Version 14 : September 2020

MTM Tutorial

Proprietary Notice

PART CREATION TUTORIALS

This section is intended to help you learn how to set up and create part geometry. As the variety and arrangements of machines are so great, the tutorials in this manual are created using special training MDDs. These MDDs are incapable of posting but accurately represent a dual-spindle, dual-turret machine and a swiss-style machine with 2 spindles, 2 turrets and a fixed post. Please use these MDDs as you may find that the MDD for your machine has a very different setup and you may not be able to complete the tutorials with it.

Please note that the background color and several elements of many of the graphics have been modified for maximum image clarity.

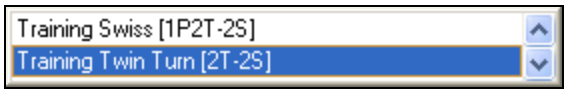
Part Setup

In this exercise we will introduce you to setting up a part. We will make a very simple part but will go into detail on the various elements.

1. Create a new part file named `Dual Spindle`.

2. Open the Document Control dialog. 

3. Select the Training Twin Turn [2T-2S] MDD.



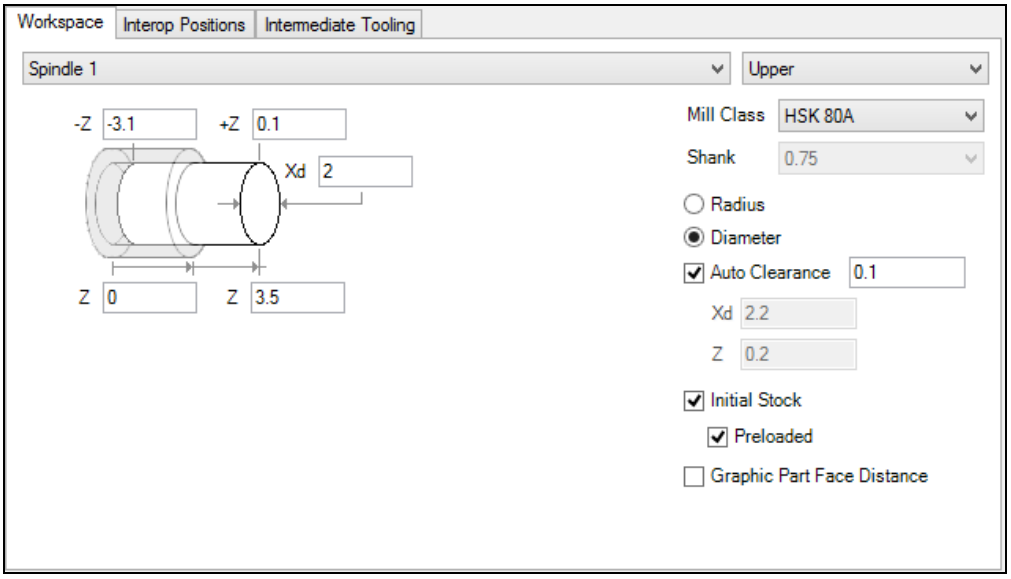
This is a basic two-turret, two-spindle machine.

Do not worry if your MDD has more than two spindles and/or turrets. When programming a part, you do not need to take advantage of all of your machine's capabilities. In this case we simply will not be needing all those axes.

If, however, your machine has one spindle, this particular exercise has minimal benefit to you. Machines with a single spindle are very easy to set up in MTM. You should follow along in the exercise and simply skip over what you cannot program.

Spindle Setup

1. Enter these parameters for Spindle 1.



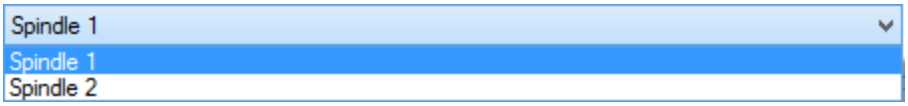
We will discuss these settings in detail. First of all, we are programming the part using a diameter measurement. We have stated that there is *Initial Stock*, meaning this is the spindle that will begin working from uncut stock. The *Preloaded* box is checked, meaning that the machine has stock loaded. There will not have to be a command to load stock. Essentially we are telling the program, "Don't worry about it, trust me, there will be stock to cut, I will take care of it myself."

The stock diagram has a lot of important information, including that we are defining the stock on Spindle 1, the stock is 2" in diameter and the part is 3.2" long. How do we know this? Easy. What is the total distance in Z? The *-Z* value is -3.1" and the *+Z* value is 0.1." Thus the total length is 3.2." We have also specified that the face of the part sticks out from the spindle by 3.5." This means we have 0.3" of bar stock sticking out of the spindle. When we feed the bar for the next run of the program, we feed out approximately 3.2", depending on the width of the cut off tool.

While the setup shown above may at first appear to have the part floating in space away from the spindle face, we should remember that these values set up the stock display size, not just the actual stock size. These values have been set to contain the part and any material to come off the part face. The extra 0.3" could be added to the *-Z* value but is not necessary and in many cases is undesirable. It is undesirable because of the size of the stock when unzooming the part.

The *Auto Clearance* and *Tool Change* settings are not our primary concern right now as we are only concerned with setting up the part and not defining machining parameters. You may either leave the values at their default or enter the values shown above.

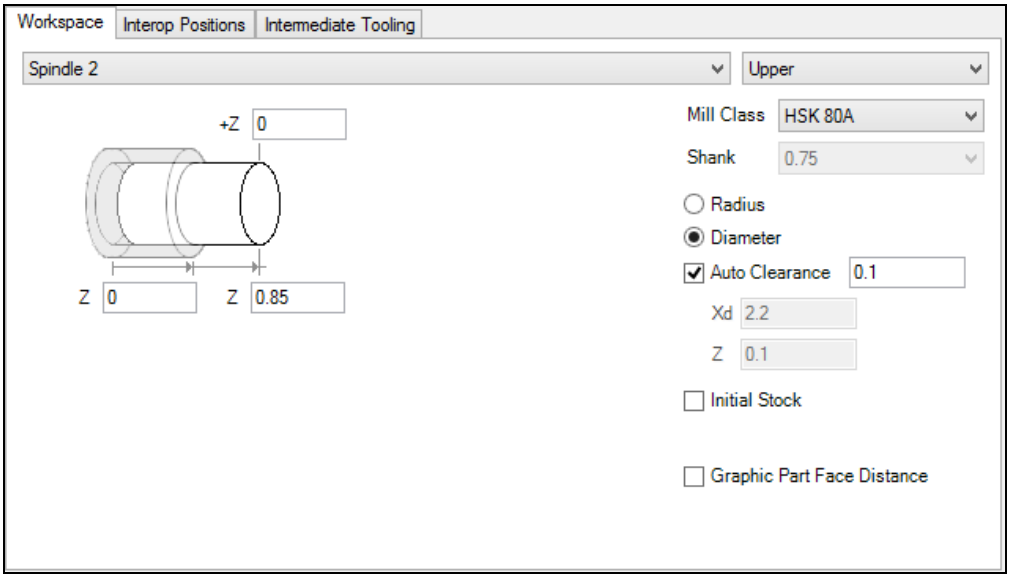
2. `Click` the Spindle selection button to change the Document dialog to setting up Spindle 2.



The DCD should display Spindle 2's initial setup as being identical to Spindle 1's. This is convenient if you are cutting multiple parts at the same time on separate spindles. This will need

to be changed to accurately reflect the part's condition after being transferred from the primary spindle.

3. Enter the parameters for Spindle 2 as shown.



Again, we should discuss these settings in detail. The Radius and Diameter radio buttons are still available. You can still change this setting, but all spindles will be programmed using the same measurement system.

For Spindle 2 we have stated that Initial Stock is off, meaning that there is no stock on this spindle and at some point a transfer will be made. Because there is no stock, the Preloaded box is not needed.

When *Initial Stock* is turned off the stock diagram changes dramatically. The -Z and Xd text boxes are removed because they are no longer needed. These settings are no longer needed because we are only concerned with the amount that sticks out of the sub spindle from the pick off. The values remaining are asking you to define how far the part sticks out from the spindle and how much stock is on the positive side of the origin. This part of the part setup needs to be carefully considered.

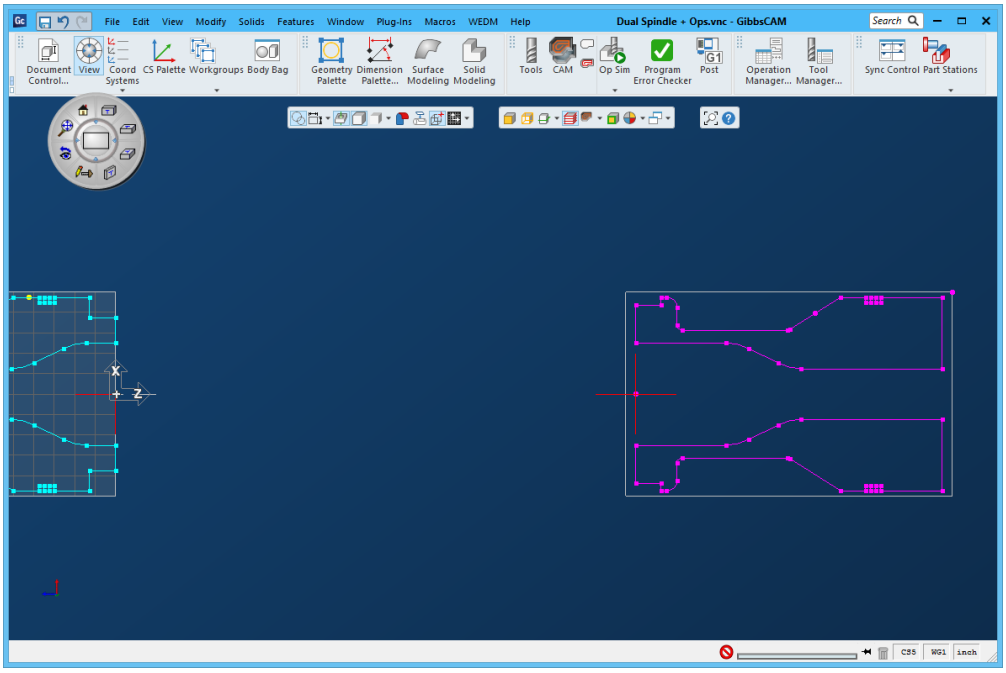
In this case, we have specified that 0.85" sticks out from the spindle. Our finished length part geometry is exactly 3.0" long. Assuming that 0.1" gets faced off on Spindle 1 and 0.1" is removed during the cut-off, we have a part that is 3.0" long. By specifying that the part sticks out by 0.85" we are implying that the chuck is holding the rest of the part and the part is picked off at Z-2.15". This information is important when setting up a part transfer.

Sometimes when setting up a part, you may wish to change this setting during or after your machining has been created.

The remaining data, Tool Change and Auto Clearance may be skipped for the time being. As for the Graphic Part Face Distance setting, we will get into that next.

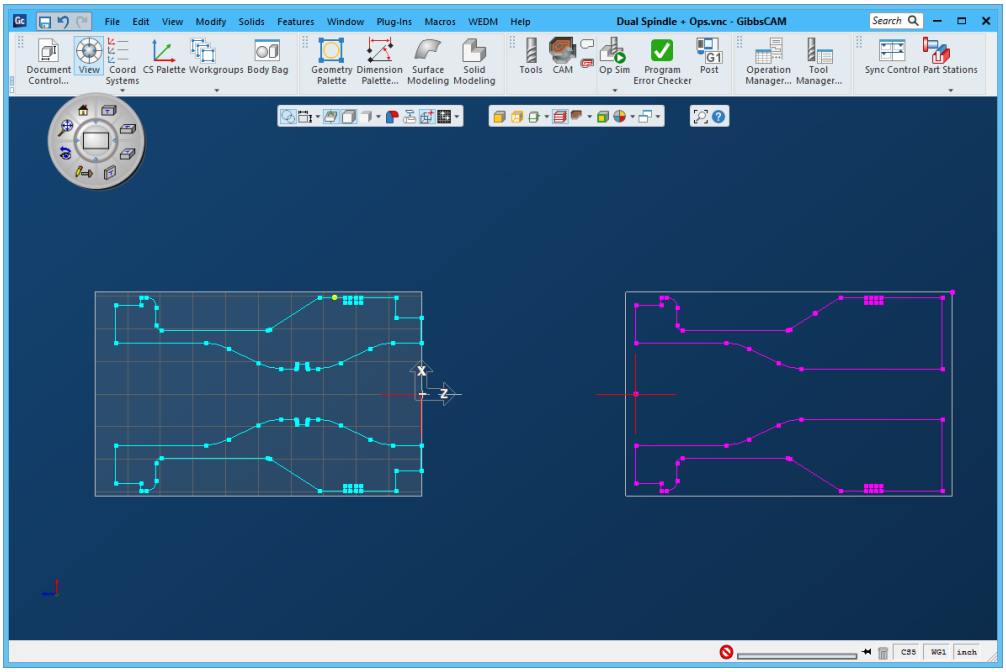
4. Close the Document Dialog so you can see all of the Workspace.

Both spindles are visible. The default distance between the origins of the spindles is the scaled distance in your MDD. This distance can be visually changed with the *Graphic Part Face Distance* setting.

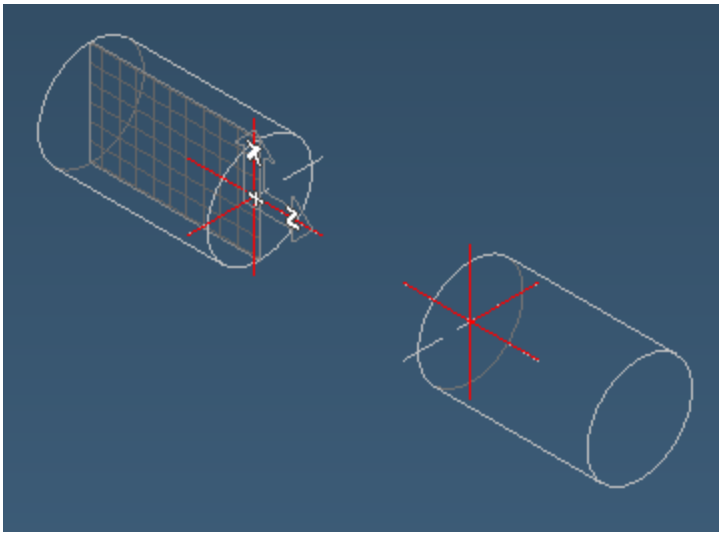


5. In the Document dialog, `click` the Graphic Part Face Distance checkbox and enter the value shown.

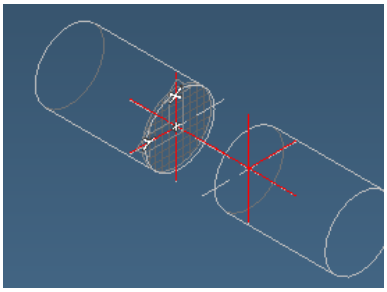
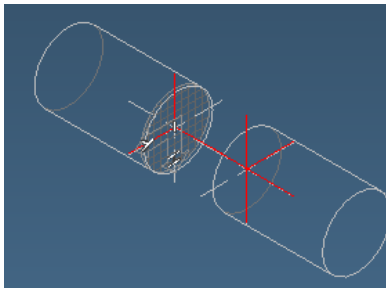
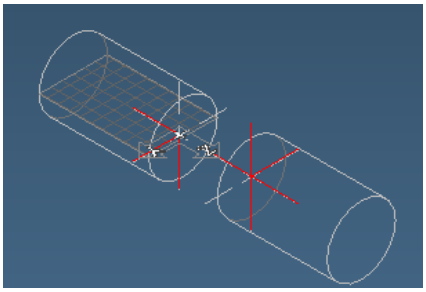
Now that the spindles appear larger on screen you will have an easier time working with them. This function acts only as a change in display. The actual distance between spindles is a fixed value in your MDD and is defined by your machine.



When a part is created, the system generates one or four coordinate systems for each spindle. If you have Lathe and Multi-Task Machining the system creates one coordinate system per spindle, the ZX plane. Note that the depths of the CSs project out towards each other. The negative depth of the ZX axis is always towards the spindle face.



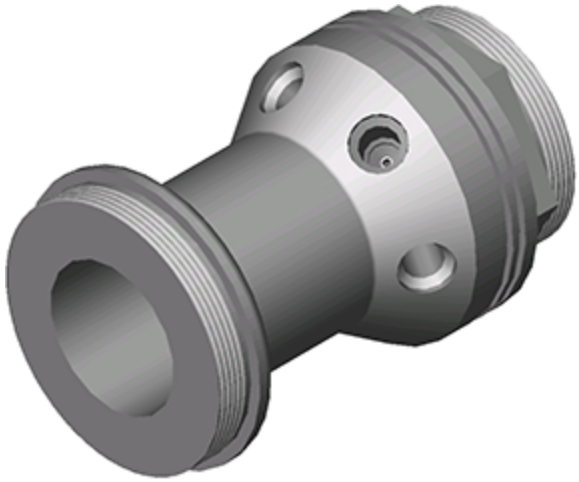
If you have Mill/Turn and Multi-Task Machining the system generates four Coordinate Systems – the ZX, XY, HY backside and YZ planes. The XY, HY backside and YZ planes are shown in the following set of graphics.

XY HY Backside YZ

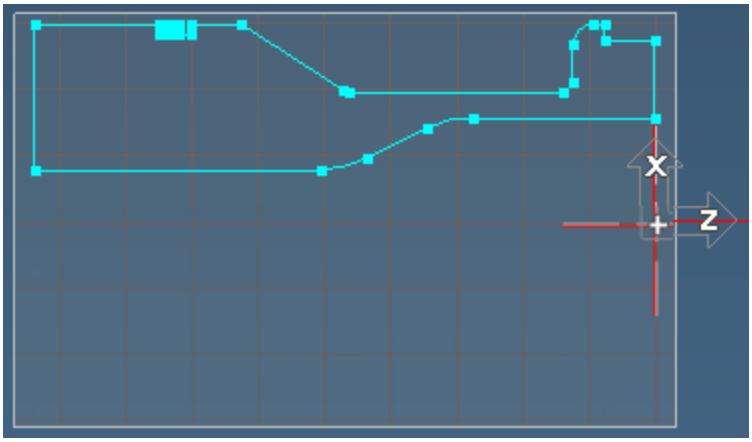
Geometry Creation and Arrangement

In this exercise we will create geometry for the "Dual Spindle.vnc" part created in Exercise 1, and transfer it to the sub spindle. For those with a single spindle machine, this exercise can be seen as a review of how to create and modify geometry.



The part we will make looks like this image. Do not worry if you do not have Advanced CS or Mill/Turn functionality to complete the milling features of the part, because the exercises are designed with this in mind.

1. For the part print, see "Main Spindle" on page 121. Create the geometry in CS1, the ZX plane.

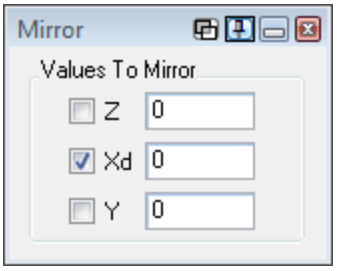


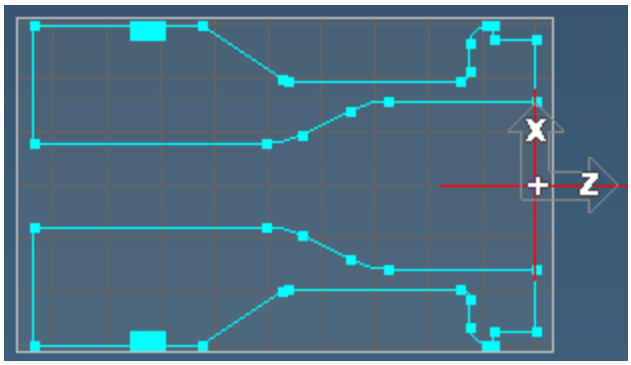
When using the Multi-Task Machining product it is often convenient to have geometry on both sides of the part's centerline. This is particularly convenient when performing -X side machining. While you can select geometry on the +X side, the toolpath will show up on the -X, thus it is good

to have geometry to show the toolpath clearly. An example of this can be seen in the machining tutorial section.

2. Select the geometry (`Ctrl+A`).

3. Modify> Duplicate and Mirror the geometry about the Xd at 0.





We will now copy the geometry to the sub spindle.

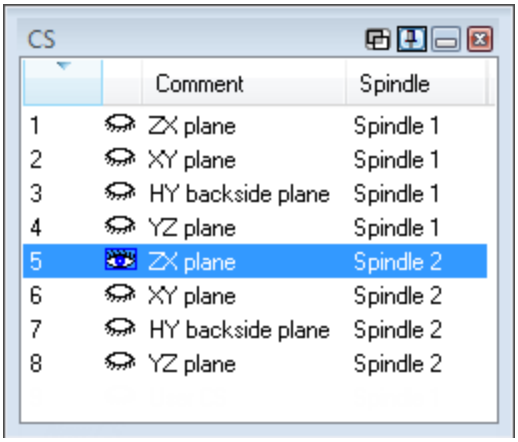
4. Select and duplicate all of the geometry (`Ctrl+A`, `Ctrl+D`).

> Having the geometry for both sides of the part can be very helpful, especially when creating operations that cut on the -X side of the part. Having geometry on both sides of the center line allows you to actually set the machining markers on the -X side in addition to comparing toolpath against part geometry.

You now have two overlapping sets of geometry: the original, which is unselected and the duplicated geometry.

5. Open the CS list .

6. Select the ZX plane as the target spindle from the CS list.



Changing coordinate systems changes the color of the geometry in Spindle 1's ZX plane. Geometry that is magenta (pink) is in a non-current coordinate system but is still in the same

workgroup. This is shown underneath the selected geometry. The selected duplicate geometry changes from red to green.

7. Choose Modify >CS> Change CS (HVD).

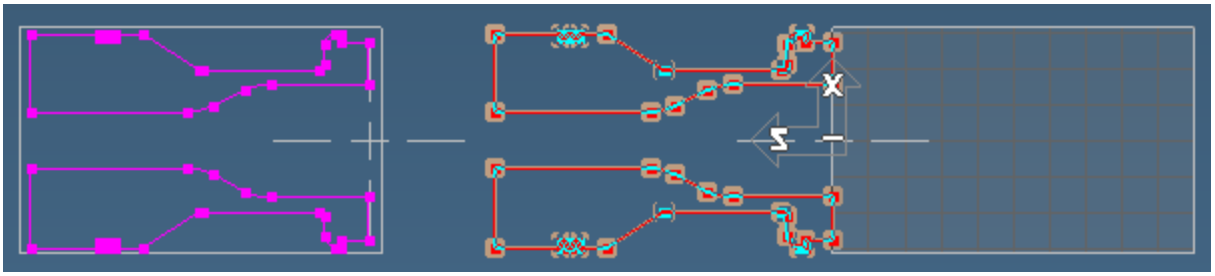
The Change CS (HVD) command moves and aligns currently selected geometry to the current CS. The geometry is placed in the target CS at the same position relative to the target CS's origin, as the geometry was in the original CS. In other words, if the geometry is a point at Z-1, X1 in CS1, it will be placed at Z-1, X1 in the target CS.

The duplicated geometry has now been moved to the ZX plane of the target spindle. It is still selected and therefore highlighted in red.

The geometry is mirrored and now needs to be flipped to align the geometry correctly.

8. Choose Modify > Mirror and mirror the geometry about Z0.

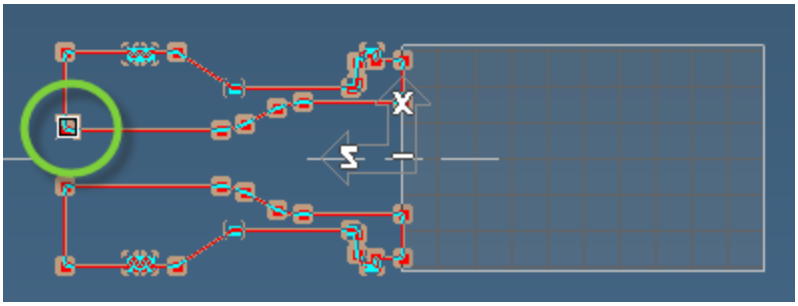
This has flipped the geometry over so that it is facing the correct direction, as shown below (the geometry is still selected). We will now translate the geometry into place.



9. Choose Modify > Translate.

We will now interrogate the Z value of a geometric element that is supposed to lie at Z0.

10. Interrogate the Z coordinate (`Alt+click`) the point shown.



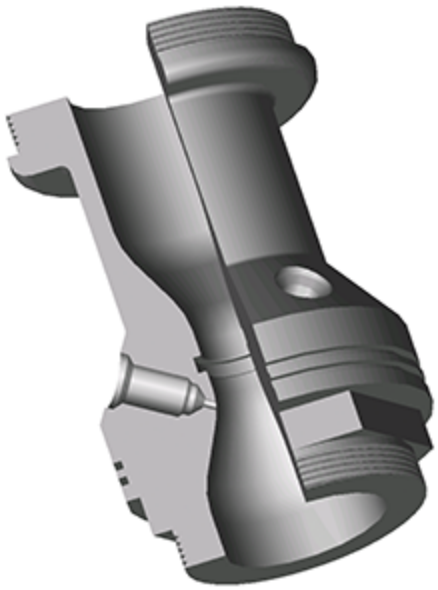
11. Change the interrogated value to a negative value.
12. Click the Do It button, and click away from the part to deselect it.

    The transferred geometry is now aligned properly.

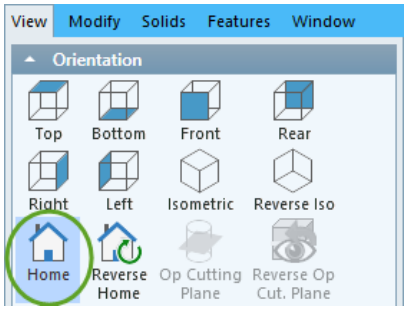
Sub Spindle Geometry

In this part of the exercise we will modify the Dual Spindle.vnc part file to add on backside geometry. In making these modifications we will be working in a different coordinate system than you may be accustomed to. Despite this, you will find this no different to working in the standard ZX plane.

We need to make modifications to the part file. We will be defining the ID geometry as shown in the cutaway view of the part below.



We need to modify the geometry that was duplicated and moved to the sub spindle. We do not need the part profile that is on the -X side of the sub spindle.

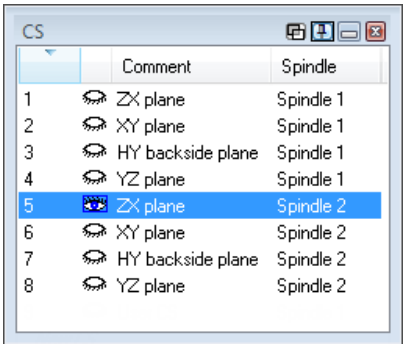
1. Switch to the Home View 

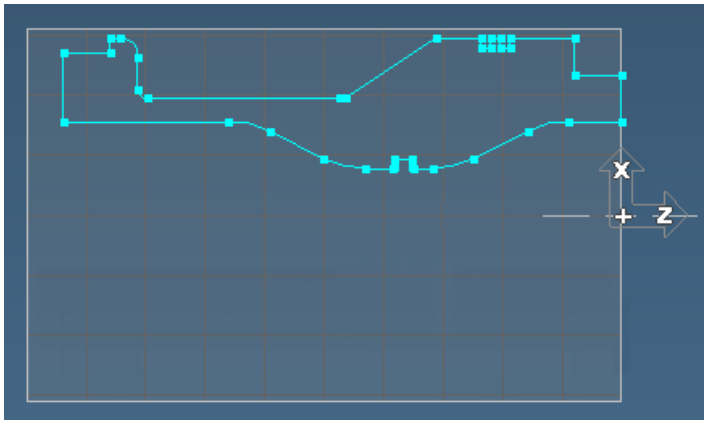
 (`Ctrl+H`).

   The Home view changes the current view to be looking down the depth of the current CS. This flips your view of the part so that the sub spindle is on the left and the main spindle is on the right. Your view is now from the back of the machine.

2. Delete the geometry on the -X side of CS5: ZX plane - S2: Spindle 2.

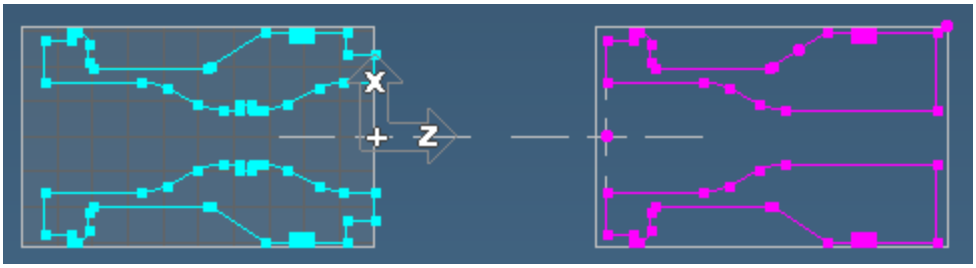
3. For the part print for this exercise, see "Sub Spindle" on page 122. Create the geometry in CS5, the ZX plane.





4. Duplicate and mirror the geometry about X0.

   The geometry for the part is complete. Save the part as `Dual Spindle.vnc`.


X
Z

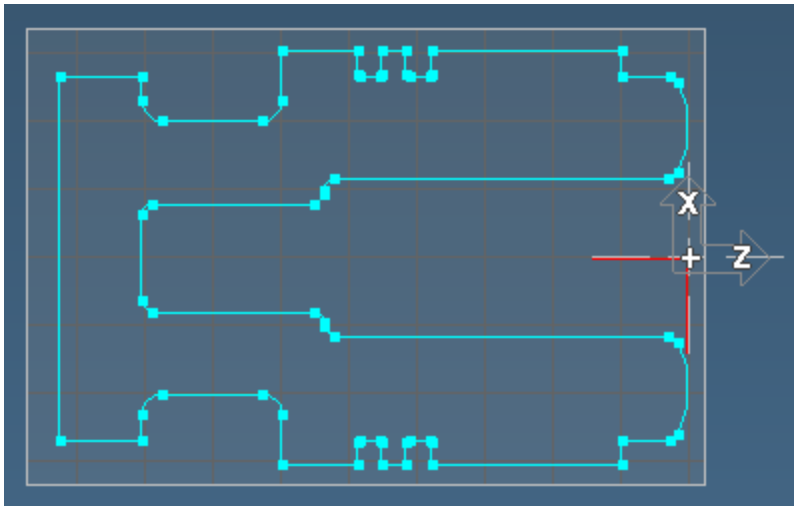
TOOLING TUTORIALS

This section of the tutorials show the creation and setup of tools for the Multi-Task Machining system. There are a total of three exercises. The first exercise is for a single spindle, dual turret machine. The second exercise is for a dual-spindle, dual-turret machine. The last exercise is for a Swiss-style machine.

The first two tutorials are created using the generic Multi-Task Machining MDD called Training Twin Turn (2T-2S). It is recommended that you go through all of the tutorials, even if you do not have a machine that matches the tutorial's description. For the Machining tutorials, see "Multi-Task Machining Tutorials" on page 41. You may either complete all of the tooling tutorials and then work on the machining exercises or you may do a tooling tutorial then move to the corresponding machining exercise.

Basic Tool Setup

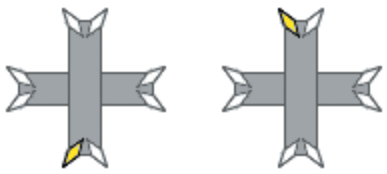
In this exercise you will be introduced to setting up tools for Multi-Task Machining parts. First we will create a part that is machined on a single spindle, dual turret machine. We will start with a part file that already has geometry. We will be creating six tools for OD and ID turning as well as OD milling. Once the tool creation is complete, we will machine the part, see "Single Spindle Part" on page 41.



1. Open the part file Single Spindle.vnc.

   If you do not have the part file you may create it using the geometry part print, see "Single Spindle Part" on page 123.

   Pay careful attention to the tool orientation diagram and the Tool Group selection. If a tool is pointing down it will be in the upper turret and if it is pointing up, it will be in the lower turret.

   Upper Lower

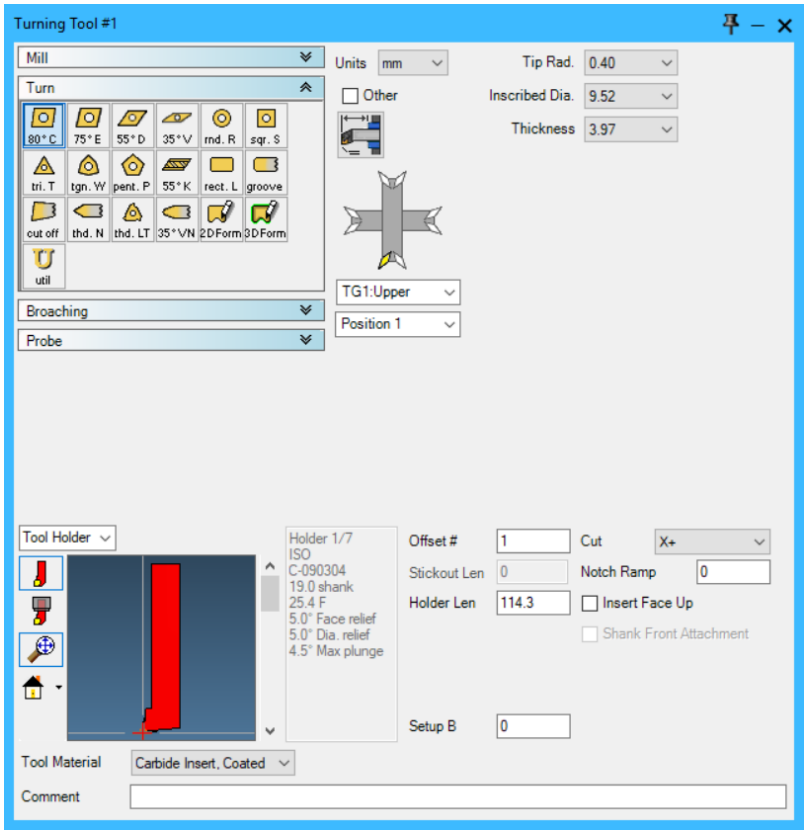
2. Open the tool list.

Tool Group 1

1. `Double-click` space #1 in the Tool list to create a new tool that will be used to face the part.

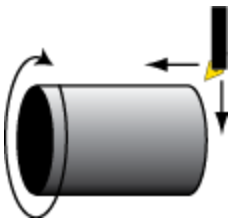
Face Tool

2. Create Turning Tool #1 as shown.



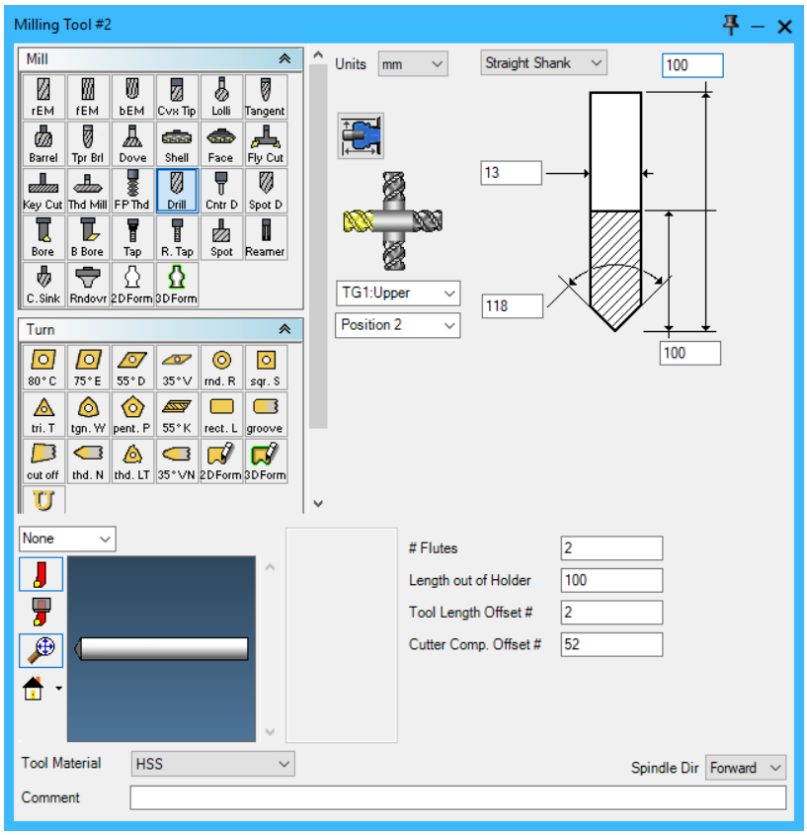
The tool is assigned to Tool Group 1, is in the first position, and is aligned to the primary spindle. The tool will be cutting on the +X side of the part and since the spindle runs counter-clockwise, the insert is Face Down. The insert Orientation is set to use a horizontal toolholder and the insert is set to cut down.

The tool diagram shows the insert on the backside of the tool holder because the tool is face down.



