Team 2485 CAM Standards

Updated: Dec 2024

This document was developed to provide our team with standard ways to use our two CnC machines, a mill and a table router. Until we developed this procedure, CnC machining, including tool selection, machining methods, feeds, speeds, and chip loads, was conducted largely according to team urban legend, resulting in broken tools and machining times as much as ten times longer than necessary.

This document describes how to setup the CAM for 3 standard types of parts; 1in X 1in 6061 Aluminum square tube, 1in X 2in 6061 square tube, and 6061 plate parts (of various thicknesses). These three part types cover about 95% of the parts needed for robot construction each year. By standardizing the procedure it makes it easier to train a larger number of machinists and the parts get manufactured faster. The other 5% of parts are done by the most experienced machinists, with skills that build off the basics in this document.

Our team uses a Supra Bridgeport style mill with a 42in bed and with an Acu-Rite 3-axis MillPwr G2 add-on CnC package and a Velox 48 X 52in table router run on Mach 3. While it is our standard procedure to make all square tube parts on the mill and all plate parts on the router, we train to make all types of parts on both machines in case one of the machines goes down. The Velox company has gone out of business so this machine is no longer available, but these procedures should work on the various Omio routers.

On the mill we use twin 6in Kurt vises for square tube parts and clamps and sacrificial boards for plate parts. On the router we use a single 48in X 48in MDF sacrificial board that we bolt to the table and replace about twice a year. We also have 3 small vices for the router that allow us to do longer parts than the mill is capable of.

We have selected a standard end mill for basic CnC milling operations on each of our machines. On the mill the standard is a 3 flute, carbide, .25in diameter, ZrN coated square end mill. On the router we use a single flute, carbide, 4mm diameter, DLC coated end mill. The latter we get from Thriftybot for $15 a pair. Clearly not all milling can be done with these tools but this is our starting point and we have empirically tested the feeds, speeds and chip loads for these tools in 6061 Al and in polycarbonate/delrin and the results are in the table below.

Although the team designs in Onshape, in our shop we use Solidworks to manage the parts, and create assemblies and mates. Then we use HSMWorks (from Autodesk) which has a seamless plugin to Solidworks. Both software packages are available with free educational licenses.

This document has 4 sections; 1)Basic Standards, 2) a discussion of tooling and feeds and speeds, 3) specific screen by screen instructions for CAMing the 3 standard parts, using templates, and 4)some information about 3D machining methods.

Basic Standards:

Order of operations:

1. Create an Assembly in Solidworks
2. Create a Job in CAM
3. Drill model holes in the part and any extra hold down holes.
4. Install all appropriate hold down screws
5. Drill plunge holes
6. 2D interior contours and bearing holes
7. 2D Adaptive/2D Pocket (if used). This is now rare.
8. 2D Exterior contours

- Try to prevent unnecessary tool lines through organization of operations
    a. Tool lines sometimes reset the z-zero on the mill, interrupting workflow, so want to be minimized (this problem is unique to the team 2485 mill)

All files should follow this standard:

- Always put parts into an ASSEMBLY, no matter the number of parts
    - If it is not a copy, you will be adding CAM data into the final robot assembly, making it harder to open and use
    - Making even single parts assemblies allows for the CAM to automatically update if design changes are made
- File naming convention: CAM-[Initials]_[Simple Part #]-x[Number of parts]_[Additional Parts]. Examples:
    - Two of 2485-20-P-S0102 → CAM-DC_S0102-x2
    - One 2485-20-P-S0102 and three 2485-20-P-F0103 → CAM-TSW_S0102_F0103-x3
    - Underscores separate pieces of info, so use between CAM header, parts and additional parts → CAM-TSW_S0102_F0103-x3

    - Dashes add info, so use within a part or to indicate CAM author → CAM-TSW_S0102_F0103-x3

All operations should be renamed clearly to this standard:

- [Tool size in decimal] [Tool Operation] [Operation Number (if there are more than one)]
    - ".201 Drill" or ".25 Contour 2"
    - This will create a comment within the code that easily identifies the next tool

All tools used should be selected from the team tool library:

- The tool definition sections of this document should be used only if the tool does not exist already in the library
- Make sure to always select tools from the same library, otherwise you may get conflicting tool numbers that break the program

For parts with bearing holes (see 2D Pocket):

- After machining the bearing hole, in a separate step and regardless of the thickness of the material, program a second machining step for all the bearing holes that is a full depth cut. This acts like a finishing pass but solves a problem in HSM Works if you use a finishing pass in the first step. What this step will do is make a cut with much less tool deflection. This will take off a tiny amount of material and make the hole closer to actual bearing size but still capable of a good press fit

Using this document:

- A contour is the same as an edge
- Hover over icons in the menu bars if specified setting in the document is not immediately obvious
- If a setting is not discussed or written into this document, then it is not important and you should use the default

Feeds and Speeds

- The feeds and speeds set for a CNC machine make reference to the feed rate (or "cutting rate") the machine moves the tool through a material and the rotational speed of the spindle. These values can be calculated by any number of feeds and speeds calculators but it is also calculated within the HSMWorks CAM software as long as the inputs are correct. If the inputs are not correct, at best, the cutter will work inefficiently but more likely the cutting tool will either clog and break or just break.

- The two most important inputs are the Surface Speed per Minute (SFM) and the chip load recommended for a given tool in a given material by the manufacturer. We have this data for our carbide tools and recommended averages across many companies for our High Speed Steel tools. The carbide tool data was based on manufacturer recommended tool data and refined with empirical testing.
    - The chip load is the size of cut (or chip) that a single flute makes each time the flute comes around. This changes based on the diameter and material of a specific cutter. For example, for our 3-flute, .25 in diameter carbide end mill we use a chip load of .0012 in, and for our 3-flute, .125 in carbide end mill the chip load is .0004 in. These numbers change significantly between carbide and High Speed Steel tools and need to be referenced to each manufacturer.
    - The SFM is also determined by the manufacturer and is an expression of how fast a tool should be pushed through a particular material (based on recommended chip load). This number is primarily used to determine the spindle speed and varies significantly based on the material that is being cut. For example, the SFM recommended by most manufacturers for cutting 6061-T6 aluminum is a range from 600-1200 feet per minute. Spindle speed (in RPM) is determined by (SFMx3.82) / the tool diameter (in inches).
    - Feed rate (in per min) at the determined spindle speed is determined by RPM X chip load X the number of flutes.
    - The complication comes where the machine has a maximum spindle speed below what the calculations determine. For Team 2485, the max RPMs are 5400 for the CNC mill and 18000 for the CNC router). If your inputs to the calculator in HSM Works lead to a spindle speed in excess of 5400 rpm (which will be the norm for small cutters in aluminum) then you must set the spindle speed at 5400 RPM. This will automatically reduce the feed rate.
- Most manufacturers recommend, and we use in most circumstances, a depth of cut of not more than .5 X D. The maximum depth of cut in the table below was determined by many test runs in those materials and several seasons of use.
- When setting up the CAM it is vital to ensure that the tool table is setup correctly for the tools you intend to use. If you are using an existing tool table, like the Team 2485 Tool Table, you should always check to make sure that tools you are calling are correct.

End mill Summary

End Mills	Stock material	Tool Diameter	Flutes	Chip Load (in)	Spindle speed (RPM)	Max depth of cut (in)
Router	6061 Al	4mm	1	.002	18000	.125
Mill	6061 Al	.25 (¼)	3	.0012	5400	.125
Mill	6061 Al	.125 (⅛)	3	.0004	5400	.08 to .125
Router	Delrin/PC	4mm	1	.004	9090	.25

- The CNC Mill is a much more powerful machine than the router and as such can make much faster cuts if cutting with large end mills and making deep cuts. The router however can take advantage of smaller end mills with fewer flutes (including single flute cutters) because of the high maximum spindle speed. This could involve spindle speeds up to 18k RPM. As a result we take advantage of the high feed rates achieved with our standard 4mm single flute end mill at 18000 rpm in .125 in thick 6061-T6 plate or up to .25 in thick polycarbonate or Delrin.
- If the cutter is getting clogged, in all likelihood, you are using a spindle speed that is too fast for the feed rate you are using and the aluminum is melting rather than cutting. Sometimes a dull cutter can give the same symptoms. Additionally, if you are cutting another aluminum alloy other than 6061-T6 (like 6063 or 5052), clogging will happen. The material will appear to turn to mush right before it breaks your end mill. If you are getting significant chatter and a rough looking cut – first E stop the machine – then you are likely using too fast a feed rate for the selected (or set) spindle speed. Always remember that for both of our CNC machines, the “S” command in the CAM code is just advice to you the operator. The command will not set the actual machine spindle speed. You must independently set the spindle speed by following the frequency to spindle speed table on each machine and setting the machine frequency on the separate control panel. For the router at least, the “S” command will start the spindle.

Tool selection and RPM

- Router. For Aluminum (whether plate or square tube), we will use the carbide, 4mm diameter, single flute, DLC coated end mills, with a chip load of .002 in, at

18000 rpm spindle speed. If we have to use the 1/8in end mills, use 16000 rpm with a chip load of .002in but the depth of cut in aluminum must be reduced to less then .1in. This does not apply for facing off the end of square tube. We will use ⅜ in 2 or 3 flute end mills for this purpose.

- CNC mill. For Al plate or square tube, we will use the carbide, 1/4in dia, 3 flute end mills, ZrN coated (if available) at 5400 RPM spindle speed with a chip load of .0012in. This does not apply to facing square tube - use the long flute ½ in end mills at 5400 rpm.

Depth of cut

- For mill and router. With the standard tools, for up to ⅛ in Al plate or square tube, use a single depth cut. If using a ⅛ in end mill for ⅛ in depth of material, reduce the chip load or use multiple depths. For 3/16in Al plate use multiple depths (2 or 3) with "even stepdowns" and a max step down of .1in. For 1/4 in Al plate use multiple depths with at least 3 cuts.

General machining processes

- For both the mill and the router we will no longer use adaptive clearance. We will use internal contours for both weight reducing holes in parts and for bearing holes.
- Use separate contour tasks within a job for the internal contours and for the bearing holes. For bearing holes, make a separate full depth contour step after the initial contour step. This acts as a finishing pass but DO NOT use "finishing passes" because there is a bug in that software that can break a tool in thick material. For multiple parts in the same piece of stock, all of the internal contours for all internal holes can be done in one step. All of the bearing holes in all the parts can be done in one step.
- Use "Predetermined Positions" rather than ramps. End mills cannot plunge. HSM works offers 2 solutions : ramps and pre-drilled holes. Ramps take a long time to machine. For all contours use pre-drilled holes (or plunge holes). Ensure that the pre-drilled holes you drill are at least as large as the end mill you plan to use.
- As a standard, we will use the .201 in drills to drill the plunge holes. In the Team 2485 Tool Table there are two .201 drills (at this writing they are tools #17 and #18). Use tool #17 for the model holes and tool #18 for the plunge holes. That way the CAM software will generate the stop codes so the machine stops automatically for you to install the hold down screws prior to drilling the plunge holes. This procedure prevents the accidental placement of a screw in a plunge hole - which always ends badly.

Cut through depth

- Mill. Generally set the cut through depth for milling and for drilling at -.030 (or just .030 for drilling).
- Router. For small jobs, set cut through to -.050. For larger parts, cut through to -.070 to -.1in (for very large parts like a belly pan). Set Z in the middle of the stock. Then check the Z height on the corners of your stock and if the variation from the corners to center exceed the cut through you set, reset the cut through to be deep enough before proceeding with machining the parts

Drilling

- Mill and router. Use 5000 rpm spindle speed and use stubby drills.

Special note: Drills on the Router. For any holes larger than 3/16 in dia, use a 13/64 drill pass first and then go back and drill with the larger drill bit. DO NOT to use drills over ⅜" diameter AT ALL!!!.

CAM - General drilling and milling procedures

Router plate job

- Aluminum. For drilling steps, use 5000 rpm spindle speed and use stubby drills. For any holes larger than 3/16 in dia, use a 13/64 drill pass first and then go back and drill with the larger drill bit. Use the first drill step to drill only the model holes and extra hold down holes. You need at least 3 hold down screws for each part and more for larger parts. Then install the hold down screws. Then drill the plunge holes. This will ensure that a screw does not get accidentally put in the plunge holes. For the internal contours within a part, only use hold down holes for very large internal pieces - greater than 4in. For milling use a 4mm end mill, 18k RPM spindle speed, and a chip load of .002in. Use internal contours for weight cut outs and other internal cavities. For bearing holes use a contour pass(es) and then a separate internal contour with a full depth cut. This acts as a finishing pass. For each contour use a pre-drilled hole adjacent to each contour. For ⅛ in Al plate use a single depth cut. For 3/16in Al plate use multiple depths (2) with even stepdowns and a max step down of .1in.

- Delrin. For up to 1/4in Delrin use a 4mm end mill, with a 375FPM surface speed (9090 RPM) and a .004 chip load. Use multiple depths (2) with even stepdowns. Use pre-drilled plunge holes. Don't screw down internal contour or bearing pockets. Do not use coolant when machining Delrin - only air to blow the chips away. This helps prevent recutting of chips that can lead to melting.

Router Square tube.   For drilling operations, use a 5k rpm spindle speed, use a stubby drill when drilling through no more than 1" of square tube. DO NOT use a jobber length drill for deeper cuts like through cutting a 2in square tube. There is no way to control where the jobber length drill starts on the internal surface. Drill just deep enough to go through one side and then flip the square tube and drill the other side. This is the same number of drilling evolutions, is more accurate and less likely to break drills. For milled holes in the square tube use pre-drilled holes for contours. For bearing holes, use an internal contour and then a separate repeat internal contour cut.. Then for up to 1/8in thick material, use the 4mm endmill, 18k rpm spindle speed and a chip load of .002in for milling operations.

CNC Mill Al 6061-T6 plate work. For drilling operations, use a 5k rpm spindle speed and a stubby drill when drilling through the plate. Use a .05 cut through. Any deeper and the hole will not be good for a hold down screw. Use the first drill step to drill only the pre-drilled plunge holes then stop and mark those holes on the plate so a screw does not get accidentally put in those holes. Then drill all other holes in the part and any hold down holes for internal contour pieces. For milling, use the carbide, 3 flute, 1/4in end mills, with a 5400 rpm spindle speed and a .0012 in chip load. For up to 3/16" plate use a single depth cut. For thicker plate use multiple depths (2 cuts). For all interior holes in the part, use internal contours ensuring that the cut is on the inside of the line. All interior contours in the entire job can be programmed in one contour program line. Use pre-drilled holes with a 1/4in hole (assuming you are using a 1/4in end mill) for each interior contour. For exterior contours, use a new contour program line for each part in the job. Use a pre-drilled hole for each exterior contour (hole size at least as large as the end mill). For large interior cutouts (roughly 4in across or greater) put 2 holes near the center of the cutout and well away from the contour edge for hold down screws. Use "points" in CAD for hold down and pre-drilled holes. In CAM ensure that you drill from "stock top" to "stock bottom".

