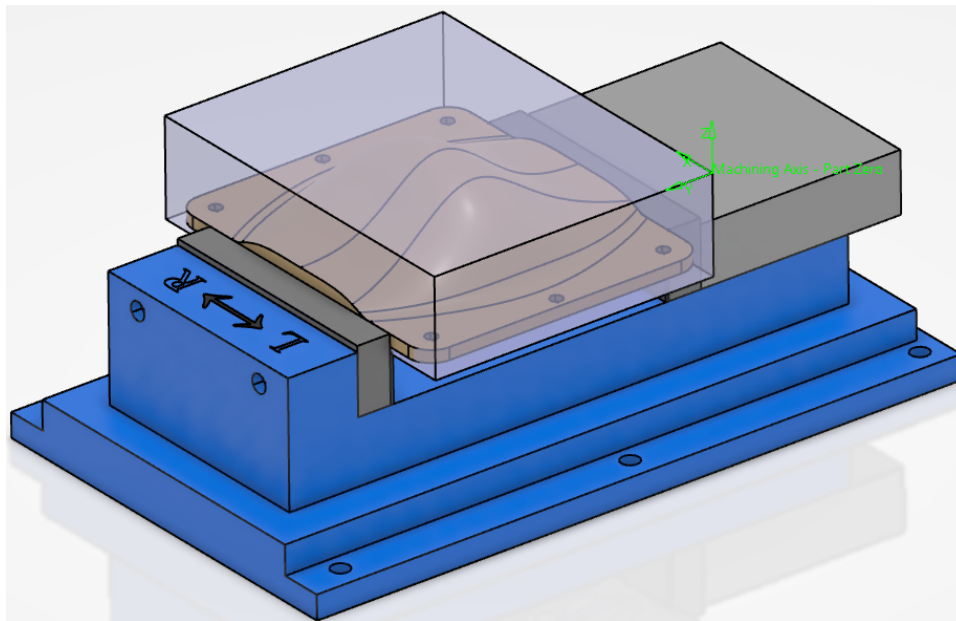
Generally, zone orders are used when you want to control the order of pocketing done in a large bulkhead. By default, in a large bulkhead with many pockets, the system will start at one end and work across the part, roughing in the most efficient manner. Although efficient, the part will gain considerable heat and may warp. By changing the order in which the pockets (zones) are machined, you can spread the heat build up around and avoid warping.

Now you will switch to the machining parameters tab and further control the toolpaths.

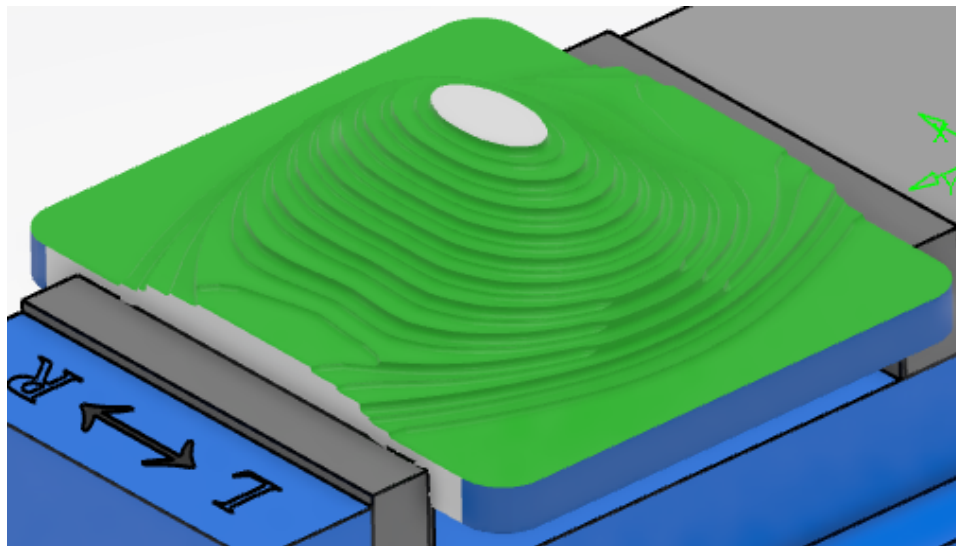
Sweeping

The sweeping operation will be the first finishing operation that you will investigate. The sweeping operation is very similar to the sweep roughing, with the exception of the ability to finely control the side steps to ensure a smooth, quality surface. The sweeping operation works best on non-vertical surfaces.

Open the SMAC050 - Sweeping process document. This is a simple model that will demonstrate the use of the sweeping operation.



Take a quick moment to review the operations that already exist. In this state, you are ready to begin finishing the surface of this part.



Start a Sweeping operation after the profile contouring operation.  The *Tool Axis* window appears.

Select **OK**. The Part definition appears.

Select the **Esc** key twice. The *Sweeping* window appears. Sweeping operations are going to be defined in a similar manner to sweep roughing operations.

Sweeping.1

New Feature

Mandatory

Part 0.25in

Tool Axis

Optional

Check 0.039in

Limiting Contour 0in

Start 0in

End 0in

Top 0in

Bottom 0.25in

Safety plane 0in

Limit Definition

Part autolimit ☒

Side to machine Inside

Stop mode Tool end

Stop position On

No Offset Group

OK Simulate Display Cancel

Geometry Tab

New Feature

Mandatory

☒ Part 0.25in

☒ Tool Axis

Optional

☐ Check 0.039in

☐ Limiting Contour 0in

☐ Start 0in

☐ End 0in

☐ Top 0in

☐ Bottom 0.25in

☐ Safety plane 0in

Limit Definition

Part autolimit ☒

Side to machine Inside

Stop mode Tool end

Stop position On


No Offset Group

You have already covered the majority of the areas for the sweeping area. There are two new options.

Optional

Start This defines the starting plane for the tool paths. The start plane must be parallel to the tool paths.

End Defines the ending of the tool paths. The end plane must be parallel to the tool paths.

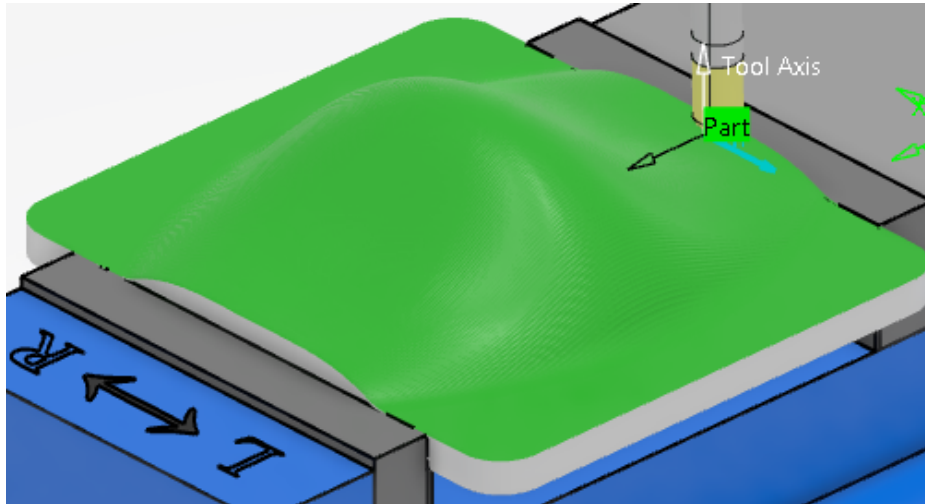
Select the **Define Part on PO icon for the *Part* definition.**  Just like with the other operations, by selecting the part definition, you will be defining the entire part body.

Make sure the offset value for the *Part* option is set to 0.0in. This will make sure you machine the final part.

For the moment, this will have enough geometry defined. You will come back and define more geometry after you have defined a different tool.

Switch to the Resource tab and select the *SMAC050 - T3 EM Tool Assembly*. This will be a more appropriate cutter for sweeping along the surface.

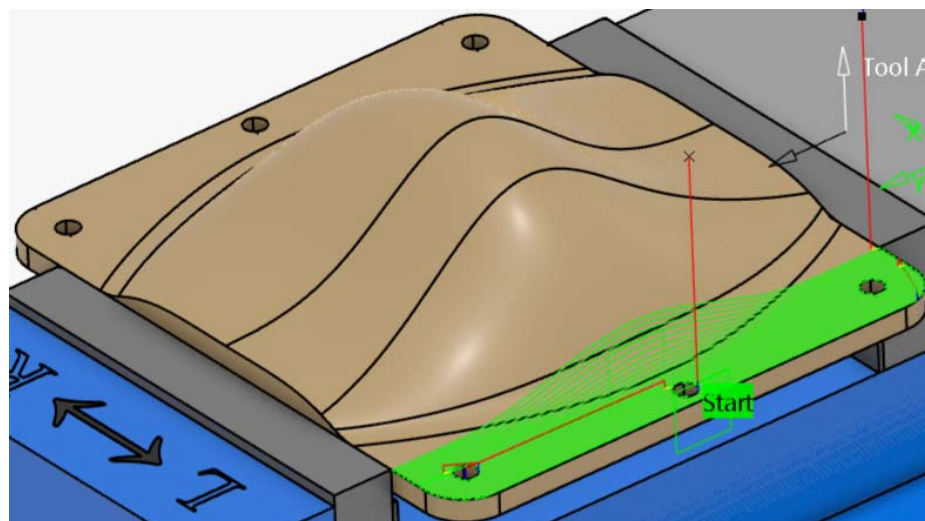
Simulate the operation. The tool simply sweeps back and forth across the part creating a moderately smooth surface.



