Team 2485 CAM Standards

Basic Standards:

Order of operations:

1. Job
2. Drill plunge holes
3. Drill holes in the part and any extra hold down holes.
4. 2D interior contours and bearing holes
5. 2D Adaptive/2D Pocket (if used). This is now rare.
6. 2D Exterior contour

- Try to prevent unnecessary tool lines through organization operations
    a. Tool lines sometimes reset the z-zero on the mill, interrupting workflow, so want to be minimized

All files should follow this standard:

- Always put parts into an assembly **ASSEMBLY,** no matter the number
    - 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 something
- CAM-[Initials]_[Simple Part #]-x[Number of parts]_[Additional Parts]
    - 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"

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):

- Normally use a ¼ in end mill but select the tool from the tool table with the .249in definition. This will slightly oversize the holes. For very small bearings that need a ⅛ in end mill use the ⅛ in definition.

Using this document:

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

Feeds and Speeds.

- The feeds and speeds set for a CNC machine make reference to the feed 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 HSM Works 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 tools or at least recommended averages across many companies for High Speed Steel tools. 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 most HSS ¼ in diameter tools we use a chip load of .0012 in, and for a 1/8 in HSS cutter the chip load is .0006in. These numbers change significantly for carbide or cobalt cutters 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 that material that is being cut. For example, the SFM recommended by most manufacturers for cutting 6061-T6 aluminum is a range from 300-600 (some say as high as 800) feet per minute. We will normally use about 400-450 FPM. This is normally based on a depth of cut of NMT 1 X D of the tool for a contour style cut. Some manufacturers simply specify the RPM for a given material.

- 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 tool table, you should always check to make sure that tools you are calling are correct. Of specific importance is the type of tool (e.g. square end mill or drill), the diameter of the tool (specifically the cutting end), and the number of flutes. The SFM X the tool diameter gives the spindle speed in RPM. At that spindle speed, the chip load X the number of flutes gives the feed rate in inches per min (ipm). The complication comes where the machine has a maximum spindle speed (5400 rpm for the CNC mill and 18000 RPM 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.
    - End mill Summary

End Mills	Stock material	Tool Diameter	Chip Load (in)	Spindle speed (RPM)	Max depth of cut (in)
Router	6061 Al	4mm	.002	18000	.125
Mill	6061 Al	.25 (¼)	.0012	5400	.125
Mill	6061 Al	.125 (⅛)	.0004	5400	.125
Router	Delrin	4mm	.004	9090	.152

- 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.

- 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. 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 carbide, 4mm diameter, single flute end mills, with a chip load of .002 in (as long as they are available). We will use 18000 rpm spindle speed. If we have to use the 1/8in end mills, use 16000 rpm with a chip load of .002in. 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 carbide, 1/4in dia, 3 flute end mills (if available). We will use 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 for that.

Depth of cut

- For mill and router. For up to ⅛ in Al plate or square tube, use a single depth cut. For 3/16in Al plate use multiple depths (2) with even stepdowns and a max step down of .152in. For 1.4 in Al plate use multiple depths with 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 most 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. You will use 2 finishing passes for the bearing holes but no finishing passes on internal or external contours. 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.

- When machining multiple parts from the same piece of stock, make a separate external contour task for each part. This prevents the system from getting confused if your parts are too close to each other.
- **Use Pre-Drilled Holes 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.

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. 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. Only use hold down holes for very large internal pieces - greater than 4in across.

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" dia!!**.

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. Try not to use drills over ⅜" dia. 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. 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, pockets with a finishing pass for bearing holes and a separate exterior contour for each part on the plate. 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 .152in.

- Delrin. For 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. 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. Do not use coolant when machining Delrin - only air to blow the chips away.

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. Use pockets with a 30% stepover with a full depth cut and 1 finishing pass for bearing holes. 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.

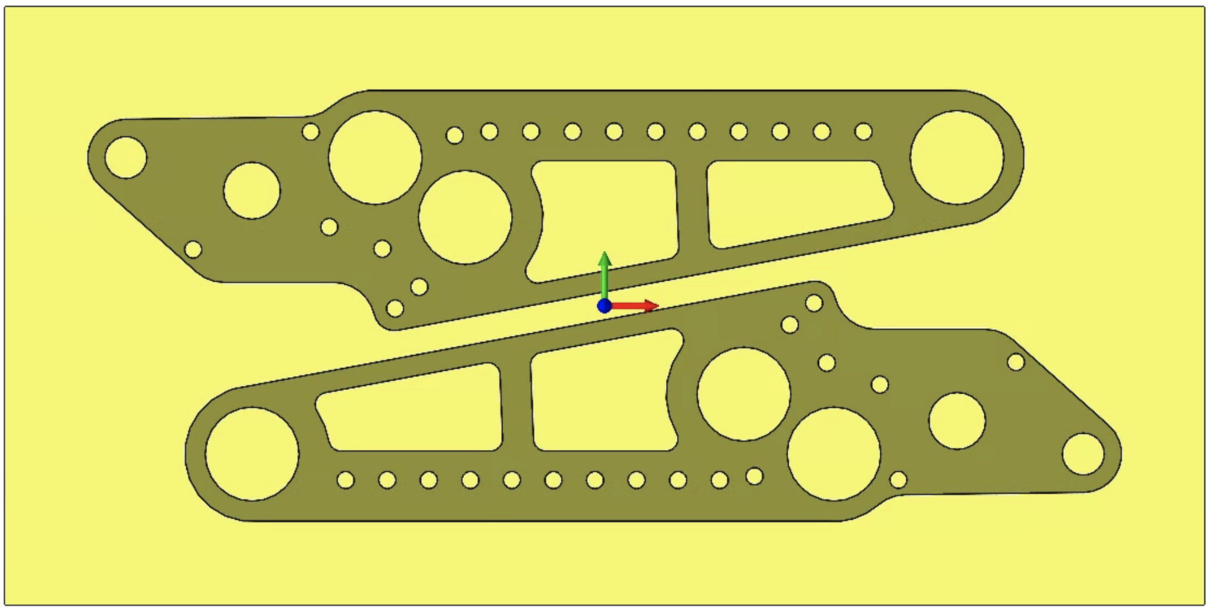
CNC Mill Al 6061-T6 plate work. For drilling operations, use a 5k rpm spindle speed and a stubby drill when drilling through the plate. Use a .05 cut through. Any deeper and the hole will not be good for a hold down screw. Use the first drill step to drill only the pre-drilled plunge holes then stop 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".

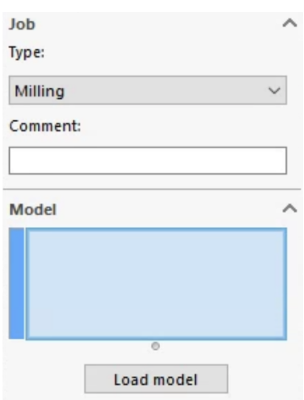
CNC Mill Square Tube: For drilling operations, use a 5k rpm spindle speed, use a stubby drill when drilling through no more than 1" of square tube. **DO NOT** use a jobber length drill for deeper cuts like through cutting a 2in square tube. There is no way to control where the jobber length drill starts 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.

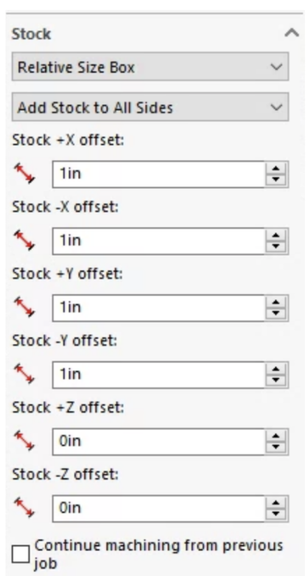
Specific Examples

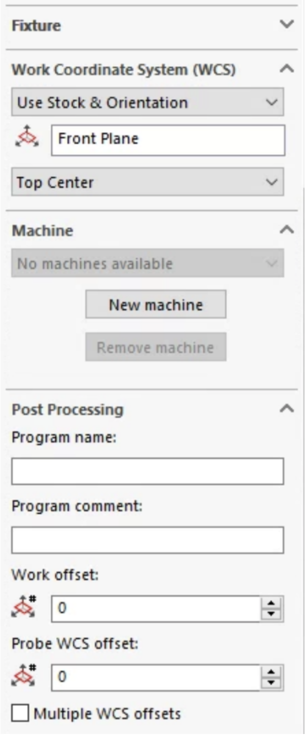
Job:

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

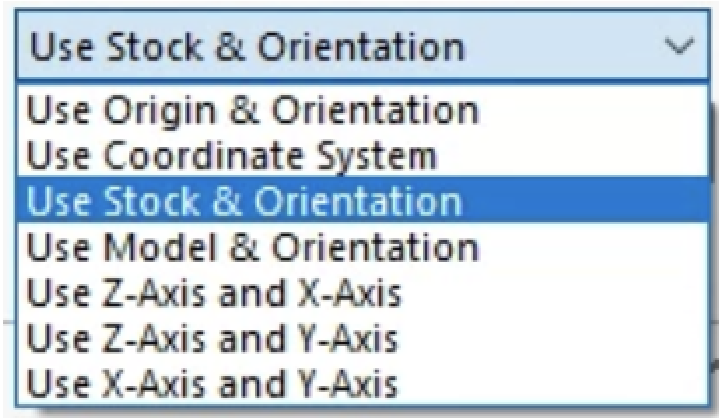






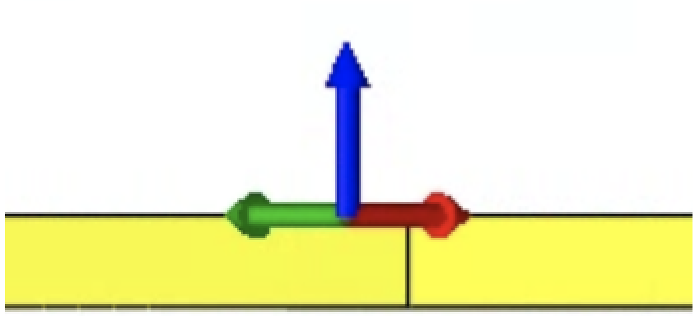


- Select parts under “model”
- Stock: Add Stock to All Sides
    - Offset by 1in to all X and Y sides
- Continue machining from previous job: OFF
- Work Coordinate System:
    - Tool orientation and direction



- 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

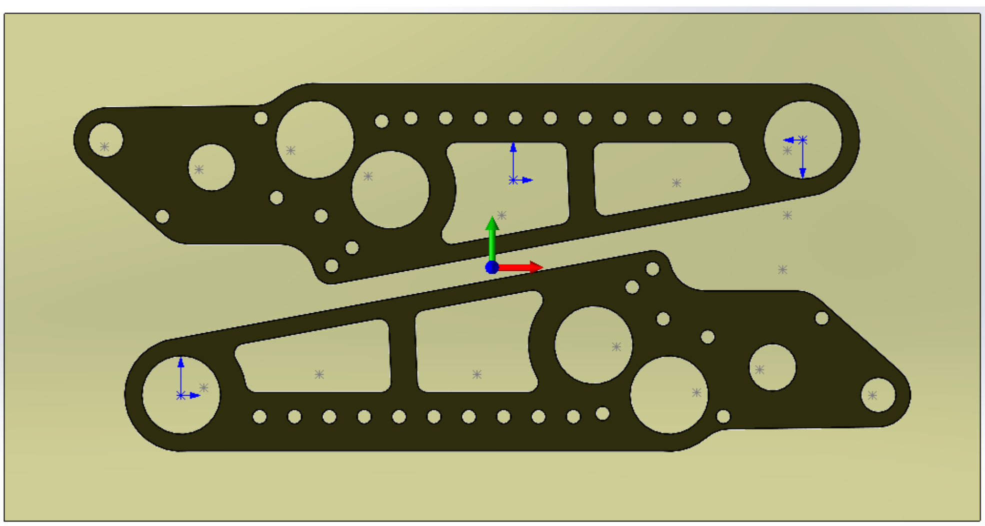


-
- Normally use “Top Center” → use to check above setting, make sure the origin is sitting on the correct side and facing the correct way

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. To begin the Drill steps, in CAM, select **Drilling** and then **Drill**
2. First drill step - Plunge 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)
- Leave as calculated defaults

In general - Tool definition:

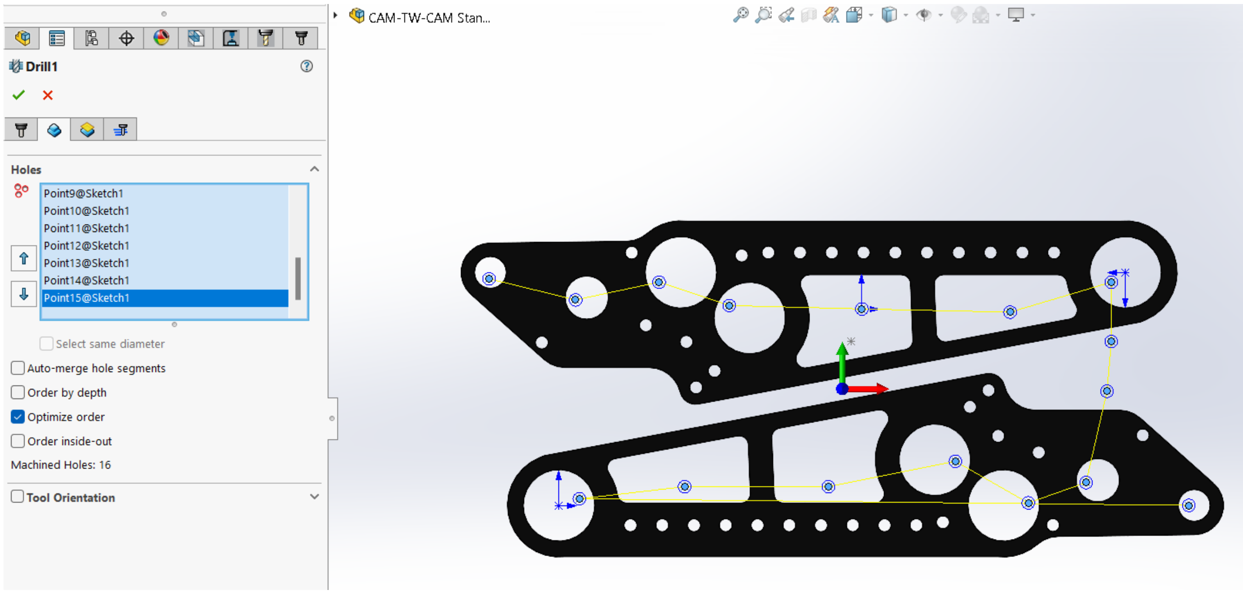
- Cutter > Type: Drill
- Cutter > Diameter
- Cutter > Tip Angle: 118° (occasionally 135°, but not necessary to check)
- Feed & Speed > # of Flutes

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	

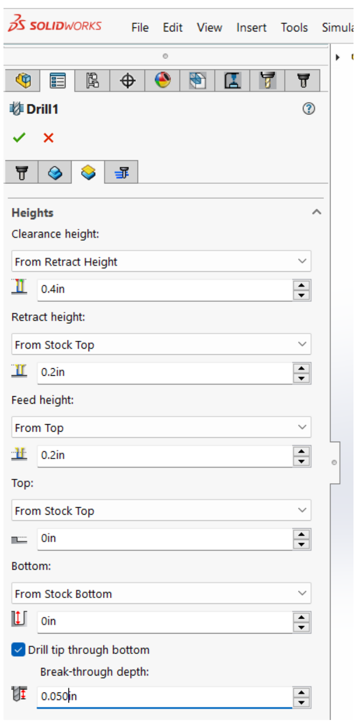
b. Geometry (Second Tab)

Select each of the Plunge holes individually

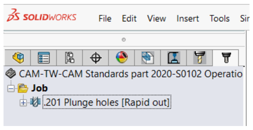


c. Heights (Third Tab)

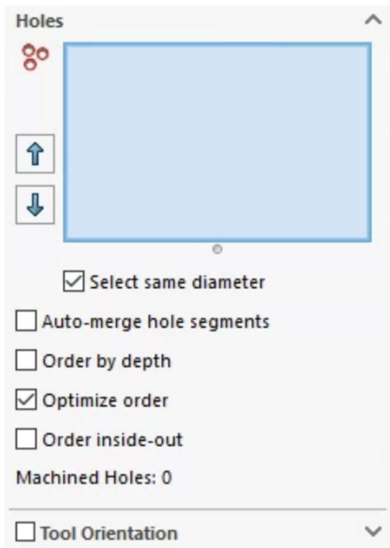
Because the plunge holes are just sketches, you must select - **stock top** and **stock bottom**. Select break through and then enter additional cut through - this is a positive value.



