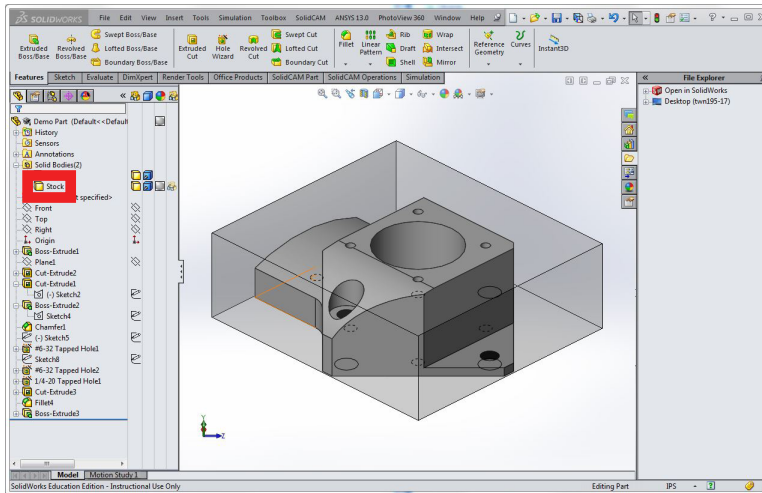


# SolidCAM Quick Start Guide

This tutorial demonstrates an overview of the basic process of generating toolpaths for CNC Milling using SolidCAM.

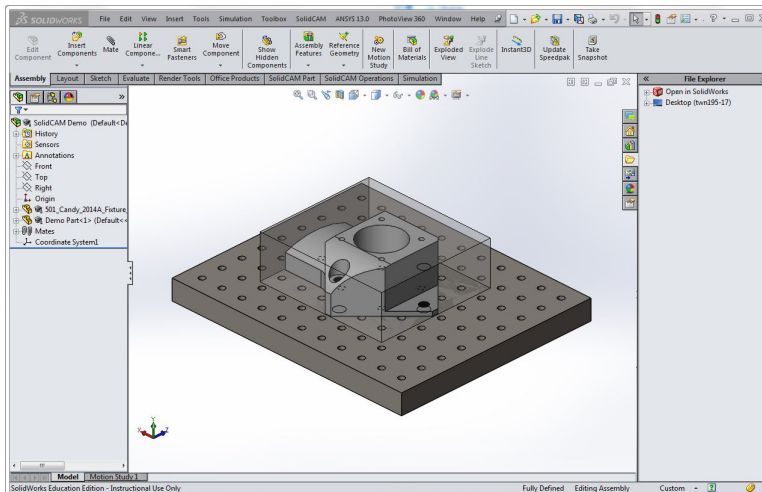
## SolidWorks Part Set Up



### Open the part

We will be making this part. Notice two things:

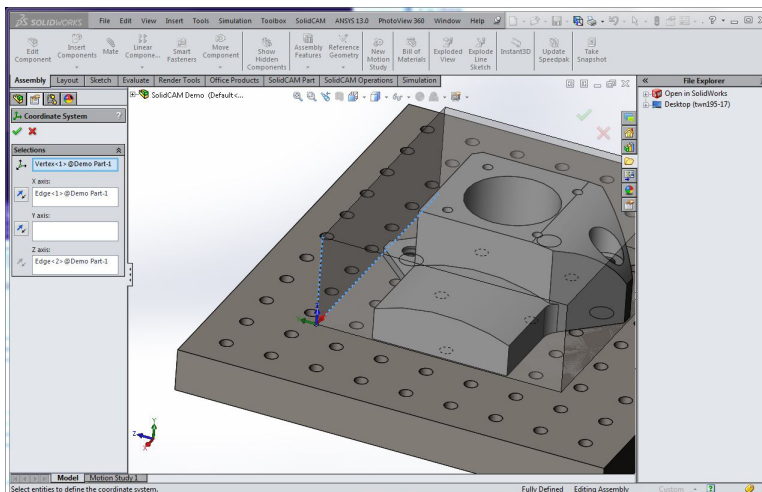
- Stock material was created within the part as a separate solid body. This can be done by performing a boss/extrude and deselecting "Merge Result"
- The view state of the stock material is transparent



### Machining Assembly

Create a new assembly that contains the part and stock model as well as any fixtures. In this case, the MiniMill fixture plate. Locate your part where you would like it on the fixture plate.

Create a fixture plate for each machining orientation, keeping your part in a single position.

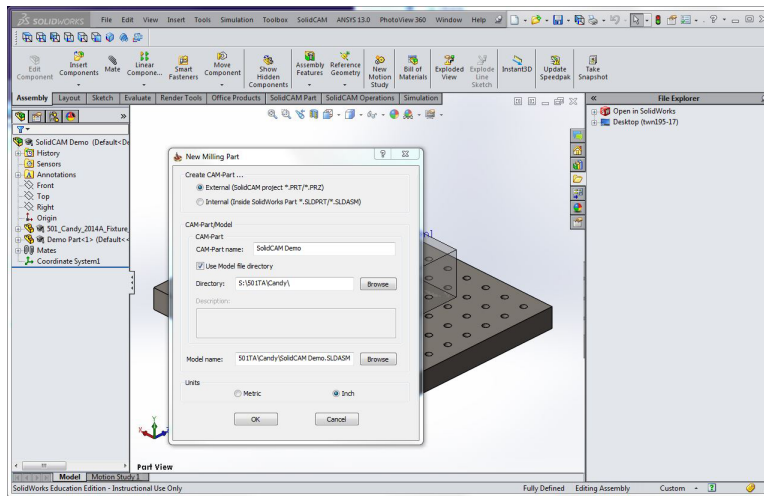


### Machining Orientation Coordinate System

Now, create a machining coordinate system for each machining orientation.

Reference Geometry > Coordinate System

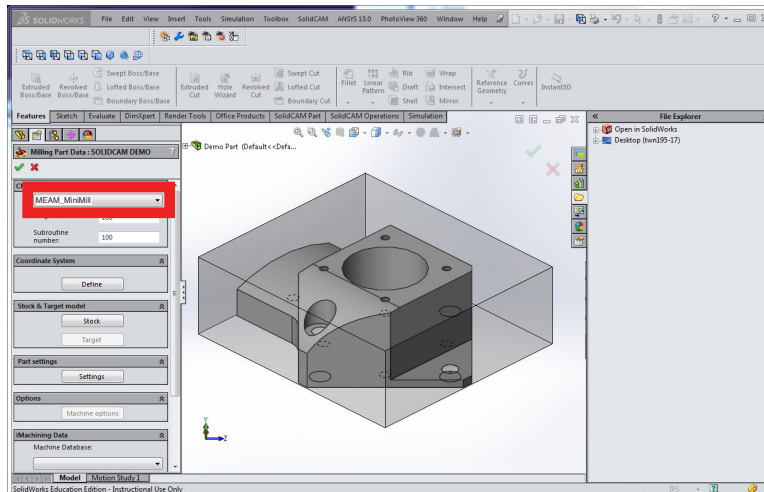
- For this part, the Coordinate System was created at the upper left of the part with the z origin at the top of the fixture plate
- Always create the origin at a point that can be re-found.
- Another, possibly better, option for the coordinate system for this part would be the upper left of the fixture plate itself.



