TABLE OF CONTENTS

Introduction

DELMIA Version 6 Surface Machining

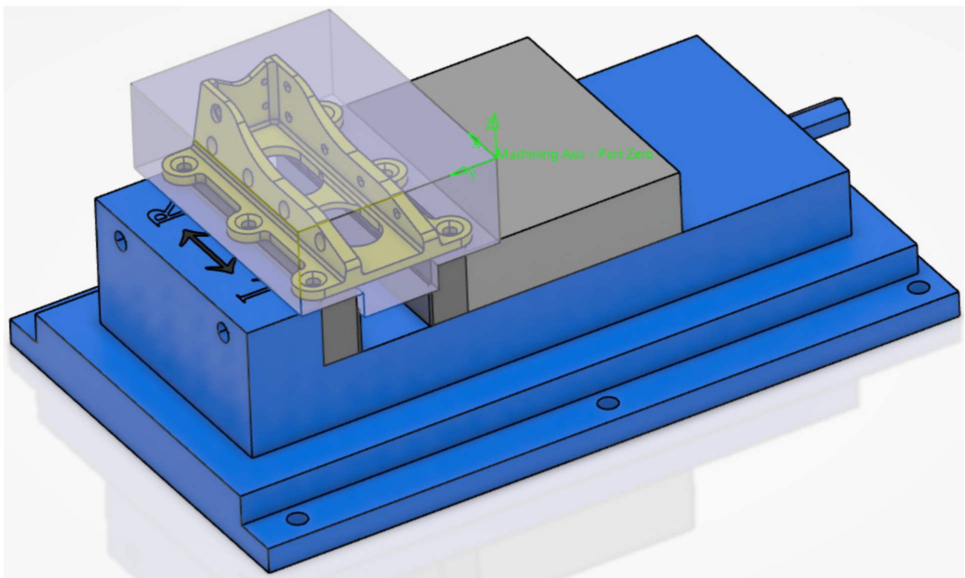
Upon completion of this course, you should have a full understanding of the following topics.

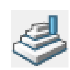
- Use the Surface and Advanced Machining workbench to machine complex contours and part surfaces
- Machine through the use of various roughing, sweeping, and reworking operations
- Integrate Prismatic Machining, Surface Machining, and Advanced Machining workbenches together to get a full range of manufacturing 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.

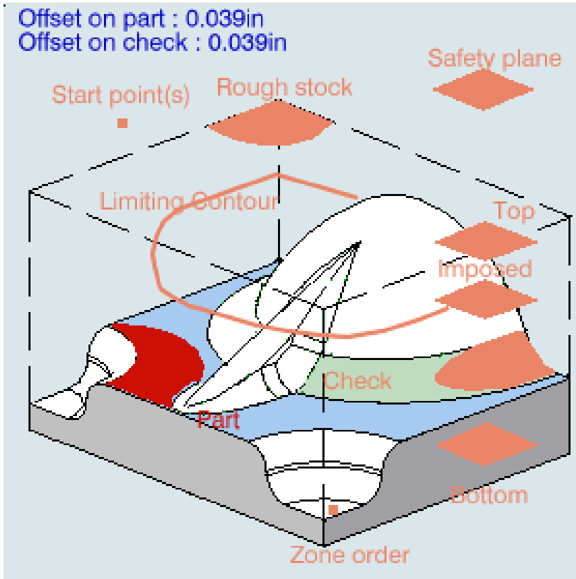


Select the Roughing icon and select *Manufacturing Program.1*.  The *Roughing* window appears.

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



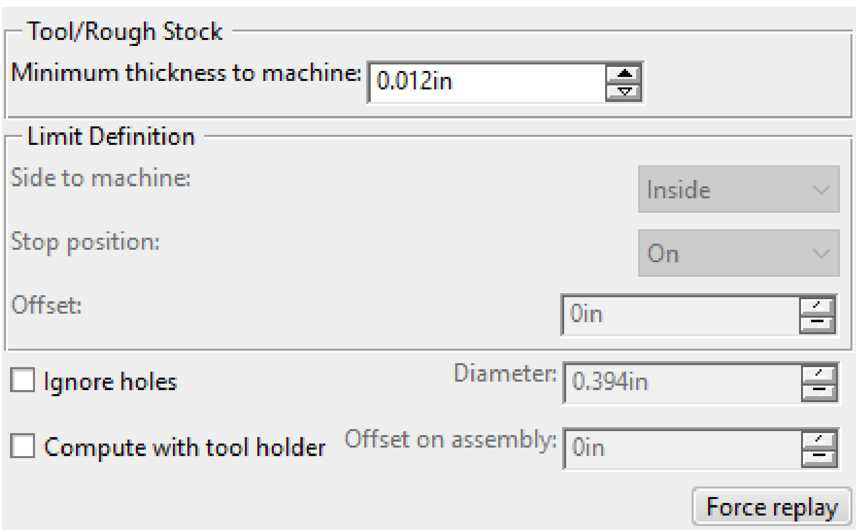
Start point(s) Defines a point or multiple points that the roughing operation can start from.

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.

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.

Zone order Defines what areas are to be machined in what order. With the contextual menu, you can also require that only ordered zones get machined.

Tool/Rough Stock



Minimum thickness... Defines the minimum amount of material that will be removed when performing a reworking operation

Limit Definition These options become available when you define a *Limiting Contour*. The options are the same as what you learned in the Sweep Roughing.

Ignore holes on... Ignores holes in the rough stock

Diameter Defines the size of holes to ignore

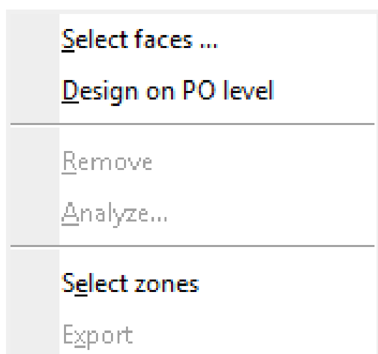
Compute with tool... Computes the tool path to avoid collisions with the tool holder

Offset on assembly Defines the offset on the tool holder assembly

Force Replay When using the Roughing operation as a reworking operation, *Force Replay* will calculate the rough stock remaining on the finished part, allowing for tool paths to be computed. Typically, forcing the replay is not needed.

If you remember from the Sweep Roughing operation, you can select the part definition, and then select the part body. There is also a contextual menu available for the part definition.

With the third mouse button, select on the *Part* definition. This will display the contextual menu.



Select faces... Allows you to select only a few faces to machine

Design on PO... Allows you to use the part defined at the Part Operation level

Remove Allows you to remove a body, or surfaces from the part selection

Analyze... Displays a part analysis window that will highlight the surfaces or bodies

Select zones Allows you to select pre-defined zones to machine through the *Zone Selection* window

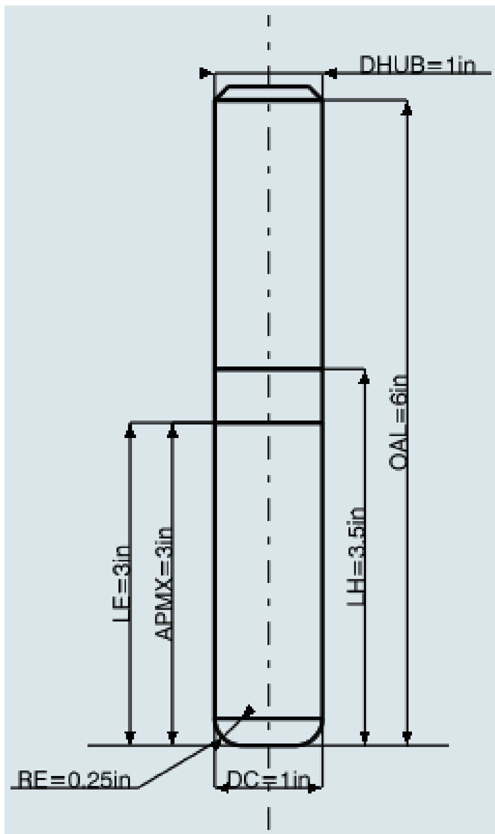
Export Allows you to save the part definition as a zone, later to be selected with *Select Zones*

Now to actually define geometry to machine the part.

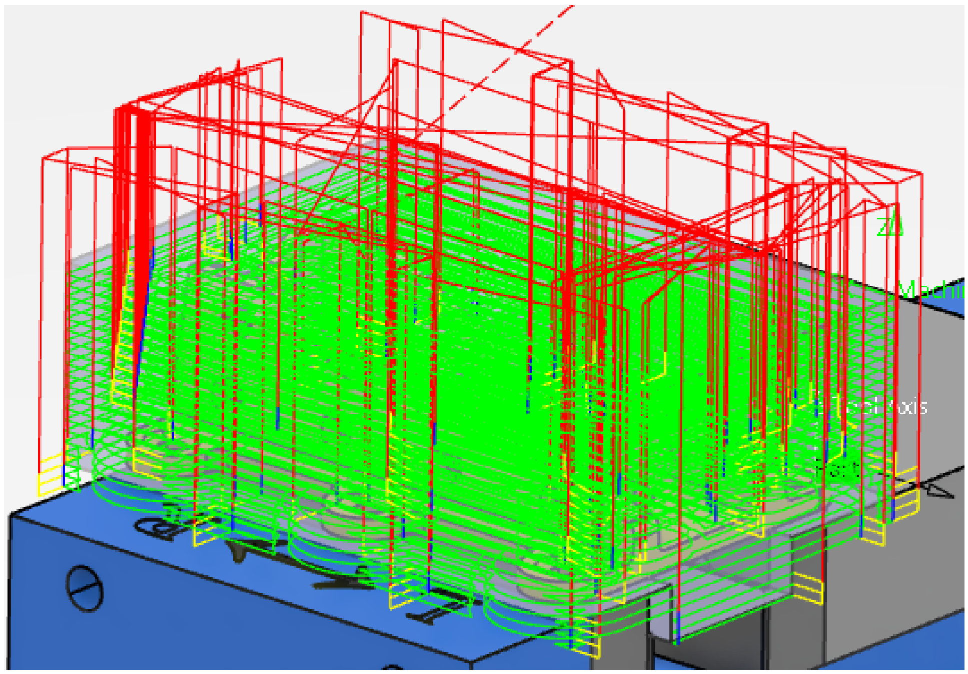
Select the *Part Definition*, then select the design part. You may have to hide the stock model or select the design part from the specification tree.

Double select in space when done. Remember, you can always define multiple parts to machine. Therefore, when done selecting the parts you want to machine, you must always double select in space to finalize the selection.

Select the Tool parameters tab and define the *SMAC030 - T0 End Mill D 1.0 x 0.25 x 6.0 L* tool for the operation. It should appear as shown.



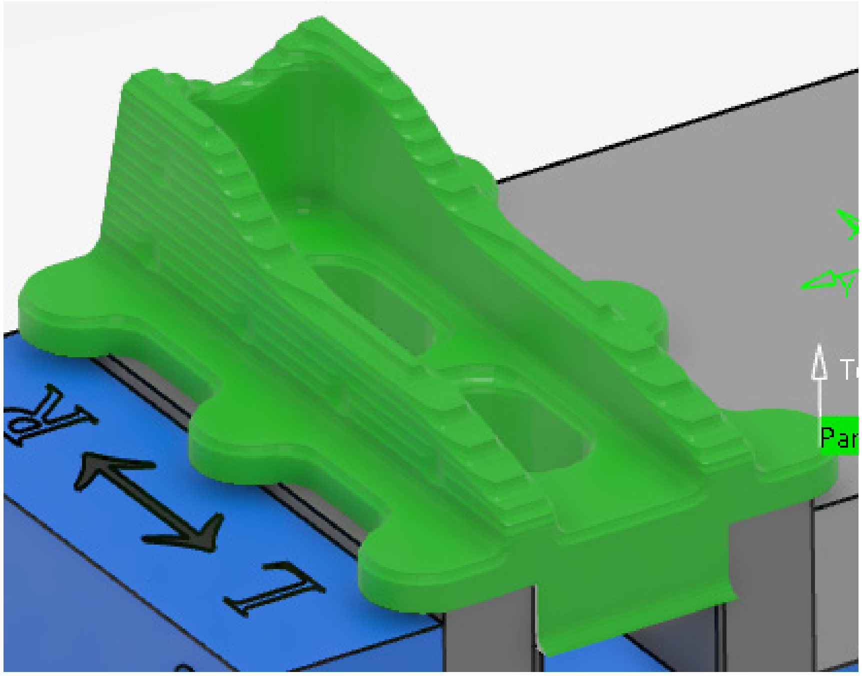
Replay the operation. Notice that by simply defining the part definition, you have given the roughing operation enough information to machine the part. As you can see, the roughing operation is designed to take a piece of stock material and convert the entire solid into chips until it reaches the design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



Change the *Offset on part* 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.