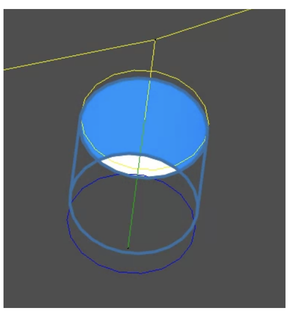
d. Rename the step: .201 Plunge Holes

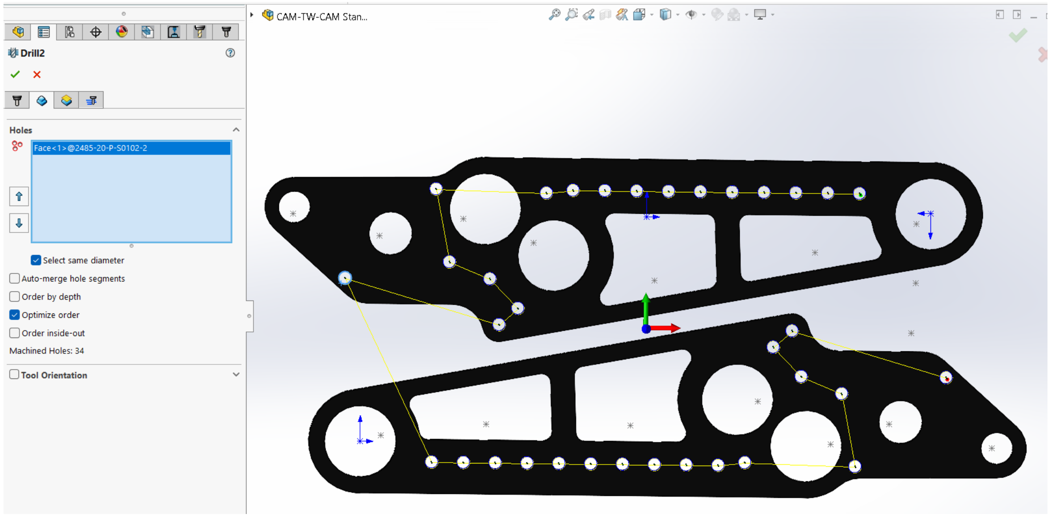


3. Second Drill Step - The holes designed in to the part
   a. Tool selection - in this example same tool



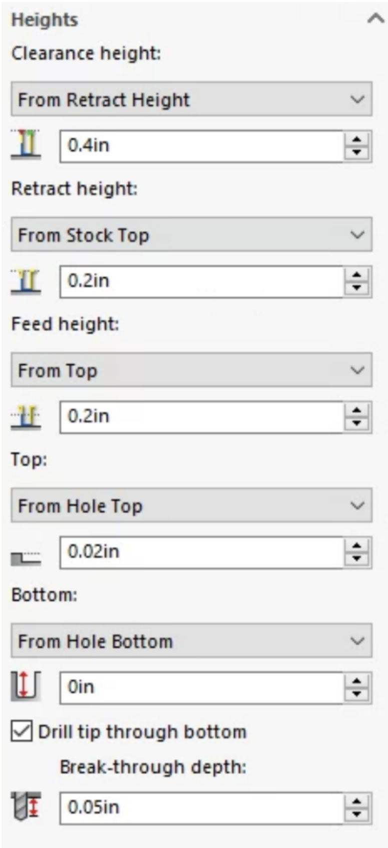
   b. Geometry (Second Tab):

- 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



CAM-TW-CAM Stan...
Drill2
Holes
Face<1>@2485-20-P-S0102-2
Select same diameter
Auto-merge hole segments
Order by depth
Optimize order
Order inside-out
Machined Holes: 34
Tool Orientation

c. Heights (Third Tab):



- Top: From Hole Top - 0.02in
- Bottom: From Hole Bottom - 0in
- Drill tip through bottom: ON - 0.05in
    - 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

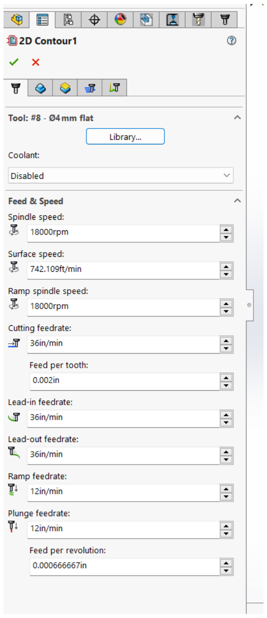
d. Passes (Fourth Tab):

- Cycle: Drilling - rapid out

Bearing Holes (and other circular holes)

Use *2D Contour*

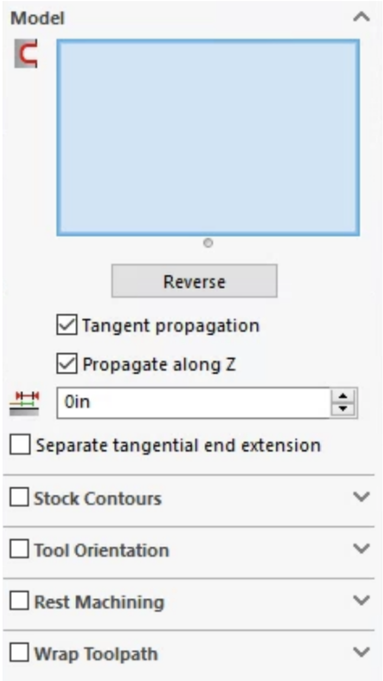
1. Tool definition: Select end mill (4mm for the router), ¼ in, 3 flute for the mill



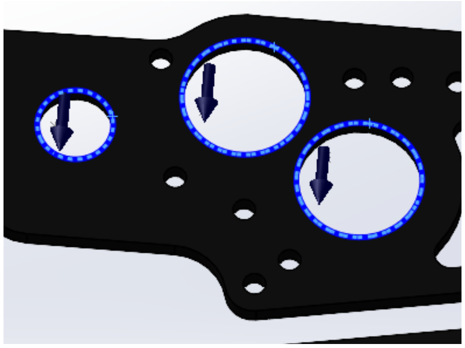
Tool:

- 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:

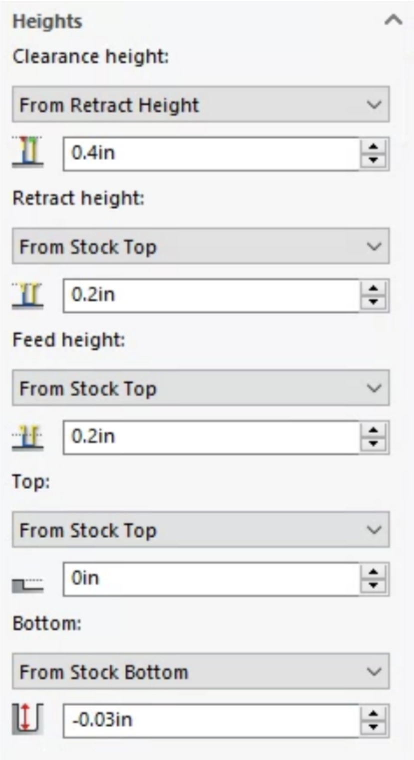


- Select upper contour



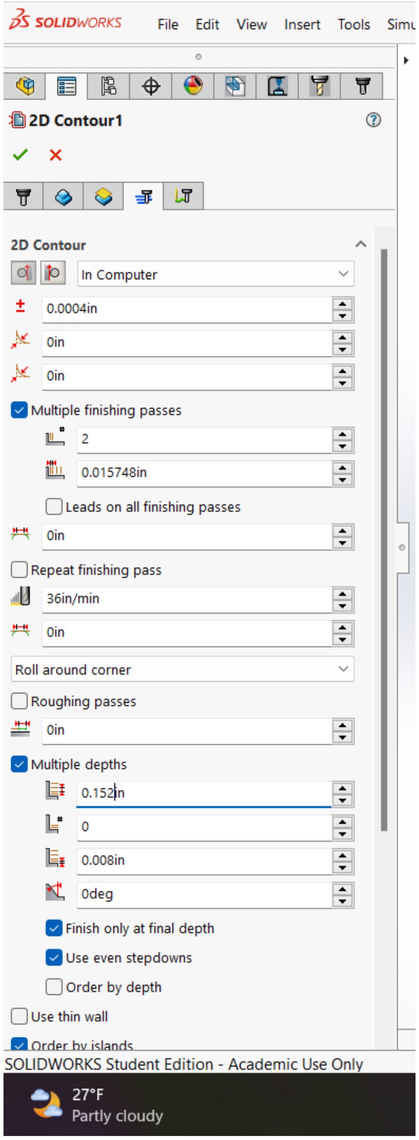
  - Make sure that the arrow is on the correct side (in space, not inside the material) → if not, select troublesome edge in selection menu, then press "Reverse"
- Tangent propagation: - No affect
- Propagate along Z: - No affect
- Stock Contours: OFF
- Tool Orientation: OFF
- Rest Machining: OFF
- Wrap Toolpath: OFF

3. 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.

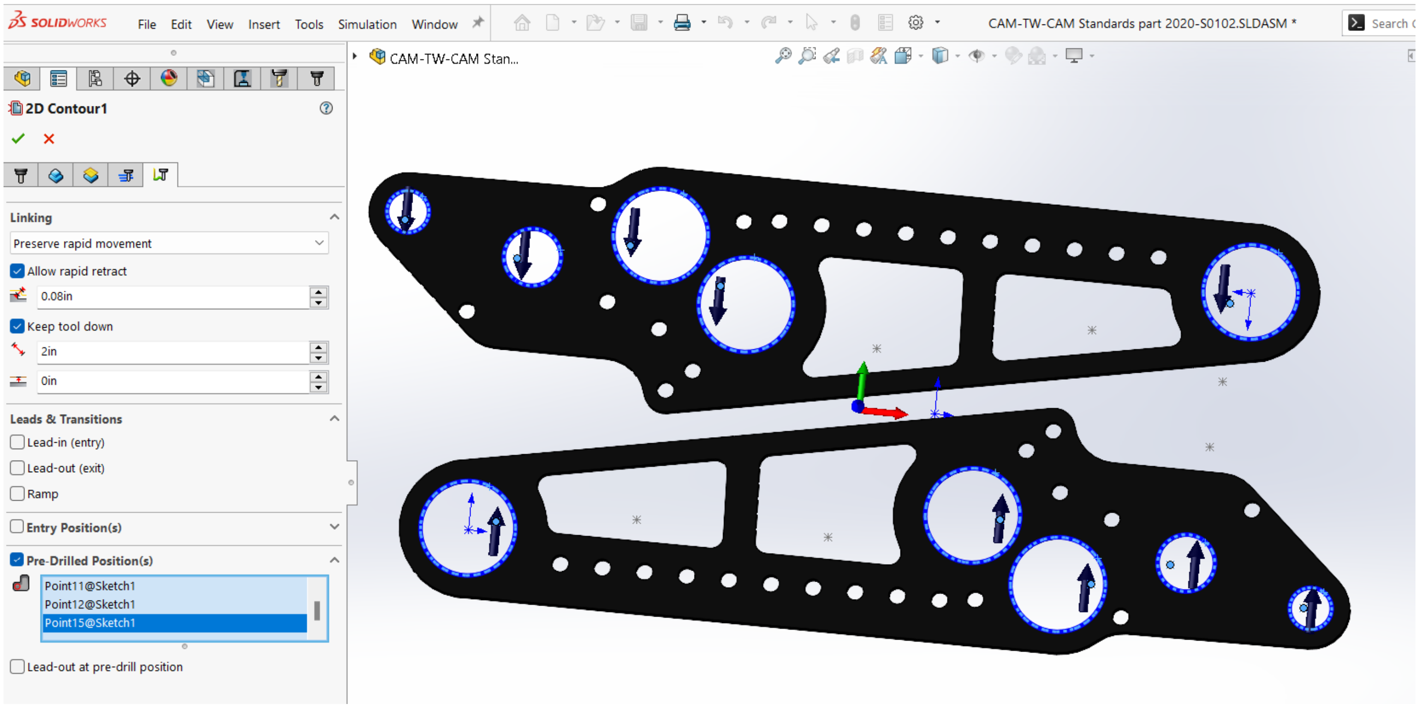
4. 2D Contour: Depths.: For <= 1/8in plate, click off multiple depths. For thicker material, select multiple depths and and a Maximum Roughing Step down of .125in. The program will determine the depth of each step. Set Use Even Stepdowns. Finishing passes. Because this is for bearing holes, set finishing passes to 2. Leave the amount as the default. Select Finish at Final Depth.



