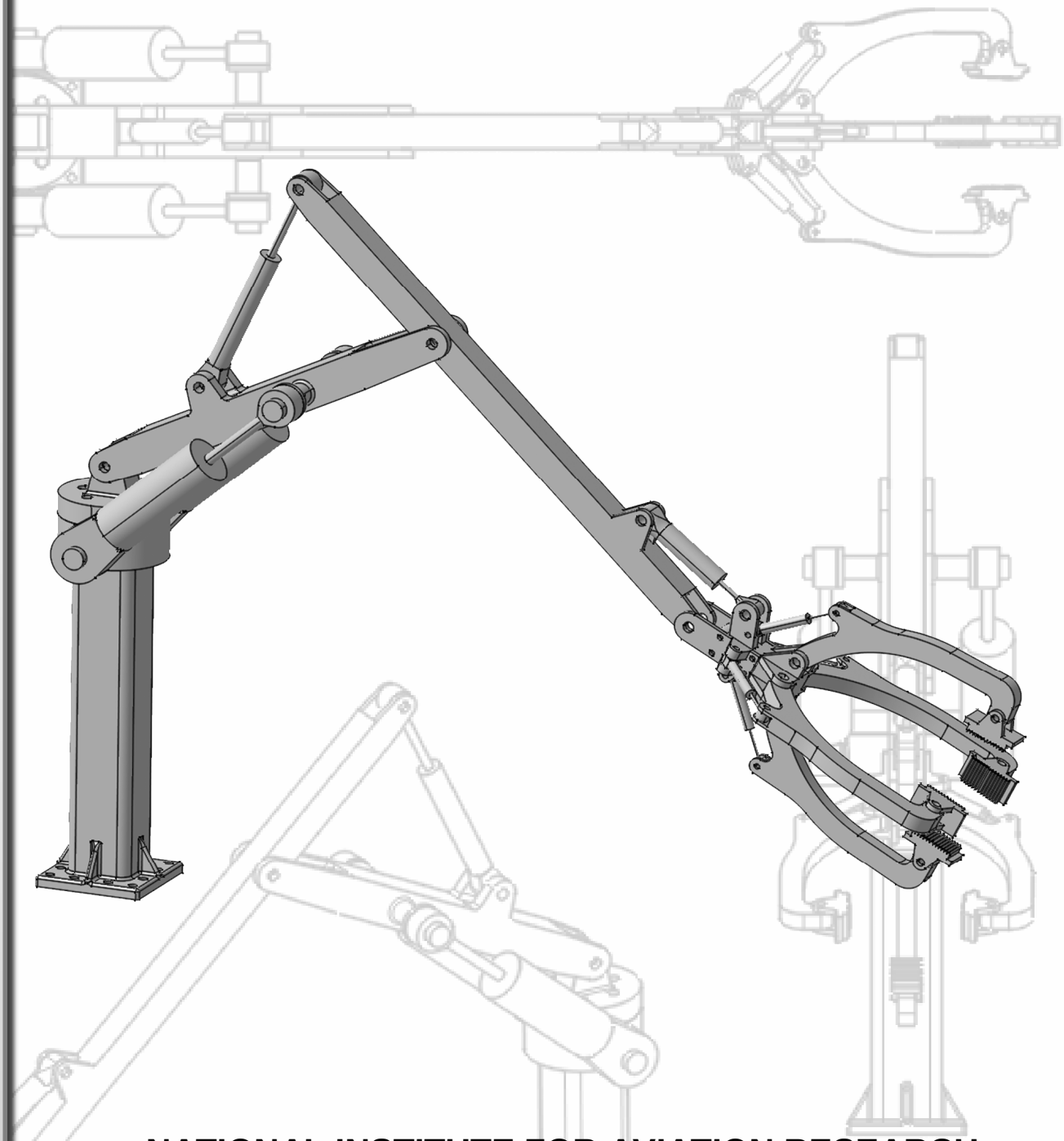
CATIA V5

Surface Machining



NATIONAL INSTITUTE FOR AVIATION RESEARCH
Wichita State University

Revision 5.13

www.cadcamlab.org

National Institute for Aviation Research
Wichita State University
Wichita, KS

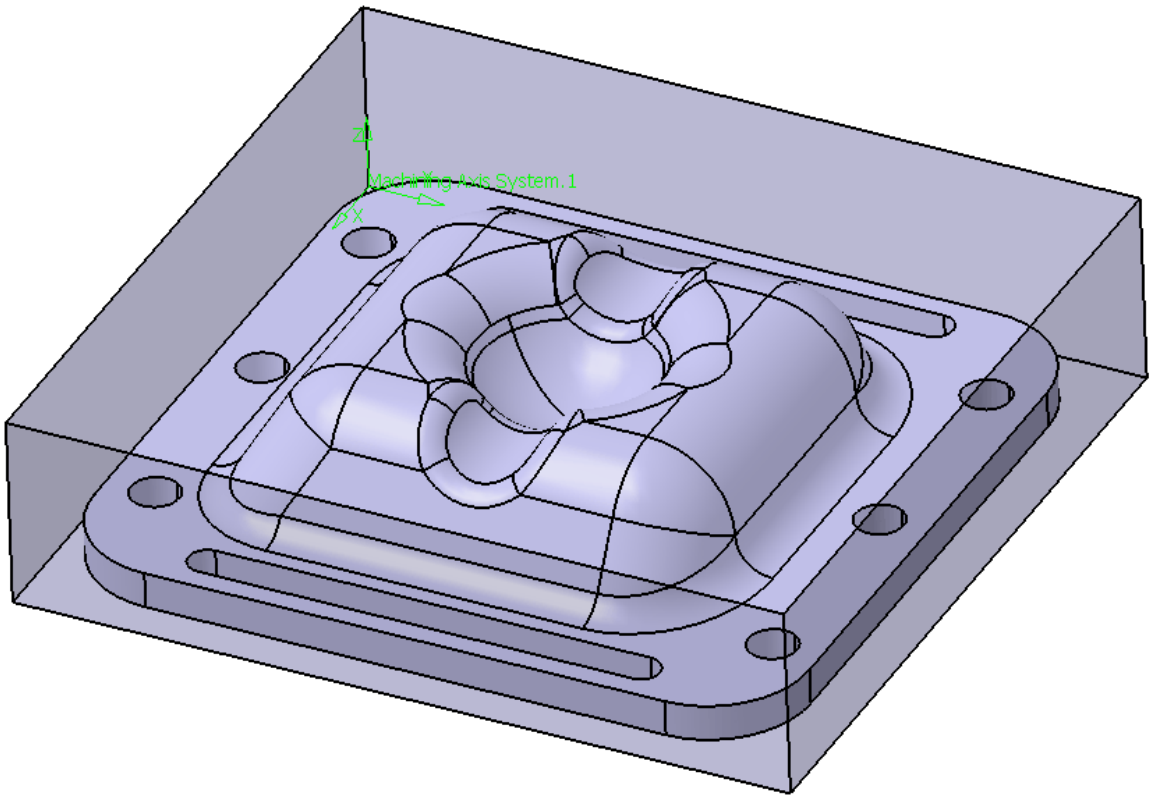
www.cadcamlab.org

TABLE OF CONTENTS

Additional Roughing

In many cases you may want to utilize some of the Prismatic Machining tools as well as Surface Machining tools.

Open the Rocker Mount Machining Process **file from the *Rocker Mount*** folder. This problem will use both prismatic and surface tools to complete the exercise. This problem will also be used with the Sweep Roughing operation, so be sure to save as you go along.

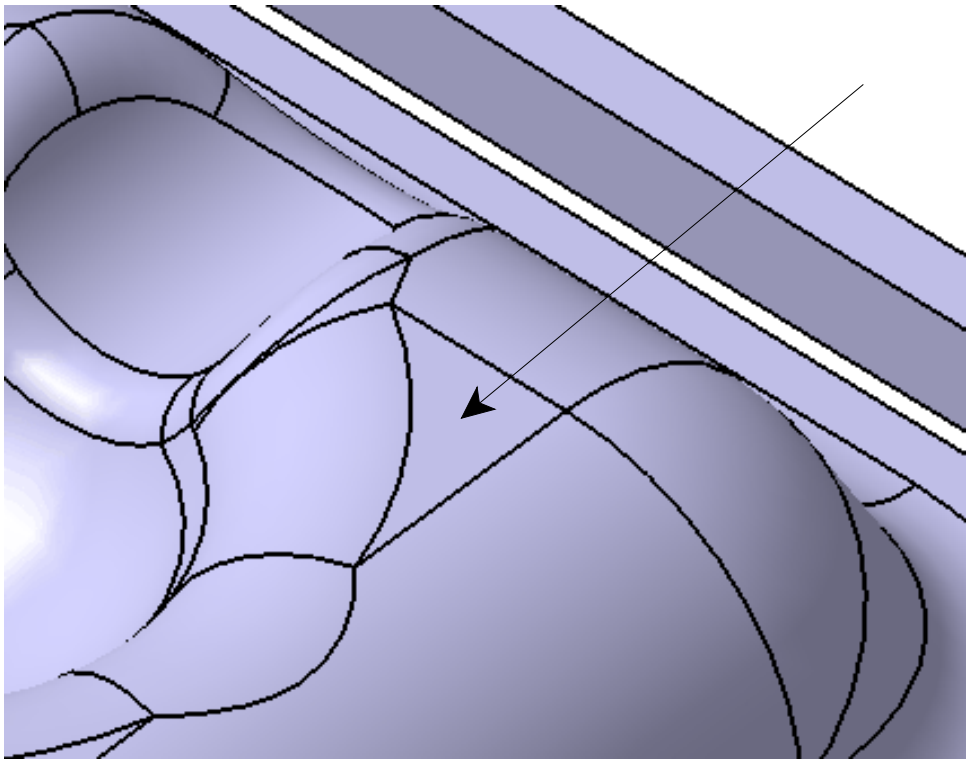


The very top of this part is flat, therefore a facing operation would work well to remove all of the material above the part. Profile contouring would also remove the outside area very well, much better than the roughing operation.

The Part Operation has already been defined for you. If you are not already in the Prismatic Machining workbench, you will need to go there.

Start a Facing operation in *Manufacturing Program.1*. You should already be very familiar working with the Prismatic Machining tools. The next few steps are going to be very general, so if you have trouble, refer back to your Prismatic Machining book.

Define the bottom of the facing operation as the top of the final part. There are small triangular areas that you can use for the bottom definition.

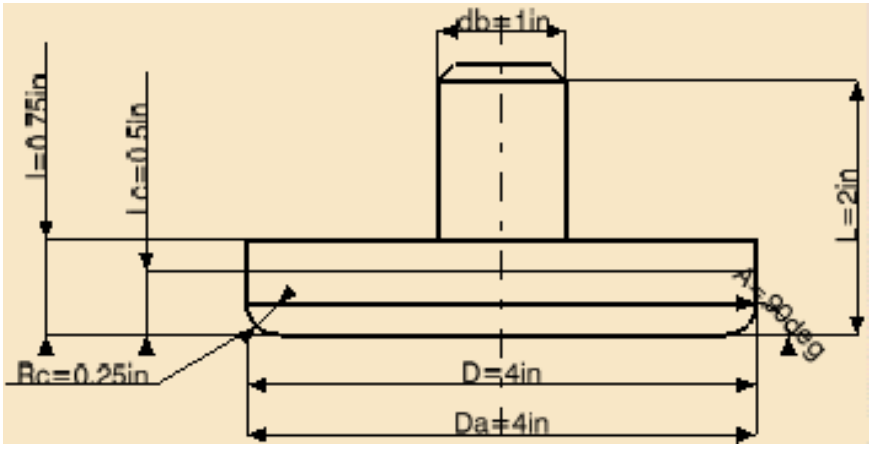


Define the boundary of the facing operation as the boundary of the stock material. Also define the top of the facing operation as the top of the stock material. This should have all of the geometry defined. Next you will define the machining parameters.