At this point, the surface quality is still rough, however, the machining parameters allow you to fine tune the tool paths to ensure a quality, smooth surface. Before adjusting machining parameters, take a moment to look at some of the additional geometry options.

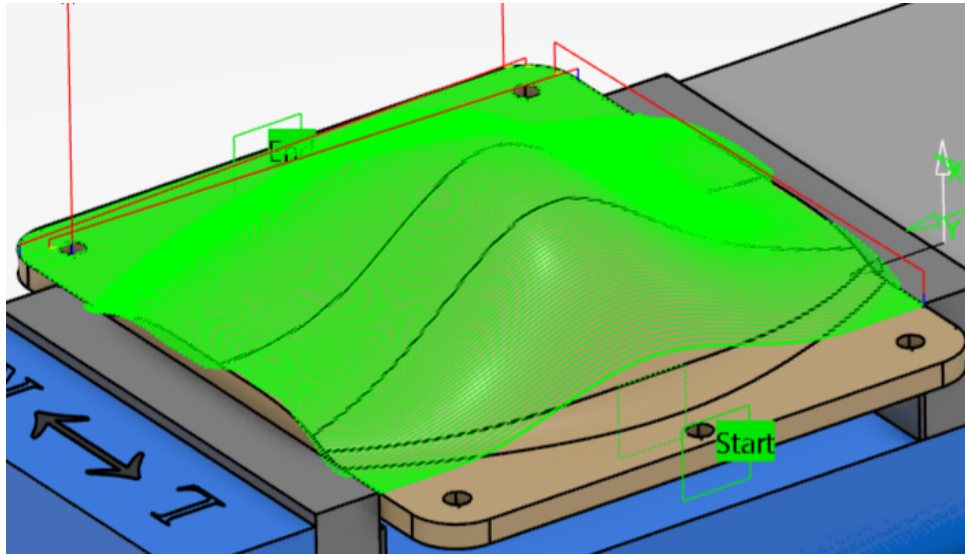
Switch back to the Geometry tab and select the *Start* option, then select the flat side as shown below. This will define a start plane to be on the side of the part. The start plane does no good until you define an offset on the plane. Keep in mind that the *Start* and *End* planes must be parallel to the tool paths.

Change the offset value for the *Start* option to -2.0in. By defining an offset of -2 inches, you will notice a small dashed plane appear inside the part.



The tool path now starts at the offset value from the start plane. By defining the start and end planes, you can control how much of the surface gets machined. Unfortunately, you cannot limit the length of the tool paths. For example, you cannot remain 2 inches away from all the boundaries with the start and end planes. For the best results, you should use a limiting contour to control the tool paths. If you wanted to do the other side of the offset plane, you need to define the End option.

Select the *End* option and define it as the opposite side of the part. It should appear as shown when you simulate.



Remove both the *Start* and *End* options. This will go back to machining the entire surface.

Select the **Define Part on PO** icon to remove the *Part* definition.

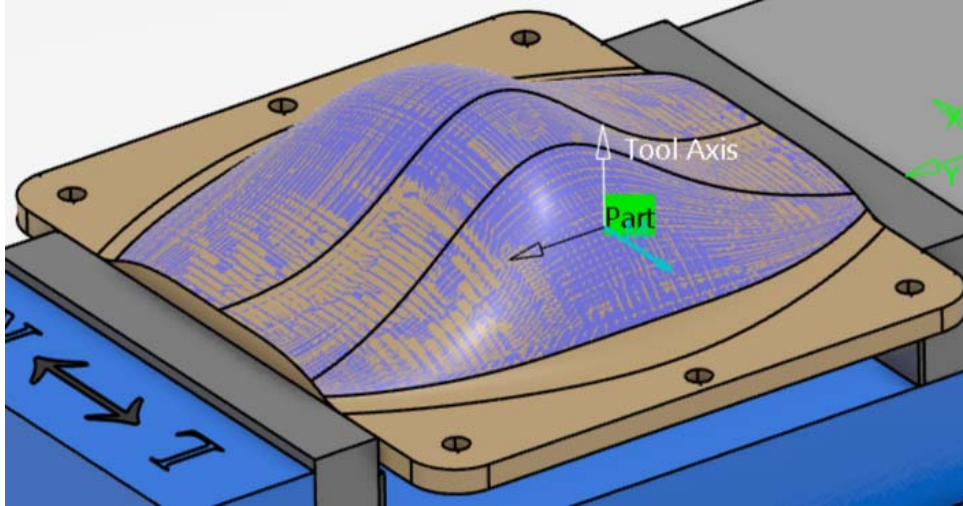


Select the **Part** definition, then select the **Select a Face** icon from the toolbar.

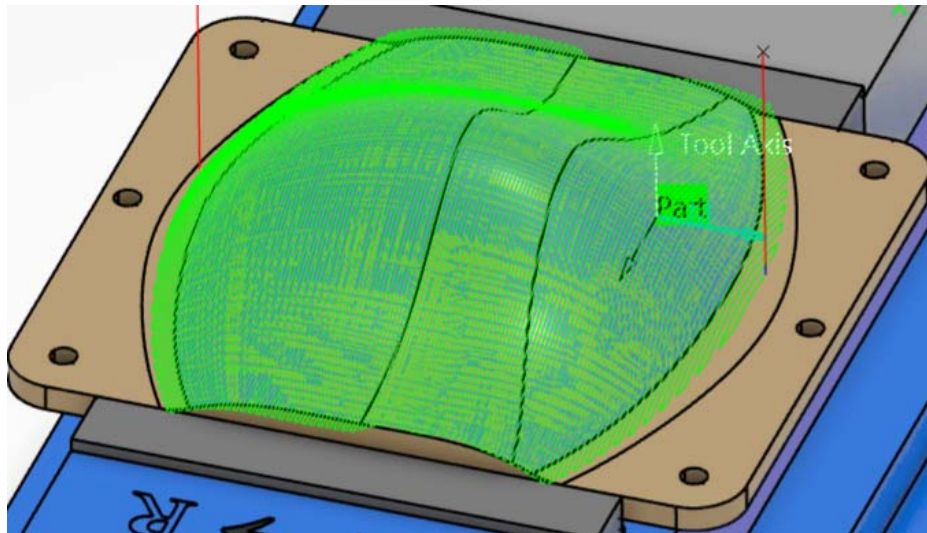


This will allow you to select just the few faces that you want to machine.

Select the two faces on the top of the part. Remember to double select in space to finalize the selection.



Simulate the tool path. Notice only the two surfaces get machined. Take a close look at the tool paths on the side. You should see that the tool paths do not conform to the contours of the faces selected.



No matter the geometry of the part or faces that you want to machine, you will find that the tool paths are always straight and parallel.

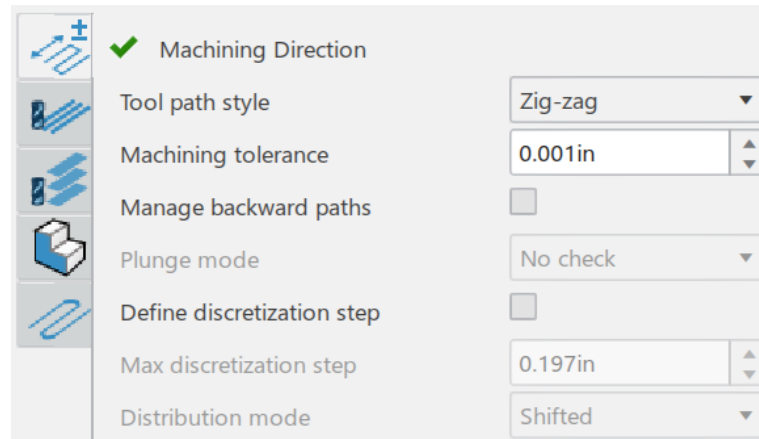
Remove the face selections and re-define the entire part as the *Part* definition. Now you will take a look at the machining parameters.

Switch to the Strategy parameters tab. You will now investigate the new options for controlling the tool paths.


Strategy Parameters

Take a quick look at the various machining parameters.

