# Carvey G-Code Generation

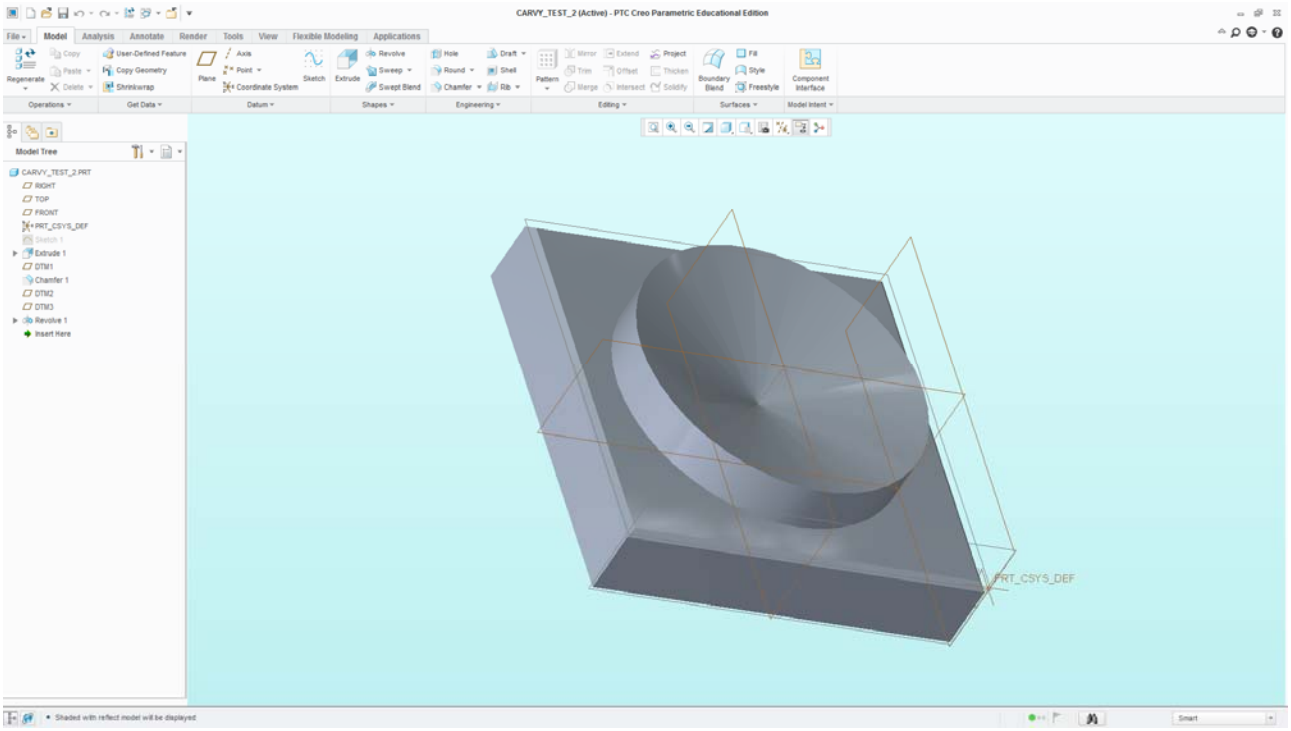
Last Updated 07/2016

## 0. Set up Creo

Before starting any project in Creo, it is good practice to:

- Set your working directory to C:\Temp
- Ensure that you are working in your intended units.

## 1. Make a part to be sent to the Carvey in Creo.

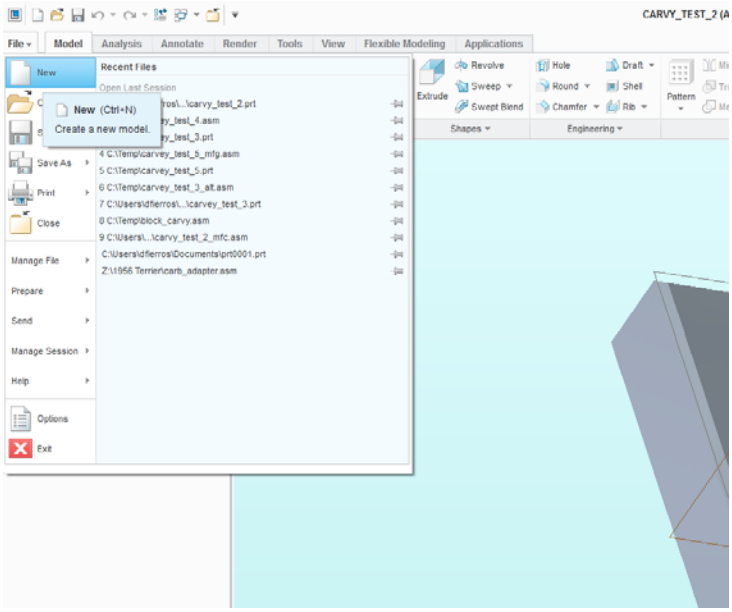


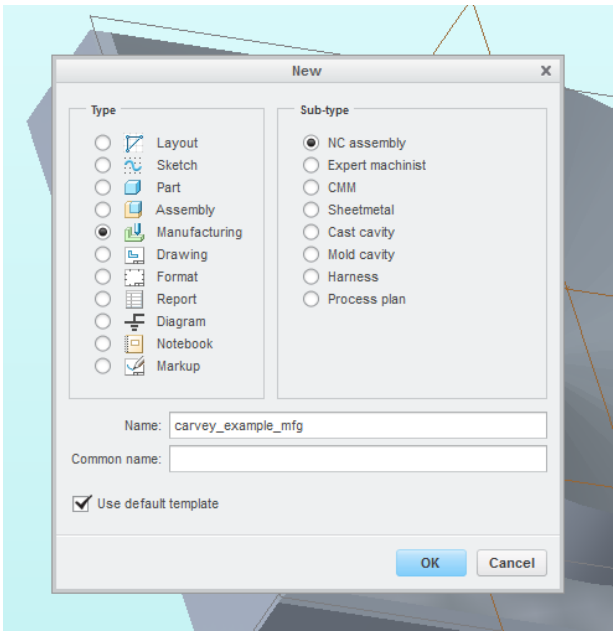

See some of Glen's tutorials if you need help with the basics of part design.

Part restrictions:

- Part must fit within an 11.6" X 8" X 2.75" (X, Y, Z) space.
- Metal should only be etched, so **cuts of more than 0.125"** into metal can damage the Carvey and therefore **should not be attempted.**
- The part should not take up the whole work area, and should avoid all safety clamps inside the Carvey.