Face Drill Tool

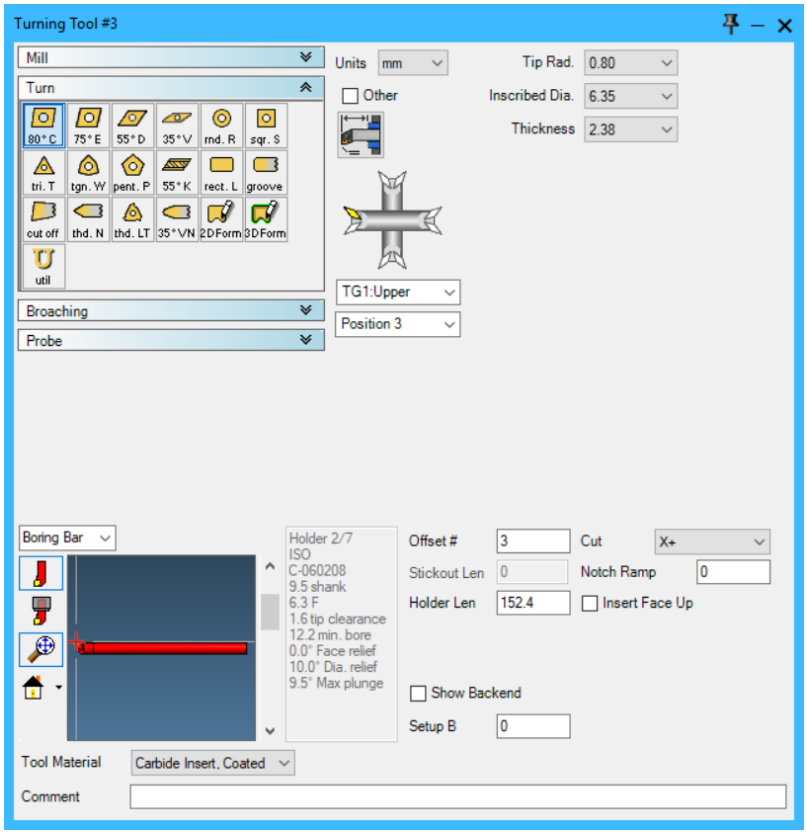
1. Create Milling Tool #2 as shown.



This drill is set to drill the face of the part. The drill is in Position 2 of Turret 1 and aligned with Spindle 1.

We will now create a tool that will be used for ID roughing and contouring.

2. Create Turning Tool #3 as shown.

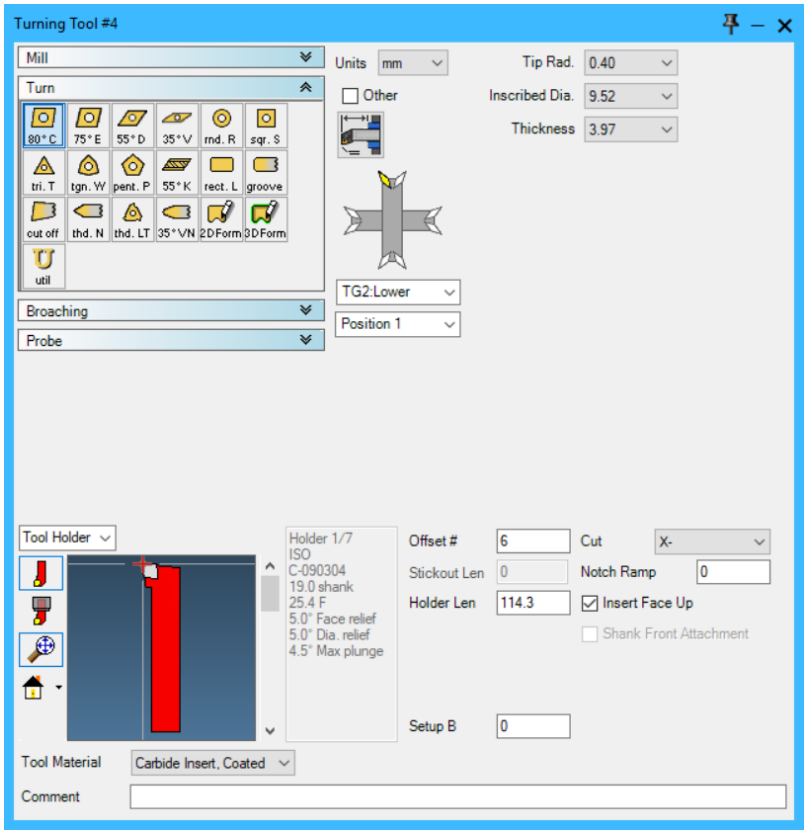


This tool has a similar setup to Tool 1 but the insert is at the top of the holder.

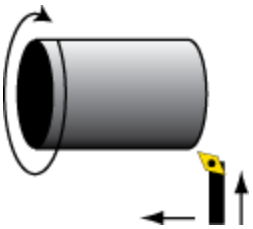
Tool Group 2

OD Roughing

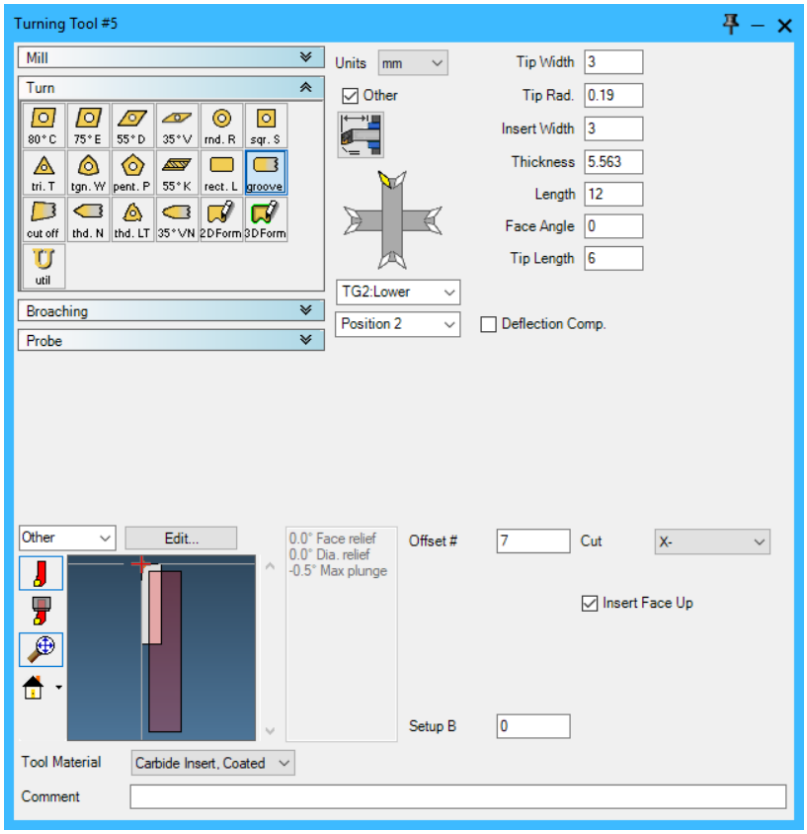
1. Create Turning Tool #4 as shown.



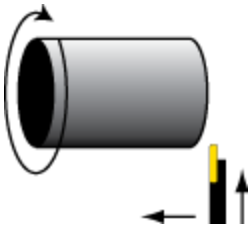
This tool will be used for OD roughing. This is the first tool we have defined in Tool Group 2 and is in Position 1. The tool is set to cut from the –X side of the spindle. Because the tool is cutting on the –X side, the Insert is Face Up.



2. Create Turning Tool #5 as shown.



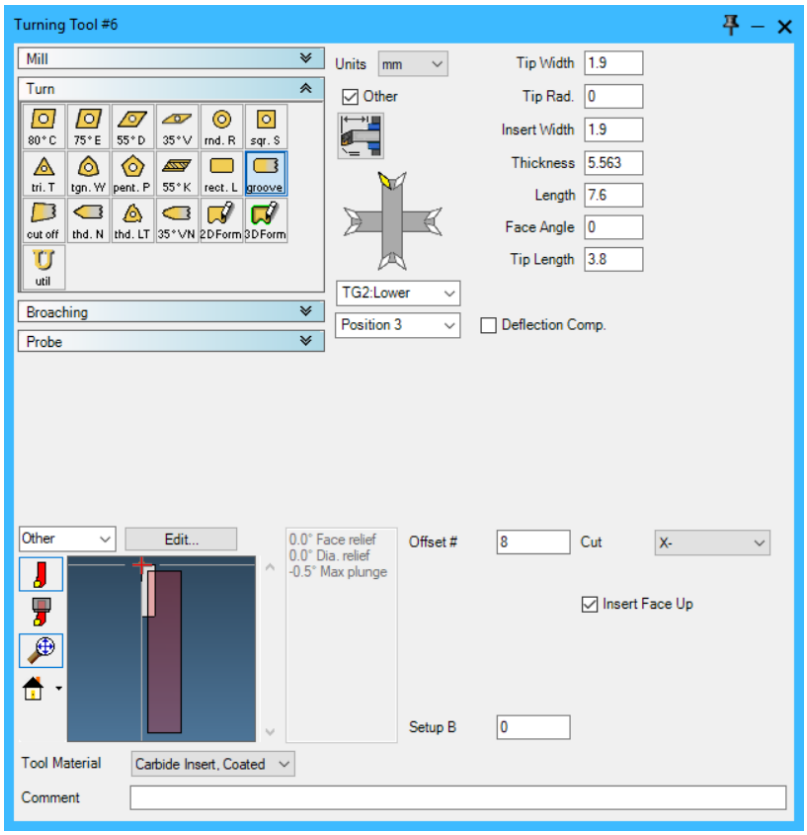
This tool will be used in a Material Only OD Roughing operation.



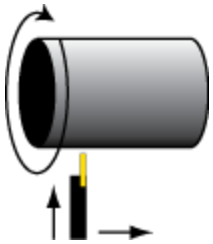
This tool is in the second position of TG2. The setup is identical to Tool #4, the tool is cutting on the -X side of the part and is Face Up.

OD Groove and Finish

1. Create Turning Tool #6 as shown.

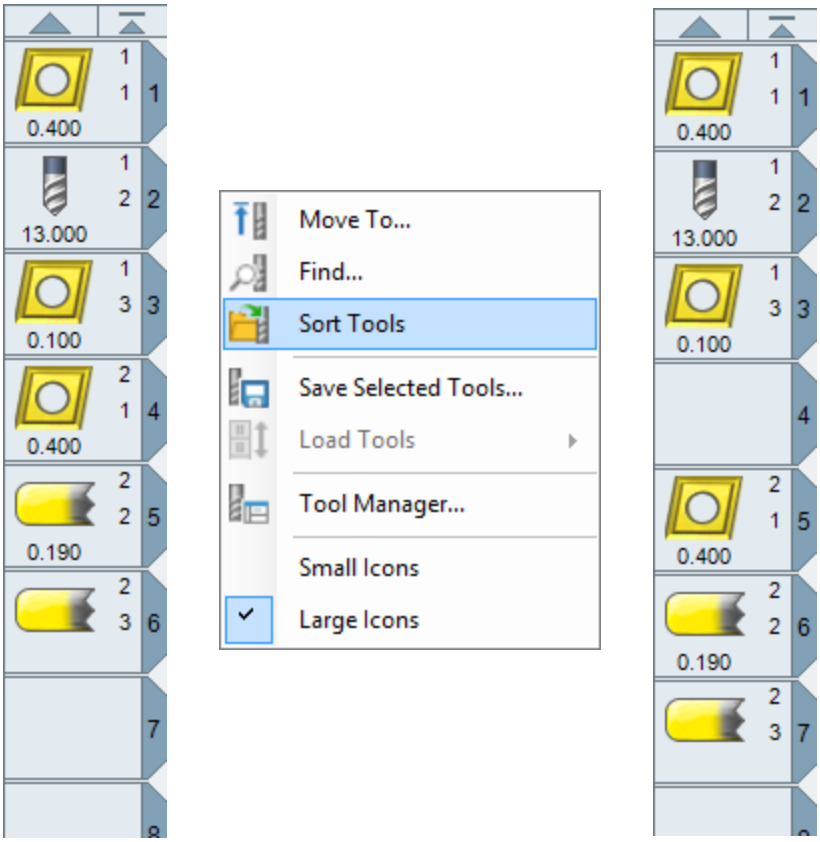


This groove tool will be used to cut the grooves and finish the part. The tool is set to cut from the bottom of the part but will cut from the back of the part. We will now organize the Tool list.



2. `Right-click` any of the Tool tiles and choose Sort Tools.

The tools are sorted by Tool Group.

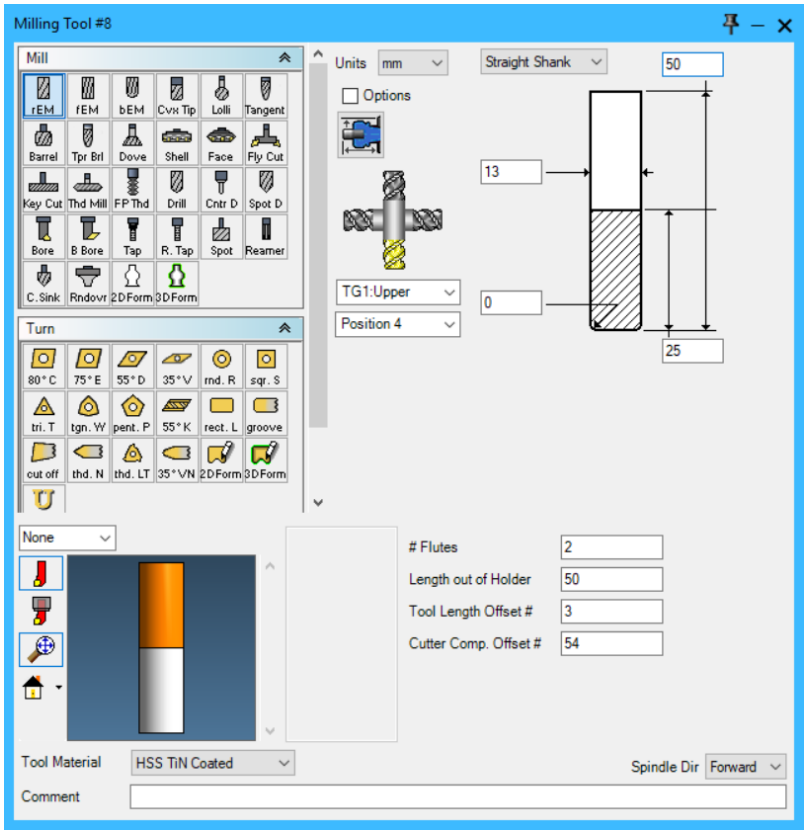


3. Save the part. If you do not have the Mill option, skip to the next exercise.

Mill Turn Tools

OD Mill

1. Create Milling Tool #8 as shown.

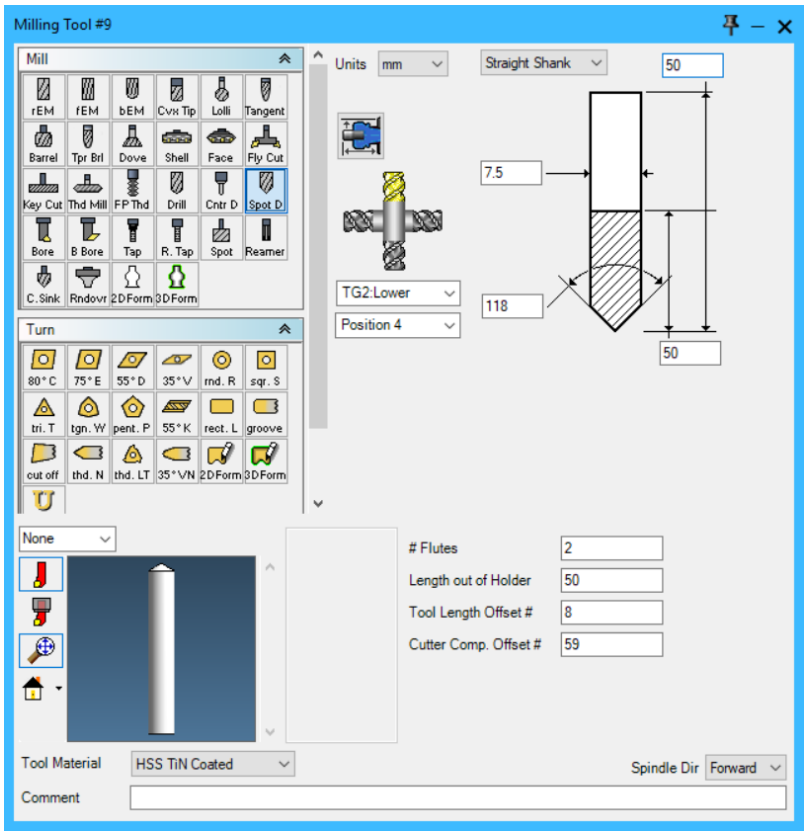


This endmill will be used to mill a hex on the OD of the part from the +X side.

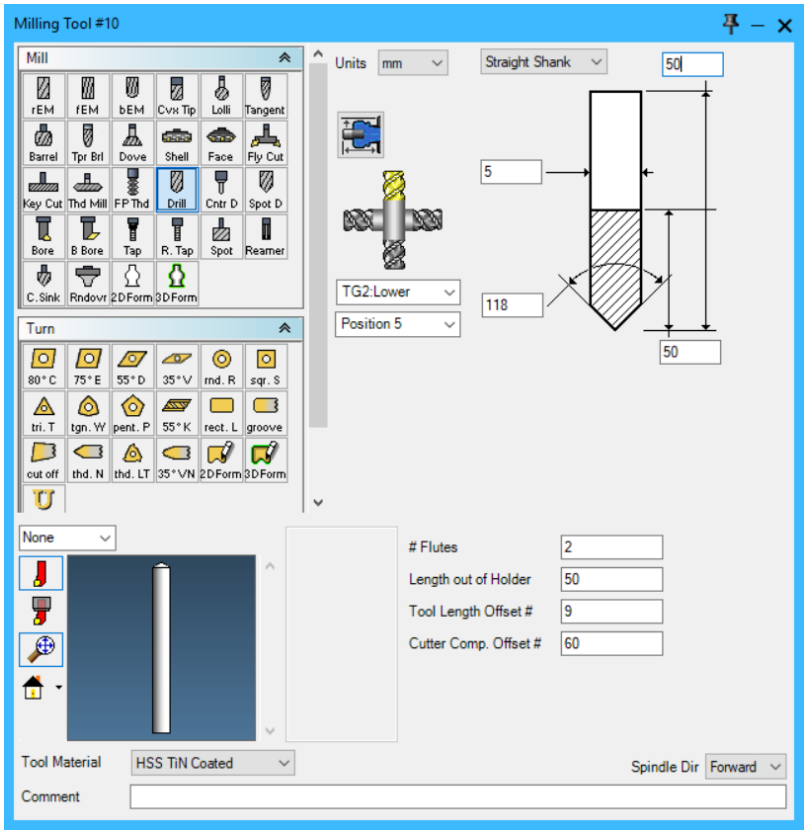
The next two tools will drill holes on the OD from the -X side in TG2.

OD Drills

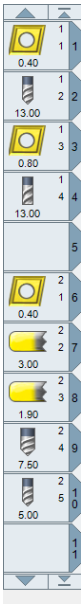
1. Create Milling Tool #9 as shown.



2. Create Milling Tool #10 as shown.



3. `Right-click` a tool tile and choose Sort Tools again.



4. Save your part file.

For the machining tutorial for this part, see “Single Spindle Part” on page 41. You may either continue on to the next tooling exercise or move on to the machining of this part.

Dual Spindle Setup

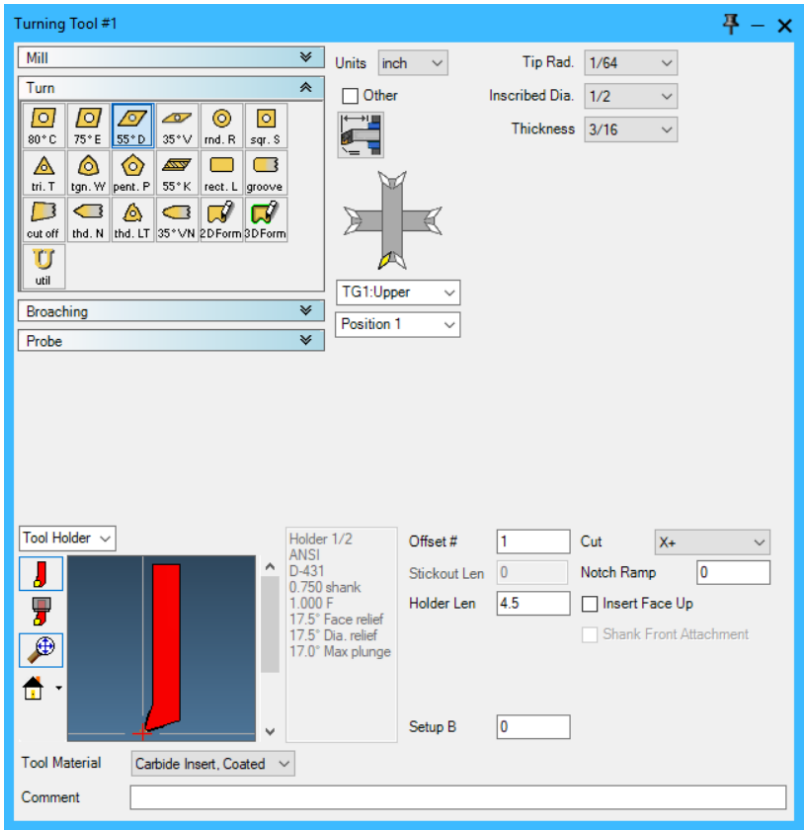
In this exercise we will set up the tools for cutting the dual-turret, dual spindle part earlier in the tutorial. For details, see “Geometry Creation and Arrangement” on page 8. The MDD (Machine Definition Document) we are using is a dual-spindle, dual-turret machine. Turret 1 and 2 may access either spindle. Turret 1 is on the +X side of Spindle 1 while Turret 2 is on the -X side of Spindle 1. The tooling we will set up will have Tool Group 1 accessing only Spindle 1 and Tool Group 2 will access both spindles. We will define both turning and milling tools. As with previous exercises, you do not need the milling option to do this exercise. We will define a total of 21 tools to cut this part.

1. Open the part file Dual Spindle.vnc.

2. Open the Tool list. 

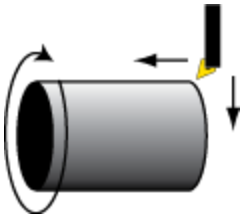
The tools are created in the order of their accompanying operations, by Tool Group. This is done strictly for convenience in this exercise and does not need to be true for other part files.

3. Create Turning Tool #1 as shown.

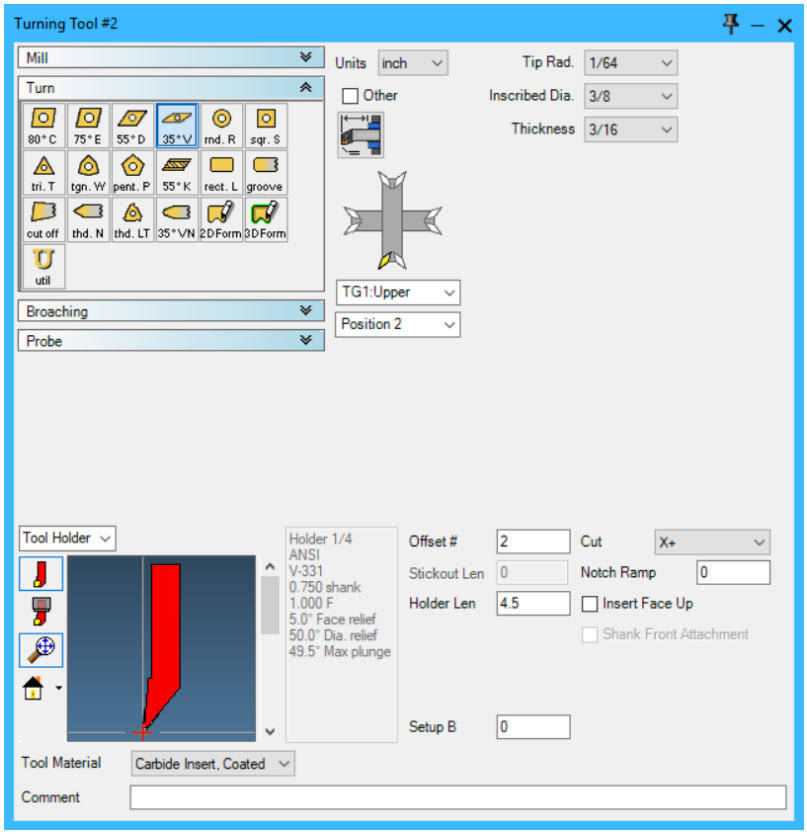


This tool is Face Down in Position 1 of Tool Group 1. As with all tools in TG 1, this tool cuts on the +X side of the part. This tool will be used to face and rough the OD.

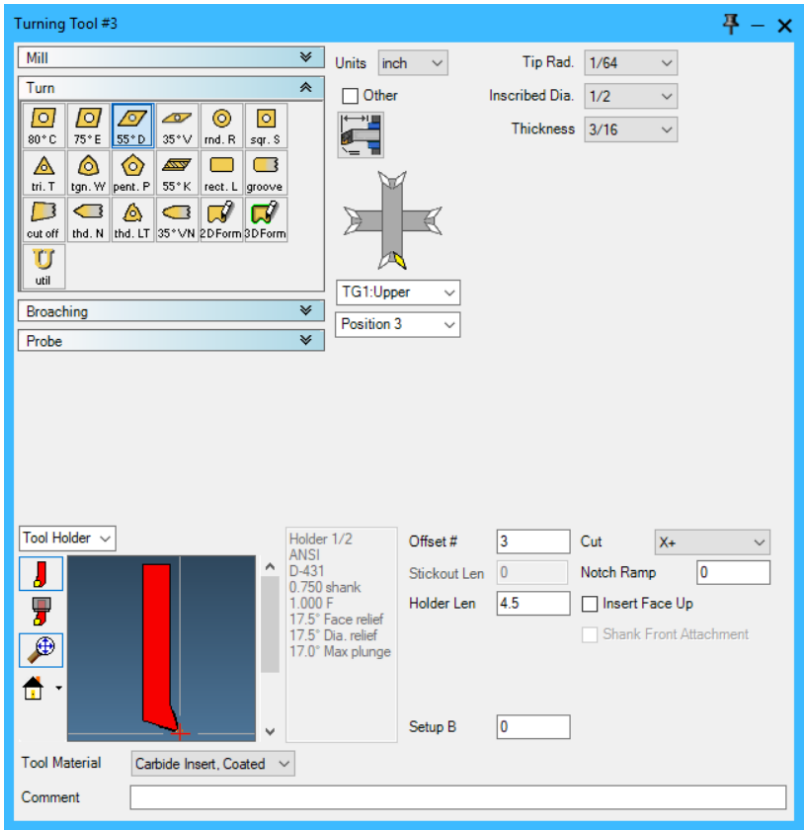
The tool and insert orientation as described above is similar to the image above. The insert is Face Down, approaching from the top of the spindle.



4. Create Turning Tool #2 as shown.

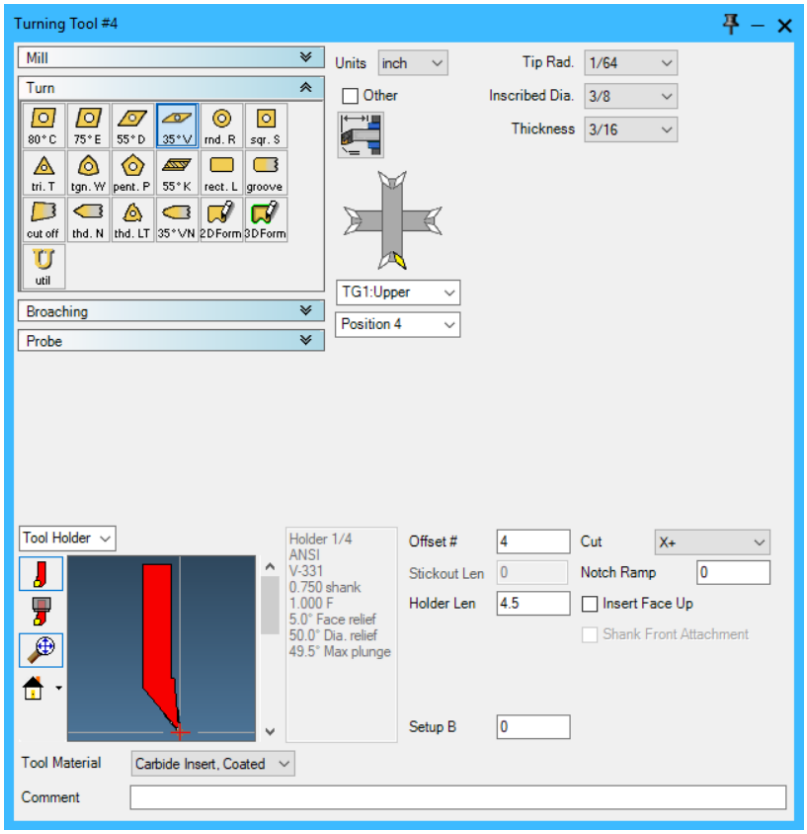


5. Create Turning Tool #3 as shown.



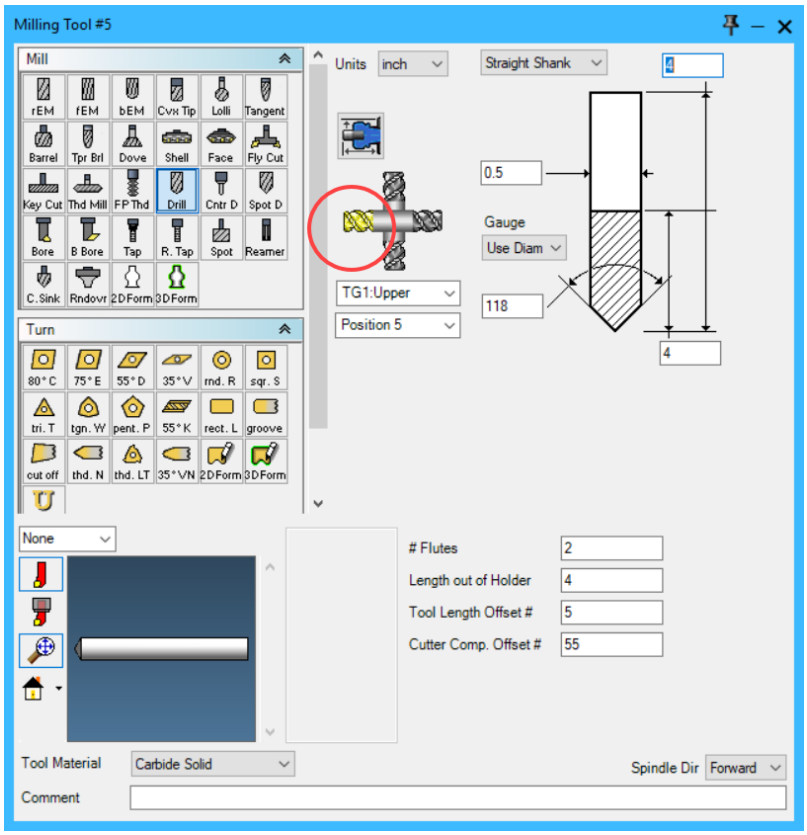
The tool is oriented to cut from the spindle towards the face of the part.

6. Create Turning Tool #4 as shown.



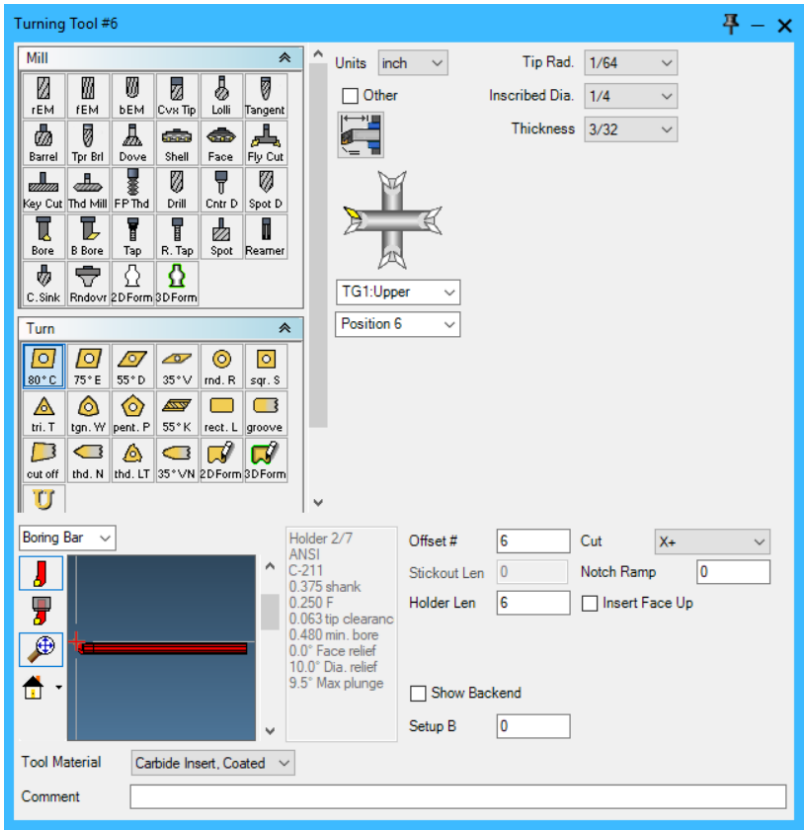
The arrangement of this tool is identical to tool #3.

7. Create Milling Tool #5 as shown.



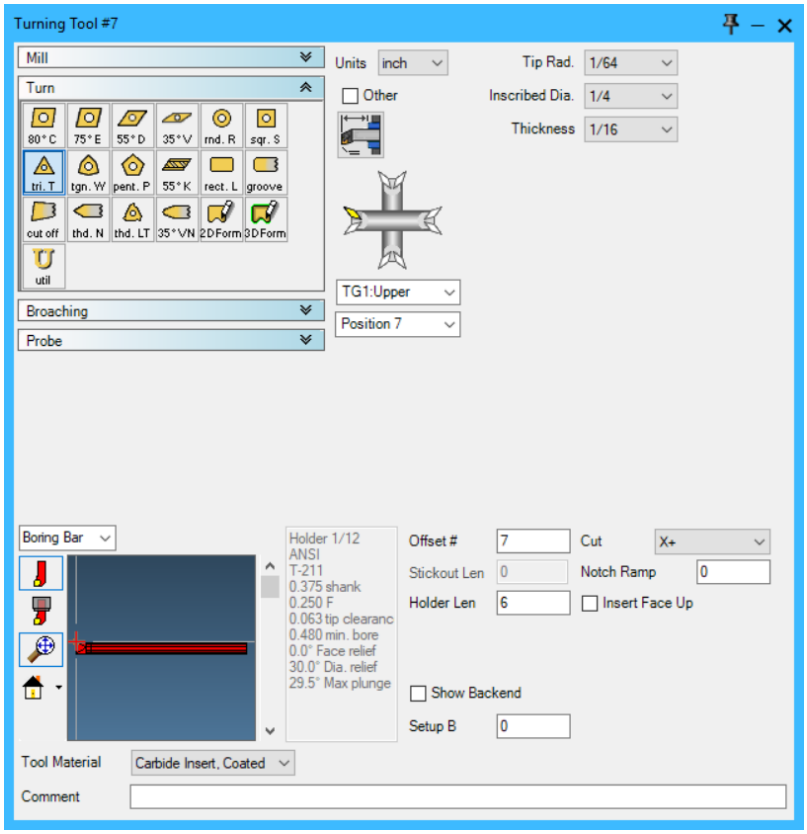
Note that, for clarity, the correct orientation is circled.

8. Create Turning Tool #6 as shown.



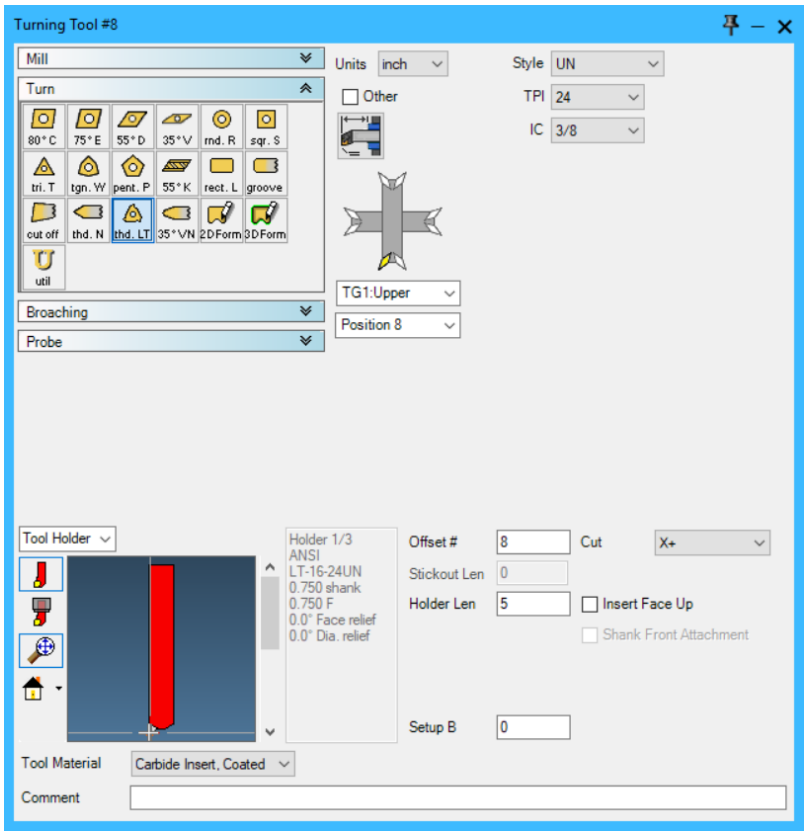
This tool will rough the ID of the part.

