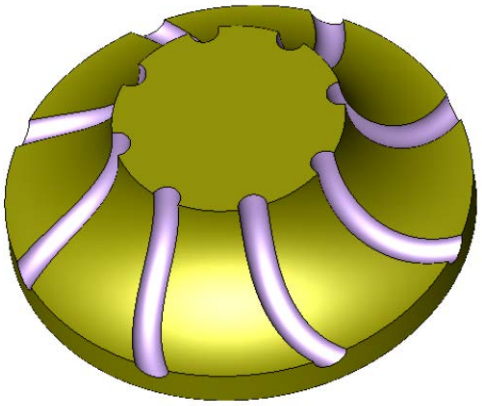
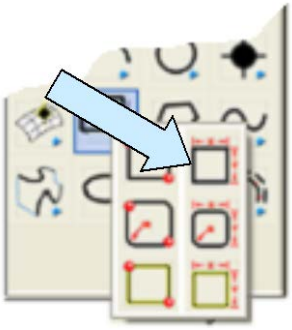
KeyCreator Lesson KC8215

Machining a Rotor with Spiral Veins (Using Rotate Tool Path)



In this exercise we're going to create the wild-looking rotor illustrated to the right. We'll use a roughing tool path, a 2D profile path to finish the rim, a finishing tool path for the curved faces, and a geometry to tool path run for one spiral vein that will then be copied using the Rotate Tool Path Function to get the other seven veins.



Let's build a model of the part first.
Start with a new file in View 2. (The Front View.)

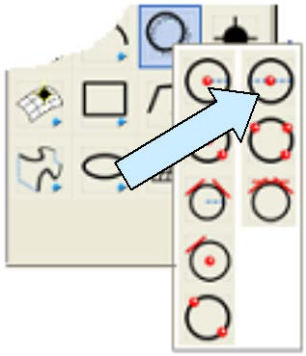
Click on the CREATE RECTANGLE BY WIDTH HEIGT Icon. Use the TopLeft Anchor Option.

Type 6 for the Width and 2 for the Height.
Using the KeyIn Option, hit the ENTER Key three times to place the rectangle.

Select a new construction color.

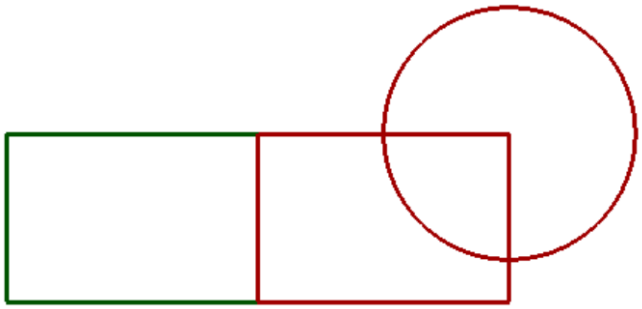
Click on the CREATE RECTANGLE BY CORNERS Icon.
Using the CtrMid Option, click on the top edge of the existing rectangle.

Then, using the EndEnt Option, click on the bottom, right corner of the existing rectangle.

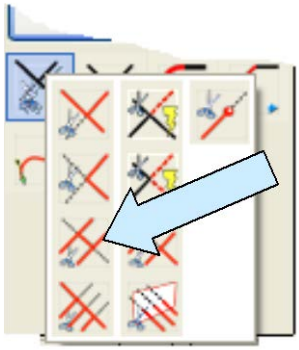


Next, click on the CREATE CIRCLE BY DIAMETER Icon.
Type 3 for the Diameter and hit the ENTER Key.

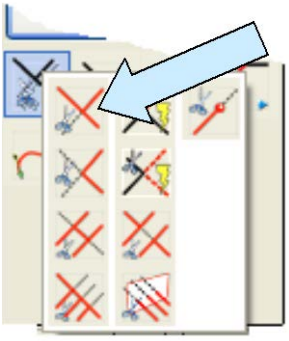
Using the EndEnt Option, click on the top, right corner of the rectangle.