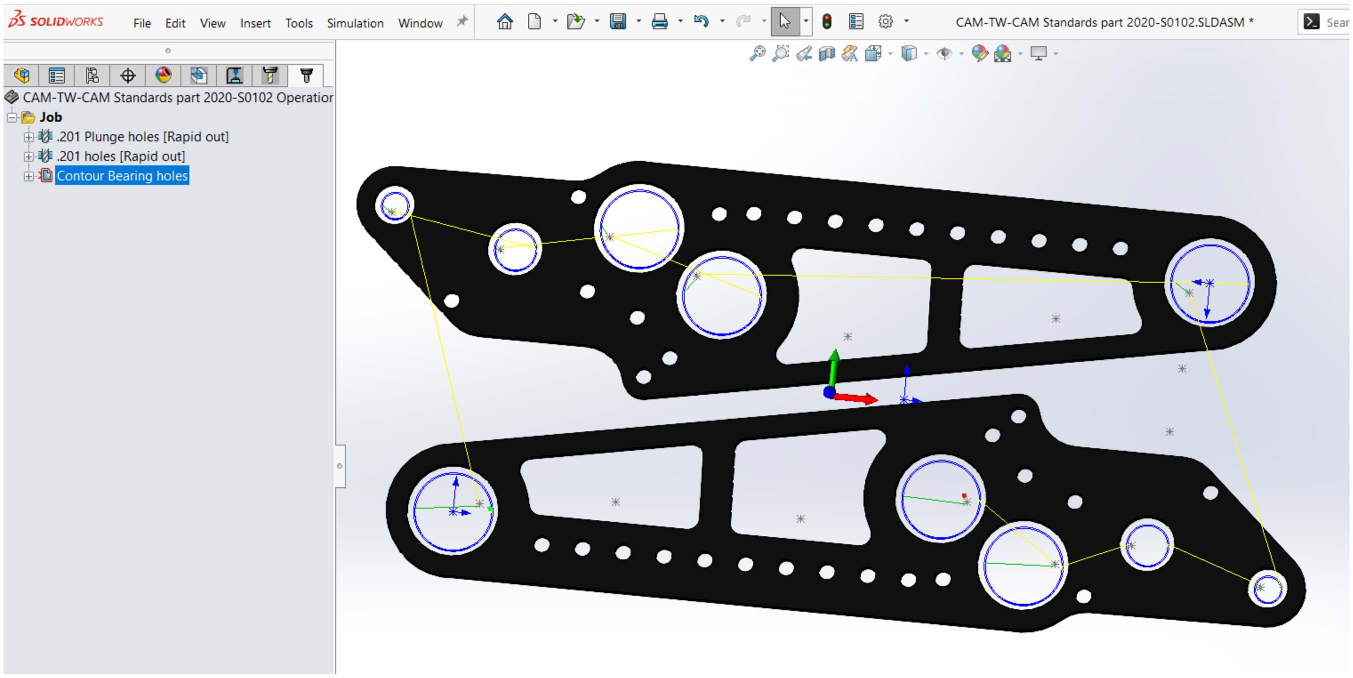
- Multiple finishing passes: 2
- Multiple depths: ON
- Maximum stepdown: 0.125in
- Use even stepdowns: ON
    - Maximum stepdown of 0.1 + even stepdowns ensures a good amount of passes → 2 passes for 3/16 in, 3 passes for ¼ in
- Stock to Leave: OFF
- Smoothing: Doesn't matter
- Feed Optimization: OFF

5. 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.



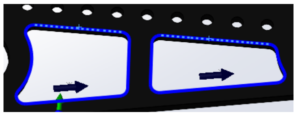
- Lead in: OFF
- Lead out: OFF
- Ramp: OFF
- Ramping angle: 2deg
- Entry positions: OFF
- Pre-Drilled Position(s): ON

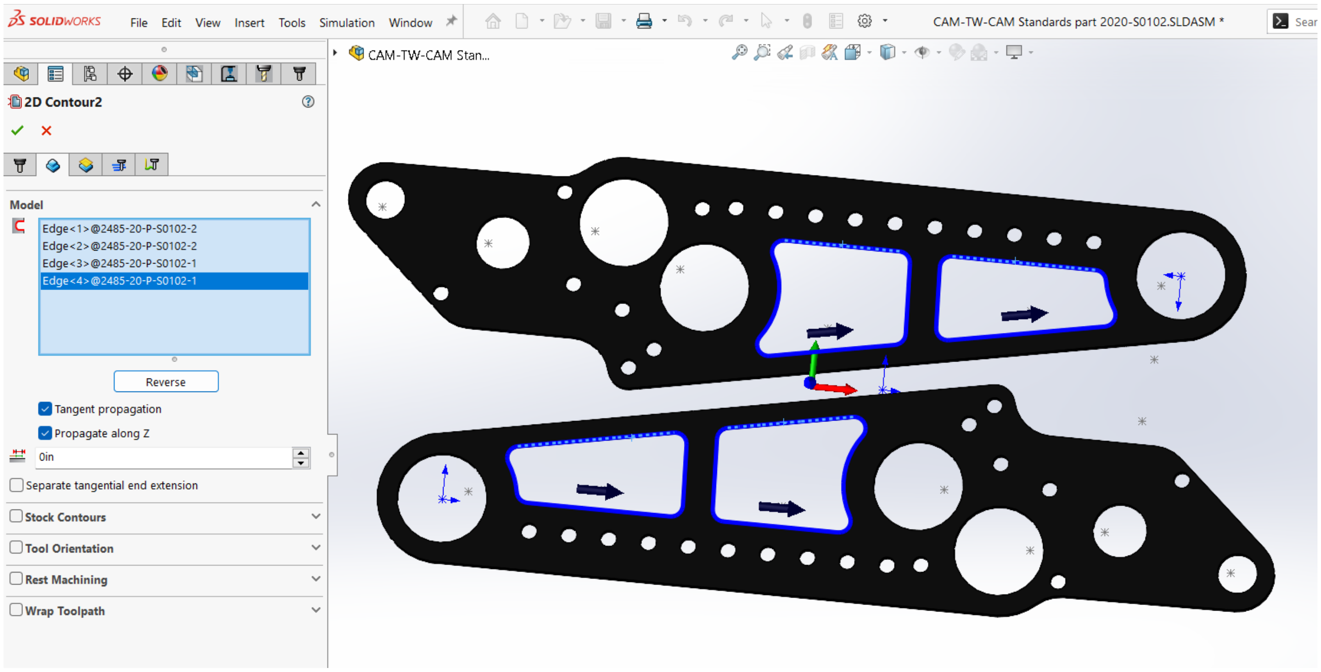


Interior Contours

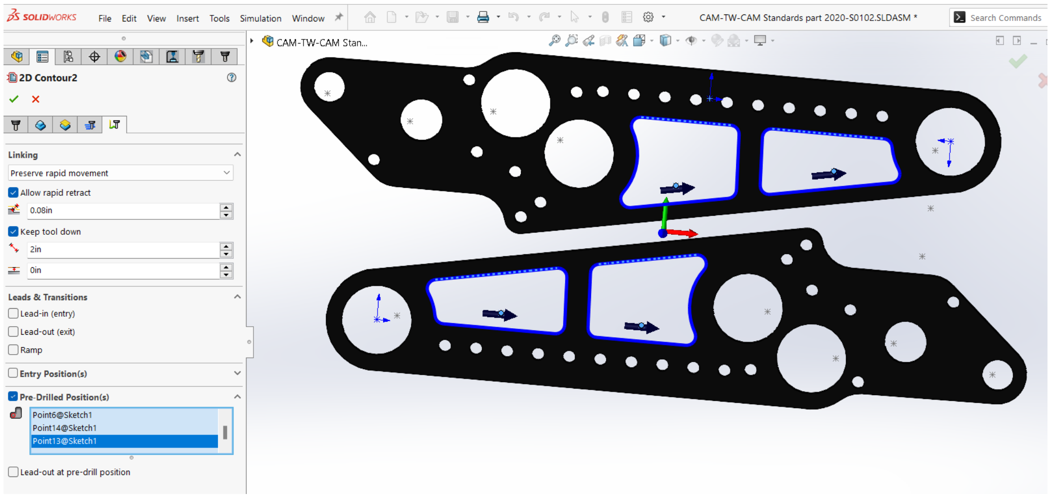
Use 2D Contour - Create a new contour line for all interior contours.