9. Create Turning Tool #7 as shown.

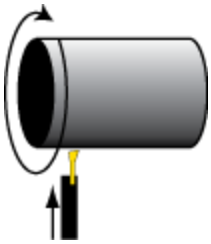


This trigon will be used to finish the ID of the part. This tool's arrangement is identical to tool #6.

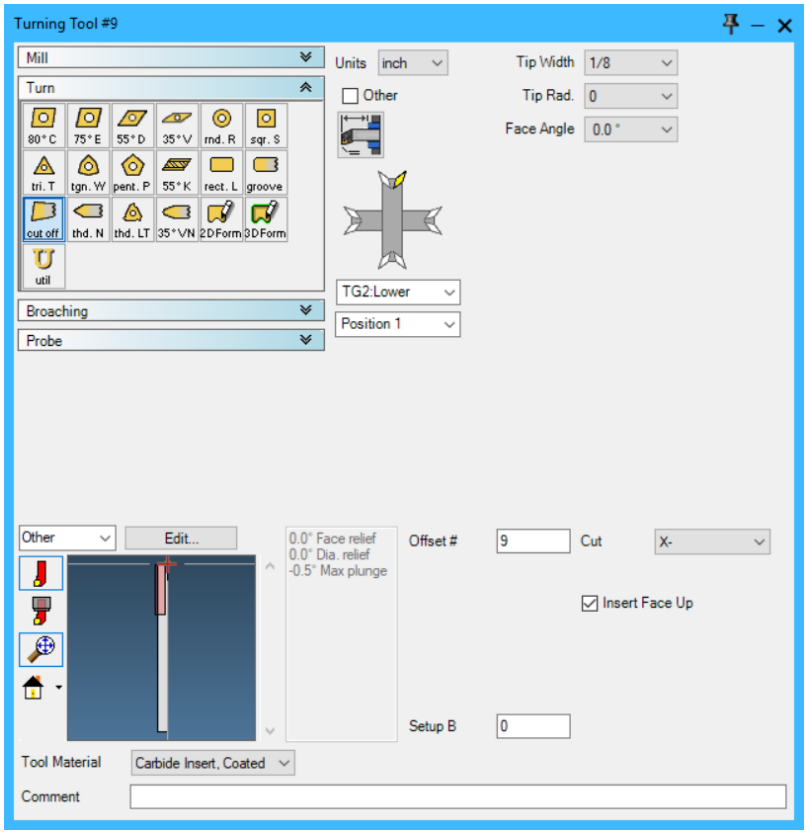
10. Create Turning Tool #8 as shown.



We will now create the tools contained in Tool Group 2. Tool Group 2 approaches the part from the -X side. Most of the tools will be set to cut on Spindle 2 but several will access Spindle 1.



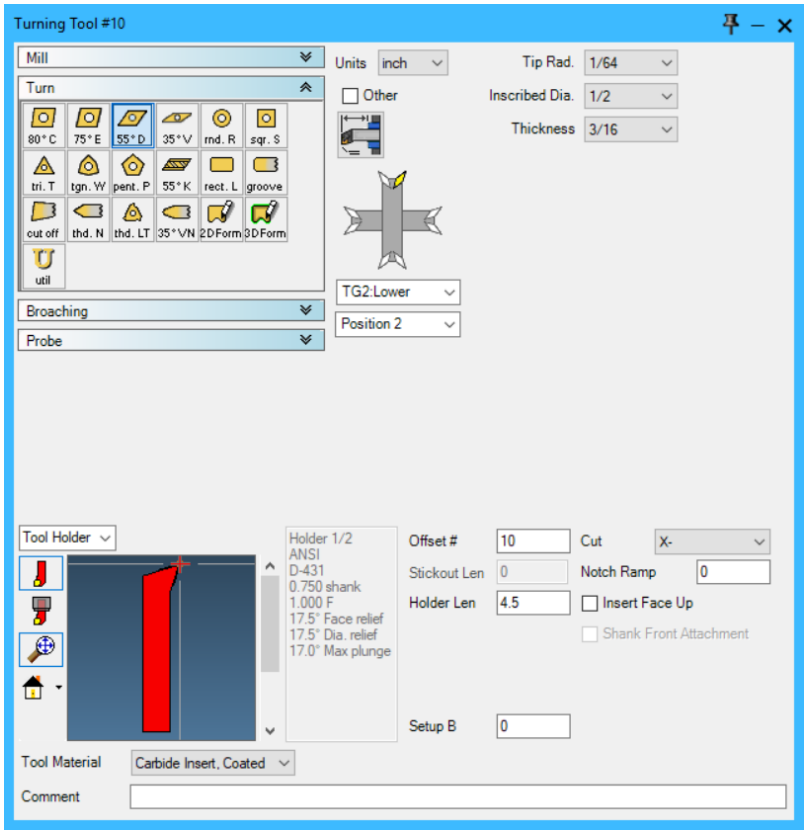
11. Create Turning Tool #9 as shown.



This tool will be the last tool used in the program.

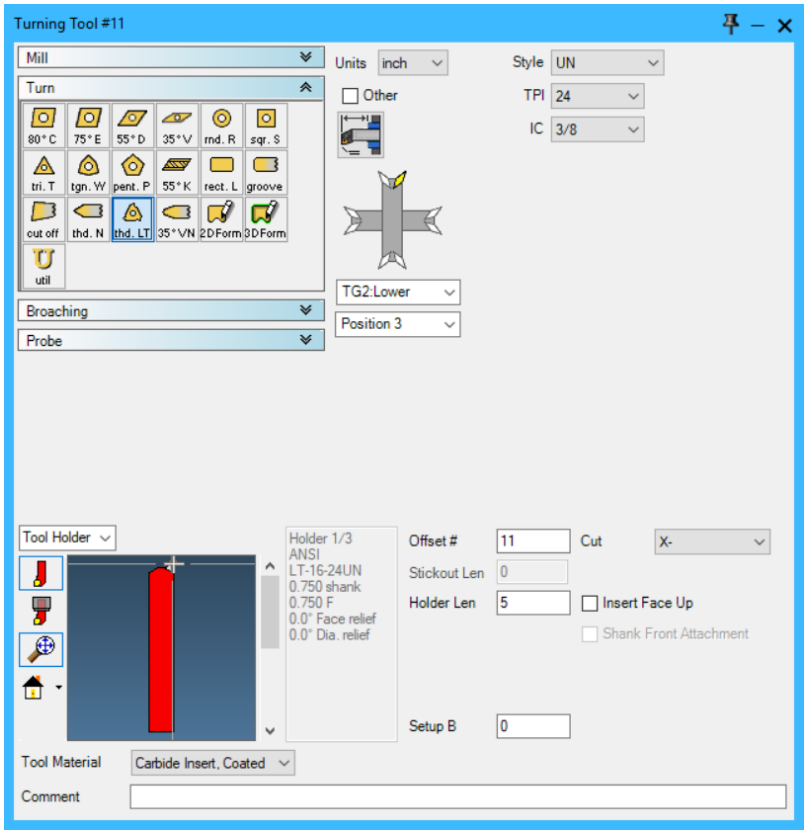
This tool is being placed in Position 1 of TG2. Second, the tool is oriented to Spindle 1. That means the Insert Orientation will be set in relation to Spindle 1, not Spindle 2. Third, this tool is cutting from the -X side, as are all tools in Tool Group 2. Lastly, note that this tool is designated as being Insert Face Up. Were this tool being used on Spindle 2 this setting would be Face Down but Tool Group 2 is moving to Spindle 1 for this operation, As the spindle moves counter-clockwise and the tool is below the spindle, the insert should be Face Up.

12. Create tool #10 as shown.



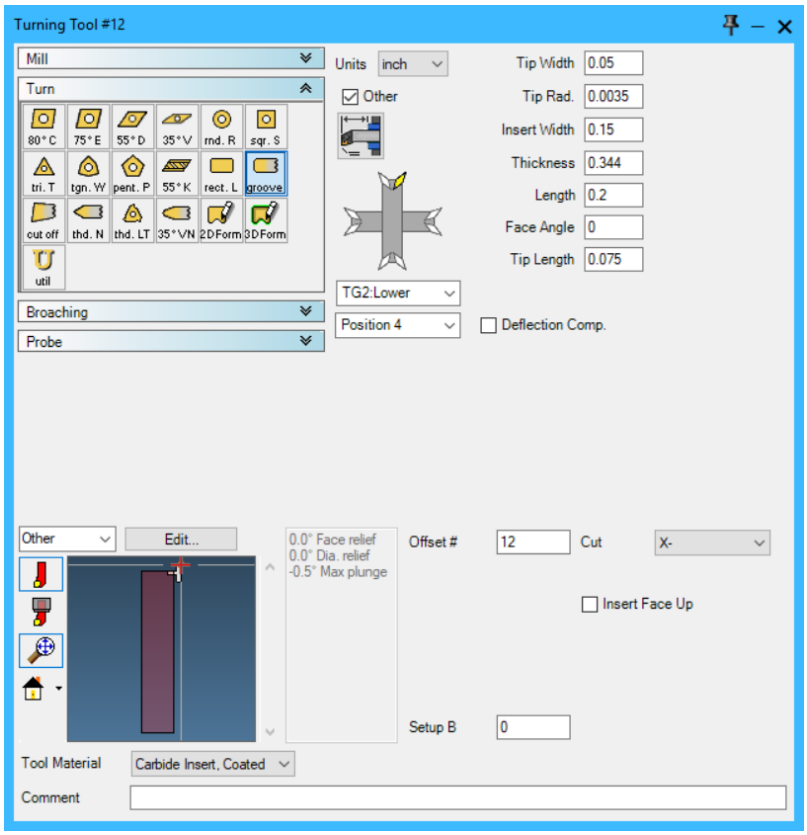
This tool will be used to rough the OD of the part once transferred to Spindle 2.

13. Create Turning Tool #11 as shown.

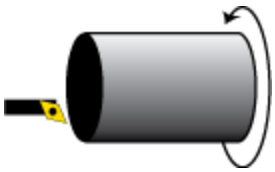


This will be used in an OD Threading operation. The alignment is identical to Tool #10.

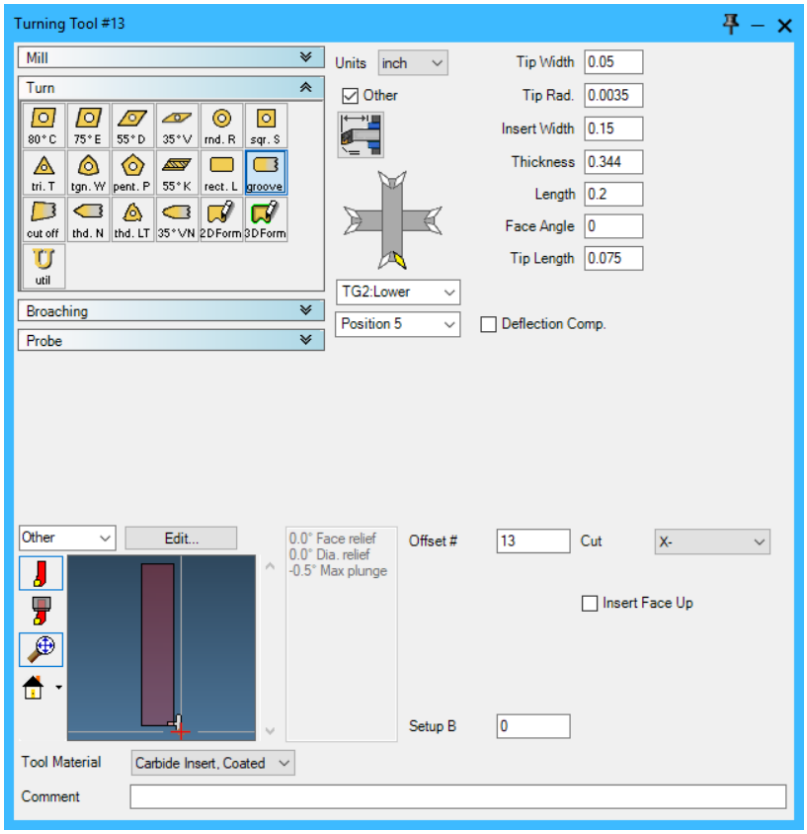
14. Create Turning Tool #12 as shown.



This tool will create the part's OD grooves. The alignment is identical to Tool #10.



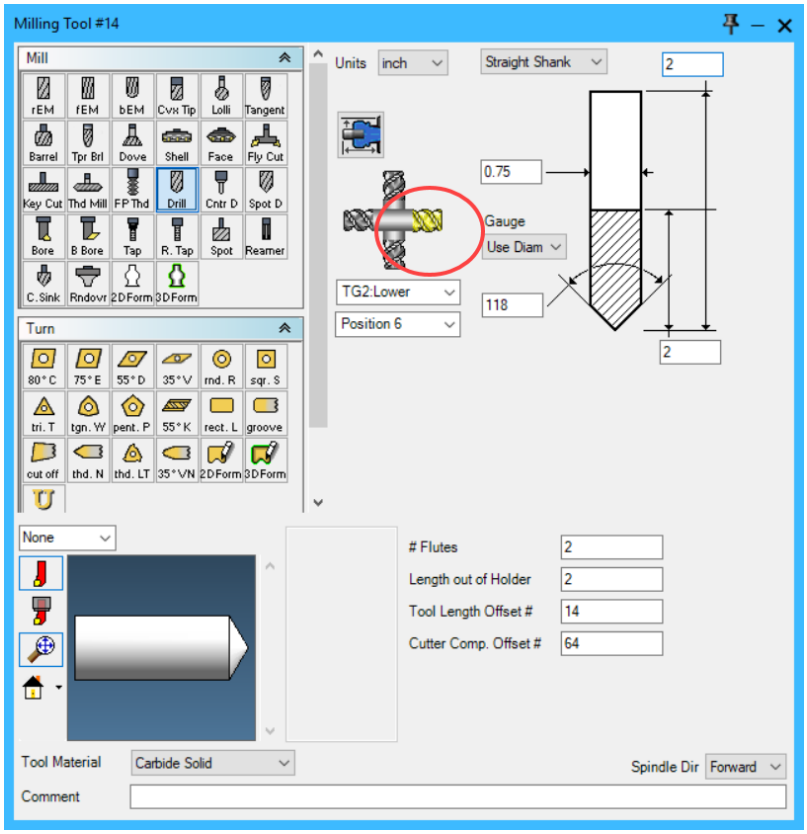
15. Create Turning Tool #13 as shown.



This tool will be used for cutting the part's ID grooves.

Perhaps you were wondering how a tool that is intended to be used on the ID can be defined with the Insert Orientation diagram shown in the above image. And perhaps you noticed that the Insert Orientation diagram and the overlapping graphic are different. The overlapping graphic shows how the insert holder would look in reality. The Insert Orientation diagram simply is used to show how the insert will be held, not how the holder is set.

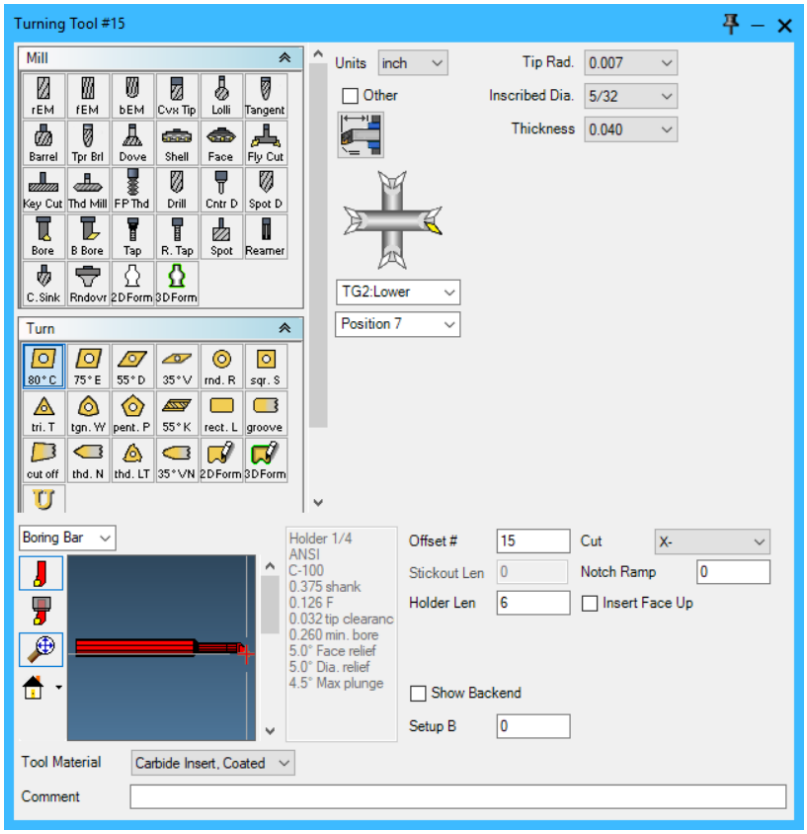
16. Create Milling Tool #14 as shown.



This drill will make a hole at the back of the part.

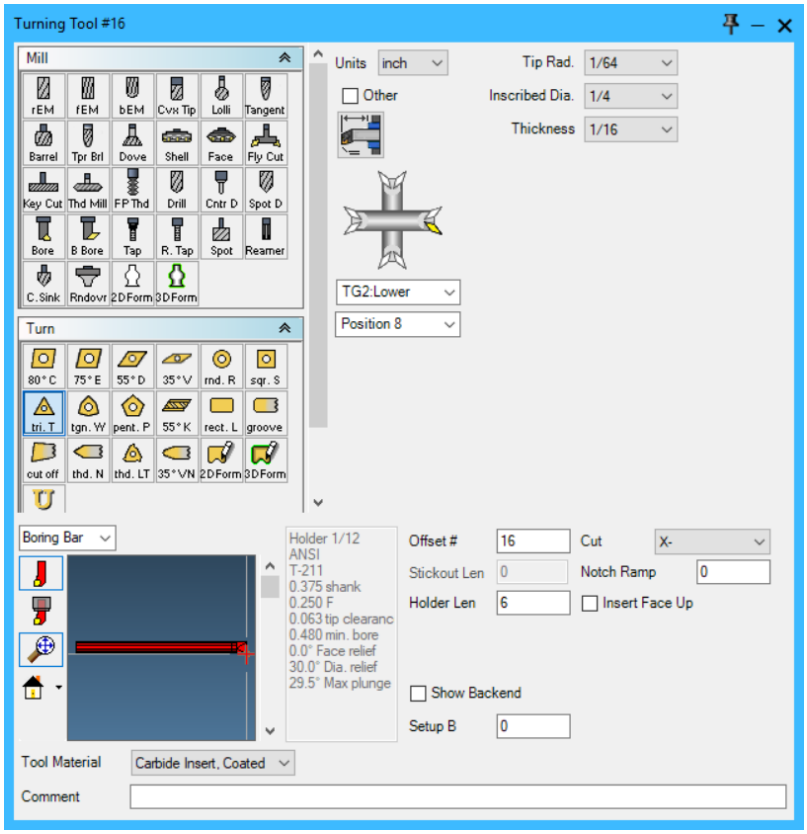
Spindle 1 is the Orientation Spindle for the tool. This drill will be cutting on Spindle 2, therefore the Tool Orientation diagram is set to cut away from Spindle 1, as shown in the image above.

17. Create Turning Tool #15 as shown.



This tool will be used to rough the ID.

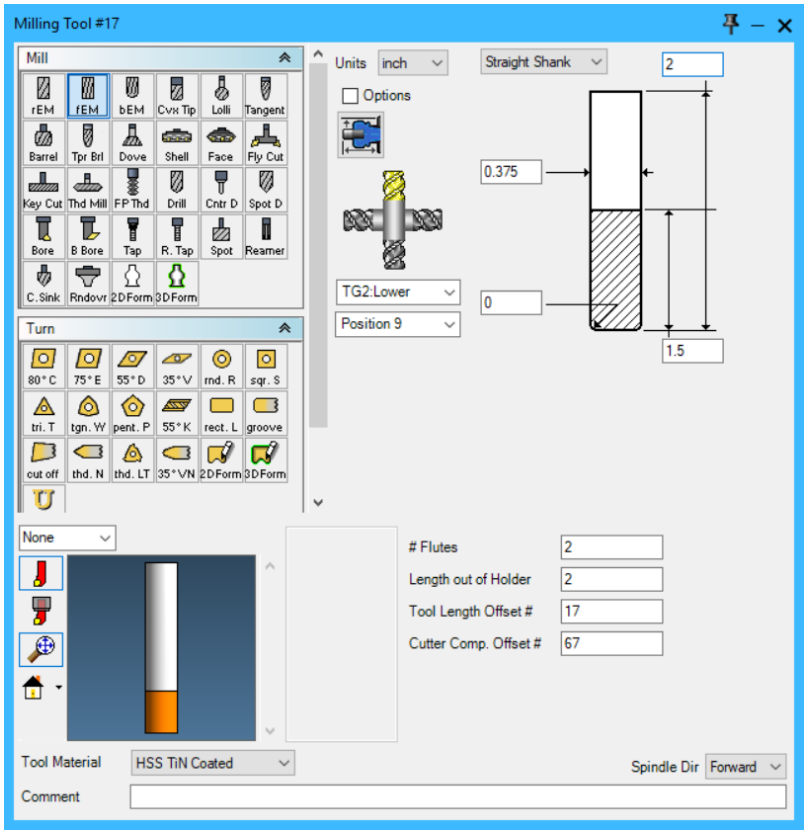
18. Create Turning Tool #16 as shown.



This tool will be used to finish the ID.

If you do not have Mill capabilities, you have completed the tool creation for this part. You may move on to the Machining tutorials. If you do have Machining capabilities we will now define these tools.

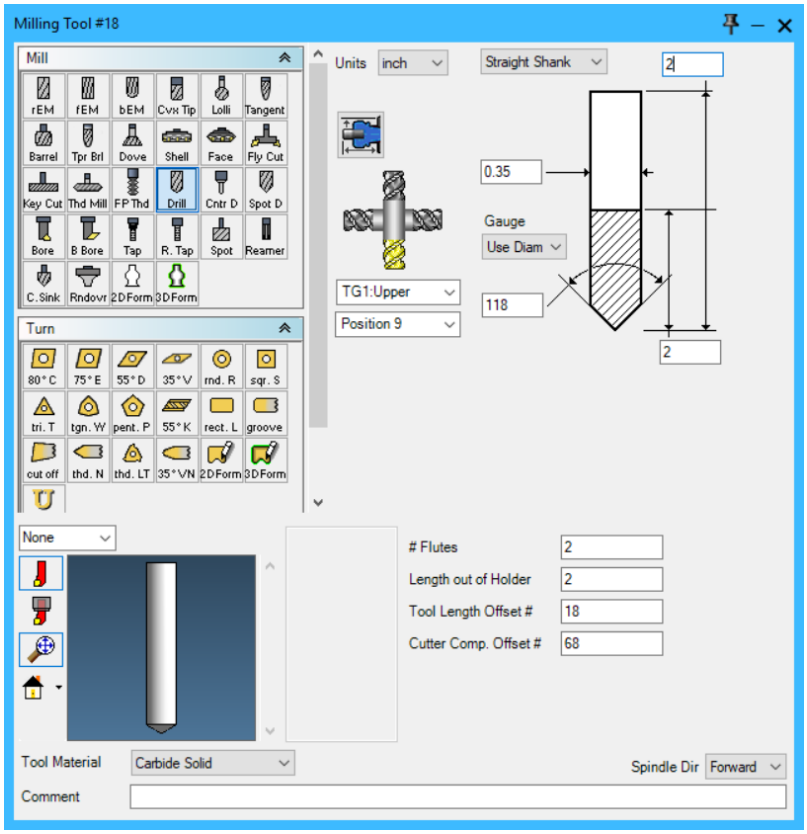
19. Create Milling Tool #17 as shown.



This tool will be used to create the hex on the back of the part.

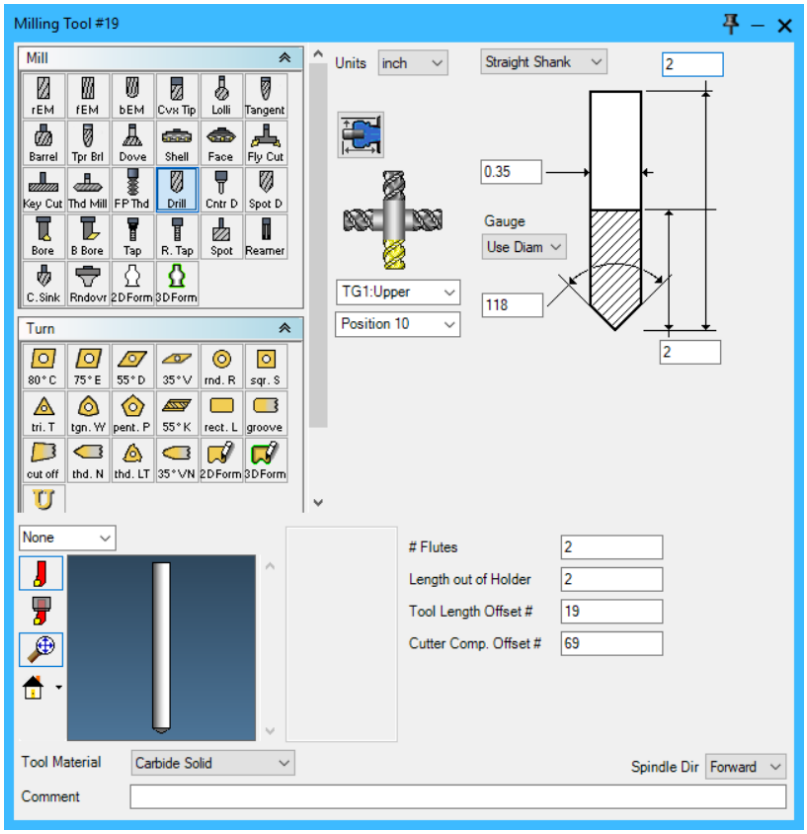
The next four tools are drills. The first two tools will be created in TG1, the others in TG2.

20. Create Milling Tool #18 as shown.



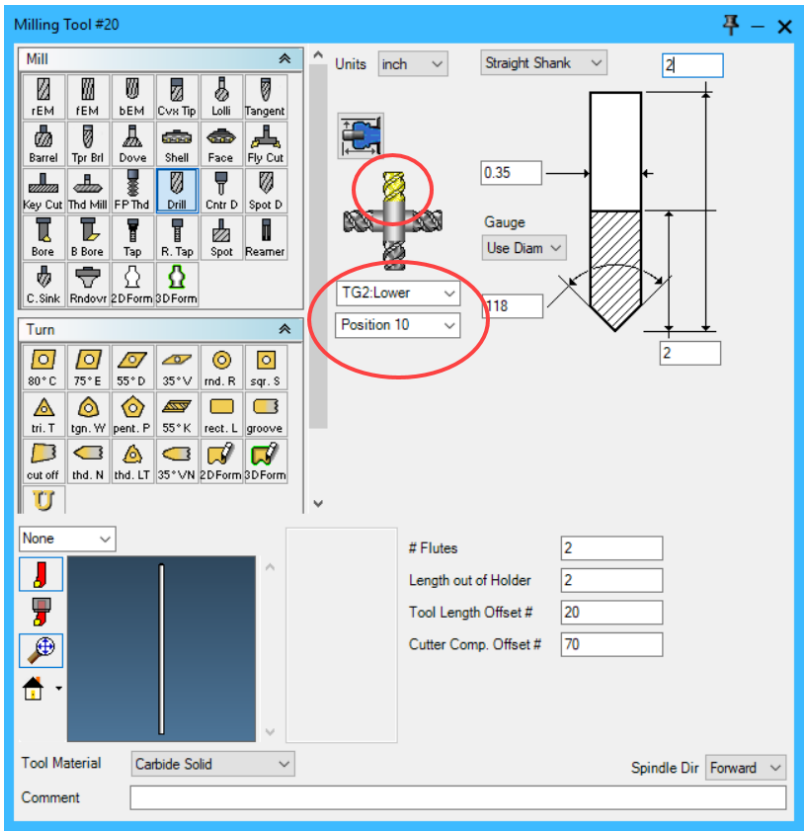
Note that this tool is in TG1.

21. Create Milling Tool #19 as shown.



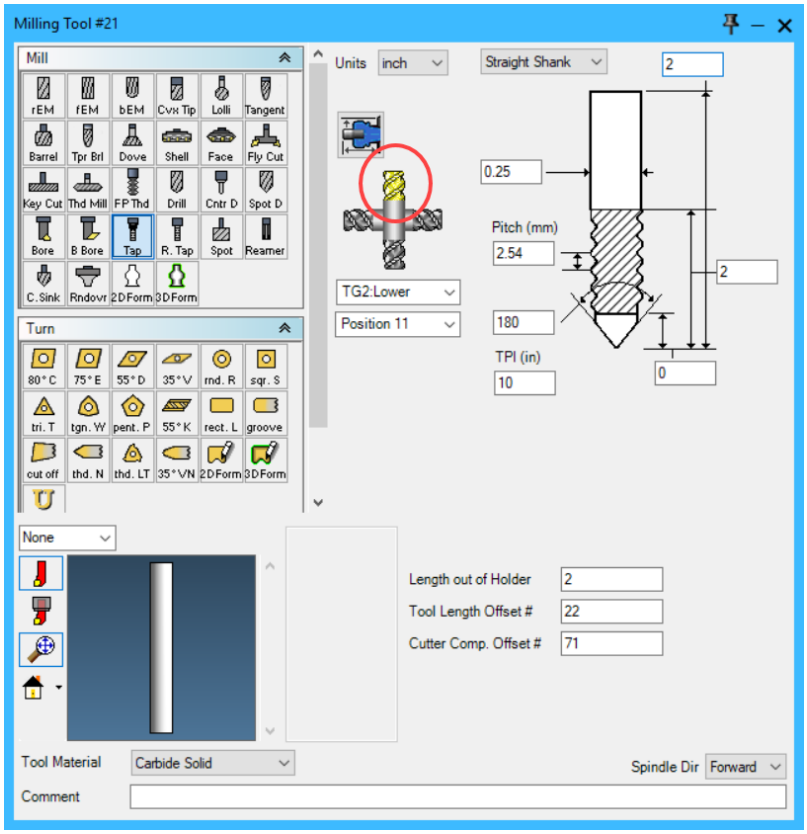
Note that this tool is also in TG1.

22. Create Milling Tool #20 as shown.



Note that this tool is in TG2.

23. Create Milling Tool #21 as shown.



Note that this tool is in TG2.

The tools for the Dual-Turret, Dual-Spindle machine exercise are now complete. Save the file as we will be using it later.

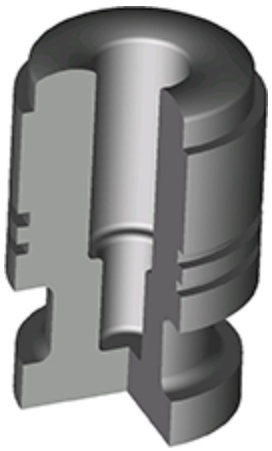
MULTI-TASK MACHINING TUTORIALS

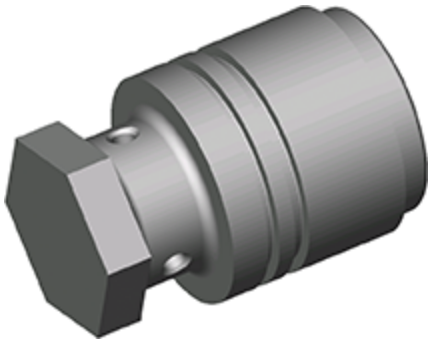
This section of the tutorials will introduce you to generating machining operations in Multi-Task Machining parts. There are three tutorials in this chapter with each increasing in complexity, starting with a single-spindle part, progressing to a dual-spindle part and finishing with a swiss-style part.

The tutorials are made using generic training MDDs (Training Twin Turn [2T-2S].mdd and Training Swiss [1P2T-2S].mdd). As such, anyone can create these parts, even if you do not have a machine of this type. It is recommended that you do all of the tutorials to maximize your familiarity with the system. Please note that these training MDDs may have options (particularly Utility operations) that you may not see when creating a part using your machine-specific MDD. This is typically because your machine does not support this function. Also, these training MDDs are generic and do not have a post processor; thus, they cannot produce G-code.

Single Spindle Part

In this exercise you will be introduced to creating machining operations for Multi-Task Machining parts. We will use the single-spindle, dual-turret part file created earlier in the tutorial, see "Basic Tool Setup" on page 14. This exercise will help you become acclimated to working with multiple flows and create simple operation synching. The exercise is designed for any user. You can complete this part whether you have Lathe or Mill/Turn functionality. Users with the basic Lathe package will complete the part shown below left. Once that is complete, users with Mill/Turn functionality will further machine the part to create the drill holes and Hex head (below right).





Setting up the Operations

Front Face

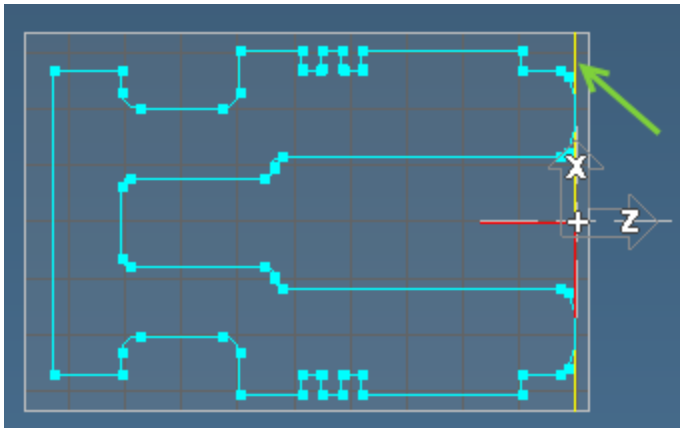
1. Open the file Single Spindle.vnc.

   This exercise does not use Auto Clearance. Auto Clearance is not used because of the relatively small amount of clearance available for ID cutting. This is the only exercise that does not use Auto Clearance.

2. Open the Tool and CAM lists.

We will begin this tutorial by creating a facing operation followed by drilling and ID turning operations. All of these operations will be made with tools from Tool Group 1. To face off the part we will need to add a line on which to set out machining markers because the front face of the part is an arc.

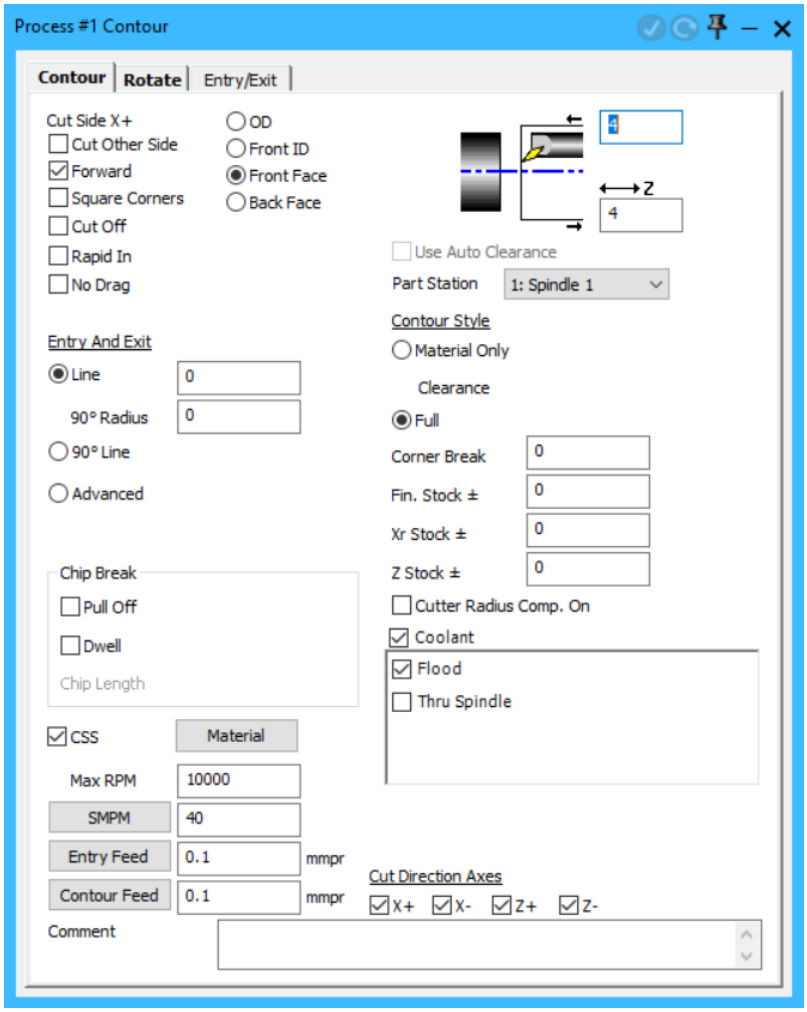
3. Create a vertical line at Z0.



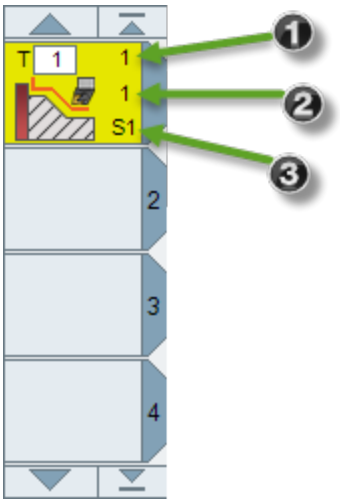
It does not matter how you draw the line. For this example we used the axis line tool.