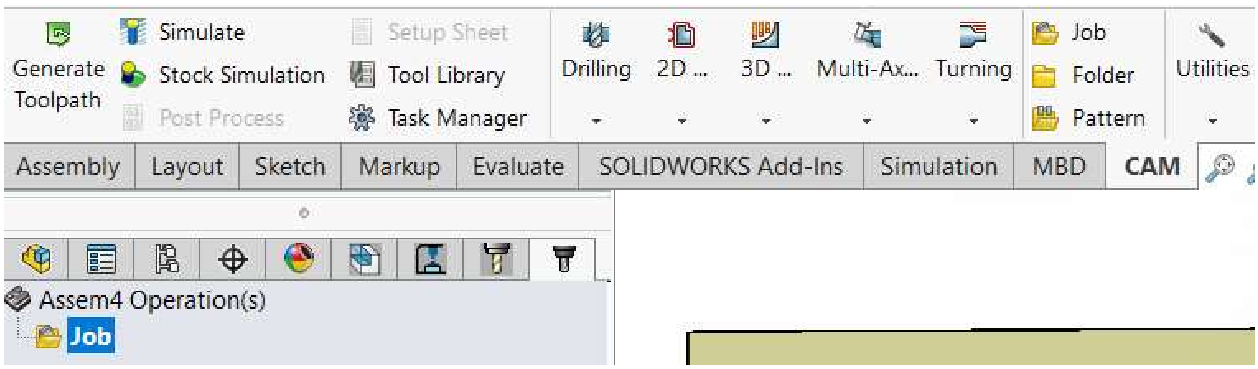
CNC Mill Square Tube: For drilling operations, use a 5k rpm spindle speed, use a stubby drill when drilling through no more than 1" of square tube. DO NOT use a jobber length drill for deeper cuts like through cutting a 2in square tube. There is no way to control where the jobber length drill starts on the internal surface. For a side > 1" deep, drill just deep enough to go through one side and then flip the square tube and drill the other side. This is the same number of drilling evolutions, is more accurate and less likely to break drills. For milled holes in the square tube use pre-drilled holes for contours. For bearing holes use an internal contour with 2 finishing passes. Then for contours up to 1/8in thick material, use the carbide, 3 flute, 1/4in end mills, with a 5400 rpm spindle speed and a .0012 in chip load. In the special case, shown below in the example, if machining bearing block holes, this requires a 1/8in end mill. Use a 3 flute, ⅛ in carbide end mill at 5400 RPM and a chip load of .0004in.

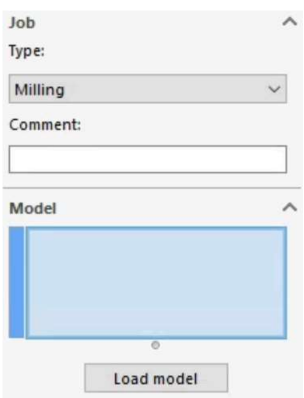
Specific Examples

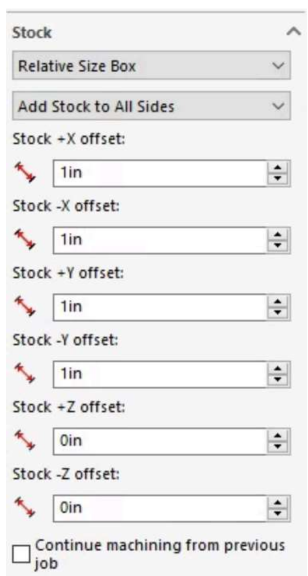
Job:

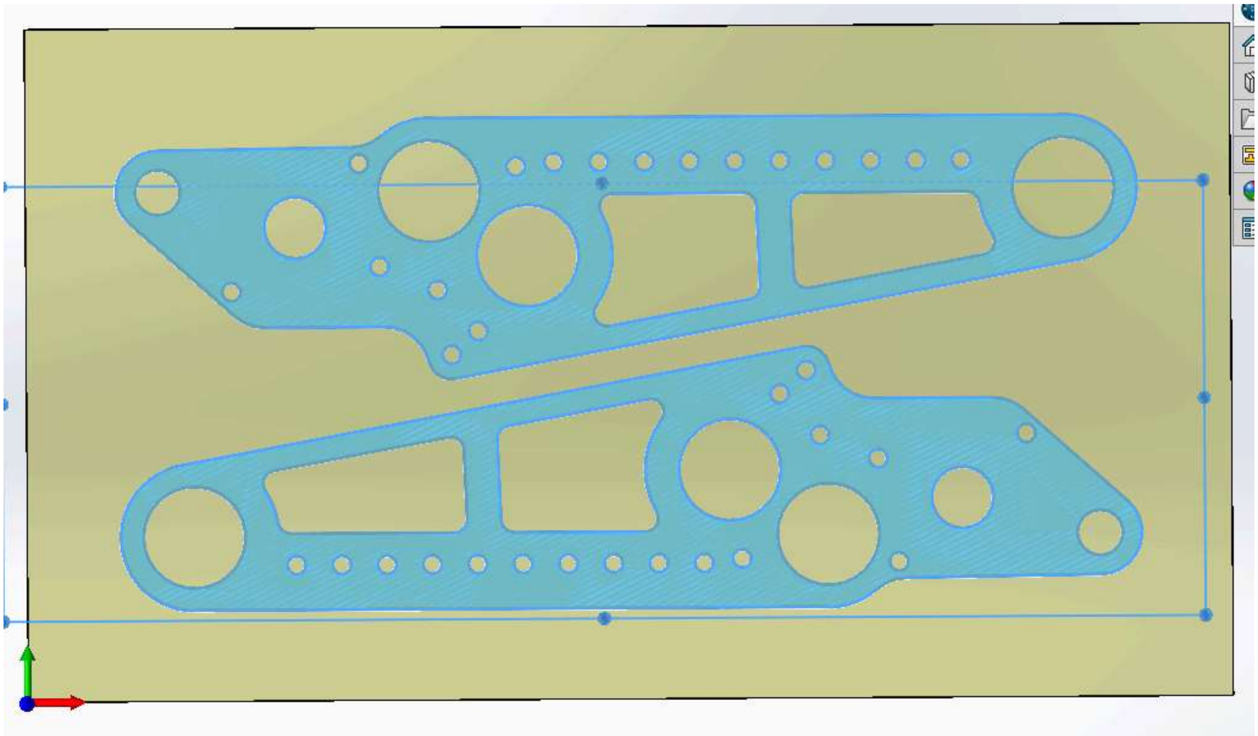
Defining of stock and coordinate system for plate parts (Example is for the router)

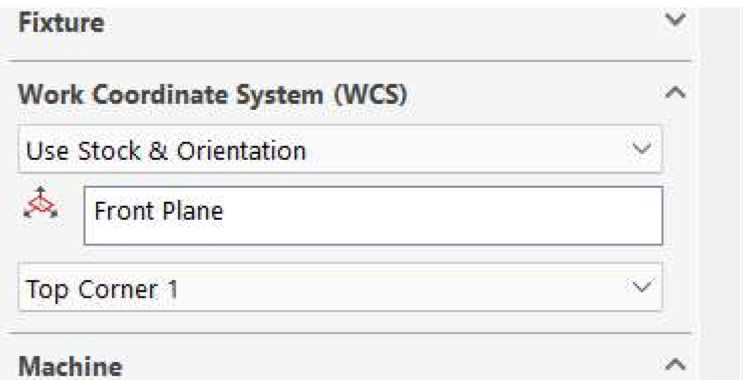
The first step is to create a Job in the CAM software. Select the CAM tab and then select “Job”.



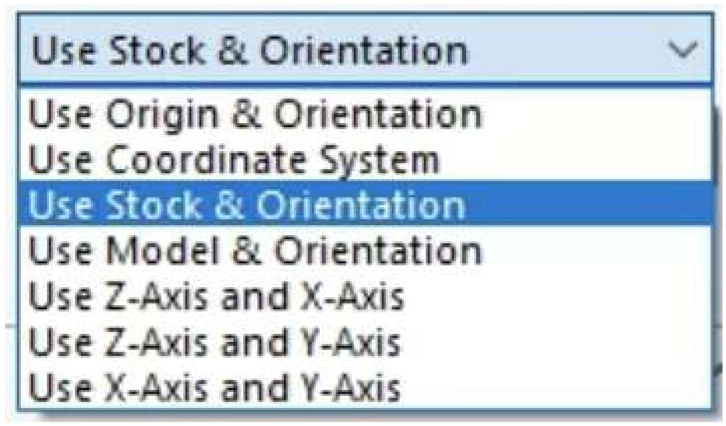






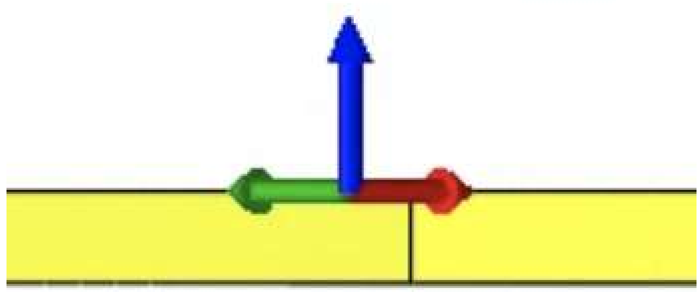


- Select parts under “model”
- Stock: Add Stock to All Sides
    - Offset by 1in to all X and Y sides
- Continue machining from previous job: OFF
- Work Coordinate System:
    - Tool orientation and direction



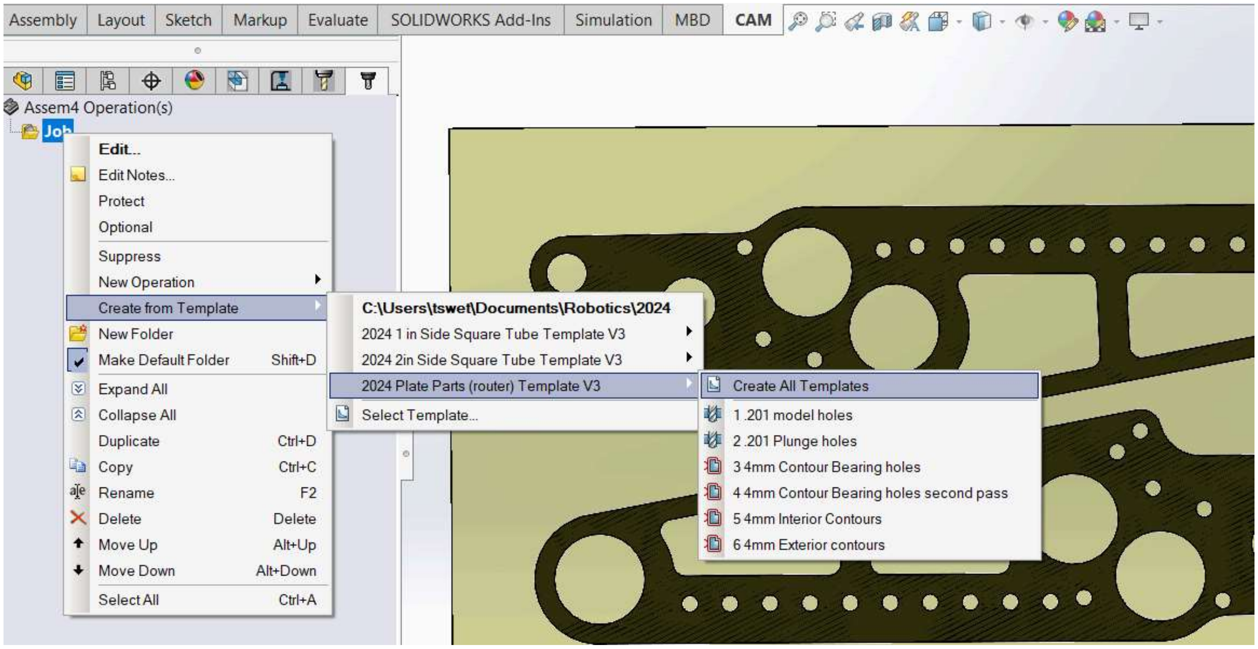
- Normally use “Use Stock & Orientation” and select proper front face of the part → may be right, top, or front depending on how the part was designed
- If it doesn’t work properly, use one of the last 3 (define by two axes) → pick two edges that run parallel to the axes you want to define

○ WCS Origin

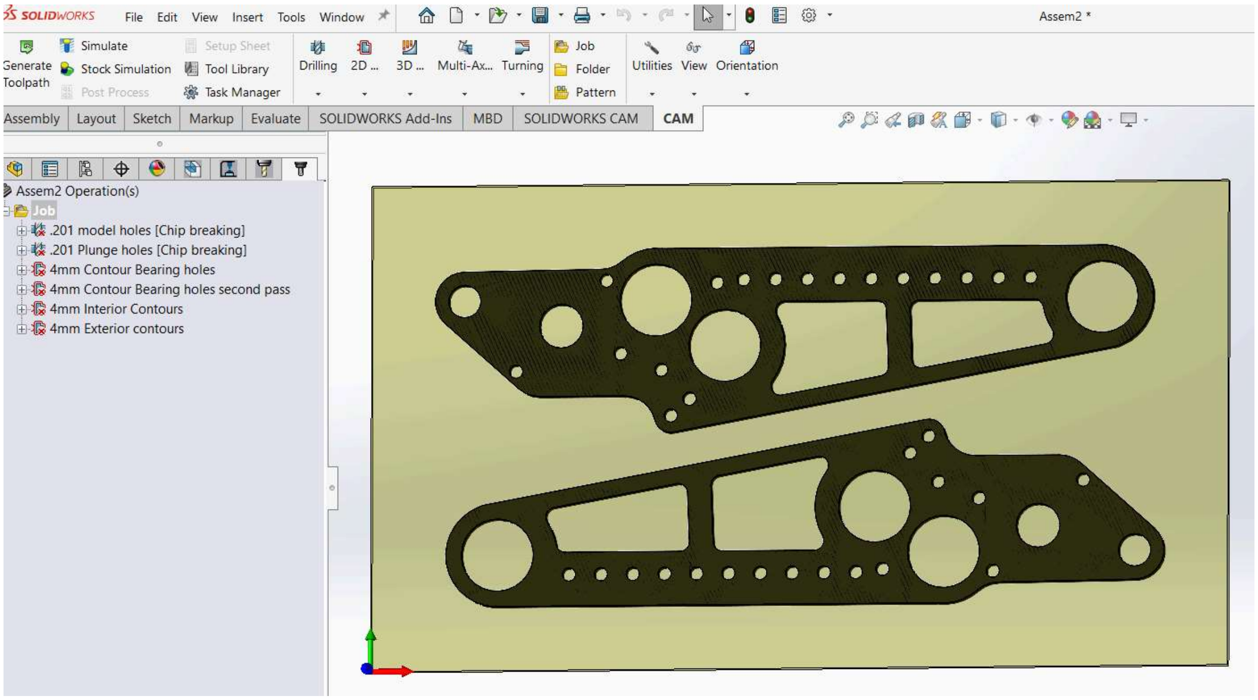


- Normally use “Top Corner 1” → use to check above setting, make sure the origin is sitting on the correct side and facing the correct way
- This will put the coordinate system at the front left corner - team standard..

Use templates to build out the CAM tasks quickly. Right click on "Job", then click on "Create from Template" and then "Plate Parts (Router)", and the "Create All Templates".