Your screen should look like this:



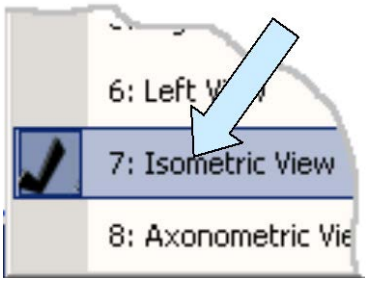
Click on the TRIM DOUBLE Icon. Click on the circle at the 8 O'Clock position and then click on the top edge of the rectangle and the right edge of the rectangle.



Next, click on the TRIM FIRST Icon.
Click on the left end of the top edge of the smaller rectangle and then on the top end of the 90 degree arc.

Next, click on the bottom end of the right edge of the smaller rectangle and then on the right end of the 90 degree arc.

Your screen should look like this:



Switch to the Isometric View. (View 7.)

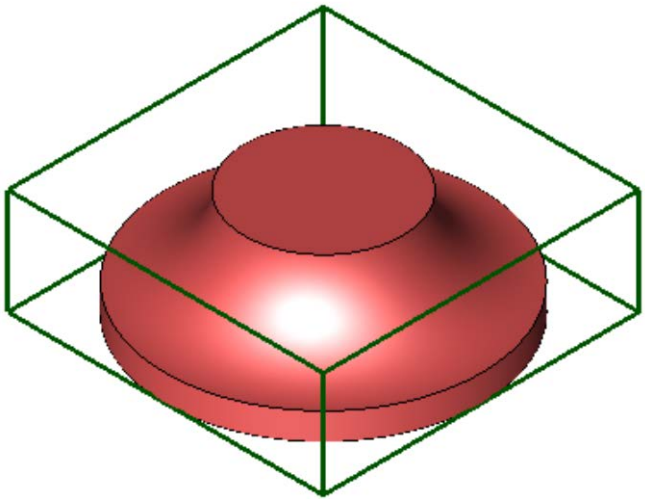
Click on the REVOLVE Icon.
A Dialog Box appears
Select the profile that you created from the trimmed smaller rectangle and 90 degree arc and hit the ENTER Key.

Next, click on the EXTRUDE Icon
A Dialog Box appears. Type 6 for the Length and click on the Midplane Option. Hit the ENTER Key.

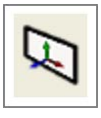
Select the large rectangle and hit the ENTER Key.

Now, click on the GENERIC BURST Icon.
Click on the large solid block and hit the ENTER Key. (Using the Extrude function to quickly make a block that we then burst into wire frame entities is a fast and efficient way to create a 3D reference grid in a file.)

Your screen should now look like this:



The geometry on the screen is all we need to create the tool paths for roughing and finishing the main body of our part. To create the spiral veins, we need to do a little more work.



Click on the CONSTRUCTION PLANE Icon and type 1 for the construction plane.

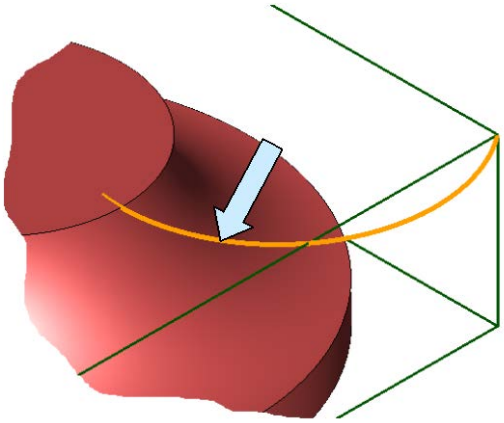
Next, click on the CREATE ARC BY THREE POSITIONS Icon.

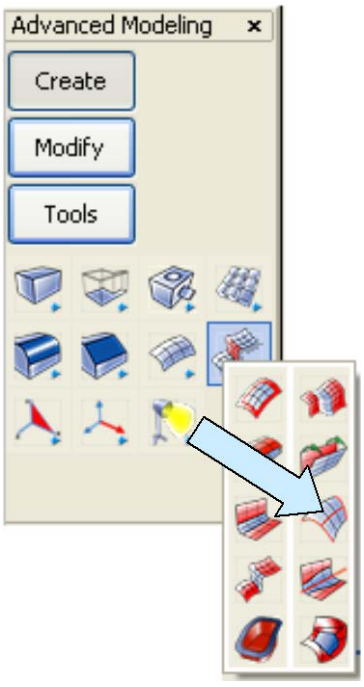
Click on the OffSet Option on the Conversation Bar.
Using the EndEnt Option, click on the top, circular edge of the revolved solid.
Type -.25 for the X value and hit the ENTER Key three times.