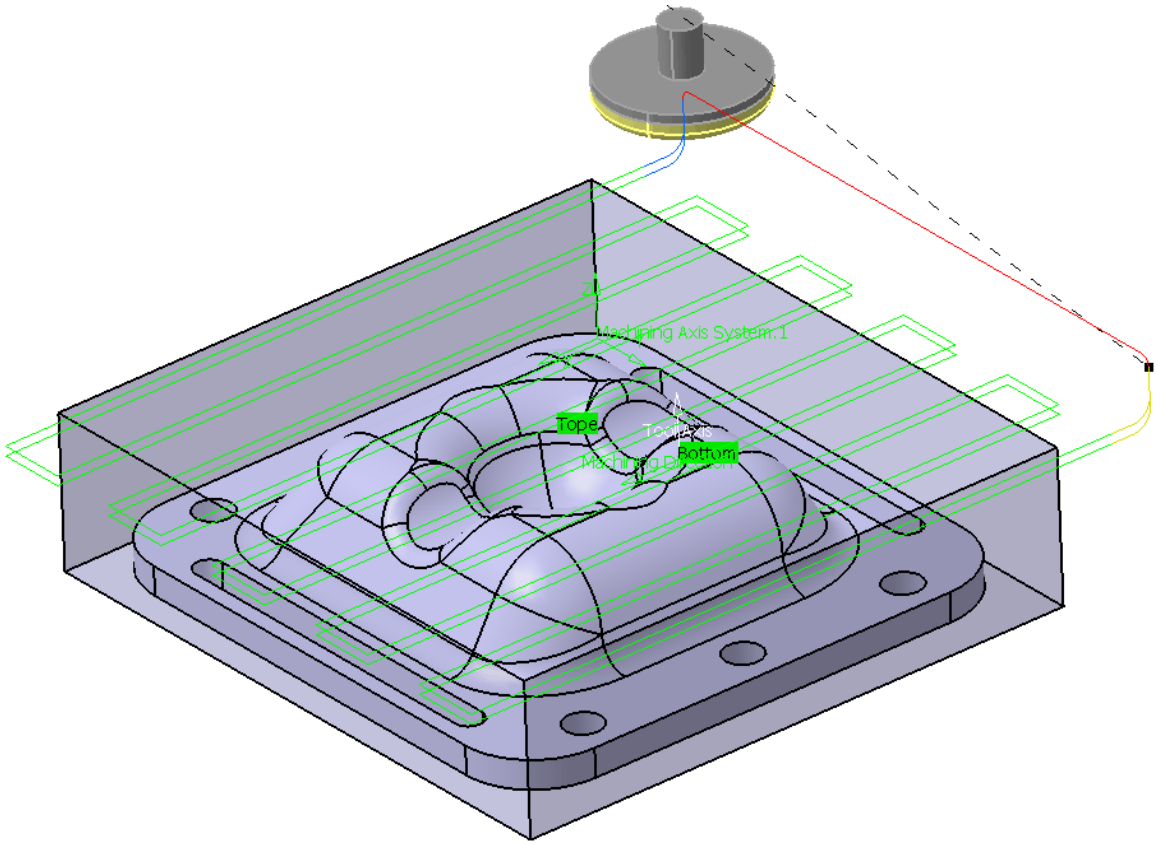
Change the *Tool path style* to *Back and forth*. Under the *Radial* tab, set the *End of path* to *Out,* and change the *Tool side approach clearance* to -0.5in. By setting the clearance to a negative value, you will be keeping the tool in contact with the stock material. This is ideal so that the tool will remain under a load.

Set a 0.25in maximum axial depth of cut. This will complete the machining parameters.

Define the following facing tool. This is a large part, so a large tool is optimum.



Define macros to finish the facing operation. It is left to you to decide what type of macros that you want.

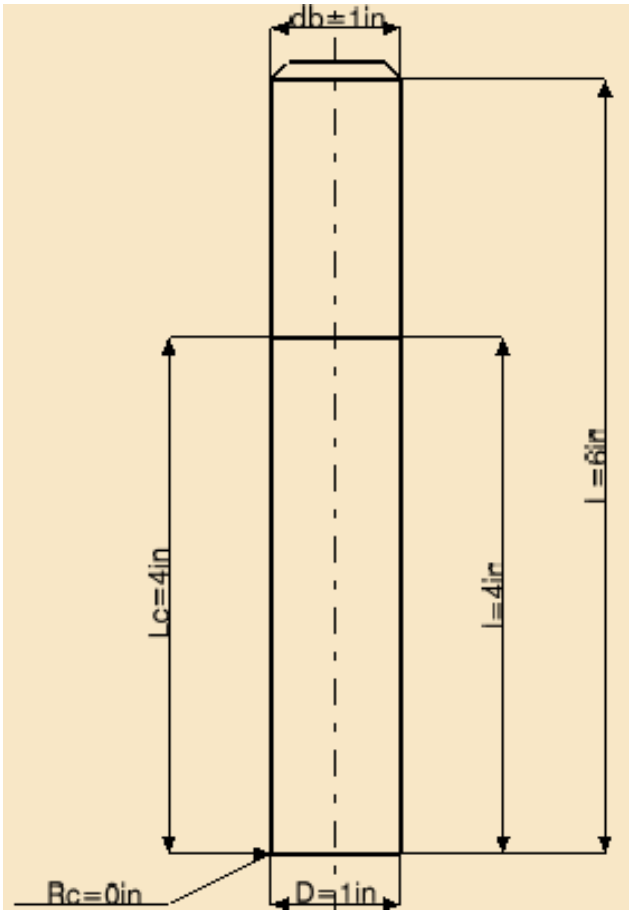


Now to profile around the outside.

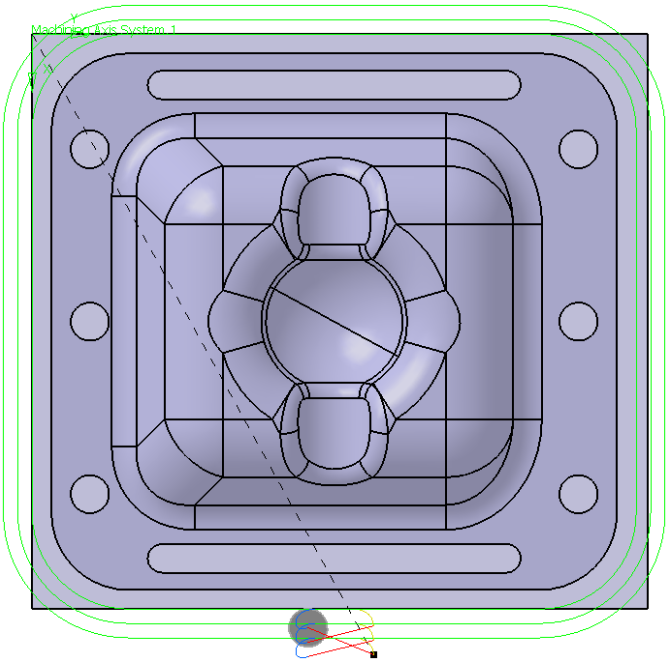
Start a Profile Contouring operation after the facing operation. Define the profile, top and bottom for the geometry definition. Be sure to make the bottom a soft bottom, as well as define the bottom on the bottom of the stock, not the part.

Define the profile as a closed tool path, with a one way tool path style. Also, set a 0.050in breakthrough, with 3, 0.375" radial paths. Cut a maximum of 0.5" deep each pass. This will have the machining parameters set. The last job is to define the tool and macros.

Change to the following tool.



Set macros that you feel will be appropriate. Again, the choice is yours on macros. You can make them as exotic or generic as you desire.



Now that you have a few prismatic machining operations, you are ready to integrate the surface machining tools.

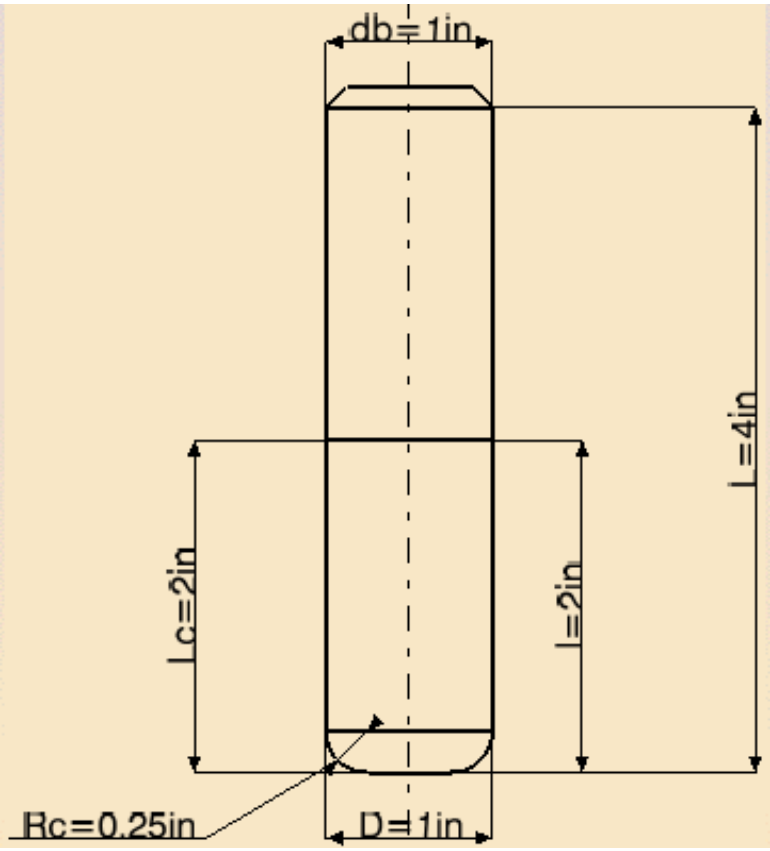
Switch to the Surface Machining workbench. Notice the workbench changes, but the part stays the same. Integrating workbenches is as simple as switching between them.

Start a Roughing operation after the Profile Contouring operation. The roughing operation will be used to remove the excess material near the part contours.

Define the *Part* definition. All you will need to do is select the part definition from the sensitive area, then select the final part partbody.

Change the *Offset on part* to 0.5in. The next set of operations will take care of the excess thickness.