**Create new SolidCAM Part**  
In the main menu, select:  
SolidCAM > New Milling Part

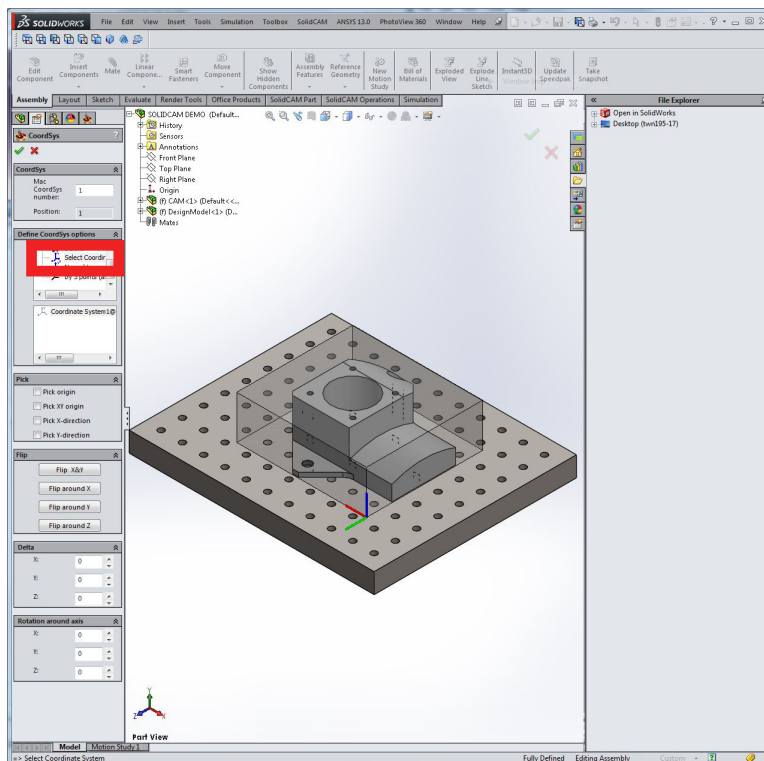
In the menu shown at left, select the options shown.

- These cannot be changed.



**Define Initial Parameters**  
Select the machine that you will be using.

- The menu on the left of the screen will now appear. This is where you will define the initial parameters for the part.
- These can be modified later.



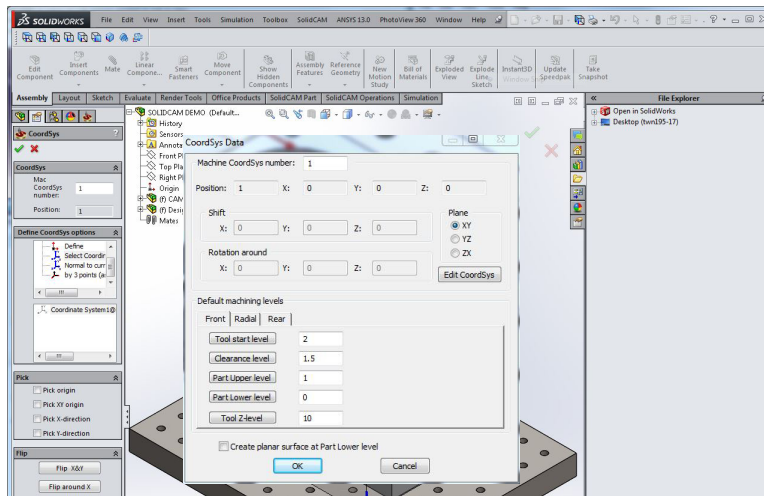
**Select a coordinate system**  
In the "Define CoordinateSys Options" box select "Select Coordinate System". In the box below, select the Coordinate System(s) that you wish to use.

- Select all coordinate systems that you will want to use. When generating toolpaths, you will select the specific coordinate system that you want to use.
- You can also create coordinate systems from edges and surfaces, but it is much easier to define the Coordinate System in SolidWorks and then select it in CAM.

Return to the Initial Parameters menu by pressing the green check.

## IPD 501 Spring 2015

### SolidCAM Quick Start Guide

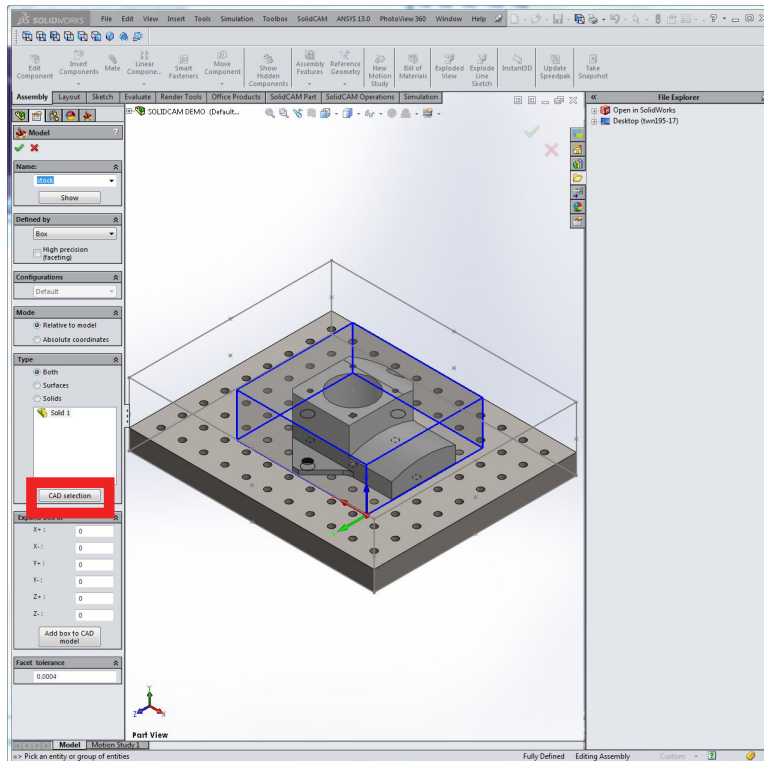


#### Define Coordinate System Levels

Selecting the button corresponding to each level allows the level to be selected in CAD. Alternatively, the numbers can be entered manually.

These are the levels that the tool travels to in different cases. As a general rule, the levels are listed in descending order.

- **Tool Start Level** height the tool moves to before starting an operation. This should be above the maximum level of the stock.
- **Clearance Level** height the tool moves to during rapid movements. This should be above the maximum level of the stock.
- **Part Upper Level** The top of the part.
- **Part Lower Level** The bottom of the part.
- **Tool Z-level** used in 4+ axis milling.