The length of the line is not important because machining markers will specify the start and end points of the operation.

4. Create a Lathe Contour Process with the 0.4” 80° Insert (Tool #1) as shown.



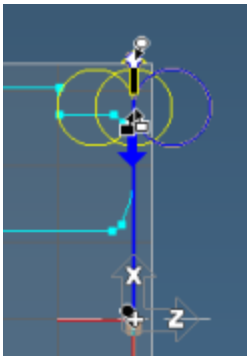
This creates a tile in the list that displays the tool number and Tool Group information.



1. Tool Group
2. Tool Group Position
3. Orientation Spindle

Note that the process states that it will be cutting on the +X side of the part.

4. Select the line you created for the contour.



5. Set the machining marker as shown.

   Drag the Start marker to a point above the stock and the End marker to just below the middle of the stock as shown in the image.

6. Create the toolpath to face the part.

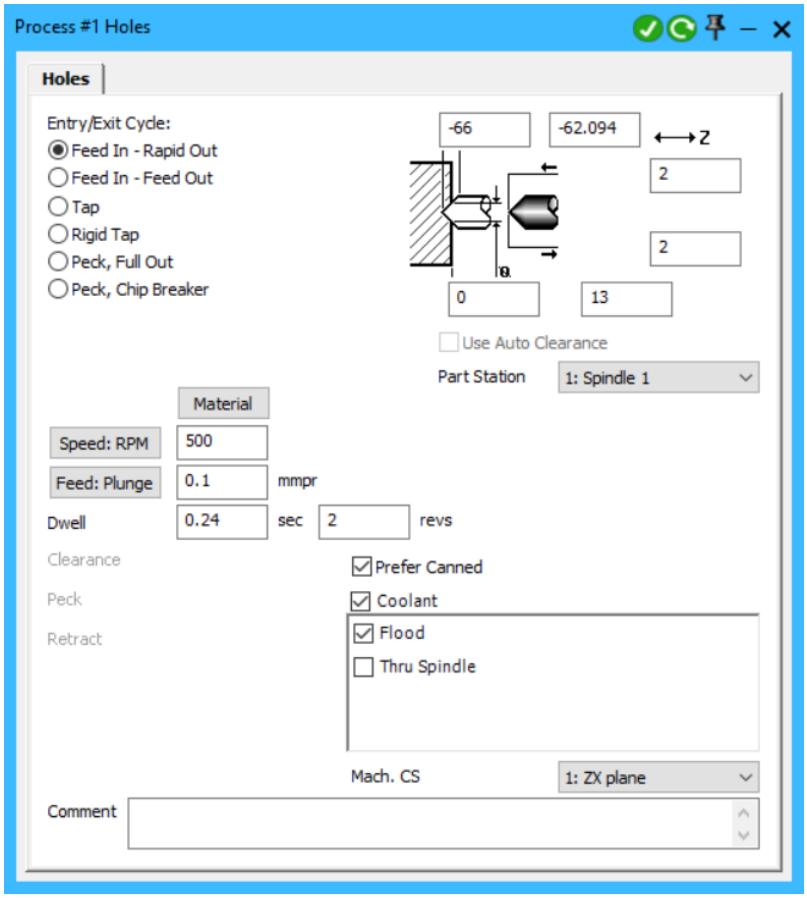
   We will now create a drilling operation to remove material from the center of the part. After the drilling operation we will create ID roughing and contouring operations to finish the ID.

7. Deselect the operation.

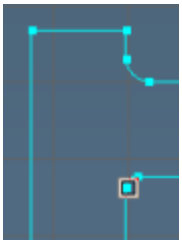
8. Clear the process list.

ID

1. Create a Lathe Drilling Process with the 13mm Drill (Tool #2) as shown.



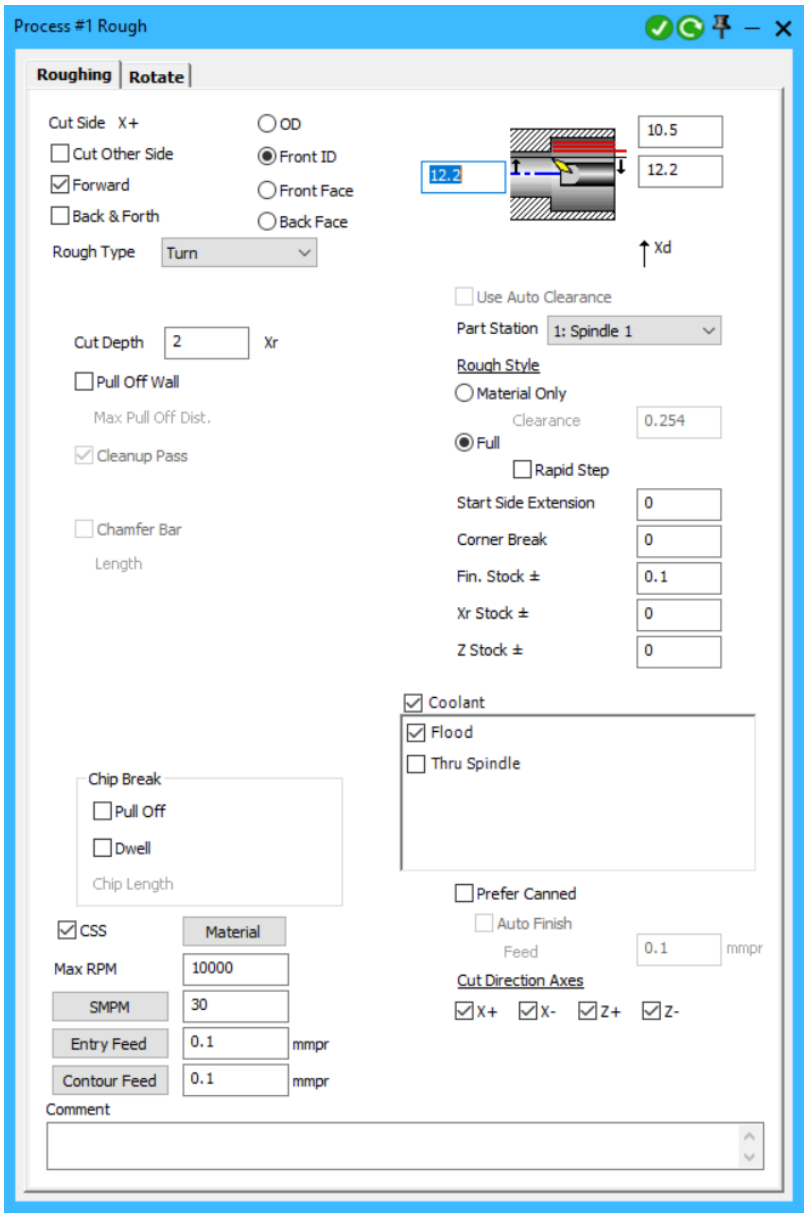
The Tip Z value of -66mm was determined by interrogating the Z depth of the ID.



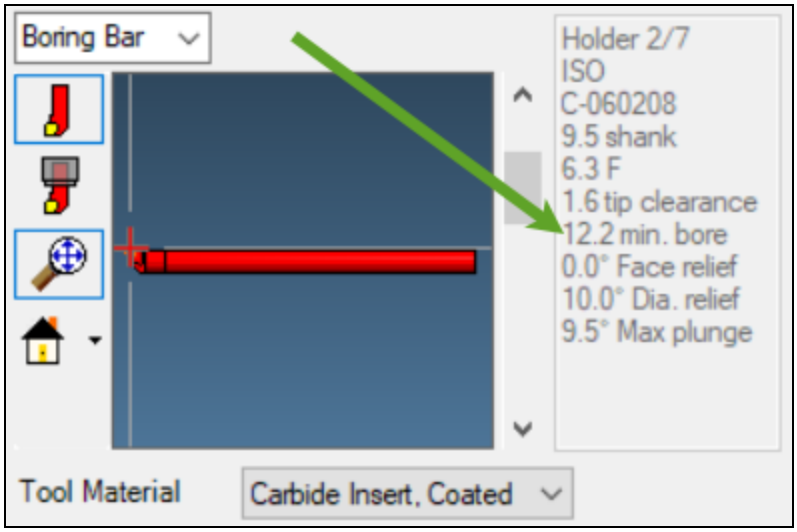
2. Create the drilling operation.

   We will now create a Rough and Contour combination process with the same tool.

3. For Process #1 create a Lathe Rough Process with the 0.8” 80° Insert (Tool #3) as shown.

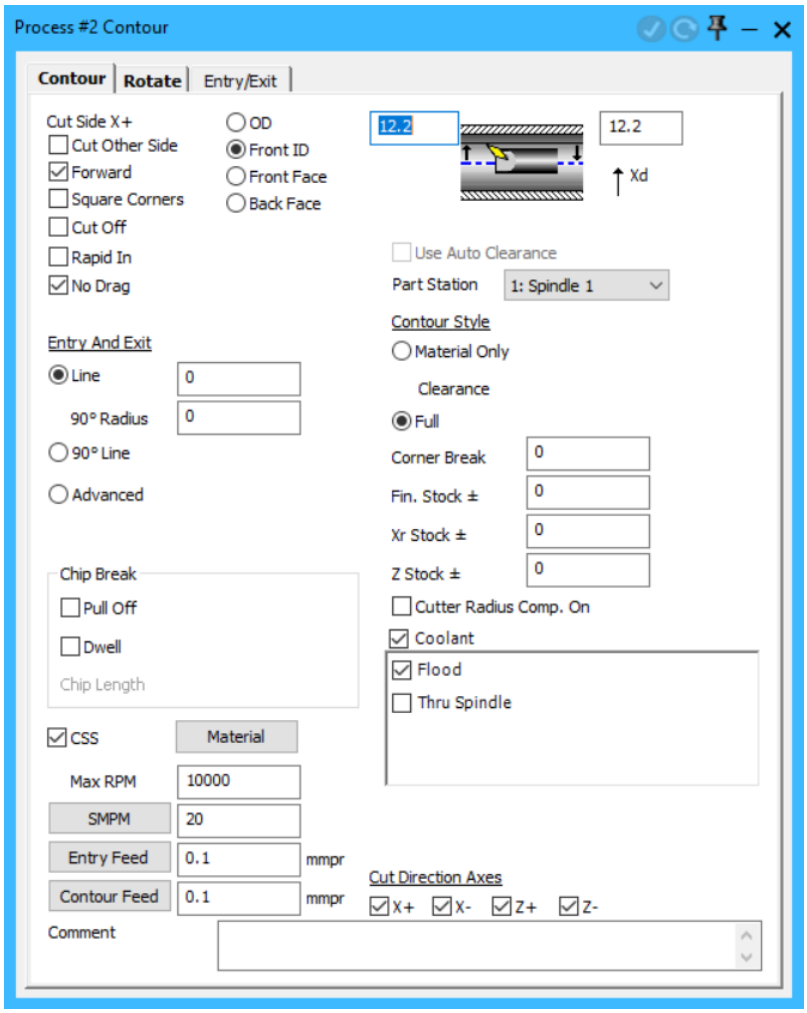


We use an Entry and Exit Clearance value of 12.2mm because that is the Minimum Bore clearance required for the boring bar we specified in the tool definition.



The value of 10.5mm is determined by interrogating the same point from the previous process. This depth ensures we will clear out all of the material at the back of this hole.

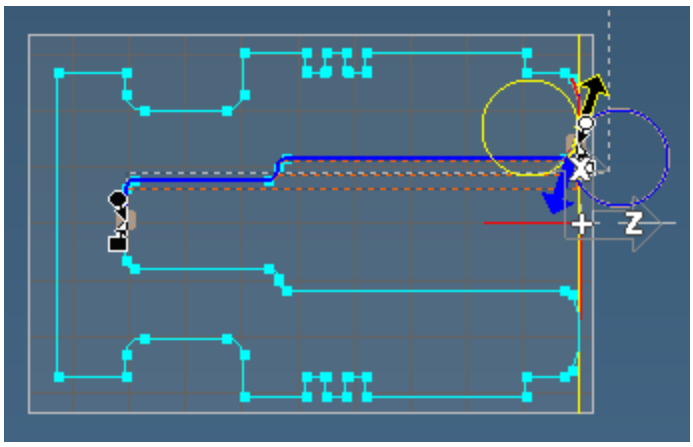
4. For Process #2 create a Lathe Contour Process again with the 0.8" 80° Insert (Tool #3) as shown.



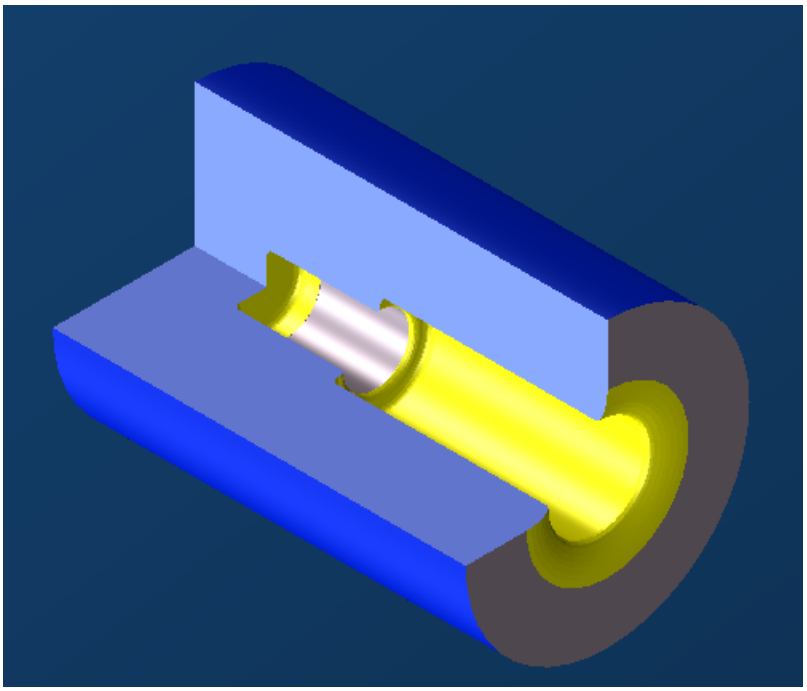
We will now set the machining markers for the processes. The operation will extend from the arc at the front of the part to the back of the ID.

5. Select the arc and set the machining markers as shown.

   The tool cuts on the outside of the shape, heading towards the inside of the part.



6. Create the Rough and Contour operations.
7. Render the operations.

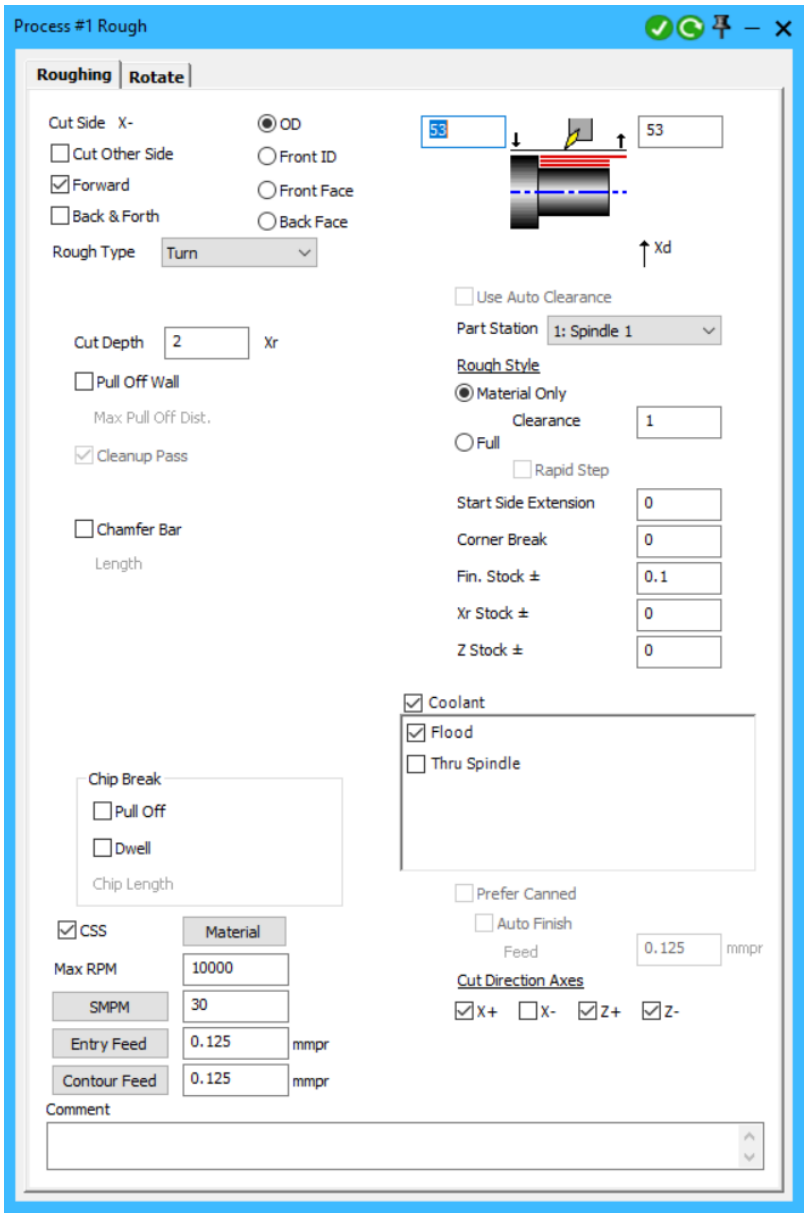


As you can see, the basics of Multi-Task Machining are the same as a standard lathe. We will now create roughing and plunge roughing operations on the OD of the part from the -X side.

8. Deselect the operations and clear any existing Process list items.

OD

1. Create a Lathe Rough Process with the 0.4” 80° Insert (Tool #6) as shown.



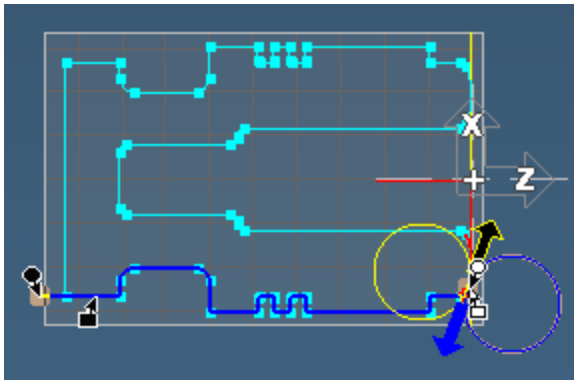
Use tool #5 if you do not have Mill/Turn.

Note that the process is set to cut from the -X side of the part. This is because the tool we are using is designated as cutting from the -X side. This value can be over-ridden, if desired, by clicking on the Cut Other Side checkbox.

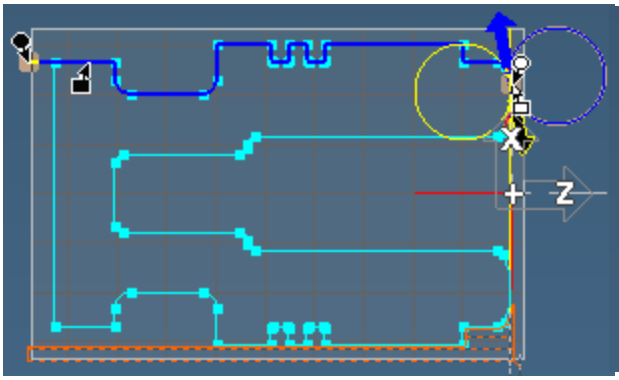
Other than the information that the process is cutting from the bottom of the part, the information in this dialog is set up no differently than any other process dialog. We now need to set the machining markers for this operation.

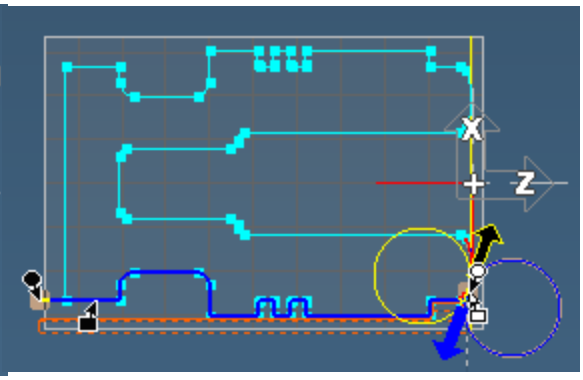
2. Set the markers as shown.

   Select the arc on the -X side of the front of the part. Be sure that the end point marker goes all the way to the back of the stock.



3. Create the toolpath.





We could have very easily set the machining markers on the +X side as shown above left and the same toolpath would be generated. It may be more convenient to have the markers in the same place as the toolpath. There is no real difference, as shown, but if there were no geometry on the -X- side of the part the toolpath could not be visually verified.

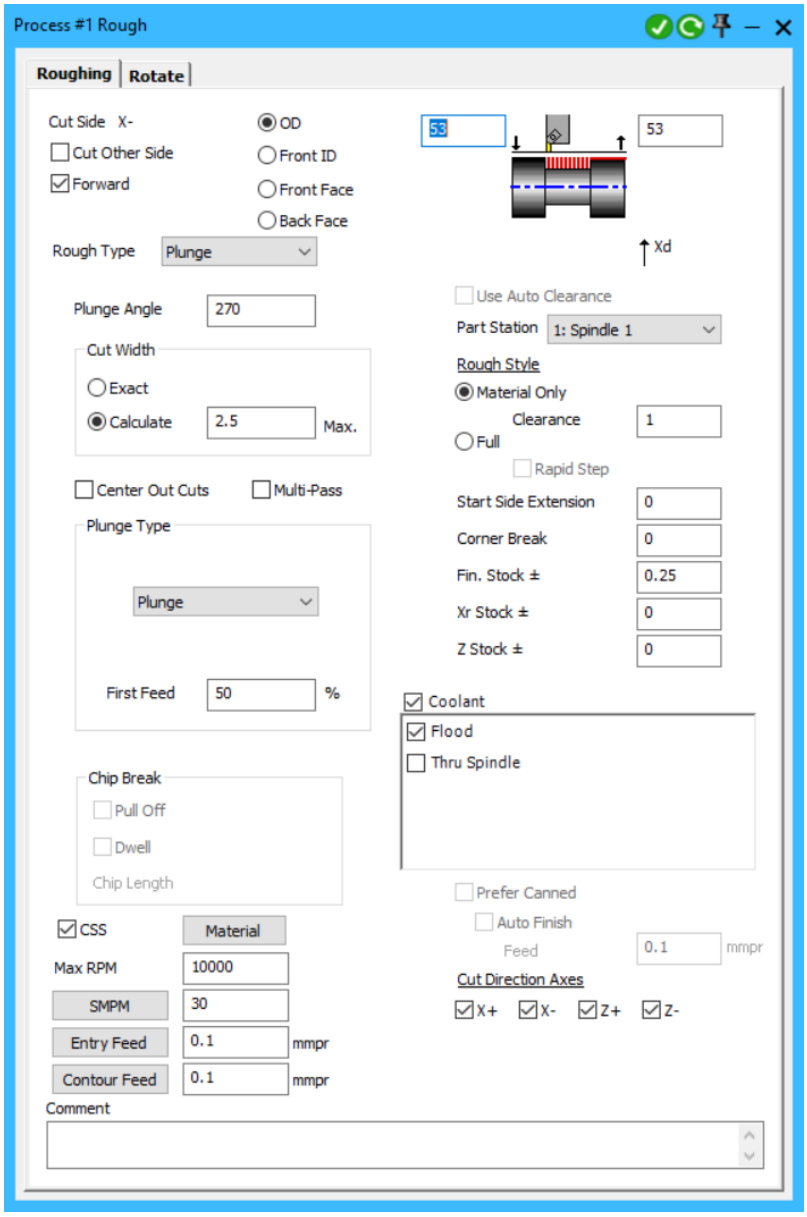
The Cut direction Axes restriction in the process dialog is not supposed to make any -X moves along the part yet it would seem that the toolpath does this. In fact, the toolpath does not make any -X moves along the shape.

All of the toolpath is first generated from the +X side of the part and then “flipped” to the -X side for graphical accuracy.

If you consider the axes as being arbitrary names, the turrets on the +X and -X side each see the part from the same point of view. The turret on the bottom of the part is potentially performing the same moves that the turret on the top of the part would make, except the bottom turret is mirrored. Therefore, it does not matter where the toolpath is first generated. It is easier to always visualize the toolpath as first cutting on the top of the part and then being moved, rather than always trying to remember exactly which direction the tool is coming from and thinking around that. If you watch the toolpath generation closely, you may even see the toolpath being generated on the +X side then getting flipped to the -X.

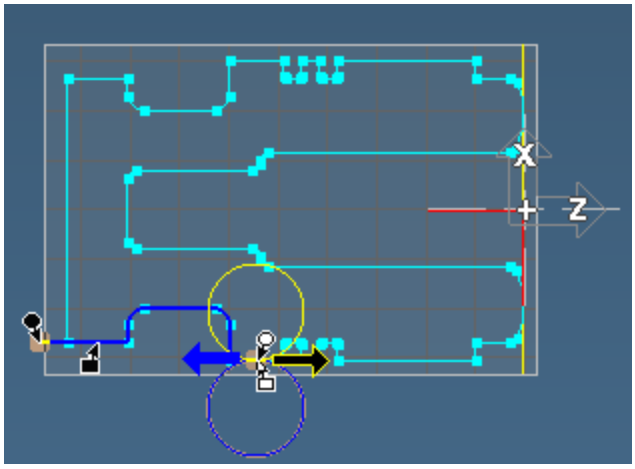
We will now continue generating toolpath on the -X side of the part by clearing out the recess not cut in the previous operation.

4. Create a Lathe Roughing Process with the 0.19” Groove tool (Tool #7) as shown.



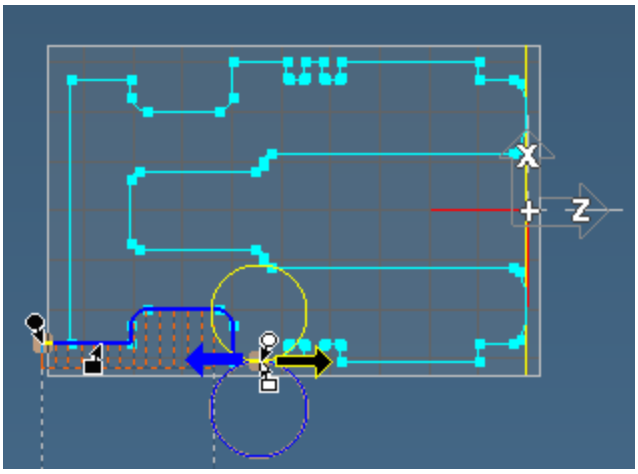
Use tool #6 if you do not have Mill/Turn.

5. Set the machining markers as shown.



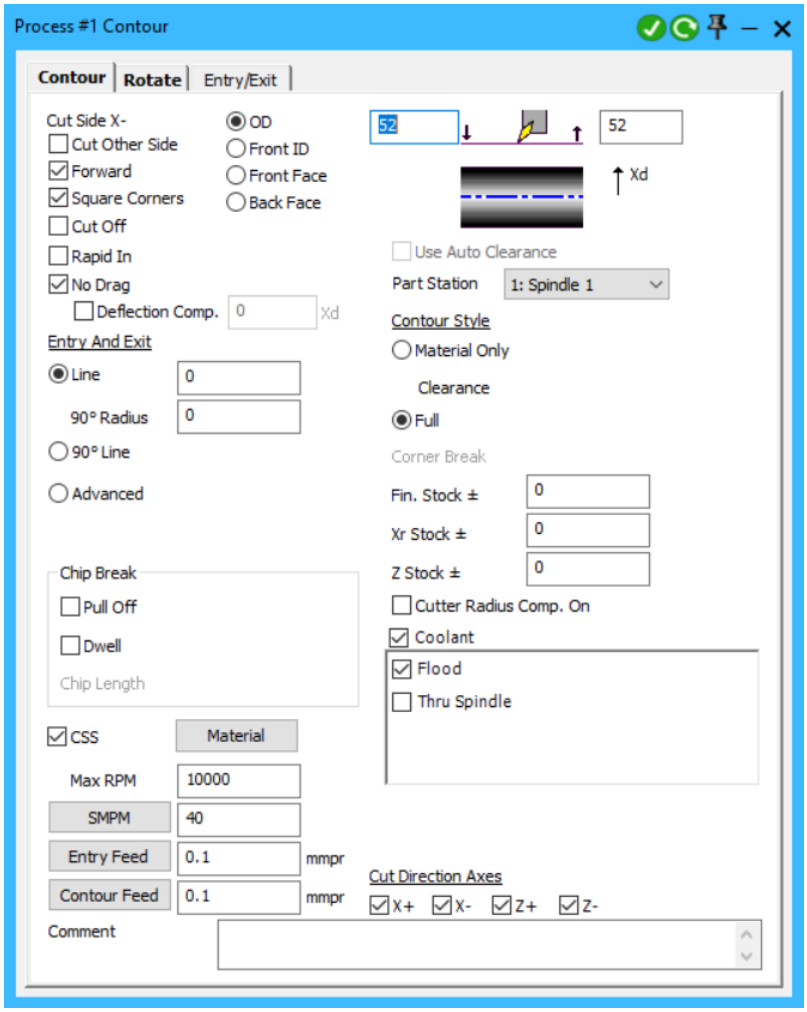
Be sure that the end point marker goes all the way to the back of the stock.

6. Create the toolpath.



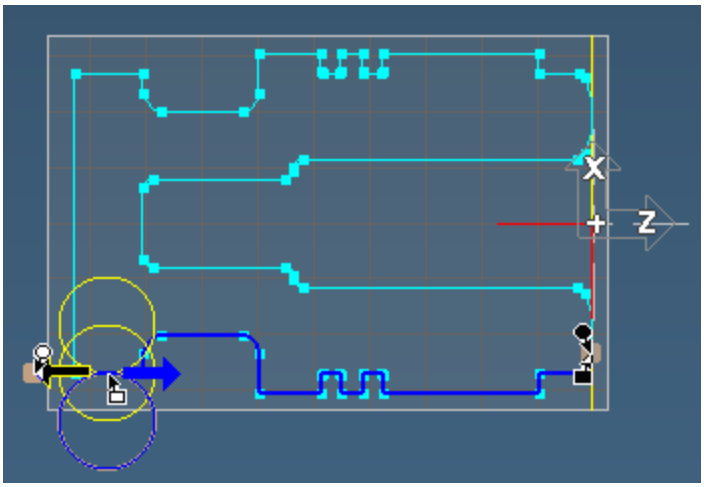
The next operation will use a groove tool to cut the grooves and finish the OD. This process will cut from the back of the part to the front.

7. Create a Contour Process with Groove Tool (tool #8) as shown.



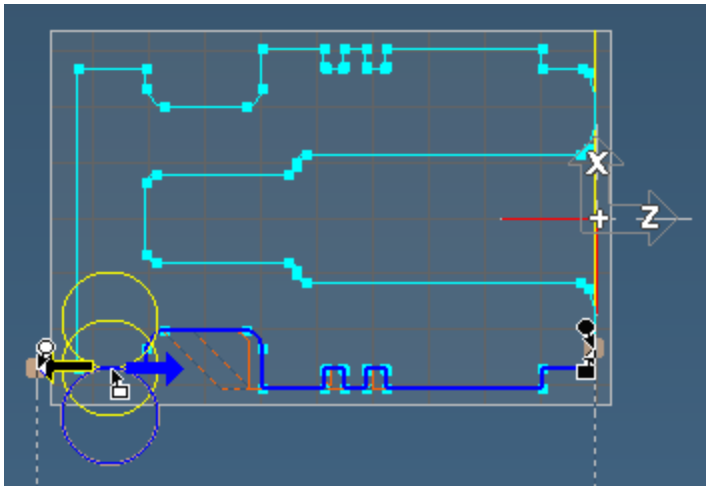
Use tool #6 if you do not have Mill/Turn.

8. Set the machining markers as shown.

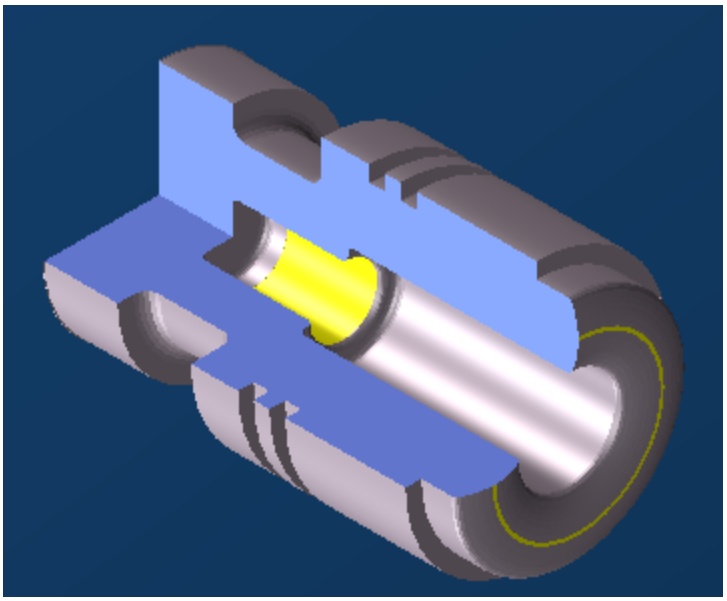


Be sure that the end point marker is in the middle of the arc at the front of the part.

9. Click Do It or (Alt+.) when you are finished, to create the toolpath.



10. Render the operations.

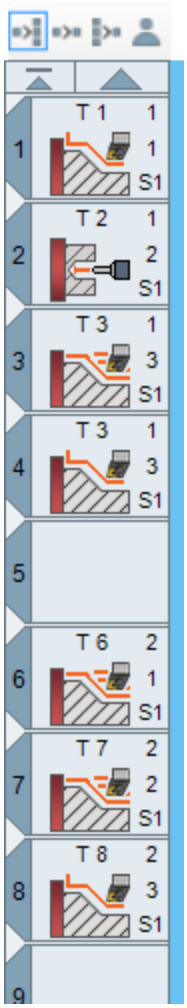


Grouping Ops by Flow

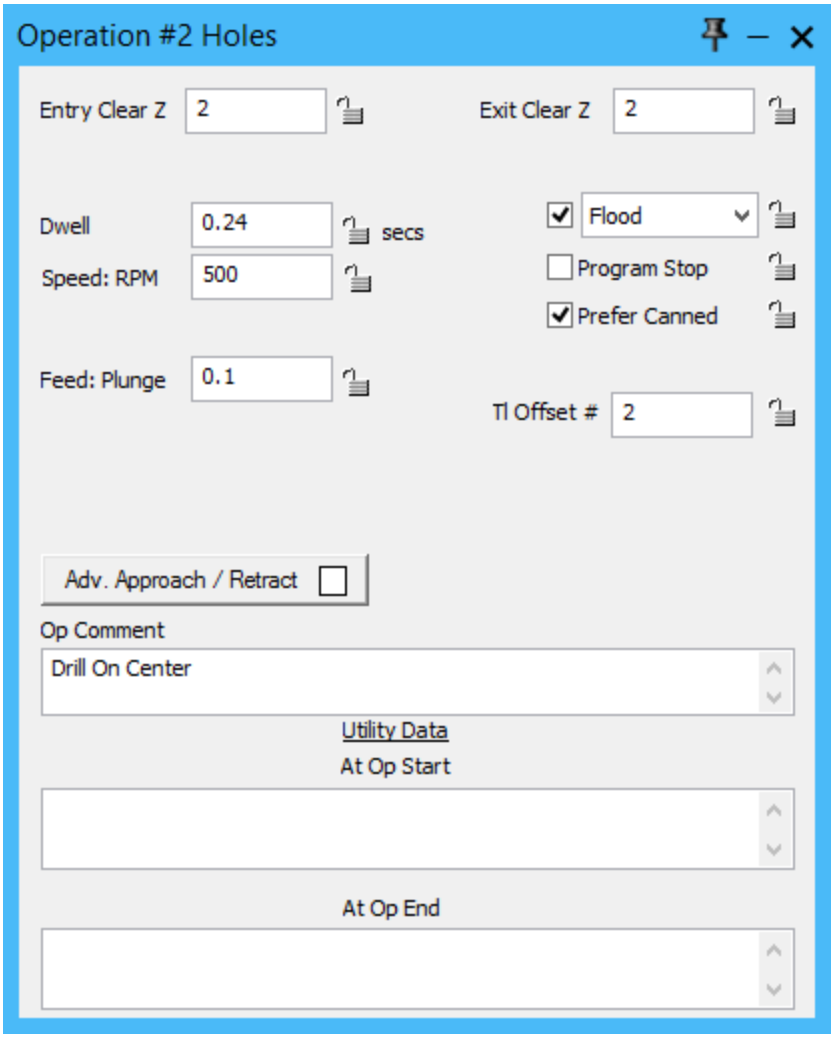
We are now going to organize this part better.

1. Click the Sort Operations option from the operations right-click menu.

The operations in this list are now divided and sorted by flows.