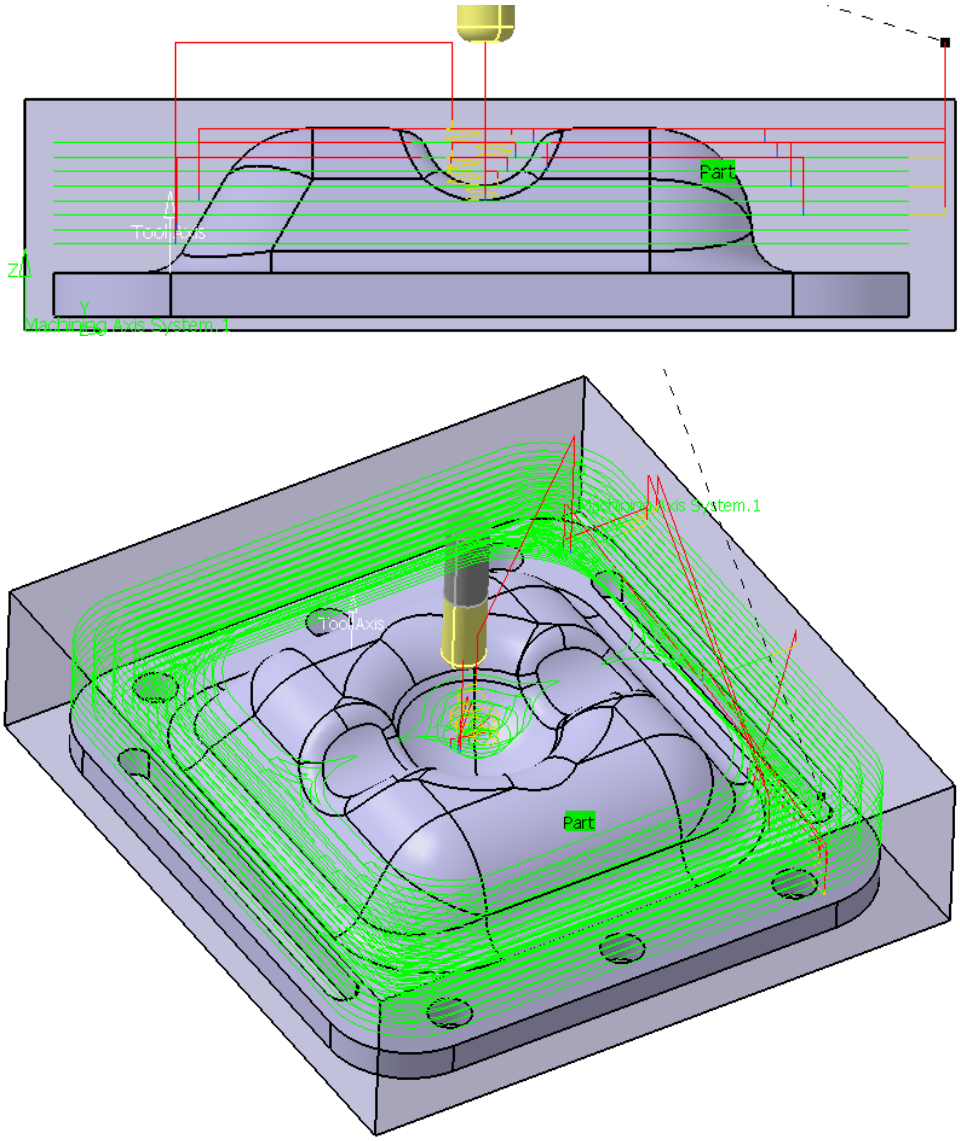
Make sure a *Spiral* tool path style is used, with a 50% stepover ratio. A 0.25" maximum cut depth should be sufficient. Once done, define the following tool.



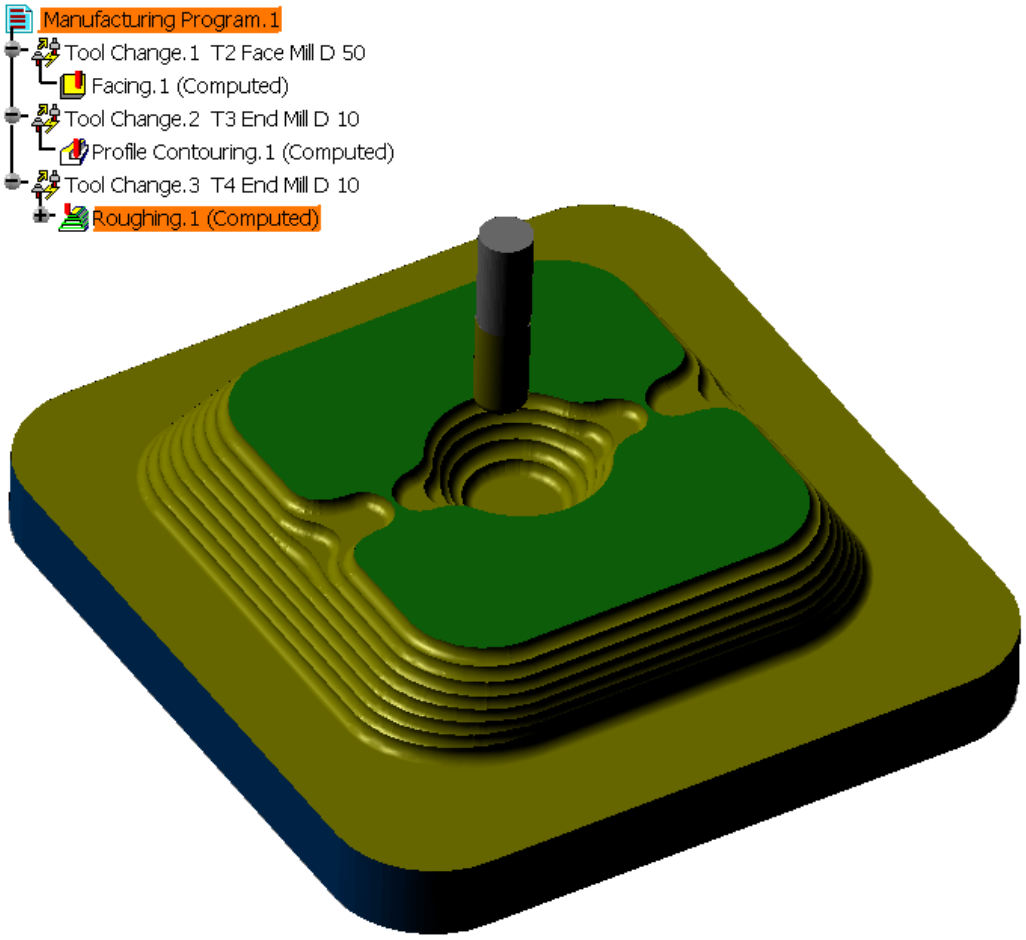
Now all that needs defined is the Automatic Motions.

Turn on *Optimize retract* with a 0.25in axial safety distance. Switch the *Automatic Motion* to a helical mode, with a 5deg ramping angle and a 60% helix diameter. An approach and radial safety distance of 0.125" should work well. This should be enough to define the macros.

Notice anything interesting?



When the roughing operation is used after any other operation, it will only cut the material that is remaining. The roughing operation is smart enough to recognize where material remains and where it does not. This will be investigated further later on. For now, be sure to save and move on.



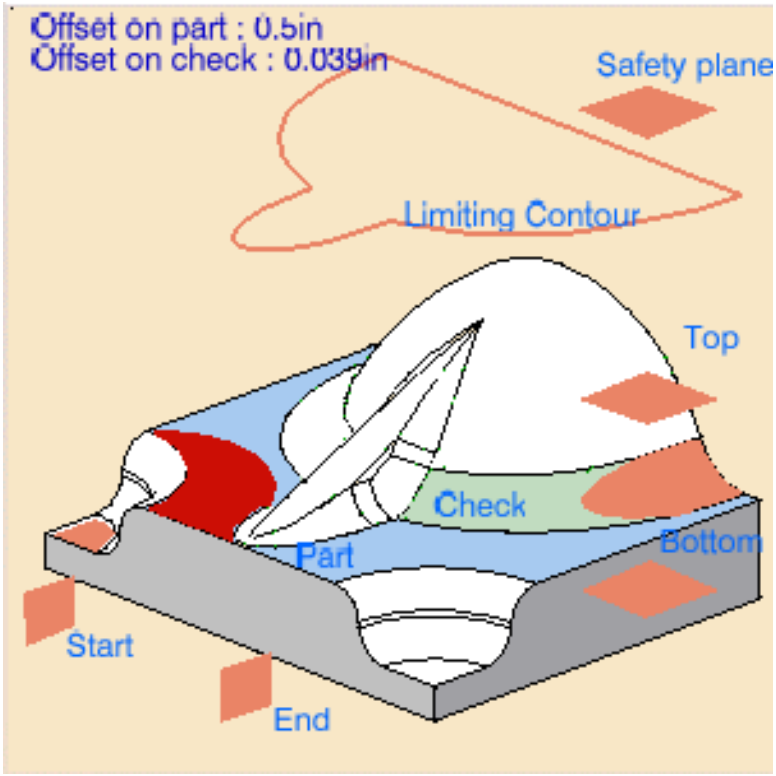
Sweeping

The sweeping operation is a good, all-around general surface sweeping operation. The tool makes passes along a surface of any contour, making the finished cut. Although the sweeping operation could be used for roughing, it is highly un-advisable. The sweeping operation is for final cuts only.

With your Rocker Mount Machining Process open, start a sweeping operation after *Roughing.1*.  Sweeping operations are going to be defined in a similar manner to sweep roughing operations.

Geometry Tab

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

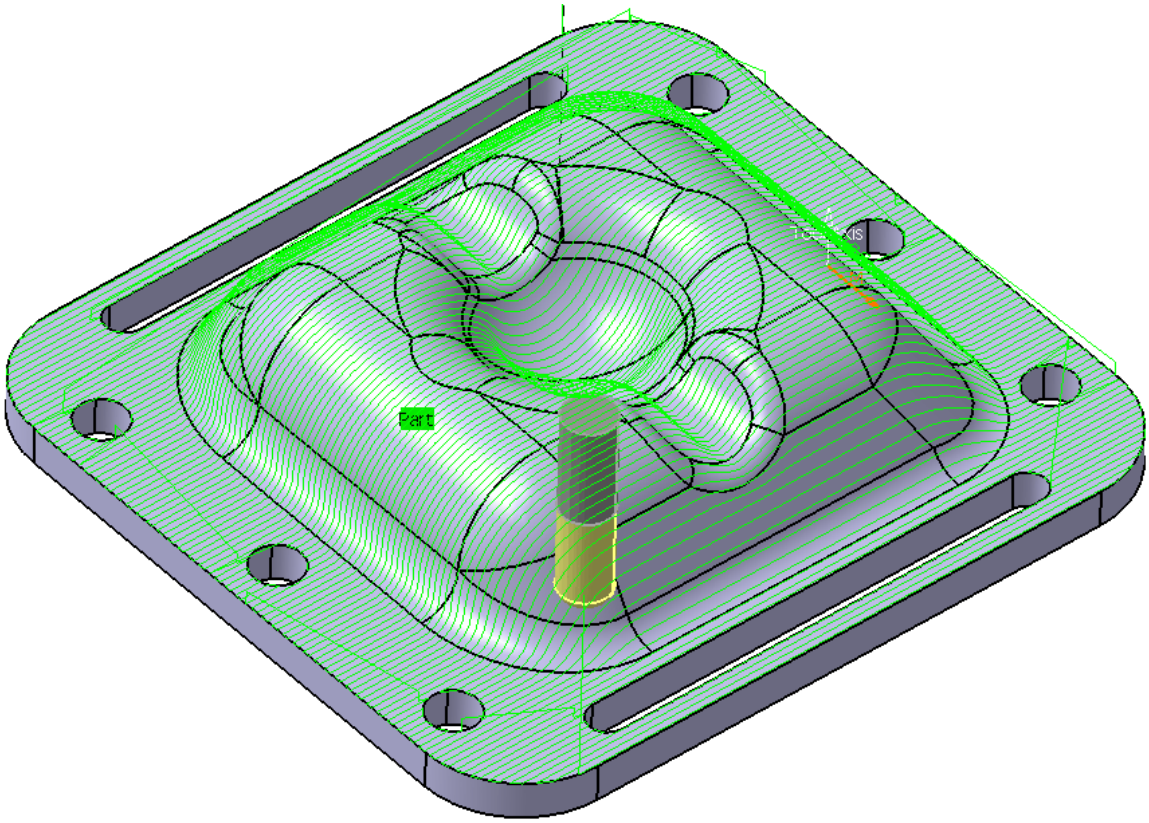


The two new additions to the sensitive window is the *Start* and *End* planes. These two planes define a limiting along the cut of the surface that will keep the tool from passing. This is different from the limiting contour in the sense that the limiting contour can be the entire surface, but the tool is only allowed to machine from the start to the end plane along the surface.