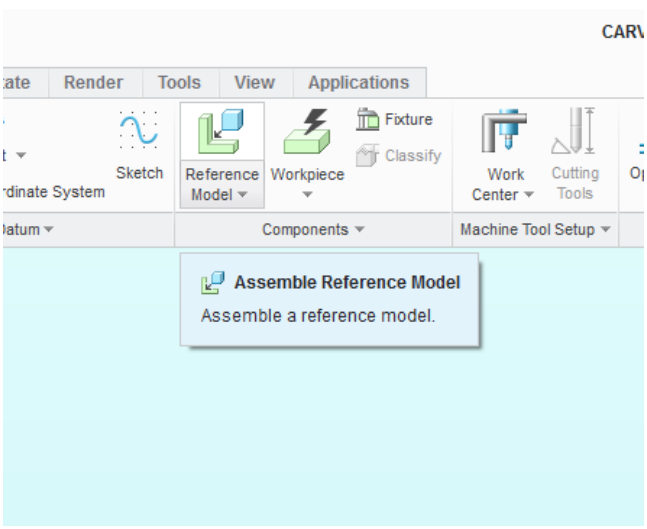
## 2. Create a new manufacturing model

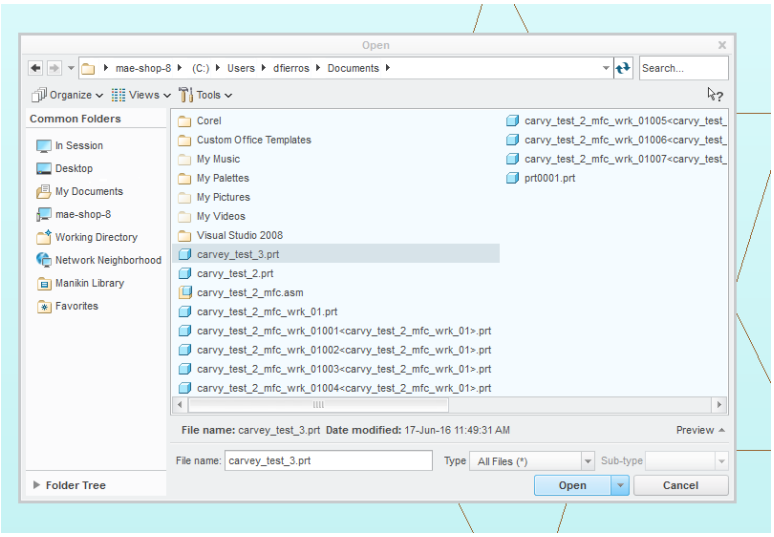



It is good form to include "_mfg" in the name of any manufacturing model. Take note that simply creating the file does not save it.

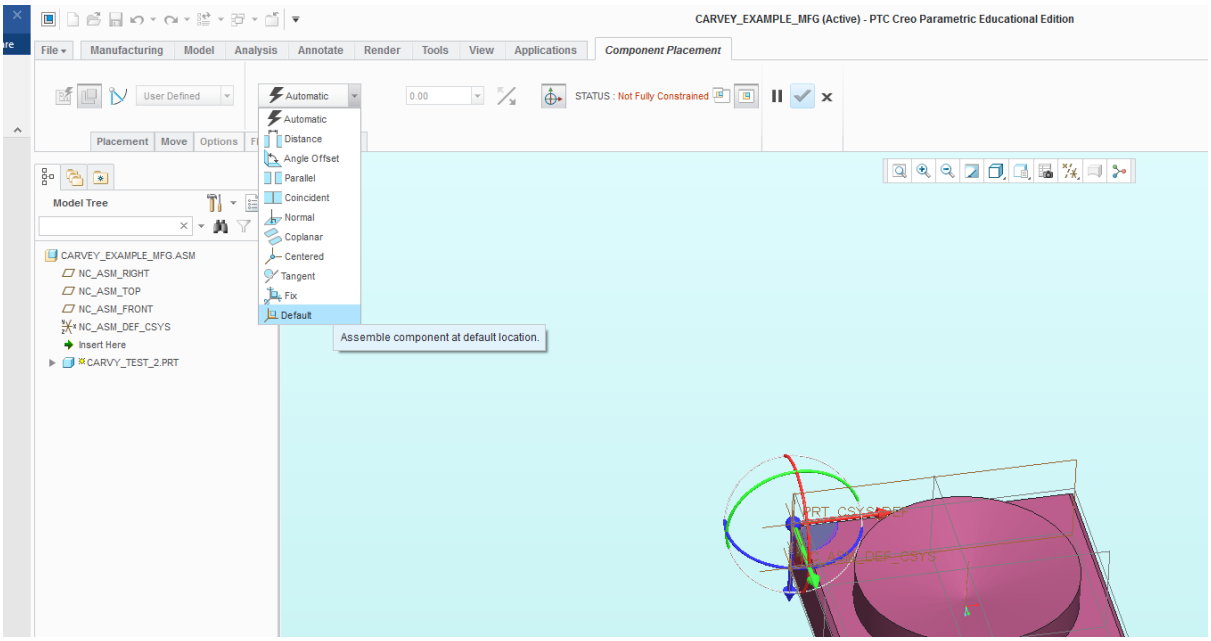
## 3. Place the part in the manufacturing model



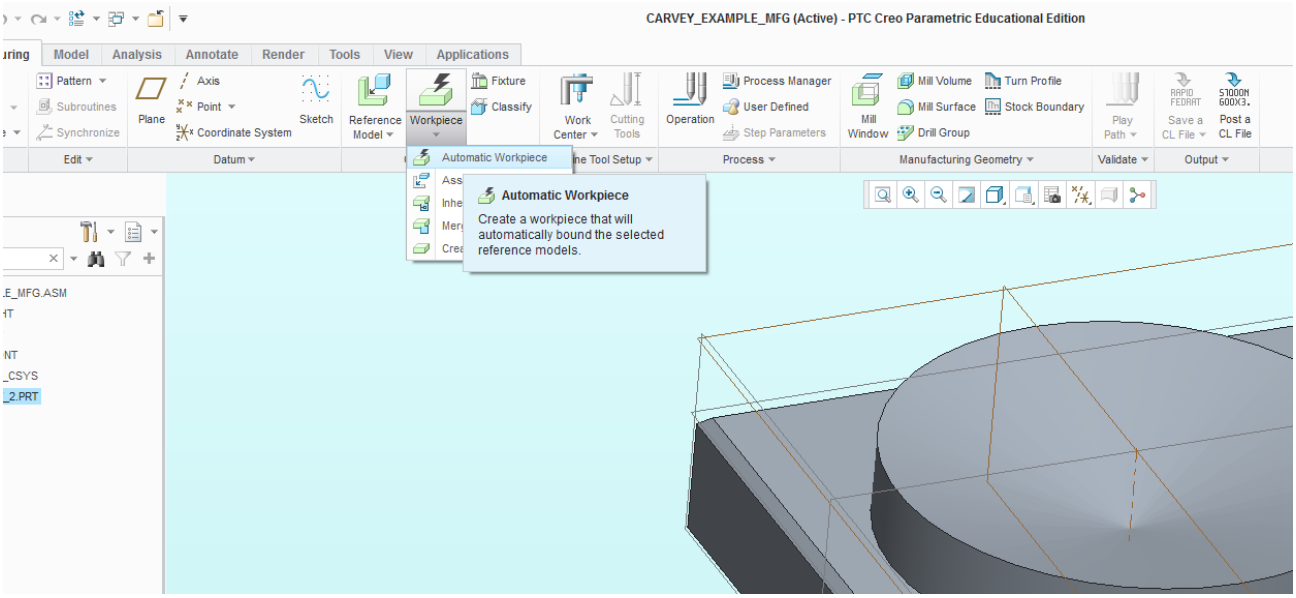

Select the part to be machined from the pop-up window and it will appear in the manufacturing modelspace.

## 4. Constrain the model

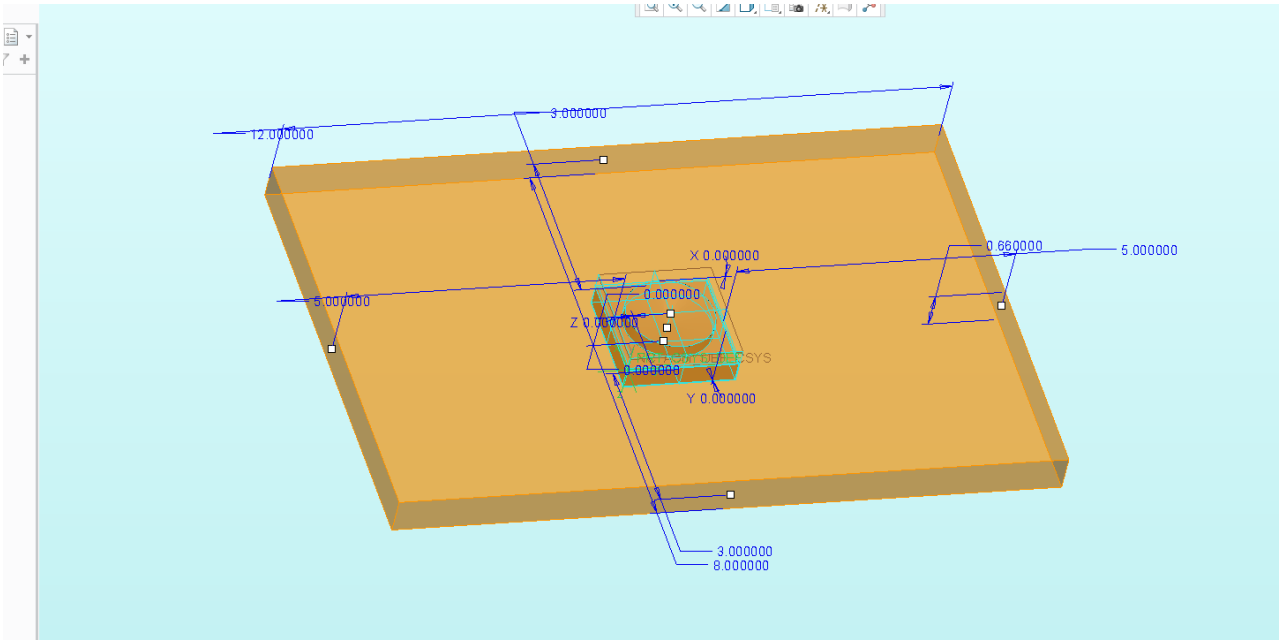



Select the "default" option on the component placement dropdown menu, you will add a more specific coordinate system and set of constraints later.