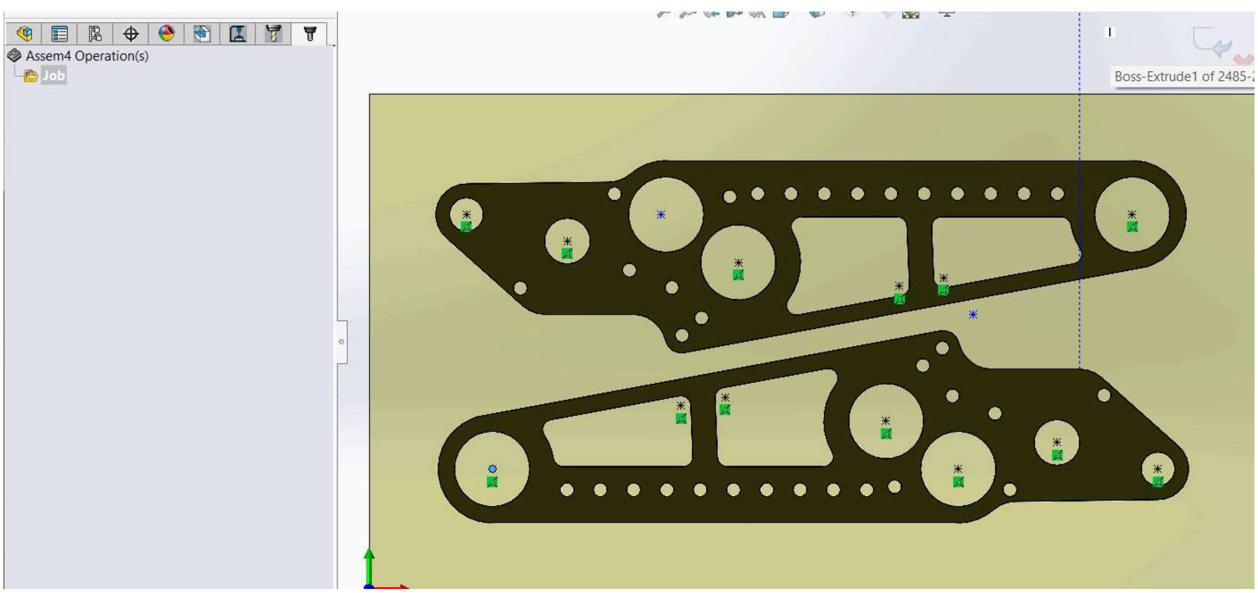
This will create all likely tasks you will need for plate parts. If you don't need individual tasks just delete them.



Holes:

Drill

First sketch in the appropriate plunge holes. This can be done within the Job. Ensure the holes are more than the radius of the drill away from the contour and that the plunge hole is at least as big as the end mill that will be used.

1. 

2. To begin the Drill steps, in CAM, select Drilling and then Drill
3. First drill step - Is to drill the model holes. We have changed this over the years and now drill model holes first. We drill the model holes and any extra hold down holes and then stop and put in the hold down screws. Then we will drill the plunge holes. This way we don't risk putting a hold down screw in a plunge hole and destroying an end mill.
4. Model Holes
   a. Tool (First Tab):
      - Specify tool: select Library, then select 2485 Tool Library
      - To minimize tool changes, we select .210 Drill from the list (for this example). There are 2 tools of the same type in the 2485 tool library - tools #17 and #18. If you chose #17 for the model holes and #18 for the plunge holes, the program will put in the stop codes so the router will automatically stop so you can put in the hold down screws.
      - Leave tool characteristics as calculated defaults

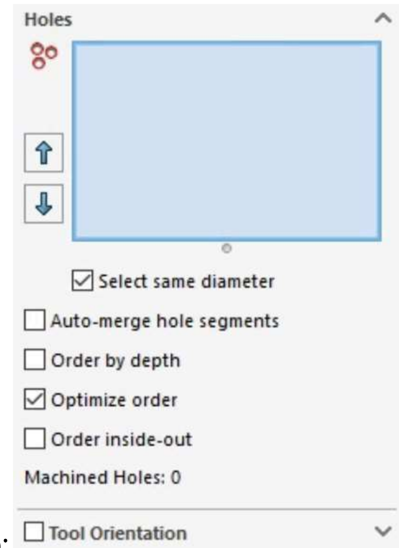
In general - Tool definition:

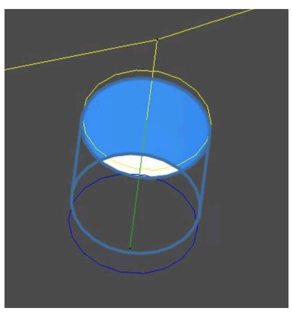
- Cutter > Type: Drill
- Cutter > Diameter
- Cutter > Tip Angle: 118° (occasionally 135°, but not necessary to check)
- Feed & Speed > # of Flutes

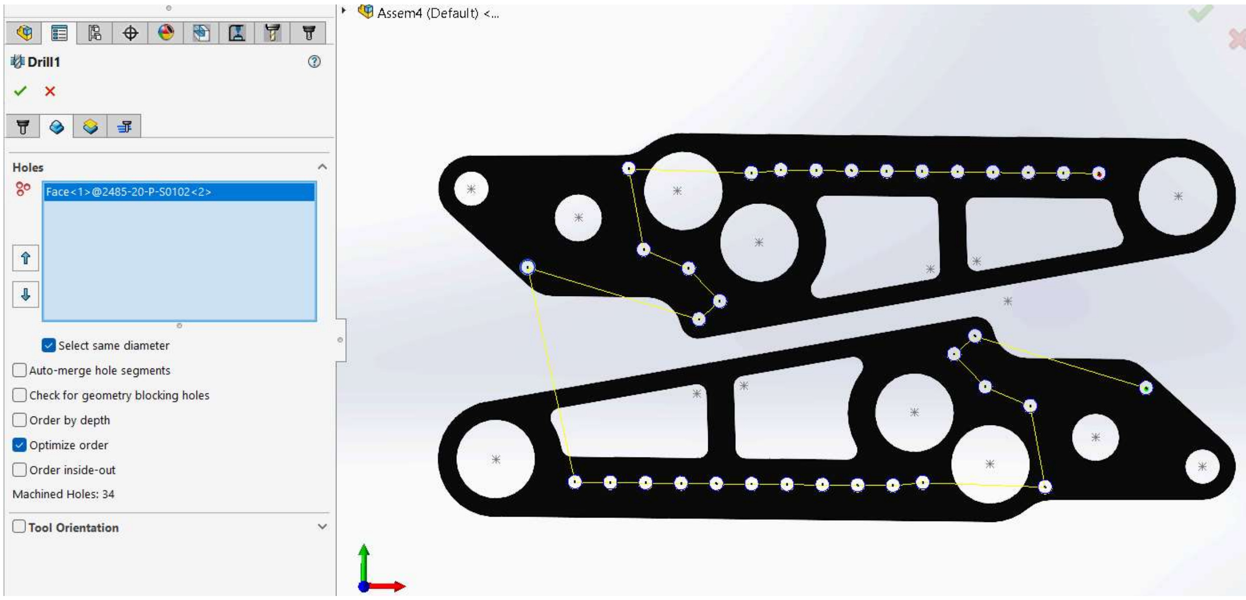
Common hole sizes:

Hole classification	Decimal conversion	Drill size
1/4-20 clearance	.257 hole	.255 drill
1/4-20 tap	.201 hole	13/64 drill (closest in size)
8-32 clearance	.177 hole	
10-32 tap	.159 hole	
8-32 tap	.136 hole	
M3 clearance (close fit)	.126in/3.2mm hole	.125 drill
6-32 tap	.107 hole	

b. Geometry

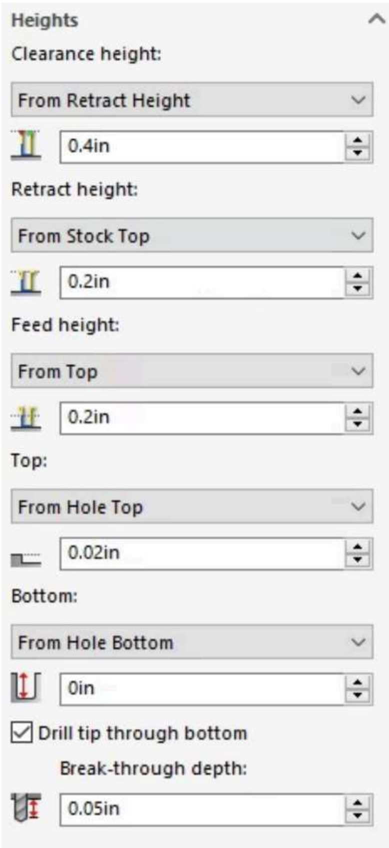


- Geometry (Second Tab):
- Select interior face
    - 
    - Select a face, not the edge → tells the CAM more about the hole and it's depth, which it likes
- Select same diameter: ON (this will select all the holes of the same size automatically)
- Tool orientation: OFF



Drill1
Holes
Face<1>@2485-20-P-S0102<2>
Select same diameter
Auto-merge hole segments
Check for geometry blocking holes
Order by depth
Optimize order
Order inside-out
Machined Holes: 34
Tool Orientation
Assem4 (Default) <...

a. Heights (Third Tab):



- Top: From Hole Top - 0.02in
- Bottom: From Hole Bottom - 0in
- Drill tip through bottom: ON - 0.05in (or up to -.1 in depending on how rough the sacrificial board is).
  - Don't add depth to the bottom height, select the "Drill tip through bottom" → this accounts for the angled tip of the drill and adds the specified depth
  - Positive because it is a specified distance it is traveling, not a coordinate

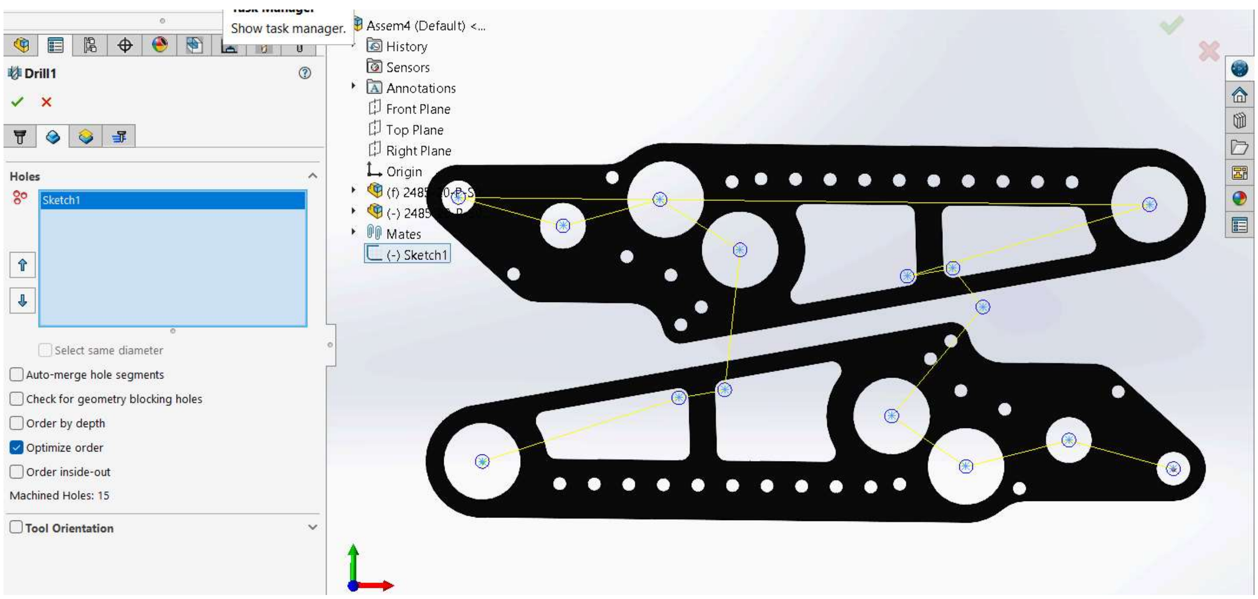
Note: At this point Stock top/bottom or model top/bottom will both work

b. Passes (Fourth Tab):

- Cycle: Drilling - rapid out

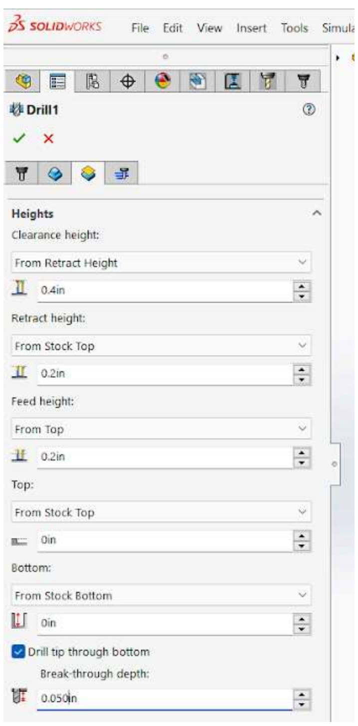
5. Now for the Plunge holes
   a. Tool selection is tool #18 (explained above)
   b. Geometry (Second Tab)

Click on the Feature tree at the top of the main window and select the sketch you just made. It will probably be Sketch #1. This will automatically select all the holes in the sketch. Alternatively you can select each point separately.

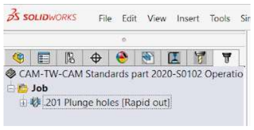


c. Heights (Third Tab)

Because the plunge holes are just sketches, you must select - stock top and stock bottom. Select break through and then enter additional cut through - this is a positive value.



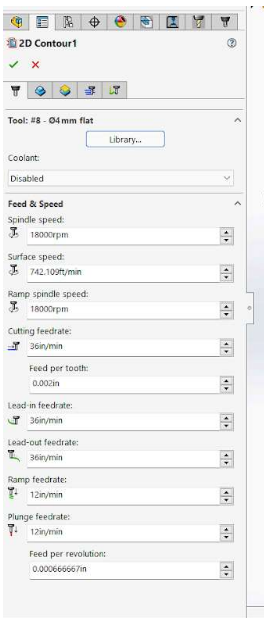
d. Rename the step: .201 Plunge Holes


SOLIDWORKS
File Edit View Insert Tools
CAM-TW-CAM Standards part 2020-S0102 Operatio
Job
201 Plunge holes [Rapid out]

Bearing Holes (and other circular holes)

Use *2D Contour*

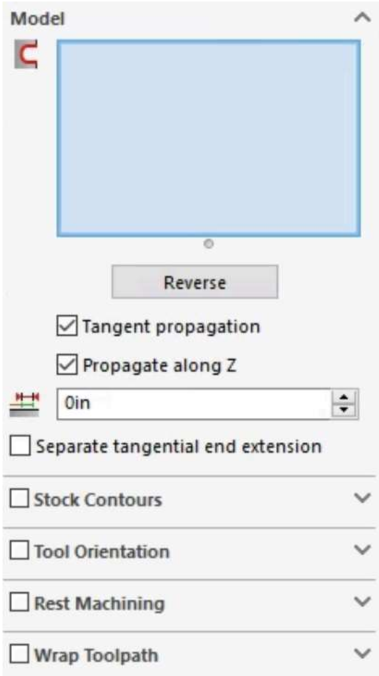
1. Tool definition: Select end mill (4mm for the router), ¼ in, 3 flute for the mill



Tool:

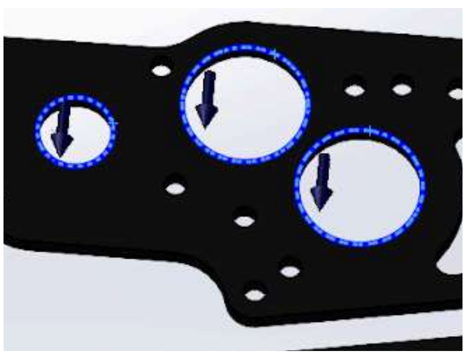
- Leave as calculated defaults. Note for the 4mm single flute, carbide end mill, normally use .002in chip load at 18000 RPM. For very large jobs we may change this to .003in to reduce machining time.