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

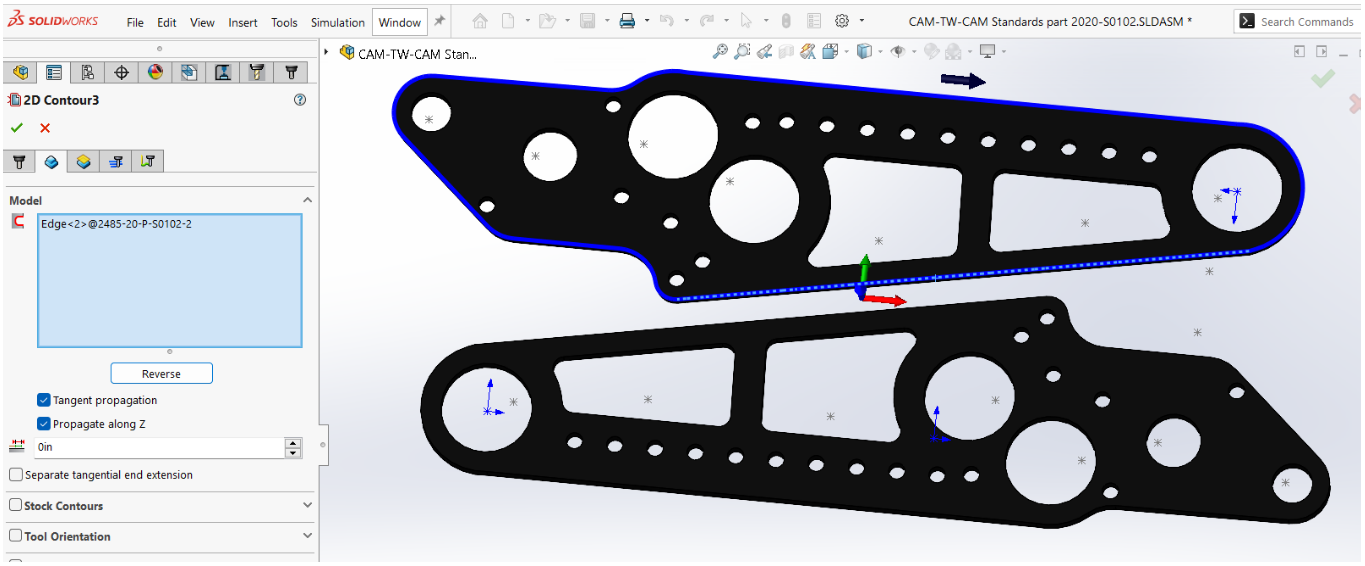
Used for the outside contour

IMPORTANT NOTES:

- **Only select one part at a time**
  - Each part has its own contour operation
  - The CAM doesn't like generating tool paths so close together, so it does weird things to accommodate that slows down the program
- **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.

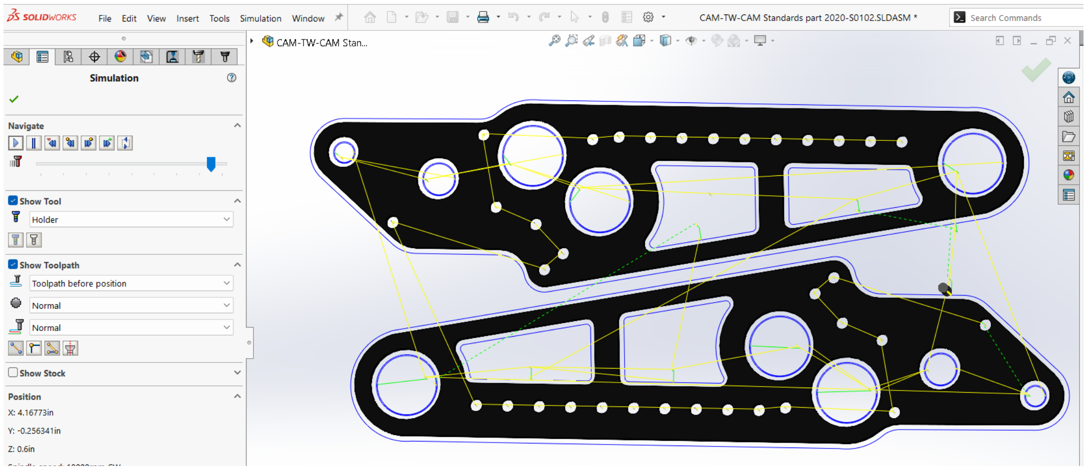
1. Tools. Same as bearing holes
2. Geometry. Like the bearing holes, choose the upper contour line of one of the outside contours and ensure the arrow is on the outside of the exterior contour.



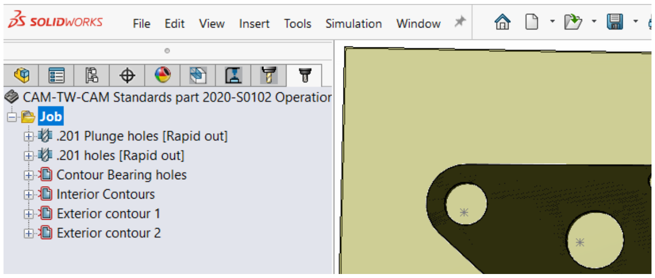
- Tangent propagation: - No Affect
- Propagate along Z: - No Affect
- Stock Contours: OFF
- Tool Orientation: OFF
- Rest Machining: OFF
- Wrap Toolpath: OFF

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.

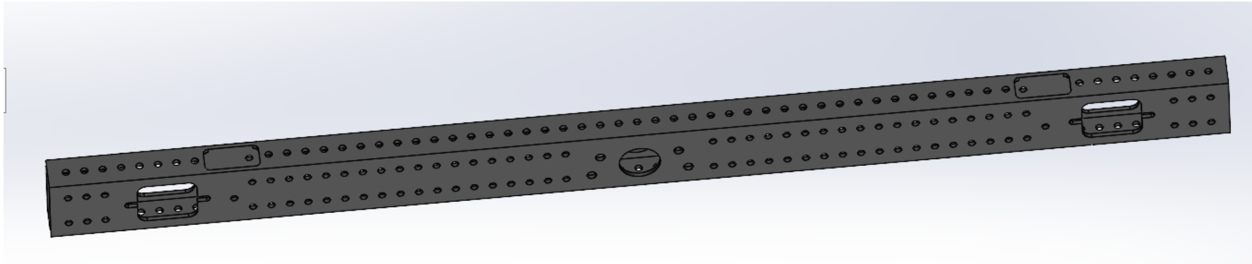
Run the Simulation and check the tool paths and ensure there is proper cut through for each operation.



Ensure all operations have been renamed to be descriptive.



Drive Train Rails (and all square tube)

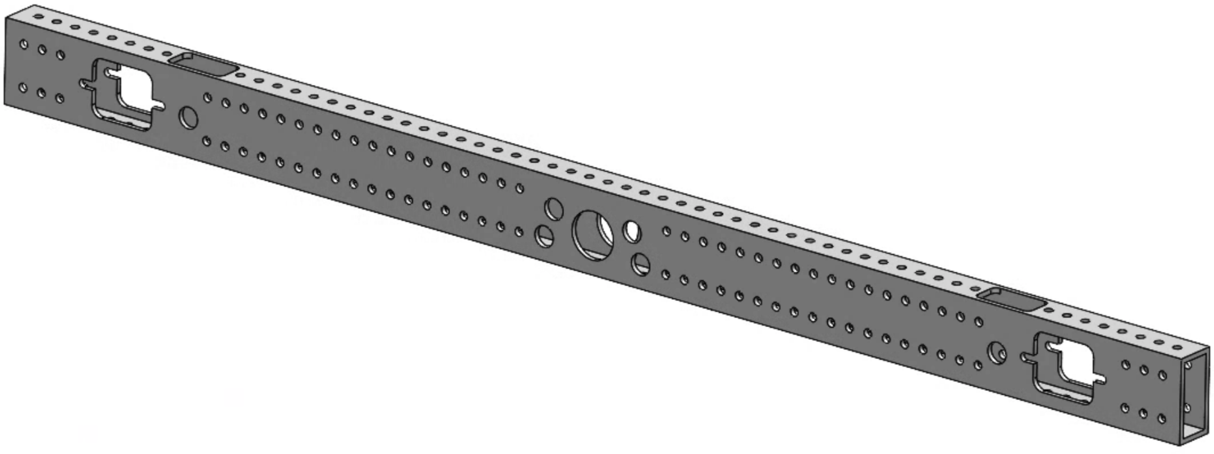


IMPORTANT NOTES

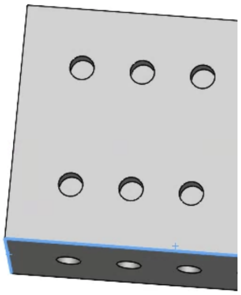
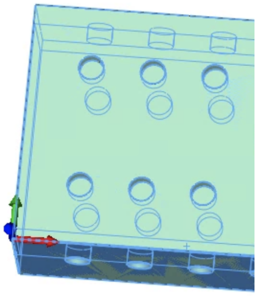
1. This will require as many as four separate sets of CAM (possibly). This can be done in a single CAM file with up to 4 separate jobs. The jobs would then be separately post processed. There would be one job for each side of the 2in side and 1 or 2 for the 1 inch side.
2. On the two inch sides, holes 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 -0.145 from stock top (can go deeper but it might cause more passes than necessary.