## 5. Create and position the workpiece




First, simply make the workpiece by clicking on "Automatic Workpiece" from the "Workpiece" dropdown menu.

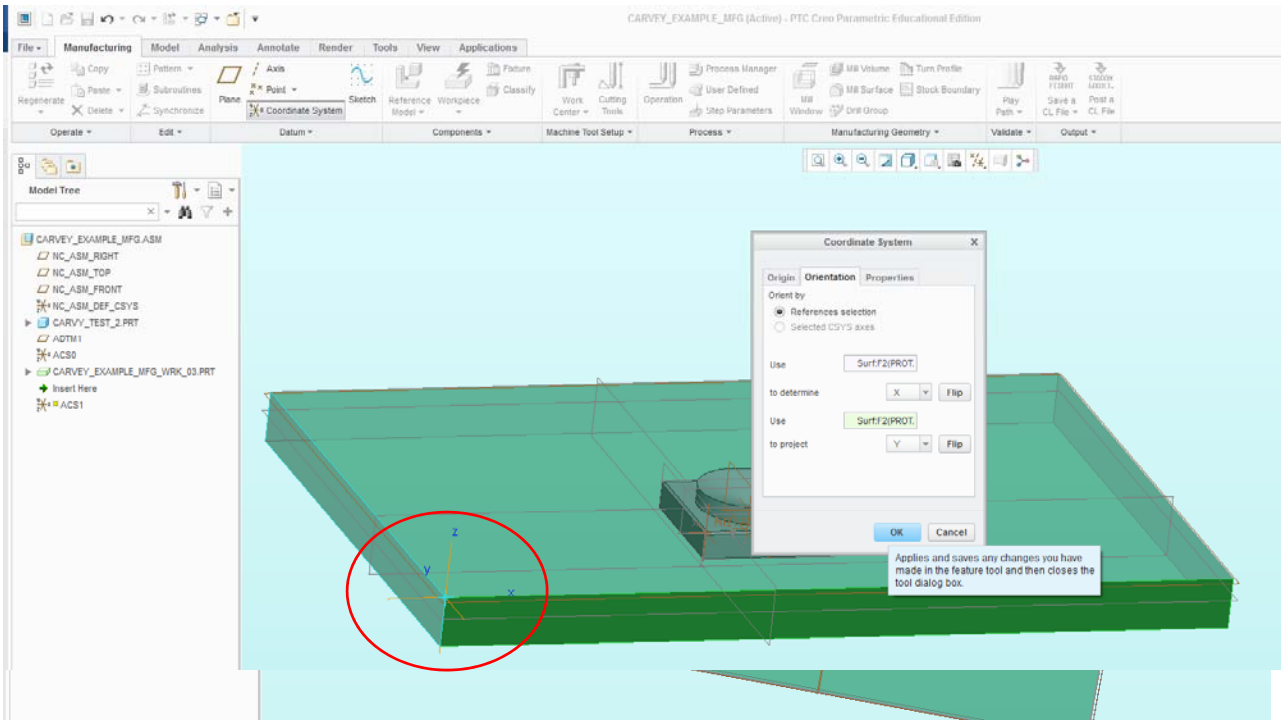



Then, adjust the workpiece dimensions until it matches the dimensions of the stock material you are using by clicking on the blue dimensions and editing them. Make sure to position the part where you want it cut out of the workpiece.

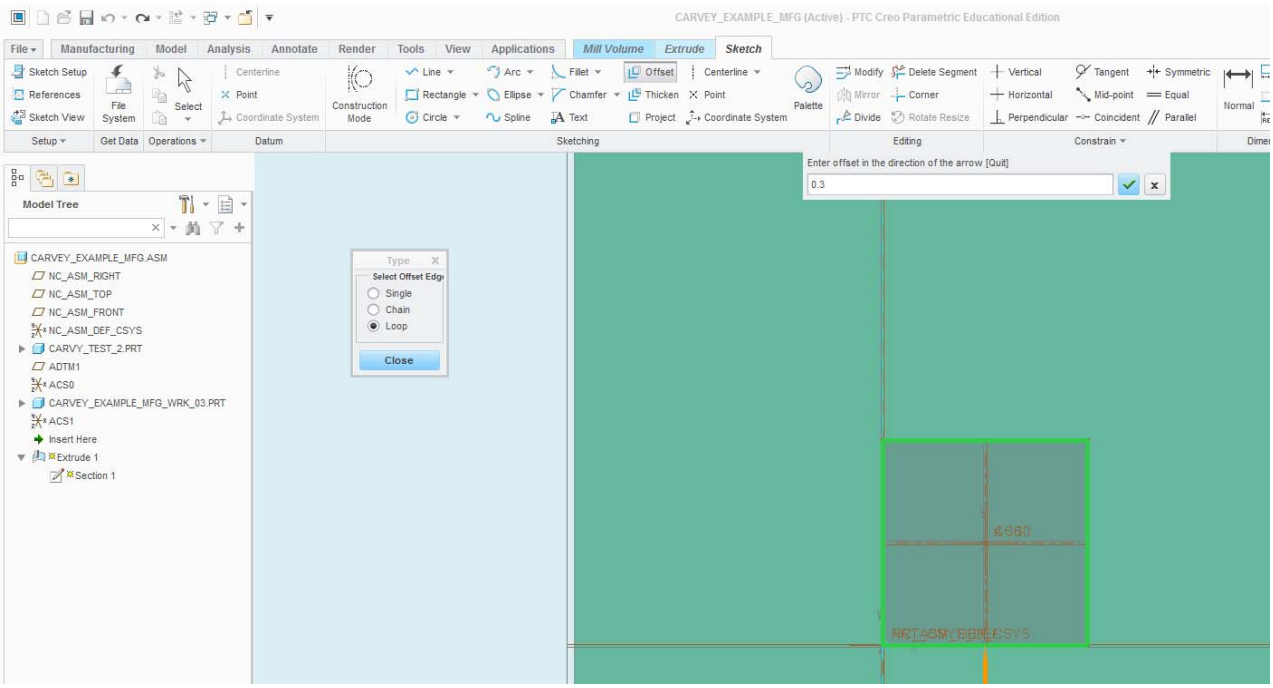
Note: The Carvey bed can fit wood with maximum dimensions of about 11.6" X 8" X 2.75" (x, y, z).