Use the Offset Option again from the same reference point, this time typing 0.25 for the X Value and hitting the ENTER Key three times.

Finally, backup and use the EndEnt Option and click on the top, right, rear corner of the three dimensional grid.

Your curve should look like this:



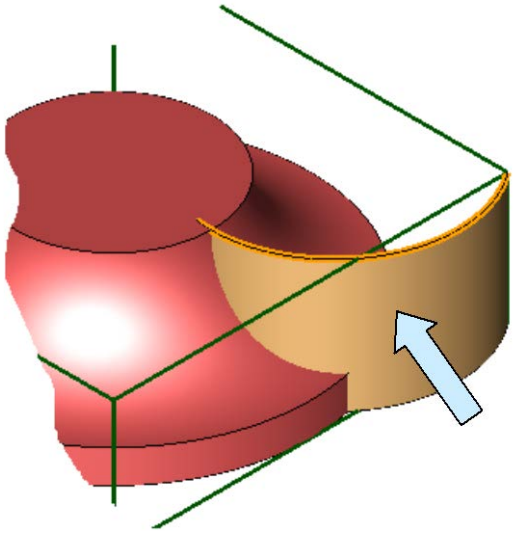


Click on the SWEPT SURFACE Icon.
A Dialog Box appears.

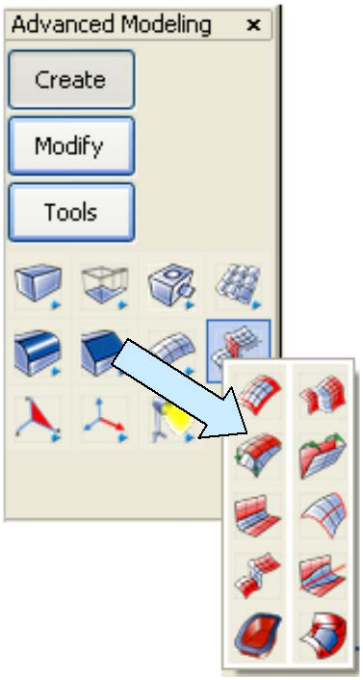
Use the One Director and One Generator Options and click on the Translate Option. Hit the ENTER Key.

Click on the arc that you just made and then, using the EndEnt Option, click on the back end of the same arc.

Next, click on the right, rear vertical edge of the grid and then, using the EndEnt Option, click on the top end of the same edge.



You will now have a vertical surface that looks like this:

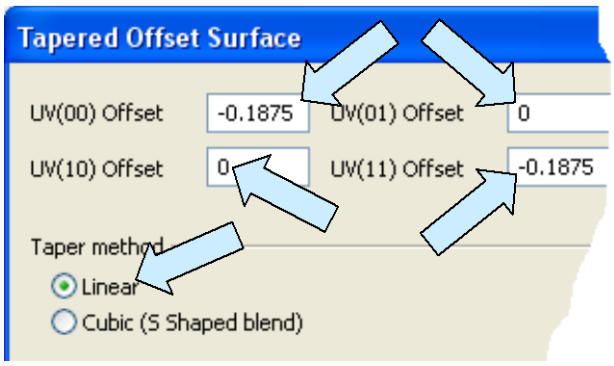


Our next task is a little more complex. We would like the tip of the 0.375 ball mill to follow a path that starts on the curved surface at the upper circular edge and that gradually drops below the surface of the part until it is at the ball mill radius (0.1875) at the outer 3 inch diameter edge.

This will give us a vein that has a semicircular profile along the entire length.

Click on the TAPERED OFFSET SURFACE Icon.