2. Geometry:



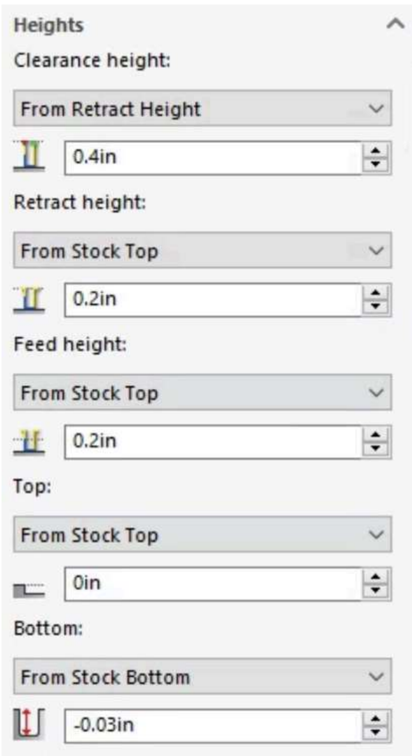
3.

- Select upper contour



  - Make sure that the arrow is on the correct side (in space, not inside the material) → if not, select troublesome edge in selection menu, then press "Reverse"
- Tangent propagation: ?? - unsure of function
- Propagate along Z: ?? - unsure of function
- Stock Contours: OFF
- Tool Orientation: OFF
- Rest Machining: OFF
- Wrap Toolpath: OFF

4. Heights:



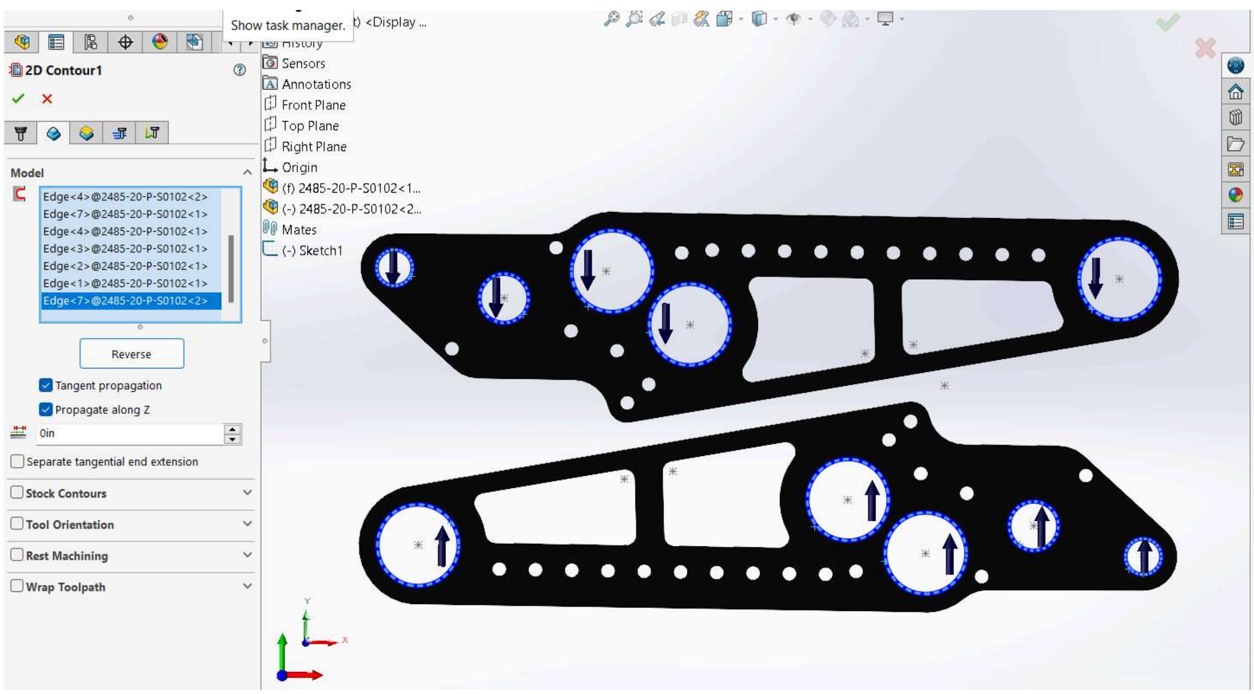
- Top: From Stock Top or From Contour - 0in
- Bottom: From Stock Bottom: -0.03in (norm for mill. For router use -.05 to as much as -.1)
  - Negative because it needs to go below the stock → is specified in relation to the z direction
  - For large parts on the router start with -.070, then measure the corner to corner difference from the center and increase the cut through as needed.

3. 2D Contour: Depths.: For > 1/8in plate, click off multiple depths. For thicker material, select multiple depths and a Maximum Roughing Step down of .1in. The program will determine the depth of each step. Set Use Even Stepdowns. Finishing passes.Do not use Finishing passes. We don't need them for any cut other than bearing holes. For bearing holes, because of an error in the HSM Works program, to get a finishing pass we add a separate Contour step immediately after

the Bearing Hole step that is a full depth contour. This will act as a finishing pass to take out the tool deflection of the first pass and help to make a better bearing fit.

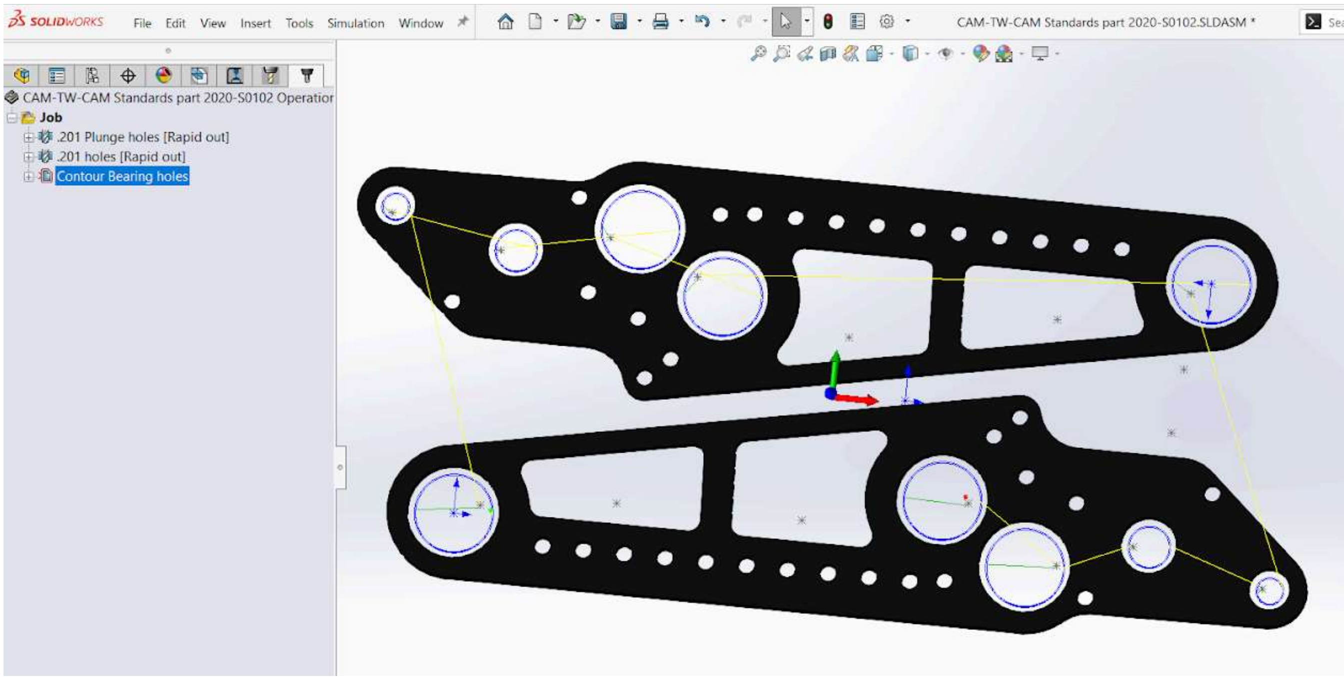
- Multiple finishing passes: 0
- Multiple depths: ON
- Maximum stepdown: 0.125in
- Use even stepdowns: ON
  - Maximum stepdown of 0.1 + even stepdowns ensures a good amount of passes → 2 passes for 3/16 in, 3 passes for ¼ in
- Stock to Leave: OFF
- Smoothing: Doesn't matter
- Feed Optimization: OFF

4. Linking:
   Lead-in and Lead-out are not necessary when using plunge holes
   Select Pre-drilled Positions, then select all the Plunge holes sketched in the bearing holes (and other round holes). They will show up in the blue window.



- Lead in: OFF
- Lead out: OFF
- Ramp: OFF
- Ramping angle: 2deg

- Entry positions: OFF
- Pre-Drilled Position(s): ON



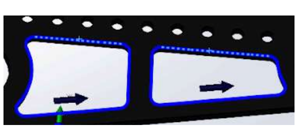
Second Bearing Contour Step

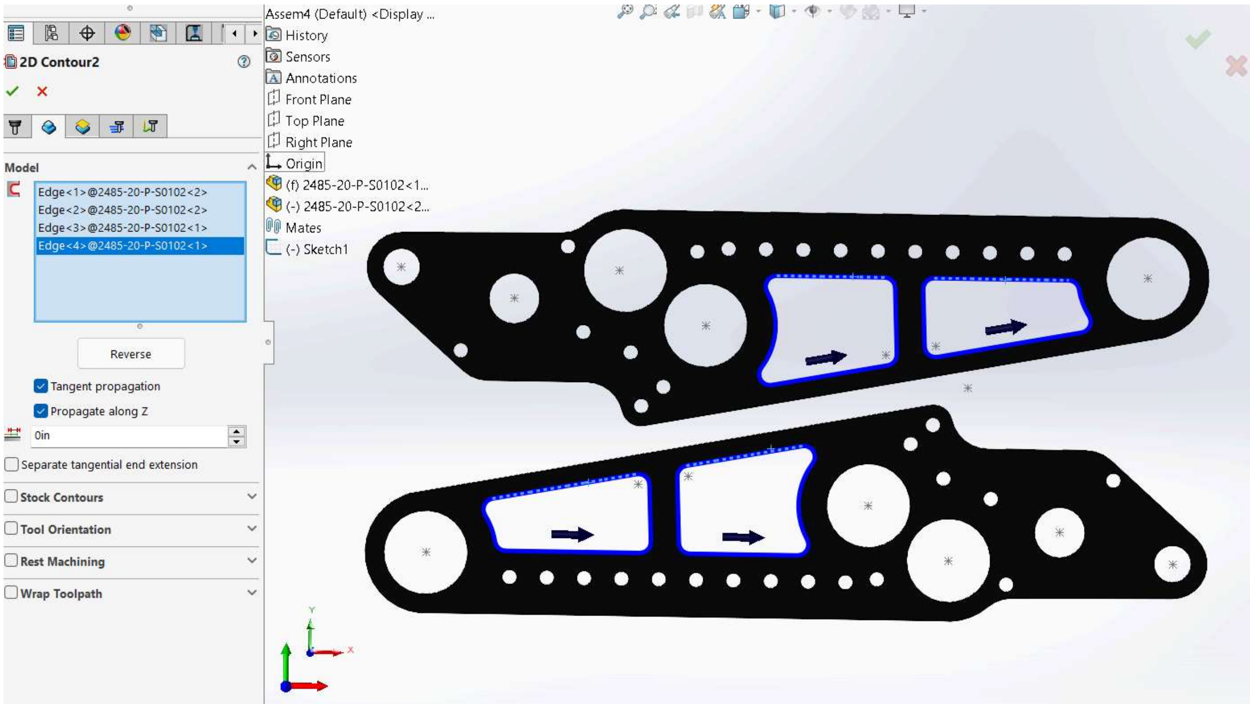
Add an additional step with a second Bearing Hole Contour. This will be a full depth cut, regardless of how thick the material is. This will act as a finishing pass and will help make seating the bearings easier because it takes a very small amount of material off.

Interior Contours

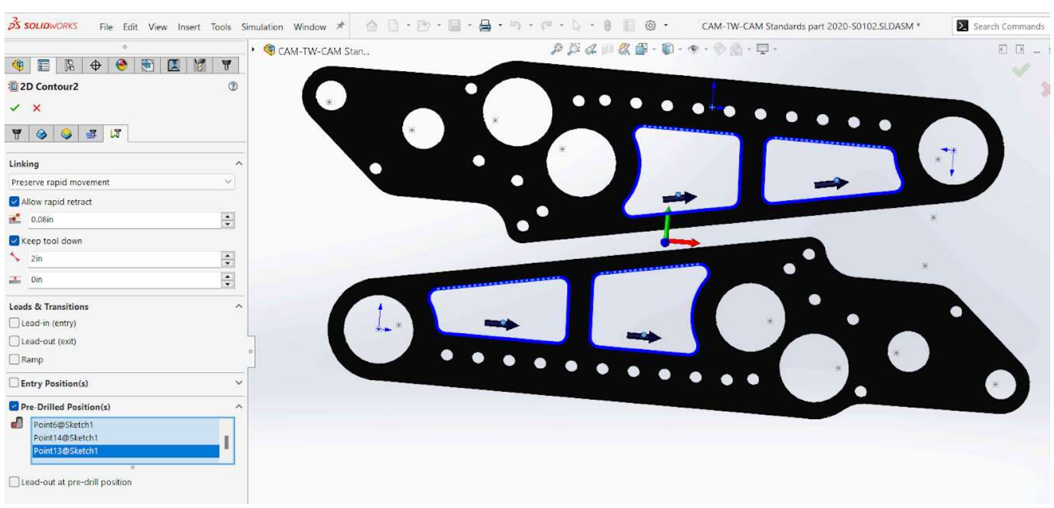
Use 2D Contour - Create a new contour line for all interior contours.

1. Tools. Same as bearing holes
2. Geometry. Like the bearing holes, choose the upper contour line of the holes and ensure the arrow is on the inside of the hole.





3. Heights. Same as bearing holes
4. 2D Contour: Depths.: Same as bearing holes
   Deselect finishing passes. (None are needed)
5. Linking: Same as bearing holes except select the plunge holes from inside the contours.