Machining



<i>Tool path style</i>	Defines the type of cut that is to be made across the part
<i>Zig-zag</i>	The tool moves back and forth across the top of the surface in a zig-zag motion
<i>One-way next</i>	The tool moves across the surface in one direction, returning to the start of the next tool pass
<i>One-way same</i>	The tool moves across the part, returning in the same path that it machined
<i>Machining Tol...</i>	Defines the tolerance of the machine being used
<i>Manage backward...</i>	Manages paths to avoid marks on the part
<i>Plunge Mode</i>	Controls if the tool plunges, or dives into the material. This option is only available with the One-Way tool path styles.
<i>No check</i>	No plunge checking is made
<i>No plunge</i>	Tool is halted before it gets into the material
<i>Same height</i>	Tool remains at the same Z-height when it is supposed to plunge
<i>Define discretizat...</i>	Overrides the step size of the machining operation
<i>Max discretization...</i>	Defines the maximum step size allowed
<i>Distribution mode</i>	Defines whether the distribution of the points will be aligned or shifted with the previous tool path

Radial


Stepover mode: Constant

Maximum distance: 0.075in

Minimum step distance: 0.039in

Scallop height: 0.01in

Stepover side: Left

Stepover mode Defines how the sidestep is calculated

Constant Tool moves over a maximum distance, keeping the scallop height at the defined size

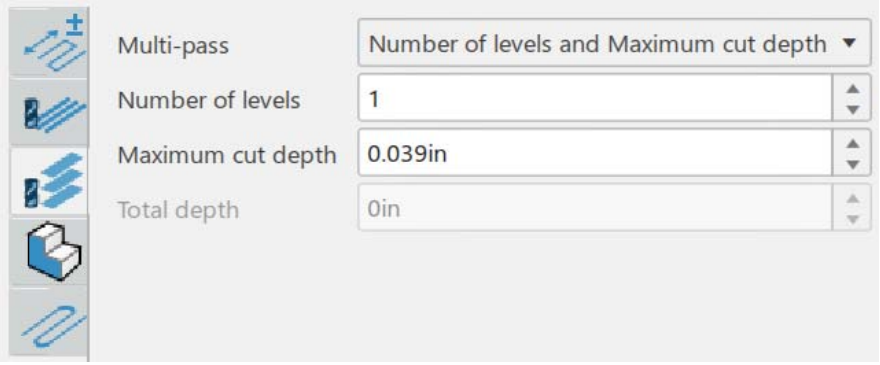
Via scallop height Tool steps over varying distances, remaining within the *Maximum* and *Minimum Distance*, while keeping a constant scallop height thickness

Maximum distance... Defines the maximum distance stepped each radial pass

Minimum step... Defines the minimum distance the tool can step each radial pass

Scallop height Defines the maximum scallop height allowed with the tool paths remaining between the maximum and minimum step sizes

Stepover side Defines which side the tool steps, *Left* or *Right*

Axial


Multi-pass: Number of levels and Maximum cut depth

Number of levels: 1

Maximum cut depth: 0.039in

Total depth: 0in

Multi-pass Defines how the multiple axial steps are calculated

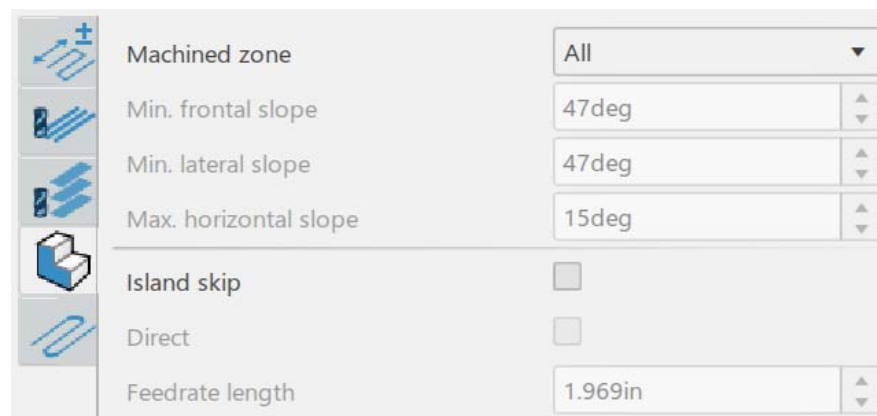
Number of levels Defines the number of axial levels

Maximum cut depth Defines the maximum depth that the tool cuts each pass

Total depth

Defines the overall depth of the part thickness

Note: Multiple axial steps with the sweeping operation should be used sparingly. The axial offset is just a vertical offset. They do not move out radially. This means that when the final pass, the pass against the part surface, is computed, a copy is made of those tool paths and they are moved along the tool axis a particular amount. This generates problems in vertical, or near vertical areas where you will be cutting a lot of material in the first pass, and then not a lot of material for all passes afterwards. Generally, you are going to want to use roughing and sweep roughing to get the part cut down to where only one finishing pass is necessary. If you find the need to make multiple axial steps, you will be better off to define multiple sweeping operations with various part offsets.

Zone


	Machined zone	All
	Min. frontal slope	47deg
	Min. lateral slope	47deg
	Max. horizontal slope	15deg
	Island skip	<input type="checkbox"/>
	Direct	<input type="checkbox"/>
	Feedrate length	1.969in

Machined zone

Controls what type of area is machined

All

All zones get machined

Frontal walls

Only the front and back walls get machined. The front and back walls are determined by the forward motion of the tool.

Lateral walls

Only machines the walls on the left and right hand side of the tool paths

Horizontal zones

Only machines areas that are within the horizontal slope angle

Min frontal slope

Minimum angle to be considered a frontal wall, typically the same as lateral slope

Min lateral slope

Minimum angle to be considered a lateral wall

Max horizontal...

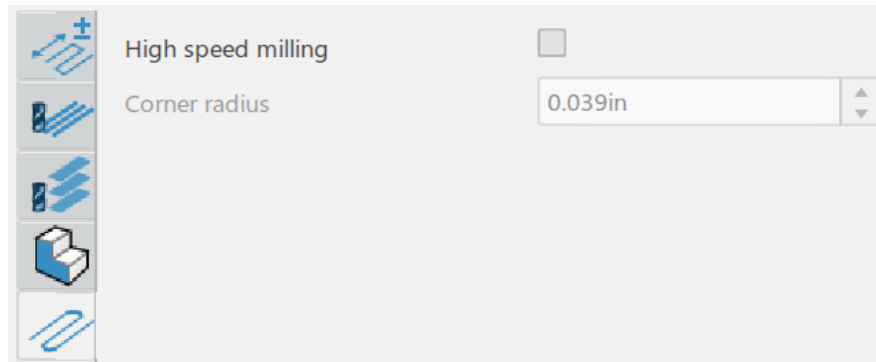
Maximum angle that will be considered a horizontal plane

Island skip

Indicates that islands, or gaps in the surface should be jumped over, not gone around

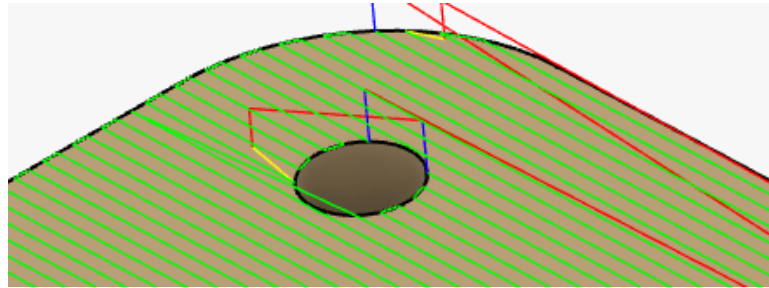
<i>Direct</i>	Causes the tool to move directly over the island without a retract and approach motion and is available only when <i>Island skip</i> is turned on
<i>Feedrate length</i>	Defines the size of a gap or island that will cause a direct island skip. Any hole or pocket larger than this length will generate a retract and approach motion.

HSM



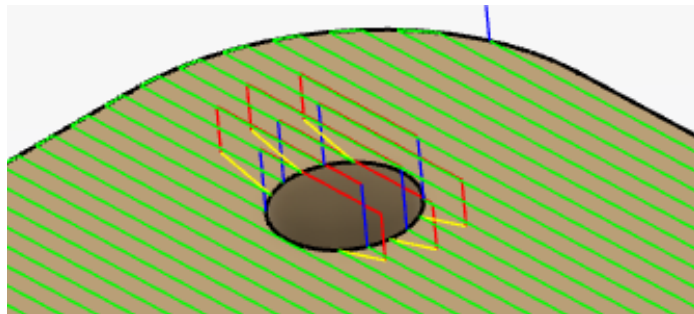
<i>High speed milling</i>	Activates high speed milling
<i>Corner radius</i>	Defines a corner radius for high speed milling

Simulate the operation. If you look closely at the holes you will notice that the tool zig-zags around them instead of machining over them.



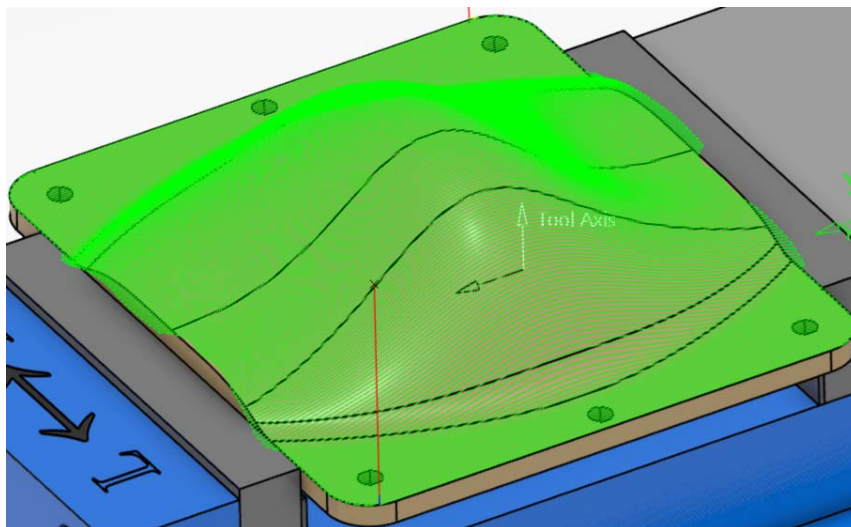
This results in a bad tool path motion and will be fixed first.

Switch to the *Zone* tab, then turn on *Island skip*. Instead of zig-zagging around the holes, the tool instead jumps up and over the holes. This is also incorrect, however, it is a step closer to what you want.