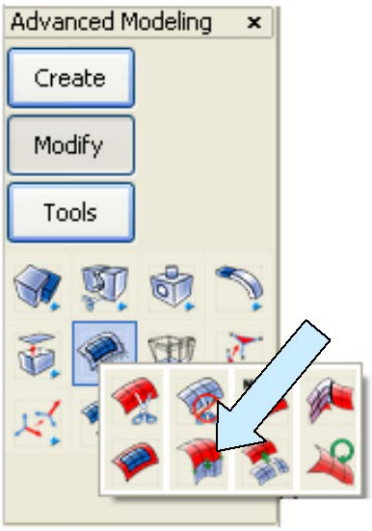
Click on the top, curved surface of the part.



A Dialog Box appears. Enter values for the four offset fields as shown in the illustration to the left.

Use the Linear Option and hit the ENTER Key.

You now have an offset surface that lies below the part.



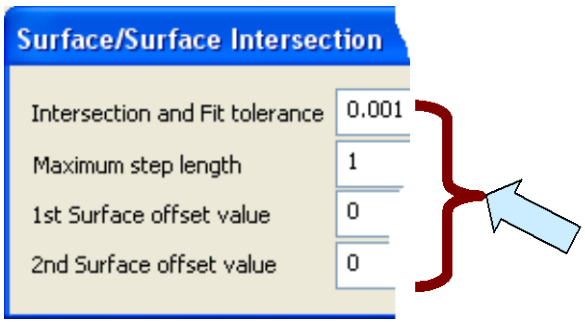
Click on the EXTEND SURFACE Icon.
A Dialog Box appears.

Use the Linear Option and type 20 for the percentage.
Hit the ENTER Key.

Click on the offset surface and then select the vector that points outward from the large, circular edge.

At this point, you might want to move the solid to a unique level. Then, remove that level from the display. (It just makes it easier to select geometry on the screen.)

Click on the CREATE SPLINE AT INTERSECTION OF SURFACES Icon.



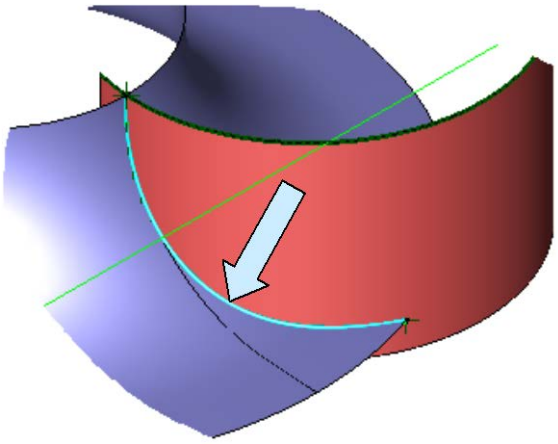
A Dialog Box appears. Use the default values and hit the ENTER Key.

Select the vertical curved surface you made a while back and hit the ENTER Key.
Then, select the offset surface you just created and hit the ENTER Key.

You will get a curve that lies at the intersection of the two surfaces. (Note: Because we extended the offset surface outward, out ball mill will flow completely off the surface it is machining prior to retracting at the end of the pass. This is another approach to ensuring clean endings on machining operations.)

You can now remove the offset surface, vertical surface, and original three position arc to a construction level in your file and remove that level from the display.

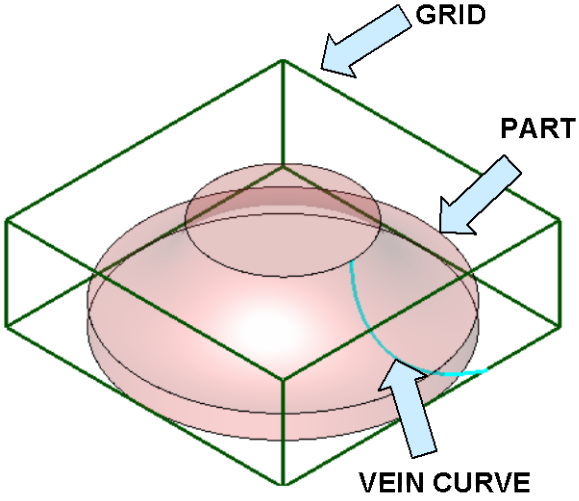
The three-dimensional curve created at the intersection of the two surfaces is the only item we'll need for creating the tool path that makes the vein.