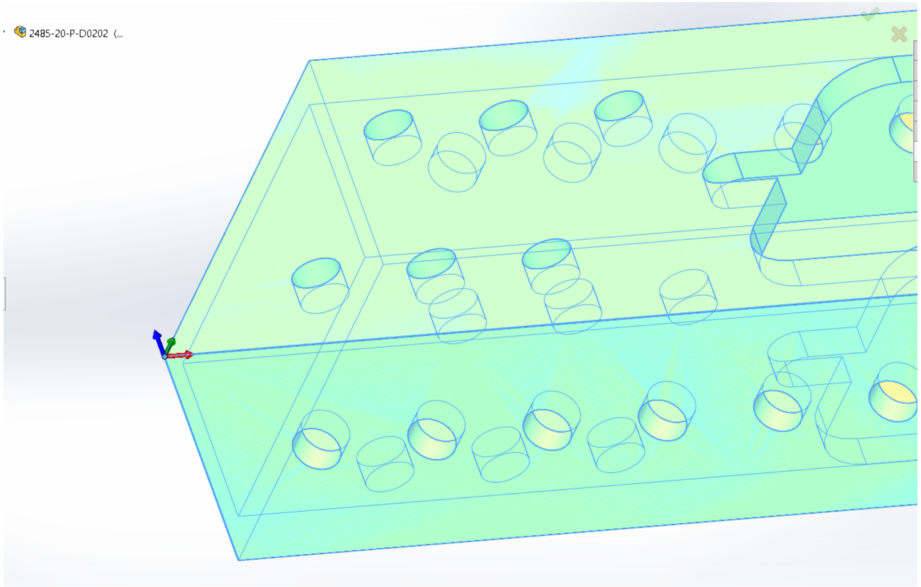
Job:

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



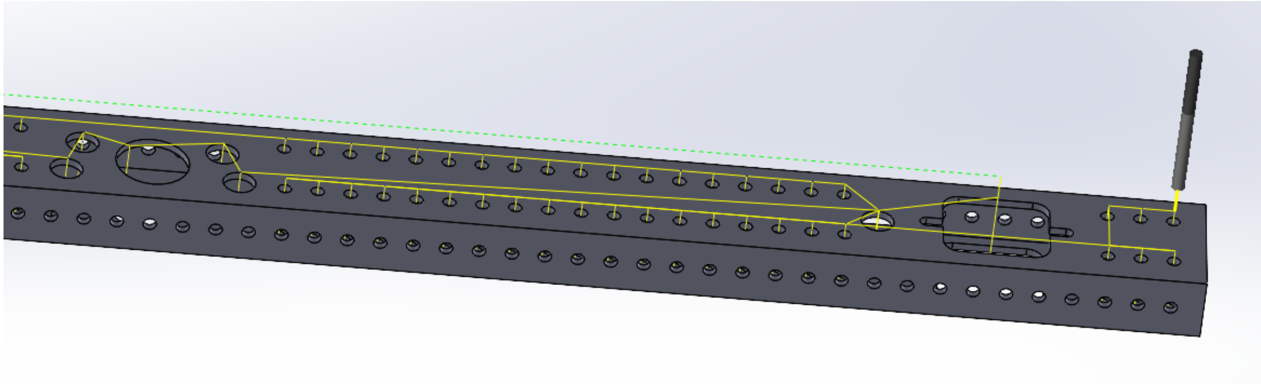
Do not add any stock.

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

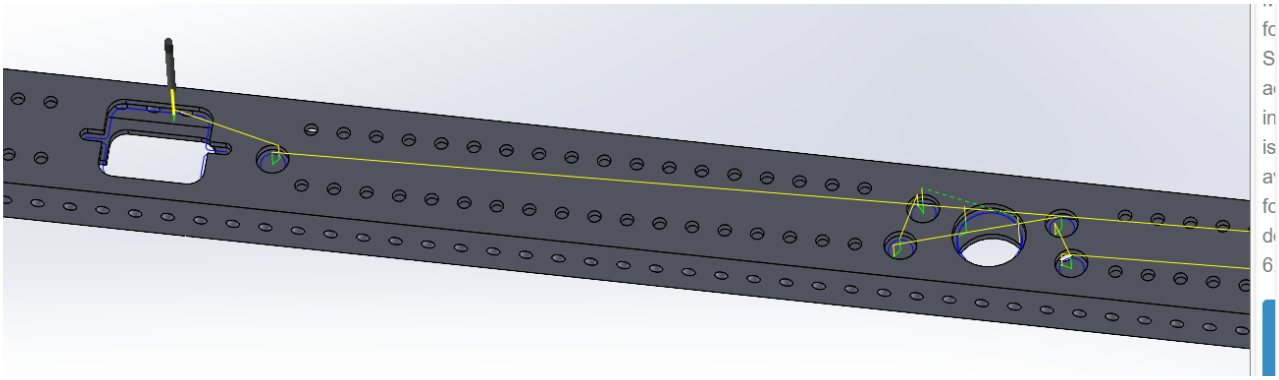
Use the “**Z Axis and X Axis**” and choose the correct edges and use **Origin - Selected Point** to achieve this orientation.

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.

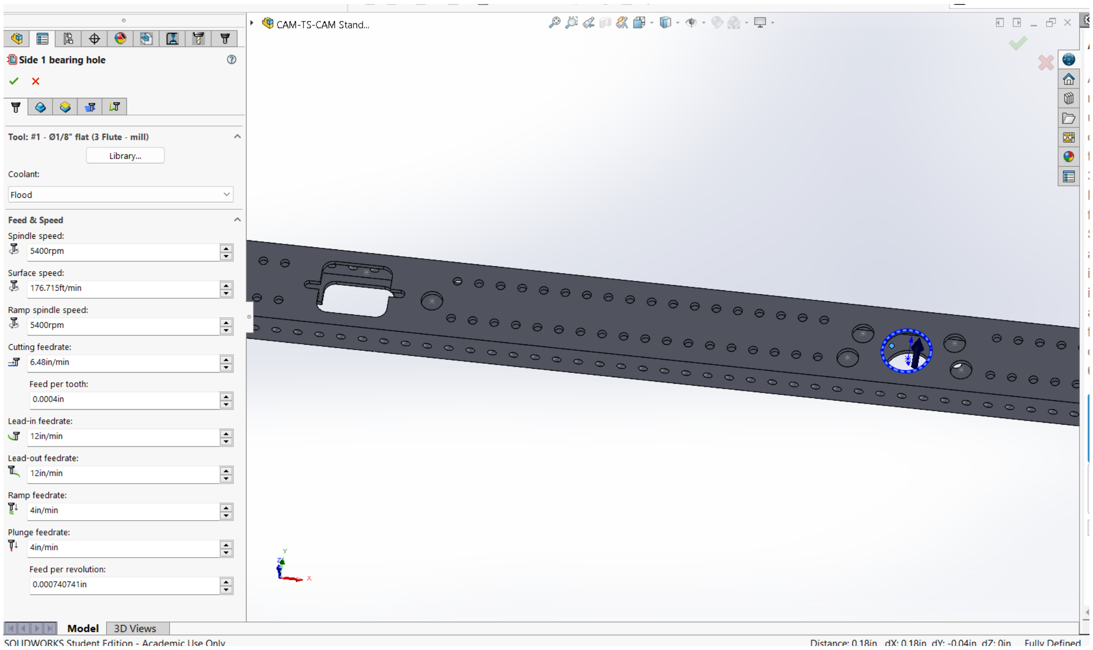


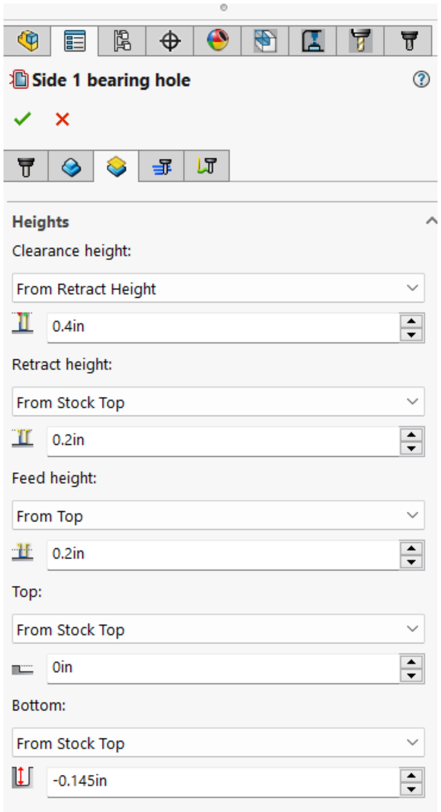
Use a ⅛" EM for the bearing block holes (this is the 3 flute, carbideEM, with a .0004in chip load at 5400 RPM) This is necessary to get the slots. Select the top contours of both bearing block holes and access holes. In the heights section set Bottom to from Stock Top to "-0.145". Then set multiple depths at .05 with even step downs. This will give 3 passes.