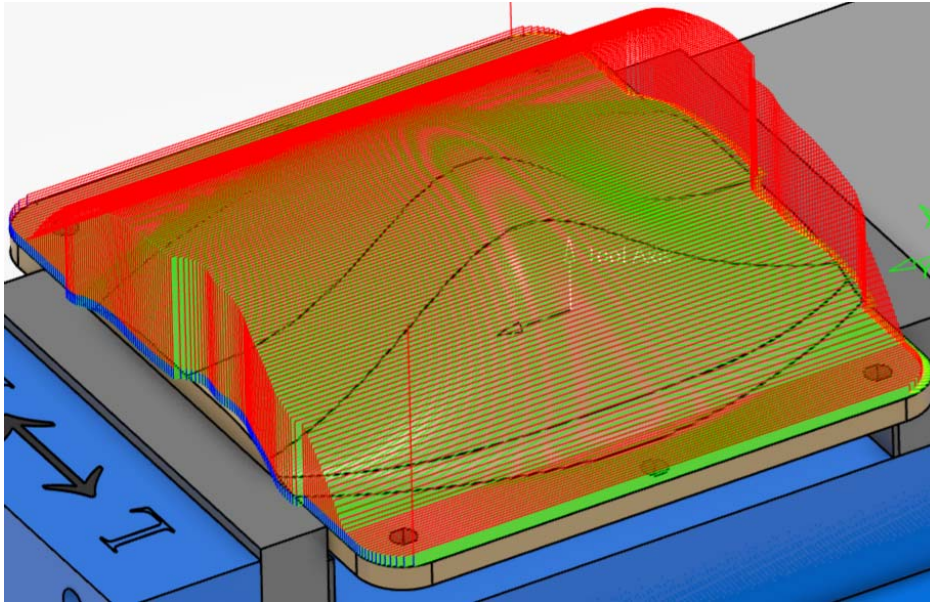


Turn on the *Direct* option under the *Zone* tab. Once you turn on direct, you have the ability to define the *Feedrate length*. If you remember from earlier, the *Feedrate length* defines the size of a gap the tool will cross in a direct motion.

Change the *Feedrate length* to 1.0in. Since the holes are all less than 1in in diameter, this will be sufficient to cause the tool to move directly across them. Direct island skipping will also work to machine directly across a pocket.

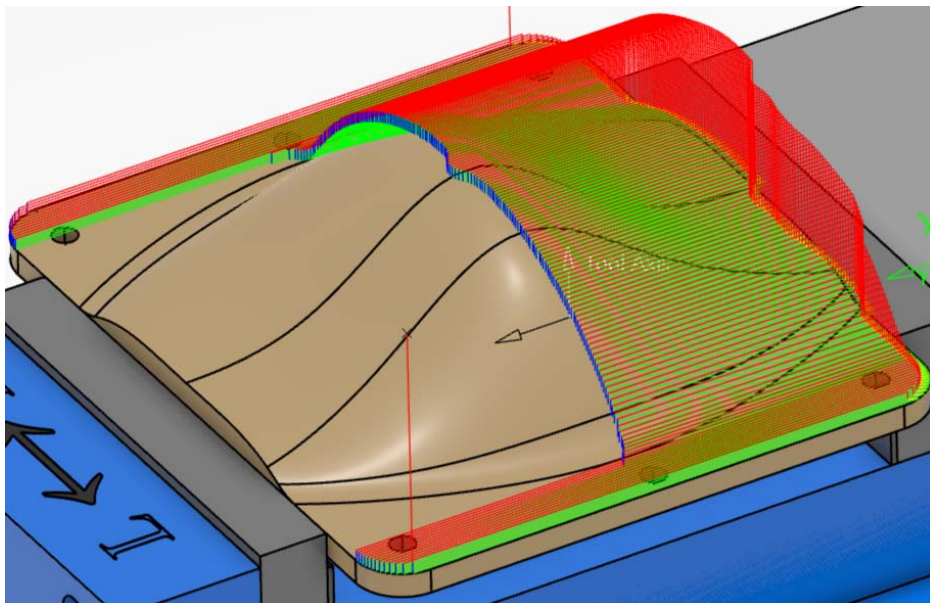


Change the *Tool path style* to *One-way next*. This will simply cause the tool to start on one side, then machine only in one direction.



Notice the tool makes rapid motions back to the start of the tool paths. You can adjust the macros to make this a smoother motion. This will be taken care of later.

Change the *Plunge mode* to *No plunge*. This will cause the tool to machine up to the highest point of the surface, then stop.

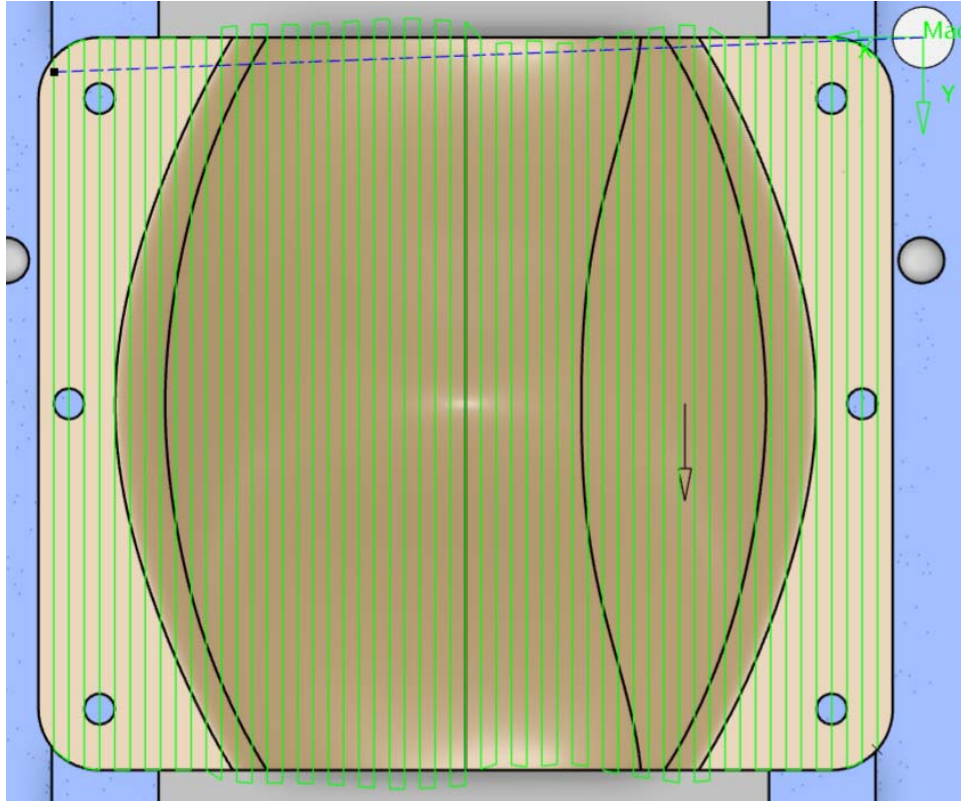


Not allowing the tool to plunge is good if you want to machine upwards along the part surface. By turning off the plunge, the tool will stop at the highest point, allowing you to go back and machine the opposite side without using limiting contours.

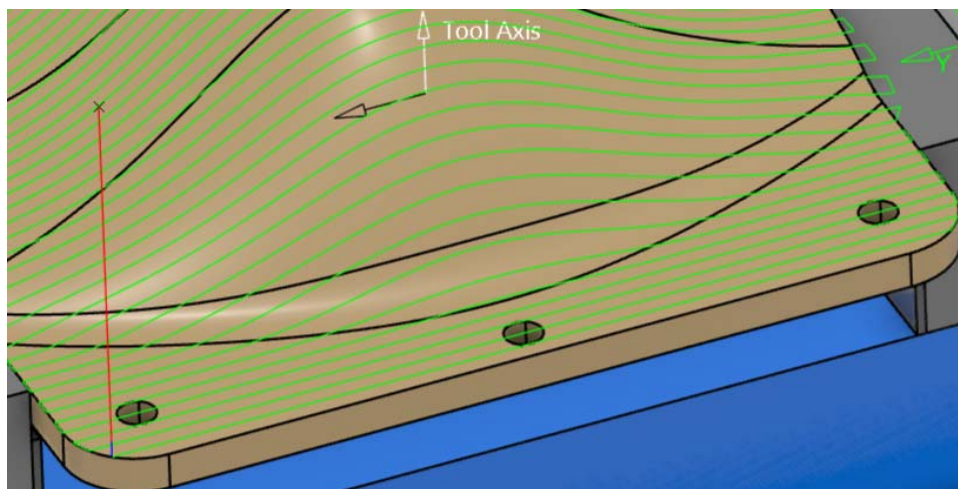
Change the plunge back to *No check*. This will go ahead and allow the tool to machine across the entire surface.

Change the *Tool path style* to **Zig-zag**. This will make it easier to see the changes made to the radial steps.

In the *Radial* tab, with the *Stepover* set to **Constant**, change the *Maximum distance* to **0.25in**. This sets all the steps to be a constant, even 0.25in.



Even though the steps are set to a constant 0.25in, if you look at the tool paths from another angle, you can see that the steps are only constant along the tool axis.