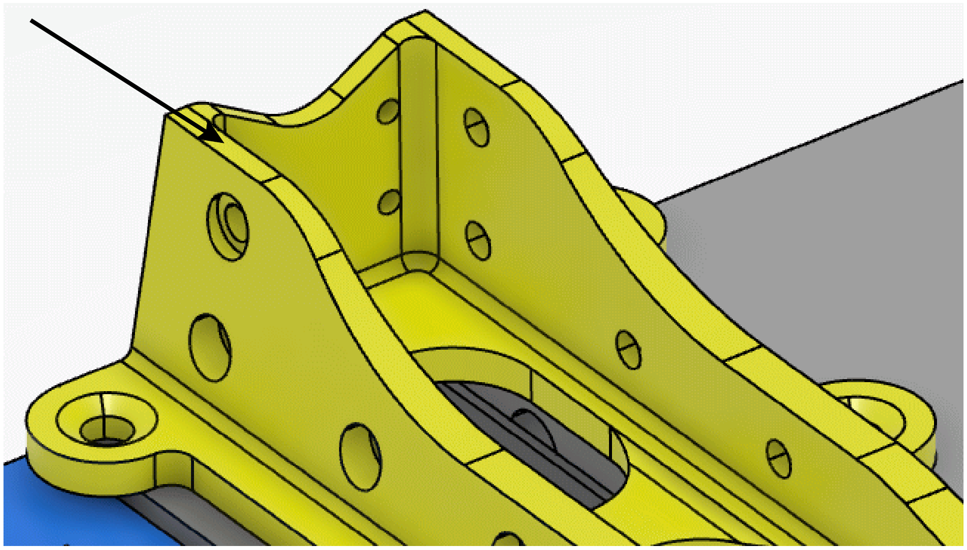
Since the rough stock is defined in the Part Operation, it is unnecessary to define it here. If you did not define the stock material, or are using a separate CGR file as the stock, then you would want to select the *Rough Stock* sensitive area, and select the appropriate stock material.

Watch the replay with 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.

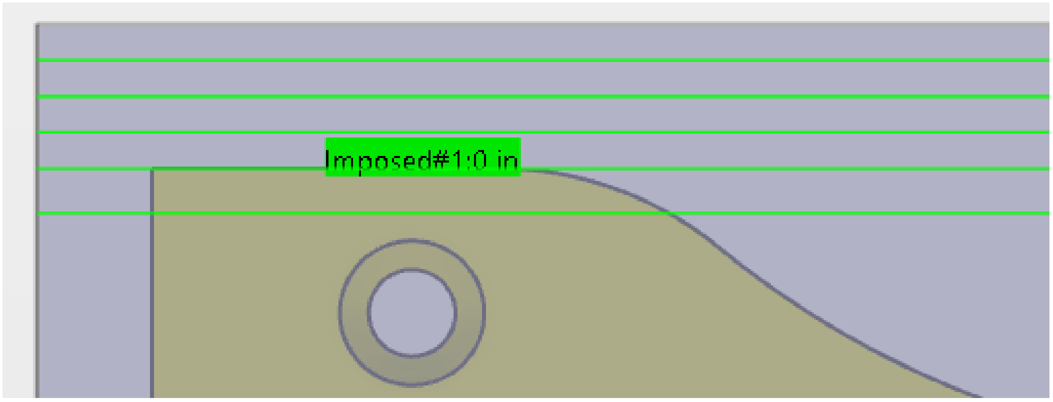


Switch to the geometry tab. You are going to work with the various *Tool / Rough Stock* options.

Select the *Imposed* plane and the plane 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.



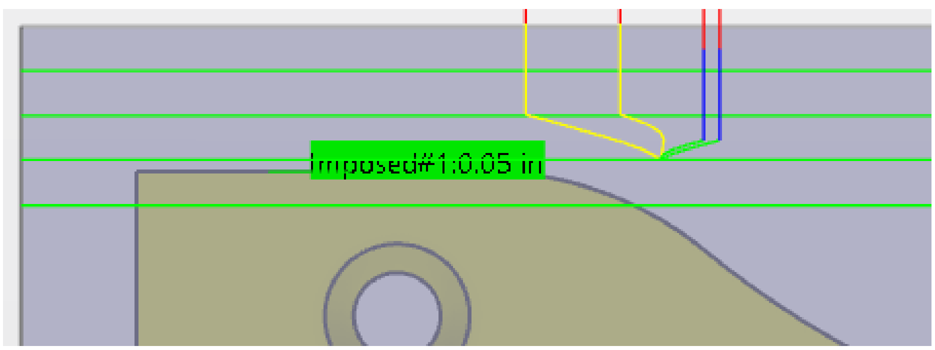
Replay the operation. There is now a tool path at the same level as the imposed plane. If you look close, you should notice that the tool does not actually cut the top of the part.



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. 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.



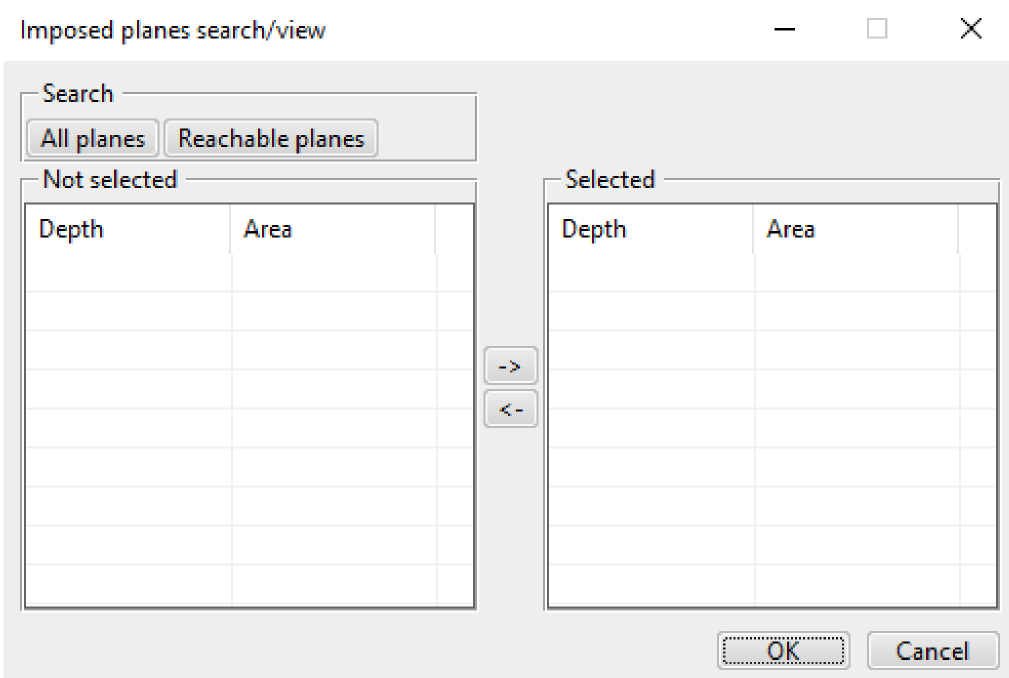
Even though you have added a 0.050in offset, the tool still does not cut at the imposed plane location. Until you make the imposed plane offset equal the *Offset on part* the tool will not cut at the imposed plane.

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.

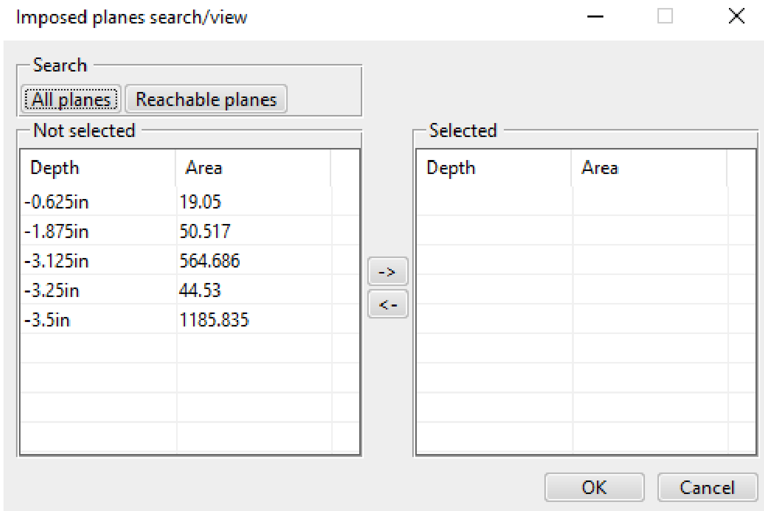
With the third mouse button, select on the *Imposed* plane and select *Remove*. This will remove all the imposed planes.

With the third mouse button, select on the *Imposed* plane again and select *Search/View...* This will display the *Imposed planes search/view* window.

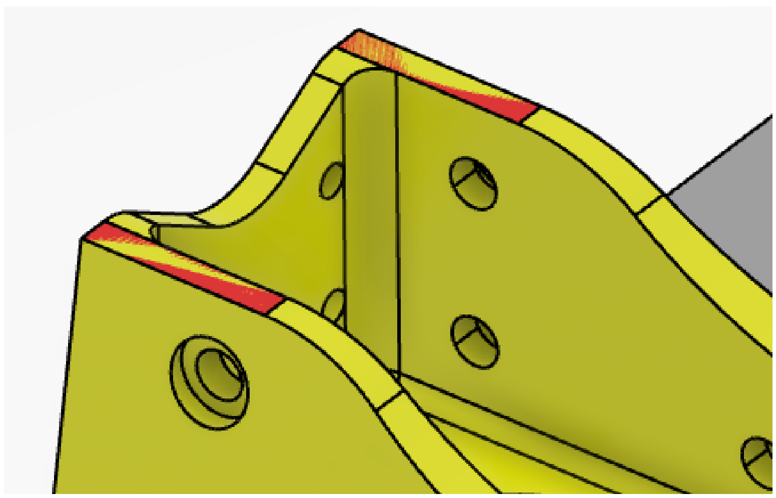


The *Imposed planes search/view* window will seek out all horizontal planes on the defined finished part and display them in the *Not Selected* frame. You can then decide which planes need to be imposed and which do not.

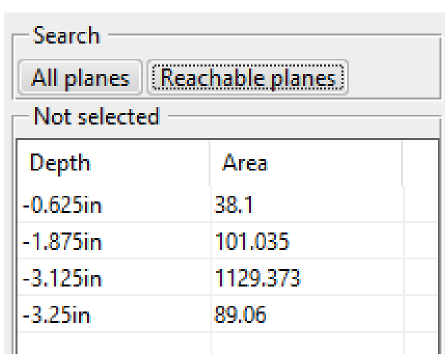
Select *All planes* from the *Imposed planes search/view* window. This will display all of the horizontal planes on the finished part.



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.

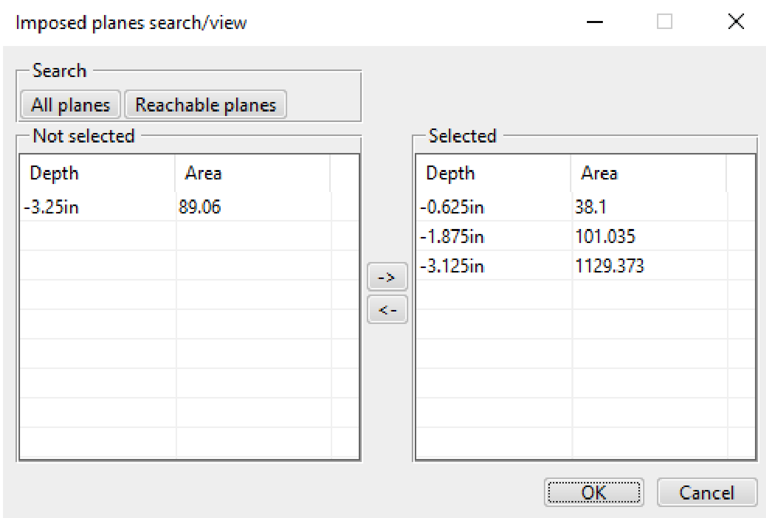


Select the *Reachable planes* button in the *Search* frame of the *Imposed planes search/view* window. This time, the list is reduced to the horizontal areas that can be reached by the tool. In this case, the 0.5in plane was removed. If other non-reachable planes existed, they would have been removed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



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

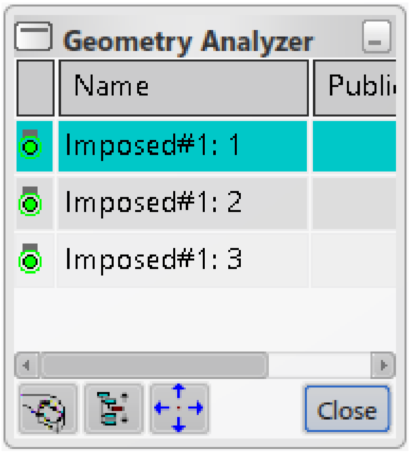
Move the -1.875in and -3.125in depths into the selected field as well. This will cause the tool to machine all three horizontal planes.