2D Contour

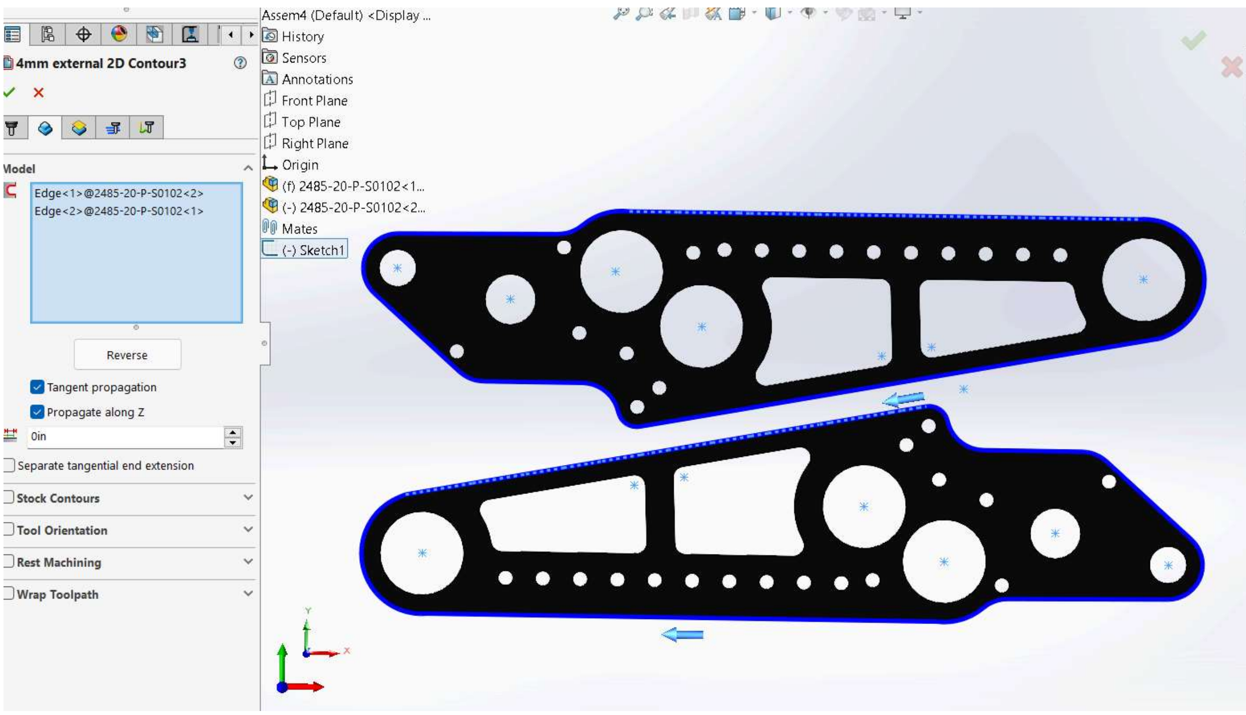
Used for the outside contour

IMPORTANT NOTES:

- Ensure that the tool path lines don't overlap
    - Overlap means that the tool is cutting into the other part since the distance between the parts would be less than the diameter of the tool

Use 2D Contour - Create a new contour line for all interior contours.

1. Tools. Same as bearing holes
2. Geometry. Like the bearing holes, choose the upper contour line of one of the outside contours and ensure the arrow is on the outside of the exterior contour.

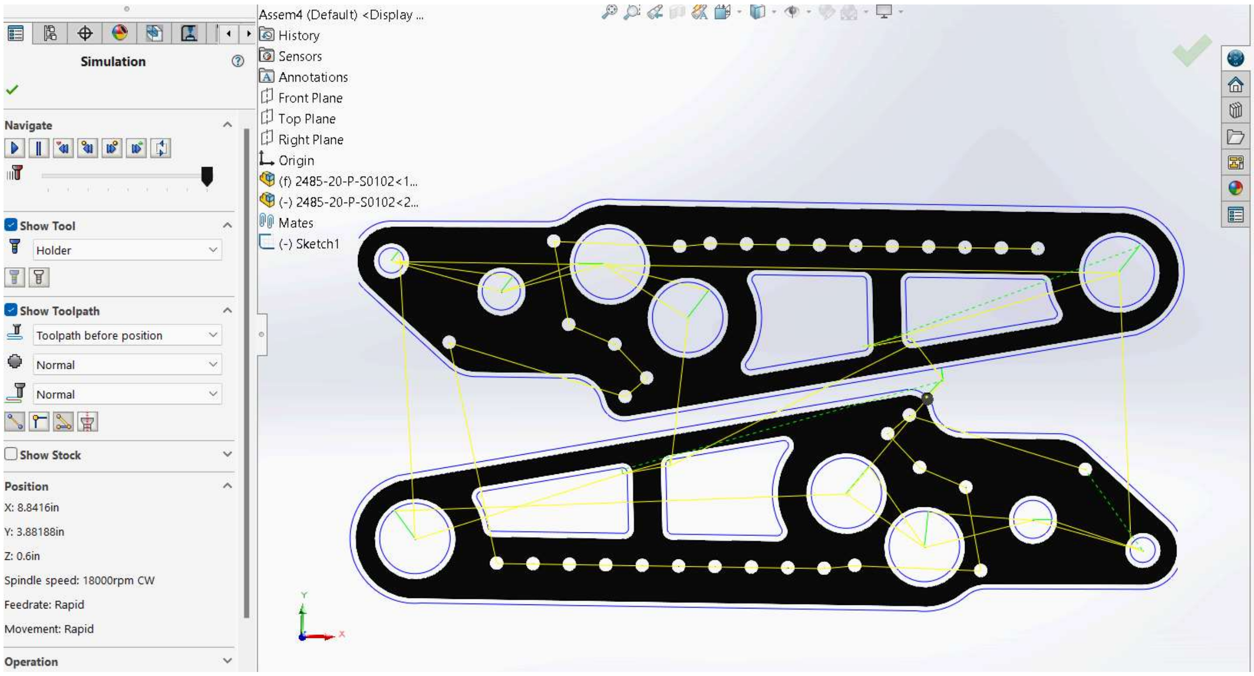


- Tangent propagation: ?? - unsure of function
- Propagate along Z: ?? - unsure of function
- Stock Contours: OFF
- Tool Orientation: OFF
- Rest Machining: OFF
- Wrap Toolpath: OFF

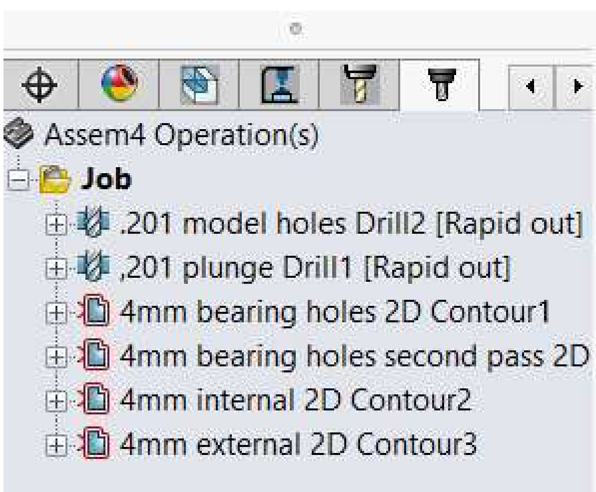
3. Heights: Same as interior contours and bearing holes

4. 2D Contour. Same as interior contours
5. Linking: Same as interior contours except select the plunge hole for each exterior contour.

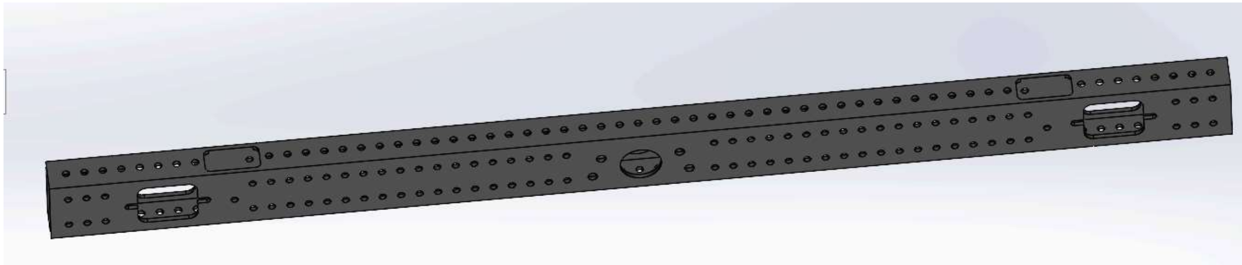
Run the Simulation and check the tool paths and ensure there is proper cut through for each operation.



Ensure all operations have been renamed to be descriptive.



Drive Train Rails (and all square tube)



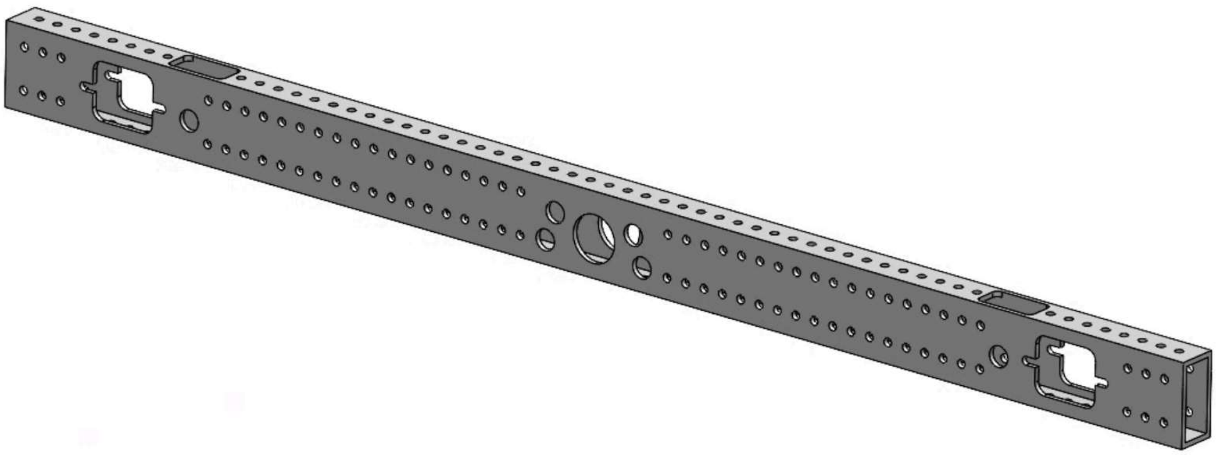
IMPORTANT NOTES

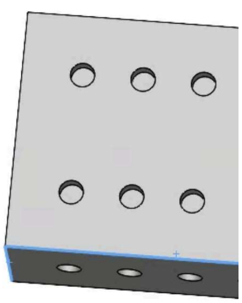
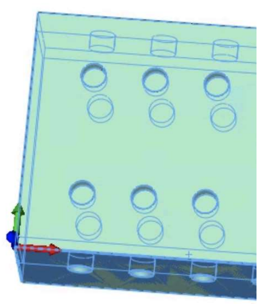
1. This will require as many as four separate sets of CAM (possibly). This can be done in a single CAM file with up to 4 separate jobs. The jobs would then be separately post processed. There would be one job for each side of the 2in side and 1 or 2 for the 1 inch side.
2. On the two inch sides, holes can be drilled with stubby drills and still go all the way through on the first side you CAM. For holes with access holes on one side, through drill the bolt size hole first.
3. On the one inch side of 1X2 square tube, drill each side separately. Use stubby drills. This approach entails one additional edge finding, but is more accurate and breaks fewer drills..
4. For the milled slots, bearing holes, etc. you must mill only one side at a time. This is achieved by setting the milling depth at -0.155 from stock top (can go deeper but it might cause more passes than necessary.

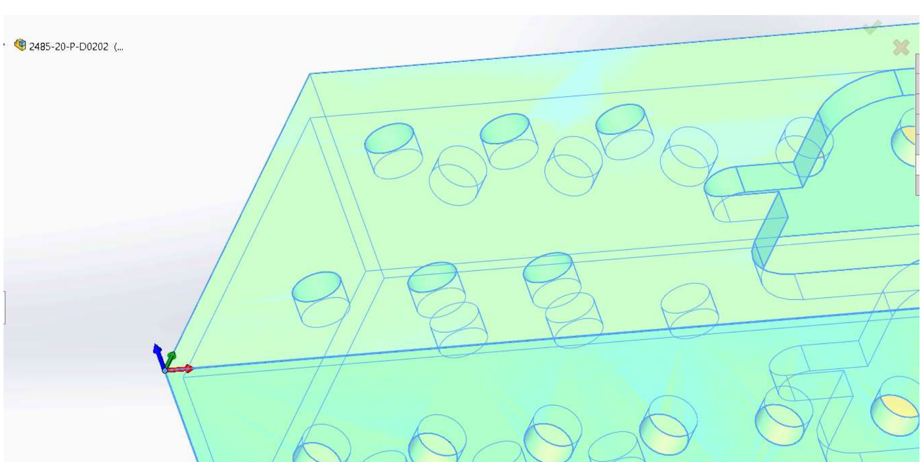
JOB

Job:

Defining of stock and coordinate system for non-plate parts like rails



- Select parts under "model"
- Stock: No Additional Stock
- Continue machining from previous job: OFF
- Work Coordinate System:
    -  
    - Use "Use Z-Axis and X-Axis"
    - [add picture]
    - Select "Top Corner 1" under Origin, then use the reverse buttons to put the origin in the proper corner
        - If this does not work, use the "Selected Point" option



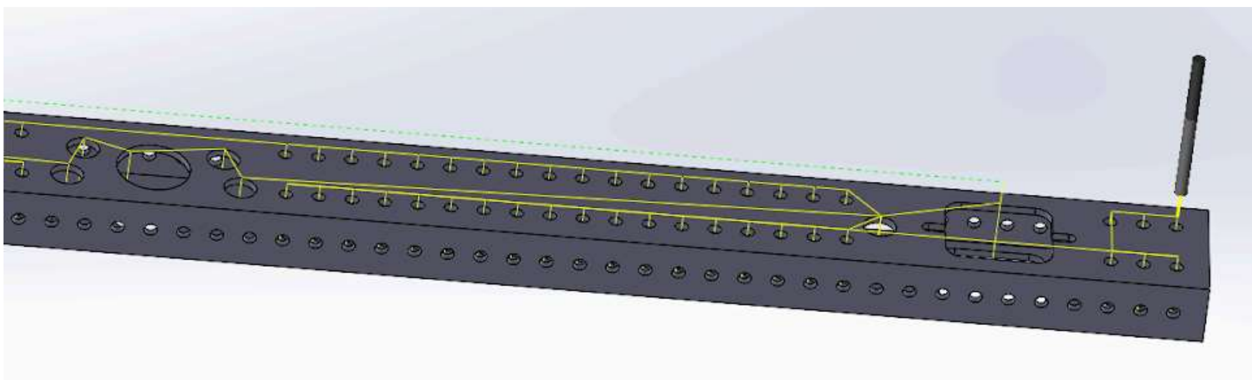
Do not add any stock.

For the Work Coordinate system,

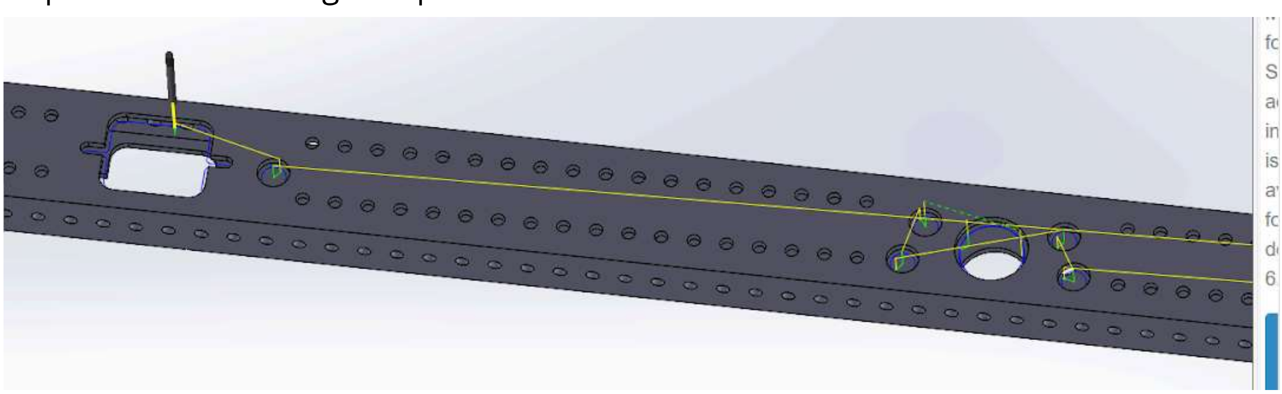
by team convention, use the front left corner - so that X and Y are always positive.