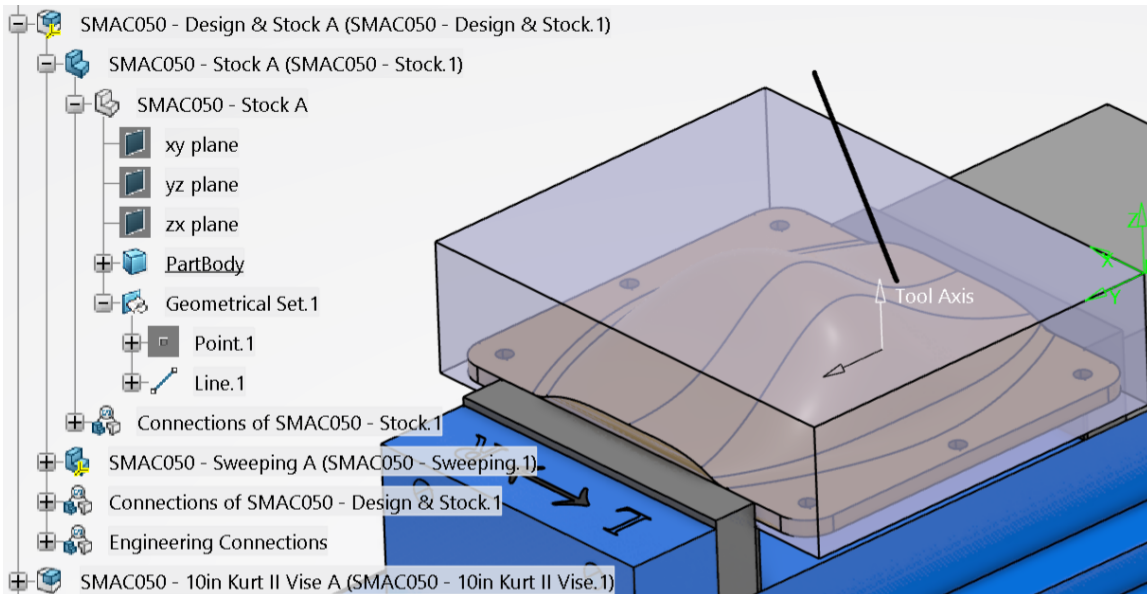


This can be impacted by modifying the tool axis. First, you will want to get a ball nose tool which will allow for better machining with a modified tool axis.

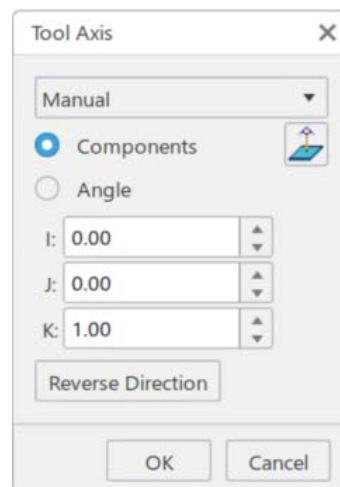
Switch to the **Resource** tab and select the **SMAC050 - T4 EM Tool Assembly**. This will convert the 1in bull nose end mill to a 1in ball nose end mill.

Switch back to the **Geometry** tab.

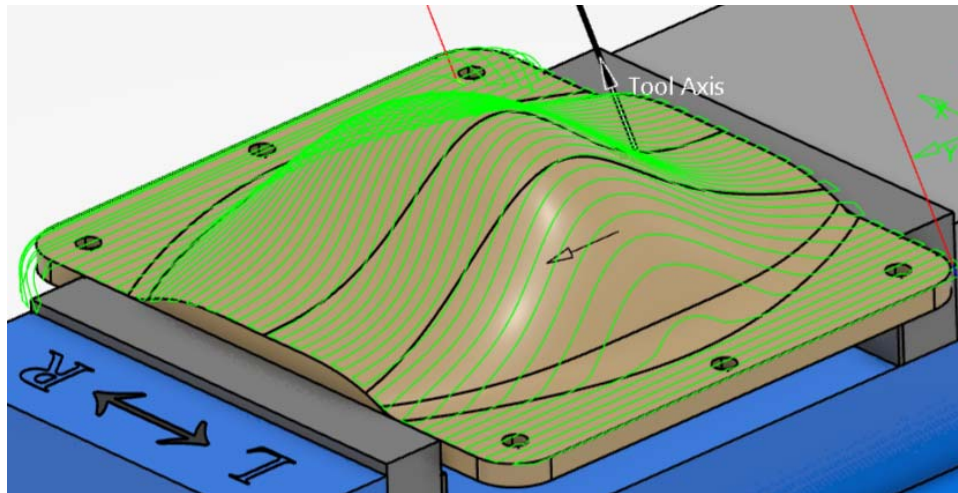
Unhide **Line.1** from the stock material in the specification tree. This line will serve as an alternative view direction.



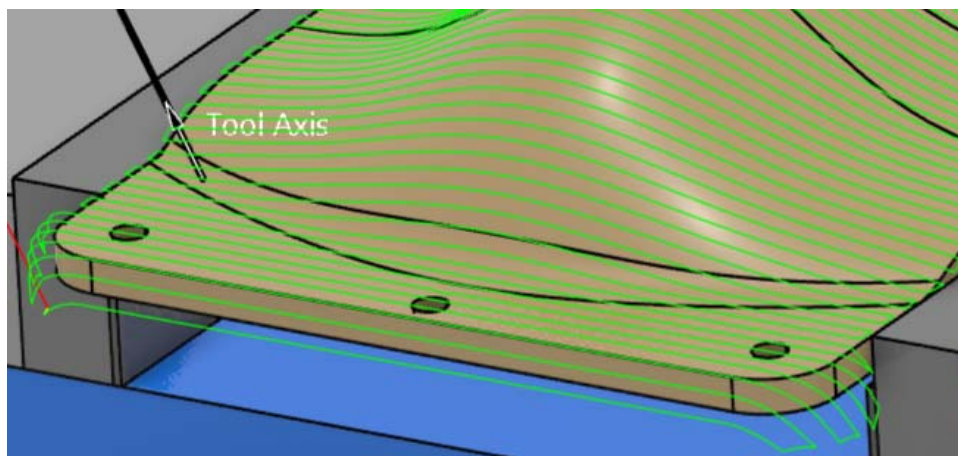
Select the **Tool Axis** definition. The **Tool Axis** window appears.



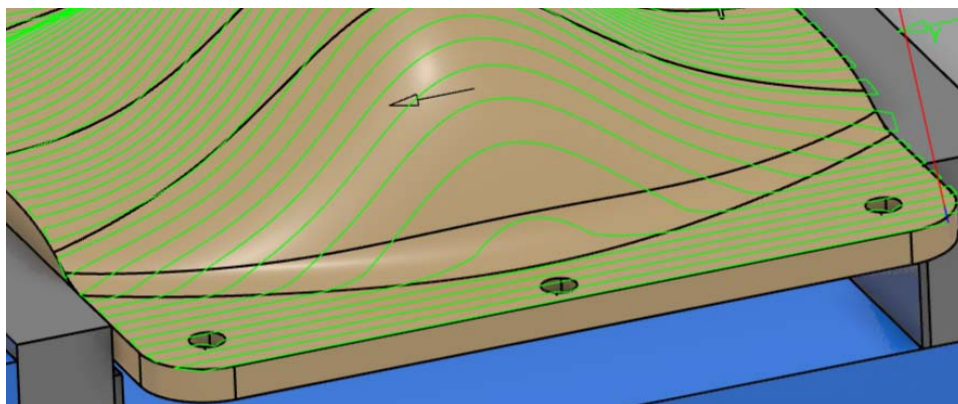
Change the drop down menu to **Selection**, then select the **Line.1** that you un-hid from the stock material. Select **OK** to the **Tool Axis** window. This will change the direction that the tool paths are computed.

Simulate the tool paths.

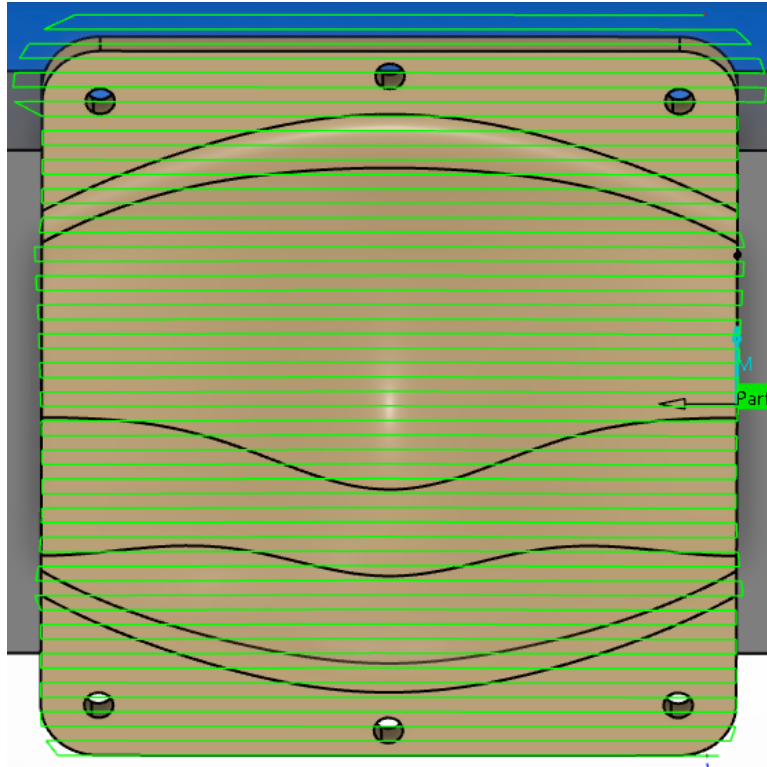
The tool paths along the flat surface are nearly the same as what you had before. Notice this time though, the tool paths extend down the side of the part and are more even against the side of the part.