## 6. Add a coordinate system to the workpiece

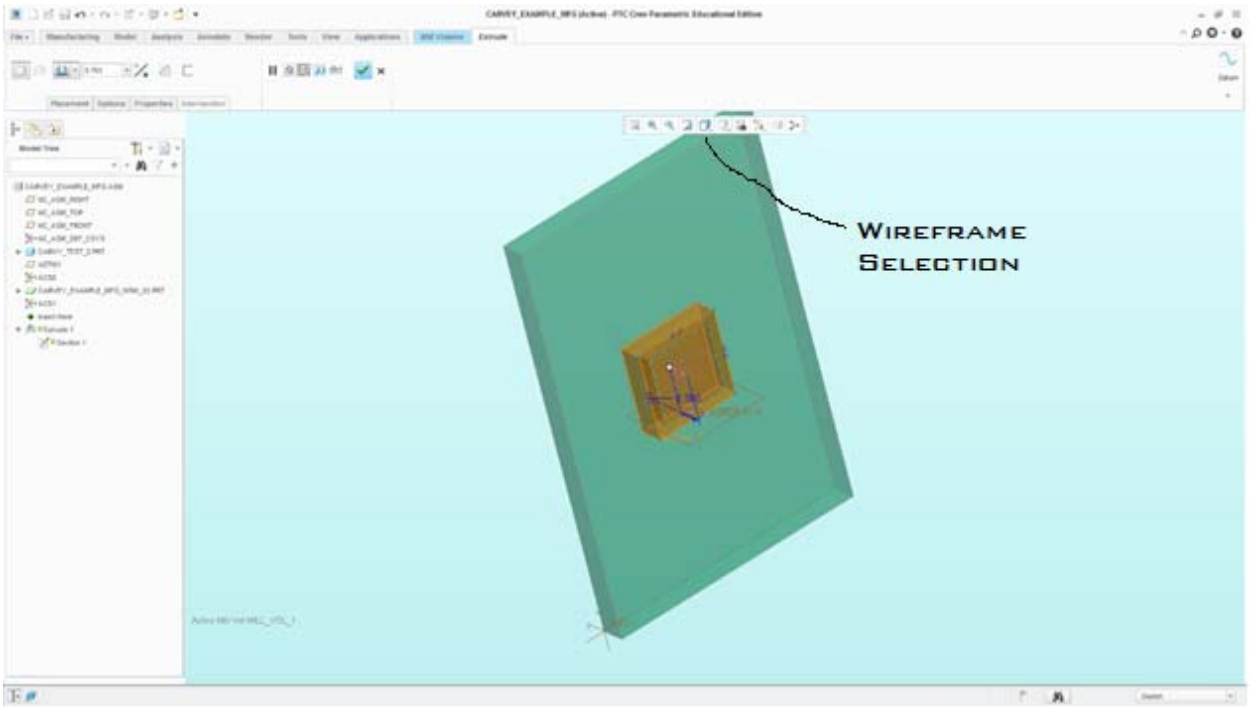
**The origin should be in the bottom left part of the top face of the workpiece.**




**The mill volume will be defined by an extrusion from the face of the workpiece**




In order to make the most efficient use of material, use the "Offset" tool loop option with an offset of about **0.3** in the sketcher for the extrusion. (If this is too difficult, simply make a rectangle cutout of the material.)




Now we must define a mill volume, which is the only part of the workpiece that the mill will cut.

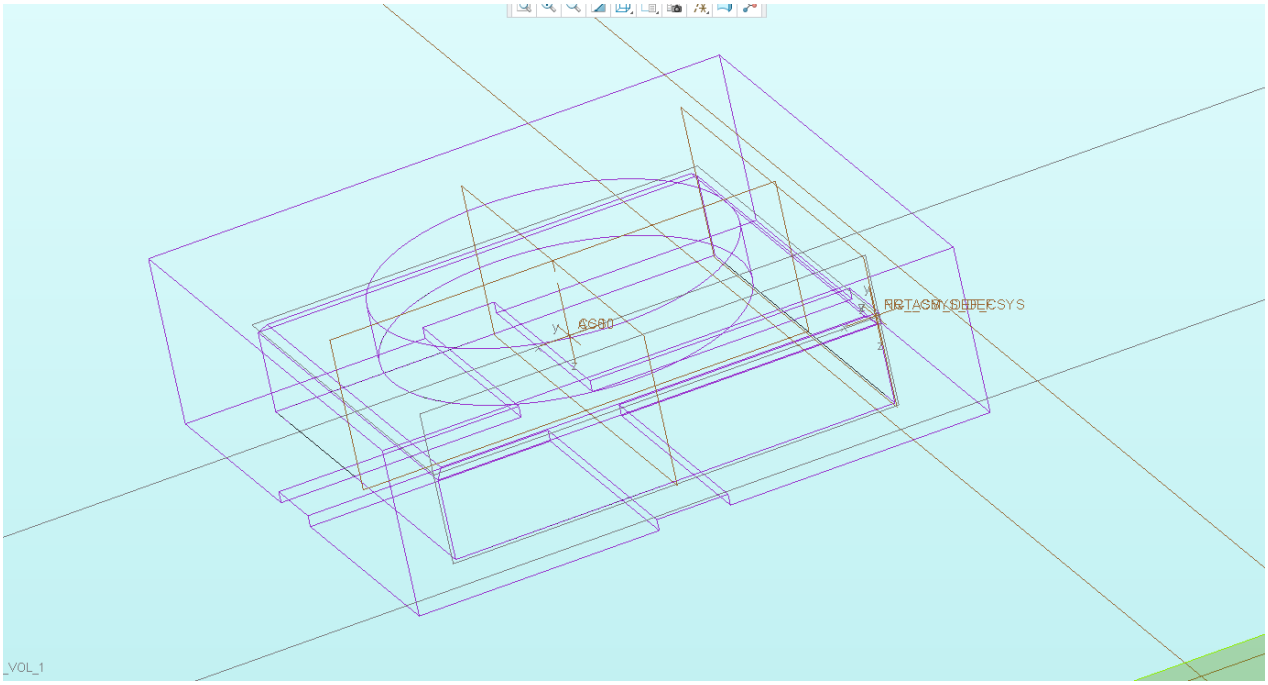
Position he mill volume until it surrounds the part, at this time, it can be helpful to set the view mode to **wireframe** as the mill volume is opaque.

## 8. Trim the mill volume

Before continuing, you must trim the part from the mill volume or else the Carvey will simply **ignore your part when cutting**. Do this by right-clicking on the mill volume you have just made in the feature list, clicking redefine mill volume, and using the trim command to remove your part from the mill volume. You should see something like this in the feature list:


Now, if you intend to cut your part out of the material you are putting in the Carvey, you will need to trim struts from the mill volume so that the part does not move. There is no prescribed method for this, but you will need to again redefine the mill

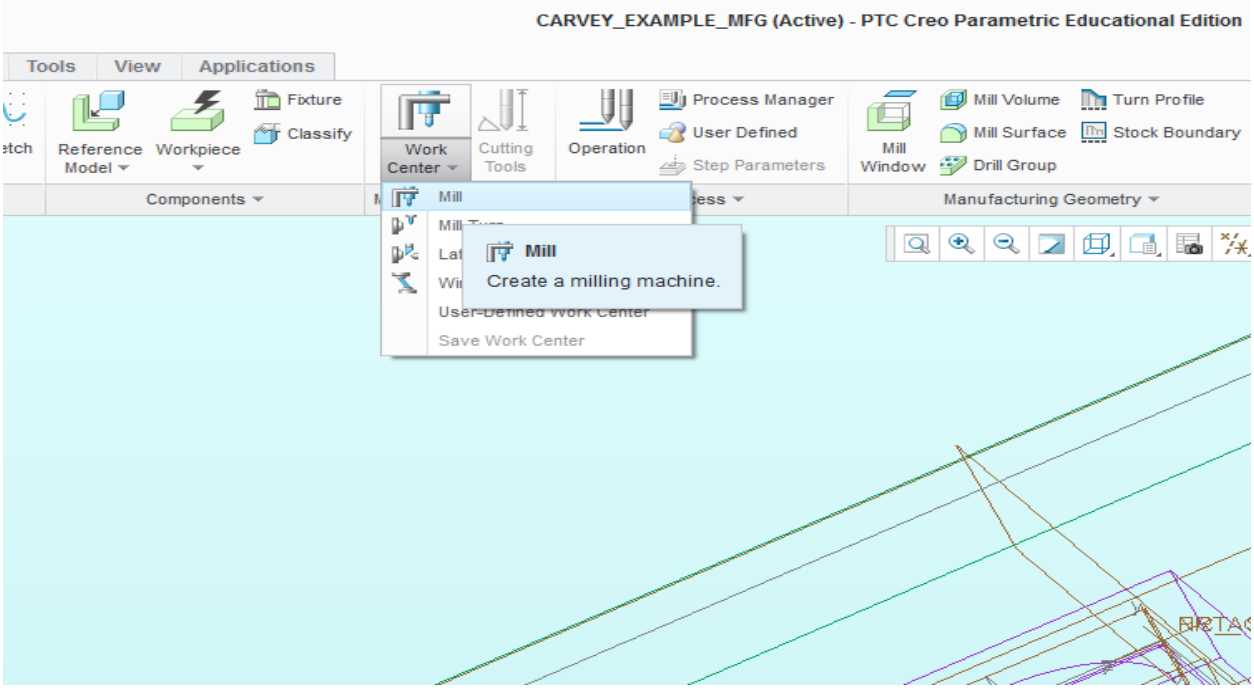
volume, this time extruding a cut from the bottom surface of your workpiece. Your struts should look something like this:



This may be unnsessary, but it is vital if you want to cut your piece completely out of the stock.

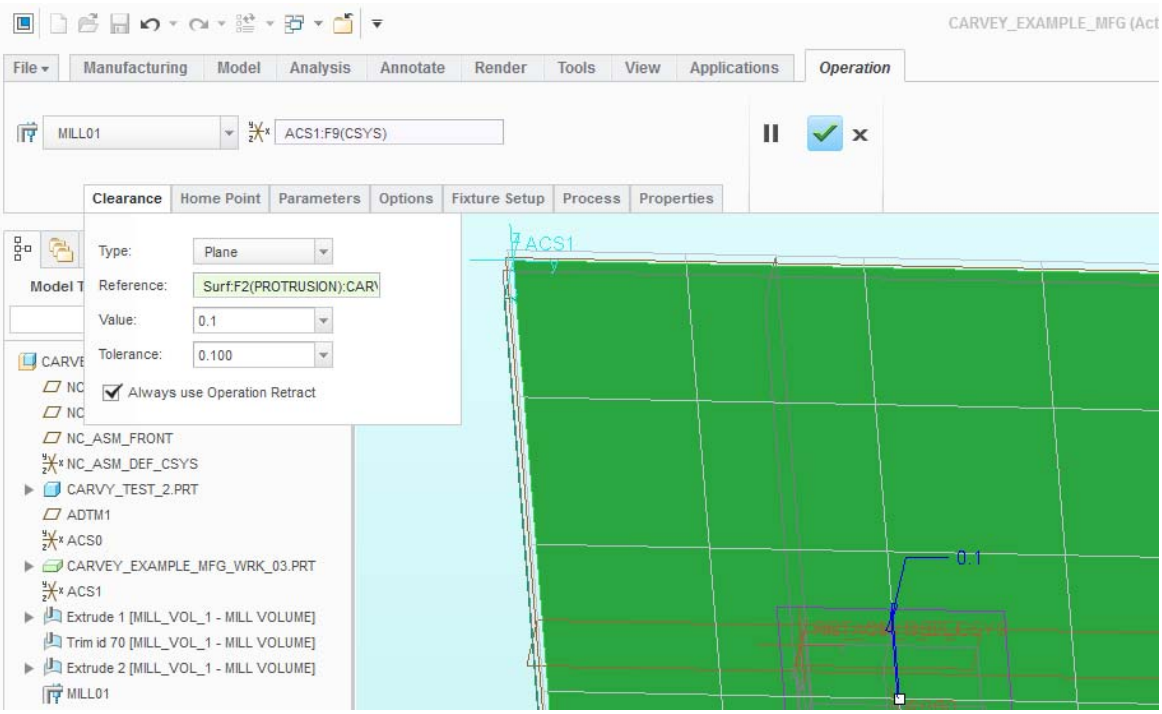
## 9. Add a mill

When the dialog box pops up, simply hit OK. This just gives Creo information about how you intend to cut the piece.



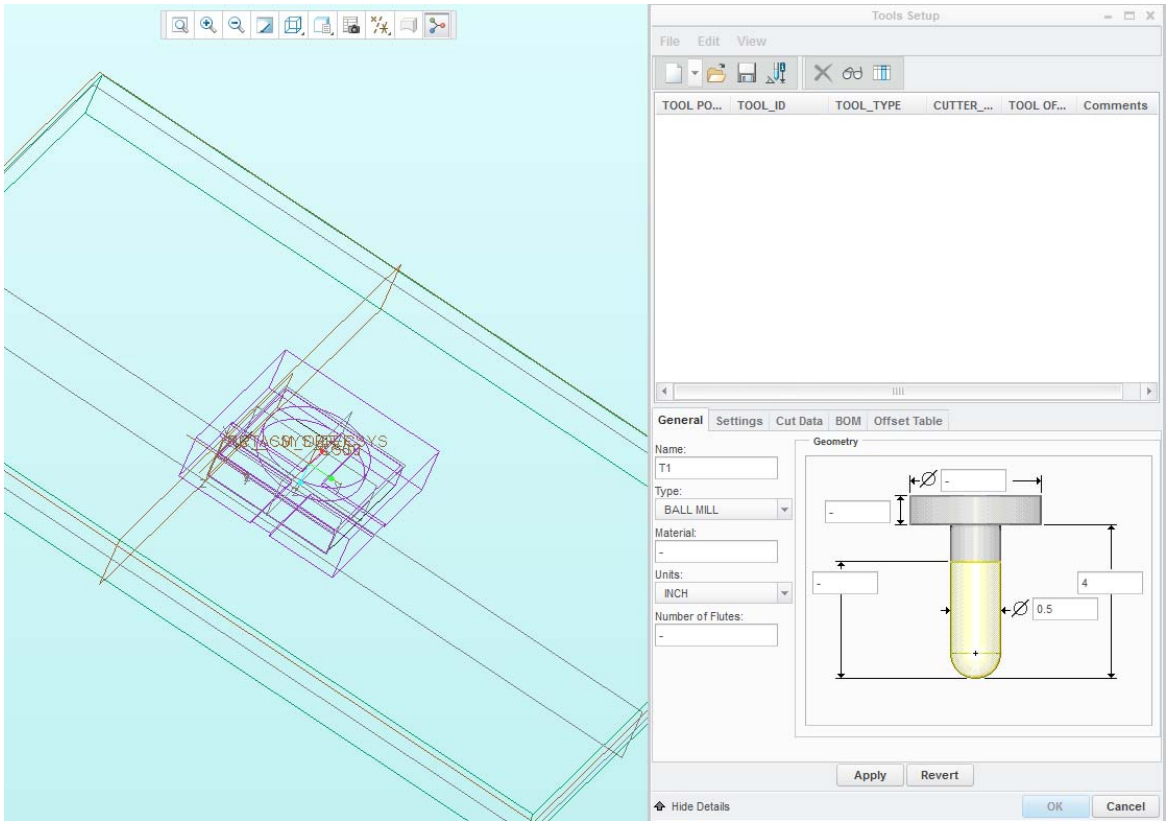
## 10. Configure the mill

Click Operation in the Manufacturing tab. Then, select the coordinate system you made earliar as the reference home position. In addition, you will need to open the Clearance tab (while still in the Operation window) and set a **plane clearance** on the **top face** of your workpiece of **0.1** so that the mill will not drag through the part.



## 11. Volume Roughing

Now click on the "Mill" tab, and in the "Roughing" dropdown menu select "Volume Rough". At this point you will need to add the dimensions of your cutting tool. If you are doing cuts with two different tools, you will need to make a separate set of G-code for each cut.