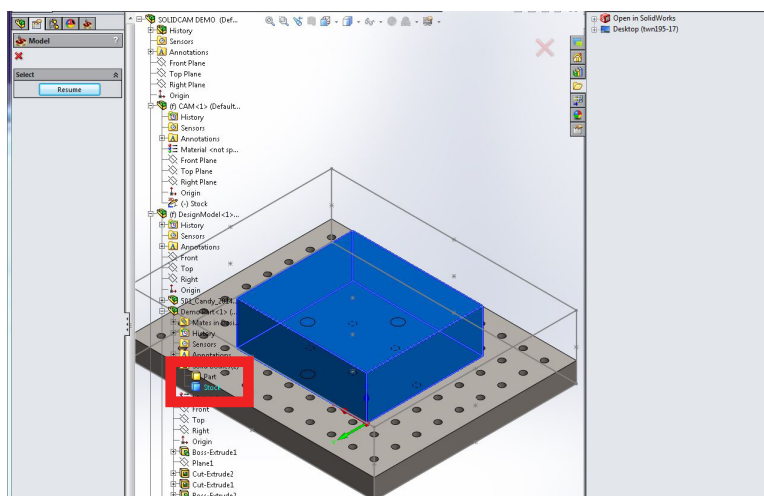


#### Select Stock and Target Material

In the Initial Parameters menu, click the button that reads "stock"

A menu menu is brought up on the left side of the screen.

Select "CAD Selection"



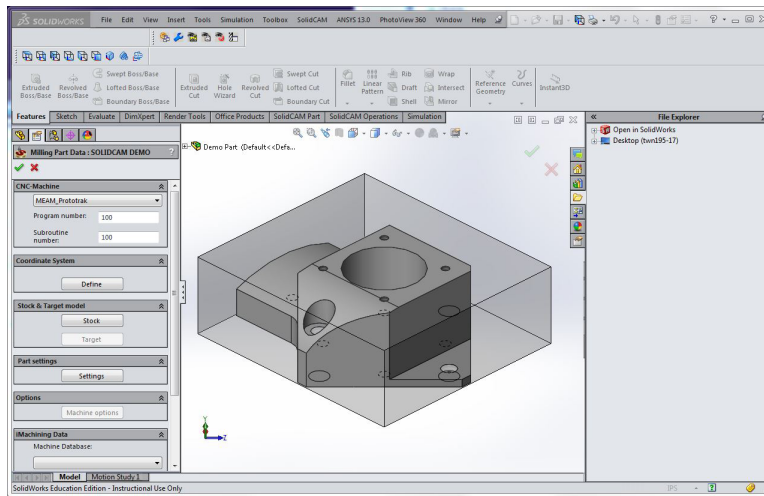
#### CAD Selection

Navigate to the part in the SolidWorks Feature Tree and select the stock material under Solid Bodies

Select "Resume" to exit CAD selection

Select the green check mark to exit Part Selection

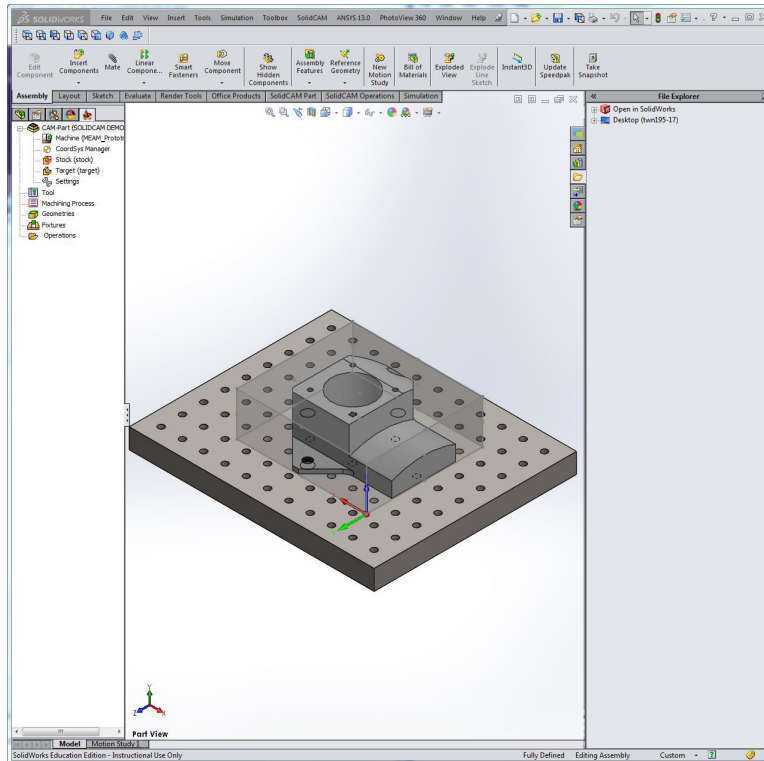
Use the same process to select the "target" - your part.



### Finish Initial Parameters

Press the green check to finish defining the Initial Parameters

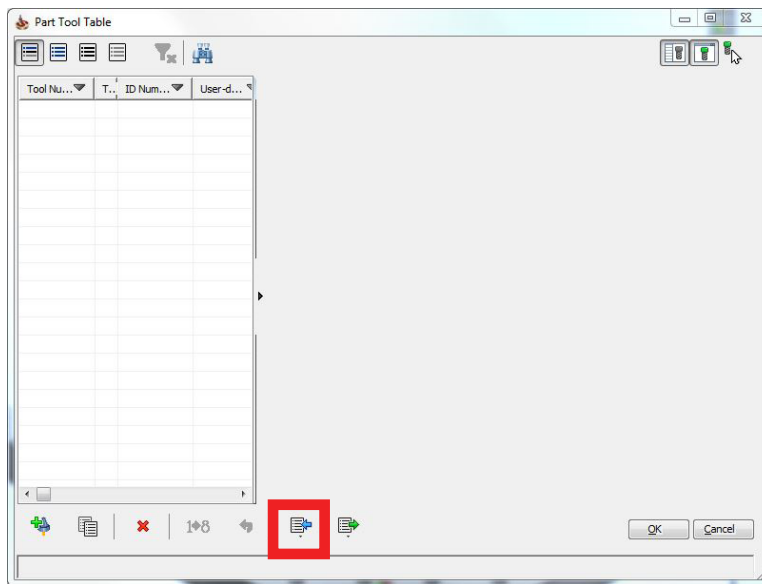
- The default "Settings" in the Part Settings box do not need to be modified.



### Main SolidCAM Page

- This is the main SolidCAM page.
- There is a similar "Feature Tree" to what you are familiar with in SolidWorks.
- The Initial Parameters that you just defined can be edited by double clicking the Parameters in the top of the Feature Tree.