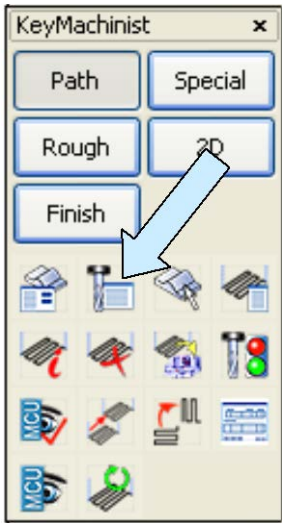


Creating the Tool Paths.

At this point you should have three items displayed on your screen.

1. Your original construction grid.
2. Your solid part.
3. The 3D curve that will define the vein tool path.



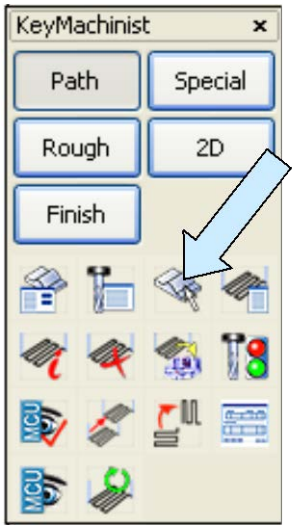


Click on the TOOL LIST Icon and define the three tools shown below. (Click for Offset Number on each one.)

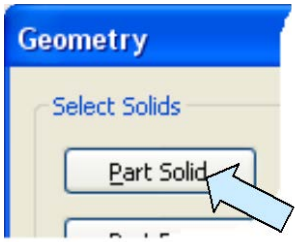
Tool Name	Tool#	Type	Diameter
FlatMill5	1	Flat End Mill	0.5
BallMill10	2	Ball End Mill	1.0
BallMill375	3	Ball End Mill	0.375

When you are done, select the 1.0 inch Ball Mill as your first tool.

Click on the SELECT GEOMETRY Icon.

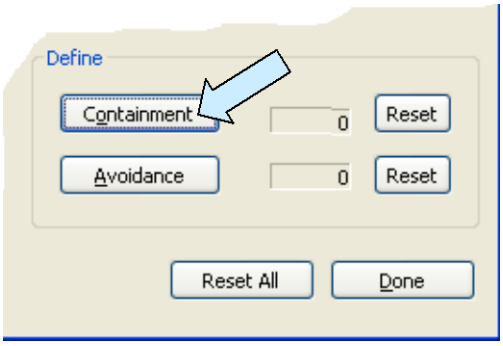


A Dialog Box appears. Click on the Part Solid Button. Click on the solid part on you screen and hit the ENTER Key.



Note: For the next operations, you should have the Restrict Chain Select and Enable Quick Chain Options selected in TOOLS/OPTIONS/SELECT.

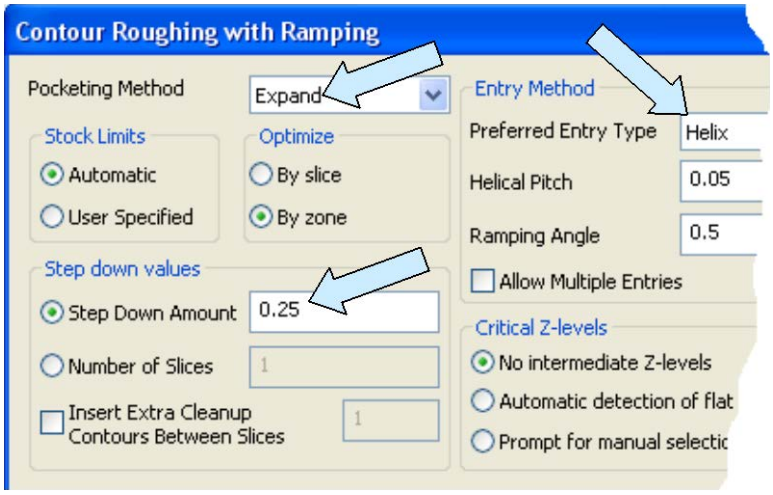
Next, click on the Containment Button. A small Dialog Box appears. Select the Contain to the Center of Tool Option and hit the ENTER Key.