Even though the tool paths on one side of the part are better, if you check the tool paths on the other side of the part, you will find that they are much worse.



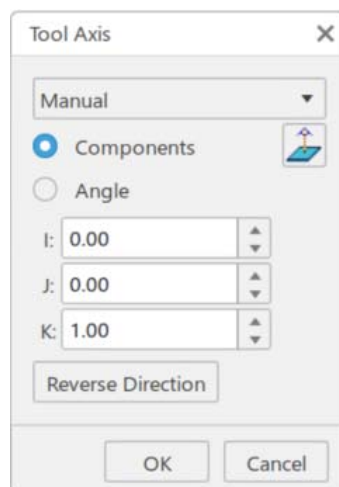
This is because when you look down along the line, you can see the side of the part, whereas, when you look down the axis, you cannot see the sides of the part.



You should also notice that when you look directly down the line, you see straight, even, 0.25in paths all the way across the part. Again, this is due to the tool axis being changed such that the 0.25in constant tool paths are computed aligned to the tool axis.

Unfortunately, if you have a surface that has a larger range of normal directions, changing the tool axis will have undesired results.

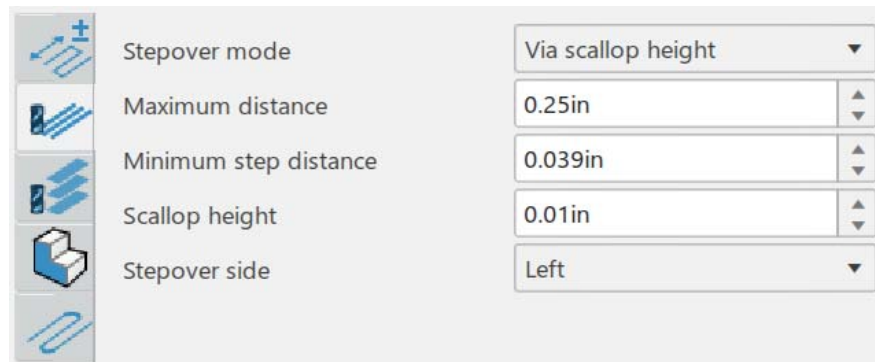
Change the *Tool Axis* back so the options match the window below. This will return the tool paths back to what they were earlier.



Hide the *Line.1* in the stock as well.

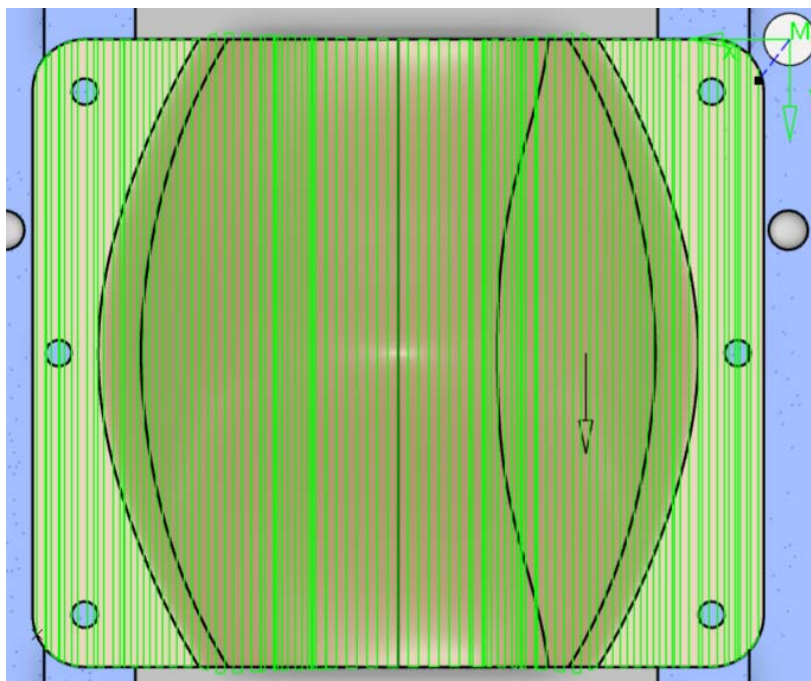
In the Strategy tab, change the *Stepover* to *Via scallop height*. This will activate the *Minimum step distance* option.

Now you have all three distance fields available to you.

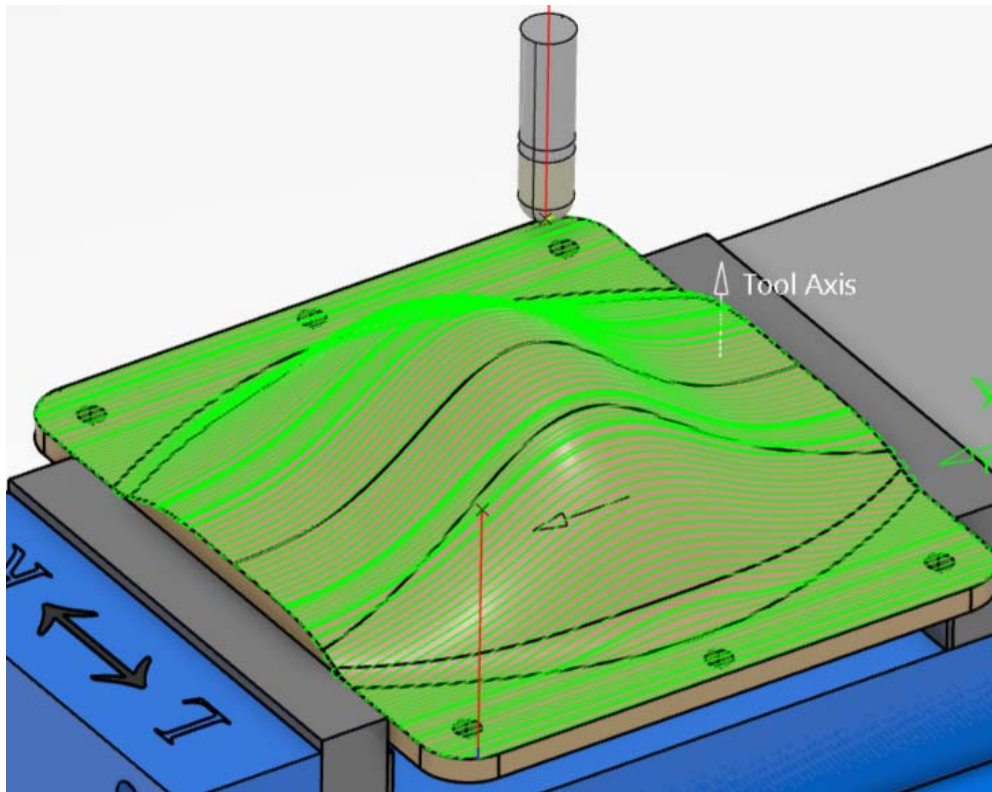


There is one rule that you must follow for the *Via scallop height* to work properly. The scallop height distance should be between the minimum and maximum distance between passes. If you have a minimum step of 0.125in and a scallop height of 0.005in, the tool will always make a step of 0.125in because that is set as a minimum. The opposite is also true. If you have a maximum distance set to 0.005in, and a scallop height of 0.020in, the tool will always make a 0.005in step, and your scallop height will be much smaller than what you are trying for. This results in various problems as you might suspect. Granted, this is based on the tool size and type of tool that you are using, but in general, you will want to make sure the scallop height falls between the minimum and maximum step sizes.

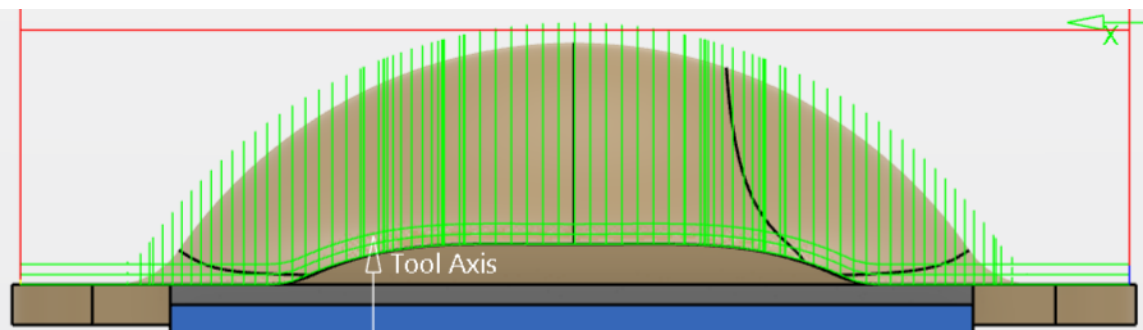
Set the *Minimum step distance* to 0.005in, with a *Scallop height* of 0.010in and the *Stepover side* set to *Right*. This will cause the tool to allow for varying sized steps as it goes across the part. The tool will do its best to maintain a scallop height of 0.010in, but will guarantee that the step size will never go under 0.005in or above 0.25in.



Change the Stepover side to Left. Notice the tool starts in the back left side, then steps right. You should also notice the blue machining direction axis change sides in the workspace. You can also flip the stepover side by selecting this axis.