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

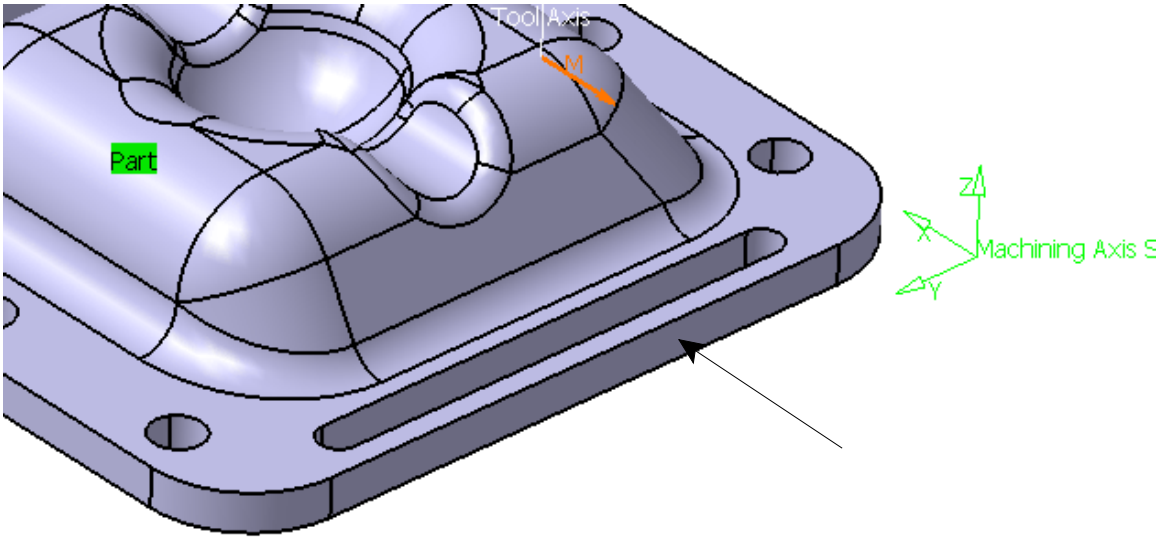
Change the *Offset on part* to be 0.0in. This will make sure you machine the final part.

Take a quick look at the tool paths.



Granted, there are a lot of things wrong with these toolpaths, but as you can quickly see, the sweeping operation will cut across the entire part, regardless of the features.

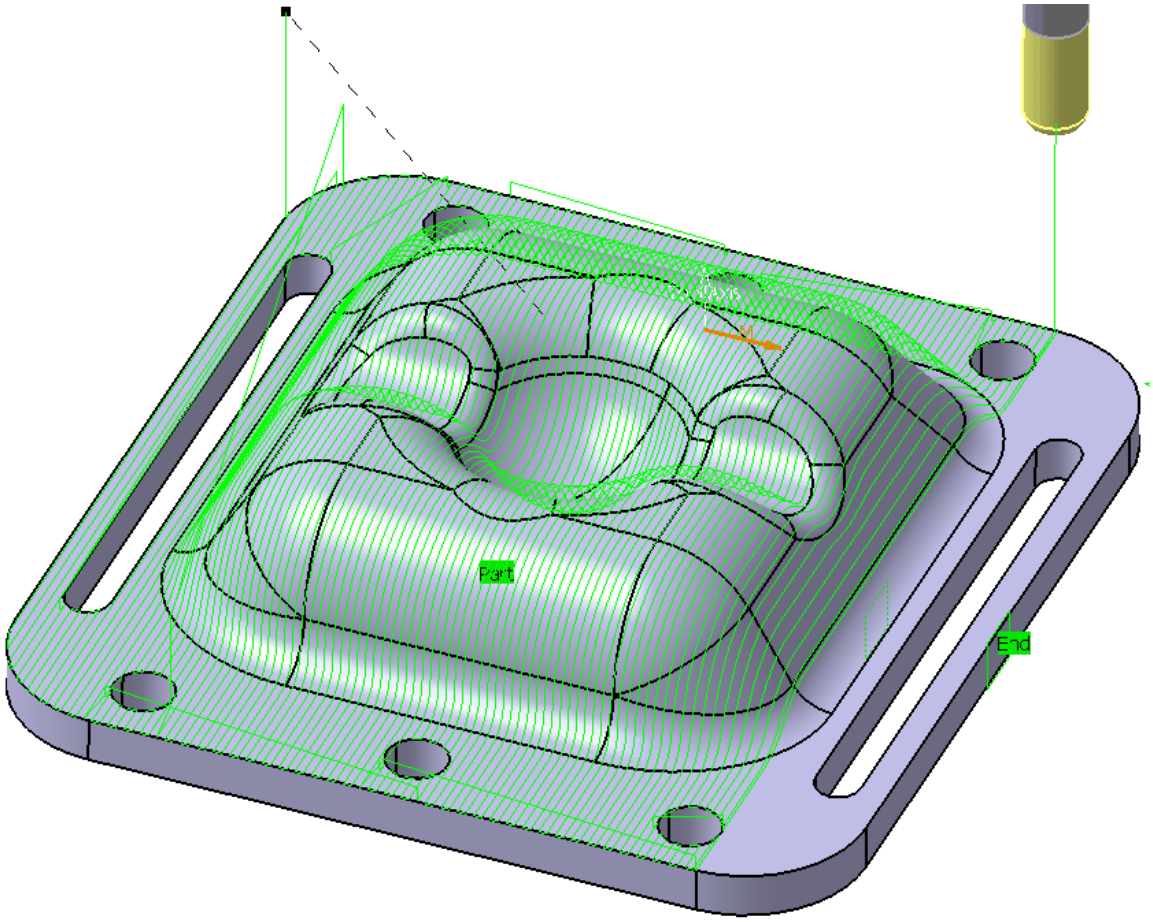
Select the *End* plane from the sensitive area, then select the flat side as shown below.



This will define the end plane to be on the side of the part. At this point, the end plane does no good, however, once you offset it, it will become more useful.

With the third mouse button, select on the *End* plane. With the contextual menu, select *Offset*. Set the offset to -2in. The -2" is required to make the plane move inward. If your plane goes outward with a negative, then use a positive value.

Take another look at the tool paths.



Notice they now stop at the end plane. The start and end planes work well for defining limits on how you want the part cut.

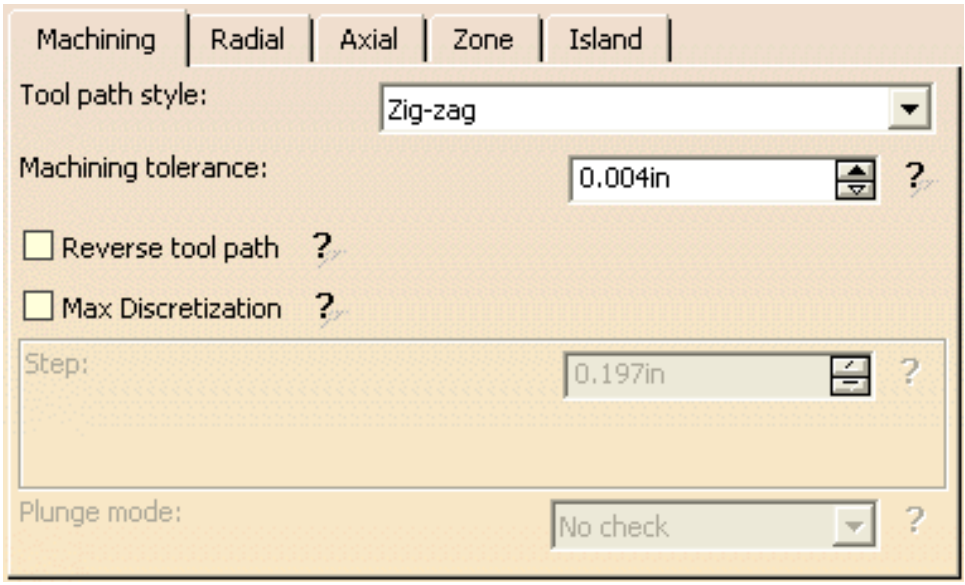
With the contextual menu, remove the *End* plane. This will go back to machining the entire surface.

Now to move on to the machining parameters.

Machining Parameters

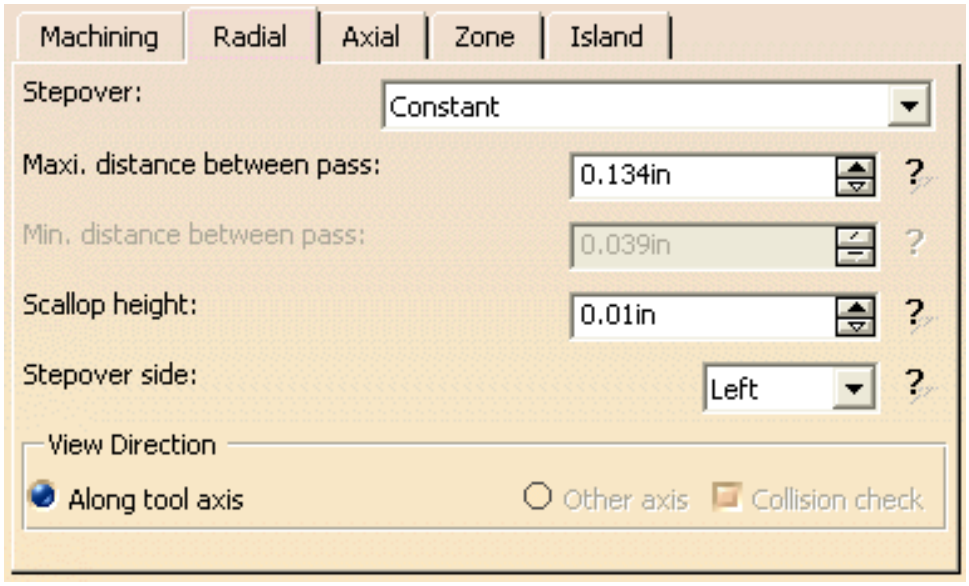
Take a quick look at the various machining parameters.