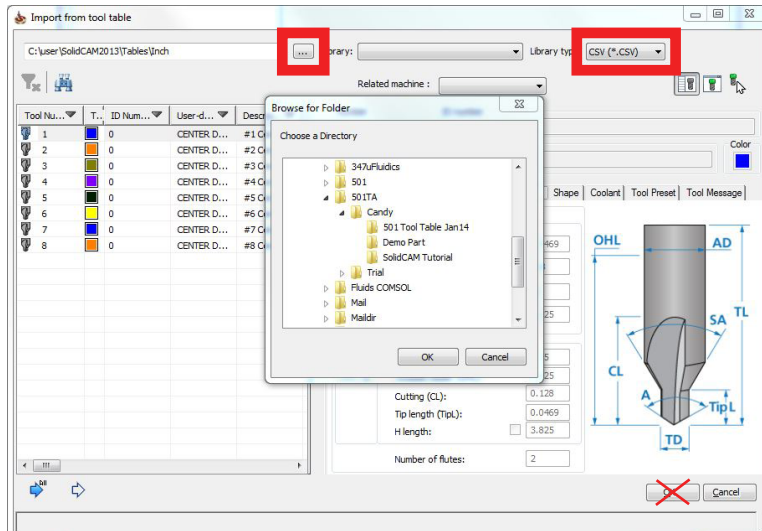


### Defining Tools

Double click the Tool button in the middle of the feature tree.

- This will bring up the Tool Table.
- The tool table is blank by default.
- You can manually enter tool parameters but for consistency, a tool table will be imported.

Select the Import Tools button



### Importing Tools

\*\*\* This will be frustrating if you don't follow these instructions in order

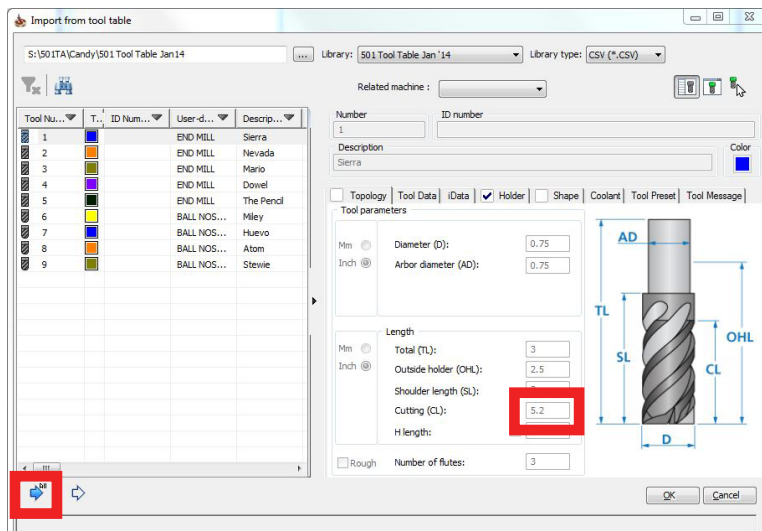
Change the file type to CSV

Select the "..." button

Navigate to the folder that contains the tool table to import

Select "OK"

DO NOT PRESS OK AGAIN

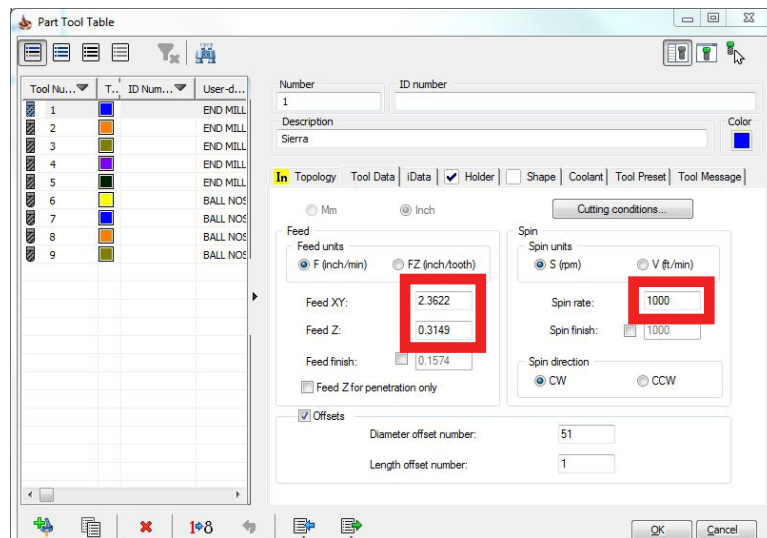


### Editing Tool Parameters

Check the tool parameters. The new tool table should work but it is always a good idea to ensure that the parameters are correct.

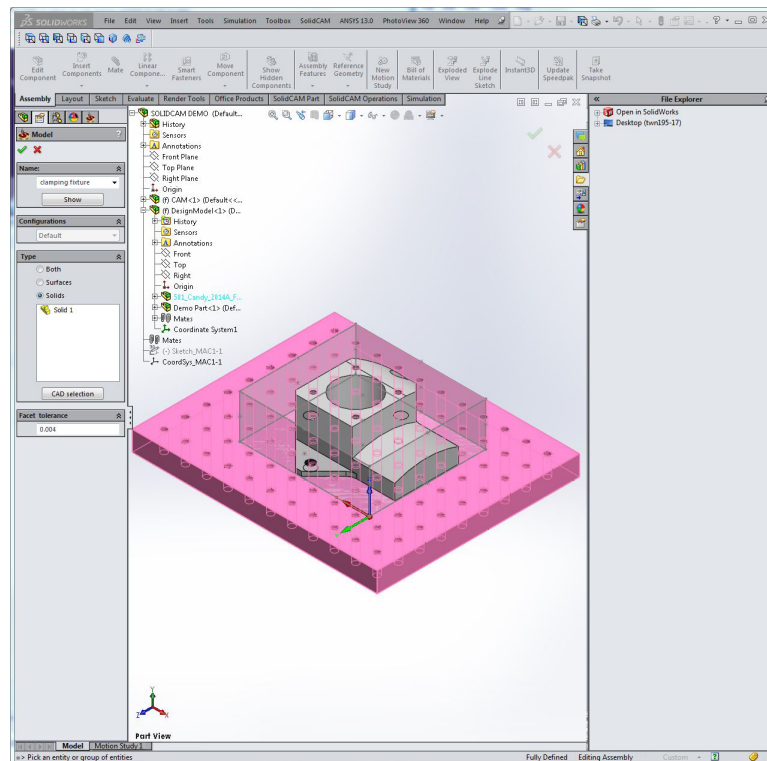
- For the Candy tool table, verify the tools with the table on the wiki.
- Verify parameters in "Topology" and "Tool Data"

The highlighted values are incorrect

**Editing Tool Parameters (2)**

Select "OK" to return to the main window.