Use the “Z Axis and X Axis” and choose the correct edges and use Origin - Selected Point to achieve this orientation.

2” side.
DRILLING. Sketch in the plunge holes for the contours and large access holes. Create a drill step for the plunge holes. In this example a small end mill is required for the contours so the plunge holes can be .201. Create a second Drill step for the hole pattern in the part.

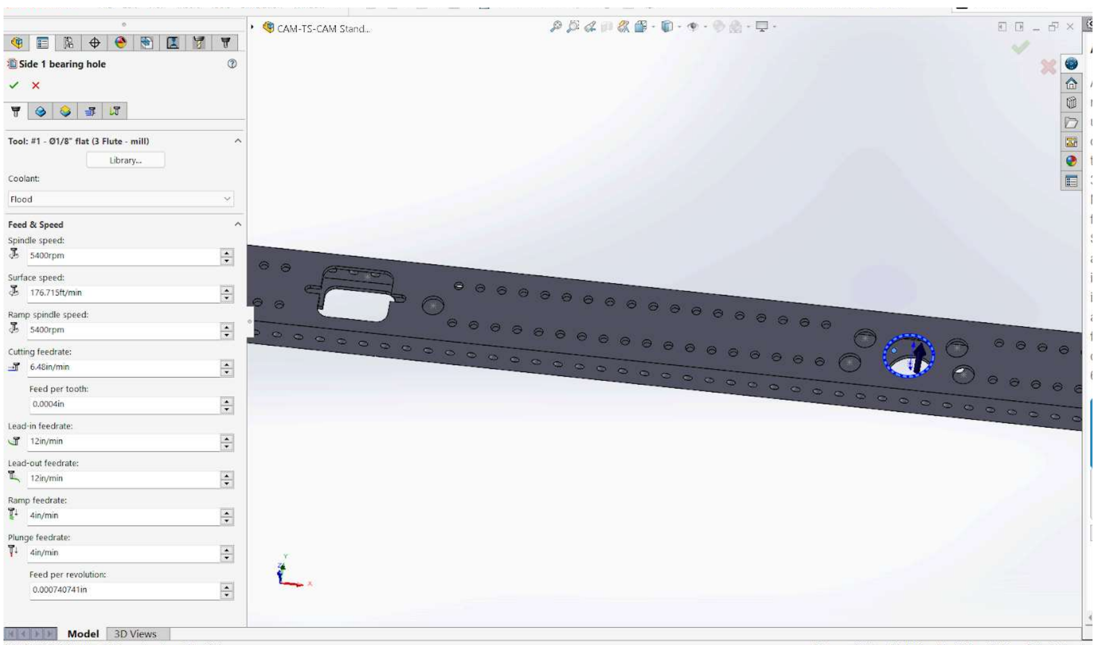


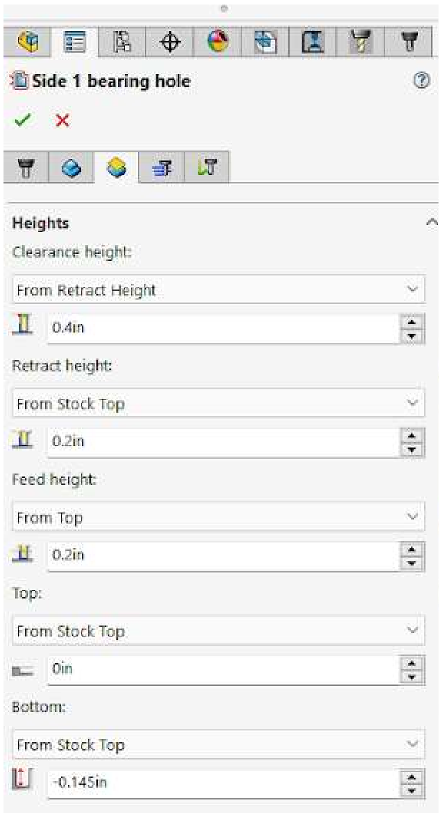
Use a ⅛” EM for the bearing block holes (this is the 3 flute, carbideEM, with a .0004in chip load at 5400 RPM) This is necessary to get the slots. Select the top contours of both bearing block holes and access holes. In the heights section set Bottom to from Stock Top to “-0.155”. Then set multiple depths at .06 with even step downs. This will give 3 passes.

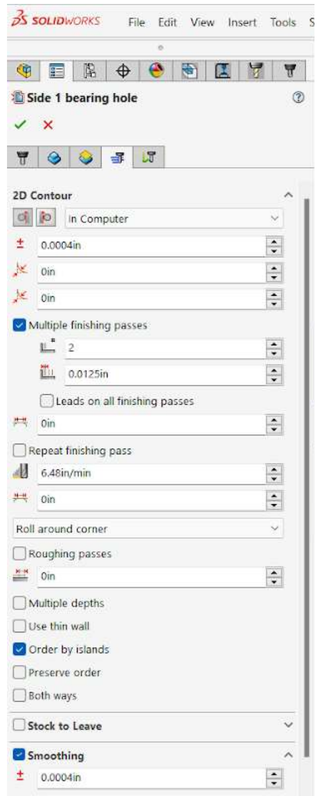


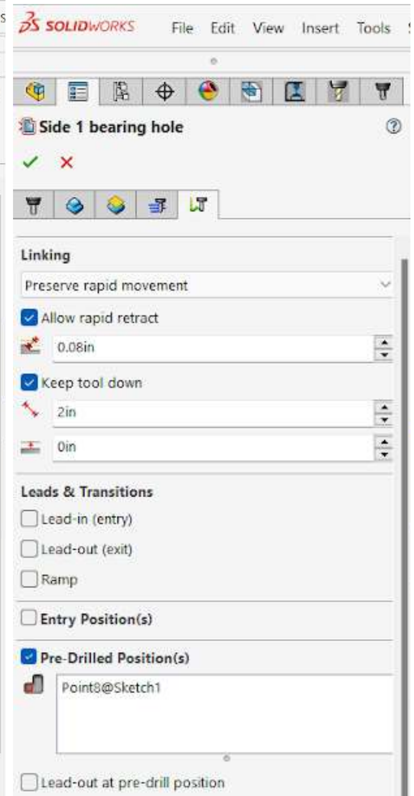
BEARING POCKETS

Use largely the same tooling/settings as the bearing block holes, except you need a second pass (separate task). Select 2 finishing passes. See settings below. This could be done with a .25in diameter 3 flute end mill as a single pass full depth cut. Or with the original 1/8in end mill with multiple depths if you don't want to or don't need to change tools. The .25 end mill is faster than the ⅛ in end mill but might require a tool change. Determine the tradeoff you want to make.



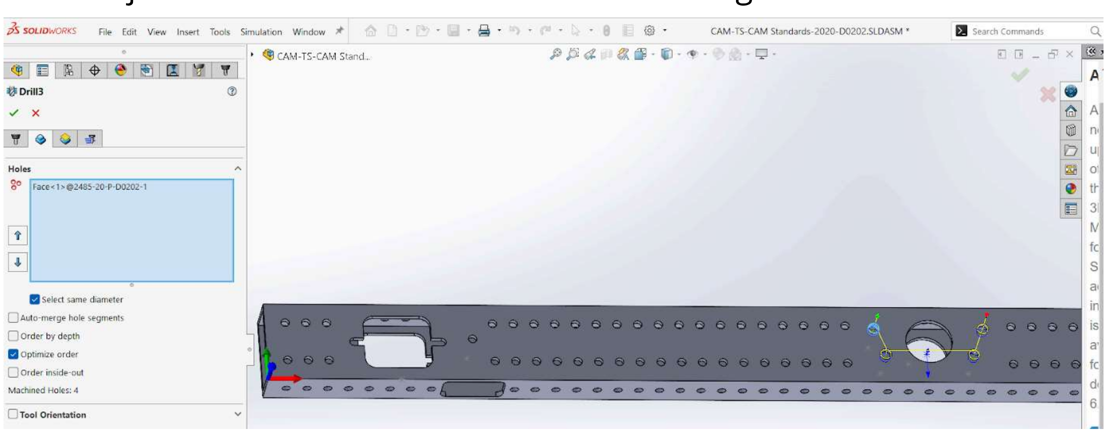




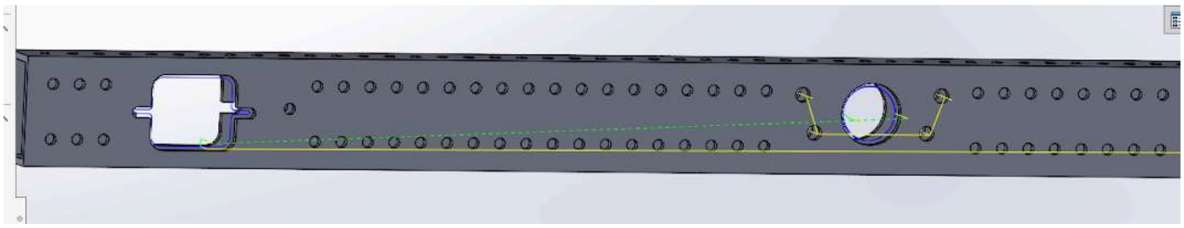


SECOND 2” SIDE

Start a second CAM or a second job for the second 2” side. Set up the stock and WCS the same way. The .201 holes have already been drilled. Create a Drill line in the new job for the 1/4in holes around the bearing hole.



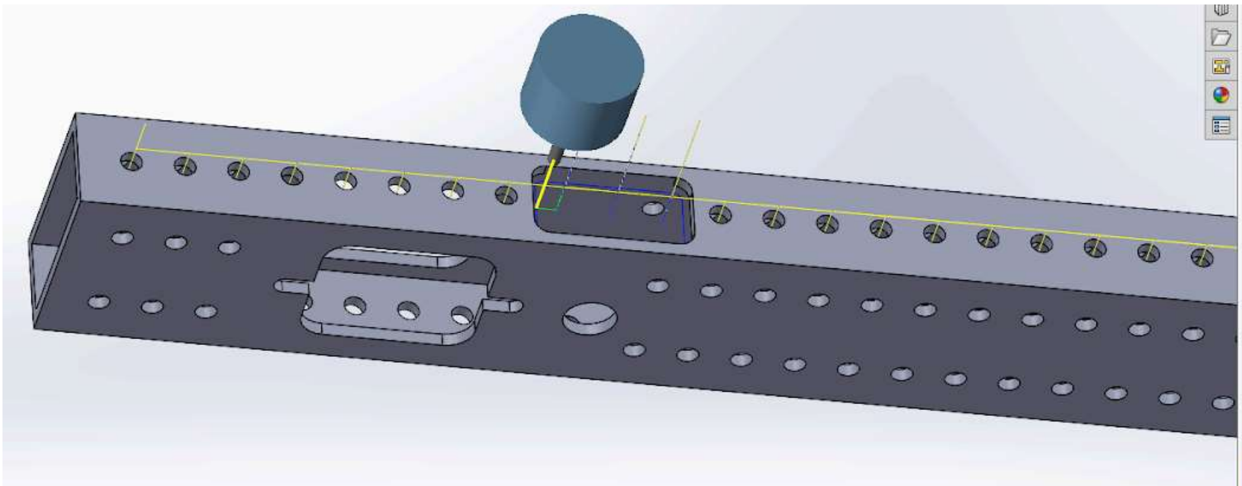
Similar to the first side, create a contour line to mill the bearing block holes using the same tooling/settings. You can use the same plunge hole sketches even though they are on the other side.



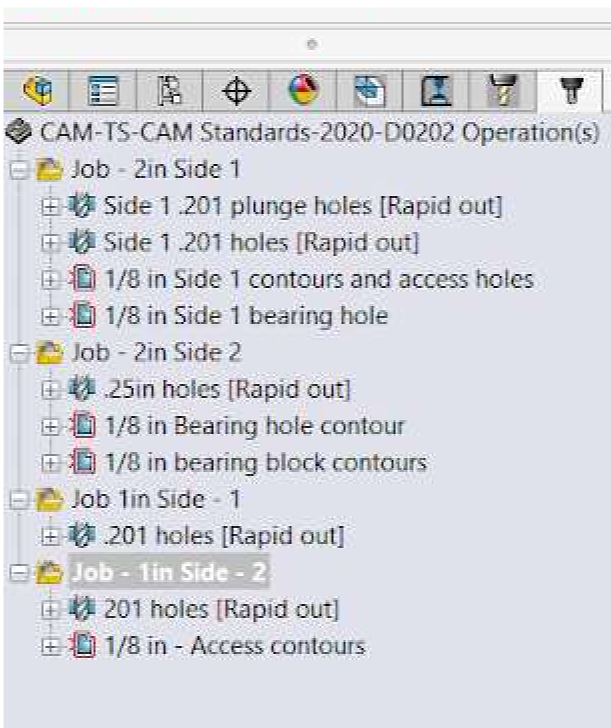
ONE INCH Side

Start a new CAM file or new job for the one inch side. Both sides will probably need to be CAM'd. You will use a stubby size drill (.201) because both sides must be drilled. Start with the side with only holes. Set up the job as before and select all the holes. Set the bottom to stock top with a depth of -.155.

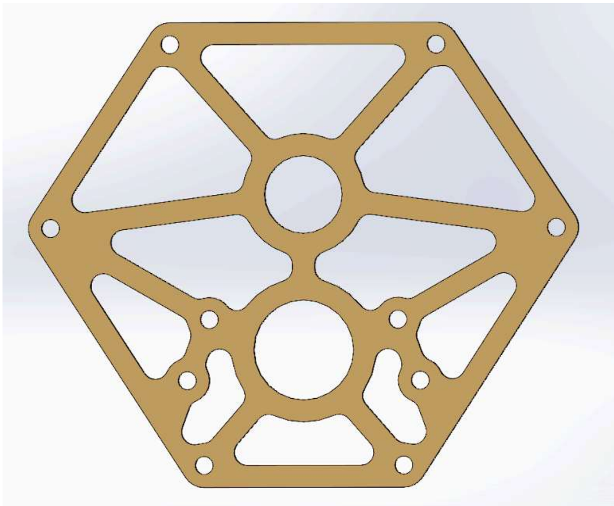
For the side with the slots, set up a new job the same way. Do a drill step as on the other side, select similar size holes. This will drill all the holes including the ones that will be cut out on the slot. This will provide us with a handy plunge hole for the slots. Sketch a point into the center of one of the holes inside the slot (you can see from the other side). Use those points for the pre-drilled positions in a contour line. Set depth by setting the bottom to Stock Top with a value of -.155in.



This is the completed job.