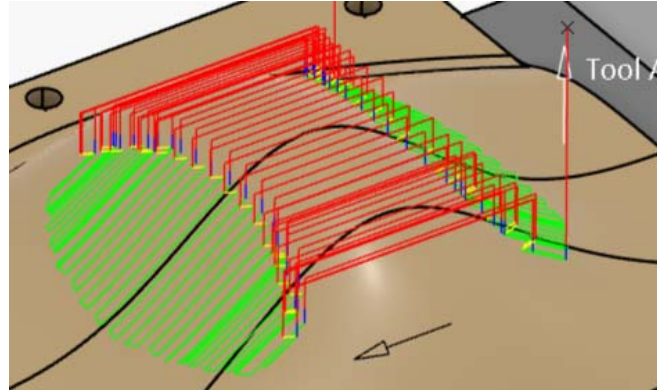
Under the Axial tab, set the Number of levels to 3, with a Maximum cut depth at 0.125in. This will indicate that you wish to have three sets of tool paths that are 0.125in apart. Take a look at the resulting tool paths, you will notice that it is a vertical offset from the bottom set of tool paths.



Since this is a purely vertical offset, it does not work very well to remove excessive material. You are better off to remove the excessive material with a sweep roughing operation prior to the sweeping operation, or to define multiple sweeping operations with part offsets.

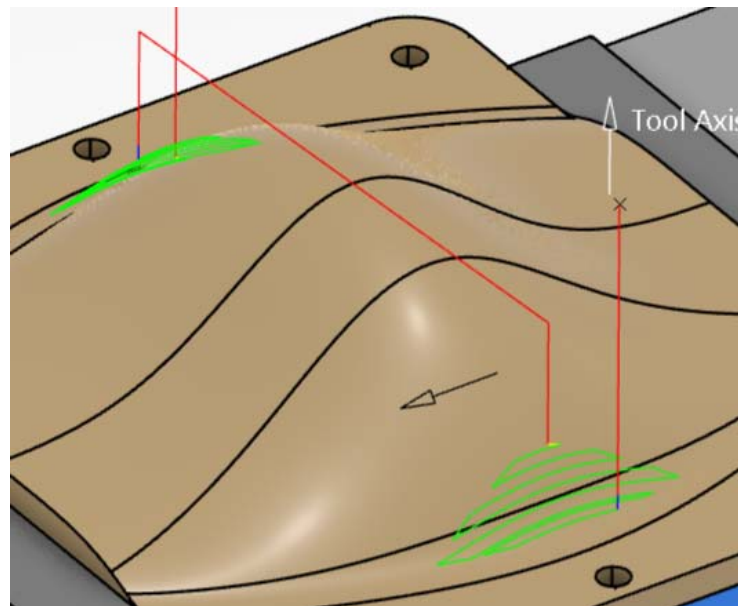
Set the Number of levels back to 1. Since the multi-pass option is not very good for performing multiple axial steps, you will set it back to one for now.

Switch to the **Zone** tab, then change the **Zone** to **Frontal walls**. Set the **Min. frontal slope** to **30deg**. This will cause the sweeping operation to only cut the frontal walls, or walls that the tool will hit head-on. By defining a 30deg minimum angle, you are defining the minimum angle from vertical that the tool will cut. The larger the angle, the less area the tool will machine.



Notice only a small frontal area gets machined. Basically, anything within 60deg ($90^\circ - 30^\circ$) of vertical, and on the frontal plane gets machined. Everything else is ignored.

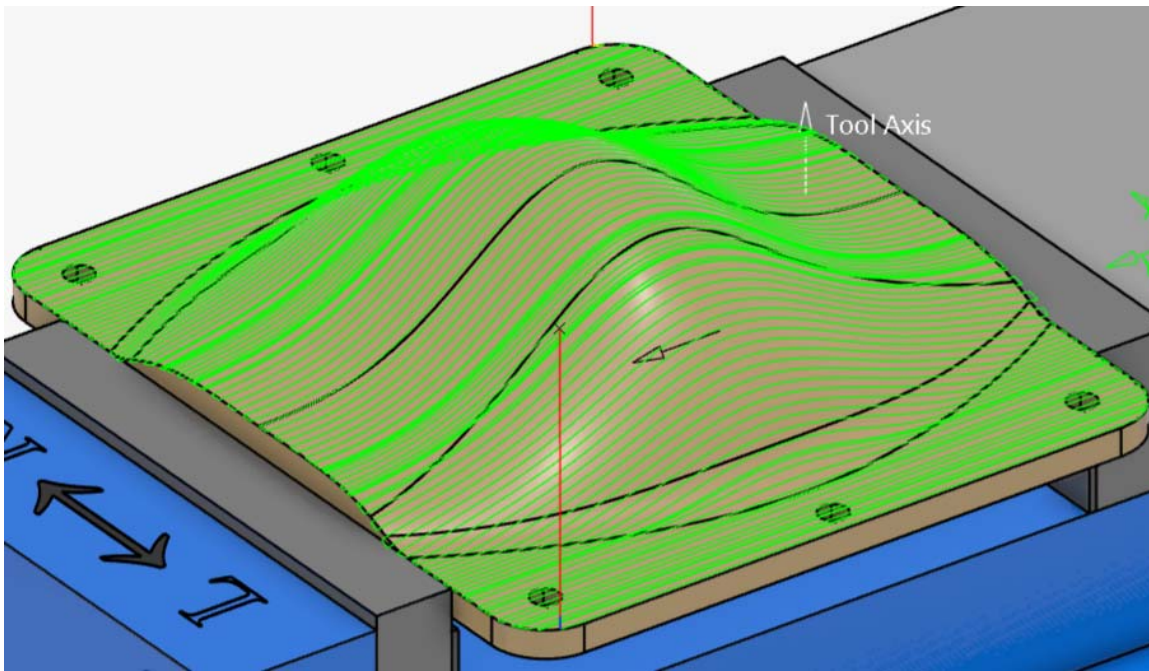
Change the **Zone** to **Lateral Walls**, with a **Min. lateral slope** of **20deg**. This time the walls on the sides get machined.



As you might suspect, the **Horizontal zones** option works in a very similar fashion where it will machine only horizontal areas, and none of the lateral or frontal walls. In most cases, you will find that you want to machine all zones.

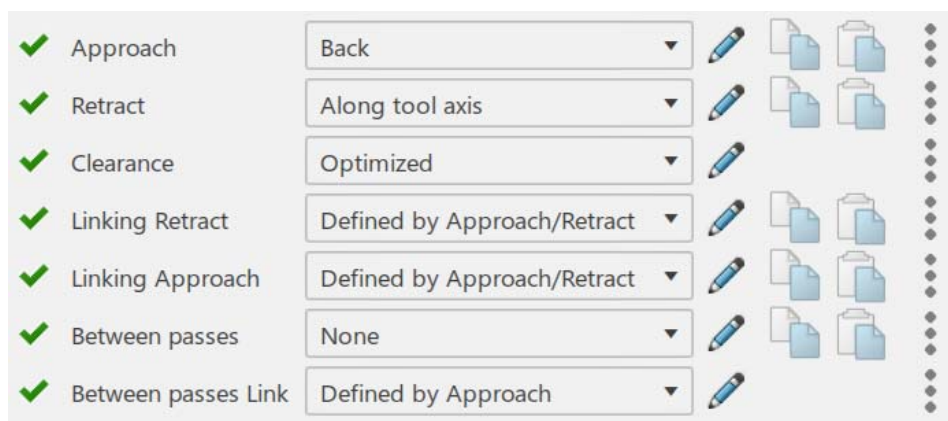
Change the **Zone** back to **All**. Now, take notice of what happens when the tool approaches the holes in the bottom of the part.

Switch to the **HSM** tab, then turn on **High speed milling**. Set a **Corner radius of 0.25in.** Notice that you do not see anything different. Very rarely will you find changes to the tool paths when working with sweeping and high speed milling. The only time you will notice a difference is if the sweeping operation had sharp corners internal to the operation.



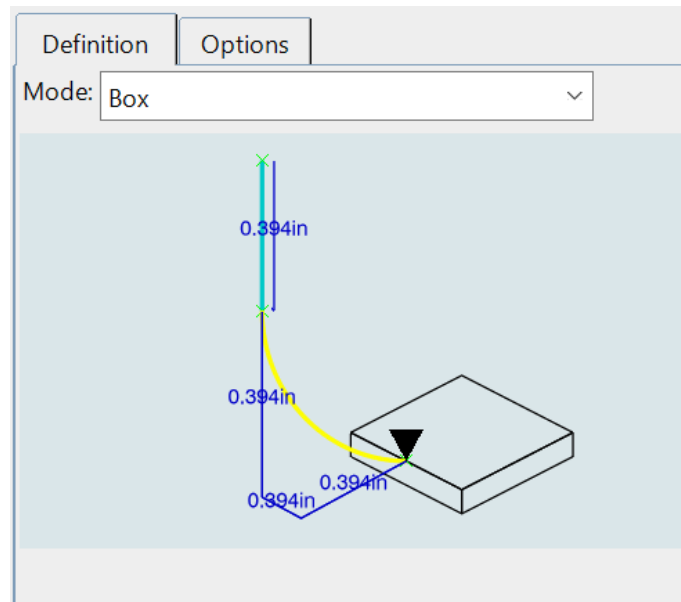
This has the sweeping operation almost complete. The only thing left is to define the macros.


Switch to the **Macros** tab. Notice that you are now back to the standard surface machining macro style. If you want to review the different macro types and available macros, refer back to the sweep roughing exercise.