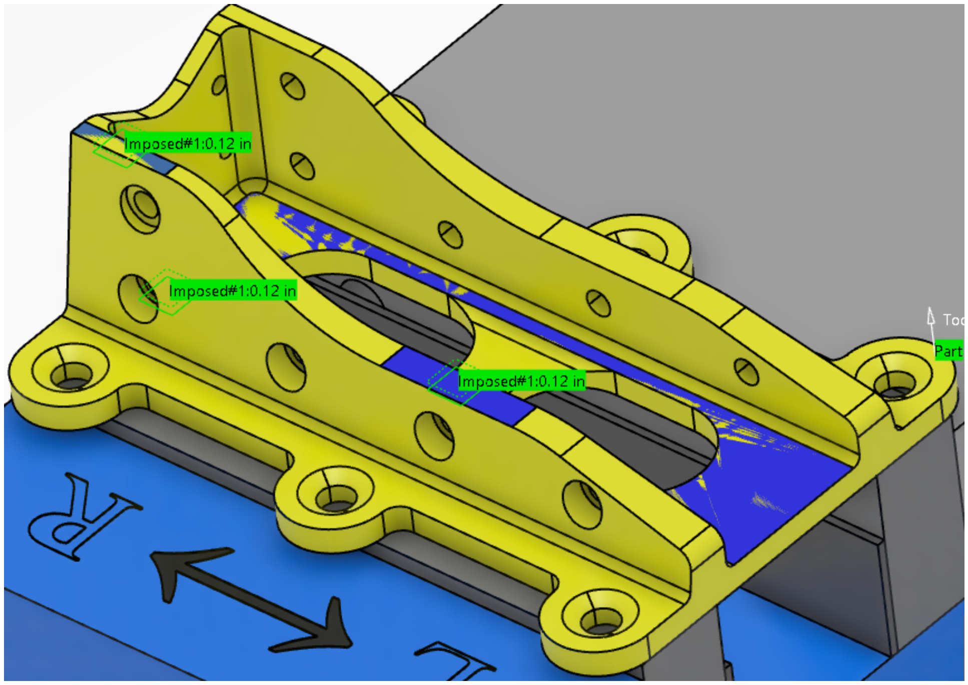
Select *OK*. This will make all the planes imposed. They should be called out in the display.

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

With the third mouse button, select on the *Imposed* plane definition, and select *Analyze*. This will display the *Geometry Analyzer* window.



Select the *Imposed#1: 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#1: 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.

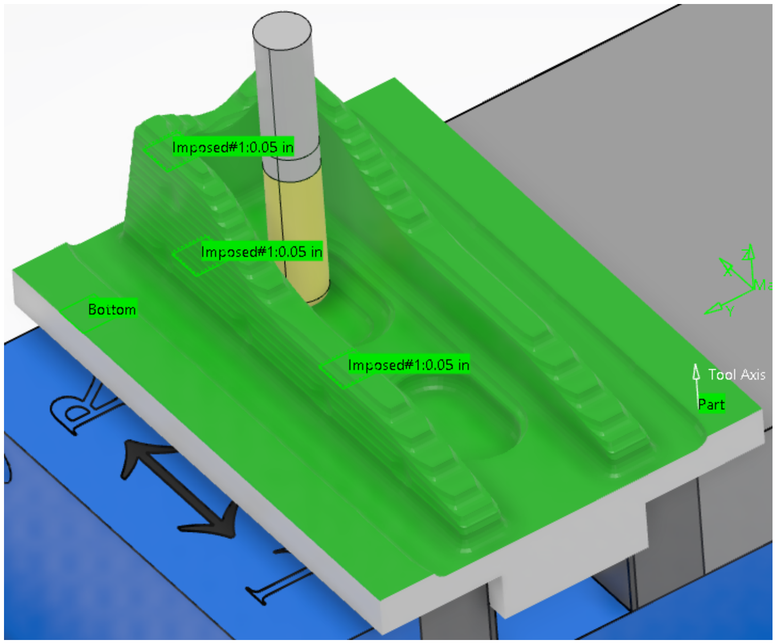
Select *Close*. It is not always necessary to ensure the selection of the surfaces, but sometimes, it is nice to know what was defined.

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.

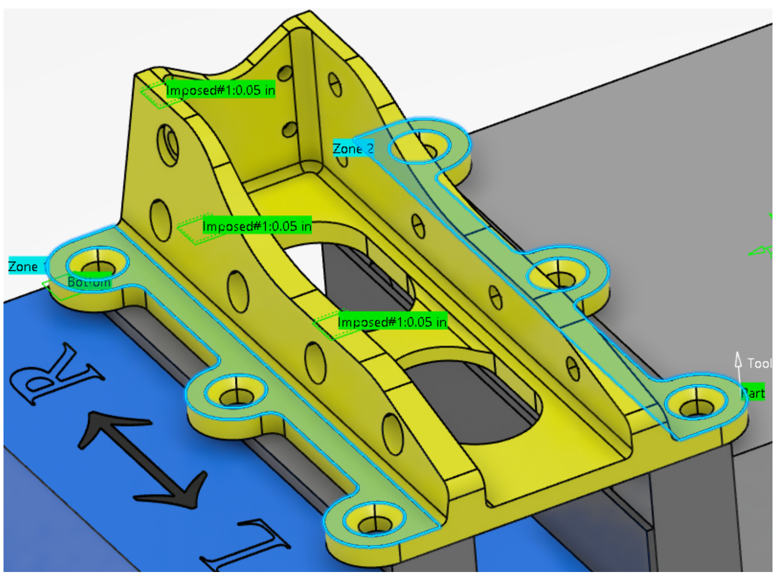
Change the offset on the *Imposed* plane to 0.050in.

Watch the replay 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.



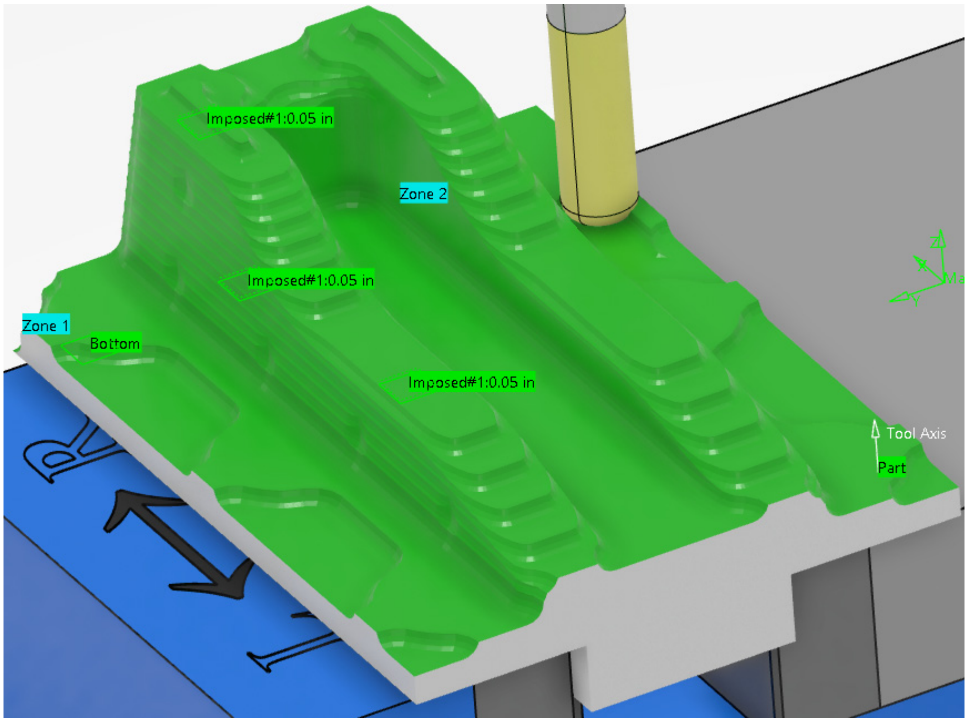
Select the *Zone order* from the geometry definition area. The zone order allows you to define pockets or areas in the order you want them machined.

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.

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



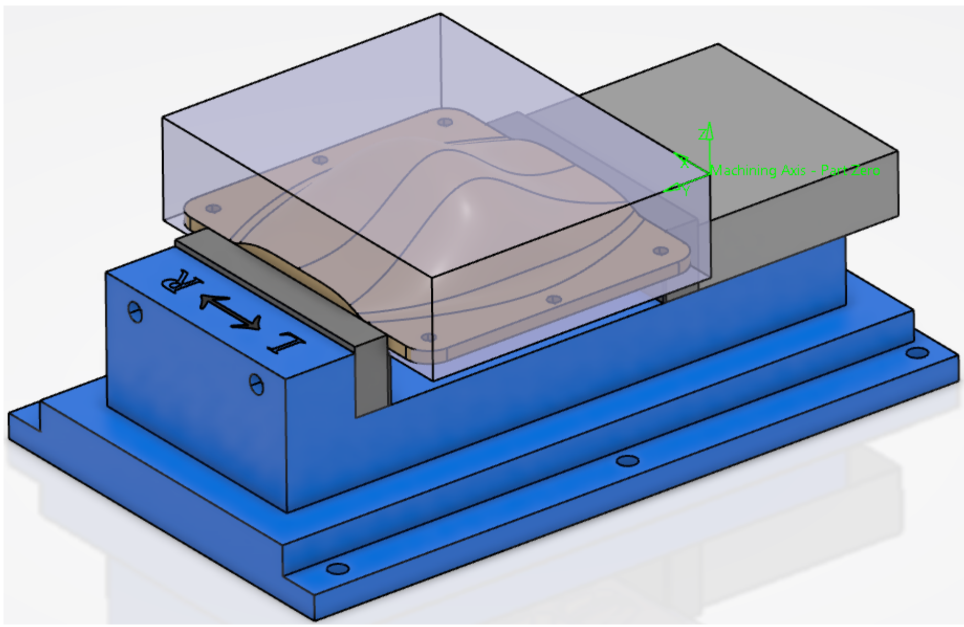
Generally, zone orders are used when you want to control the order of pocketing done in a large bulkhead. By default, in a large bulkhead with many pockets, the system will start at one end and work across the part, roughing in the most efficient manner. Although efficient, the part will gain considerable 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.

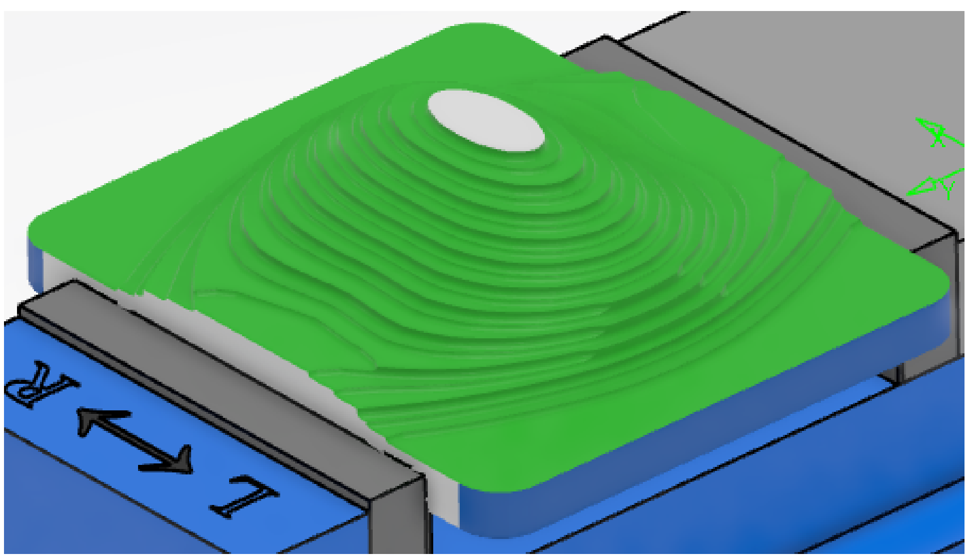
Sweeping

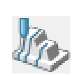
The sweeping operation will be the first finishing operation that you will investigate. The sweeping operation is very similar to the sweep roughing, with the exception of the ability to finely control the side steps to ensure a smooth, quality surface. The sweeping operation works best on non-vertical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

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



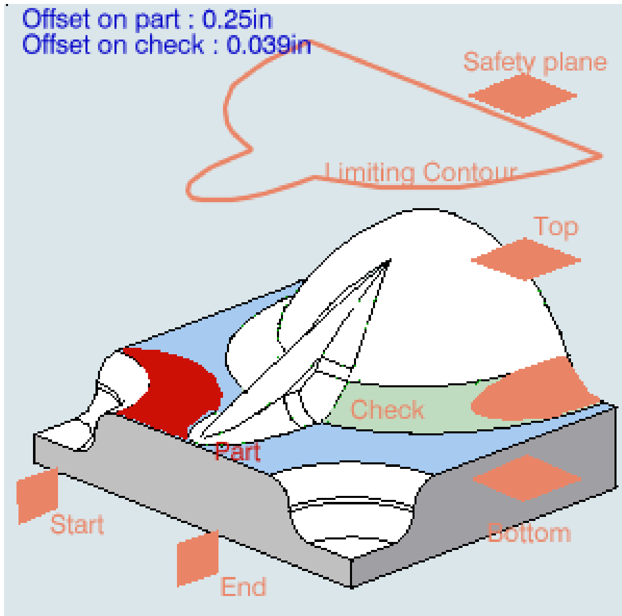
Take a quick moment to review the operations that already exist. In this state, you are ready to begin finishing the surface of this part.



Start a Sweeping operation after the profile contouring operation.  Sweeping operations are going to be defined in a similar manner to sweep roughing operations.

Geometry Tab

Sweeping is very similar to the sweep roughing operation. Take a quick look at the sweeping operation sensitive area.