Machining Tab



Tool Path Style	Defines the type of cut that is to be made across the part. There are three options available.
Machining Tolerance	Defines the tolerance of the machine being used
Reverse tool path	Reverses the computed tool path
Max Discretization	Allows you to override the step size of the machining operation
Step	Defines the maximum step size allowed
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 Tab

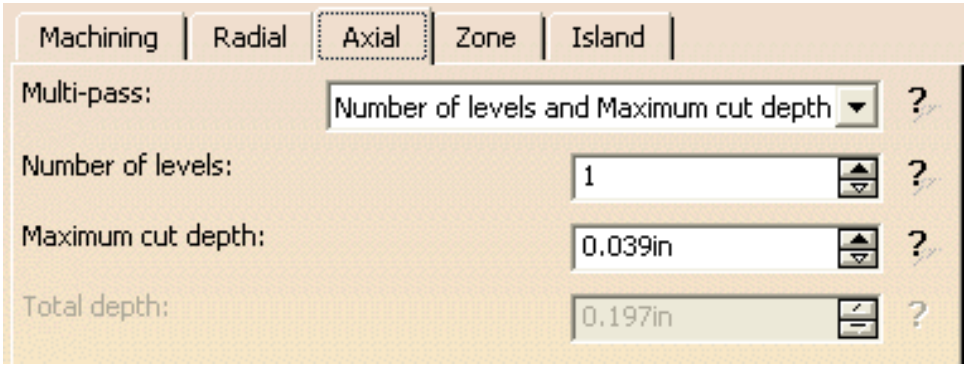


Stepover Defines how the sidestep is calculated

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
Maxi. distance between pass	Defines the maximum distance stepped each radial pass
Min. distance between pass	Defines the minimum distance the tool can step each radial pass
Scallop height	Defines the maximum scallop height allowed
Stepover side	Defines which side the tool steps, left or right
View Direction	Defines the normal direction for the tool path computation
Along Tool Axis	The normal for the tool path is computed looking down along the tool axis
Other Axis	The tool paths are calculated looking along another tool axis to determine the stepover distance
Collision Check	Collisions between the part and the tool would be avoided

Note: The only time the View Direction would need to be changed would be if you were machining a large sloped surface. Otherwise, by changing the view direction to something other than the tool axis, you run the risk of not machining all the surfaces.

Axial Tab



Multi-pass Defines how the multiple axial steps are calculated

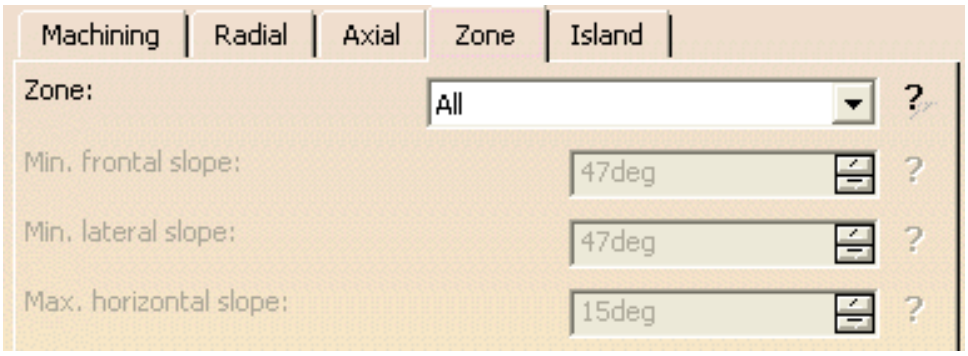
Number of levels Defines the number of axial levels

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 the final pass, the pass against the part surface is computed, then a copy is made of those tool paths and they are moved up 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 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.

Zone Tab



Zone Controls what type of area is machined. There are a number of options.

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

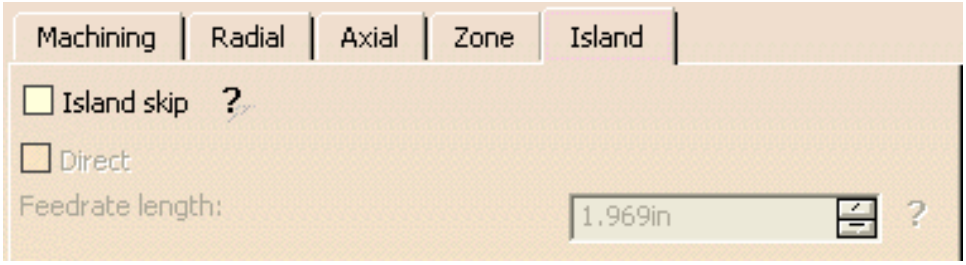
Horizontal Zones Only machines areas that are within the horizontal slope angle

Min lateral slope Minimum angle to be considered a lateral wall

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

Max horizontal slope Maximum angle that will be considered a horizontal plane

Island Tab



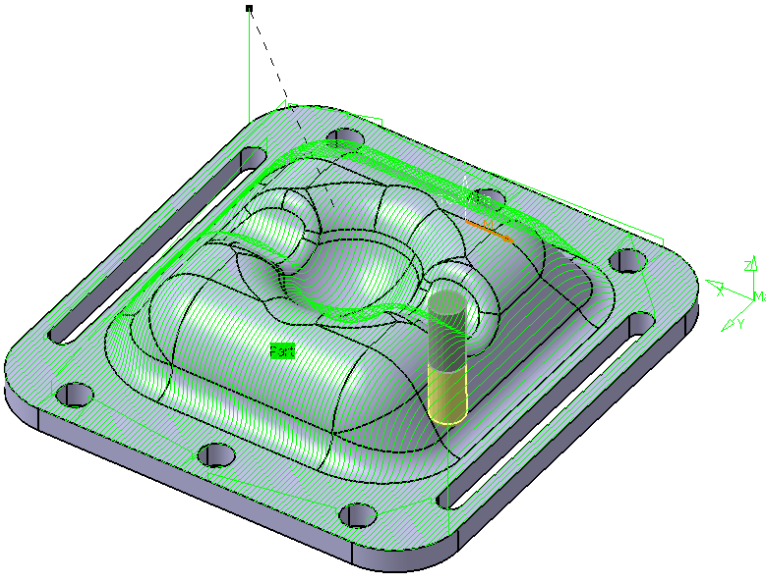
Island skip Indicates that islands, or gaps in the surface should be jumped over, not gone around

Direct This option, available only when Island skip is turned on, will cause the tool to move directly over the island, and not make a retract and approach motion

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.

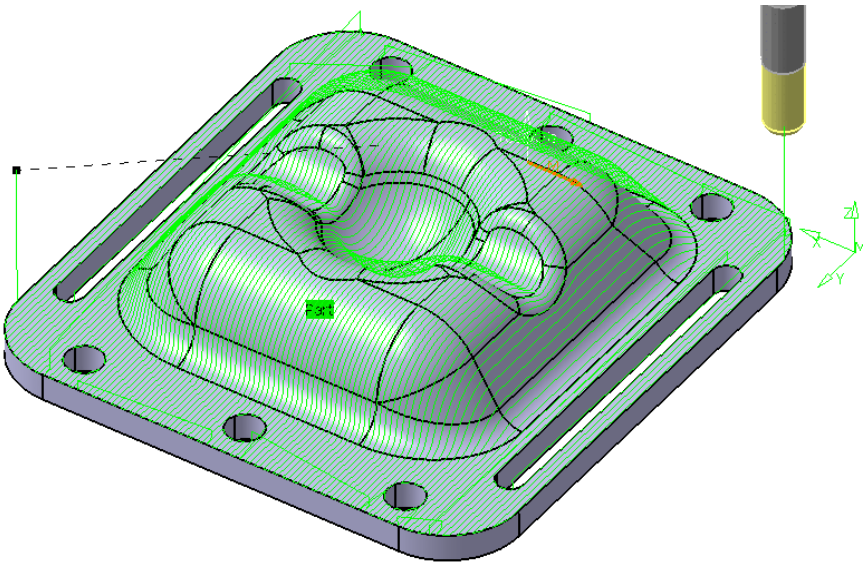
Now it is time to change a few machining parameters to determine the effects they make on the tool paths.

Take a quick note of the unchanged tool paths.



Note that the tool starts in the back left corner of the part and ends in the front right corner.

Turn on *Reverse tool path*. Instead of the tool starting in the front right and ending in the back left like you might expect when you reverse the tool paths, it instead starts in the front left and ends in the back right.



Since the tool is in a zig-zag motion, the cutter is continually switching between climb and conventional milling. If you were machining in a one way motion, the reverse tool path would make the difference between climb and conventional milling.

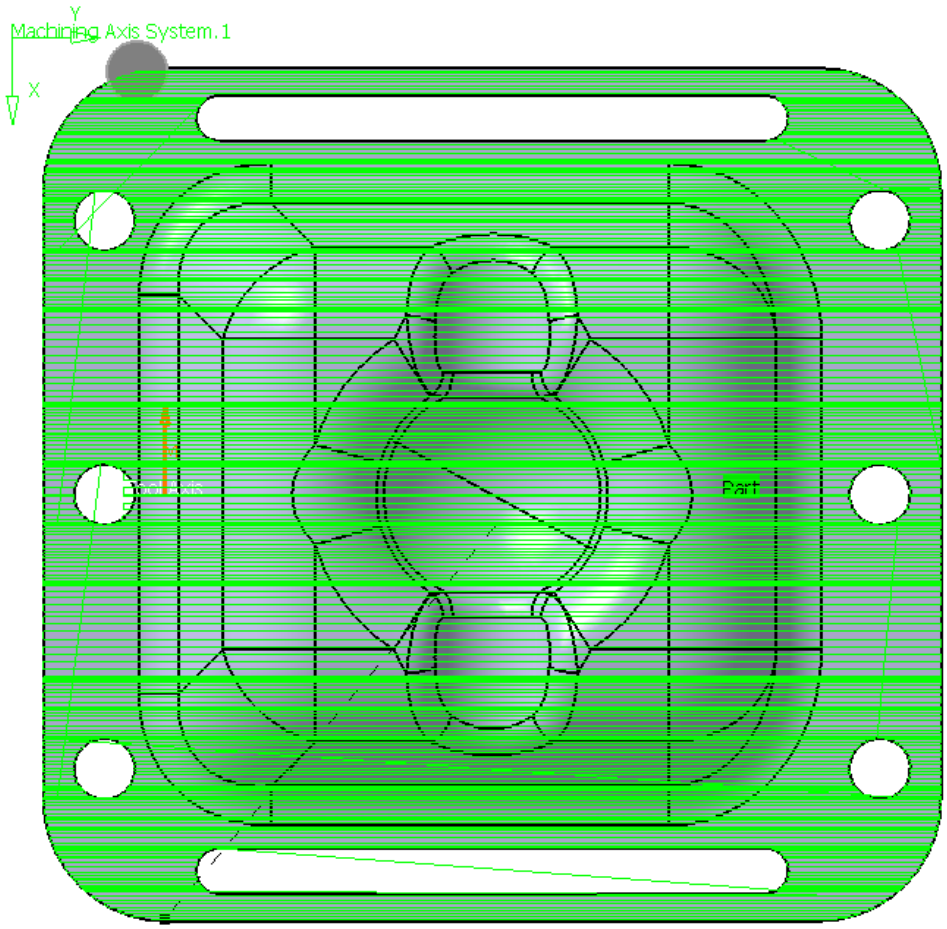
Turn off *Reverse tool path*. In the *Radial* tab, change the *Scallop height* to 0.005in. Notice when you change the scallop height, the *Maxi. distance between pass* changes to 0.155". The two fields are linked together by some means. Fortunately, most times you will not have a *Constant* stepover.

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 0.125" and a scallop height of 0.005", the tool will always make a step of 0.125" because that is set as a minimum. The opposite is also true. If you have a maximum distance set to 0.005", and a scallop height of 0.020", the tool will always make a 0.005" step, and your scallop height will be much smaller than what you are trying for. This results in various problems as you might suspect.