Now, click on the Chain Option on the Conversation Bar. Move the cursor over one of the top grid lines and click when all four highlight. Hit the ENTER Key. Then, click on the Done Button.

Click on the CONTOUR ROUGHING Icon.

A Dialog Box appears. Type “Rough1” for the Tool Path Name and hit the ENTER Key.

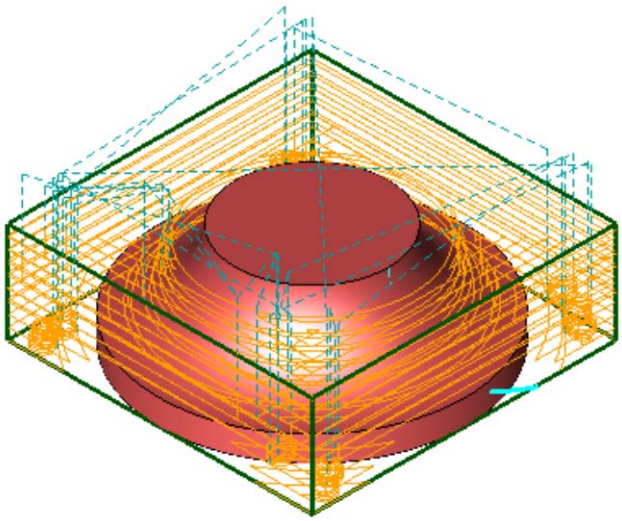


The second sheet of the Contour Roughing Dialog Box appears.

Use the Expand Option and type 0.25 for the Step Down Amount.

We'll use a Helix Entry. Hit the ENTER Key.

Your tool path should look like this:



Next, click on the 2D PROFILE Icon.

The first sheet of the Profile Milling Dialog Box appears. Click on the Select New Tool Button and select the 0.5 diameter Flat End Mill. Then, click on the NEXT Button.

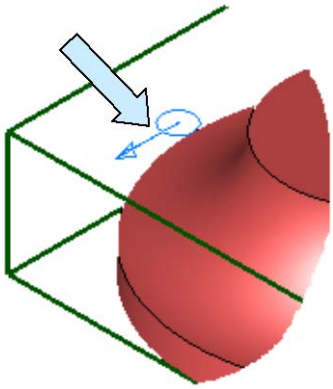
Sheet 2 of the Profile Milling Dialog Box appears. Click on the Select Profile Button.

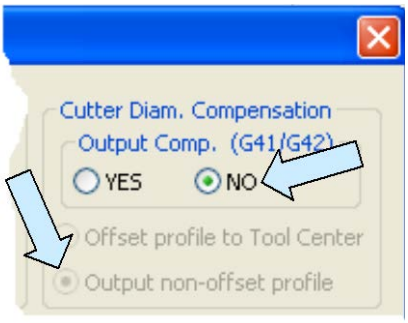
Using the Single Option, click on the top, outer circular edge of the solid part and hit the ENTER Key.

You are asked if you want to reverse the machining direction. Click on the NO Button.

You are next asked what side of the profile to cut on. Click on the Right Option.

For the Entry Point select the Mid Option and then hit the ENTER Key.





For the Cutter Diam Compensation click on Output non-offset profile and click on the NO Option. **(Note: If you had used the On Curve Option in your previous step, you would then leave these settings on YES and Offset Profile to Tool Center.)**