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

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

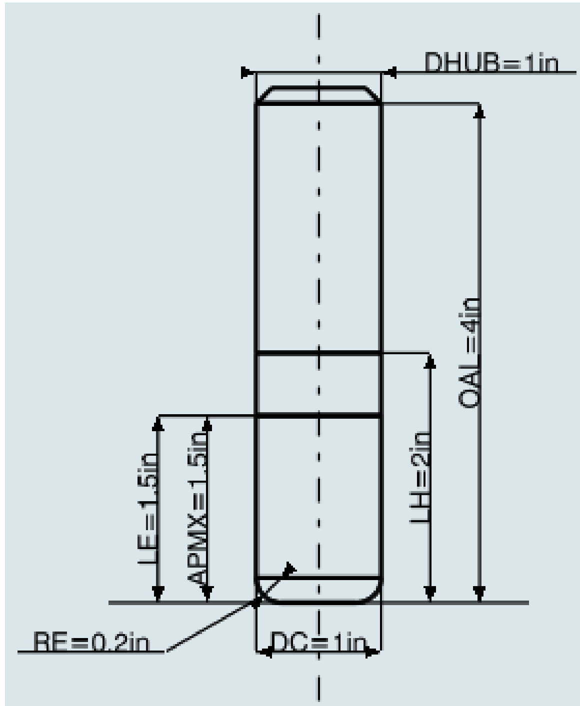
End Defines the ending of the tool paths. The end plane must be parallel to the tool paths.

Select the *Part* definition, and select the finished part. Just like with the other operations, by selecting the part definition, you will be defining the entire part body.

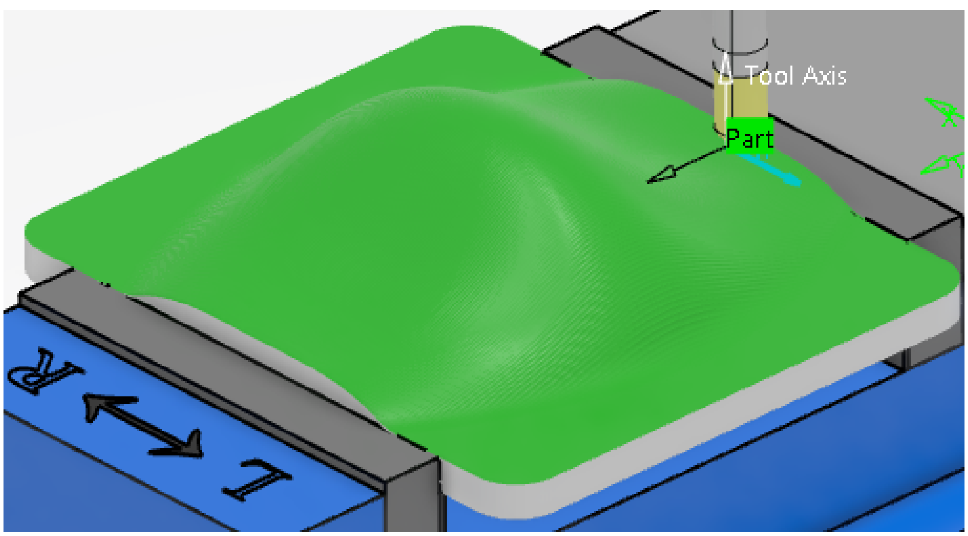
Make sure the *Offset on part* is set to 0.0in. This will make sure you machine the final part.

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 Tool parameters tab and select the *SMAC050 - T2 End Mill D 1.0 x 0.20 x 4.0 L* tool. This will be a more appropriate cutter for sweeping along the surface.

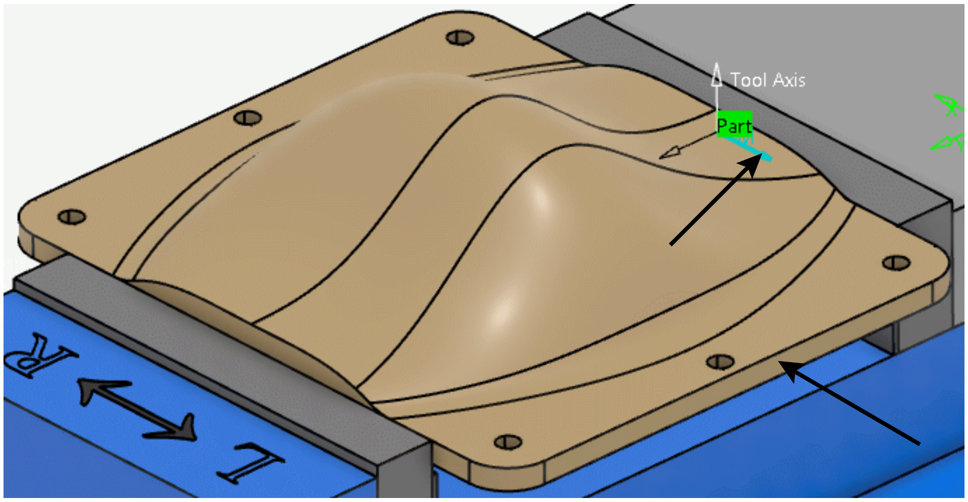


Replay the operation. The tool simply sweeps back and forth across the part creating a moderately smooth surface.



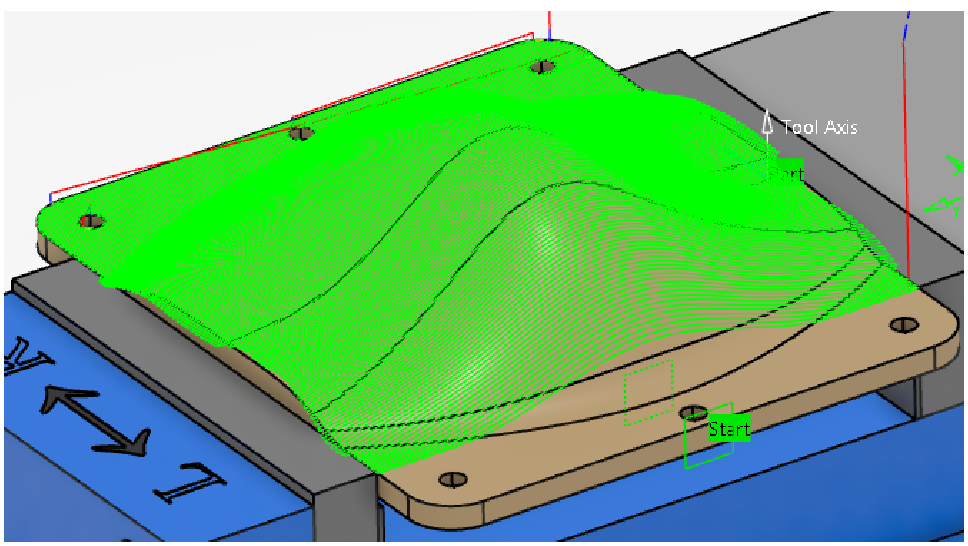
At this point, the surface quality is still rough, however, the machining parameters allow you to fine tune the tool paths to ensure a quality, smooth surface. Before adjusting machining parameters, take a moment to look at some of the additional geometry options.

Switch back to the geometry tab and select the *Start* plane from the sensitive area, 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.



Invert the direction arrow above, so it is pointing in the R direction on the tool. This will change which direction the part is machined in.

With the third mouse button, select on the *Start* plane. In the contextual menu, select *Offset*. Set the offset to -2.0in. By defining an offset of -2 inches, you will notice a small dashed plane appear inside the part.

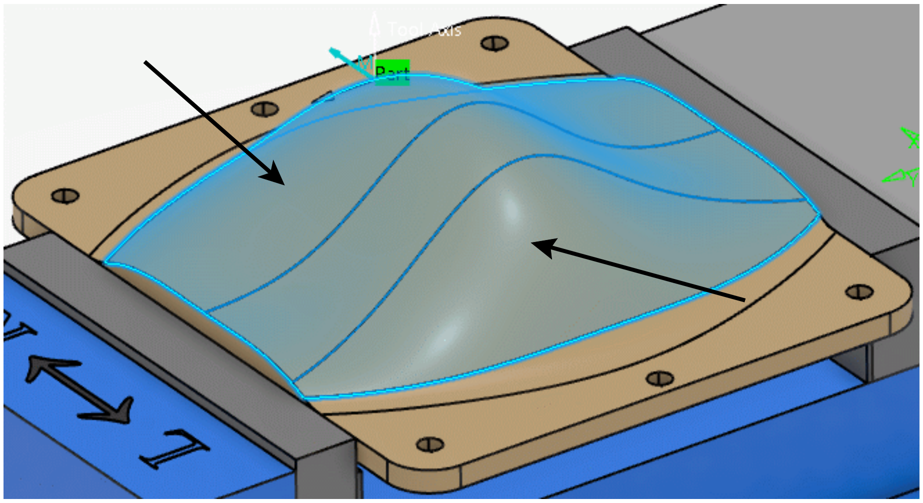


The tool path now starts at the offset value from the start plane. By defining the start and end planes, you can control how much of the surface gets machined. Unfortunately, you cannot limit the length of the tool paths. For example, you can cannot remain 2 inches away from all the boundaries with the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

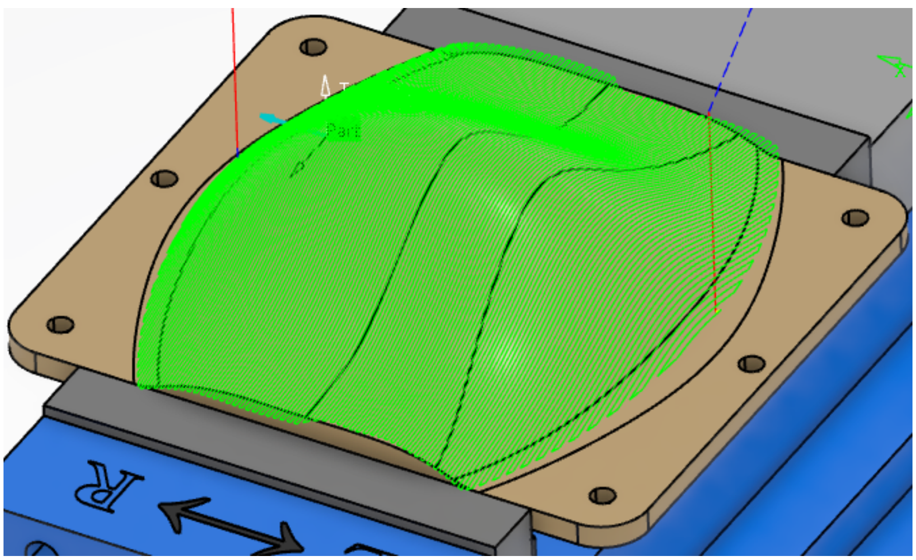
With the contextual menu, remove the start plane. This will go back to machining the entire surface.

With the third mouse button, select on the *Part* definition and select *Remove*. Then right mouse select again on the *Part* definition and select the *Select faces* option. 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.



Replay 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.



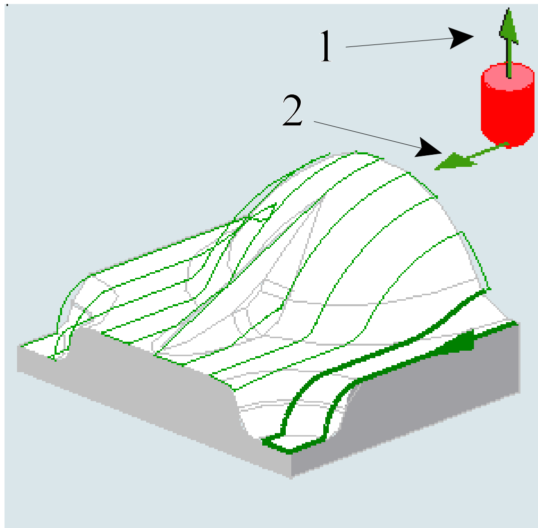
No matter the geometry of the part or faces that you want to machine, you will find that the tool paths are always straight and parallel.

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.

Machining Parameters

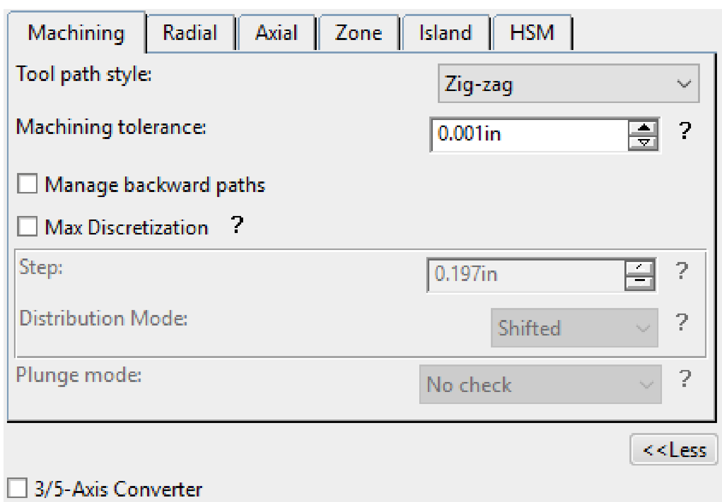
Take a quick look at the various machining parameters.



1 – *Tool Axis* Defines the tool axis direction

2 – *Machining Direction* Defines the tool path direction

Machining



Tool path style Defines the type of cut that is to be made across the part. There are three options available.