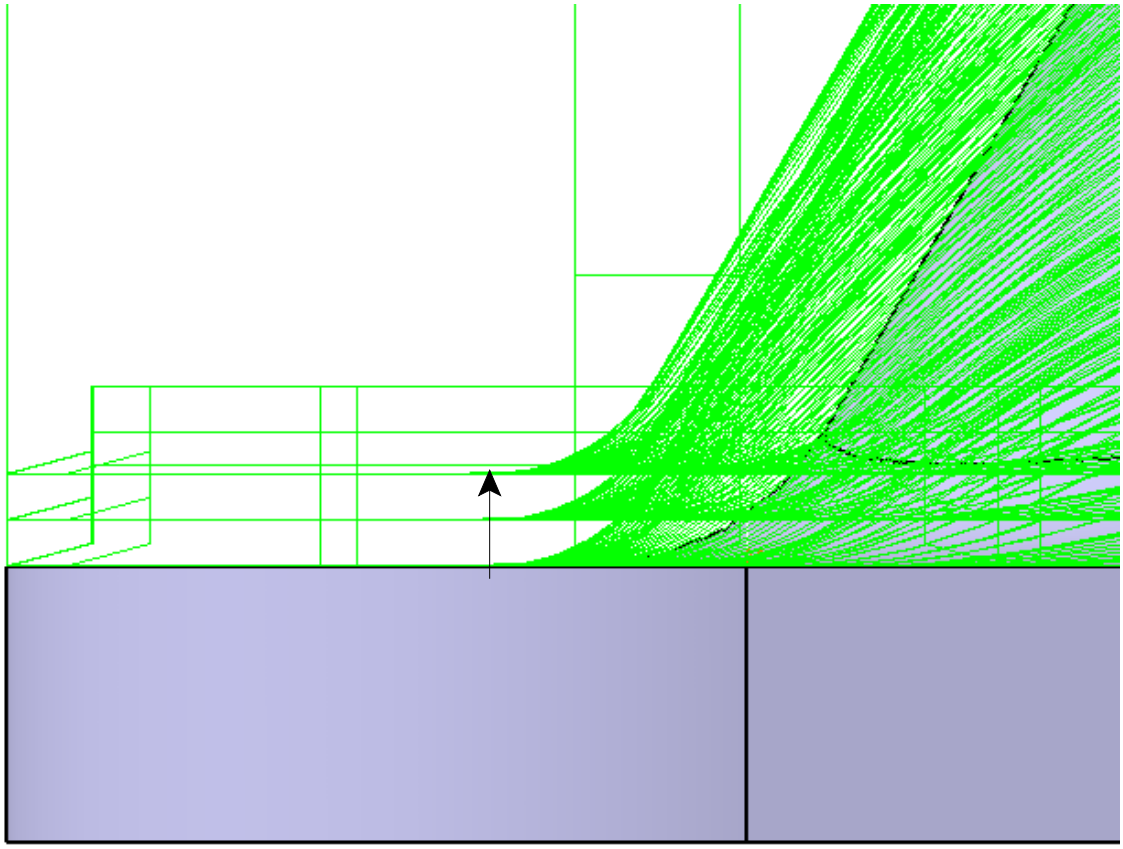
Change the maximum distance to 0.125", the minimum distance to 0.005", with a scallop height of 0.010". This will cause the tool to allow for varying sized steps as it goes across the part. The tool will do its best to maintain a scallop height of 0.010", but will guarantee that the step size will never go under 0.005" or above 0.125".



Again, take a quick note of where the tool starts and stops. Currently it starts in the back left corner of the part, then makes its way to the right.

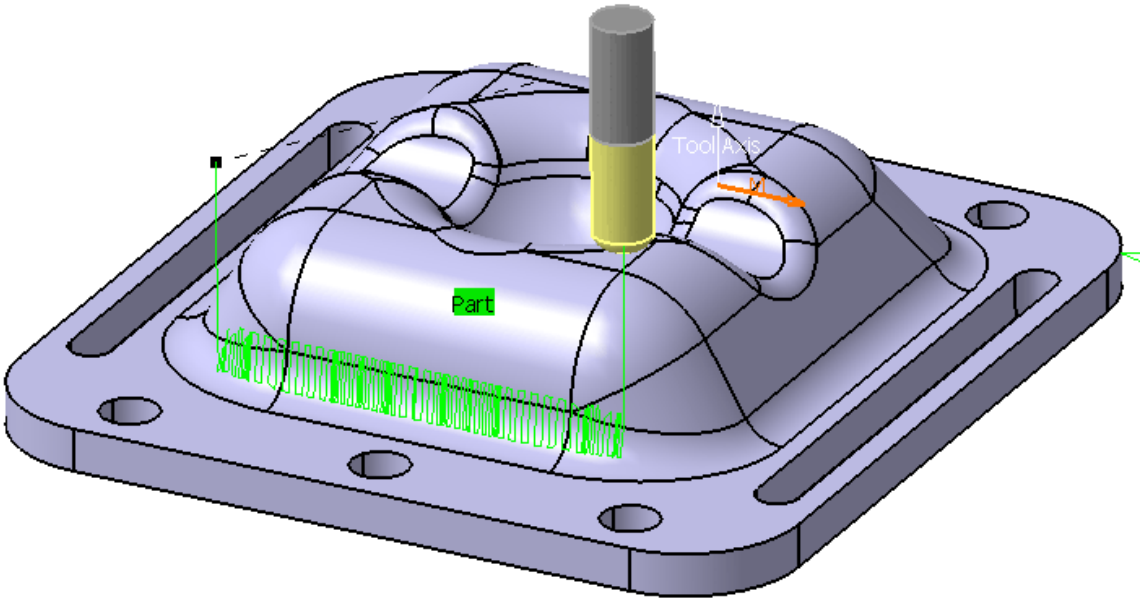
Change the *Stepover side* to *Right*. Notice now that the tool starts in the back right side and then steps left. Between this option and the *Reverse tool path* option, you can change the overall starting and machining direction of the tool paths.

Change the *Stepover side* back to *Left*. Under the *Axial* tab, set the *Number of levels* to 3, with a *Maximum depth of cut* at 0.125". Now the sweeping operation makes three levels of cut. If you look closely at the tool paths, you should notice that the tool paths are only a vertical offset from the bottom set of tool paths.



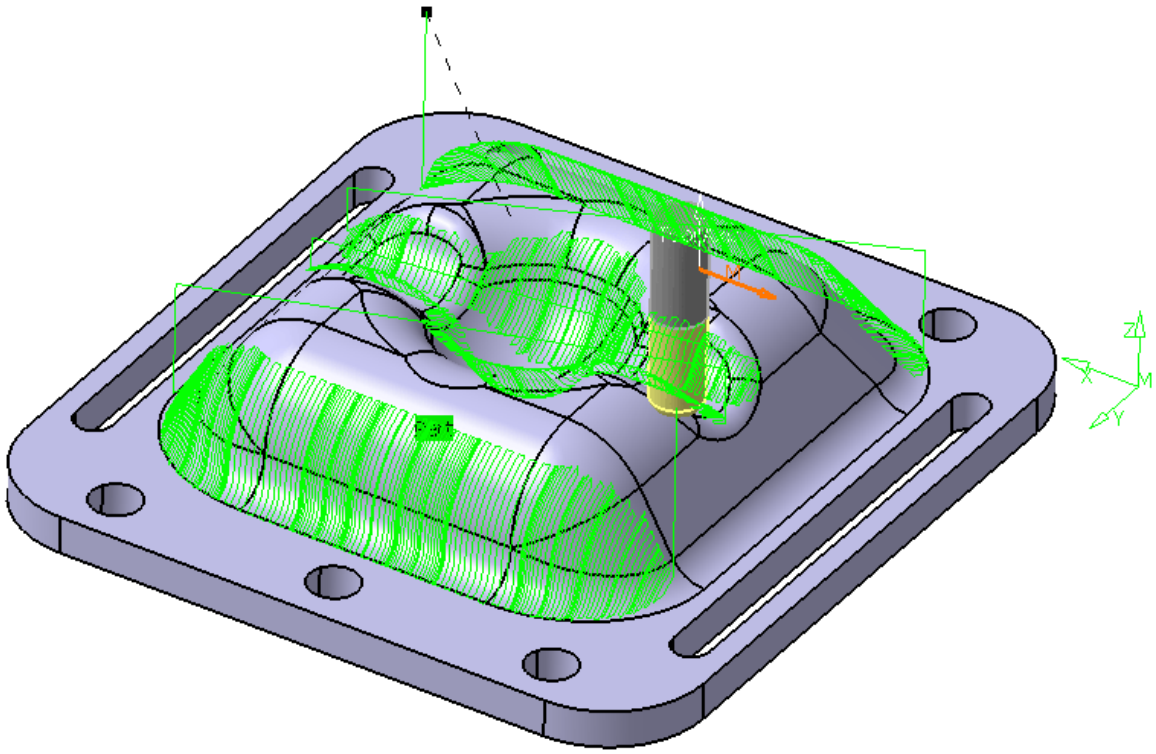
Since this is a purely vertical offset, it does not work very well to remove excessive material. You are much better off to remove the excessive material with a sweep roughing operation prior to the sweeping operation.

Set the *Number of levels back* to 1, and then change the *Zone* to *Frontal walls*. Set the *Min. frontal slope* to 75deg. This will cause the sweeping operation to only cut the frontal walls, or walls that the tool will hit head-on.

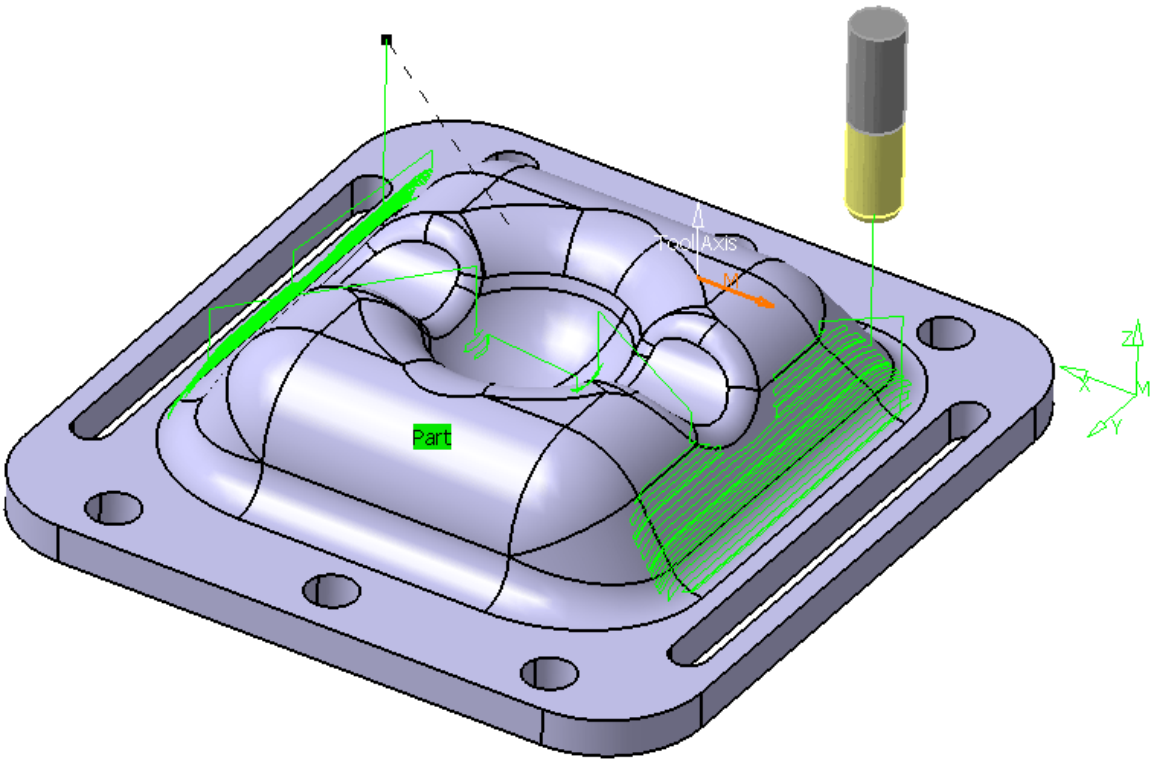


Notice only a small frontal area gets machine. Basically, anything within 15deg (90-75) of vertical, and on the frontal plane gets machined. Everything else is ignored.