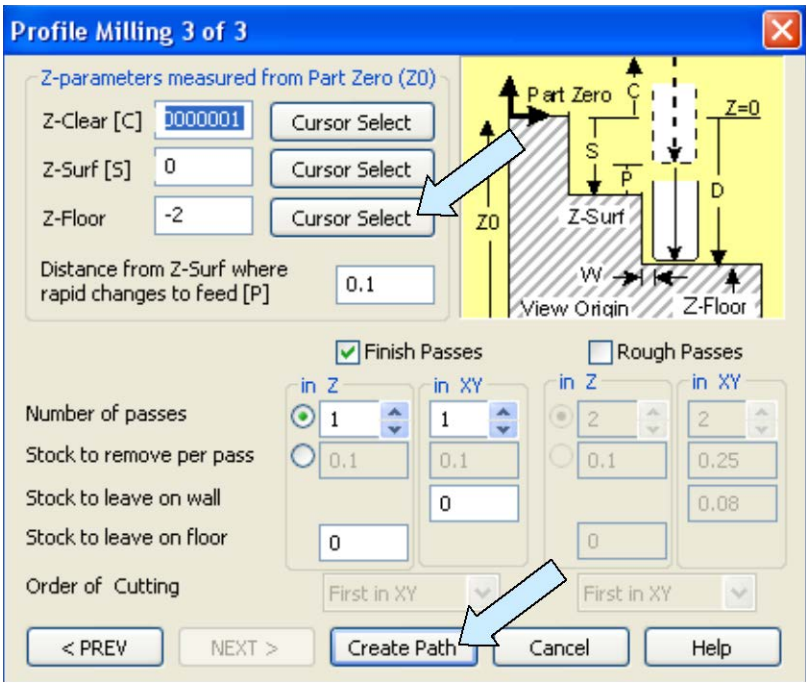
Click on the NEXT Button.

Sheet 3 of the Profile Milling Dialog Box appears.

For the Z-Floor, click on the Cursor Select Button.

Using the EndEnt Option, click on the bottom, outer, circular edge of the solid.

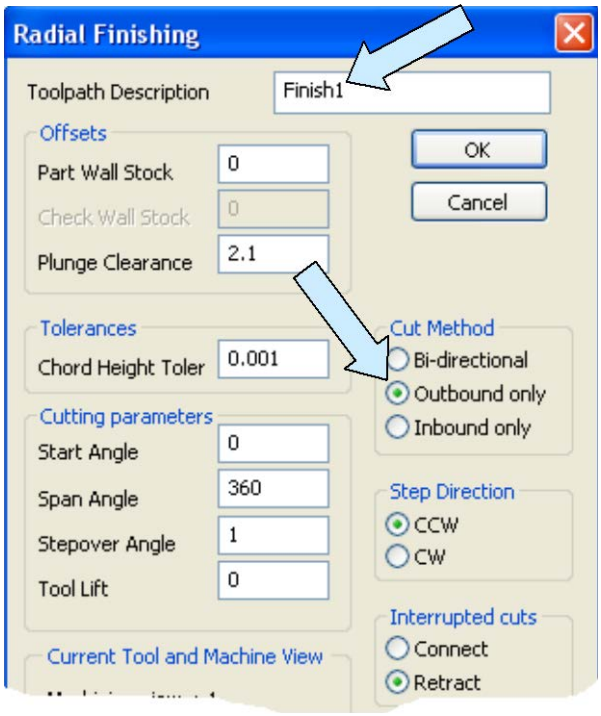
Then, click on the Create Path Button.



A small Dialog Box appears. Type “EdgeClean” for the name and hit the ENTER Key. You now have a second tool path.

Now, let's create a finishing Tool Path to smooth out the curved top surface of the part.

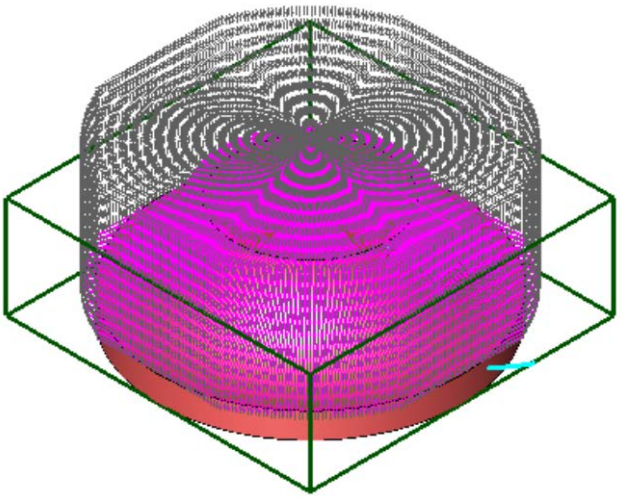
Click on the RADIAL FINISHING Icon.