Zig-zag The tool moves back and forth across the top of the surface in a zig-zag motion

One-way next The tool moves across the surface in one direction, returning to the start of the next tool pass

One-way same The tool moves across the part, returning in the same path that it machined

Machining Tol... Defines the tolerance of the machine being used

Manage backward... Manages paths to avoid marks on the part

Max Discretization Allows you to override the step size of the machining operation

Step Defines the maximum step size allowed

Distribution Mode Defines whether the distribution of the points will be aligned or shifted with the previous tool path

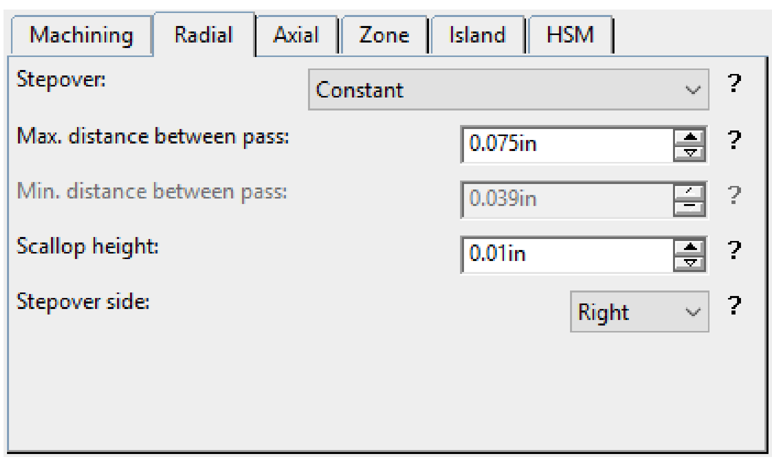
Plunge Mode Controls if the tool plunges, or dives into the material. This option is only available with the One-Way tool path styles.

No check No plunge checking is made

No plunge Tool is halted as soon as it is to plunge into the material

Same height Tool remains at the same Z-height when it is supposed to plunge

Radial



Stepover 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

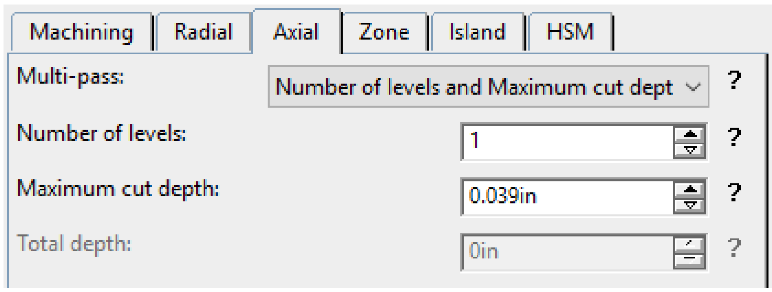
Max. distance... Defines the maximum distance stepped each radial pass

Min. distance... 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 minium step sizes

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

Axial



Multi-pass Defines how the multiple axial steps are calculated

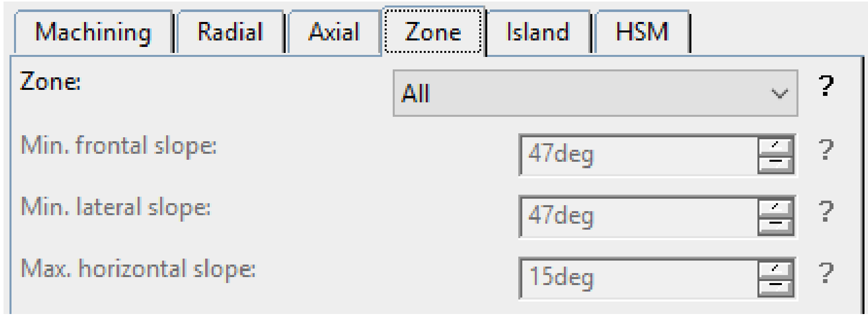
Number of levels Defines the number of axial levels

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



Zone — Controls what type of area is machined

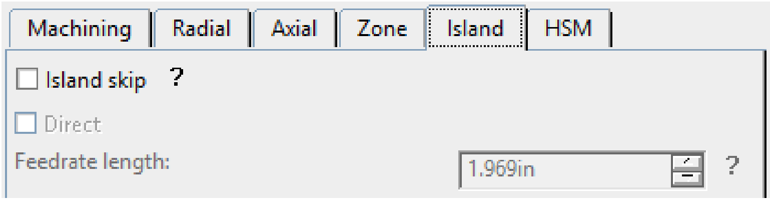
- *All* — All zones get machined
- *Frontal walls* — Only the front and back walls get machined. The front and back walls are determined by the forward motion of the tool.
- *Lateral walls* — Only machines the walls on the left and right hand side of the tool paths
- *Horizontal zones* — Only machines 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

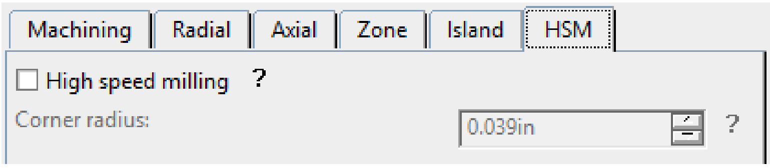
Max horizontal... — Maximum angle that will be considered a horizontal plane

Island



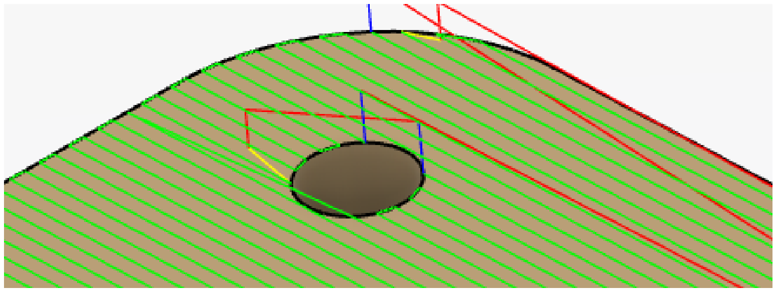
Island skip	Indicates that islands, or gaps in the surface should be jumped over, not gone around
Direct	Causes the tool to move directly over the island without a retract and approach motion and is available only when *Island skip* is turned on
Feedrate length	Defines the size of a gap or island that will cause a direct island skip. Any hole or pocket larger than this length will generate a retract and approach motion.

HSM



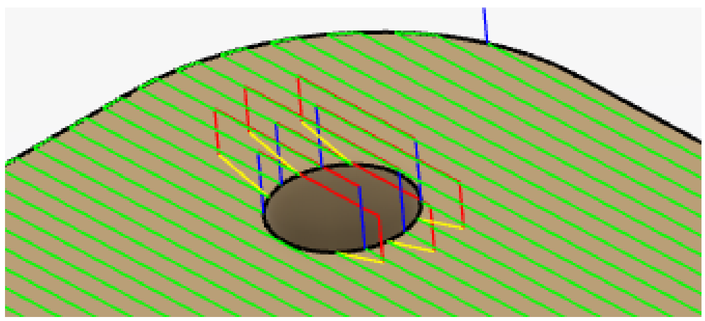
High speed milling	Activates high speed milling
Corner radius	Defines a corner radius for high speed milling

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



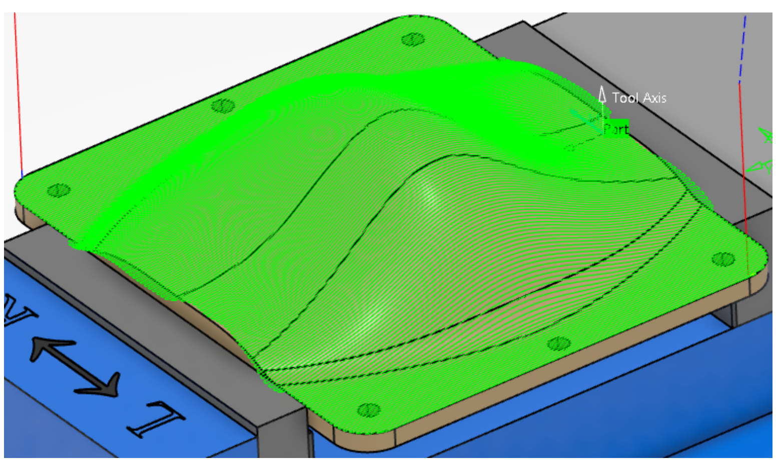
This results in a bad tool path motion and will be fixed first.

Switch to the *Island* 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.

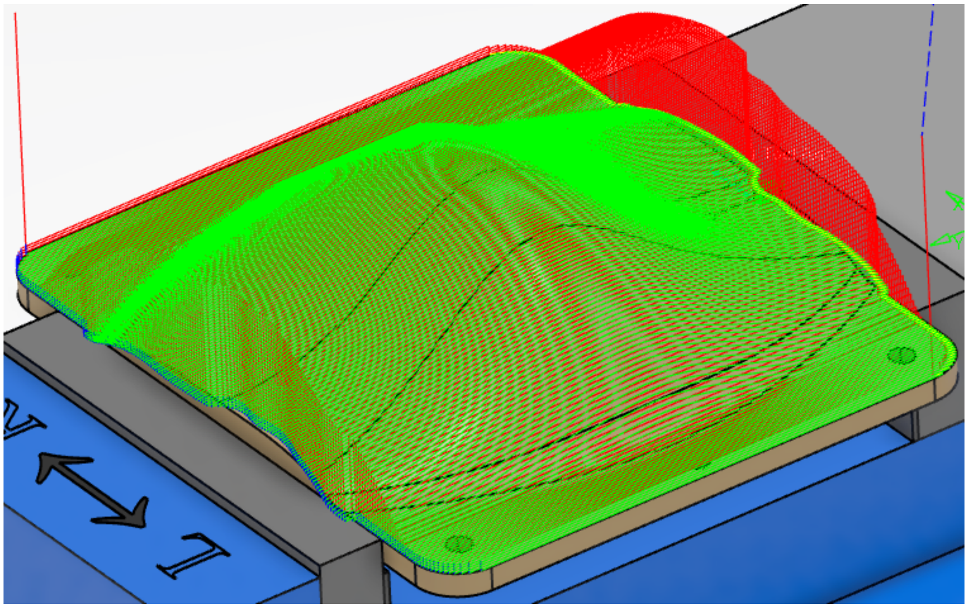


Turn on the *Direct* option under the *Island* tab. Once you turn on direct, you have the ability to define the *Feedrate length*. If you remember from earlier, the *Feedrate length* defines the size of a gap the tool will cross in a direct motion.

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.

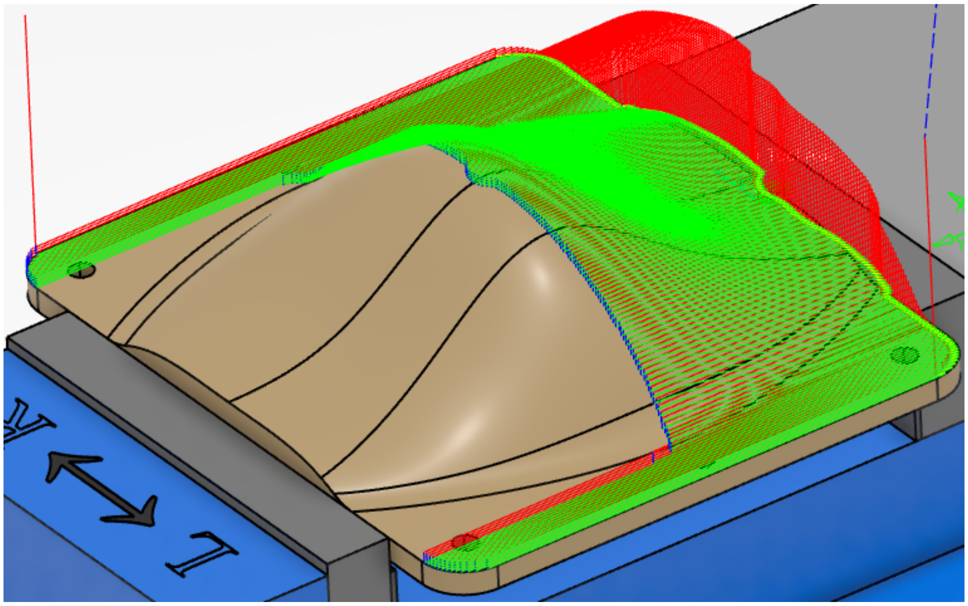


Change the *Tool path style* to *One-way next*. This will simply cause the tool to start on one side, then machine only in one direction.



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.

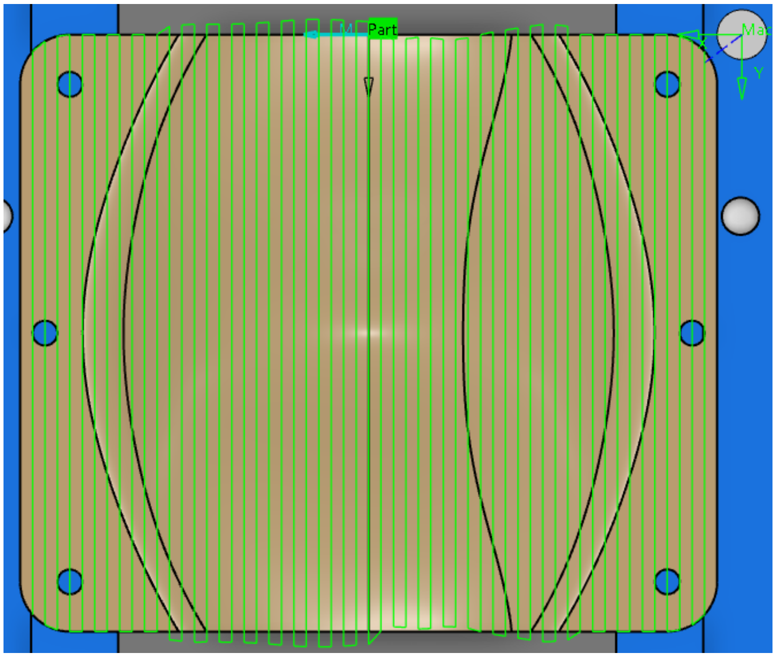


Not allowing the tool to plunge is good if you want to machine upwards along the part surface. By turning off the plunge, the tool will stop at the highest point, allowing you to go back and machine the opposite side without using limiting contours.