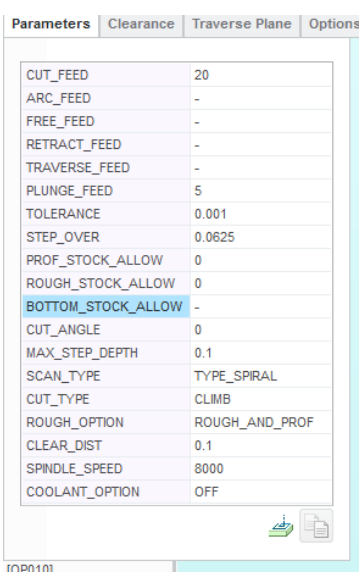



In the above picture the dimensions of a simple ball-end mill have been entered.

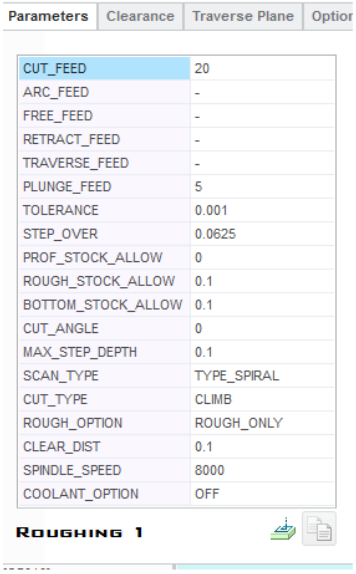
Hit "Apply" when your tool is set up, then hit "Ok".

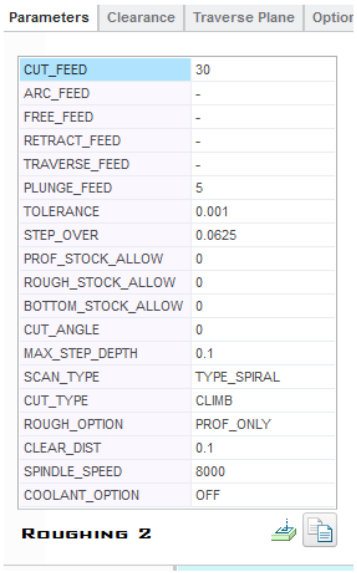
After entering the tool information, click the "Reference" tab and simply click on your mill volume. Then in the parameters tab you will need to make the following entries:

- "Cut Feed" should be set to **20** (Feet per minute)
- "Plunge Feed" should be set to **5** (Feet per minute)
- "Step Over" should be set to **half the diameter of your cutting tool**
- "Max Step Depth" should be set to **0.1** (Inches)
- "Scan Type" should be set to **Spiral**
- "Clear Distance" should be set to **0.1** (Inches)
- "Spindle Speed" can vary, at the least the speed should be **8000** (RPM) but can be as high as **11000** (RPM) (Consult Glen in the machine shop if unsure).




*Regular Configuration I*




*Alternate Configuration I*

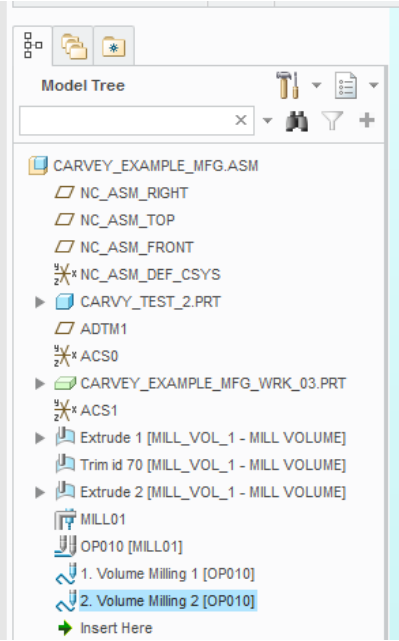
Alternate configuration for a better (but slightly slower) cut:

- **All of the above configurations**
- "Rough Stock Allowance" should be set to **0.1** (Inches)
- "Bottom Stock Allowance" should be set to **0.1** (Inches)
- "Rough Option" should be set to **Rough Only**
- **Make another Volume Milling and enter all of the original configurations**
    - "Cut Feed" should be set to **30** (Feet per minute)
    - "Rough Stock Allowance" should be set to **0**
    - "Bottom Stock Allowance" should be set to **0**
    - "Rough Option" should be set to **Profile Only**

**See the Appendix I for more information about machine settings**

## 12. Surface Milling

Before beginning this step, this is a good time to check to make sure the manufacturing sequence is correct so far. Your model tree should look something like this:



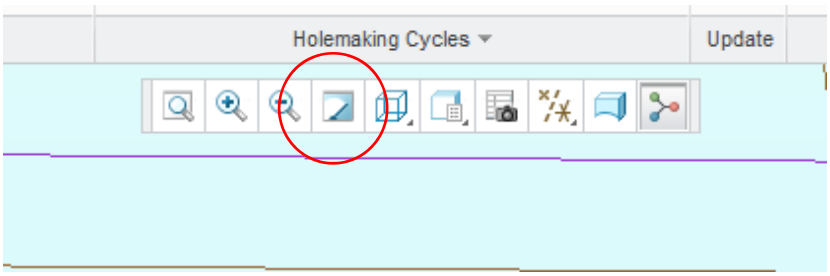

(Depending on which configuration you are using, you will have one or two volume milling entries)

As an additional tool, you can right click on the milling entry(ies) and select "Play Path" to simulate the cut—use this to make sure that the volume milling is acting as expected.

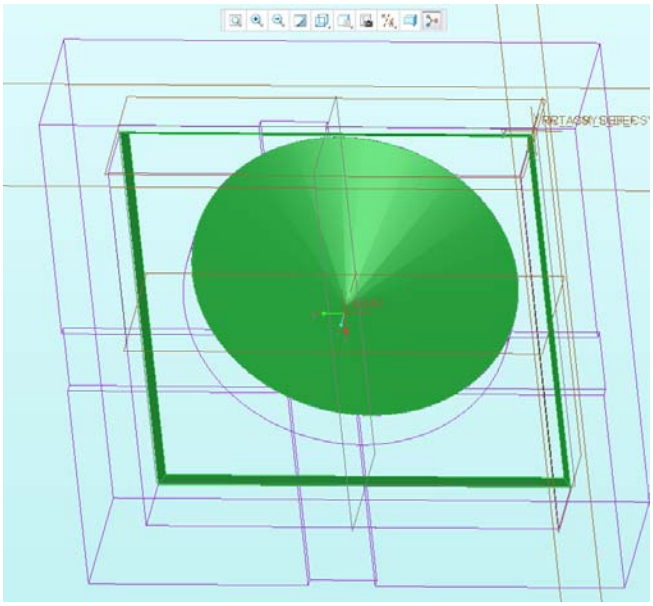
Now, in the "Mill" tab, select "Surface Milling" and click "Done" in the pop-up window. In the parameter window that pops up, make the following changes:

- "Cut Feed" should be set to **30** (Feet per minute)
- "Plunge Feed" should be set to **5** (Feet per minute)
    - You may need to select the "All" tab to see this option now
- "Step Over" should be set to anywhere from **0.030** to **0.1** (Inches). A smaller number will take much longer but will look better.
- "Clear Distance" should be set to **0.1** (Inches)
- "Spindle Speed" should be set to **8000** (RPM)

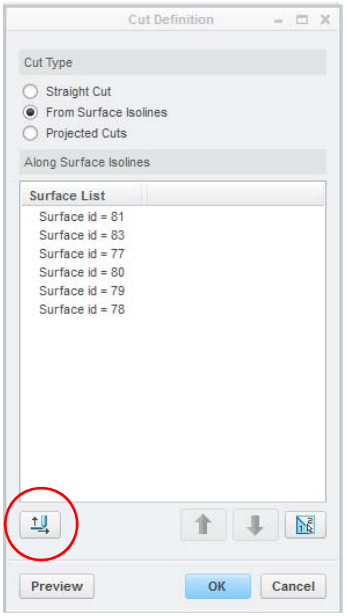
In the next tab, select "Mill Volume" and select the mill volume. Then, hit "Done". In the next tab, select the mill volume again then hit the "Repaint" icon in the small toolbar above your part.