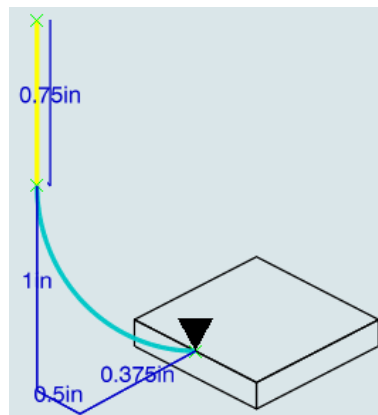
Select the Edit icon for the **Approach** macro.  The Approach window appears.

Change the *Mode* to *Box*. This will create the imaginary box around the tool path to create a nice, S-Curve shape into the part.



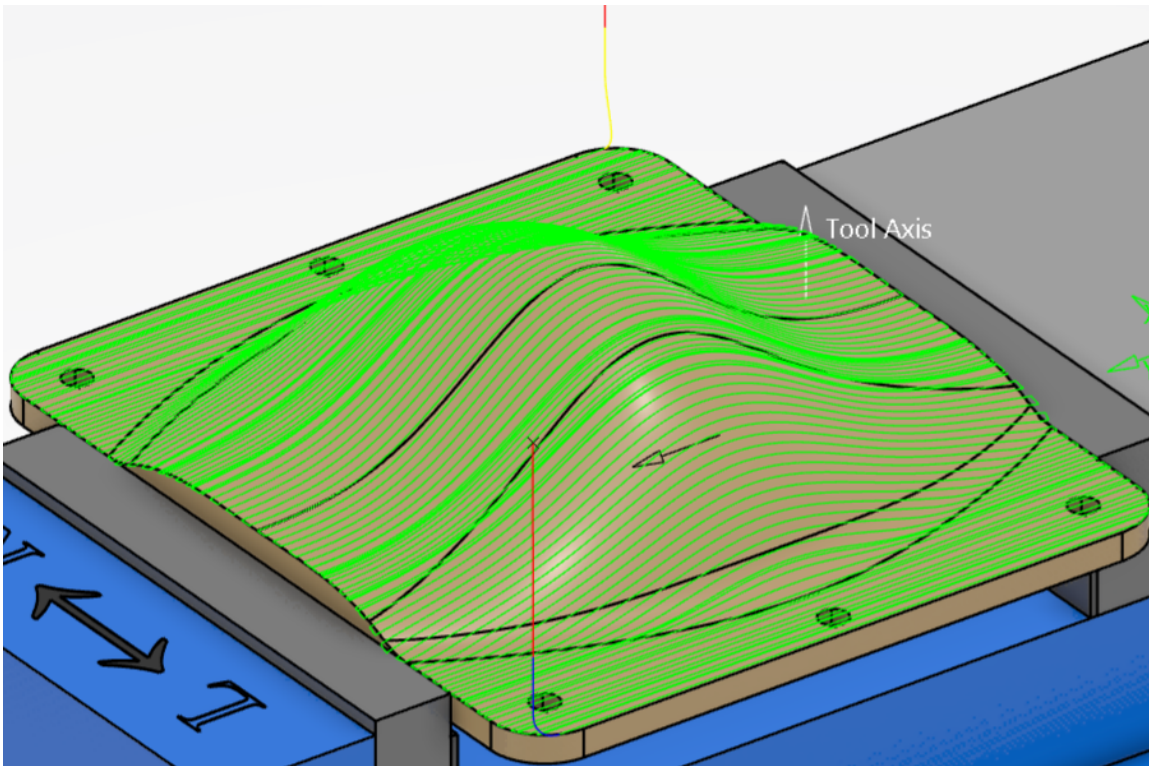
Select the **Switch Macro Definition Mode** icon.  This mode can be easier to define the macros than using the graphical picture. It is good to know how to define the macros in both modes.

Change the parameters as shown. This will make the box have a 1.0in height, 0.375in width, and 0.5in depth.



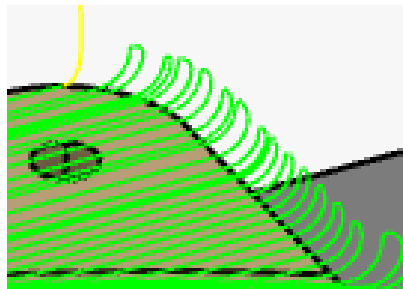
Change the *Retract* macro to a circular macro, with the *Angular sector* at 90deg, the *Angular orientation* at 90deg, the *Radius* at .5 and the axial distance at 1.0in. As you might have noticed, using the pre-defined macros allow for a quick definition of the macros. Many times it is advantageous to use the pre-defined macros instead of trying to custom build macros all the time.

Simulate the operation. It was mentioned earlier that the high speed milling options were primarily defined in the macros. Take a moment to look at the HSM options that you can use to help get HSM movements.



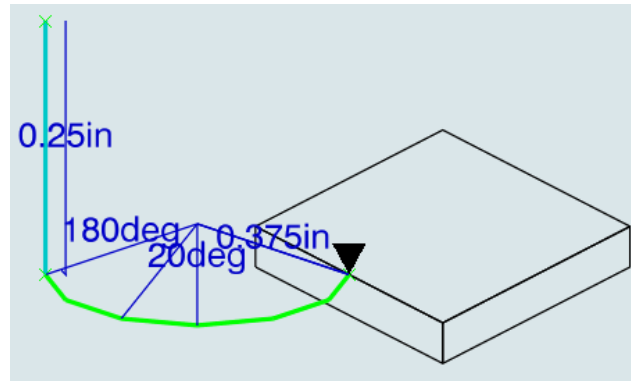
Set the *Between passes link* macro to *High speed milling with a 0.25in Transition Radius*. This will indicate that you wish to have a high speed milling motion between the passes with a 0.25in radius off the part. If the *Between passes* macro is not defined, there may be some situations where not all of the paths will have the radius.

Simulate the tool paths.

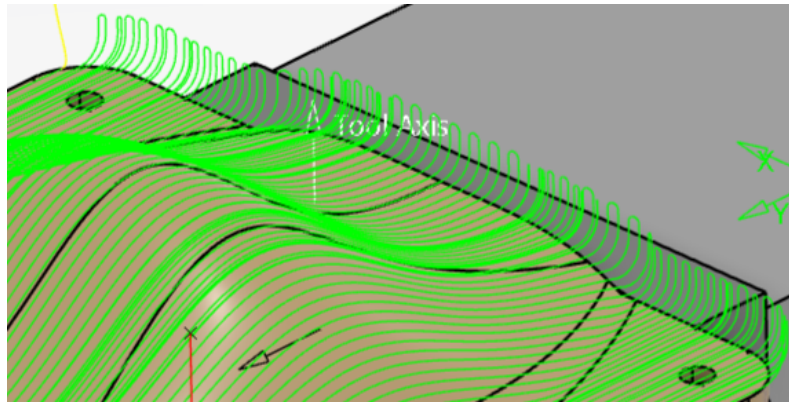


Set the *Between passes* macro to **High speed milling with the following parameters.**

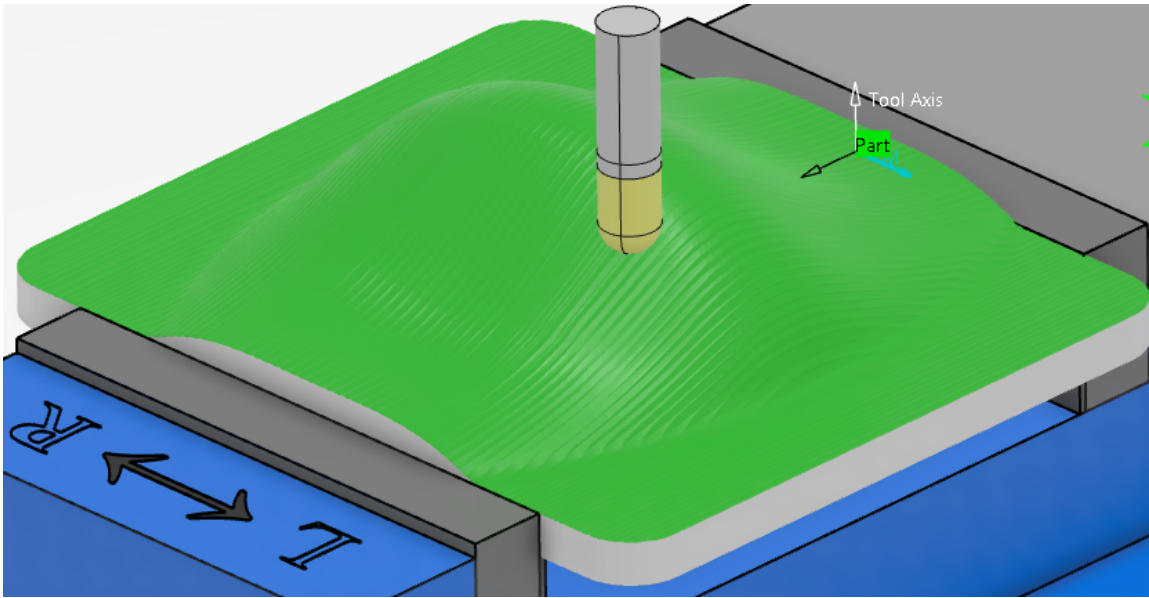
This is will define a 0.375in radius at the end of each pass, then move vertically 0.25in before circling around and returning to the part.



Simulate the tool paths. Now all of the paths move off the part before machining in the opposite direction.



Notice that this takes care of all the “bad” locations that were left behind before. Through the combination of the between passes and between passes link macros, you can obtain full high speed milling motions at the end of the passes.



Save and close the document.