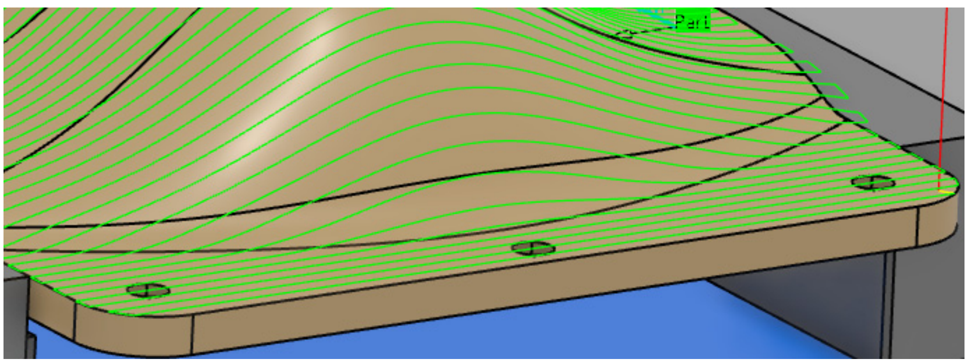
Change the plunge back to *No check*. This will go ahead and allow the tool to machine across the entire surface.

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 *Max. distance between pass* to 0.25in. This sets all the steps to be a constant, even 0.25in.

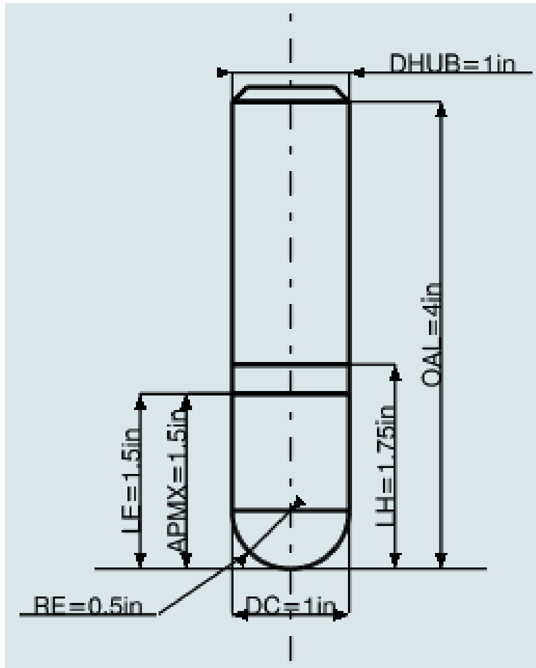


Even though the steps are set to a constant 0.25in, if you look at the tool paths from another angle, you can see that the steps are only constant along the tool axis.



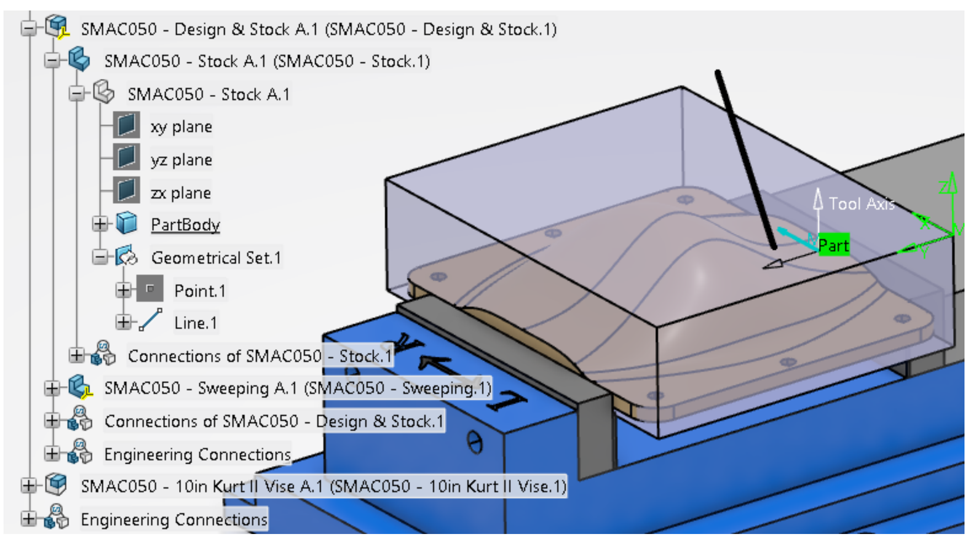
This can be impacted by modifying the tool axis. First, you will want to get a ball nose tool which will allow for better machining with a modified tool axis.

Switch to the Tool parameters tab and select the *SMAC050 - T3 End Mill BALL NOSE D 1.0 x 4.0 L* tool. This will convert the 1in bull nose end mill to a 1in ball nose end mill.

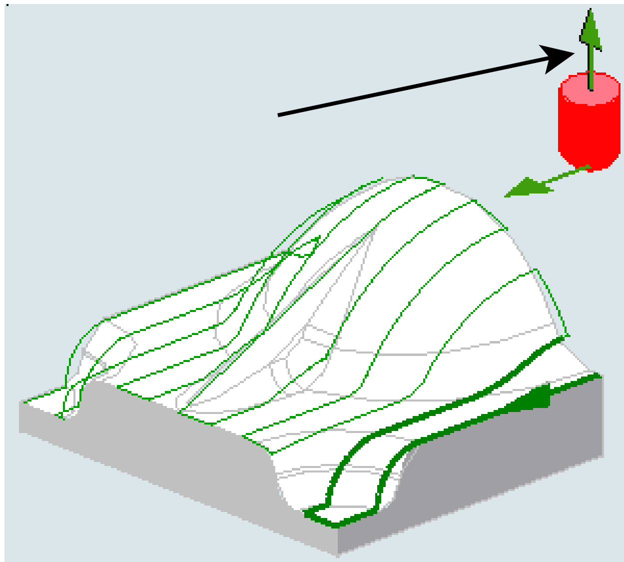


Switch back to the Stategy parameters tab.

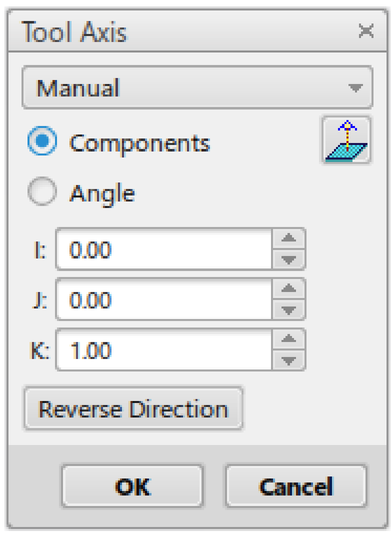
Unhide *Line.1* from the stock material in the specification tree. This line will serve as an alternative view direction.



Select the tool axis definition as shown.

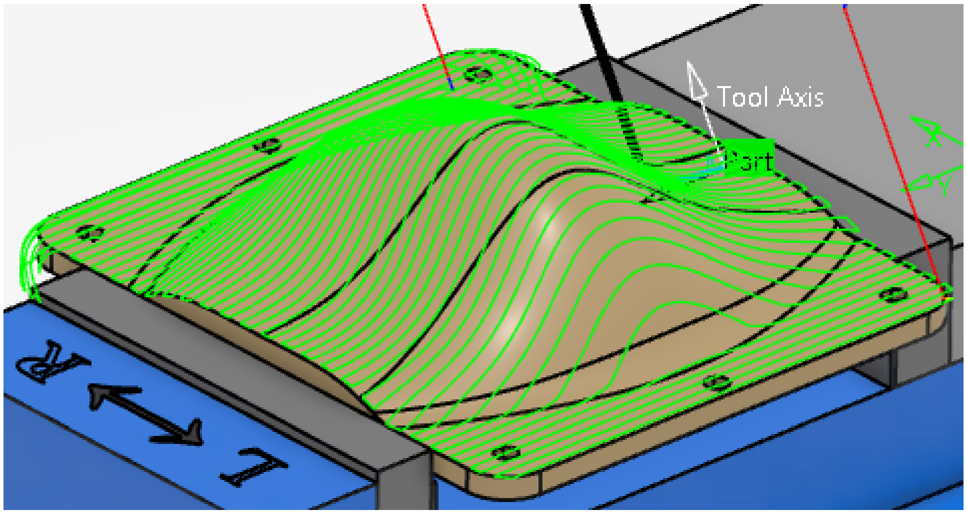


The Tool Axis window appears.

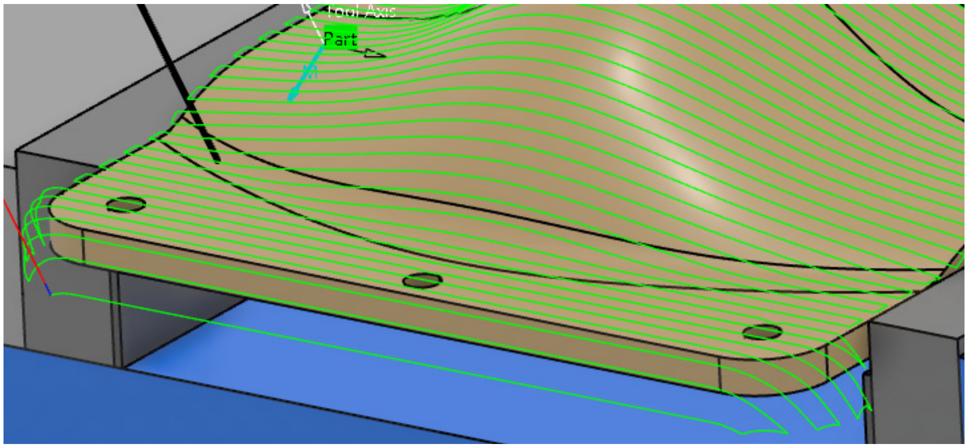


Change the drop down menu to Selection, then select the *Line.1* that you un-hid from the stock material. Select *OK* to the *Tool Axis* window. This will change the direction that the tool paths are computed.

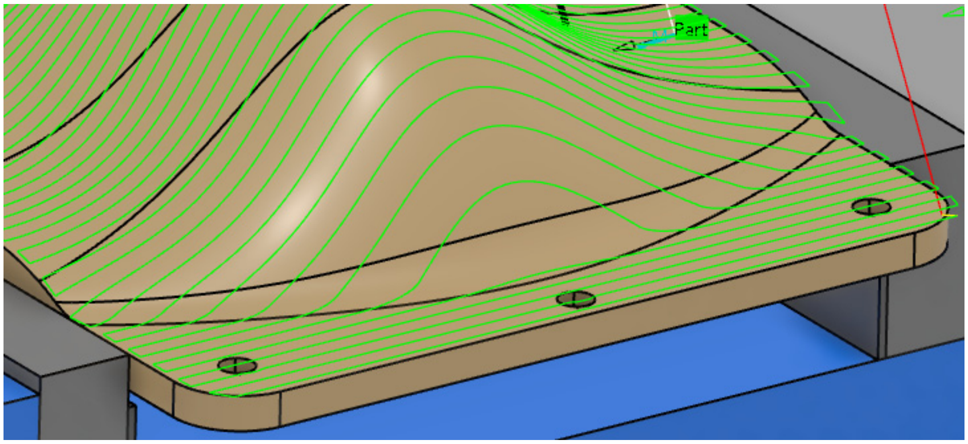
Replay the tool paths.



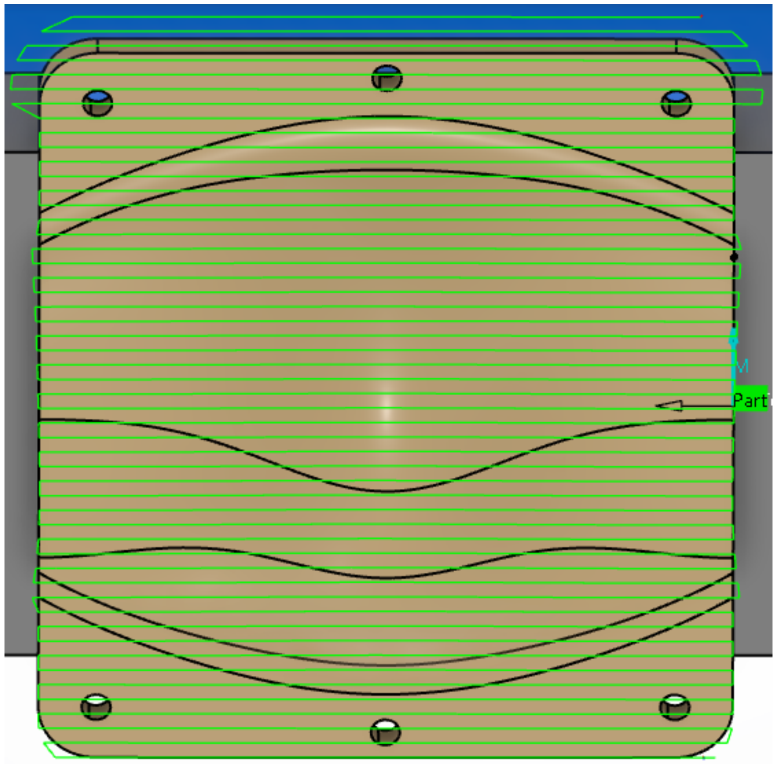
The tool paths along the flat surface are nearly the same as what you had before. Notice this time though, the tool paths extend down the side of the part and are more even against the side of the part.