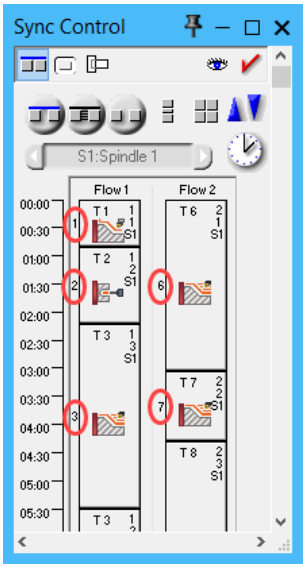


The Operation Data dialog, also on the operation tile `Right-click` menu, allows you to enter information about the operation that will appear in the post, set overrides to values and lock values so other users cannot override your settings. Additionally, syncs are shown at the bottom in the Utility Data section. We recommended that you do not change sync data the system creates here.


Operation #2 Holes
Entry Clear Z
2
Exit Clear Z
2
Dwell
0.24
secs
Speed: RPM
500
Flood
Program Stop
Prefer Canned
Feed: Plunge
0.1
Tl Offset #
2
Adv. Approach / Retract
Op Comment
Drill On Center
Utility Data
At Op Start
At Op End

Control of Flows

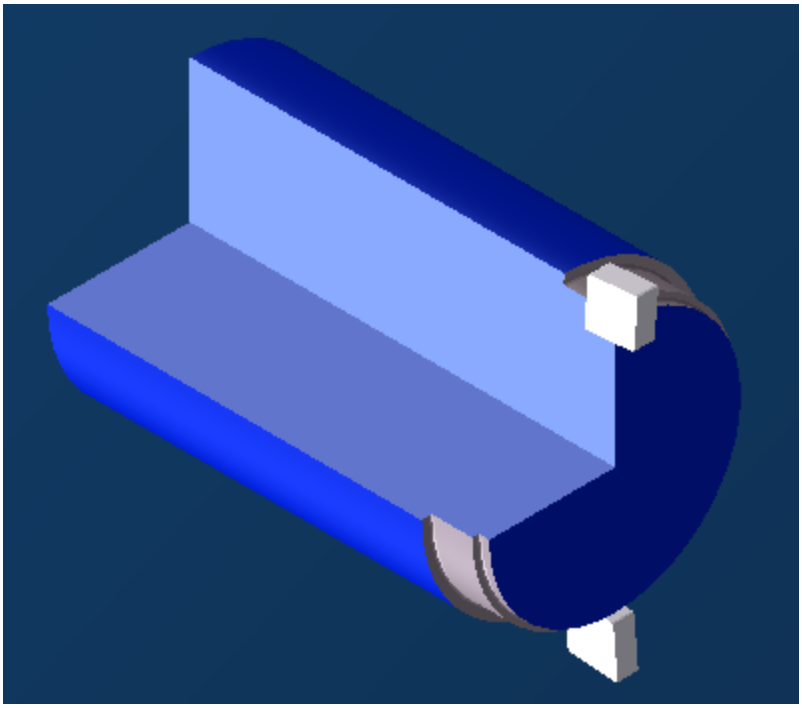
Syncing Operations



1. Open the Sync Control dialog. 

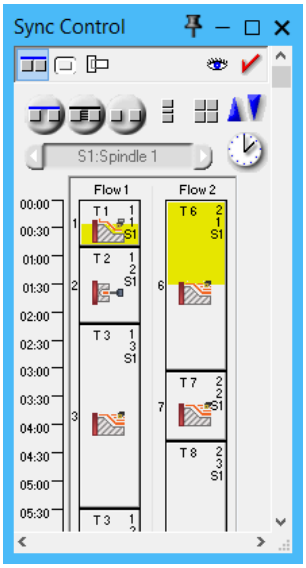
   This Sync Control dialog displays all of the operations in the part. The operations are sorted into flow columns. Each operation tiles length, is an accurate representation of the time needed for the operation. This is where we will set syncs to control the operations in each flow. Note the operation numbers highlighted in the screenshot.

If we look at the rendering, the start of the two flows is simultaneous. Even though there should not be any interference among the turrets, we will put a sync in to make the OD roughing operation wait until the face contour operation is complete.



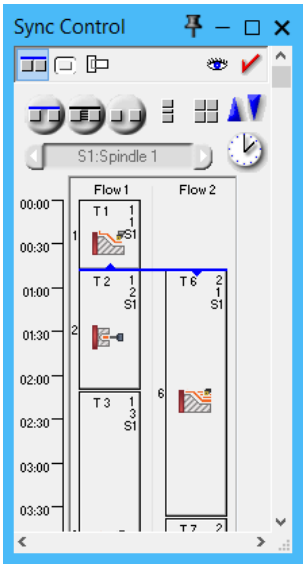
2. Ensure the sync mode is on. 

While in Sync mode each operation tile has two selections, the start and end of the operation. To select the operation start, select the top of the tile. To select the operation end, select the bottom of the tile.



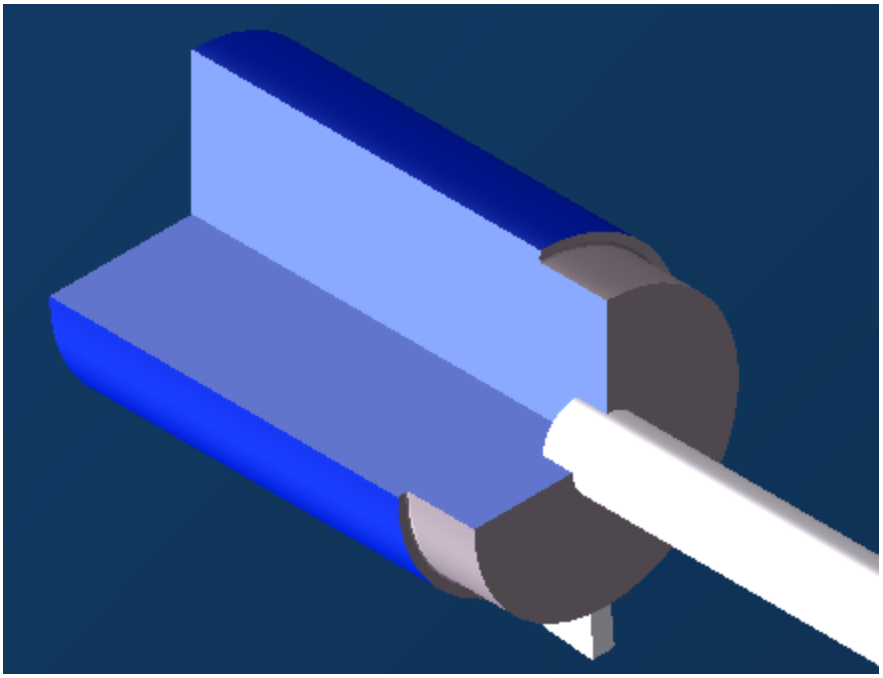
3. Select the bottom (Op end) of Op1 in Flow 1.
4. Select the top (Op start) of Op6 in Flow 2.

5. Click the Operation Sync button. 



Both flows are updated and Flow 2 will now start once Operation 1 has finished. The run-time of the part has also been updated. Setting syncs is that easy.

You may notice that the run-time of Op 1 is different after the sync is set. Before the sync was set two flows were cutting on the spindle at the same time – one flow was performing a facing op and the other flow was performing an OD Rough, each using the Constant Surface Speed setting. Since both operations cannot have control over the spindle speed at the same time, the system gave Flow 2 the spindle control. By setting the sync, the run-time for Op 1 is recalculated.



Operation 1 now completed, Operations 2 and 6 begin.

6. Save the part file.

   If you do not have the milling option, you have completed this exercise and may move on to the next tutorial, see "Dual Spindle Part" on page 66. If you do have the milling option, we will now mill and OD drill this part. First we need to sort the tools to get the mill tools into order.

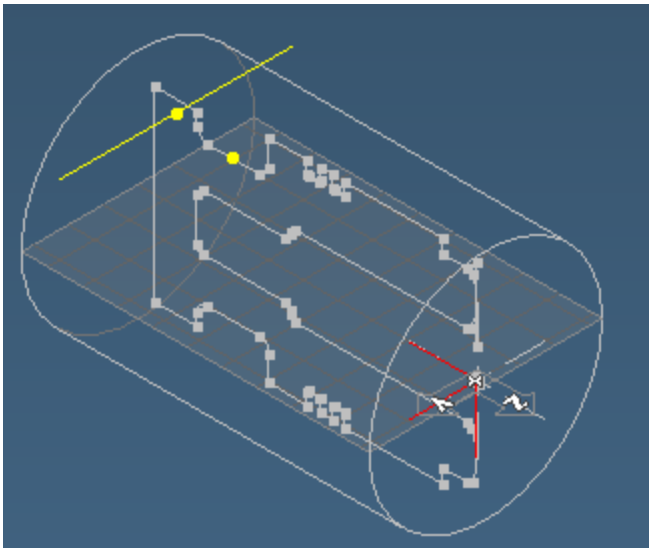
7. Close the sync dialog.

   If the dialog is left open some warnings may appear as we add more operations. We will fix these sync errors at the end.

Milling Ops

Hex Head

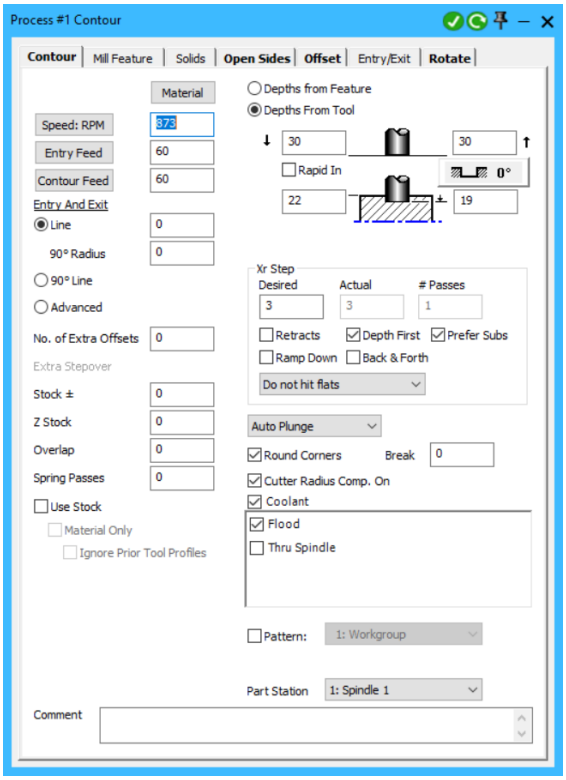
1. Switch to Workgroup 2: Milling and CS4: YZ plane.

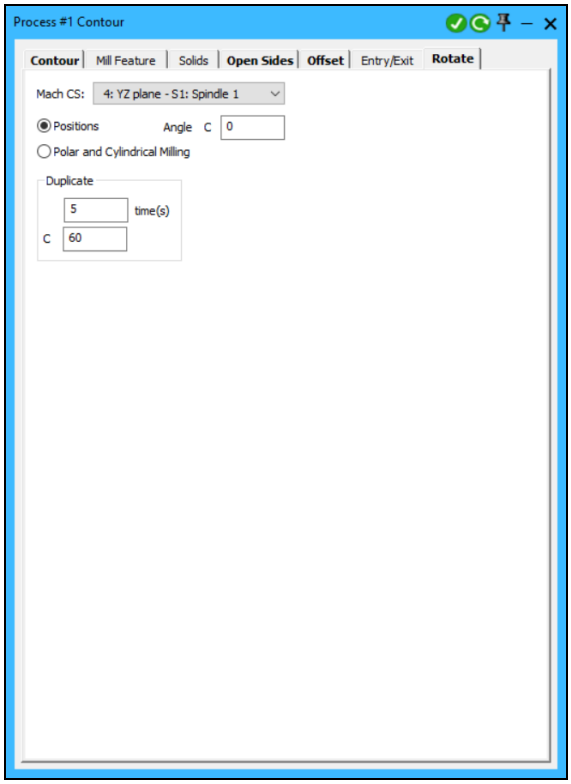


All of the geometry for the mill components of this part are contained in Workgroup 2 and belong to CS4: YZ plane. This includes a point at the middle of the line at the bottom of the recess of the part and a point and line that will be used to mill a hexagon near the back of the part.

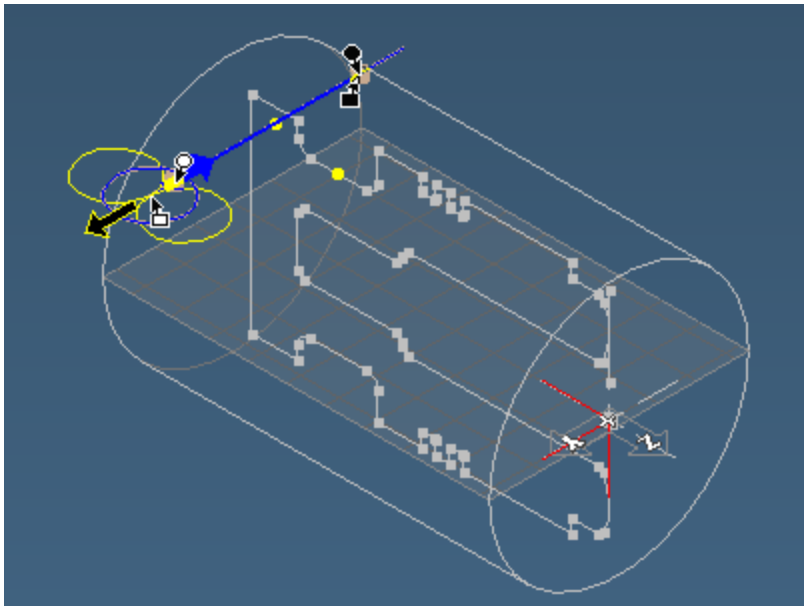
The first milling operation we will create will be to mill a hexagon by using six rotary positioned contour cuts.

2. Create a Milling Contour Process with the 13mm Rough Endmill (Tool #4) as shown.





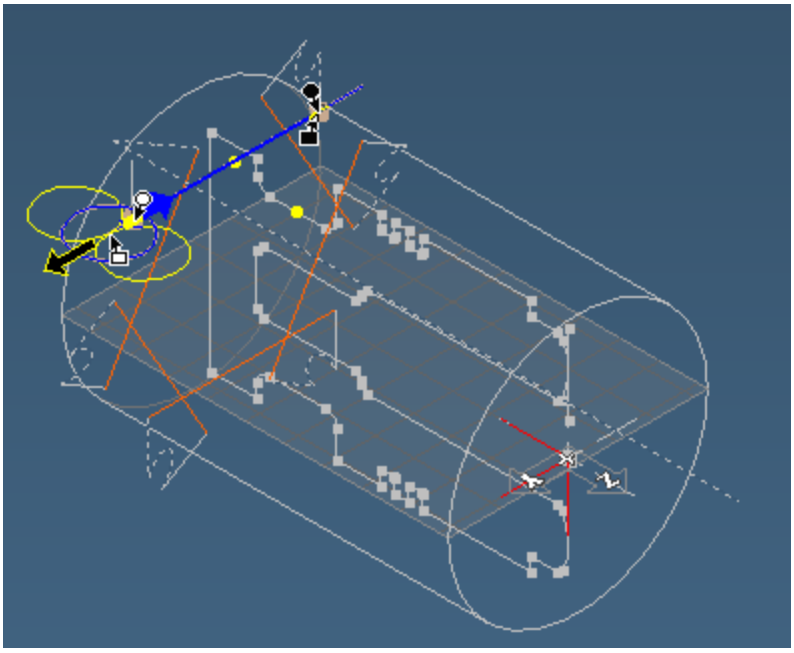
3. Select the horizontal line.



4. Set the markers as shown.

This will be a centerline cut from the +Y to the Y- side of the part. The tool should cut on-center

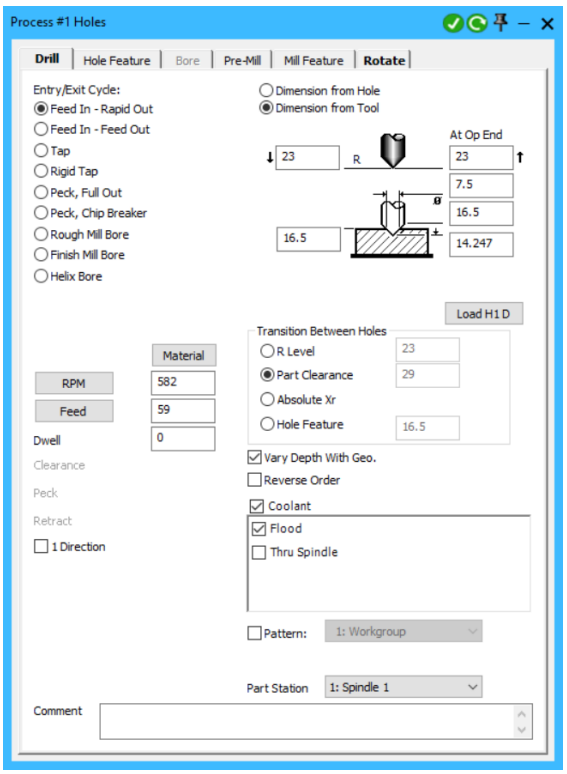
5. Create the toolpath.

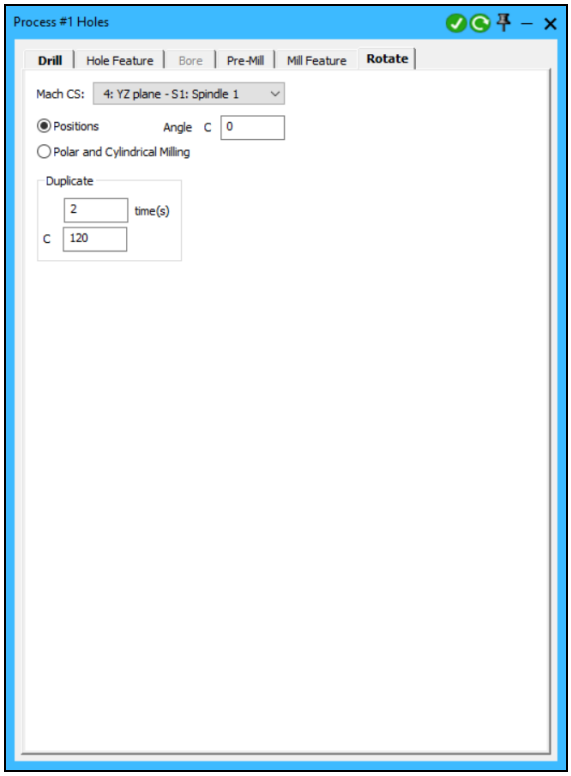


Be sure you are not cutting from the Y- side. To cut down on the amount of movement by the tool, experiment with the placement of the machining markers to optimize the amount of feeding required.

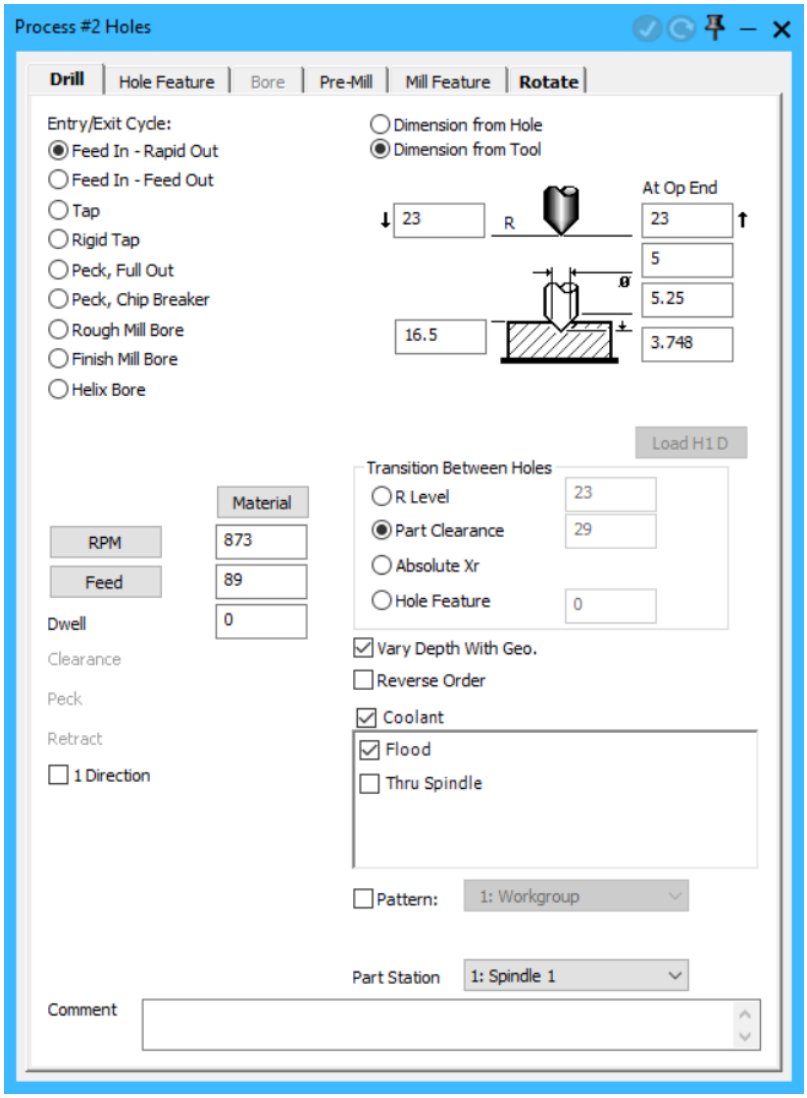
The next set of operations will spot and drill three holes in the recess of the part.

6. For Process #1 create a Mill Hole process with the 7.5mm Spot Drill (Tool #9) as shown.



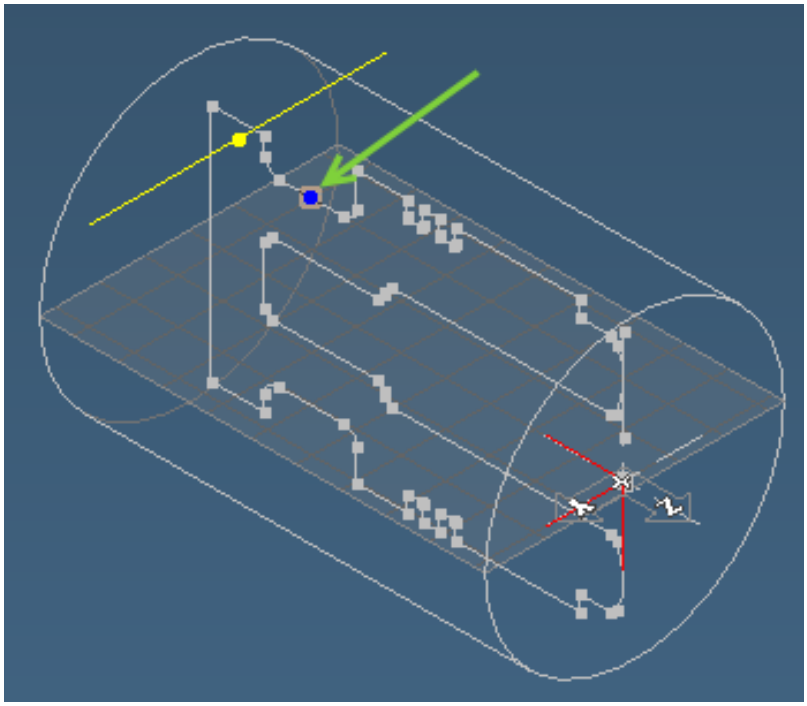


7. For Process #2 create a Mill Hole process with the 5mm Drill (Tool #10) as shown.

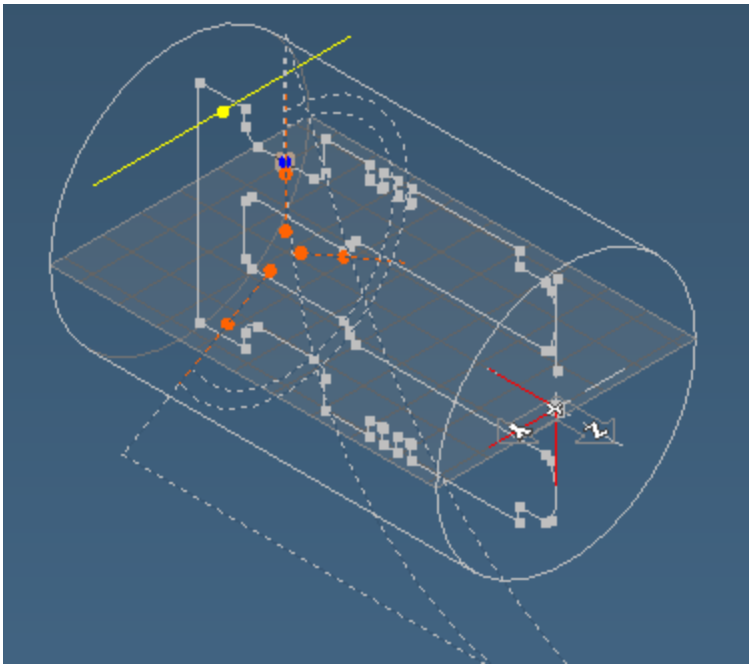


The Full Diameter Z value is determined by interrogating the same point used for the ID drill depth. There is no need to enter rotary positioning information for this process. Linked processes have the same positioning information by default.

8. Select the point shown.



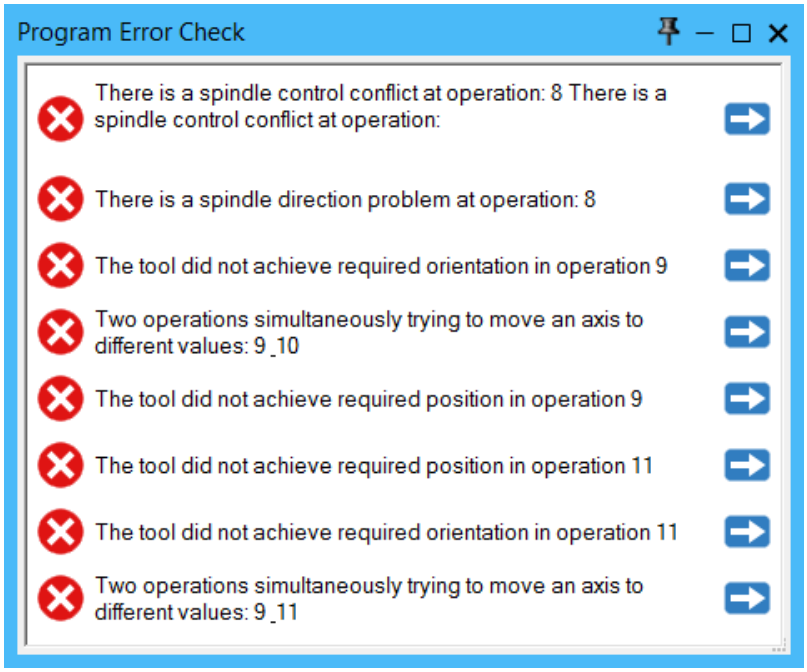
9. Create the toolpath.



Even though these drills are approaching from the bottom of the part, we can still use geometry defined at the top of the part. The post will simply handle the position of the spindle when the program is run.

One thing that will not be handled by the post or the machine is two milling operations occurring on the same spindle at the same time. We need to sync the contouring and drilling operations. Additionally, we have a milling operation happening concurrently with a turning operation.

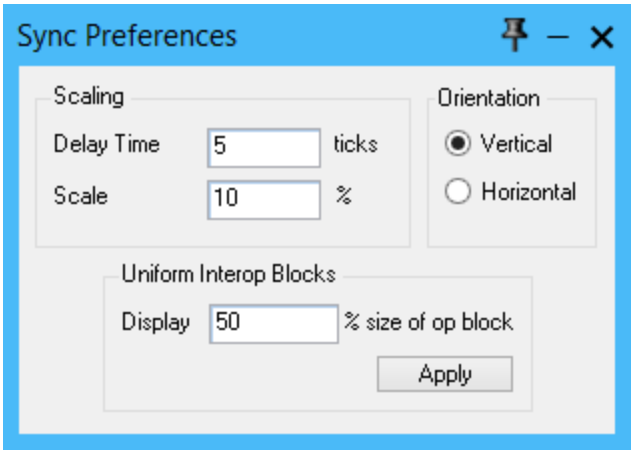
10. Open the Sync Control dialog. 



When the Sync Control dialog is opened we get warnings that there are conflicts with spindle control. We will set a sync and fix the problem. First we will need to adjust the scaling control in the sync dialog so that we have more precise control over expanding/contracting the tile display.

11. Click the Eye symbol on the Sync Control Dialog. This opens the Sync Preferences dialog. 

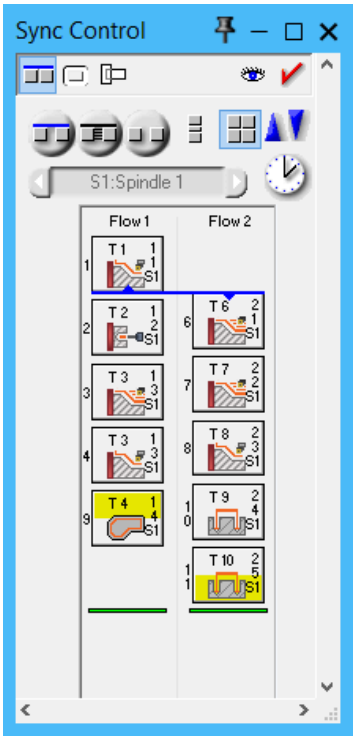
12. Change the scale to 10%.



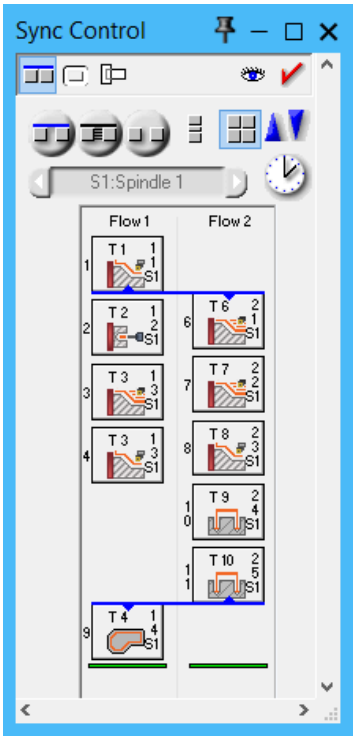
13. Now change the tiles to Uniform view and use the down arrow to fit the operations into the sync control window. 

You can drag the edge of the dialog to expand the Sync control dialog if necessary. It is also possible to view the dialog horizontally - for this, select the Horizontal button in the Sync Preferences dialog.

14. Select the start of operation #9 and the end of operation #11.



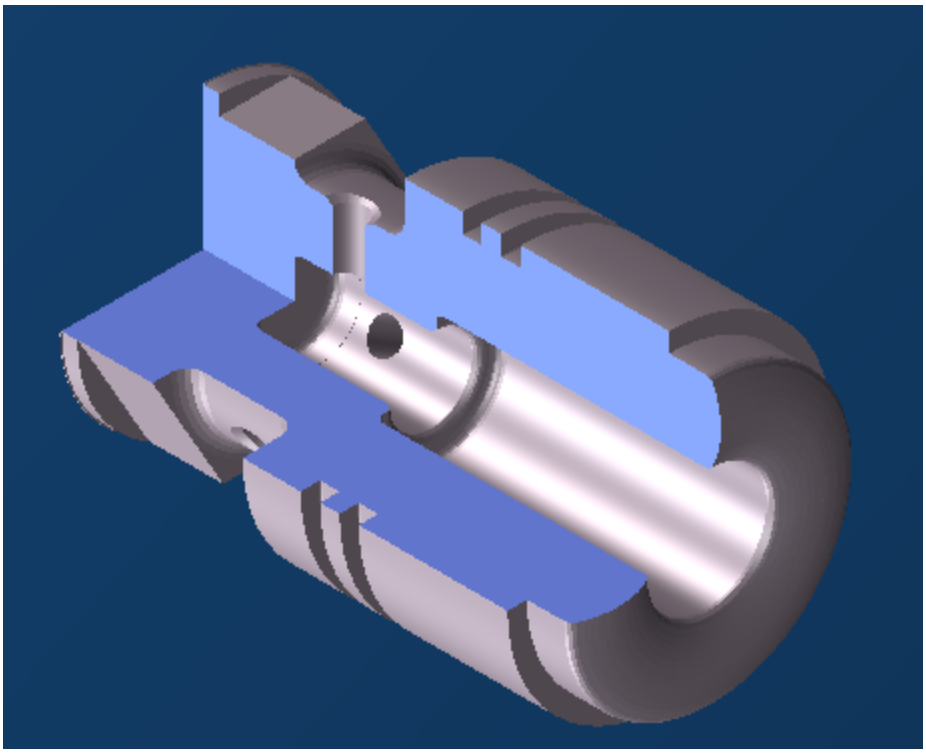
15. `Click` the Operation Sync button.



16. `Click` the Uniform view button to switch it off, then click the Recalculate button.

The operations have now been synced. Click the Error check button. It will now display All Checks Passed.

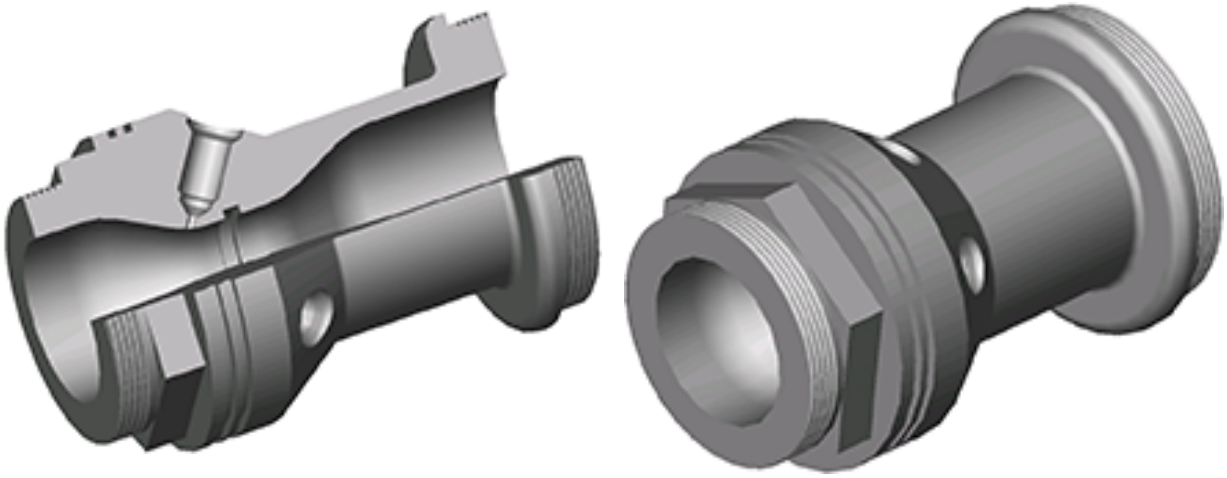
17. Render the operations.



18. Be sure to save the part.

Dual Spindle Part

In this exercise we will machine the Dual Spindle.vnc part created earlier in the tutorial, see "Geometry Creation and Arrangement" on page 8. This tutorial will create a part using both turning and milling operations. As with other tutorial exercises, you do not need to have the milling option to complete the exercise. In this tutorial we will have Turret 1 perform machining operations on the +X side of Spindle 1 only. Turret 2 will primarily cut on Spindle 2 except for the part cut-off or if you are performing the milling operations. Turret 2 is set up to cut on the -X side of the part. When complete, the part will look like the following images.



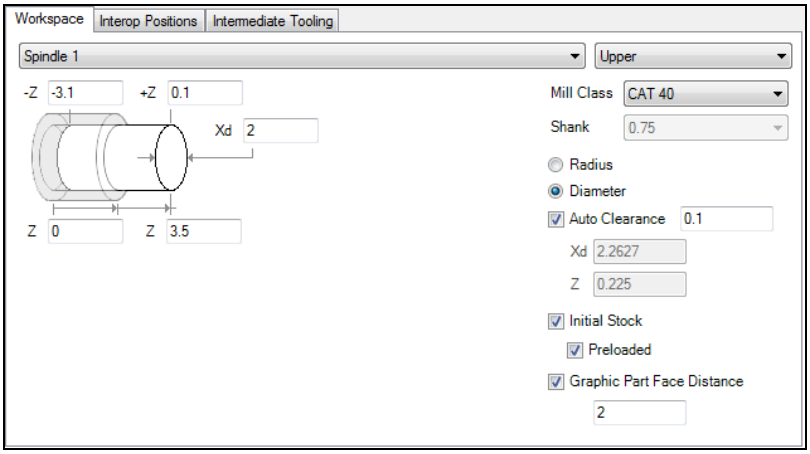
Part Setup - Tool Change

1. Open the part file Dual Spindle.vnc.

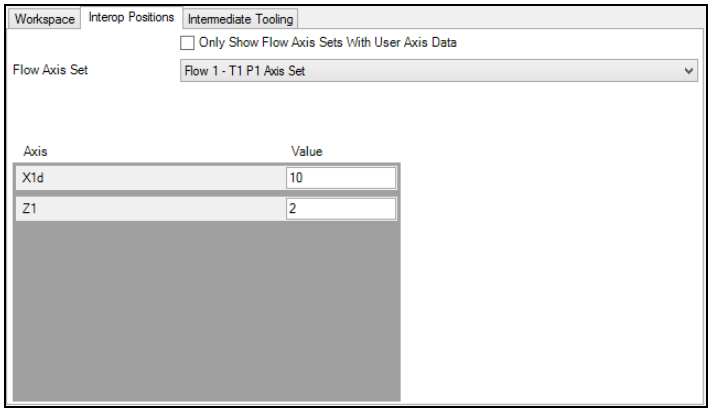
   For details on the tooling for this exercise, see “Dual Spindle Setup” on page 25. If you have not completed this exercise, please do so now.