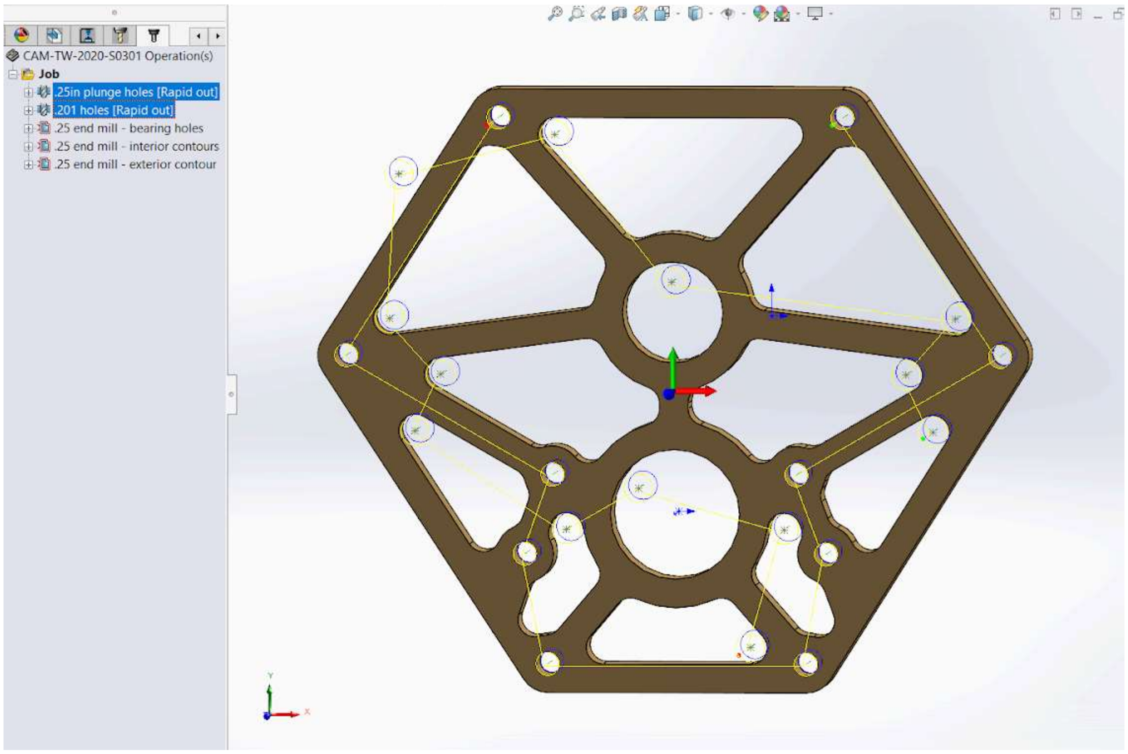


Gear Box Plate - CNC Mill



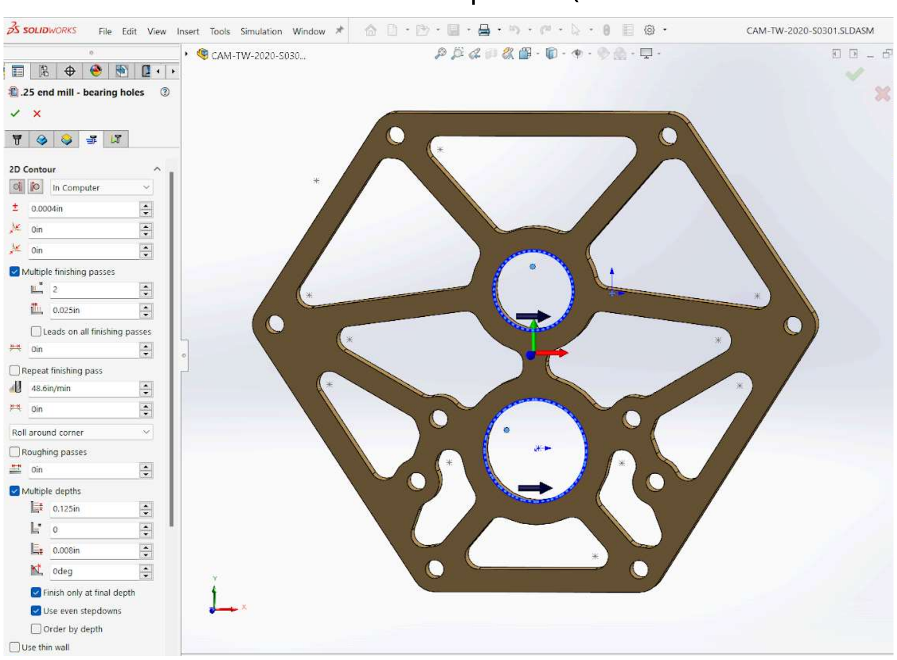
Gear box plates are typically made out of 1/4in 6061 Aluminum. We will use the 1/4in, 3 Flute, carbide end mills, with a chip load of .0012in, at 5400 rpm spindle speed for the milling. We will use contours with plunge holes for all the milling steps, bearing holes and interior and exterior contours. Doing this on the mill requires that the stock and the sacrificial board be clamped to the table. The clamps typically cover the corners of the stock. In this case set the job up with the coordinate system at top center. When setting up zeros on the mill, use the method of visually lining up each axis at zero and then the other side and divide by 2. This will give you x and y zeros at about the center of the stock.

Drilling:
We will drill the plunge holes first, using a ¼ in drill, then mark the plunge holes. Then we will drill the .201 hales in the part.



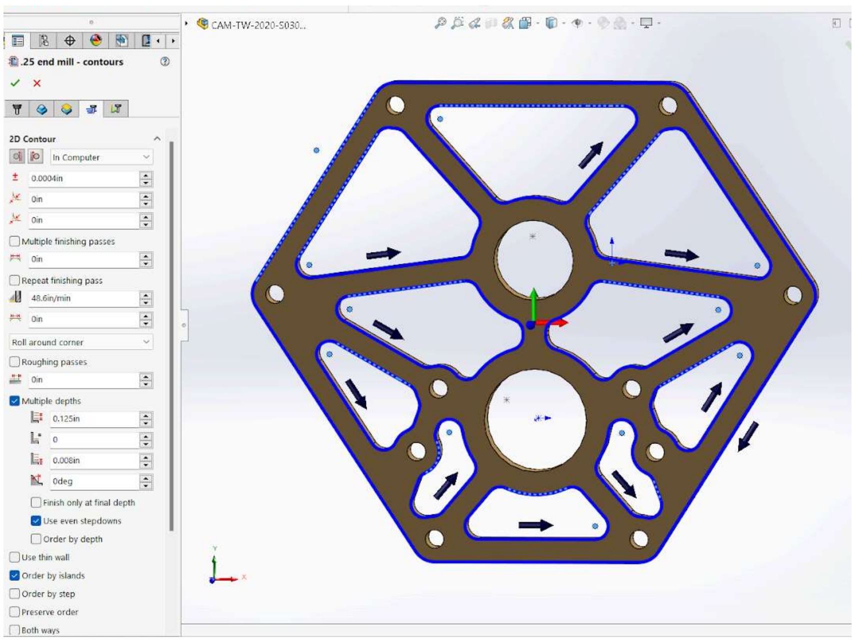
Milling:

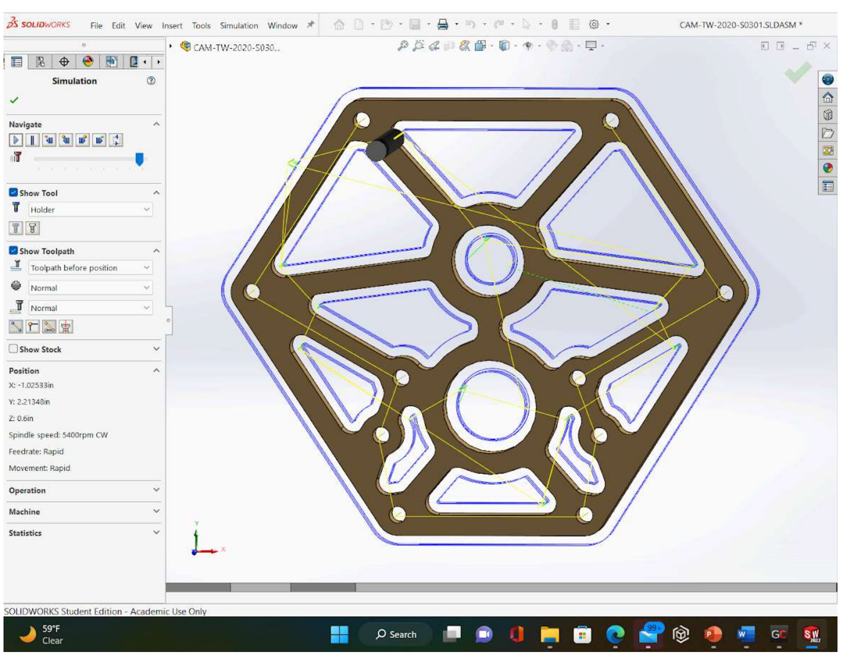
Bearing Holes: Start with the bearing holes. Use the ¼ in end mill (carbide - 3 flute). Select the .249 end mill (for the mill) from the Tool table. This is the same tool but will make the bearing hole slightly larger and fit the bearing better. This mill will use 5400RPM with a .0012in chip load (which is all set in the Library).



Set finishing passes to 0. Also set multiple depths with a max stepdown of .125 in. Also set Even Stepdowns and Finish Only At Final Depth. Select the plunge holes for the Predetermined positions.

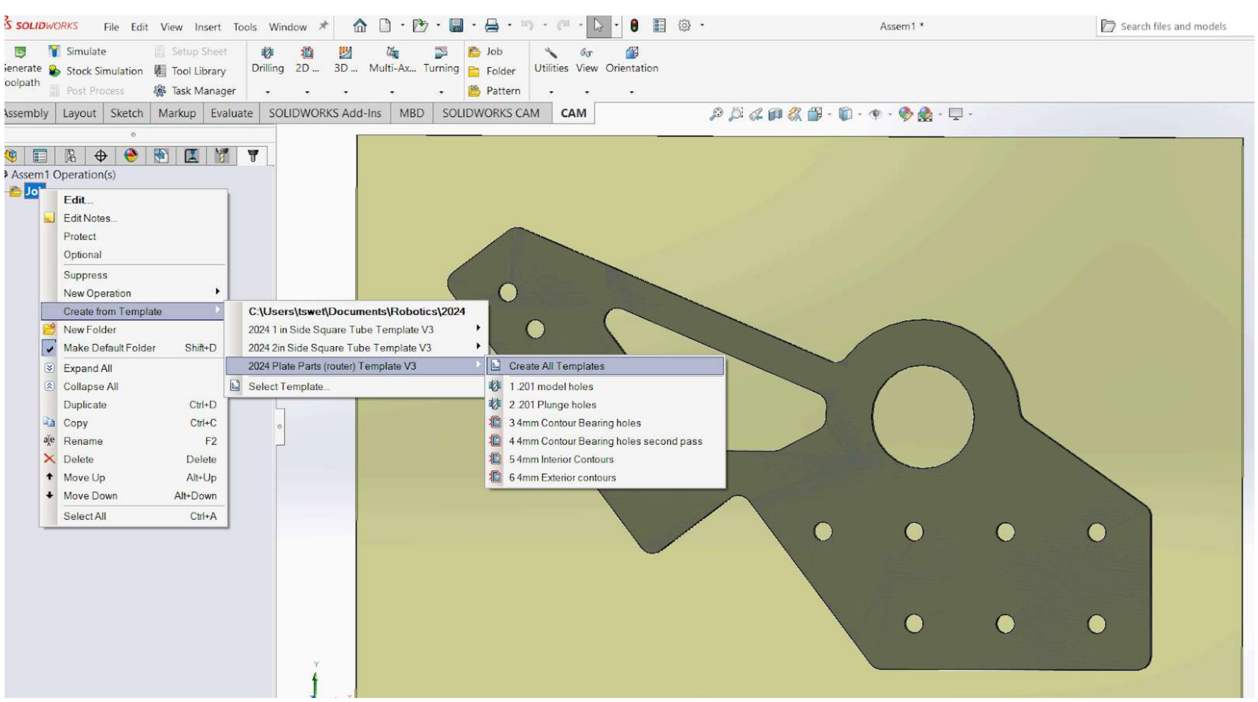
Contours: Do both interior and exterior contour together in this case (because there is only one part) Same as the bearing holes except do not select finishing passes. Bearing holes need a second pass at full depth (not a finishing pass).

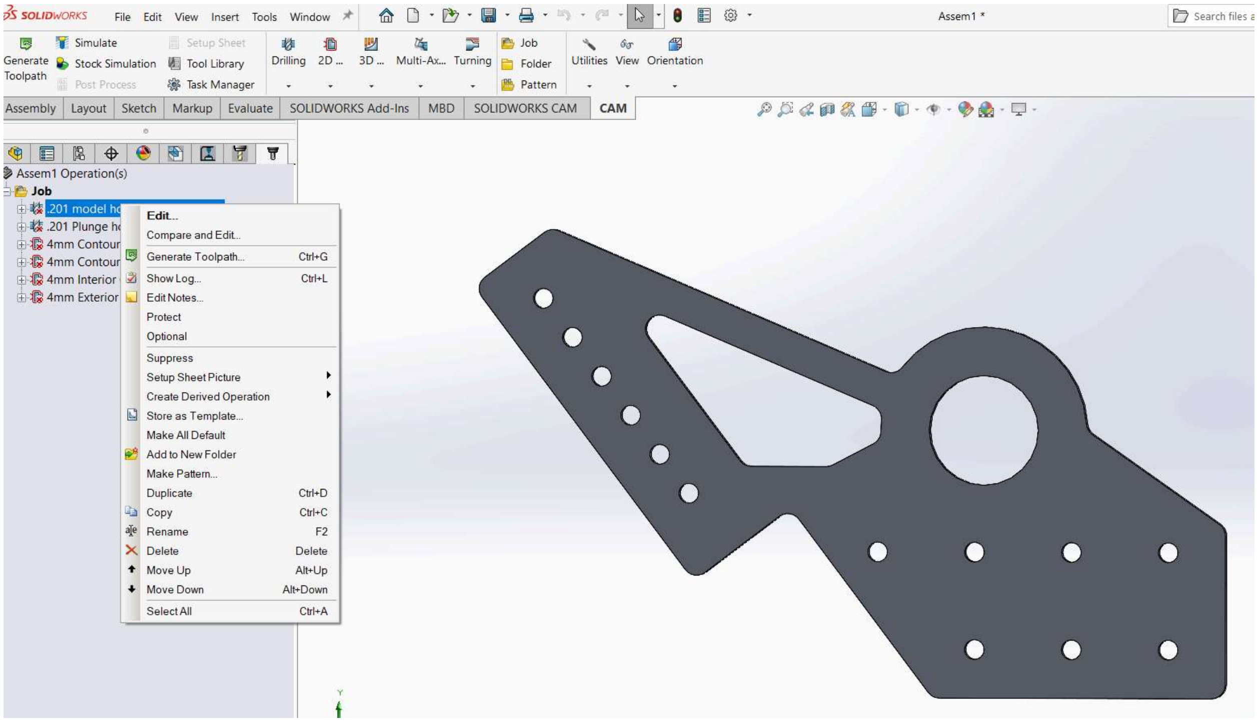




USING TEMPLATES

The CAM process for all three of the standard types of parts (plate parts, 1X1 square tube and 1X2 square tube have been developed into CAM templates. Using a template allows a much more rapid CAM process by filling in standard tools and other standard settings that only need to be checked. The images below show the process. After creating a Job, select that job and right click. In the menu, select "Create from Template". In the next menu select the type of part you are making. In the third menu select "Create All Templates". This will preload all the standard steps under the job. Now right click on each step in order and click edit. Note each step will have a red "X" on it. These X's will go away once you have edited the step and a tool path has been created. If you don't need some of the steps (for example a plate part does not have any bearing holes), then just delete the step from the job.