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.



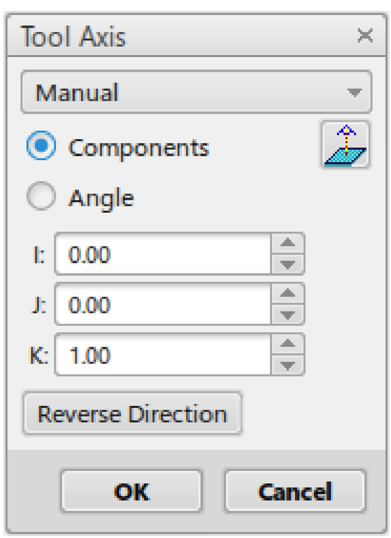
This is because when you look down along the line, you can see the side of the part, whereas, when you look down the axis, you cannot see the sides of the part.



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.

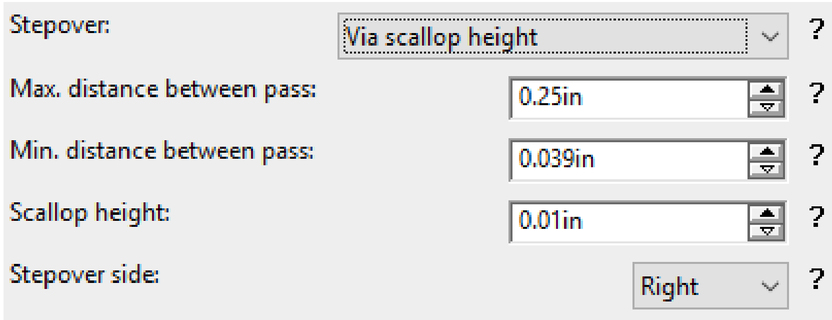
Unfortunately, if you have a surface that has a larger range of normal directions, changing the tool axis will have undesired results.

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



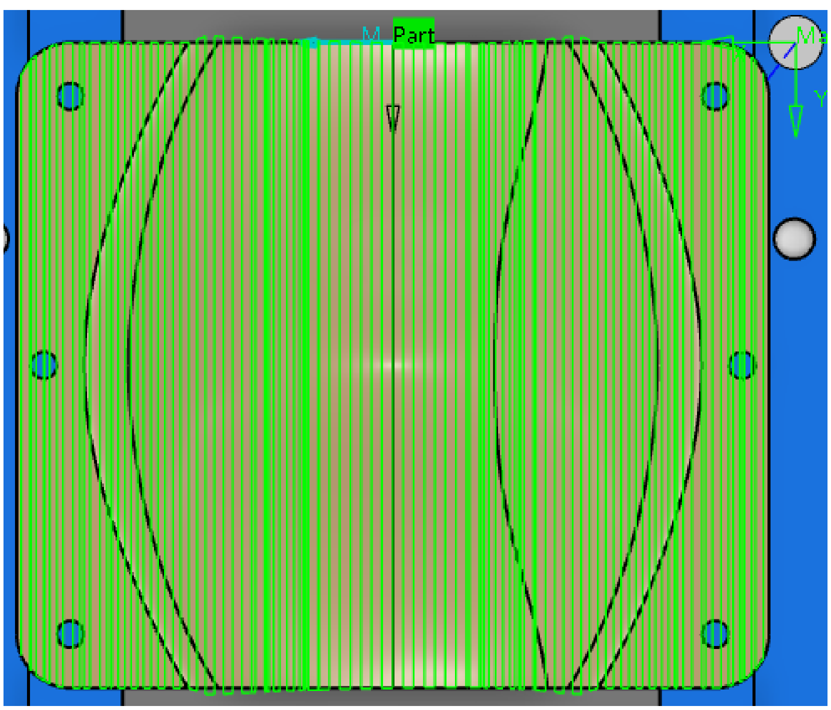
Change the *Stepover* to *Via scallop height*. This will activate the *Min. distance between pass* option.

Now you have all three distance fields available to you.

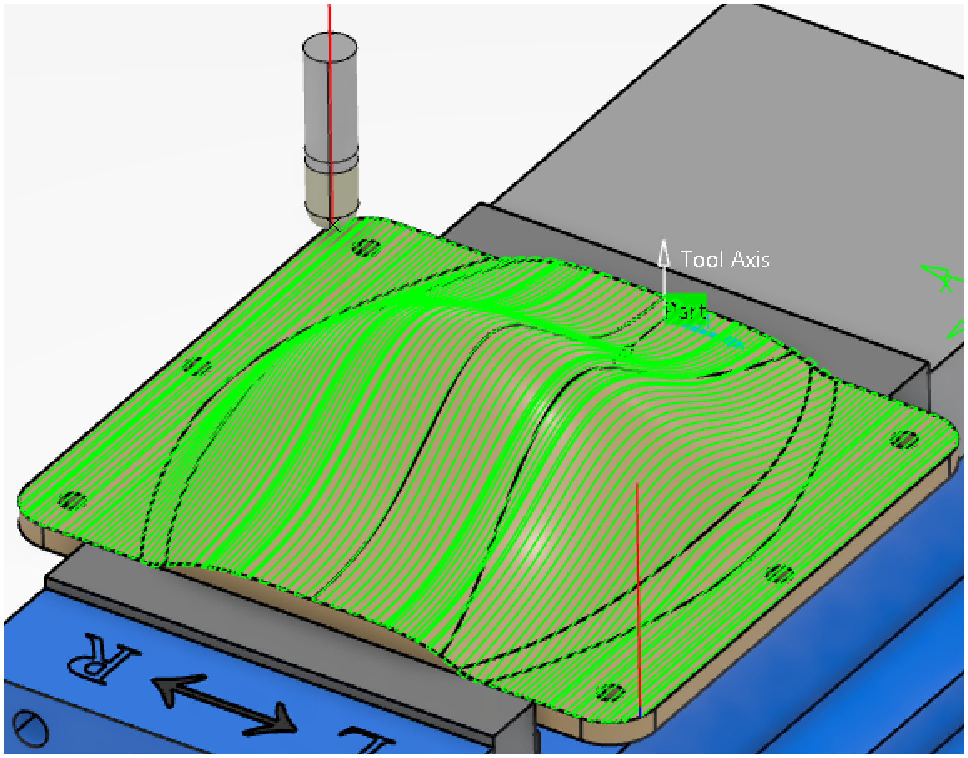


There is one rule that you must follow for the *Via scallop height* to work properly. The scallop height distance should be between the minimum and maximum distance between passes. If you have a minimum step of 0.125in and a scallop height of 0.005in, the tool will always make a step of 0.125in because 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.

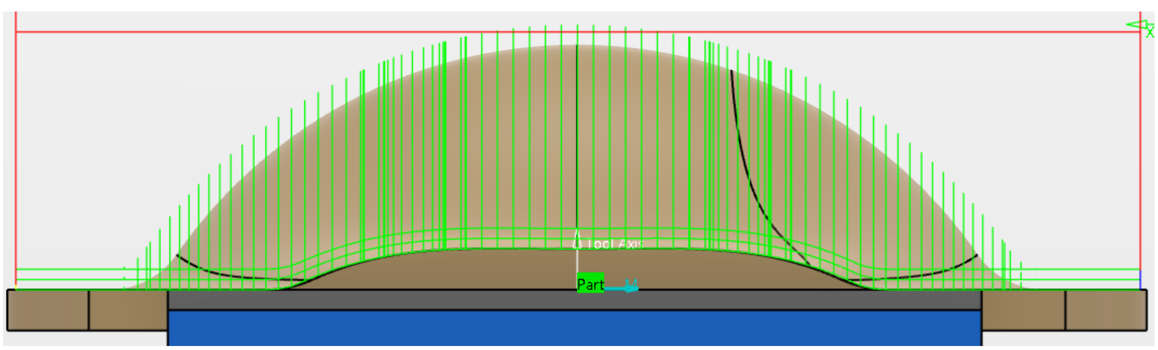
Set the minimum distance to 0.005in, with a scallop height of 0.010in. This will cause the tool to allow for varying sized steps as it goes across the part. The tool will do its best to maintain a scallop height of 0.010in, but will guarantee that the step size will never go under 0.005in or above 0.25in.



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.



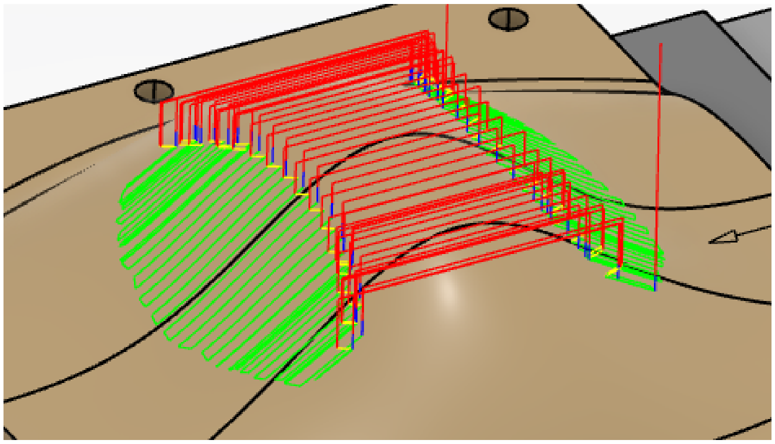
Under the *Axial* tab, set the *Number of levels* to 3, with a *Maximum cut depth* at 0.125in. This will indicate that you wish to have three sets of tool paths that are 0.125in apart. Take a look at the resulting tool paths, you will notice that it is a vertical offset from the bottom set of tool paths.



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.

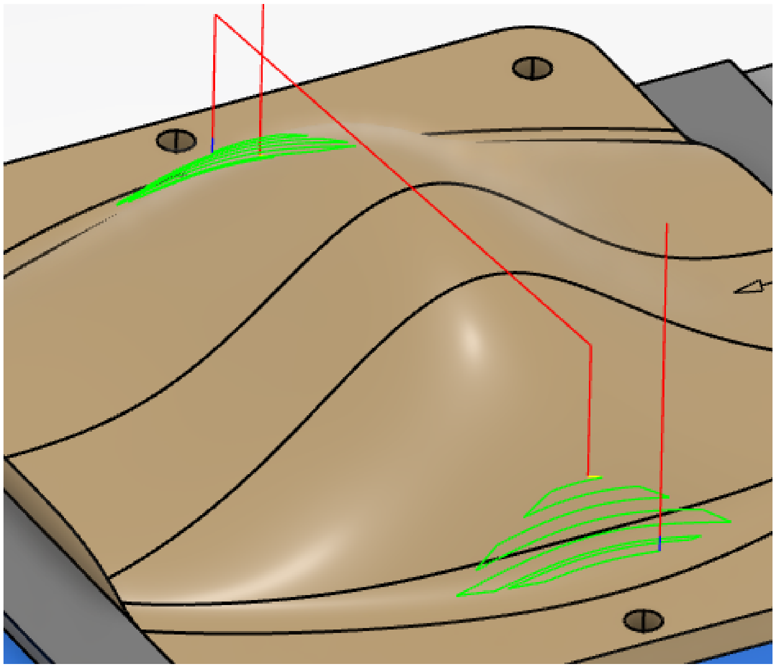
Set the *Number of levels* back to 1. Since the multi-pass option is not very good for performing multiple axial steps, you will set it back to one for now.

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 machine. Basically, anything within 60deg (90°-30°) of vertical, and on the frontal plane gets machined. Everything else is ignored.

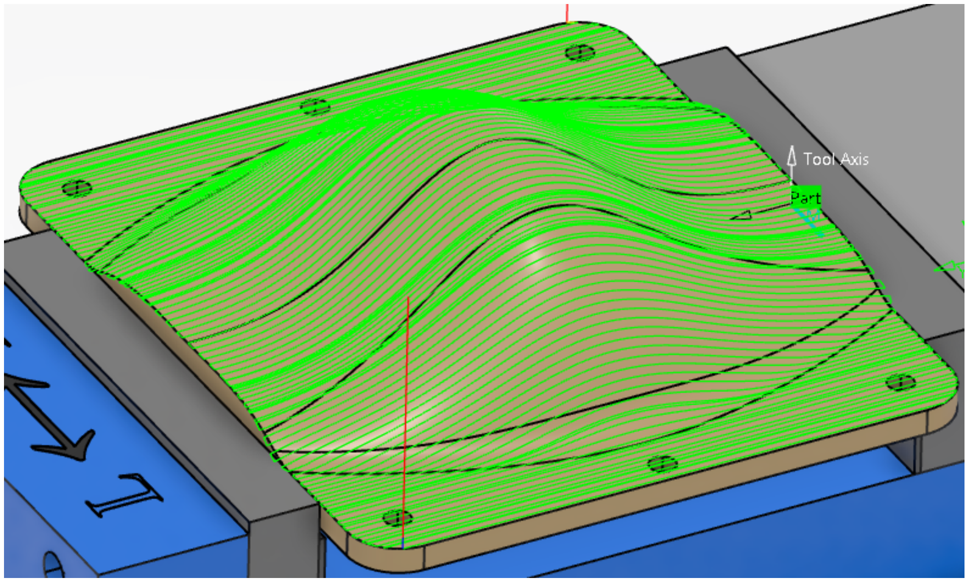
Change the *Zone* to *Lateral Walls*, with a minimum angle of 20deg. This time the walls on the sides get machined.