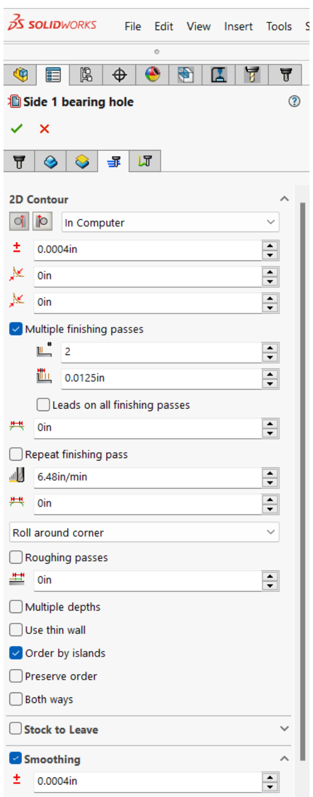


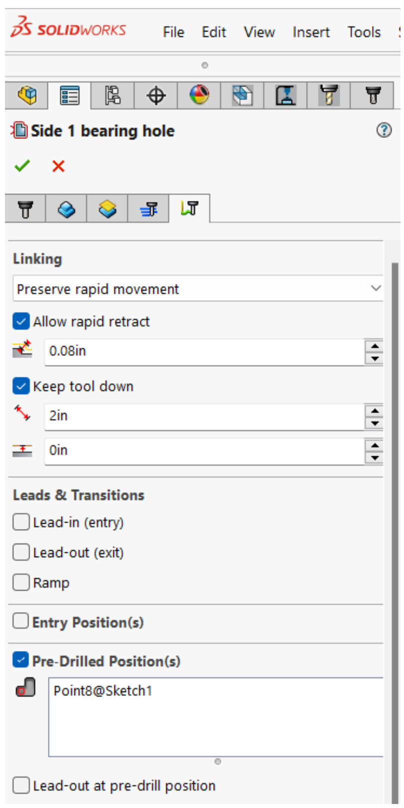
BEARING POCKETS

Use largely the same
tooling/settings as the bearing
block holes, except you need a finishing pass for the bearing pockets. Select 2 finishing passes. See settings below.



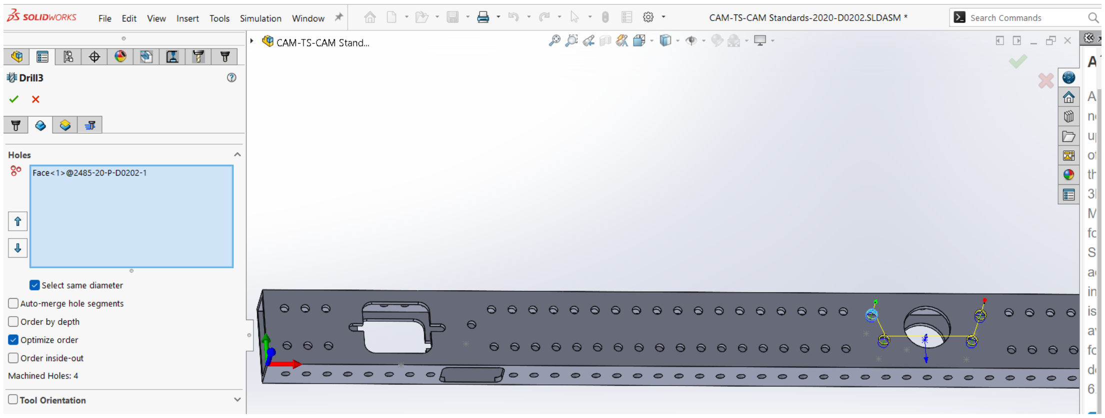

Side 1 bearing hole
Heights
Clearance height:
From Retract Height
0.4in
Retract height:
From Stock Top
0.2in
Feed height:
From Top
0.2in
Top:
From Stock Top
0in
Bottom:
From Stock Top
-0.145in


SOLIDWORKS
File Edit View Insert Tools
Side 1 bearing hole
2D Contour
In Computer
0.0004in
0in
0in
Multiple finishing passes
2
0.0125in
Leads on all finishing passes
0in
Repeat finishing pass
6.48in/min
0in
Roll around corner
Roughing passes
0in
Multiple depths
Use thin wall
Order by islands
Preserve order
Both ways
Stock to Leave
Smoothing
0.0004in

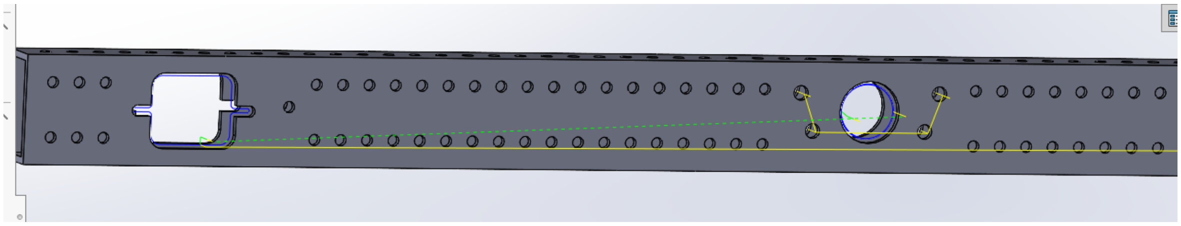

SOLIDWORKS
File Edit View Insert Tools
Side 1 bearing hole
Linking
Preserve rapid movement
Allow rapid retract
0.08in
Keep tool down
2in
0in
Leads & Transitions
Lead-in (entry)
Lead-out (exit)
Ramp
Entry Position(s)
Pre-Drilled Position(s)
Point8@Sketch1
Lead-out at pre-drill position

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.

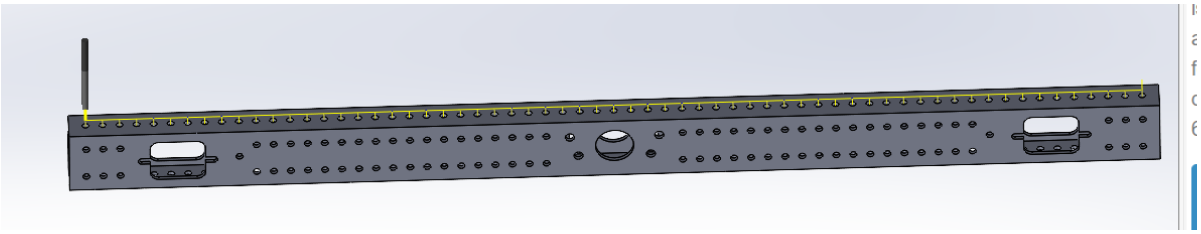


Similar to the first side, create a contour line to mill the bearing block holes using the same tooling/settings. You can use the same plunge hole sketches even though they are on the other side.

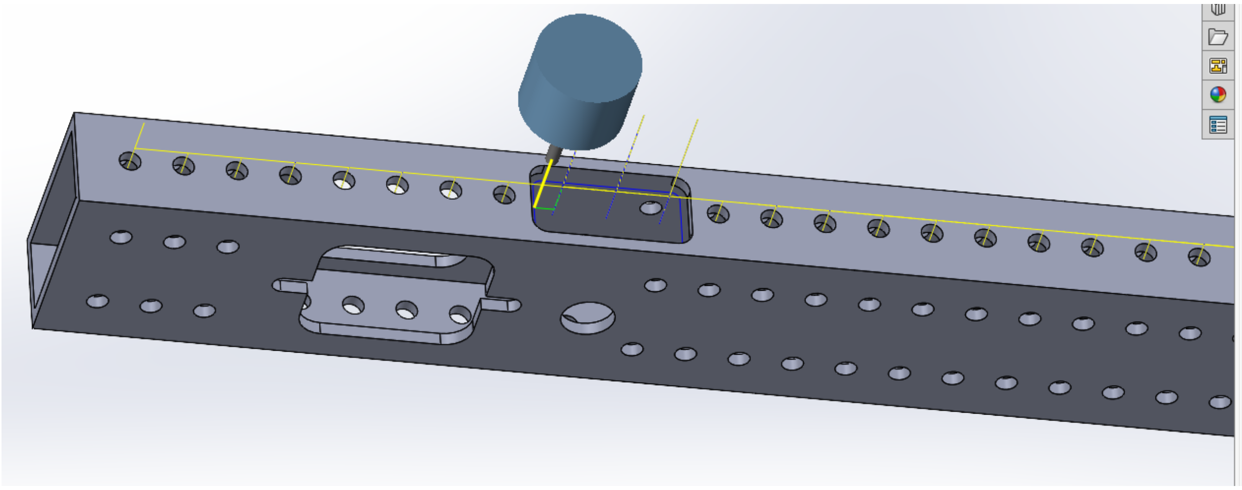


ONE INCH Side

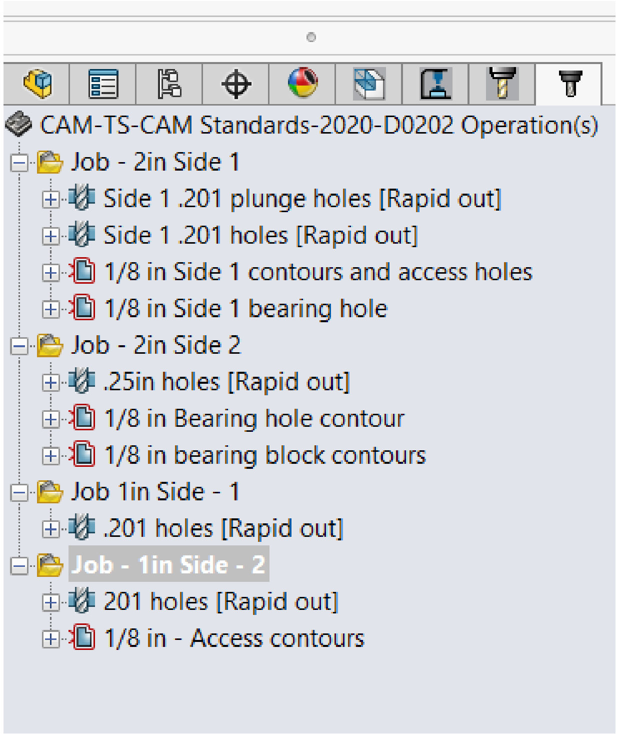
Start a new CAM file or new job for the one inch side. Both sides will probably need to be CAM’d. You will use a stubby size drill (.201) because both sides must be drilled. Start with the side with only holes. Set up the job as before and select all the holes. Set the bottom to stock top with a depth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