   We first need to check our Clearance and Tool Change settings. We will be using Auto Clearance for this part. The tool change position for both spindles will be Z2" and Xd+/-10" from the origin. The Xd value will depend on whether the turret is the Upper or Lower turret.

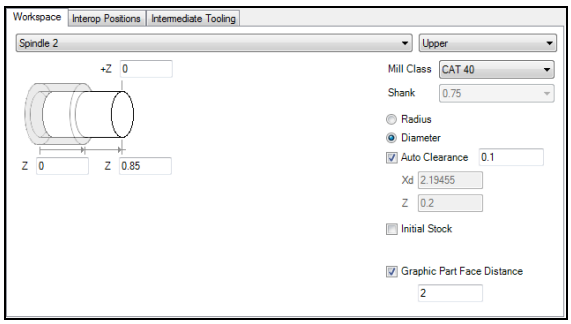
2. Enter the information in the Document dialog as shown.

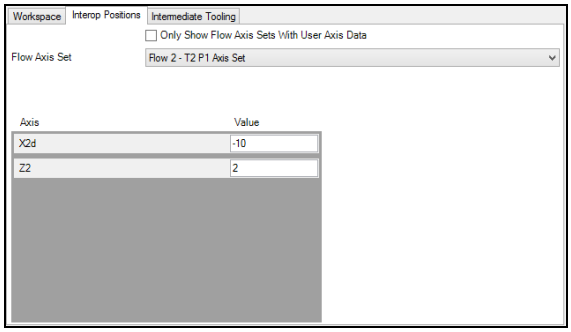


3. Switch to Lower and enter the settings shown in the Interop Positions tab.



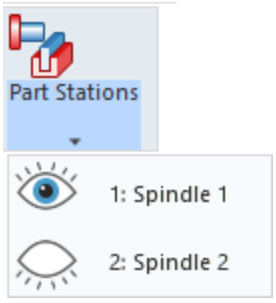
4. Switch to Spindle 2 and enter the information shown.





5. Open the Tool list and Machining palette.

6. Open the Part Stations list from the Commands Toolbar.



7. Hide Spindle 2.

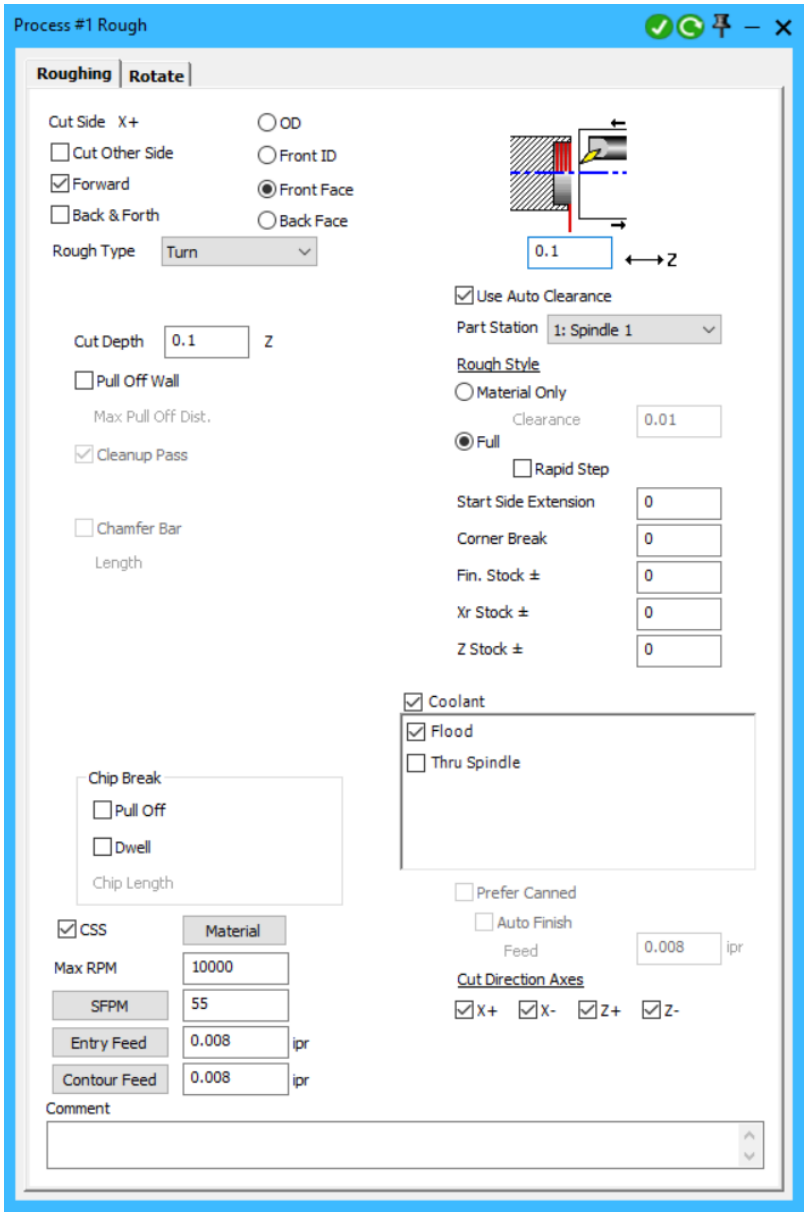
8. Unzoom (Ctrl+U).

   The main spindle is now the only spindle in the Workspace. Showing and hiding spindles will make creating parts much easier.

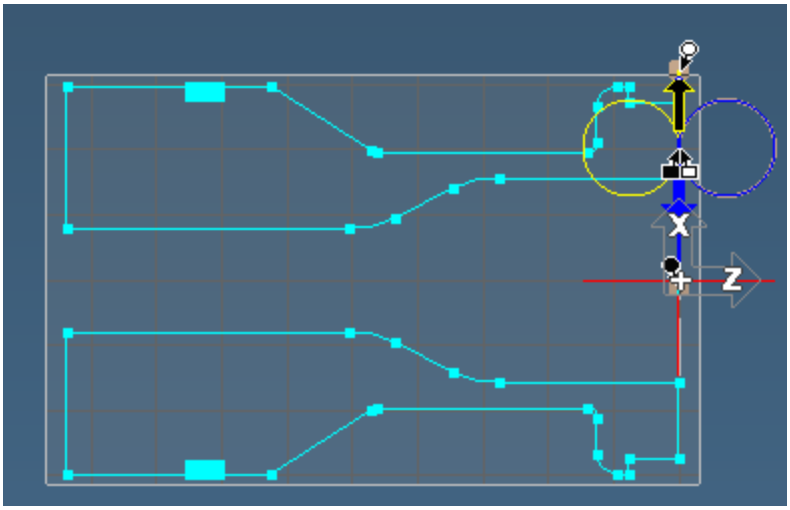
Setting up the Operations

Front Face

1. Create a Lathe Rough Process with the 1/64” 55° Insert (Tool #1) as shown.



2. Set the machining markers as shown.

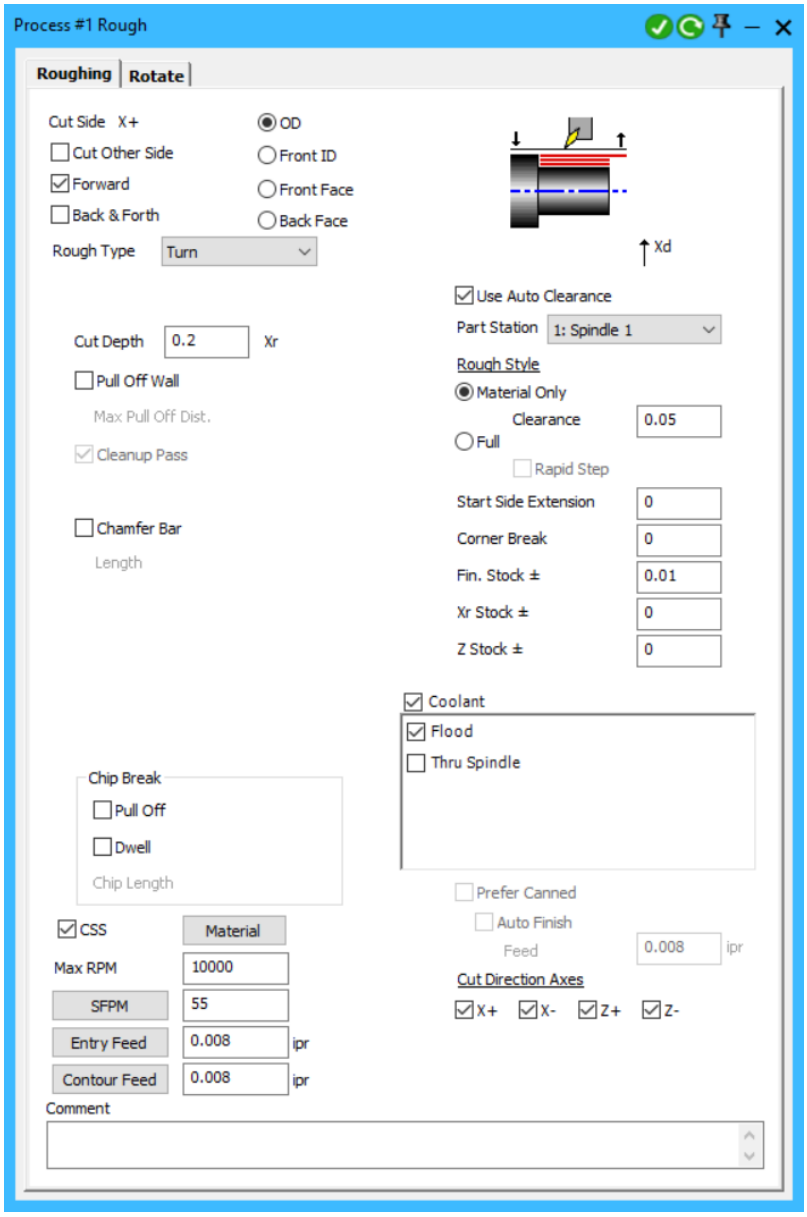


3. Create the toolpath.

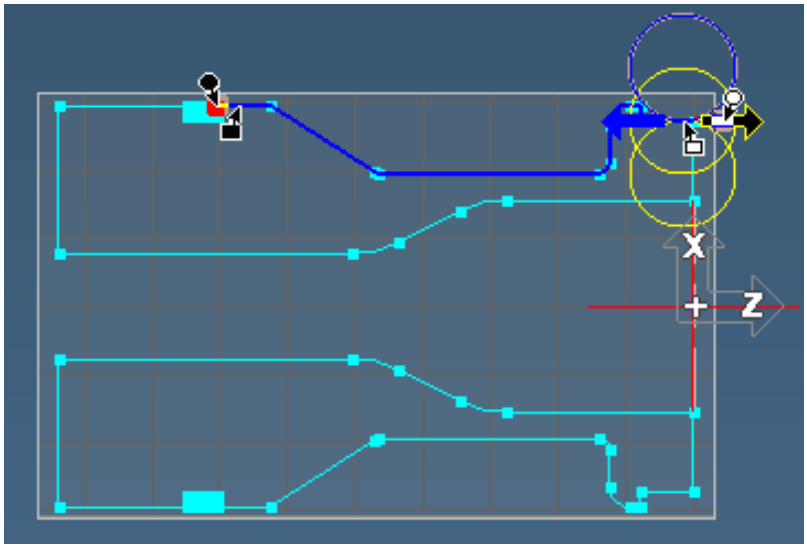
OD

We will now rough the OD of the part. To do this we will make four operations, two roughing and two contour operations.

1. Create a Lathe Rough Process with same insert (Tool #1) as shown.

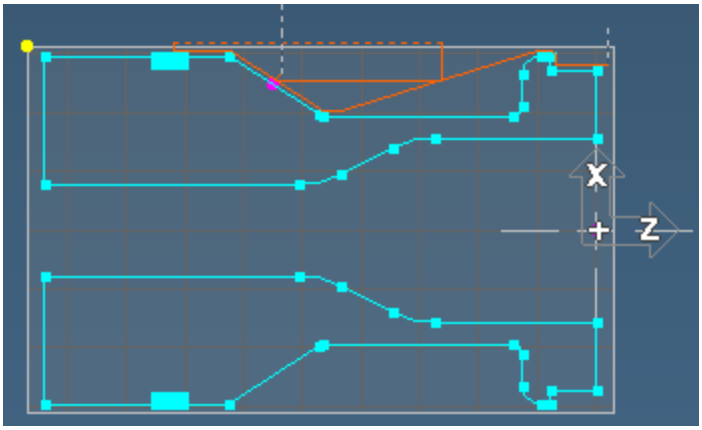


2. Set the machining markers as shown.



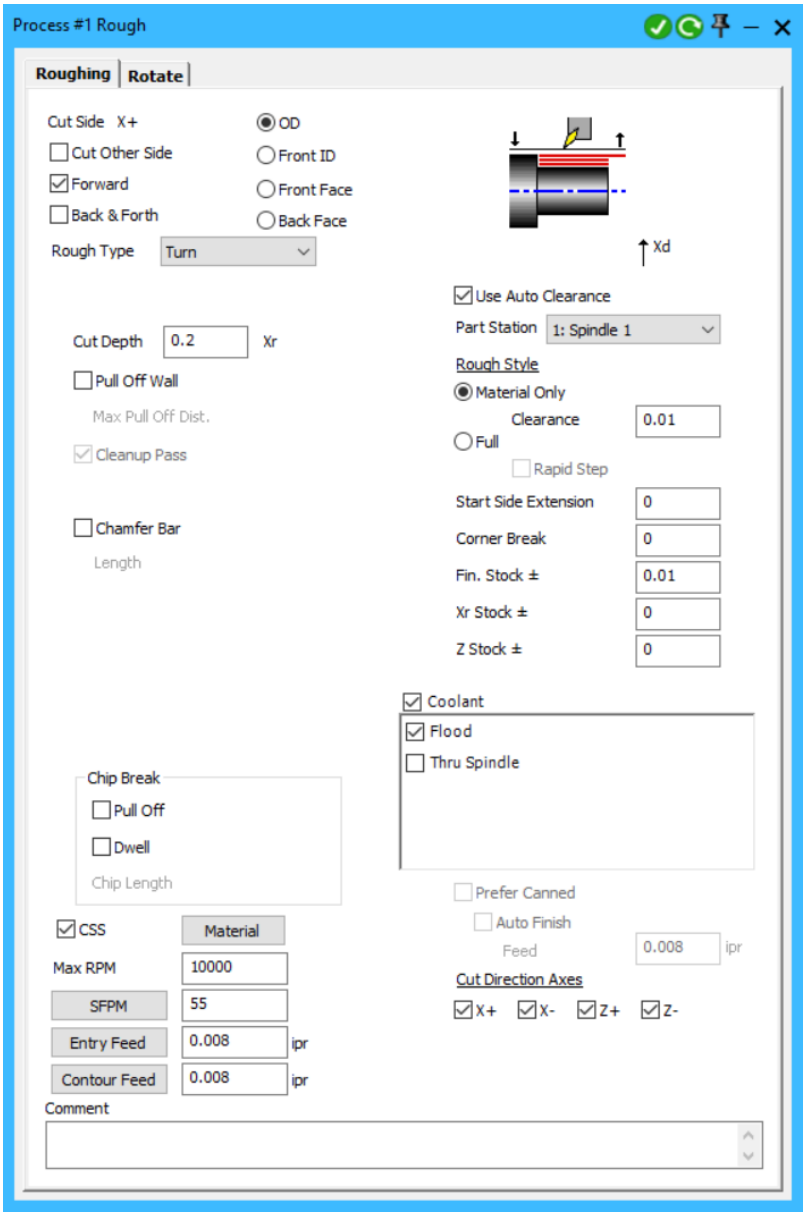
This will cut from the front to the back of the first groove.

3. Create the toolpath.

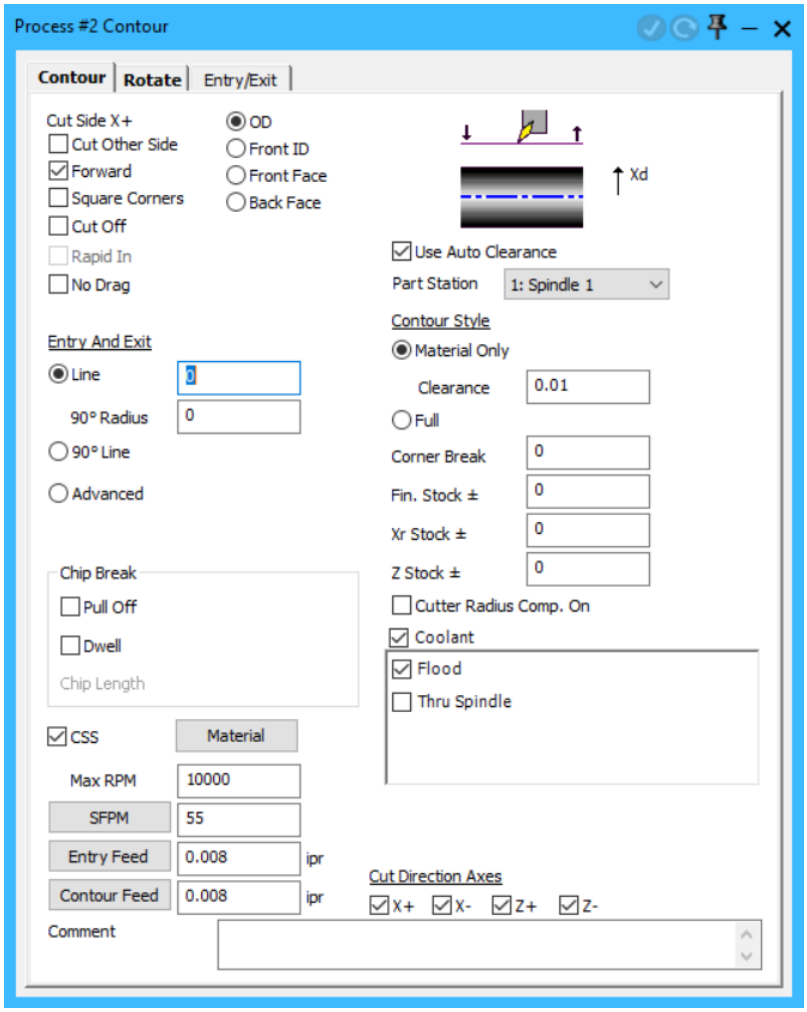


The next set of operations will continue to machine the OD of the part. We will rough and finish the back side of the front flange with tools 3 and 4 in a multi-process operation, Perhaps you recall that these tools were set to cut from the back of the part to the front.

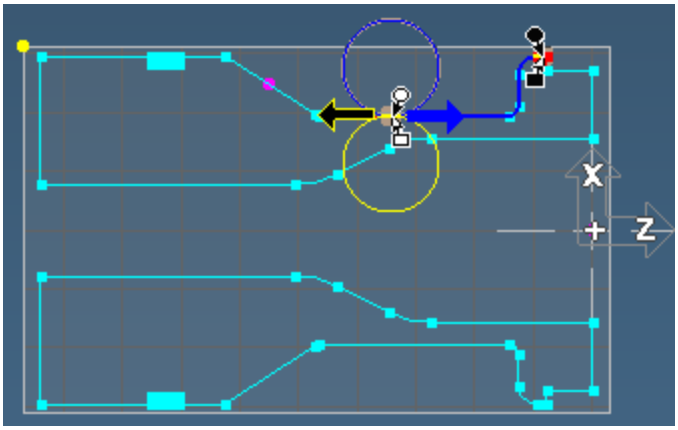
4. For Process #1 create a Lathe Rough process with the second 1/64” 55° Insert (Tool #3) as shown.



5. For Process #2 create a Lathe Contour process with the 1/64” 35° Insert (Tool #4) as shown.

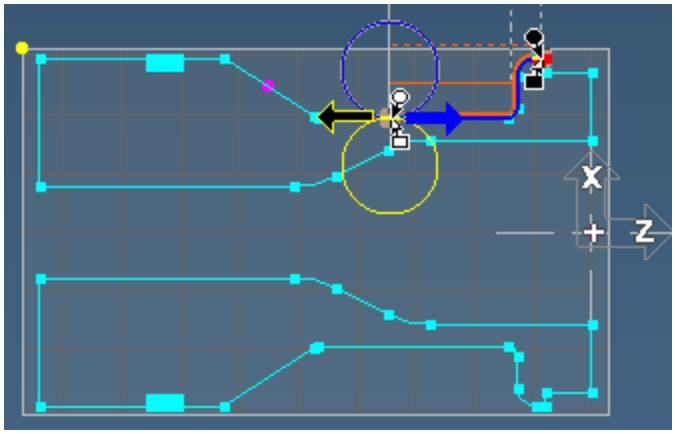


6. Set the machining markers as shown.



This will generate toolpath that will cut from the middle of the part to the front of the flange.

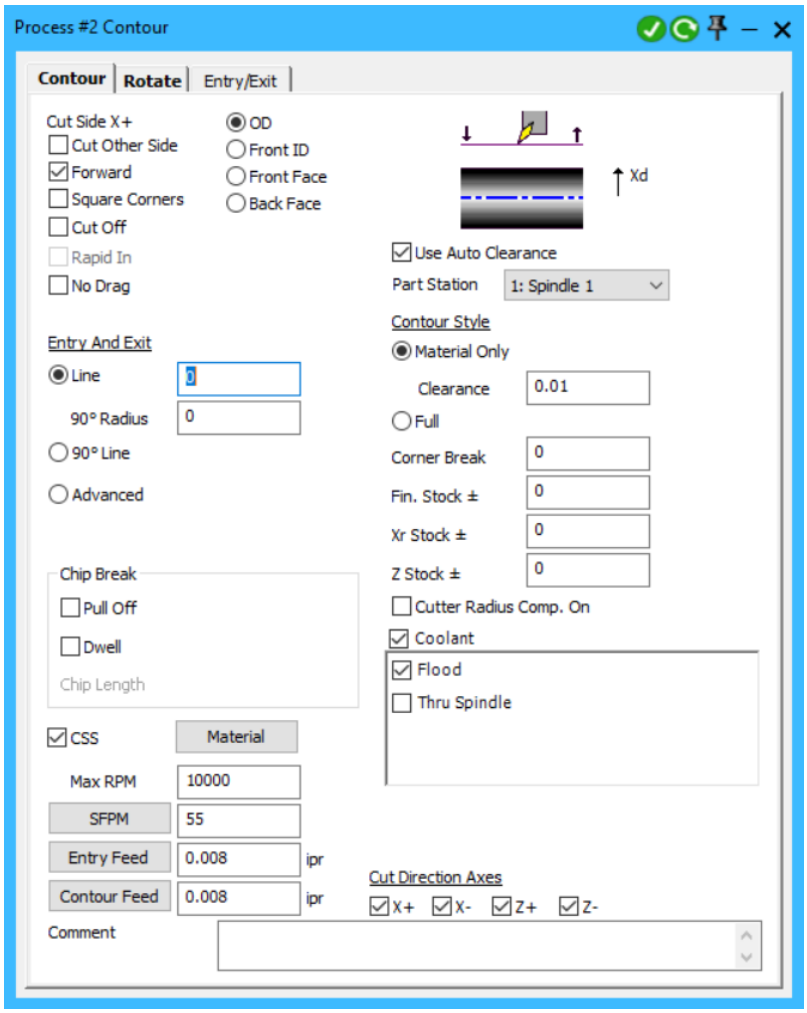
7. Create the toolpath.



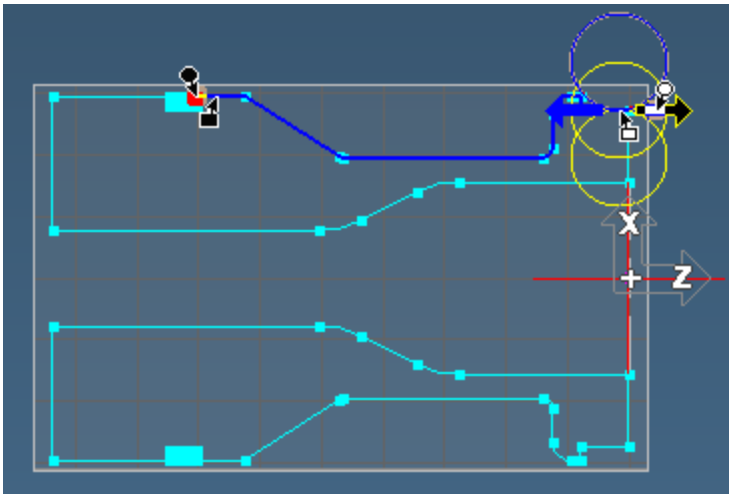
The next operation we will make use of the contouring operation that we just used.

8. Delete the Rough Process from the list.

9. Change the tool in the last Contour process to the 1/64” 35° Insert (Tool #2).

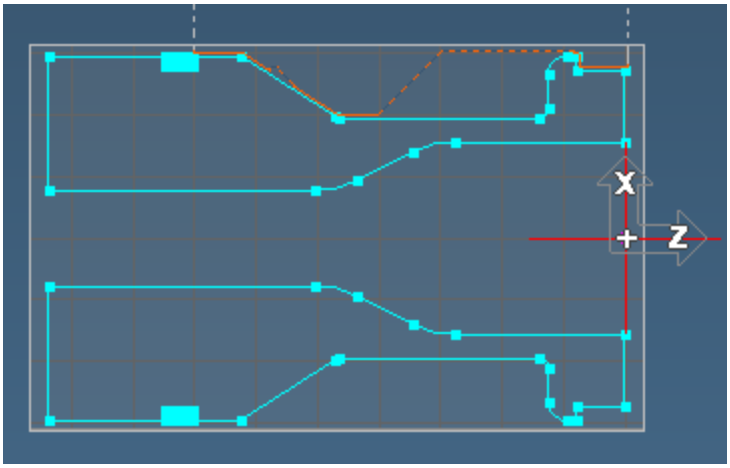


10. Set the machining markers as shown.



This will cut from the front of the part to the back of the first groove, as shown below.

11. Create the toolpath.

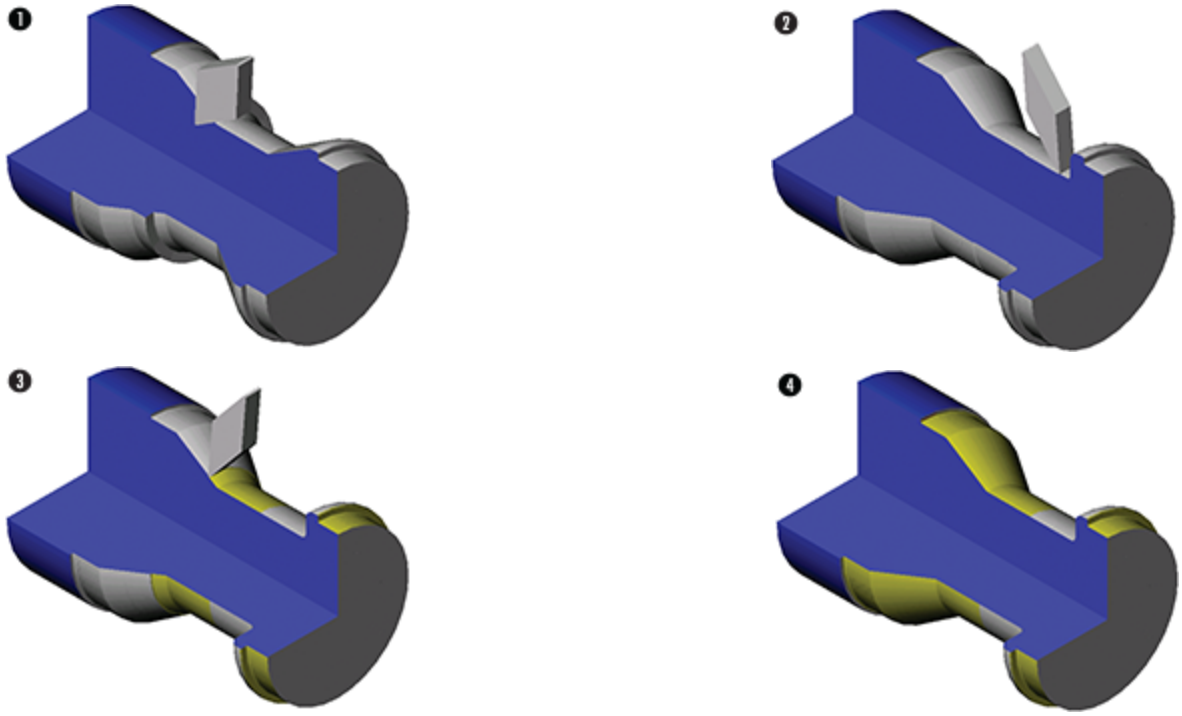


Note how the tool will rapid over the area that has been finished by operation #4.

12. Switch to the isometric view (`Ctrl+I`).

13. Render the operations.

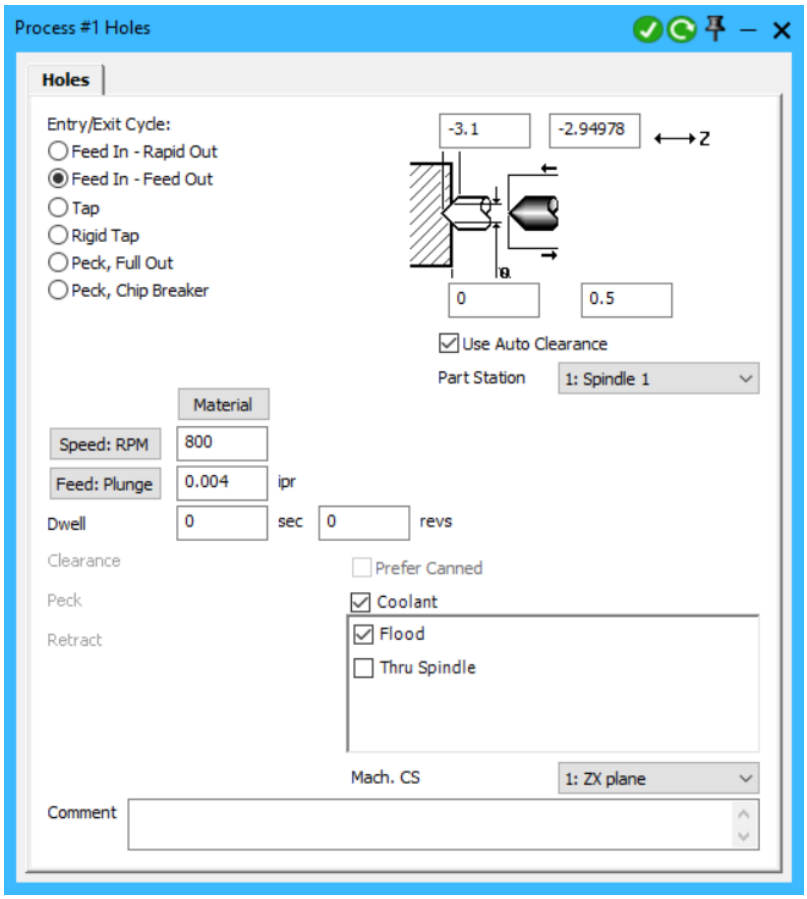
Image 1 shows Operation #2 near the end of its last pass. Image 2 shows Operation #4, a contouring operation, finishing the back of the flange. Image 3 shows Operation #5 performing a Material Only Contouring operation to finish the OD and Image 4 shows the part condition after the first five operations.



ID

Our next set of operations will be to Drill, Rough and Finish the ID of the part.

1. Create a Lathe Holes process with the 1/2” Drill (Tool #5) as shown.

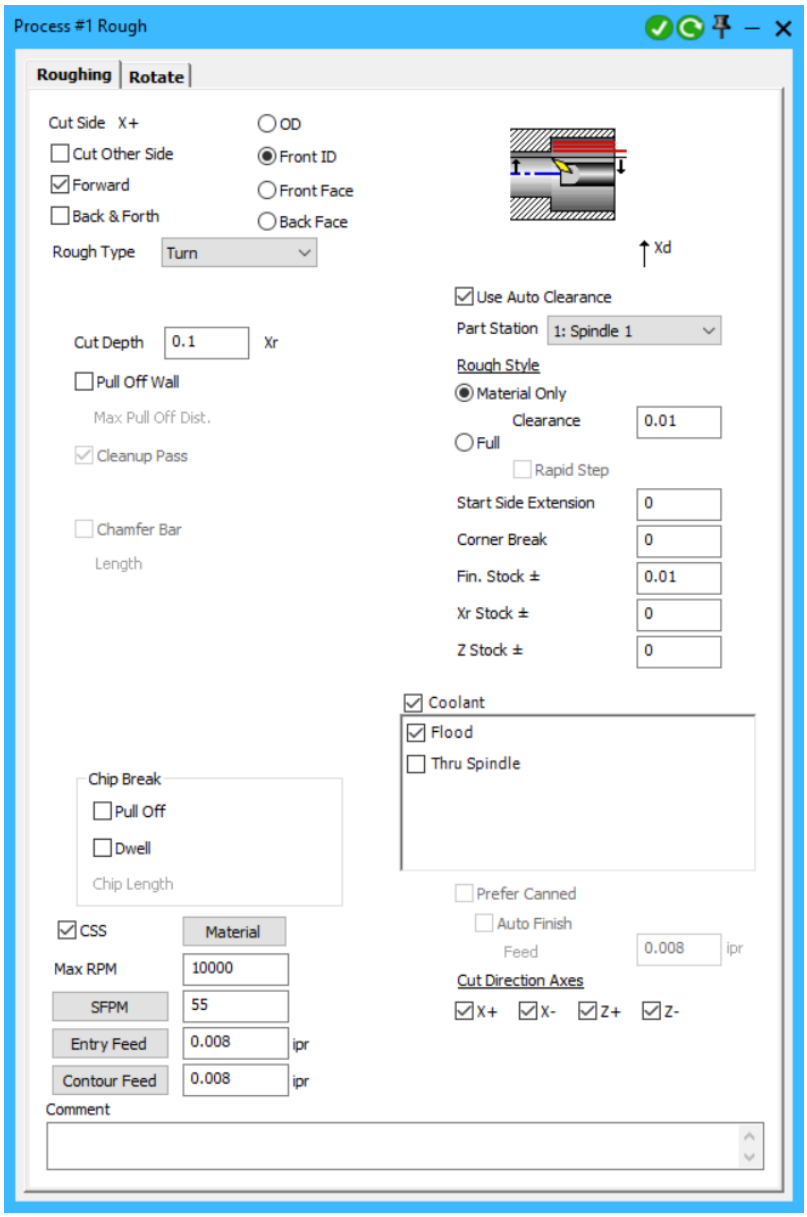


2. Create the toolpath.

   This will drive the tip of the drill to the back of the part.

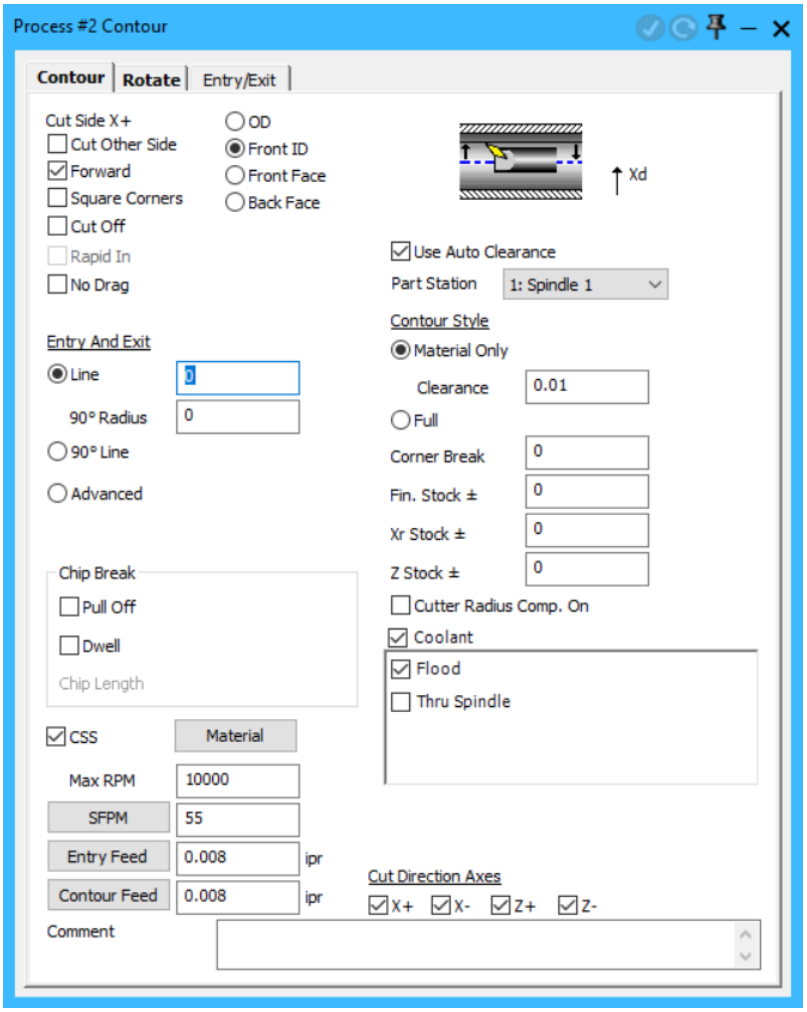
   The next set of operations will be a multi-process operation consisting of a Rough and a Contour process.