Now you will need to select any surface you want to be detailed by the surface milling process. This should include all slanted and curved surfaces that the volume roughing will not shape to an exact degree. To select surfaces, hold control and select the surfaces you want. Make sure to right click to cycle surfaces if you want to select another layer (play around with this to figure it out). Your finished product should look something like this:

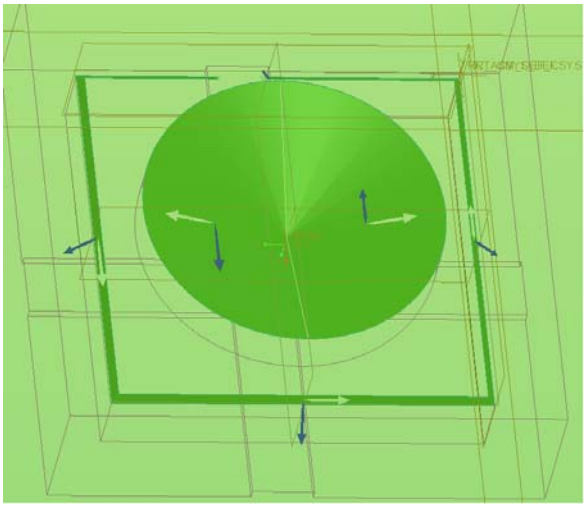


Once all of your surfaces are selected, hit "Done" and then "Done" again. When the tab labeled "Cut Definition" pops up, you will need to select the "From Surface Isolines" option. You should see something like this:




Now you will need to **click on each Surface id** and click the **redirict icon** (circled at left). When finished, your part will look something like this:

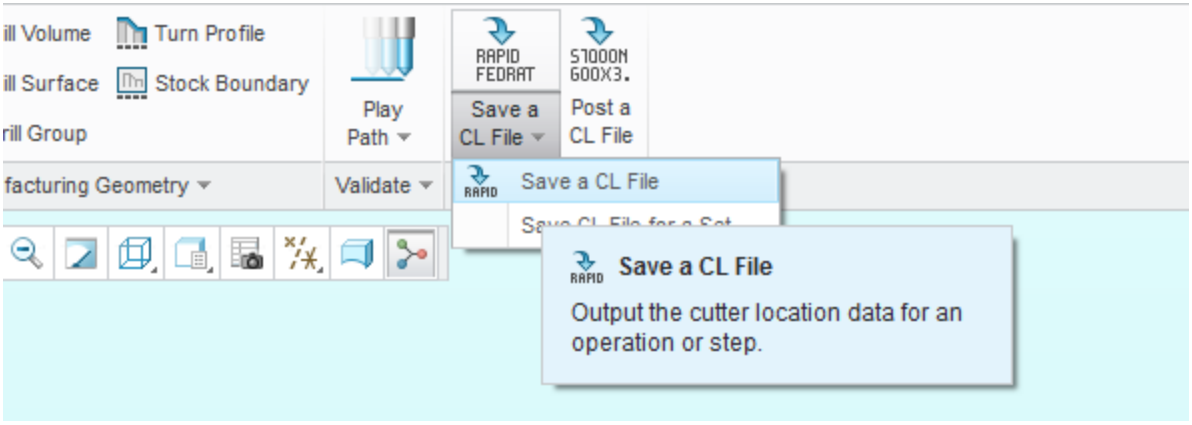
(This is just one of those Creo things)



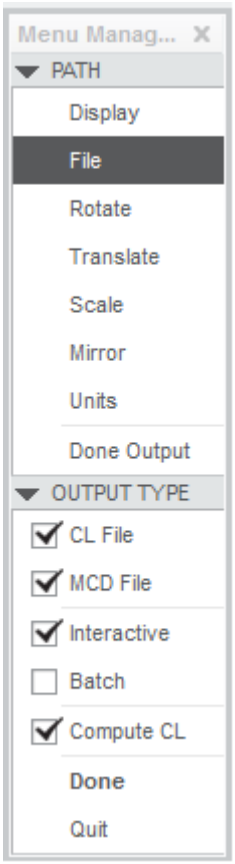
Hit "Done Seq" when finished.

## 13. Post Processing

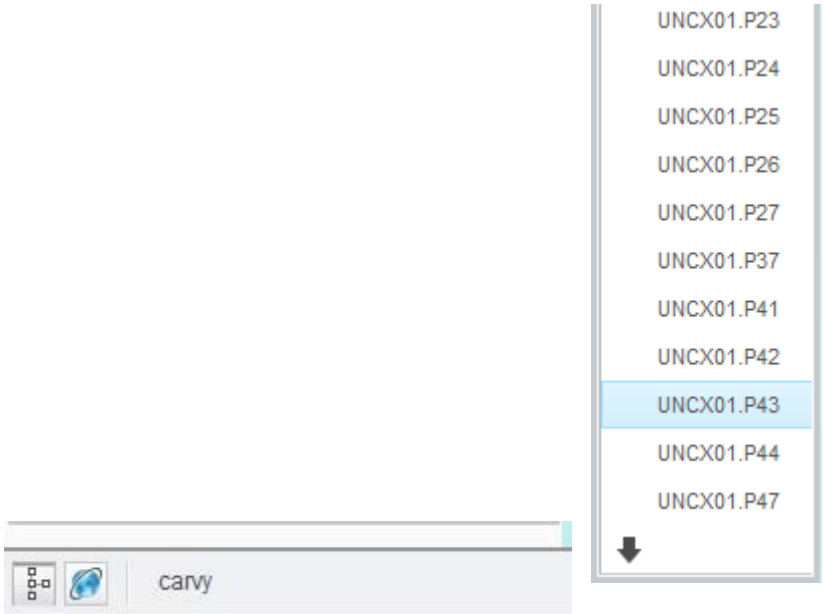
At this point, you are ready to export the manufacturing model to G-Code that the Carvey can use. Make sure to **save your work**, as post processing can crash Creo and you likely haven't saved recently. Now in the "Manufacturing" tab, click "Save a CL File" and select it in the dropdown menu.



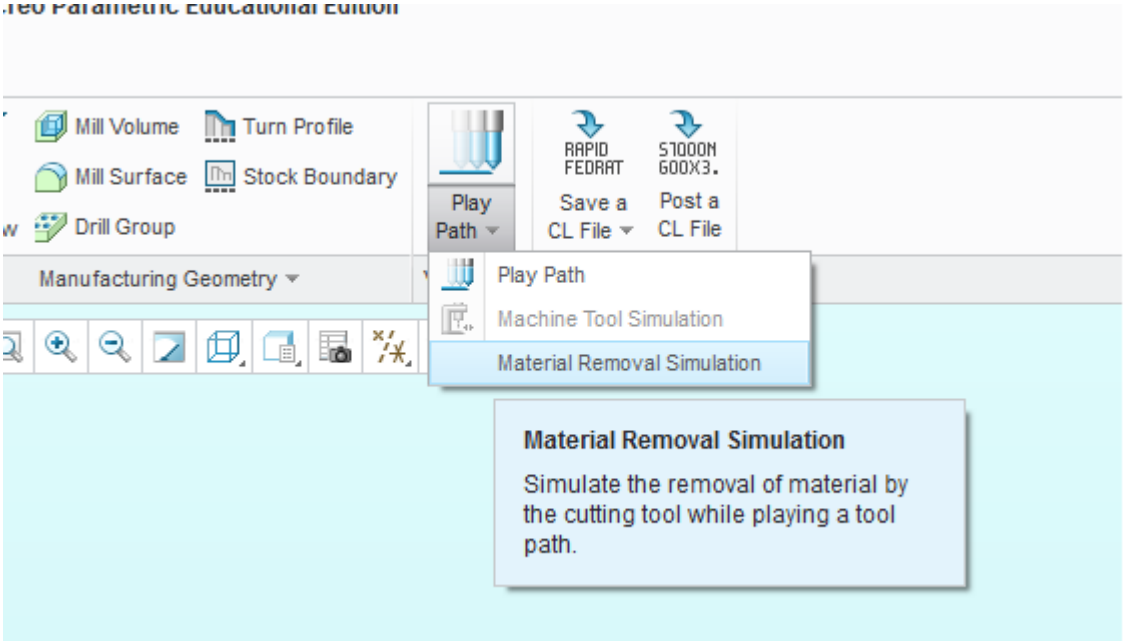

In the tab that pops up click "Operation" and then in the lower tab click "OP010". In the next window, click "File" and then select both **MCD File** and **Compute CL**. The selection plane should look like this:




Then simply name your output file and save it. In the next window that pops up hit "Done" again, then a list of post processors should pop up. The Carvey post processor should be labeled "UNCX01.P43", and when you mouse over it the bottom bar should read "carvey":



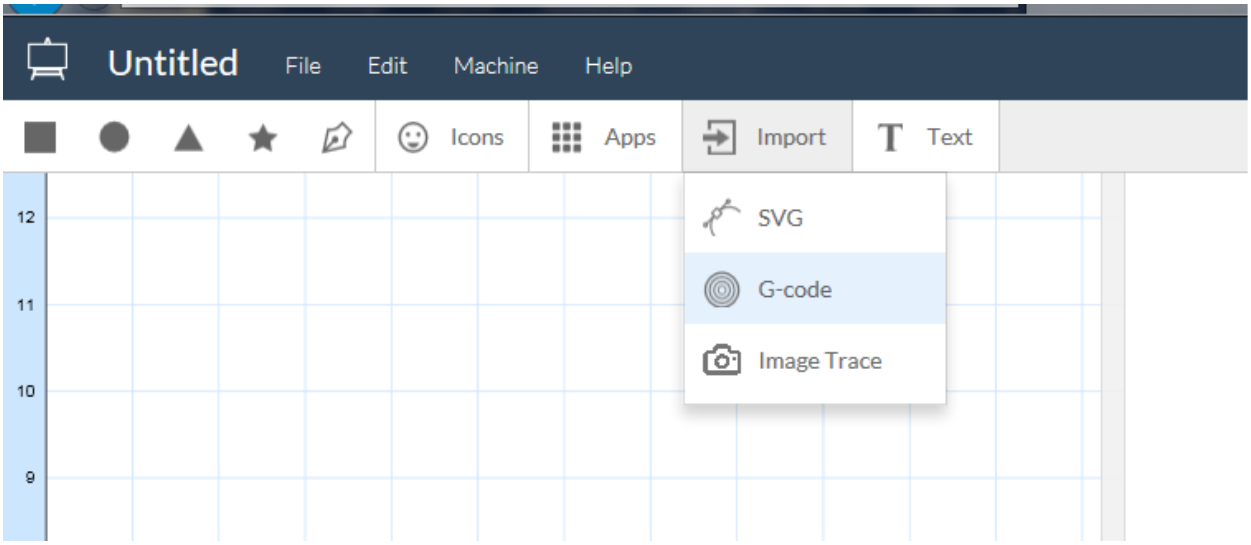
At this point, several dialog boxes will pop up followed by a processing report. Make sure that it doesn't indicate any errors, then you can validate your G-Code by going to the "Play Path" icon in the "Manufacturing" tab and selecting "Material Removal Simulation" from the dropdown menu.



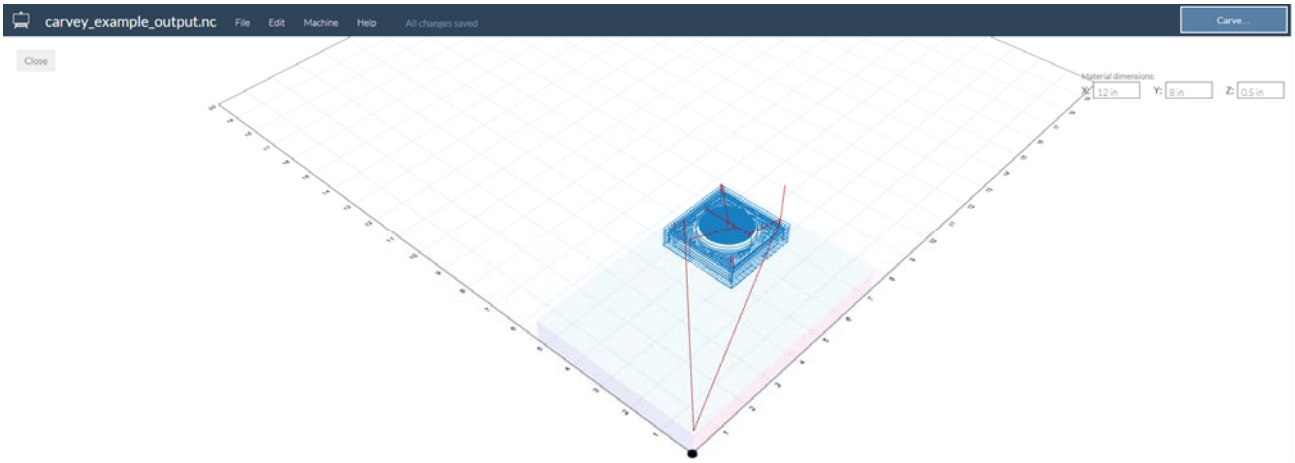

In the window that pops up, simply select the <filename>.ncl file that you just made, and hit the play button on the Vericut simulation that pops up. This can be handy to see exactly how your part will look when you make it. If everything looks good, it is now time to export your file to Easel so that the cut can be made.

## 14. Export to Easel

In the Easel application, click on the "Import" tab and select "G-Code" from the dropdown menu:



At this point, if you have done everything correctly you will see the G-Code pathing appear in the window:




## 15. Carve!

Before you begin carving, make sure to go through this checklist:

- Ensure that the part is totally secured, as any movement will cause you to lose the cut.
- Make sure to put some plexiglass or other such waste board material underneath your piece, as the cutting tool will often go slightly go slightly below your piece when a through-cut is specified.
- **WATCH THE CUT**, if there is any mistake in your coding, the carvey may attempt to either go beyond its cut area or cut into the brackets or the wasteboard. Cutting bits also have a tendency to break, which can lead to damage if the Carvey is allowed to continue after the breakage. Leaving the Carvey unattended during a cut makes both the machine and the piece susceptible to damage. DO NOT BE AFRAID TO PAUSE THE CUT IF YOU FEEL THAT SOMETHING IS WRONG.
- If your cut is a long one, you may consider pausing the cut every once in a while to vacuum out the work area. This is likely unnecessary, but it can avoid some chipping and other damage due to sawdust buildup.

# 16. Appendix

I. Machine Settings
   a. "Cut Feed" defines the speed at which the machine's spindle will move through your material.
      i. **Increasing this** puts you at risk of chipping your stock but will make your cut faster
      ii. **Decreasing this** can make for a very long cut.
   b. "Plunge Feed" defines the speed at which the spindle moves downward into the stock. Creo will allow this to default to your Cut Feed, and this can easily lead to broken mill bits.
   c. "Step Over" is the distance that the milling bit moves between parallel cuts.
      i. The minimum this can ever be is the radius of the drill bit, any higher will cause stock to be left in the cut.
      ii. Setting this value lower can make for a more detailed and clean cut, but it will take much longer.
   d. "Max Step Depth" is about the same as Step Over, but in the downward direction
   e. "Scan Type" defines the pattern that the mill will move in to clear out material.
      i. **Spiral** is one of the best settings for volume milling, as it best preserves the stock material.
      ii. **Type 3** is a dangerous choice for volume milling (especially at high speeds) but is fine for surface milling, where less material is present.
   f. "Clear Distance"
   g. "Spindle Speed" as it implies defines how fast the spindle spins
      i. **Increasing this** can usually be done without too much risk, but make sure not to exceed (or even reach) the spindle's maximum RPM
      ii. **Decreasing this** can lead to stock chipping, bit breaking, and even motor damage—do this with caution.
      iii. Note that this setting is irrelevant on the X-Carve, as the Dewalt spindle has manual speed control only.