Change the *Min. frontal slope* to 25deg. Now a lot more gets machined. By decreasing the minimum frontal slope angle, you are allowing more and more of the non-frontal, non-vertical walls to get machined.

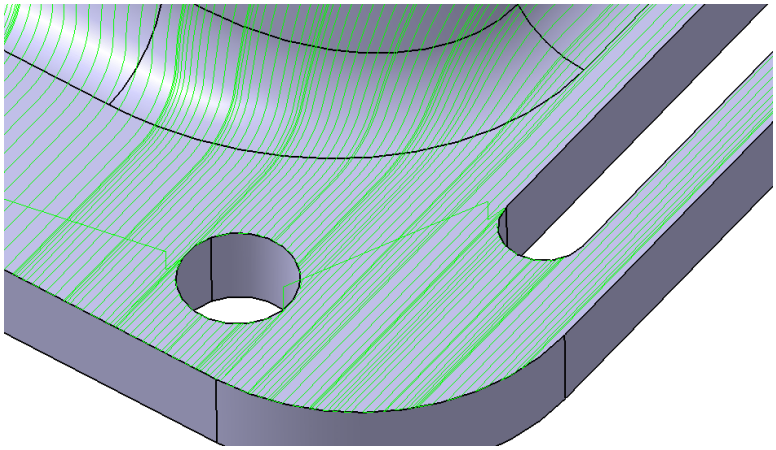


Change the *Zone* to *Lateral Walls*, with a minimum angle of 45deg. This time the walls on the sides get machined. The 45deg allows the part to be partially machined on the frontal zones.

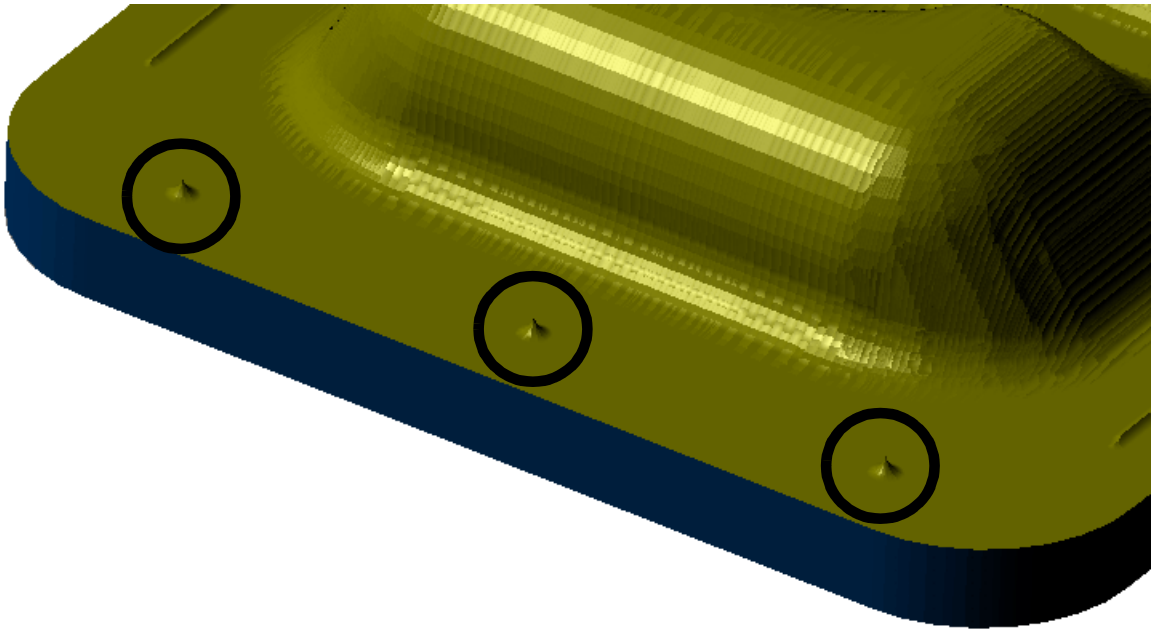


As you might suspect, the *Horizontal zones* option works in a very similar fashion where it will machine only horizontal areas, and none of the lateral or frontal walls. In most cases, you will find that you want to machine all zones.

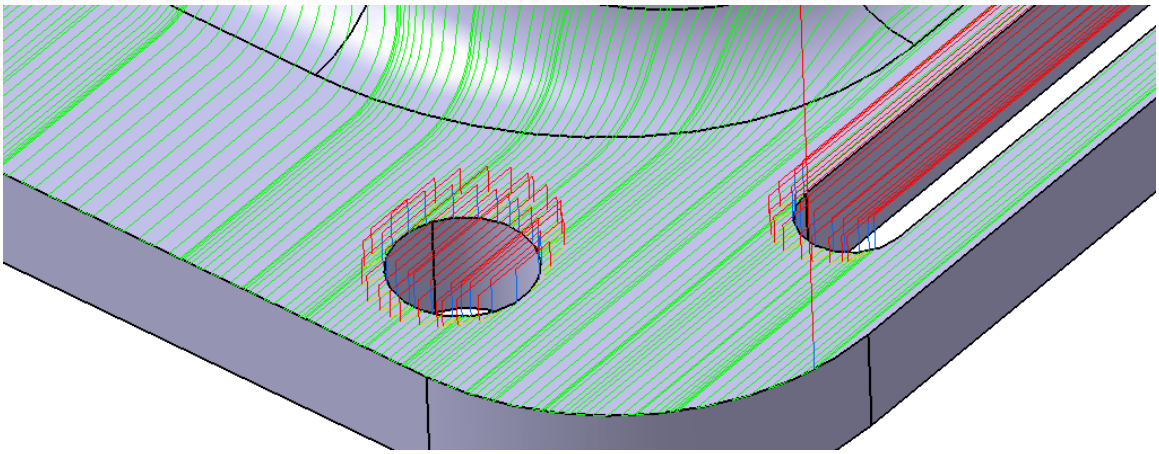
Change the *Zone* back to *All*. Now take notice of what happens when the tool approaches the holes in the bottom of the part.



Notice how the tool paths do not go over the hole, nor do they retract from one side and approach the other. Instead, the tool zig-zags its way around the hole and pocket area, then comes back later to clean up what was left behind. Even though, there will still be small peaks that will make drilling a straight and accurate hole quite difficult.

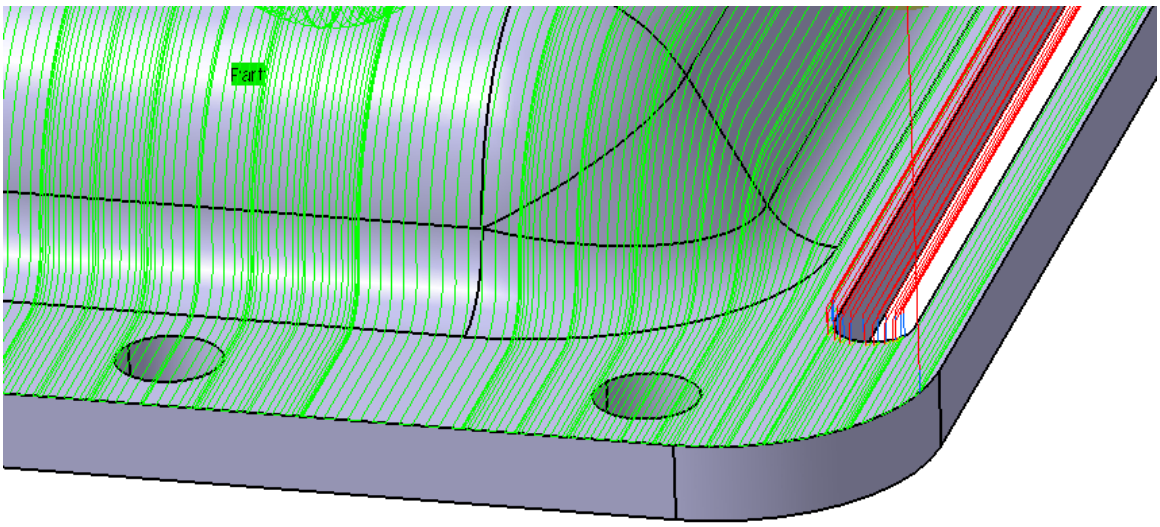


Turn on *Island skip* under the *Island* tab. Although somewhat better, this will still yield the same problem as before.



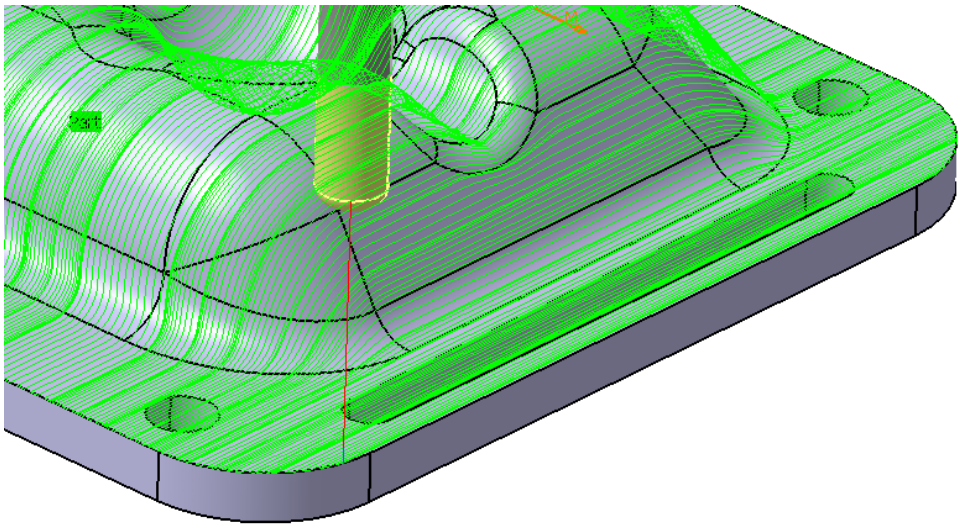
The tool now retracts once it gets to the hole, rapids over, then approaches back down on the other side. A small mound will still be present with this scenario.

Turn on *Direct*, with a *Feedrate length* of 2in. Now the tool paths are a little more of what you would want.



Notice now, the tool passes over the hole as though it is not there. You should also note though that the tool does still retract and approach over the pocket area.

Change the *Feedrate length* to 10in. This will now cause the tool to pass over the slot as though it is not there either. Increasing the feedrate length is one way to make sure you pass over all the open areas.