3. Create a Lathe Roughing process with 1/64” 80° Insert (Tool #6) as shown.

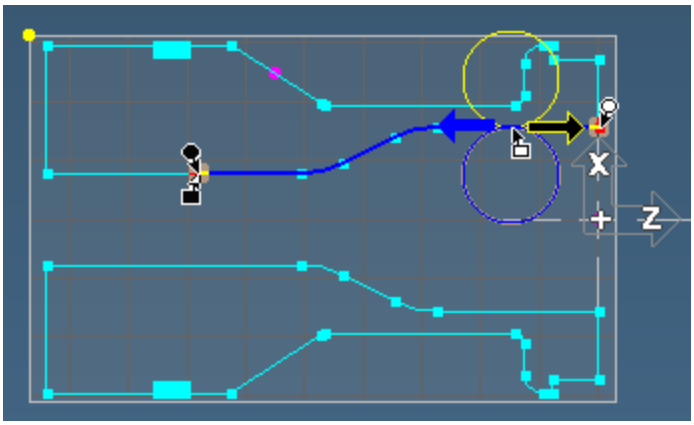


   Be sure to enter a stock value for the Contouring operation to clean up.

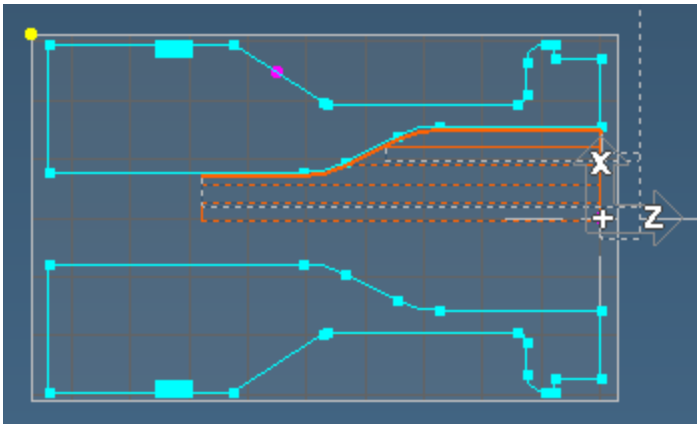
4. Create a Contour process with the 1/64” Triangle Insert (Tool #7) as shown.



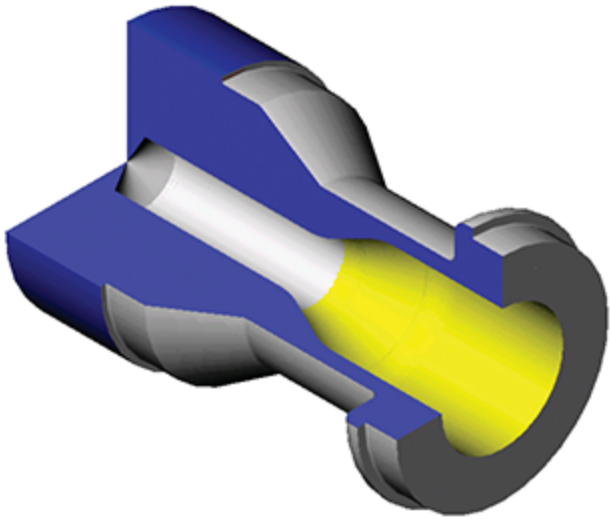
5. Set the machining markers as shown.



6. Create the toolpath.



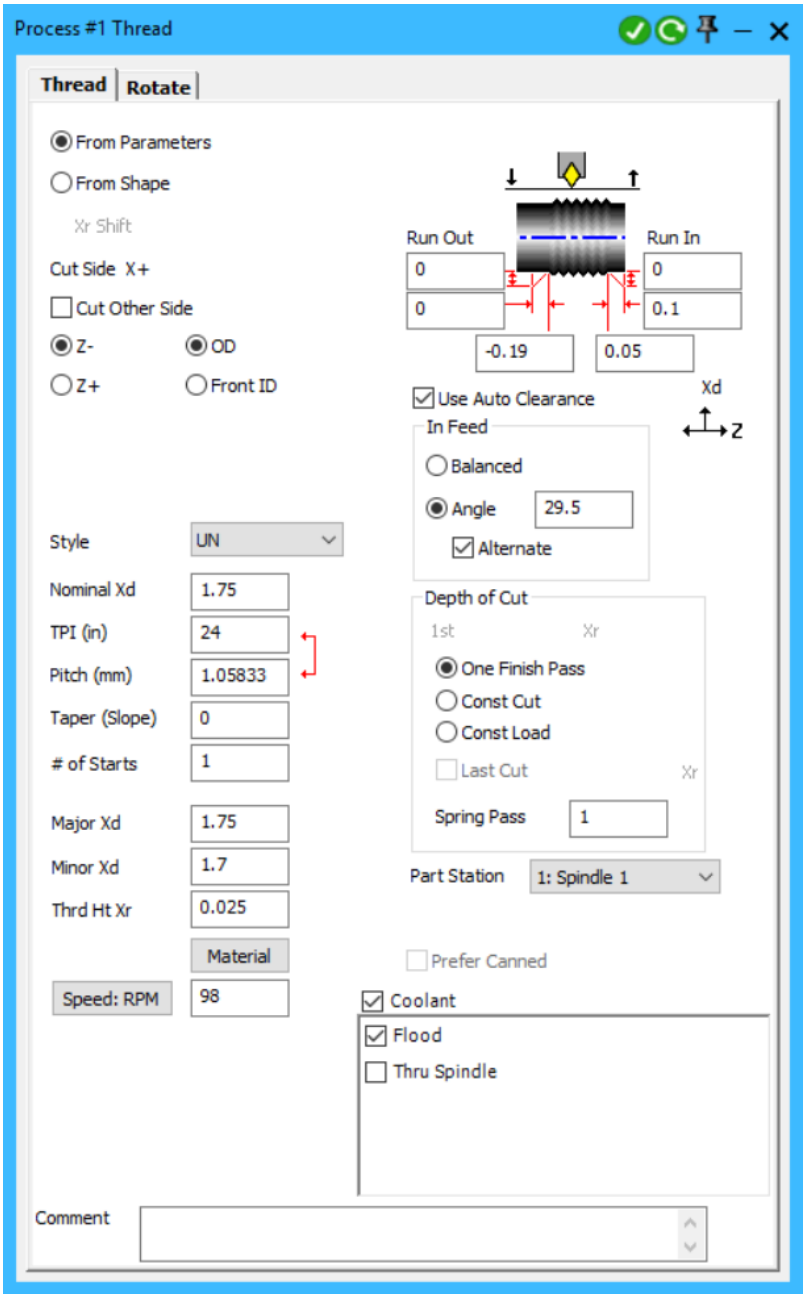
7. Render the Operations.



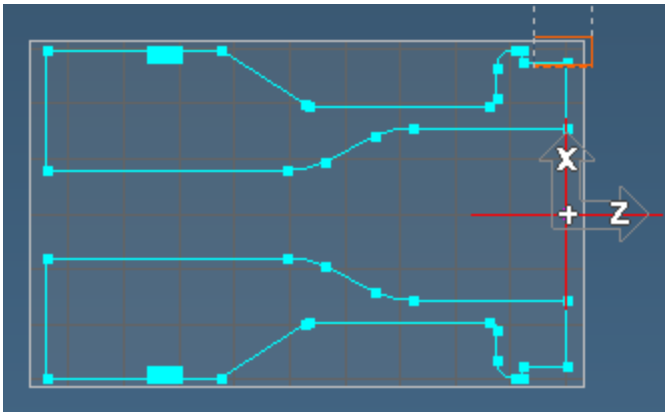
OD Thread

The last turning operation we will perform on Spindle 1 will be an OD thread at the front of the part.

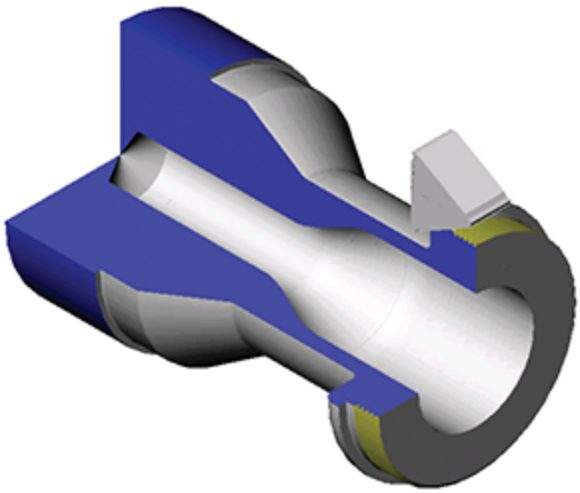
1. Create a Lathe Thread process with the Threading tool (Tool #8) as shown.



2. Create the toolpath.



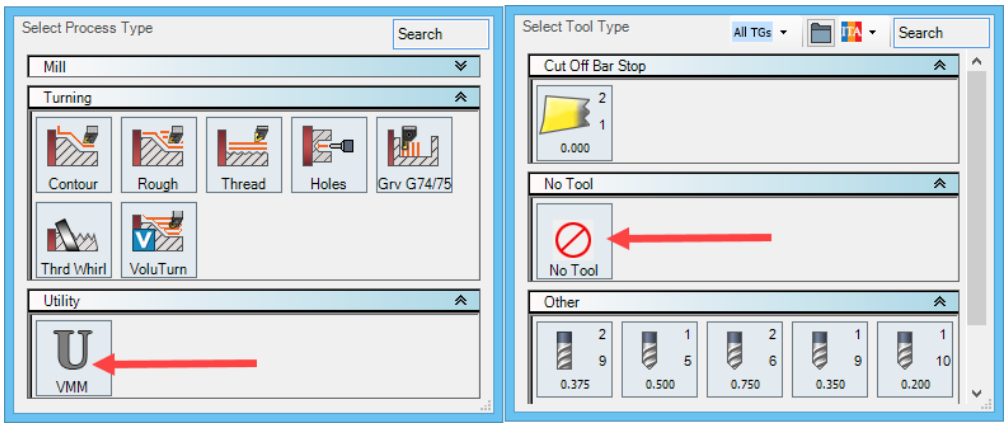
3. Render the operations.



At this point we will create Utility operations to place the sub spindle on the part and return with the part. While this can be created at any time, we will do this now so that we can render operations we will create on the sub spindle. While operations can be created on a spindle that does not have stock, rendering will not show up if stock has not been transferred to the spindle.

Utility Ops

1. Double click a process tile and select the Utility function tile. A tool is not needed.

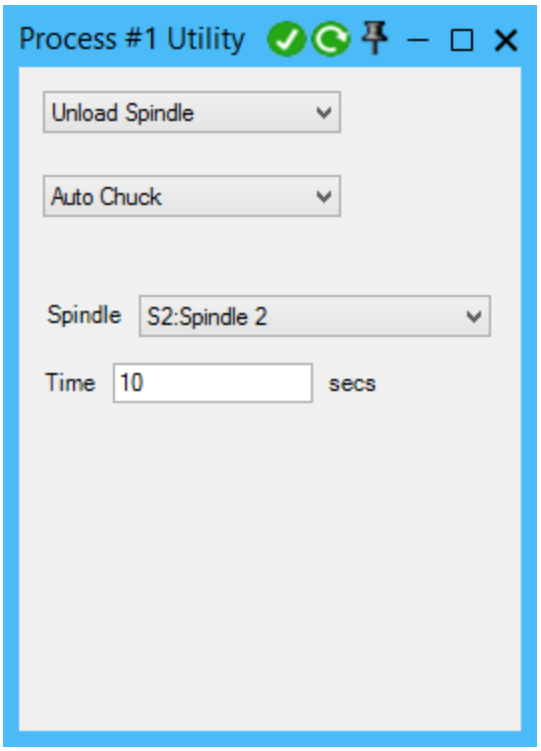


The Utility process tile seen in the CAM palette is a capital “U”. No tool is required so the “No Tool” tool type should be selected.

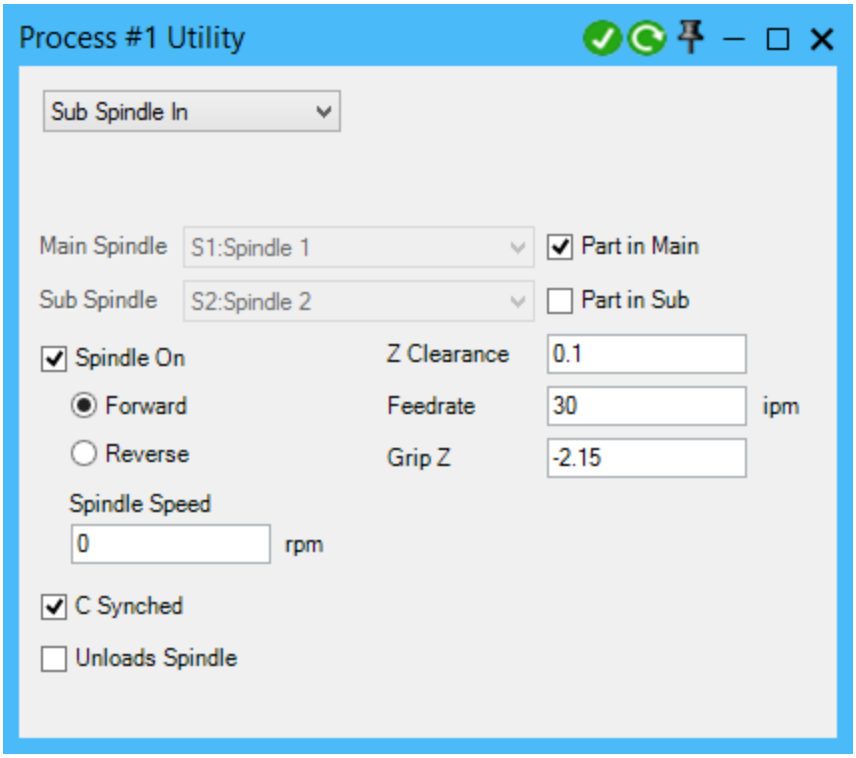
Once “No Tool” is selected the icon in the process tile changes to match the type of Utility operation being created. By default this is typically “Load Spindle” but depending on your MDD may be something else.

The first Utility operation we will make is Unload Spindle. Before bringing our sub spindle to the main we want to be sure the spindle is empty.

2. Create an Unload Spindle operation as shown.

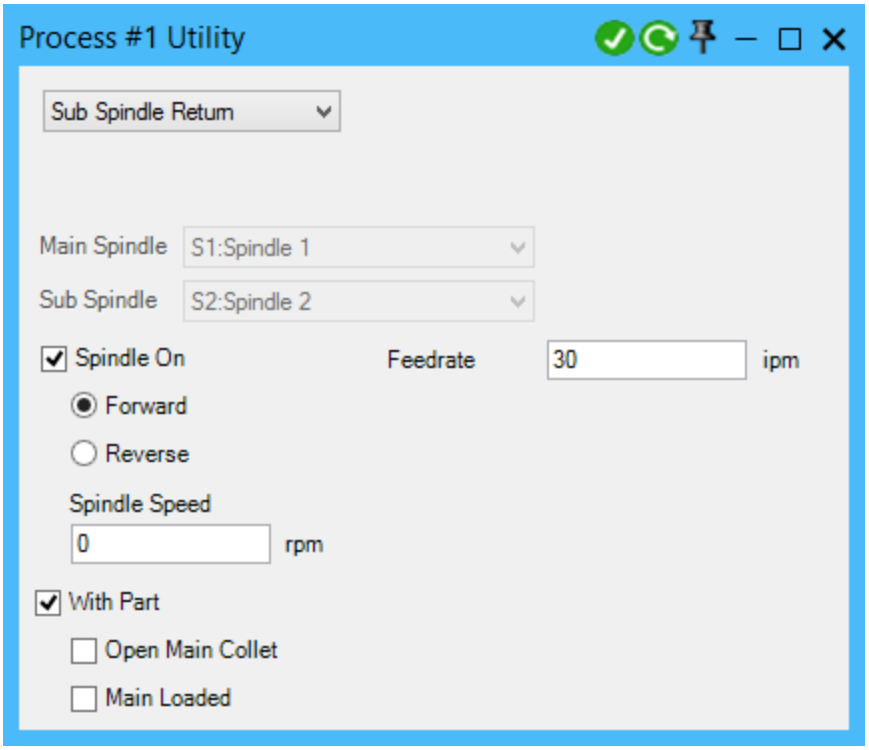


3. Create a Sub Spindle In operation as shown.



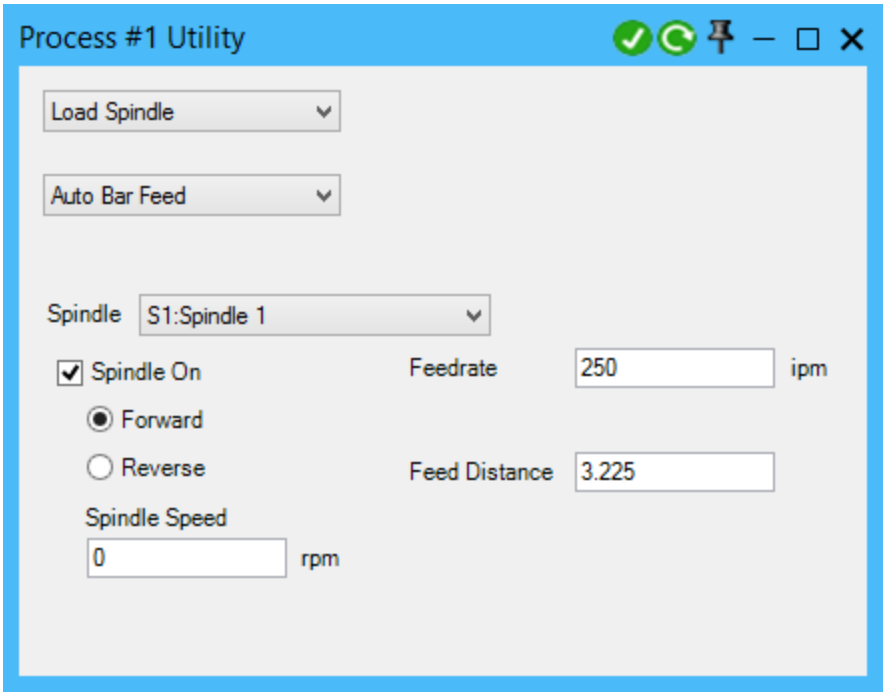
The Grip Z value specifies the depth at which the sub spindle will grab the part. The sub spindle will grab the part 0.100" from the first groove. If you look at the Operations list you will see that these Utility processes actually created two operation tiles each. This is for synching purposes; the operations represent the start and end of the Sub Spindle On Part operation. We will now generate two more Utility Operations. The first will return the sub spindle to its normal position and bring the cut off part with it, and the second operation will load stock into Spindle 1.

4. Create a Sub Spindle Return operation as shown.



Be sure that the With Part checkbox is checked.

5. Create the Utility operation shown.



Click Do it (Ctrl+.).

Sub Spindle Ops

OD

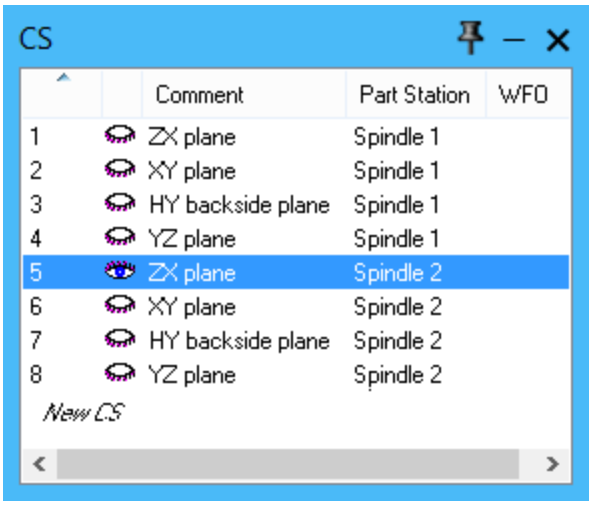
We will now create the sub spindle operations.

> A recommended method for creating operations is to create all ops on the primary spindle first. Once the operations are complete and valid, they may be changed to the sub spindle. This is an excellent method for new users as it minimizes the complexity of the part.

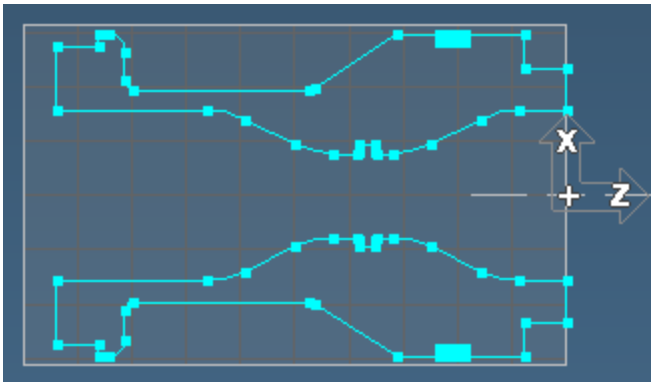
All milling operations will be created last. We will be moving the operations performed on the sub spindle to the empty tile slot so we may leave the empty space.

1. Open the Spindles list and show Spindle 2.

2. Switch to CS 5: ZX plane of Spindle 2 and Unzoom.

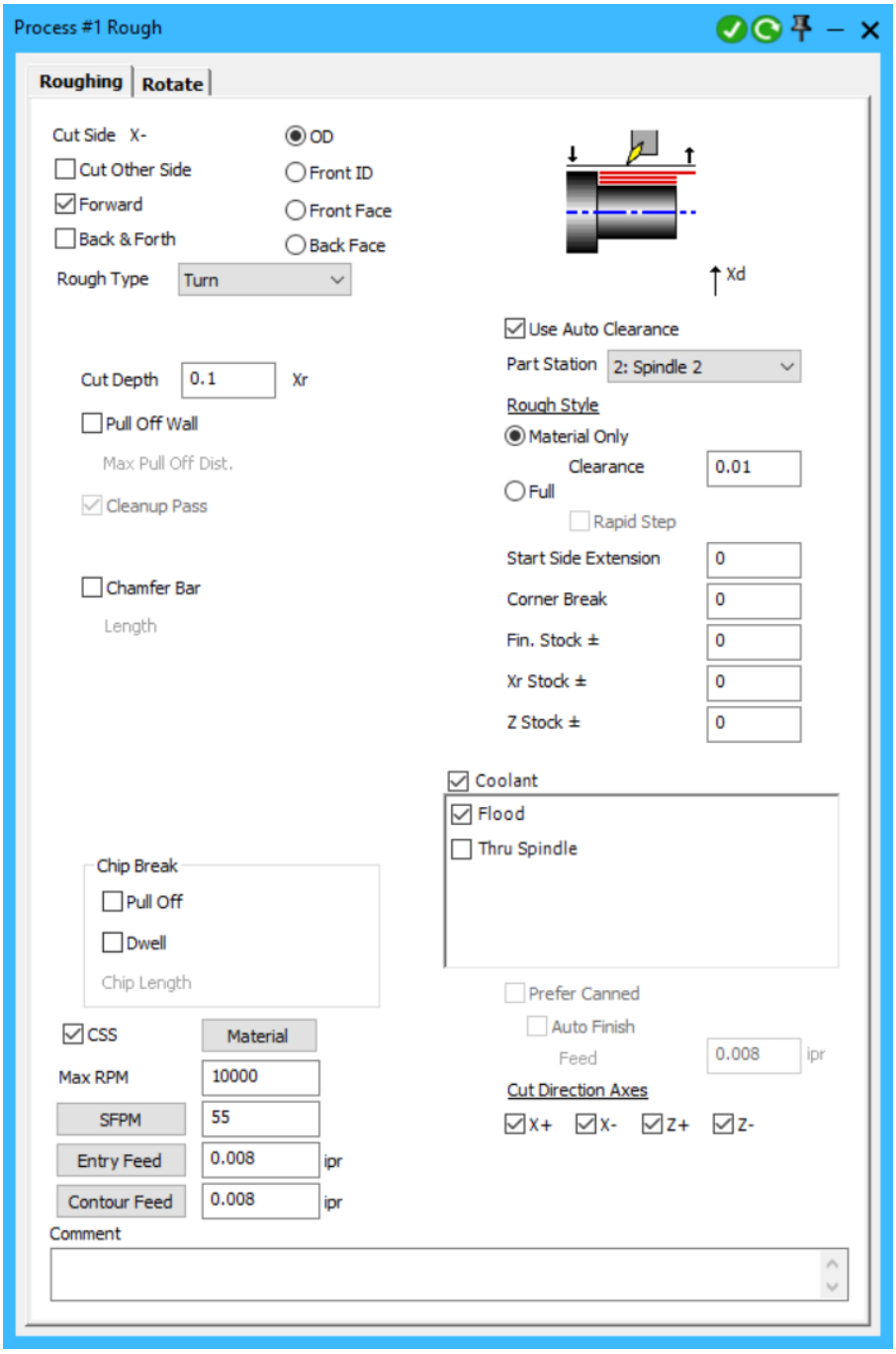


3. Switch to the Home view (`Ctrl+H`).

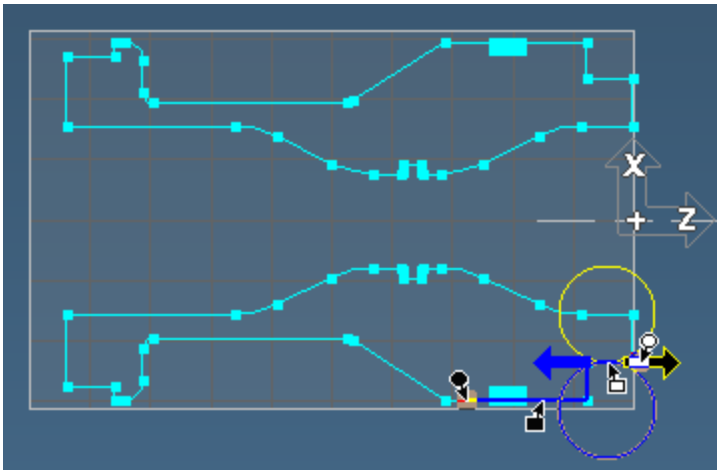


This will re-orient your view of the ZX plane of Spindle 2.

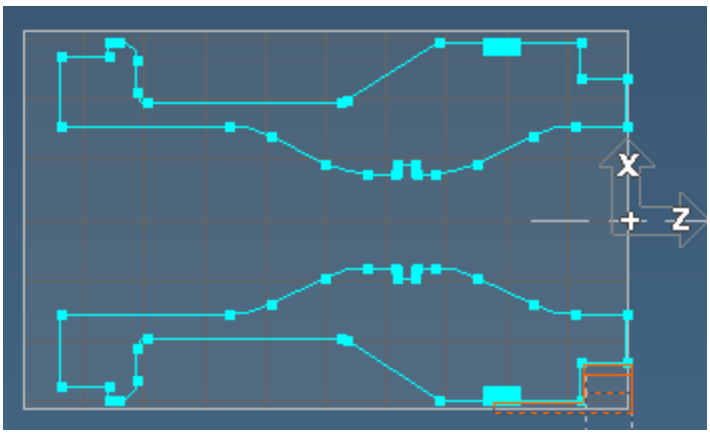
4. Create a Lathe Rough process with the 1/64” 55° Insert (Tool #10) as shown.



5. Set the machining markers as shown.

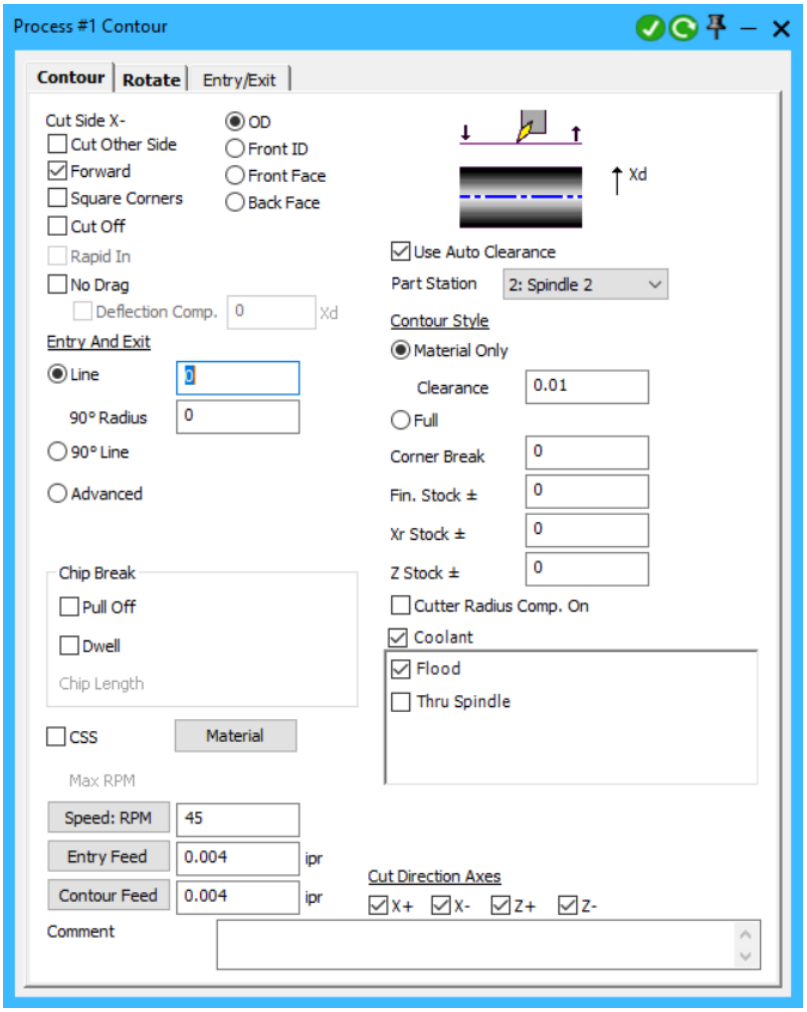


6. Create the toolpath.

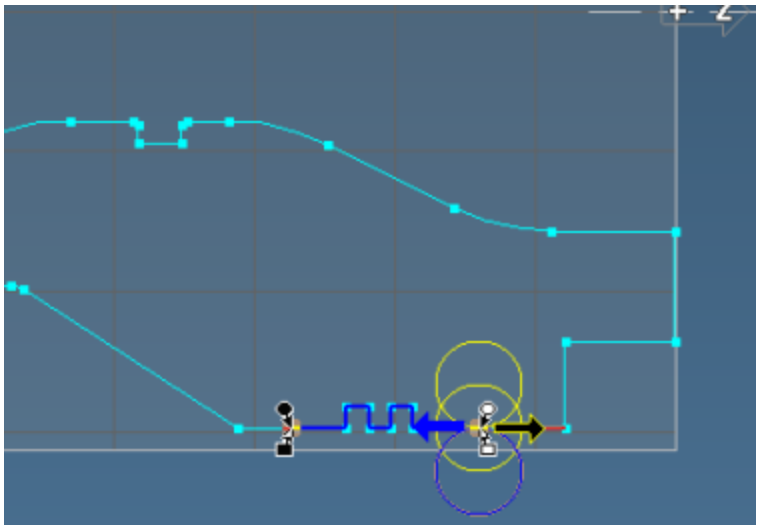


The operation we just created will rough the backside of the part. The machining markers were set on the bottom of the part for ease of visualizing what the tool will do. Remember that all tools in Tool Group 2 are approaching from the bottom. The next operation we will create will machine the OD grooves.

7. Create a Lathe Contour process with the Groove Tool (Tool #12) as shown.

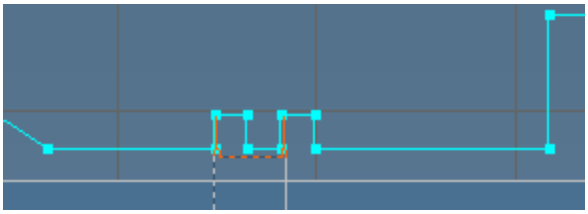


8. Set the machining markers as shown.



The groove tool simply dips into each of the grooves.

9. Create the toolpath.



We will next generate the ID machining, including drilling roughing and finishing operations.

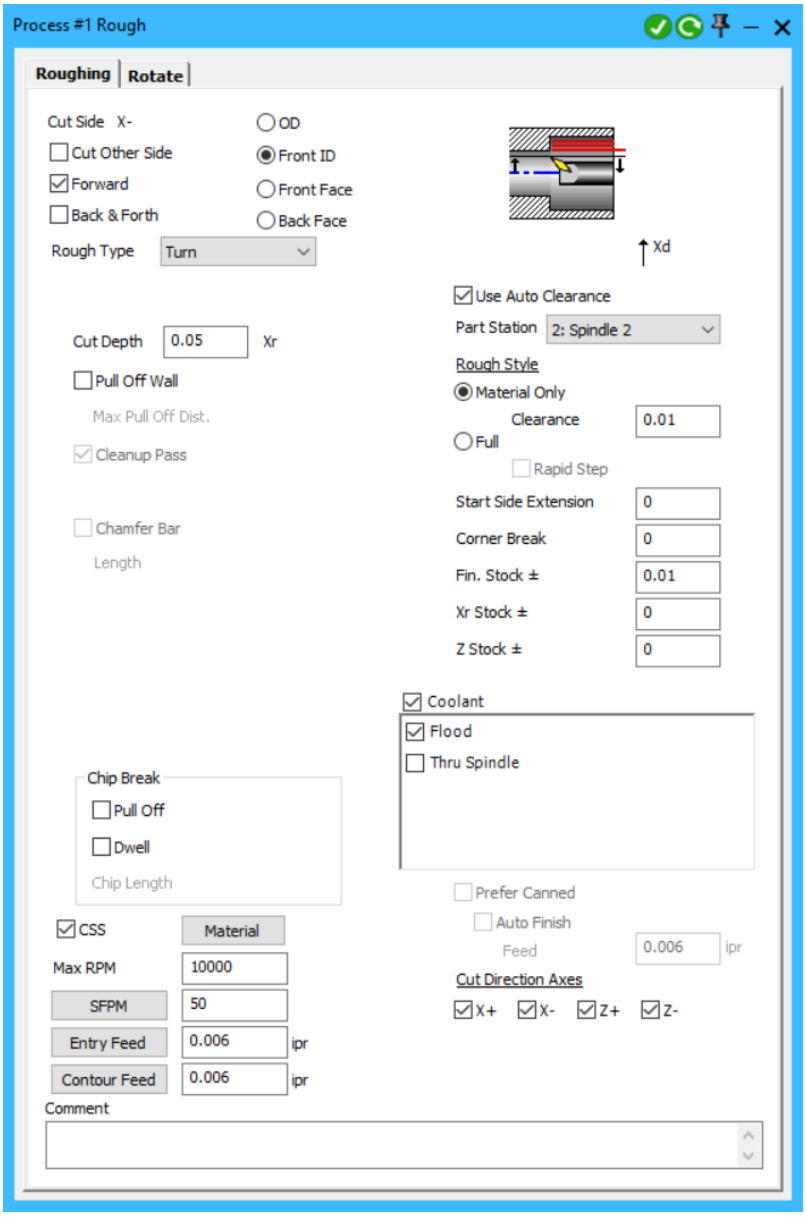
ID

1. Create a Holes process with the 3/4” Drill (Tool #14) as shown.

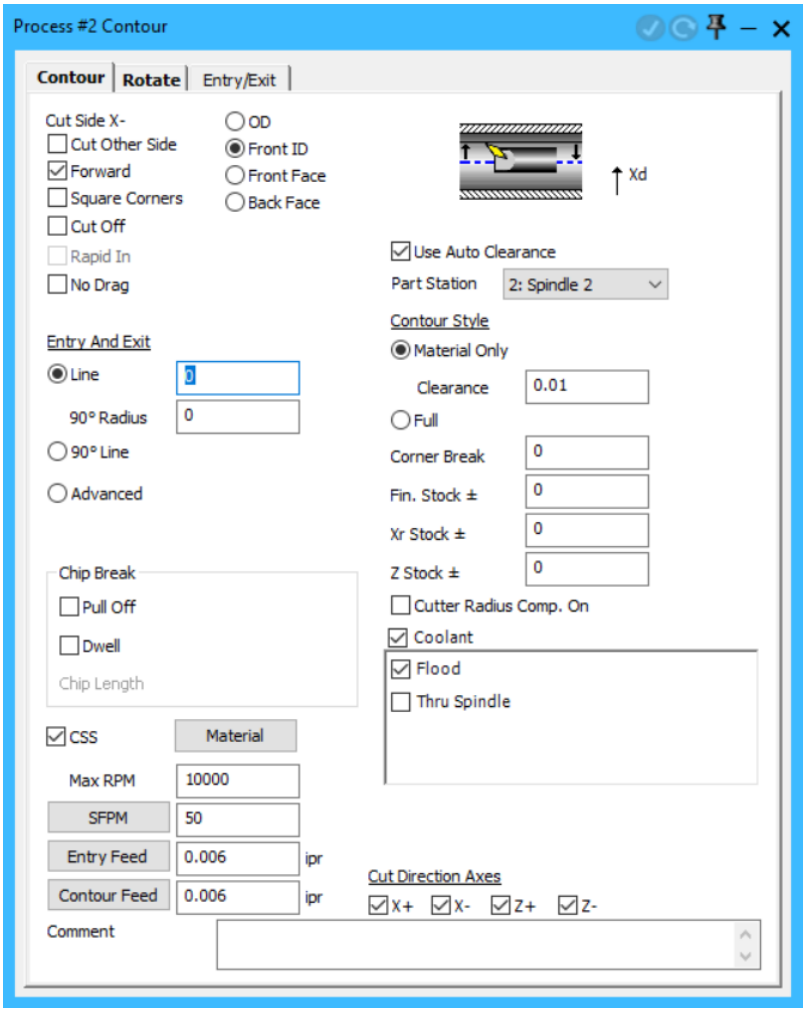
2. Create the toolpath.