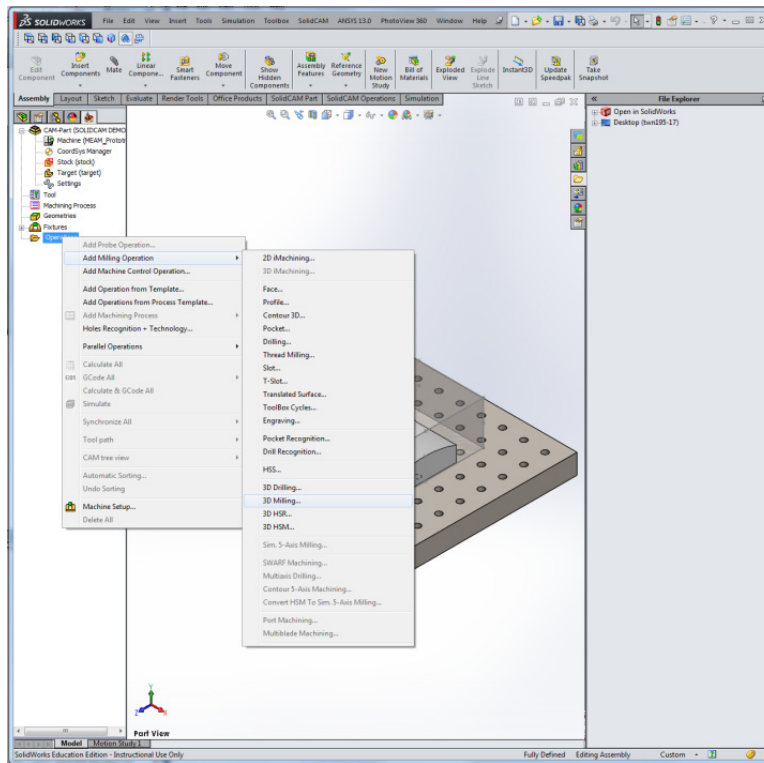
The highlighted values are incorrect

**Selecting a Fixture**

In the main SolidCAM window, double click the "Fixtures" menu and use "CAD Selection" to either directly select the fixture or select it in the SolidWorks Feature Tree.

Press the green check to return to the main menu.

# Toolpath Creation

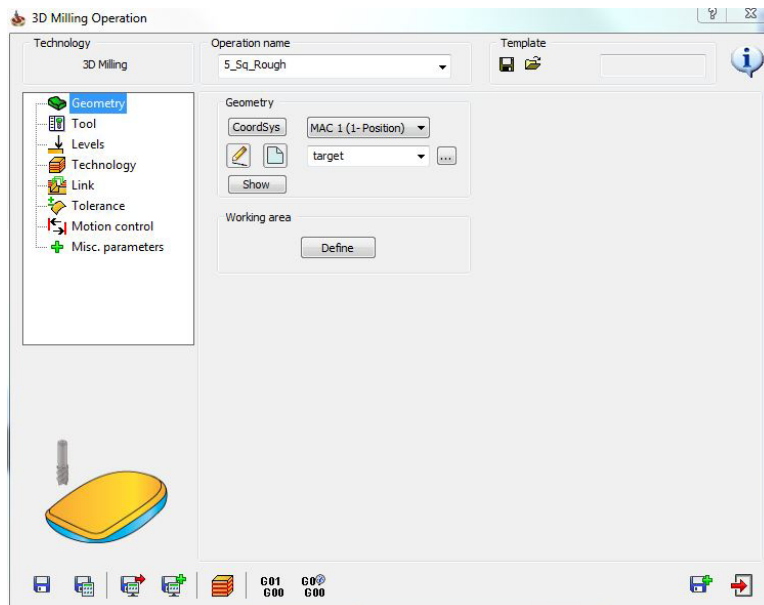


## Create Milling Operation

In the SolidCAM menu on the left right-click on the operations folder and select:

Add Milling Operation > 3D Milling...

- This is the broadest machining operation and the one that you will use 90% of the time.
- Other milling operations will not be covered in this tutorial.

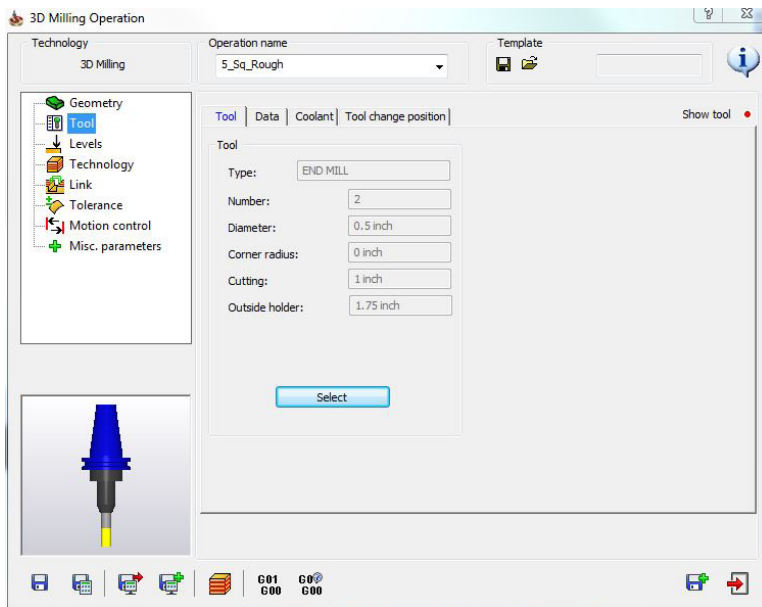


## Define Geometry

Adding a new milling operation opens this window and starts you in the geometry tab.

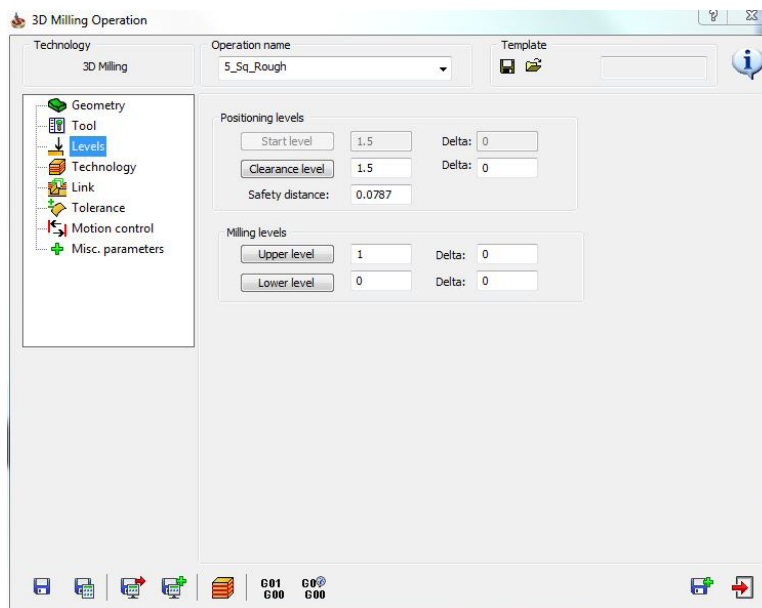
Name your operation to indicate what tool it will use and if it a roughing or finishing pass.

You can "Define" the working area but that is not necessary for the roughing pass of this part.



### Select Operation Tool

The next tab is the Tool tab. Press the select button and choose the tool that you want to use.