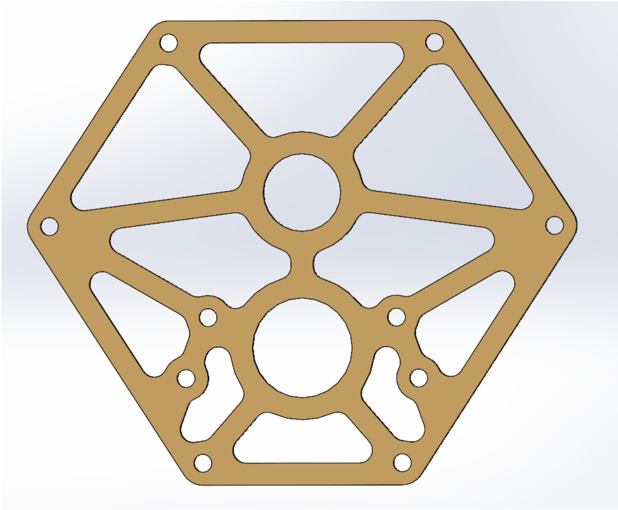
For the side with the slots, set up a new job the same way. Do a drill step as on the other side, select similar size holes. This will drill all the holes including the ones that will be cut out on the slot. This will provide us with a handy plunge hole for the slots. Sketch a point into the center of 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.



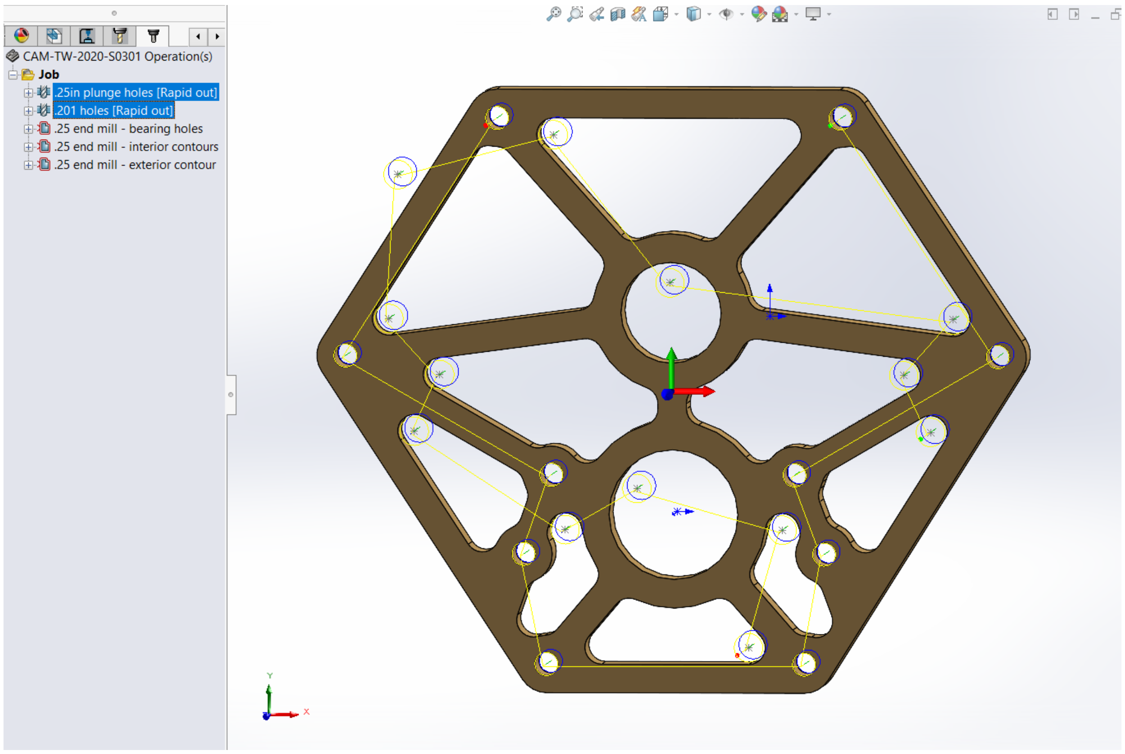
Gear Box Plate - CNC Mill



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..

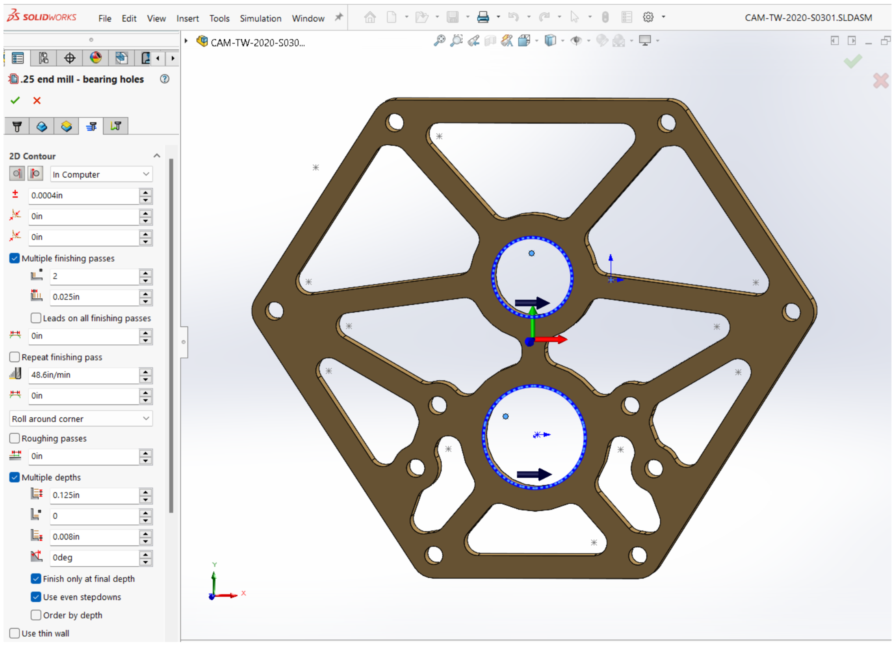
Drilling:

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.



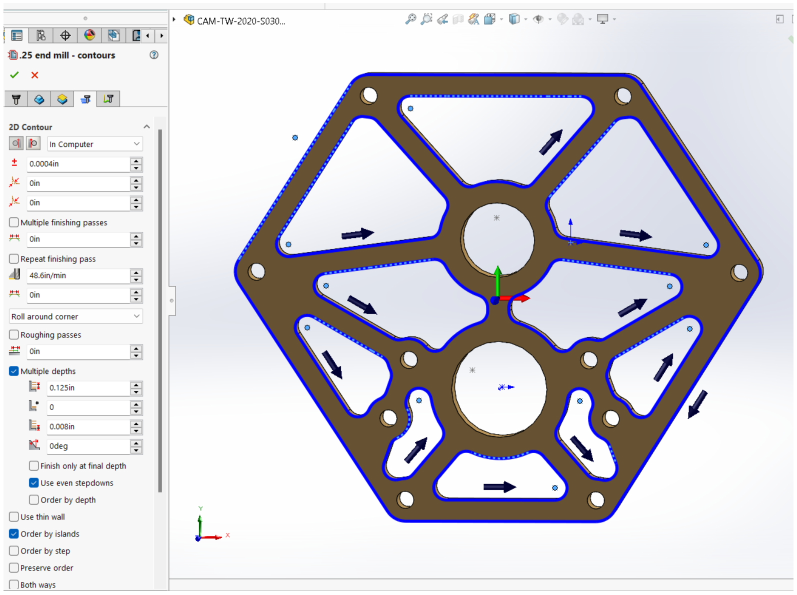
Milling:

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 5400RMP with a .0013in chip load (which is all set in the Library).



Set finishing passes to 2. 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.


.25 end mill - contours
2D Contour
In Computer
0.0004in
0in
0in
Multiple finishing passes
0in
Repeat finishing pass
48.6in/min
0in
Roll around corner
Roughing passes
0in
Multiple depths
0.125in
0
0.008in
0deg
Finish only at final depth
Use even stepdowns
Order by depth
Use thin wall
Order by islands
Order by step
Preserve order
Both ways
CAM-TW-2020-S030...