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 Macro parameters 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.

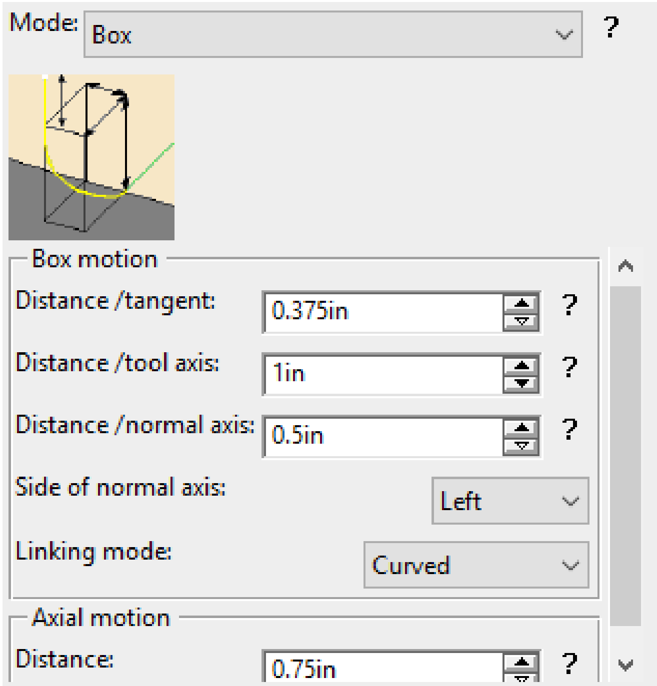
Macro Management

Macro	Name	Mode
Approach	Approach.1	Back
Retract	Retract.1	Along tool axis
Clearance	Clearance.1	Optimized
Linking Retract	Linking.1	Defined by Approac...
Linking Approach	Linking.1	Defined by Approac...
Between passes	Between passes.1	None
Between passes Link	Between passes.1	Defined by Approach

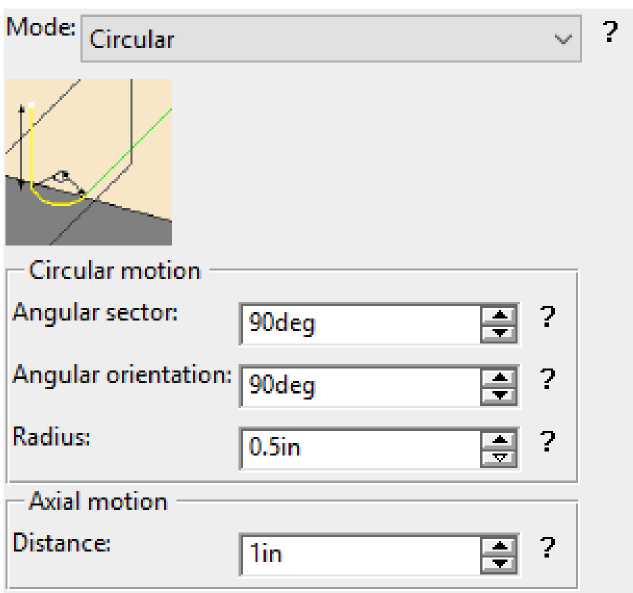
Select the *Approach* macro and change the *Mode* to *Box*. This will create the imaginary box around the tool path to create a nice, S-Curve shape into the part.

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.

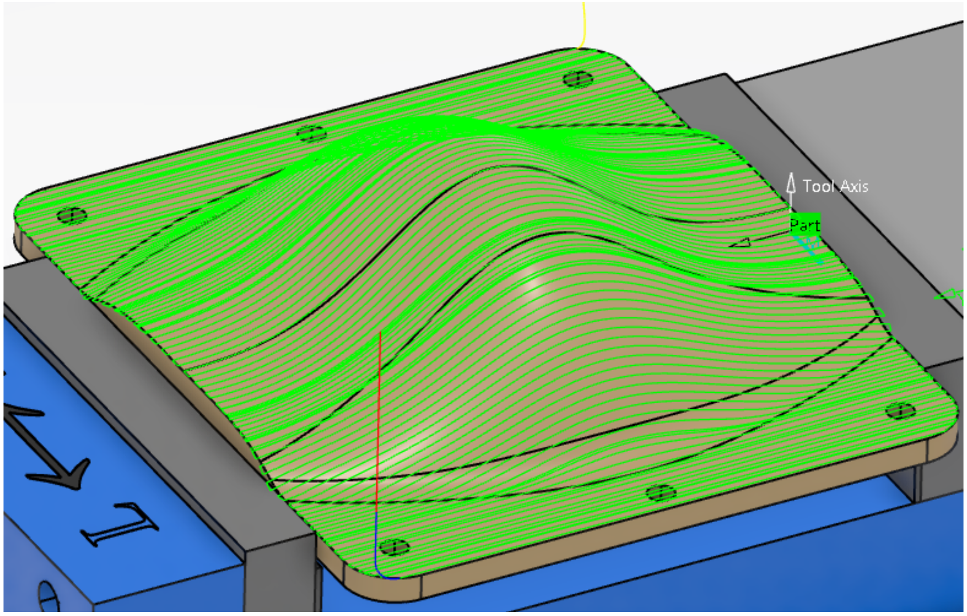
Change the box macro to have the following parameters. This will make the box have a 1.0in height, 0.375in width, and 0.5in depth.



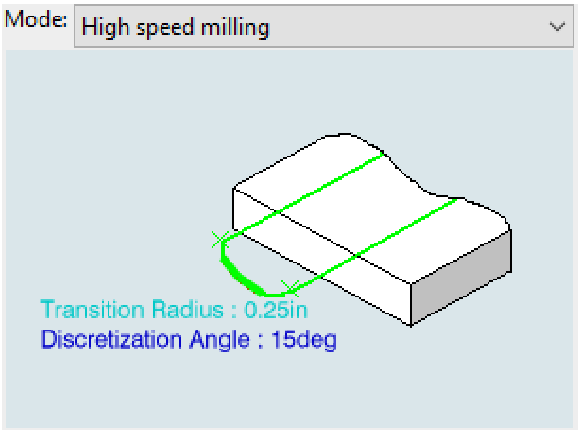
Change the *Retract* macro to a circular macro, with the following parameters. As you might have noticed, using the pre-defined macros allow for a quick definition of the macros. Many times it is advantageous to use the pre-defined macros instead of trying to custom build macros all the time.



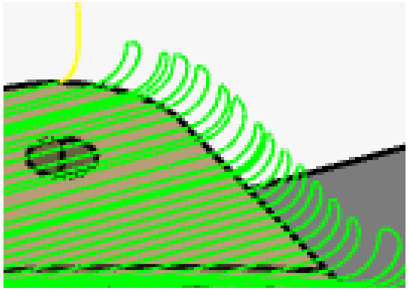
Replay the operation. It was mentioned earlier that the high speed milling options were primarily defined in the macros. Take a moment to look at the HSM options that you can use to help get HSM movements.



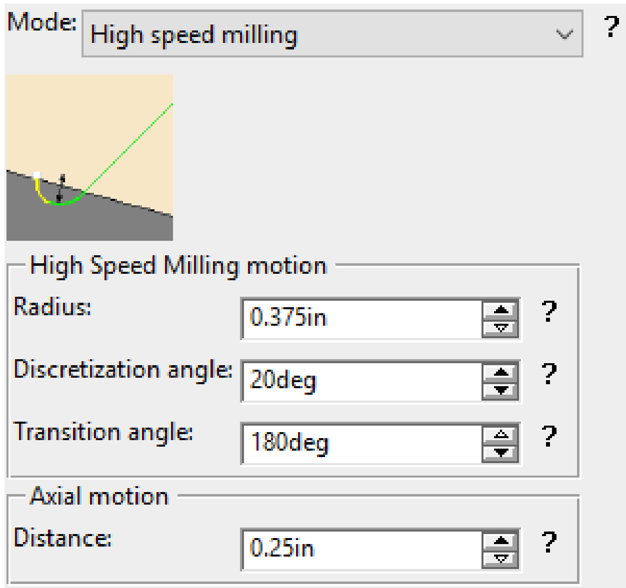
Set the *Between passes link* macro to *High speed milling* with a 0.25in *Transition Radius*. This will indicate that you wish to have a high speed milling motion between the passes with a 0.25in radius off the part. If the *Between passes* macro is not defined, there may be some situations where not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



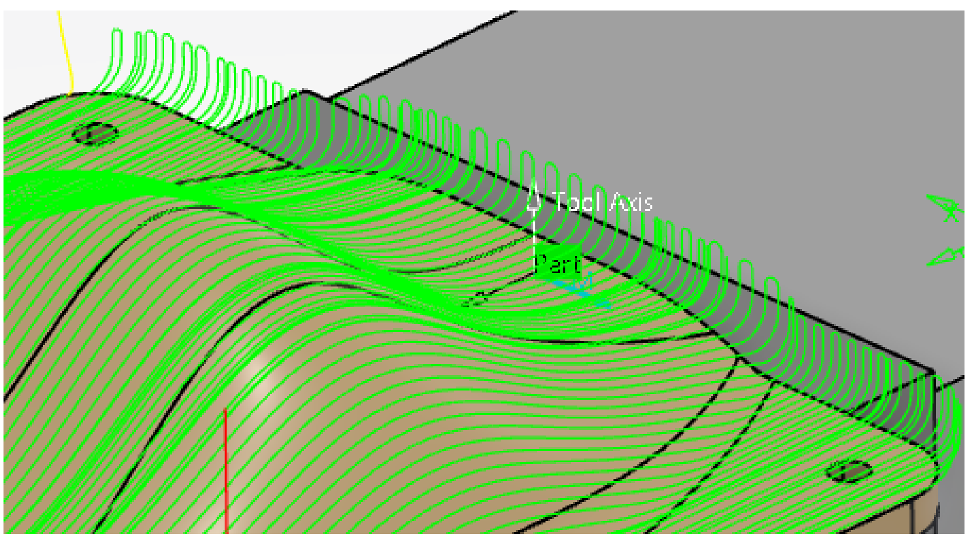
Replay the tool paths.



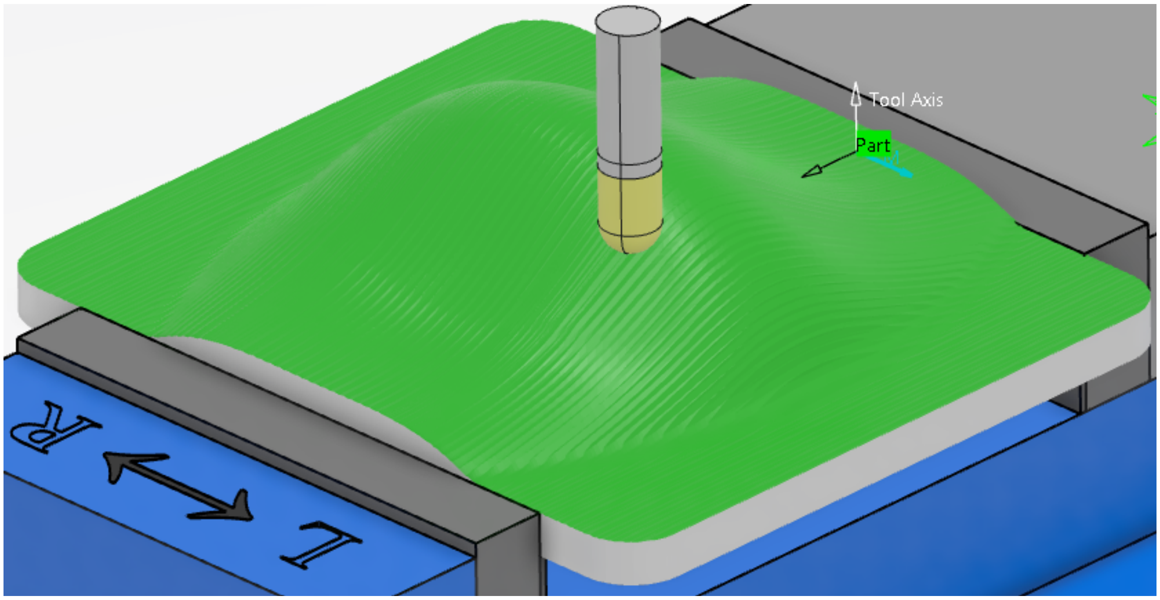
Set the *Between passes* macro to *High speed milling* with the following parameters. This is will define a 0.375in radius at the end of each pass, then move vertically 0.25in before circling around and returning to the part.



Replay 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.