### Define Levels

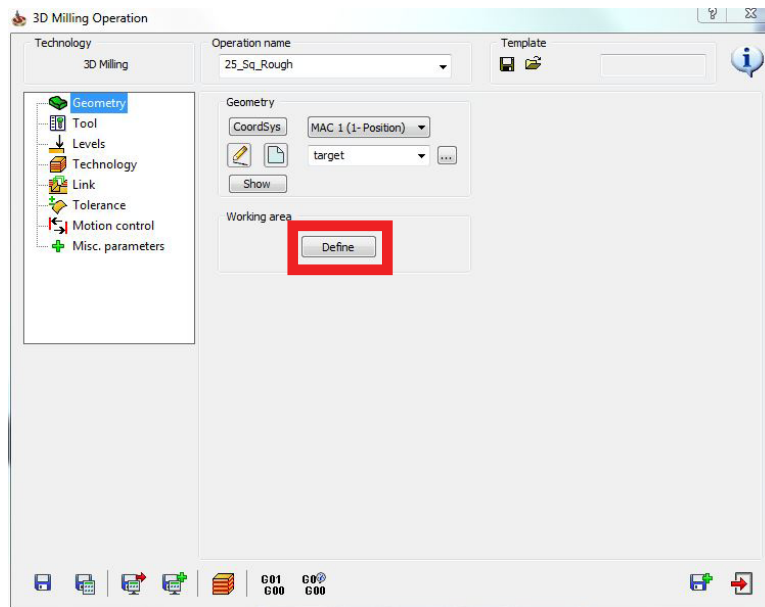
These are usually fine from the level you defined when defining the coordinate system.

- Levels can be used to focus on sections of a part or split up a large file.



Link, Tolerance, and Motion Control will not be covered. They are unnecessary for most 3D.

- **SolidVerify** is the highest quality simulator that does not require major setup. With very complicated surfaces, it can have a tendency to crash but it is generally reliable.
- **RapidVerify** works well but does not always render the most accurate results.
- **Rest Material** shows the material that is not part of your part after a milling operation.



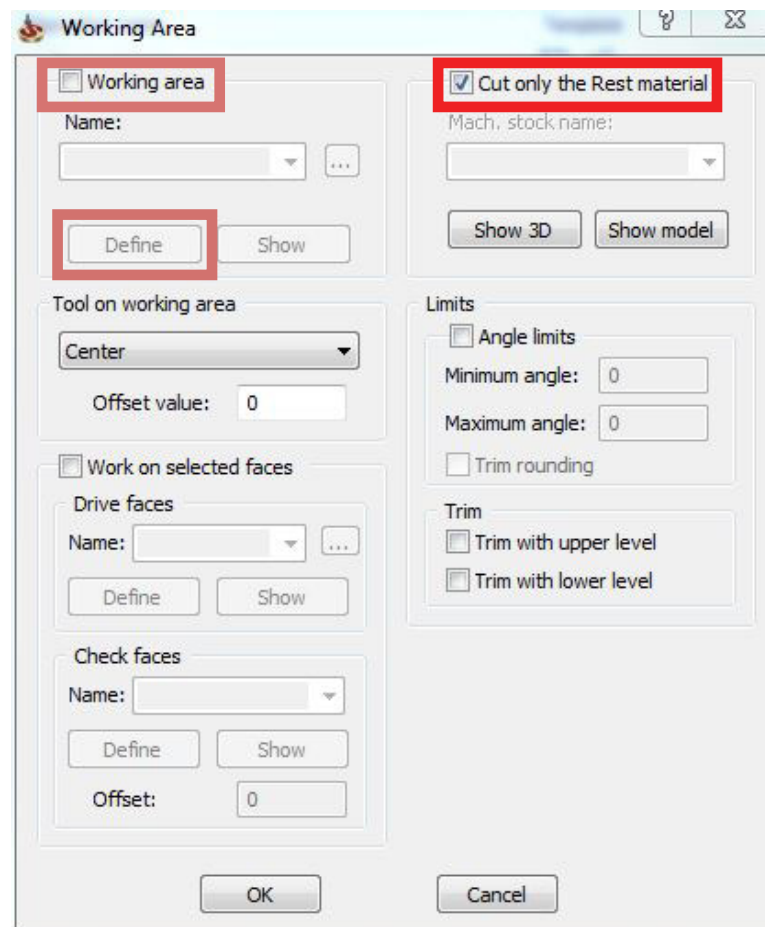
### Second Roughing and Finishing Pass

Follow the same procedure as before to create a new Milling Operation (Add Milling Operation > 3D Milling...)

Name your operation as before.

This time, we will edit the working area. Select the "Define" button in the Working Area box.

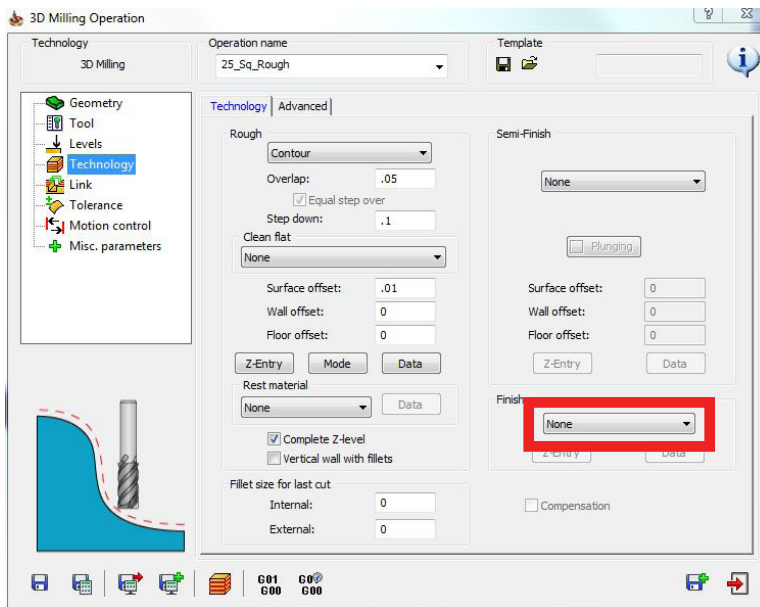
- Both operations will be performed with the same tool so they can be combined into a single milling operation.
- A second roughing pass is performed to remove more material before going to a finish pass. This will lead to a better surface finish but is not necessary in all cases.



### Working Area

Select the "Cut only the Rest material" checkbox. This will force SolidCAM to identify the material that was removed by the previous pass.