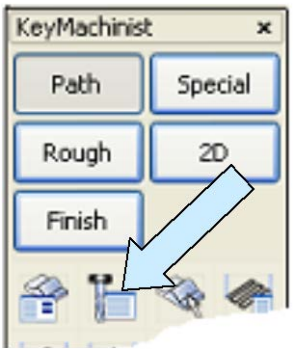
The Radial Finishing Dialog Box appears. Type "Finish1" for the Tool Path Description.

Click on the Outbound Only Option and hit the ENTER Key.

You are prompted to select a pivot point. Using the CtrMid Option, click on the uppermost circular edge on the part.



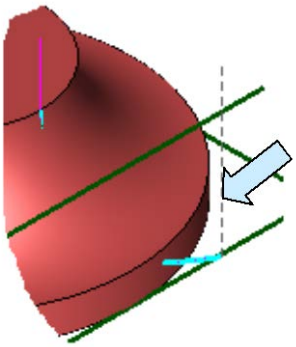
Your tool path is generated.



To create the vein tool path, click on the TOOL LIST Icon and select the 0.375 Ball Mill.

Next, click on the GEOMETRY TO TOOL PATH Icon.

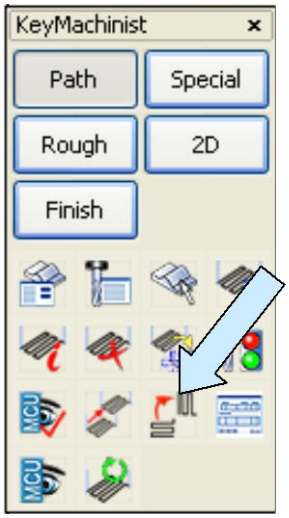
A Dialog Box appears. Type “Vein1” for the Tool Path Name and hit the ENTER Key.



Using the Single Option, click on the upper end of the spline that you created at the intersection of the vertical surface and curved surface earlier in the exercise. Hit the ENTER Key.

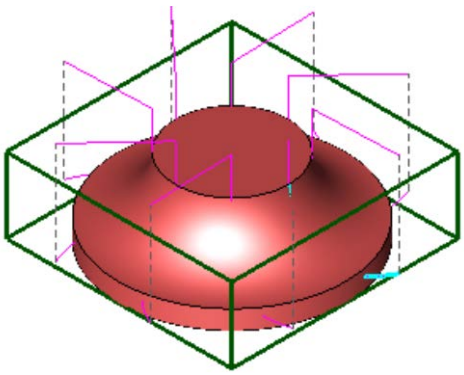
Click NO for the Reverse Machining Direction Question.

Your tool path is generated.



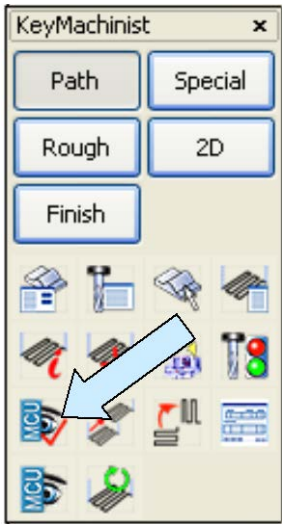
Since we need a total of eight veins, click on the ROTATE TOOLPATH Icon.

Select the Vein1 tool path and hit the ENTER Key.
Type 7 for the Number of Copies. Click on YES to combine the copies into one tool path. Type 45 for the Rotation Angle.



Using the CtrMid Option, click on the top, circular edge of the solid to indicate the rotation point.

Your complex tool path should look like this:



Click on the VERIFY TOOLPATH Icon.

Select the four tool paths in the following order:

1. **Rough1**
2. **EdgeClean**
3. **Finish1**
4. **Vein1**

Use the Corners Option and using EndEnt, click on the top, front, left corner of the grid and the bottom, right, rear corner of the grid. Then, click on the SKIP Button.

The Meta Cut Utility Screen opens. You can play the animation. You might want to start with the slider at least three-quarters over to the right because of the complexity of the tool paths.