You could have also change the machining direction to fix the problem of jumping over the slot. By machining in another direction, the tool would not see the slot as being long, but instead, just a narrow jump.

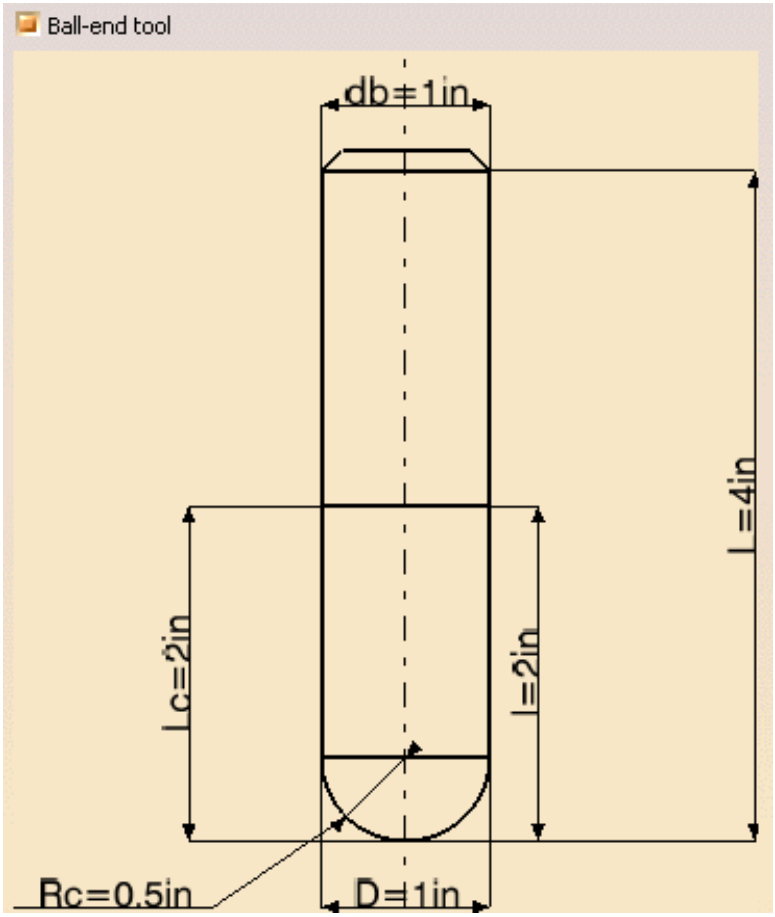
The only time that the direct island skip has troubles is when the Z-height changes between two levels of a pocket.

The tool paths are now starting to look really good. However, this is still an incorrect operation. The tool used is not a good choice. Granted, the physical dimensions of the tool are just fine, but if you remember, this is the same tool that was used for roughing earlier. Very seldom are you going to want to rough and finish cut with the same tool.

Change to the tool tab. Currently, the previous tool is used as the default tool. You need to force a tool change so you can change the parameters of the tool.

Change the *Name* of the tool to T4 Ball Mill, and press Enter. By changing the name of the tool, you are forcing a tool change.

Change the tool to be a ball nose end mill, with all the same dimensions as before. You should just need to select the Ball-end tool checkbox to convert the tool into a ball nose tool.

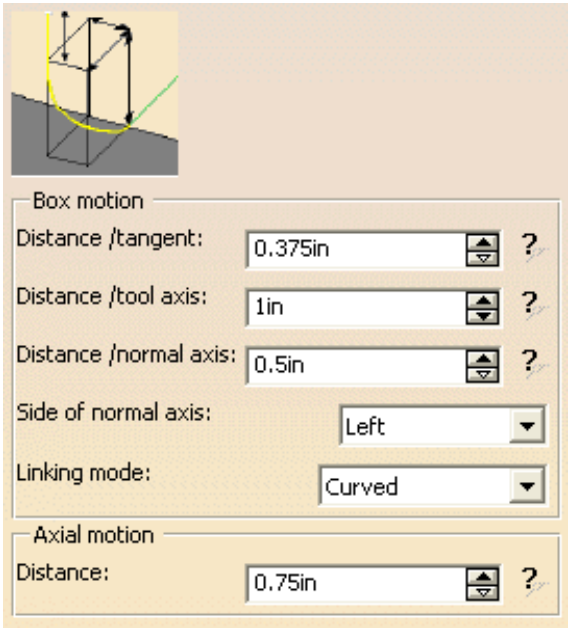


This nearly has the sweeping operation complete. The only thing left is to define good macros.

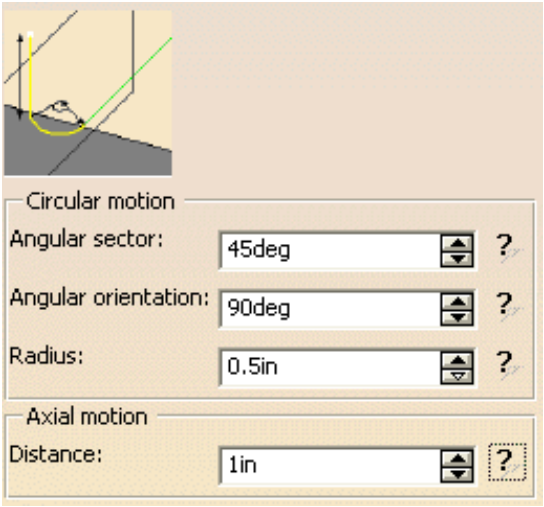
Switch to the macros tab. Notice that you are now back to the standard surface machining macro style. If you want to review the different macro types and available macros, refer back to the sweep roughing exercise.

Select the *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.

Change the box macro to have the following parameters. This will make the box have a 1" height, 0.375" width, and 0.5" depth.

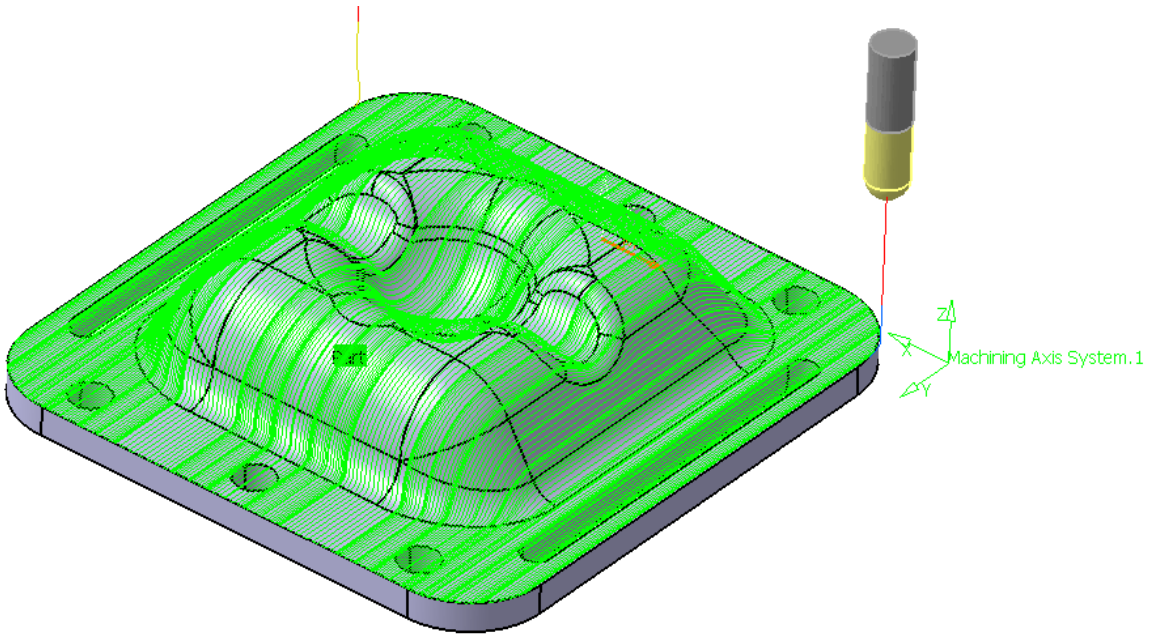


Change the *Retract* macro to a circular macro, with the following parameters. As you might have noticed, using the pre-defined macros allow for a quick definition of the macros. Many times it is advantageous to use the pre-defined macros instead of trying to custom build macros all the time.

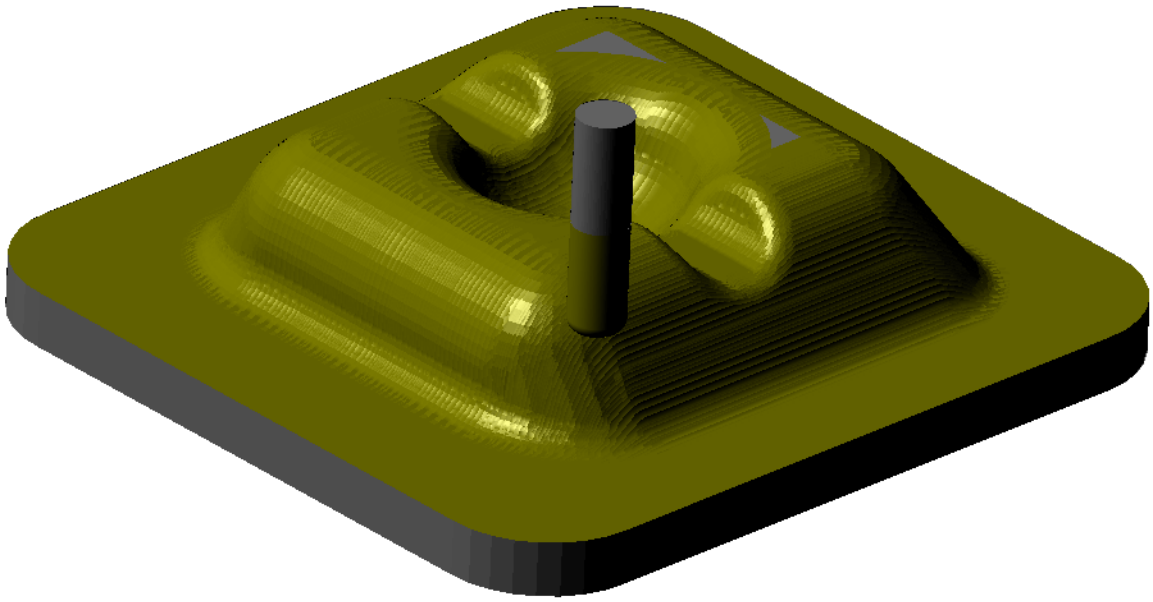


This will finish your Sweeping operation.

Sweeping, although it works good for an all-purpose finishing operation should not be used to completely finish parts all the time.



Save and close the document.



Other available courses

CATIA V5 and ENOVIA

- **CATIA Basic Concepts**
- **CATIA Part Design & Sketcher**
- **CATIA Assembly Design**
- **CATIA Drafting**
- **CATIA Wireframe & Surfaces**
- **CATIA Prismatic Machining**
- **CATIA Surface Machining**
- **CATIA Fitting Simulation & Kinematics**
- **CATIA Functional Tolerancing & Annotation**
- **CATIA Stress Analysis**
- **ENOVIA DMU Viewer**
- **ENOVIA LCA Basic Concepts**
- **ENOVIA LCA Advanced Concepts**
- **ENOVIA LCA Product Design**

To enroll in any of the above courses, contact us at: (316) 978-3283
toll-free at: 1-800-NIARWSU or email: info@cadcamlab.org