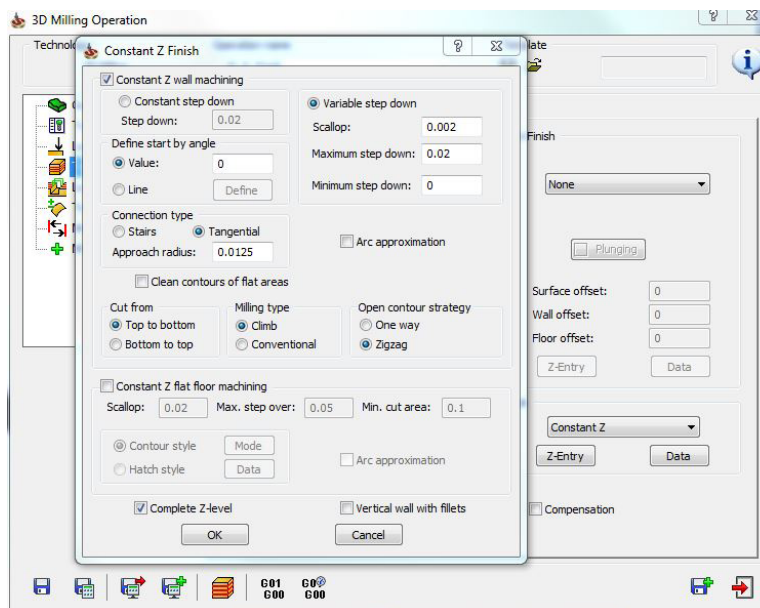
The working area can be constrained to a surface or feature by selecting the "Working area" checkbox and then selecting "Define". This will allow you to use CAD selection to either directly select areas or use the SolidWorks Feature Tree.



### Technology

Select the appropriate parameters for the rough pass.

Using the drop-down menu in the "Finish" box, choose a finishing option. We will use "Constant Z" for this pass. The tool will stay at a constant level and countour the part topography at varying Z levels.



### Constant Z Finish

Selecting the "Data" button in the "Finish" box will open the Constant Z Finish parameter window.

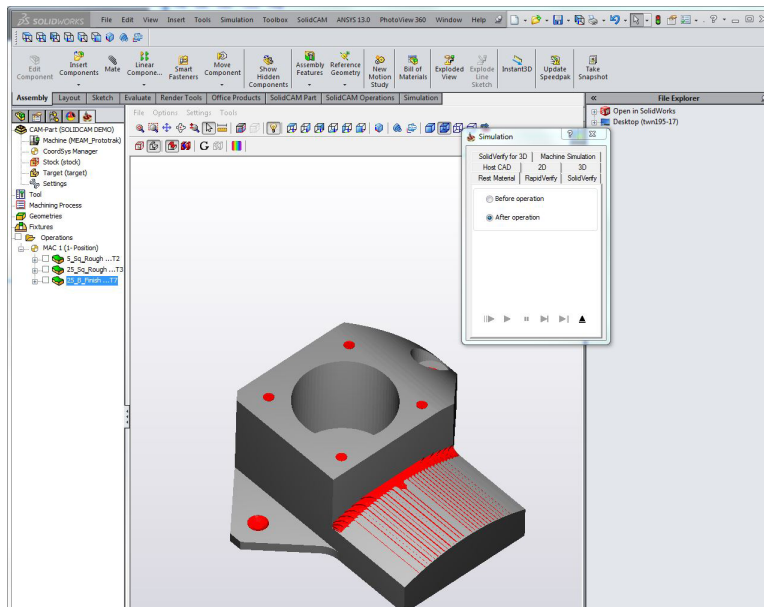
Select and define the appropriate parameters.

Select "OK"

In the 3D Milling Operation window, select "Save and Calculate" and then "Simulate"

- **Variable Step Down** allows for scallop height to be defined. It tends to be faster than constant step down and results in a more uniform surface finish when compared to Constant Step Down.
- **Zigzag** open contour strategy will reduce rapid movements and decrease machining time.

Explore other options through simulation.

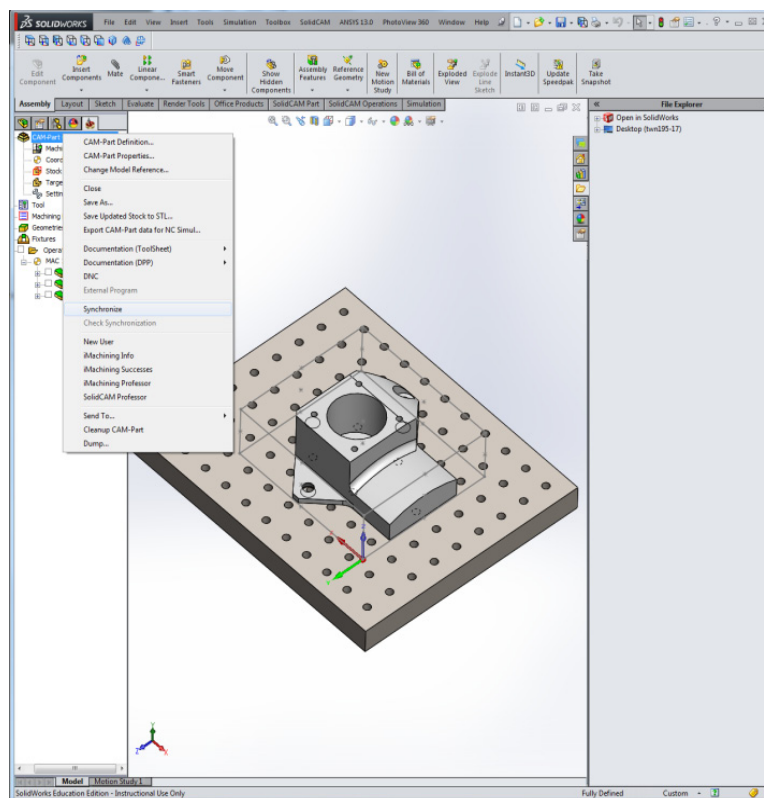


### Modifying Geometry

Simulating the part in Rest Material highlights an unmachinable region. Upon further inspection of the part, it is noted that this corner was not filleted!

Exit out of the simulation and navigate to the SolidWorks Feature Tree. Right click on the part and select "Edit Part". Add the fillet and save.

Close the part and return to SolidCAM.



### Synchronization

The update was made but it is not reflected in SolidCAM.

Right-click on the CAM-Part menu in the SolidCAM Feature Tree.

Select Synchronize.

It still does not update. WTF!

The trick is to double click on the CoordSys Manager and edit MAC-1 and the rest of your coordinate systems.

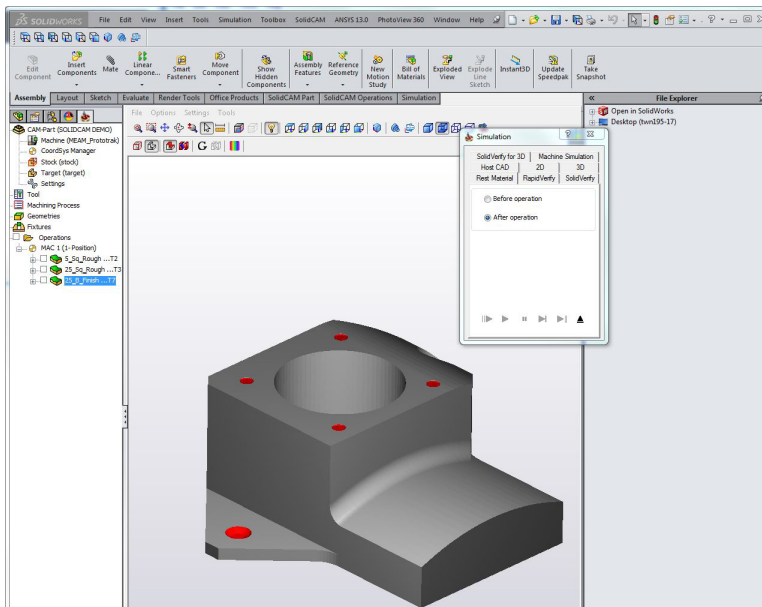
In the coordinate system edit window, right click in the selection box and select "Unselect All".