Machining of Solid Block parts

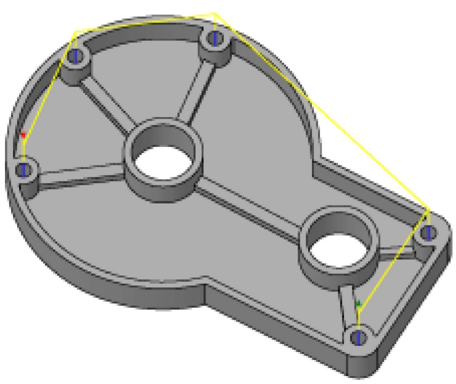
Lessons learned.

1. Try not to use the long end mills because they can cause significant tool deflection, even the 1/2in dia, 5in long end mill. Use the shortest end mill you can for the job.
2. Use 2 finishing passes of .005 each
3. Keep the stock on both sides (if you can) until the last step on each side. This keeps the depth calculations correct and ensures that you will have a zero point for you to reset zeros if they are lost for whatever reason.

Advanced Discussion of 2D and 3D Machining Strategies

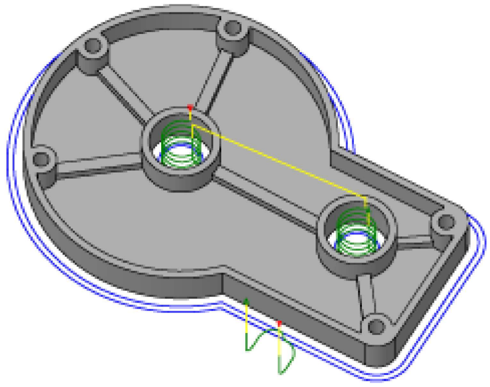
2D Machining Strategies

Drilling and Hole Making



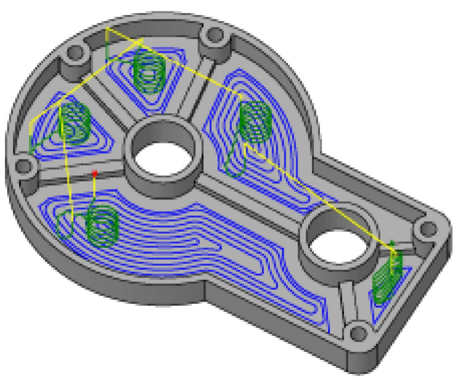
Autodesk HSM includes a powerful **Drill** tool for generating drilling, counterboring and tapping operations. The **Circular** strategy is used for milling cylindrical pockets and islands, while the **Thread** operation is used for thread milling cylindrical pockets and islands. The **Bore** operation allows you to bore mill cylindrical pockets and islands by selecting the cylindrical geometry directly. All operations are optimized to minimize tool travel and overall cycle time. Both standard and customized cycles are supported for all point-to-point operations, including spot-drilling, deep drilling with chip break, etc.

Contouring



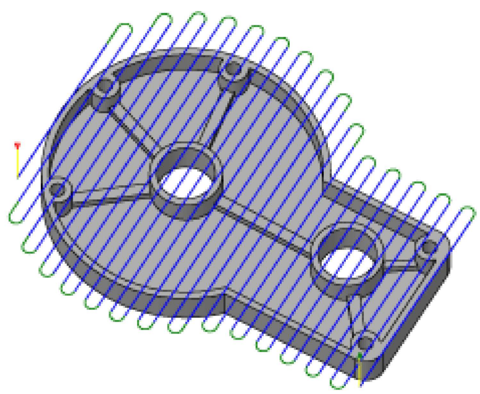
With the **Contouring** strategies, you can easily machine 2D and 3D contours with separate lead-in and lead-out, and with or without tool compensation. Choose multiple roughing and finishing passes and multiple depth cuts for any contour. Machine open and closed contours without creating additional geometry and eliminate sharp motion with corner smoothing.

Pocket



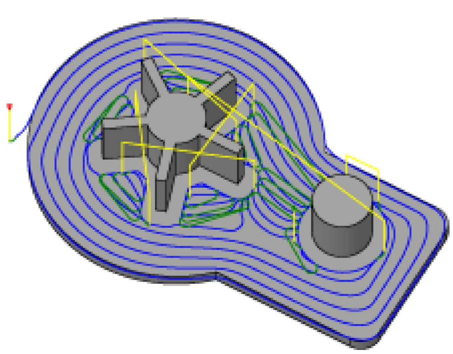
The **Pocket** toolpath is used for machining closed curves both with and without islands. The toolpath starts at the center of the pocket and works its way outward. The entry can be selected anywhere on the model and includes possibilities for plunge, ramp, or at a pre-drilled position. The special high-speed option creates a smooth toolpath and allows you to specify a maximum tool engagement. As a result, the feedrate can be increased significantly, reducing the machining time and tool wear.

Facing



The **Facing** strategy is designed for quick part facing to prepare the raw stock for further machining. It can also be used for clearing flat areas in general.

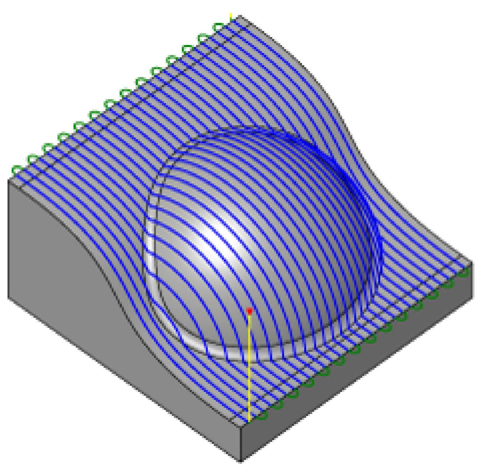
Adaptive Clearing



The **Adaptive Clearing** strategy creates a roughing/clearing toolpath inside closed curves both with and without islands. This strategy avoids full-width cuts by progressively shaving material off the remaining stock. The generated toolpath ensures that the cutting conditions remain constant with a stable load on the tool. As a result, the feedrate can be increased significantly, reducing the machining time by 40% or more which provides improved surface quality and less tool wear.

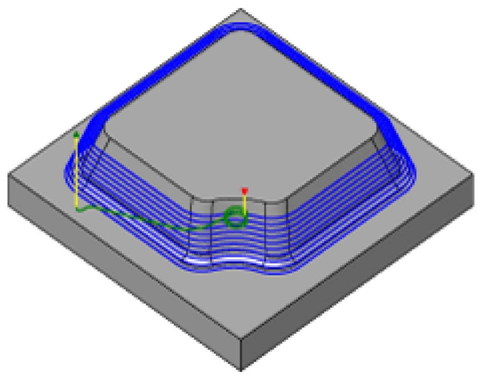
3D Machining Strategies

Parallel



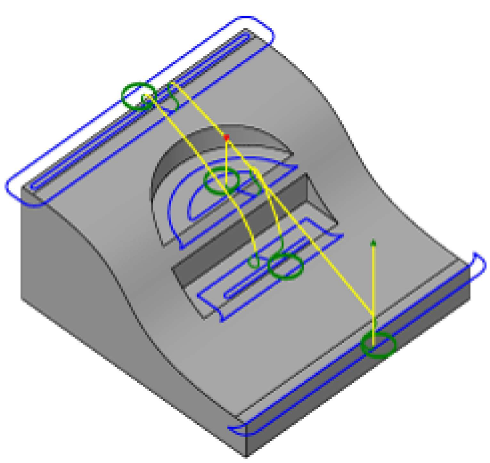
Parallel passes is one of the most widely used finishing strategies. The passes are parallel in the XY plane and follow the surface in the Z direction. **Parallel** passes are best suited for shallow areas and down milling. To automatically detect shallow areas, the machining can be limited to a maximum angle between the tool tip and the surface. By selecting the down milling option, tool deflection can be minimized when machining complex surfaces.

Contour



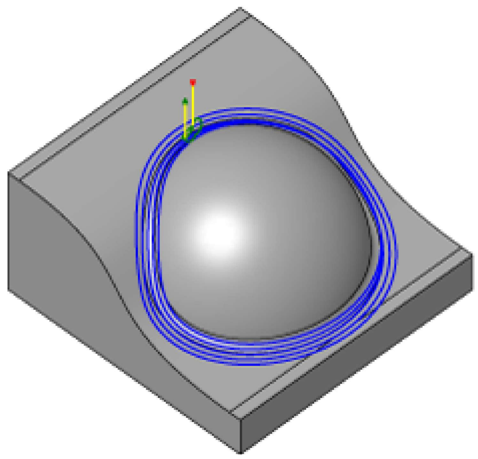
Contour passes is the best strategy for finishing steep walls, but can be used for semi-finish and finish machining on the more vertical areas of a part. If a slope angle is specified, for example 30 to 90 degrees, the steeper areas are machined, leaving the shallower areas up to 30 degrees for more appropriate strategies.

Horizontal Clearing



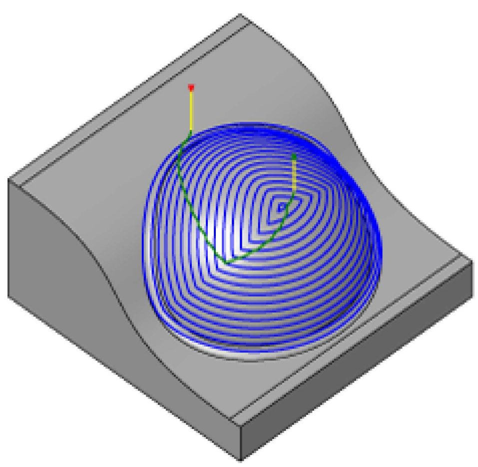
The **Horizontal Clearing** strategy automatically detects all the flat areas of the part and clears them with an offsetting path. When the flat area is shelved above the surrounding areas, the cutter moves beyond the flat areas to clean the edges. Using the optional maximum stepdown, horizontal faces can be machined in stages, making the horizontal clearing suitable for both semi-finishing and finishing.

Pencil



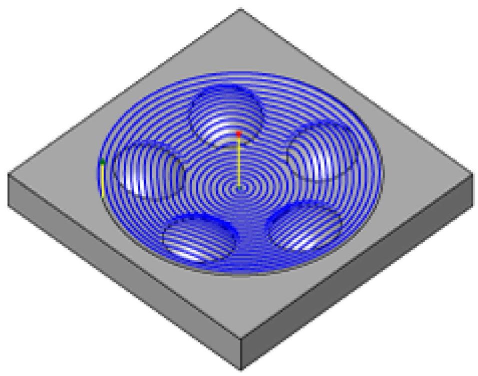
The **Pencil** strategy creates toolpaths along internal corners and fillets with small radii, removing material that no other tool can reach. Whether using single or multiple passes, the **Pencil** strategy is ideally suited for cleaning up after other finishing strategies.

Scallop/Constant Stepover



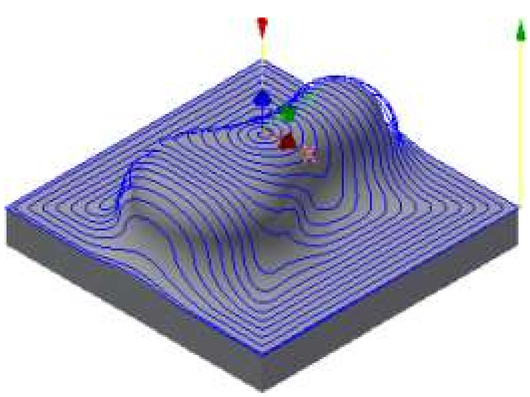
The **Scallop** strategy creates passes that are at a constant distance from one another by offsetting inward along the surface. The passes follow sloping and vertical walls to maintain the stepover. Although **Scallop** finishing can be used to finish an entire part, it is most commonly used for rest machining, following a combination of **Contour** and **Parallel** passes. Like the other finishing strategies, machining can be limited by a contact angle range.

Spiral



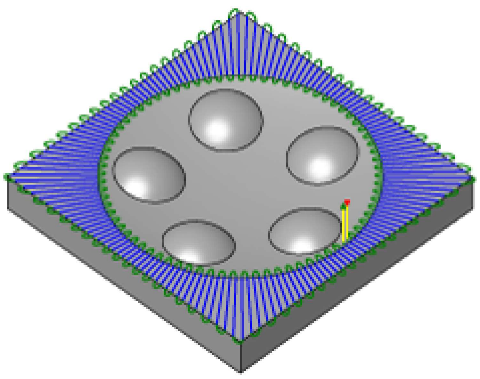
Spiral machining creates a spiral toolpath from a given center point, generating a constant contact as it machines within a given boundary. It is ideally suited for use on round shallow parts using tool contact angles up to 40 degrees, in conjunction with **Contour** passes for the more vertical faces. The center point of the detail to be machined is located automatically, or can be user-specified. This strategy also supports tool contact angles.

Morphed Spiral



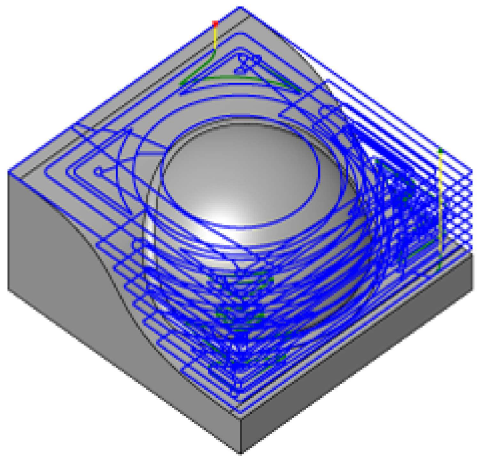
The **Morphed Spiral** strategy is very similar to the **Spiral** strategy. However, a **Morphed Spiral** operation generates the spiral from the selected boundary as opposed to a **Spiral** operation which trims the generated passes to the machining boundary. This means that **Morphed Spiral** can be used for additional surfaces for which **Spiral** is not appropriate. It can also be very useful when machining free-form/organic surfaces. Although the **Scallop** strategy is often used for these types of surfaces, both the sharp corners and the linking transitions between the generated passes can result in visible marks. The **Morphed Spiral** strategy generally provides a much smoother toolpath by avoiding these issues.

Radial



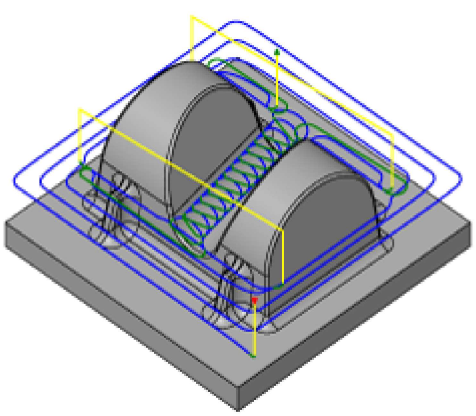
Like spiral machining, **Radial** machining also starts from a center point, providing you with the ability to machine radial parts. It also provides the option to stop short of the center of the radial passes, where they become very dense. The center point of the detail to be machined is located automatically, or can be user-specified. This routine can also be used with tool contact angles.

Pocket



Pocket is the conventional roughing strategy for clearing large quantities of material effectively. The part is cleared layer by layer with smooth offset contours maintaining climb milling throughout the operation. To avoid plunging, the tool ramps down along a helical path between levels. To maintain a high feedrate, and thereby reducing machining time, sharp changes of direction are avoided by smoothing the tool motion.

Adaptive Clearing



Adaptive Clearing is an innovative roughing strategy that offers significant improvements compared to conventional roughing strategies. The strategy avoids full-width cuts by progressively shaving material off the remaining stock. The generated toolpath ensures that the cutting conditions remain constant with a stable load on the tool. As a result, the feedrate can be increased significantly, reducing the machining time by 40% or more.