Select "Define" and reselect your coordinate system.

Asterixes should now appear next to all of your milling operations indicating that geometry has changed and the operation needs to be recalculated.

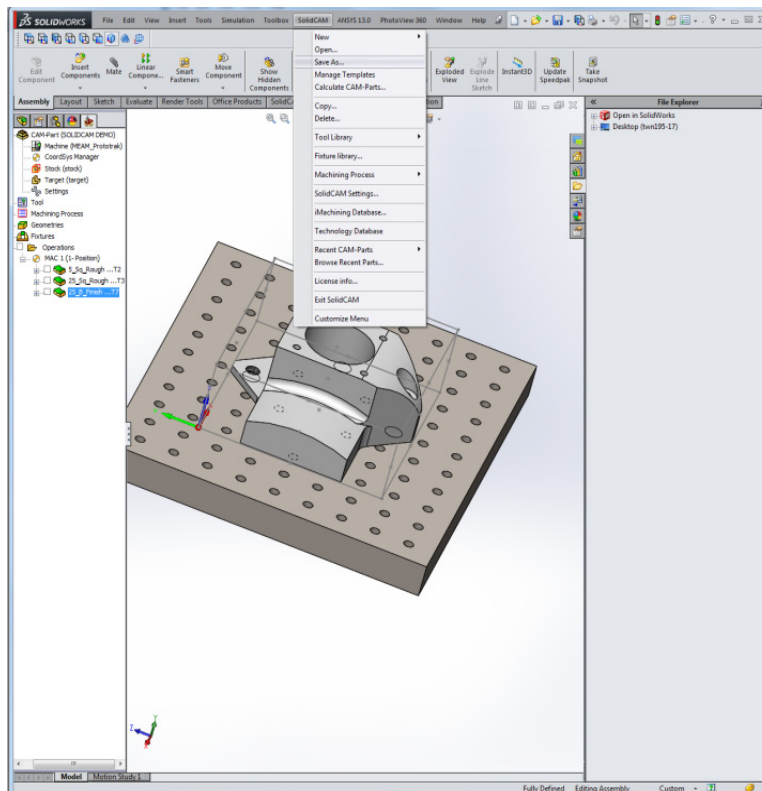
Right click on your operations and select "Recalculate". The new toolpaths should reflect the updated geometry.





### Final Simulation

Final simulation using Rest Material shows the finished part with material remaining in holes only. Surfaces are nicely finished.

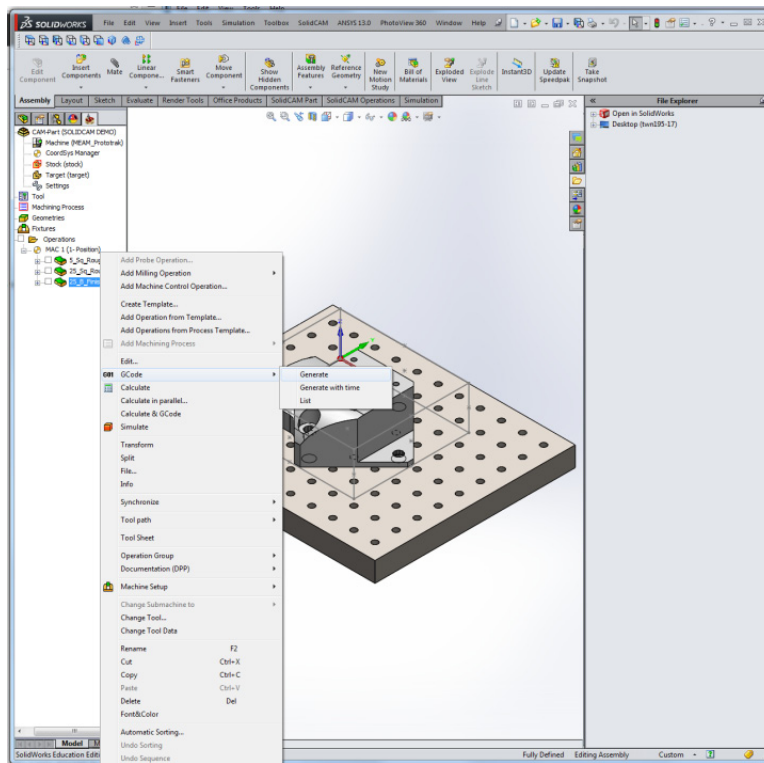


### Saving

Save by selecting: SolidCAM > Save As and giving your part a revision name.

SolidCAM will close your file and open the new saved-as model.

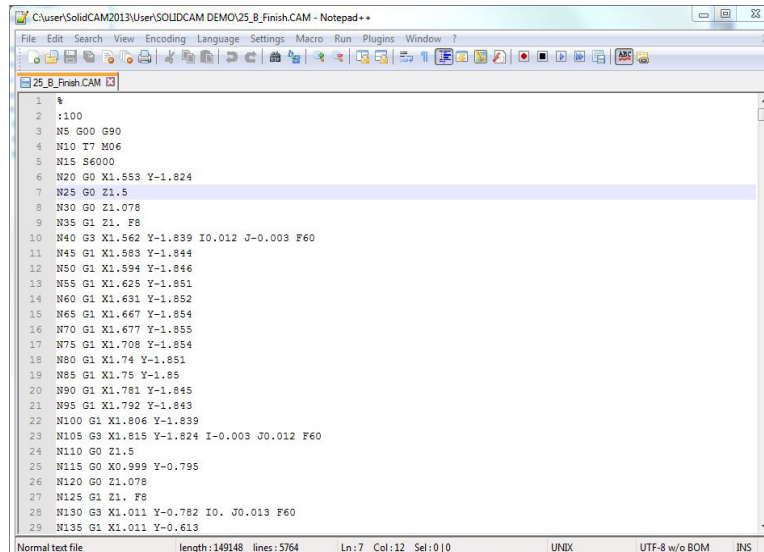
- Having this revision history can be very useful.



### Exporting G-Code

Right-click on a Milling Operation in the SolidCAM Feature Tree and select GCode > Generate.

Name your file the same name that you gave the Milling Operation earlier.



### Editing GCode

SolidCAM will open the GCode in Notepad so you can preview it before running on the mill.

**The GCode that SolidCAM is not perfect. YOU MUST REPLACE ALL INSTANCES OF G53, G55, G56, G57, G58, AND G59 WITH G54.**

Save your GCode to a safe place and copy it to a USB memory drive. We will use this drive to upload the code to the mill.