   The next set of operations will be a multi-process operation consisting of a Rough and a Contour process.

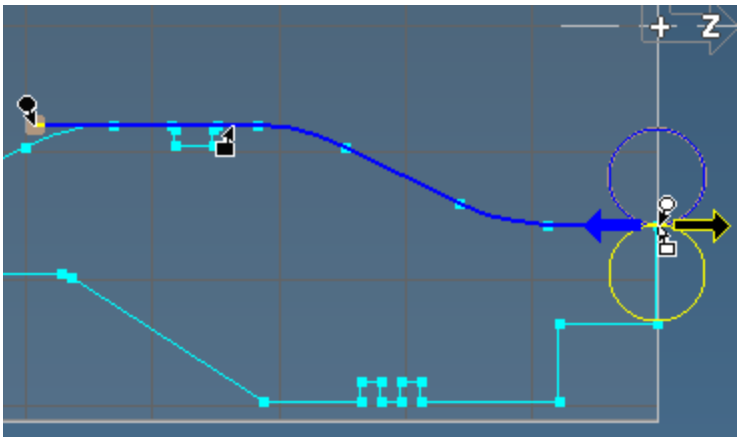
3. For Process #1 create a Lathe Rough process with the 1/64” 80° Insert (Tool #15) as shown.



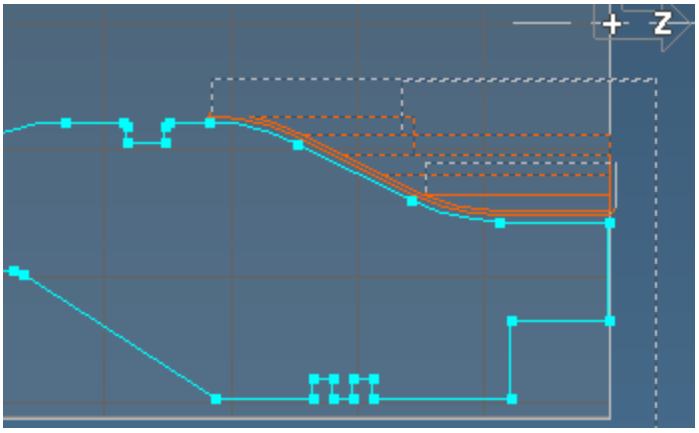
4. For Process #2 using the same tool create a Contour process as shown.



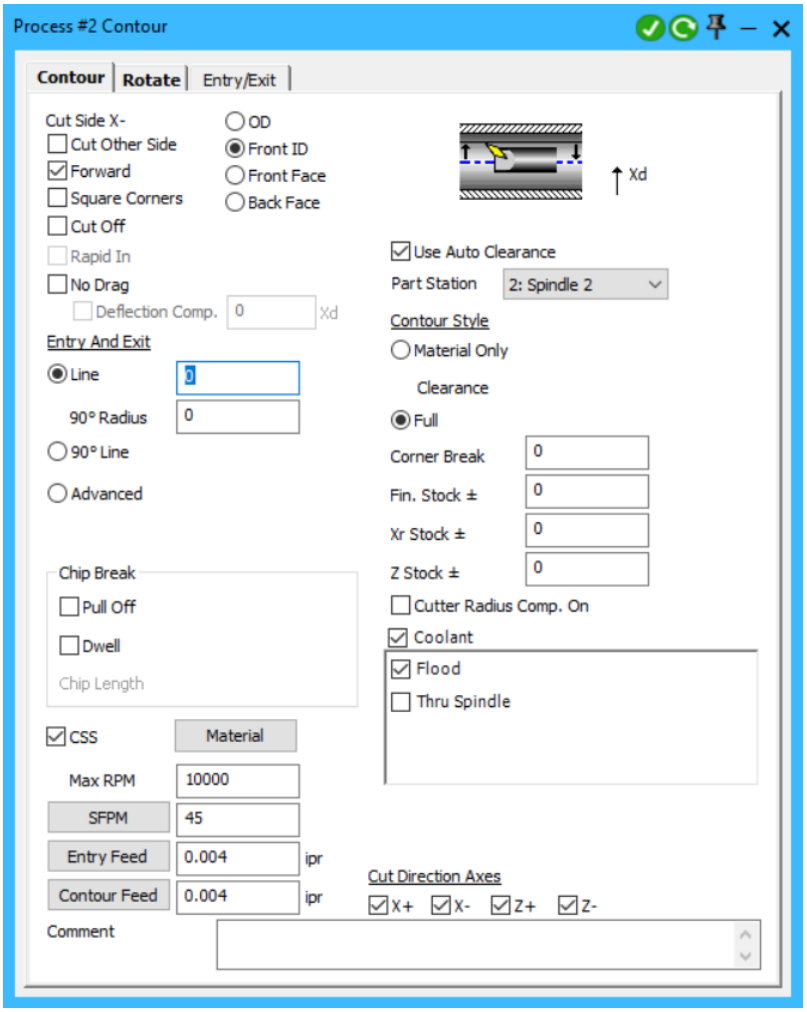
5. Set the machining markers as shown.



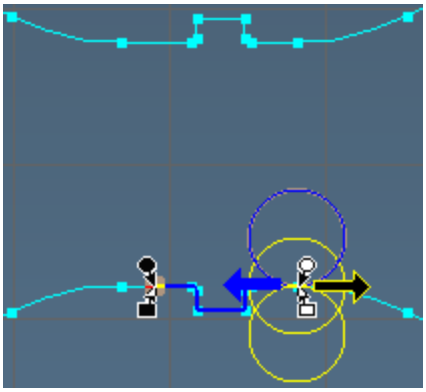
6. Create the toolpath.



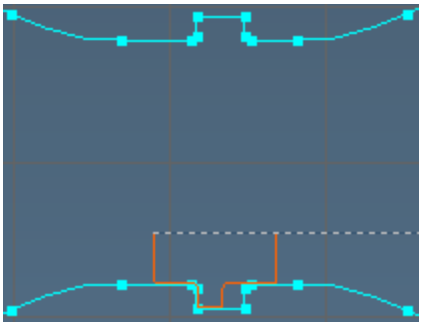
7. Create a Contour process with Groove Tool (Tool #13) as shown.



8. Set the machining markers as shown.



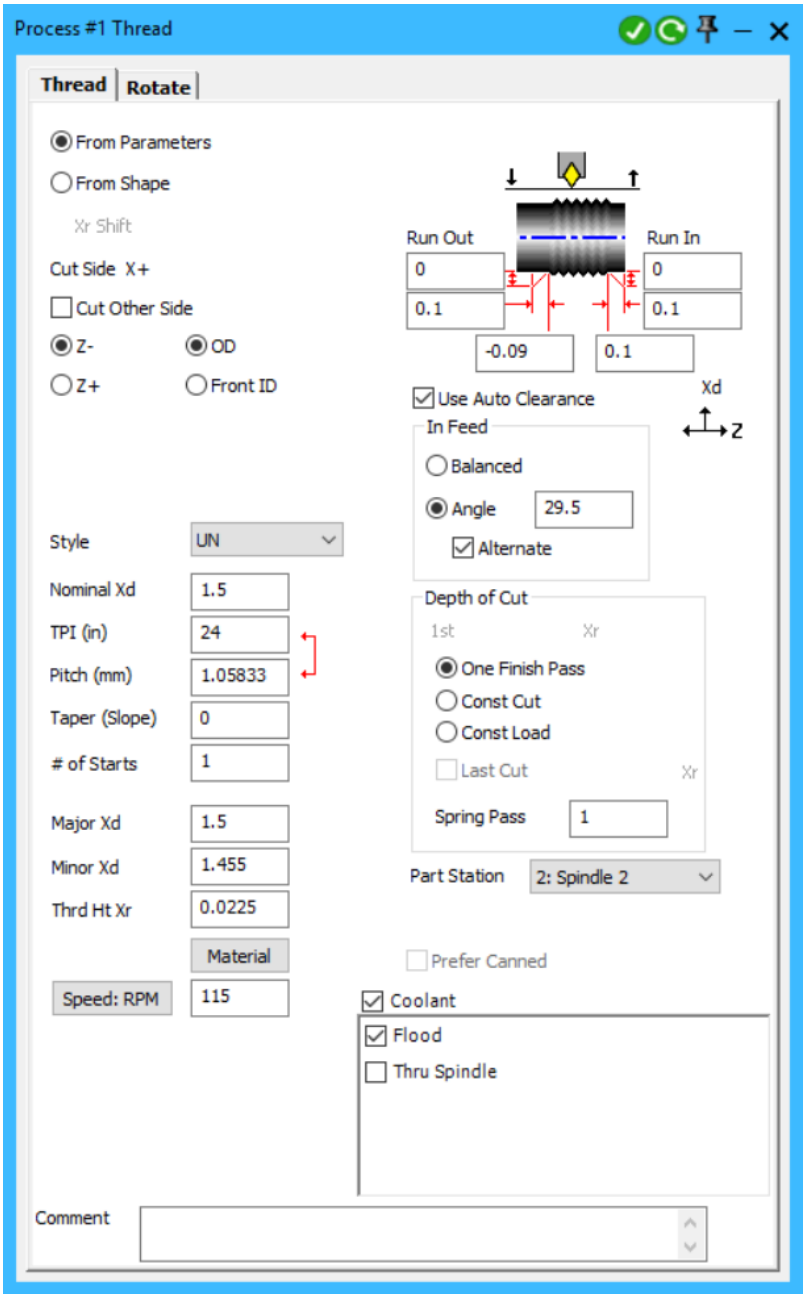
9. Create the toolpath.



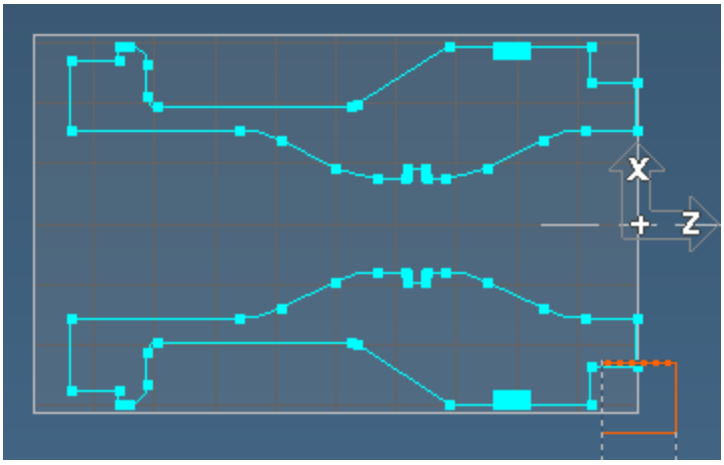
OD Thread

The last turning operation we need to create is the OD thread on the sub spindle.

1. Create a Lathe Thread Process with the Thread Tool (Tool #11) as shown.



2. Create the toolpath.

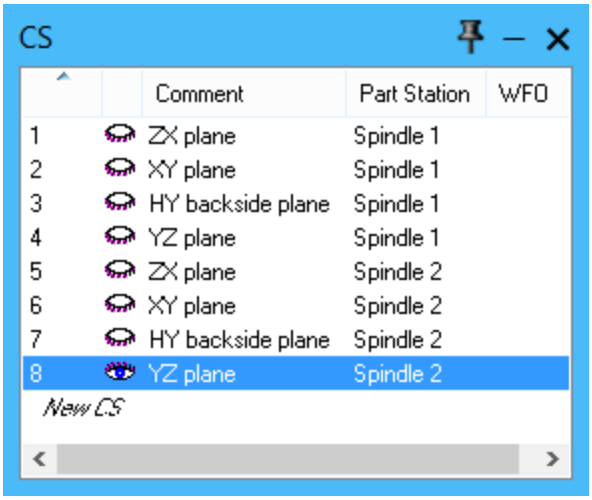


3. Save the part file.

   This completes the turning operations on this part.

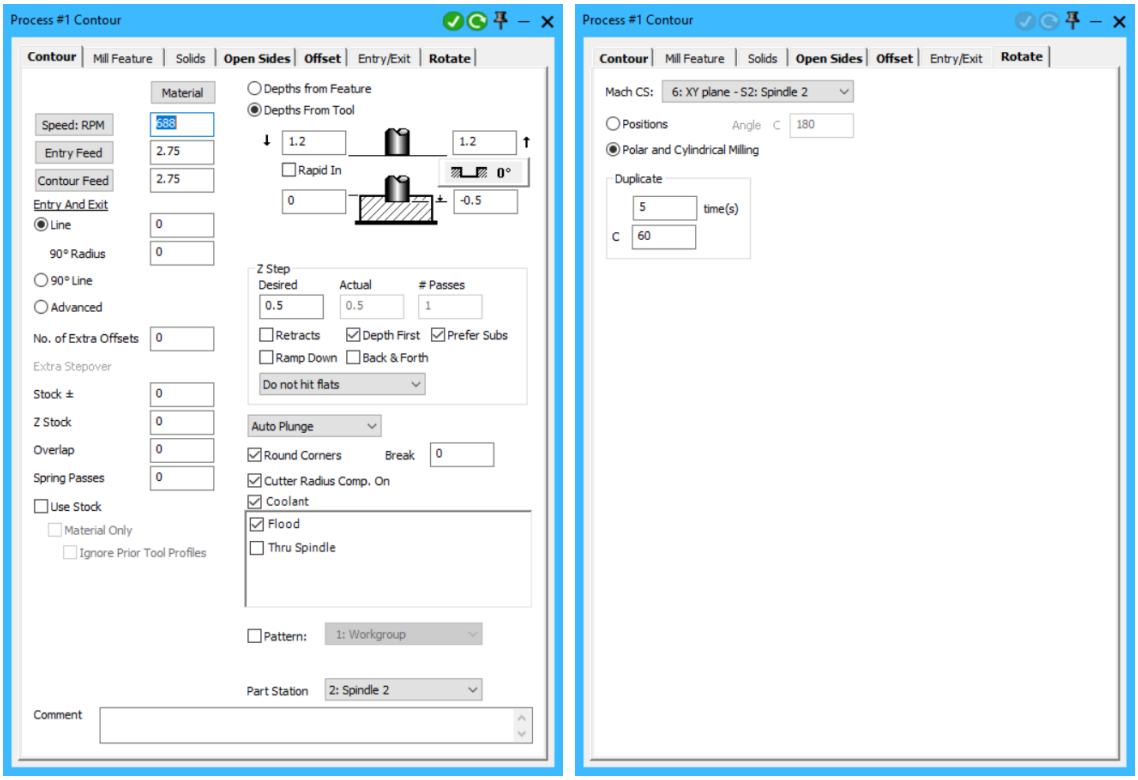
Hexagon

We will now create several milling operations. The first operation we will create is the hexagon on the backside of the part. To generate this operation we will need to create some geometry.

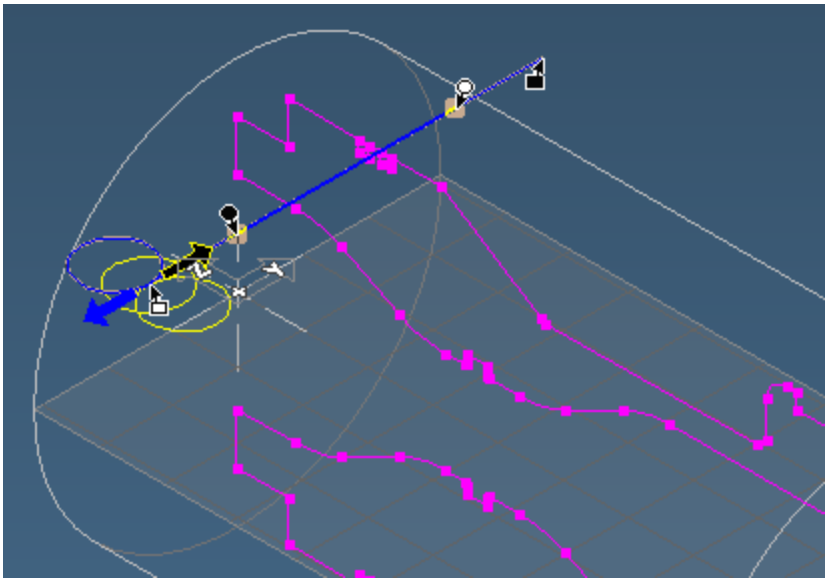


1. Switch to CS8, the YZ Plane of Spindle 2.
2. Create a horizontal line at Z: -0.5", X: 0.75".

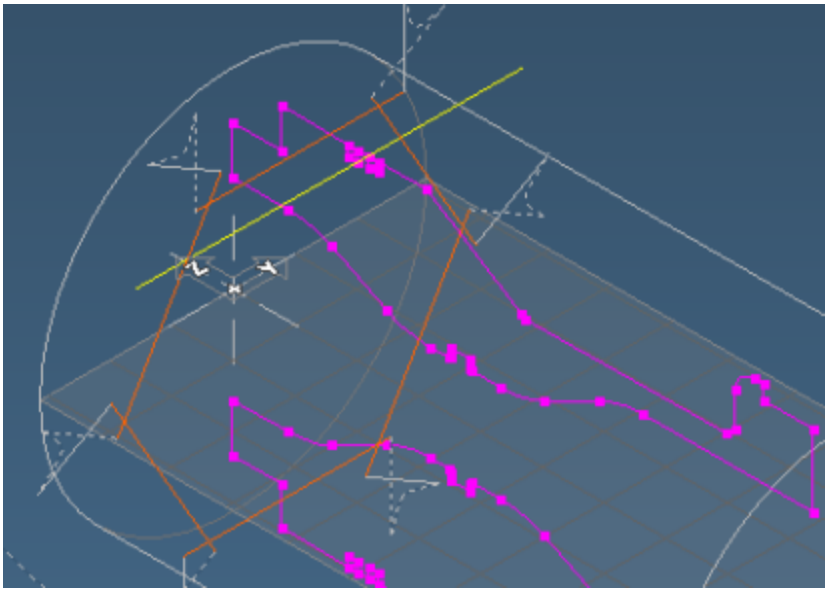
3. Create a Mill Contour process with the 0.5” Finish Endmill (Tool #17) as shown. Note the CS used in the Rotate tab.

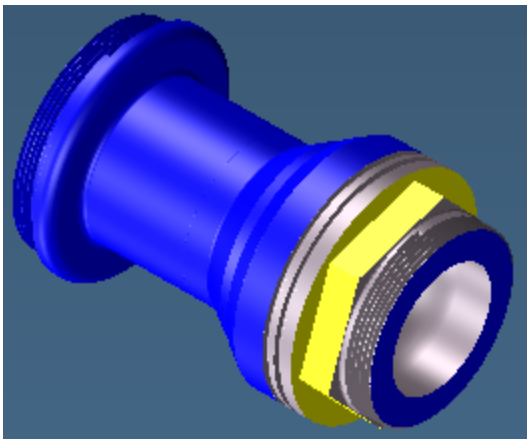


4. Set the machining markers as shown. This placement is crucial - note the positioning of the white and black markers.



5. Create the toolpath.

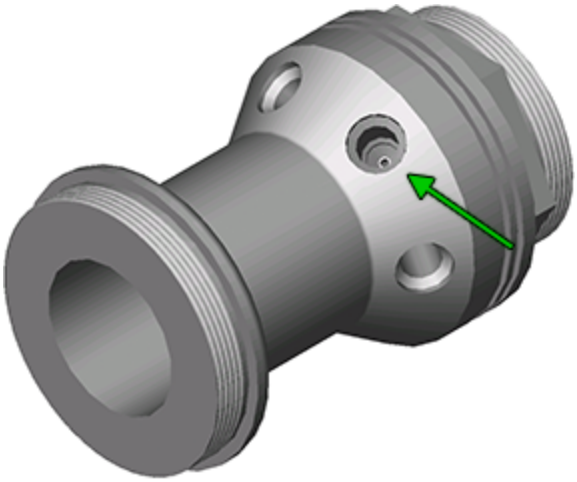




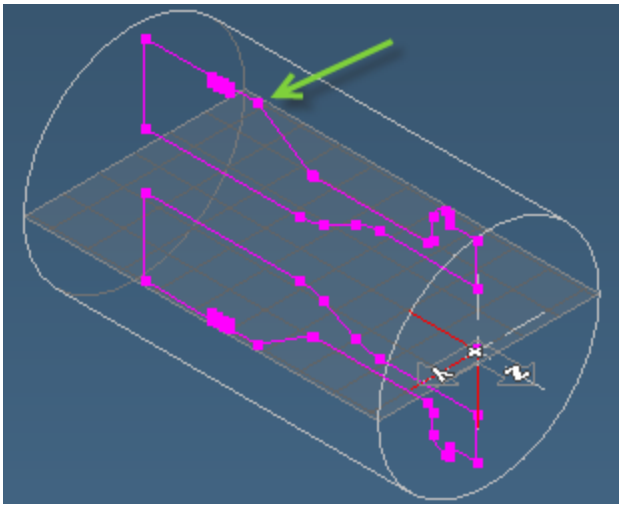
6. Render the operations.

Drilling ops TG1

We will now create drilling operations on Spindle 1 with tools in both Tool Groups. To perform these operations you must have the Advanced CS option. We will be creating a new CS to drill on the B-axis of this part.

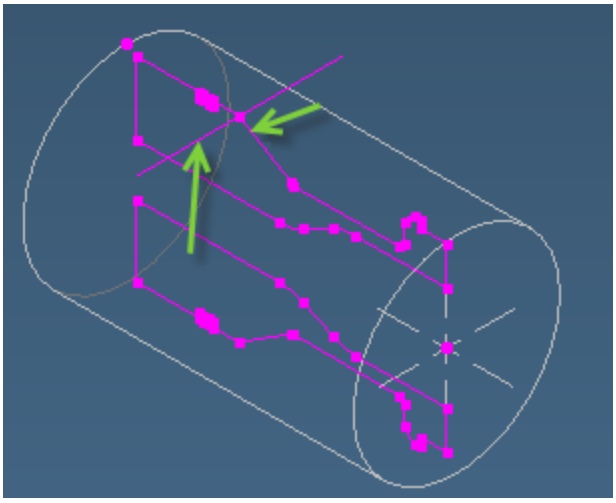


1. Create a new CS based on CS4, the YZ plane of Spindle 1.
2. Switch to Spindle 1.
3. Select the point shown and click the Change CS Origin button.

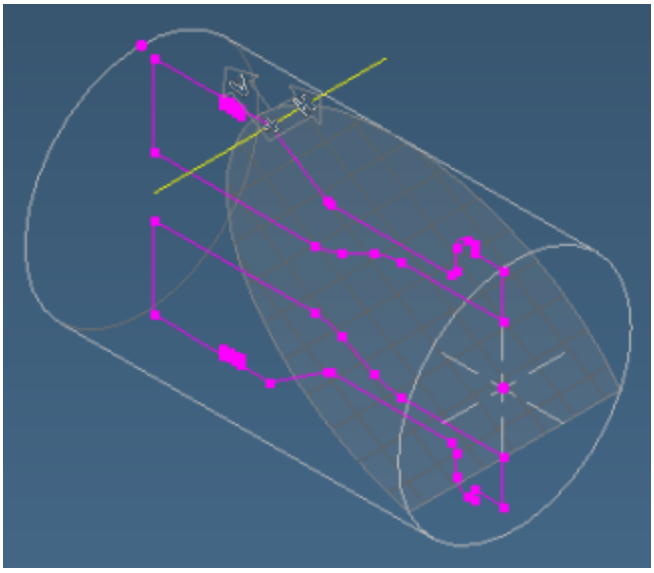


4. Create a horizontal line at Z0.

   This line will be used to help align the CS.
5. Select the two lines shown



6. Click the Align CS Plane button. 

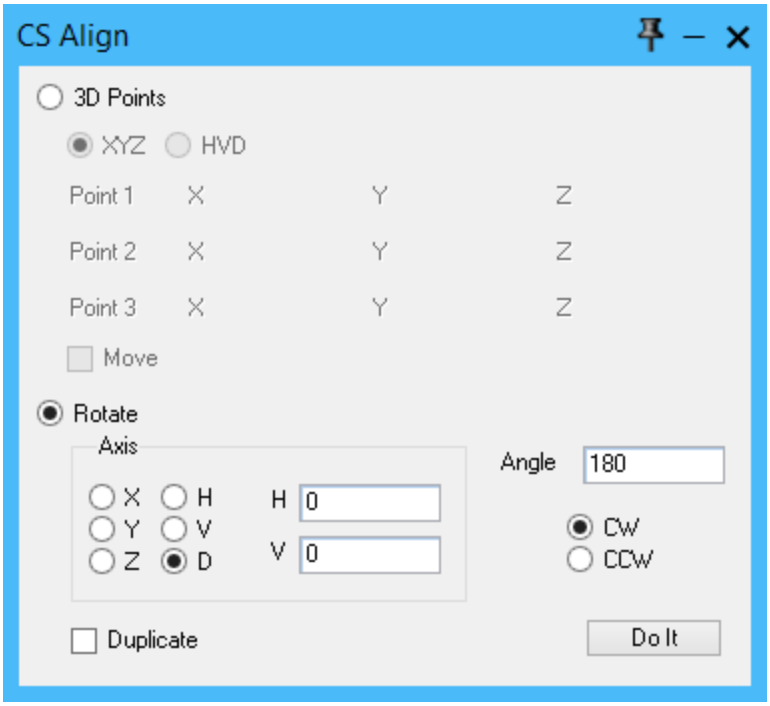


We will now rotate the CS 180° around its depth axis.

7. Deselect all geometry and click the Align CS Plane button again. 

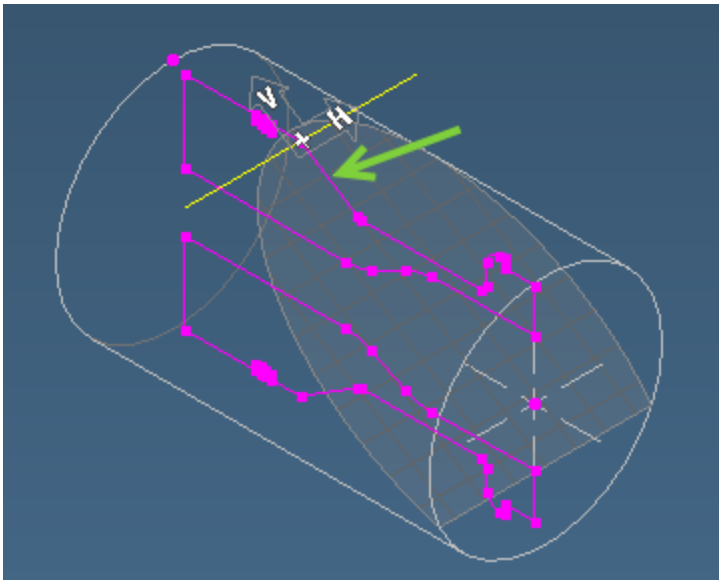
This will open the CS Align dialog.

8. Rotate the CS as shown.



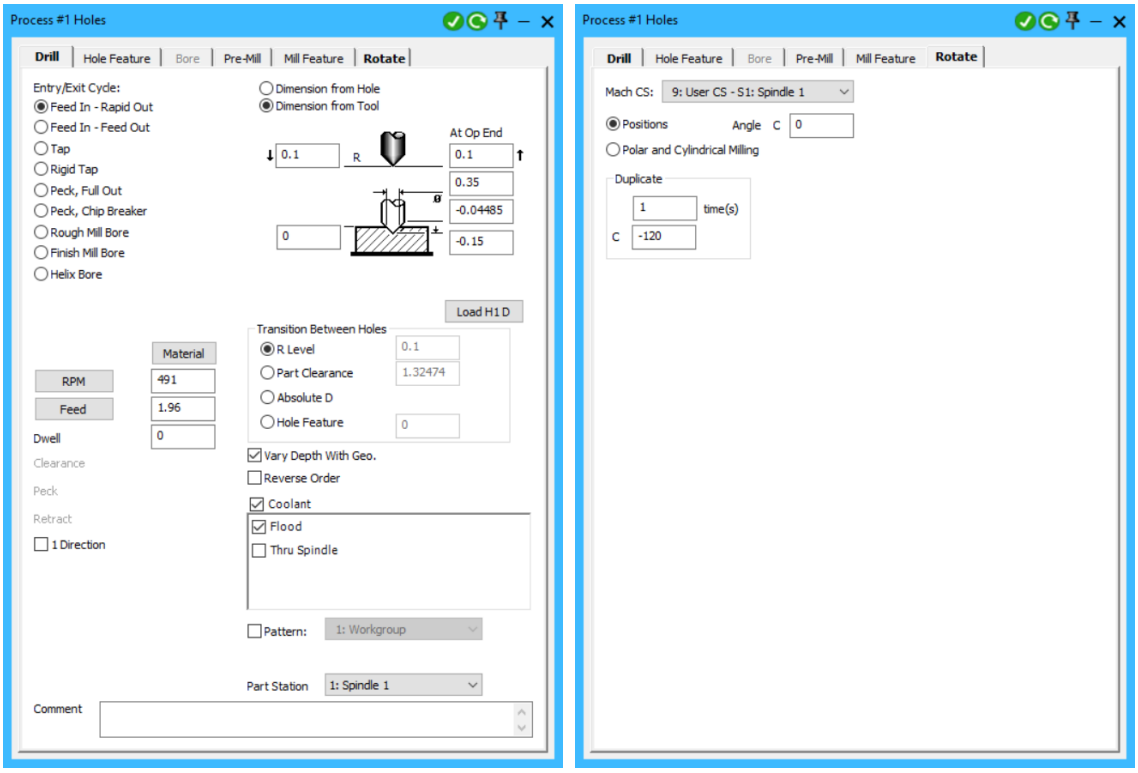
9. Rename the CS "B-Axis Drilling".

   We now need to create a point for the drilling operations we will be creating.

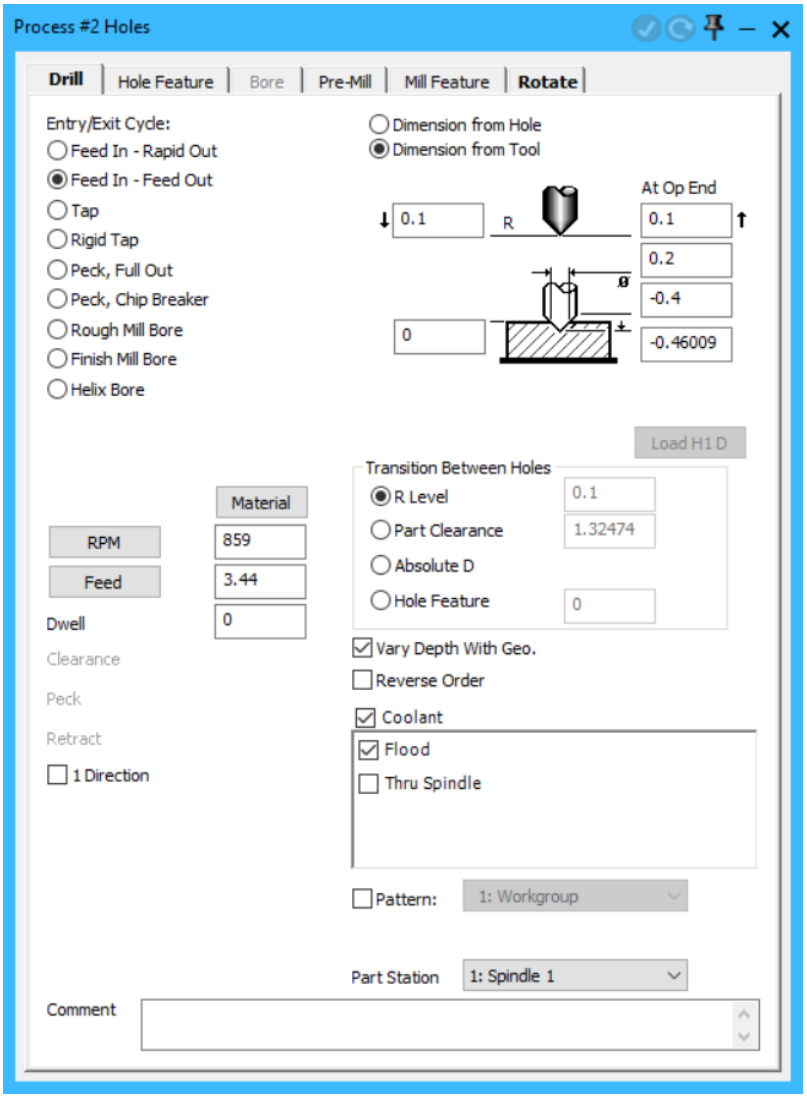


10. Create a Mid-Point for the line shown. 

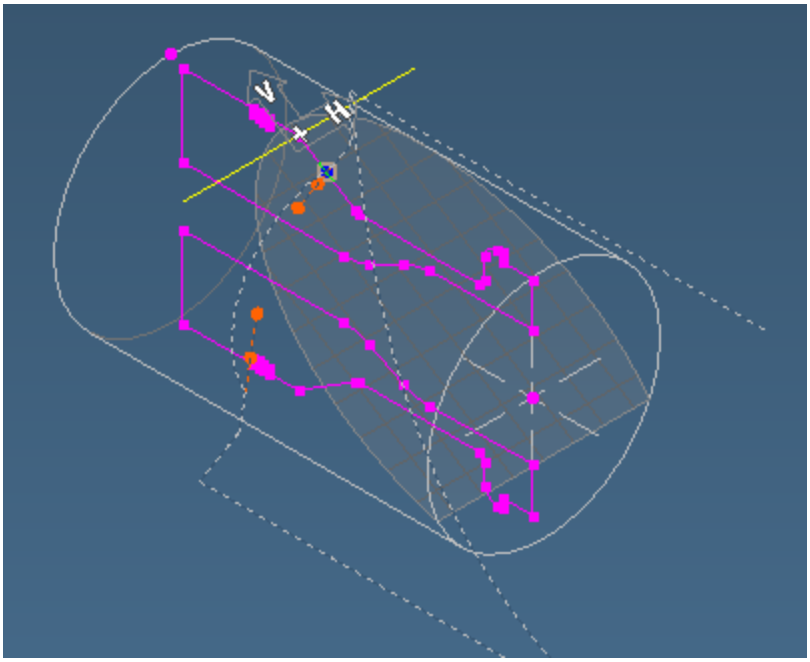
11. For Process #1, create a Milling Holes Process with the 0.35" Drill (Tool #18) as shown.



12. For Process #2, create a Holes process with the 0.2” Drill (Tool #19) as shown.



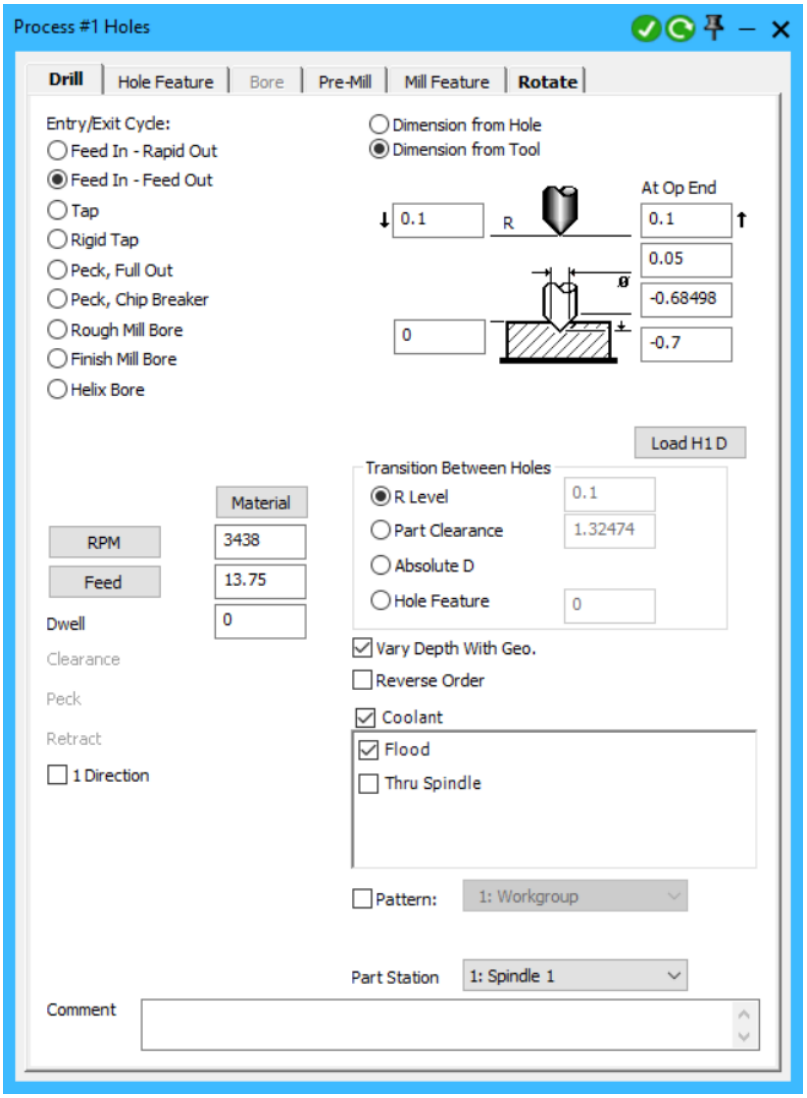
13. Select the mid-point and create the toolpath.



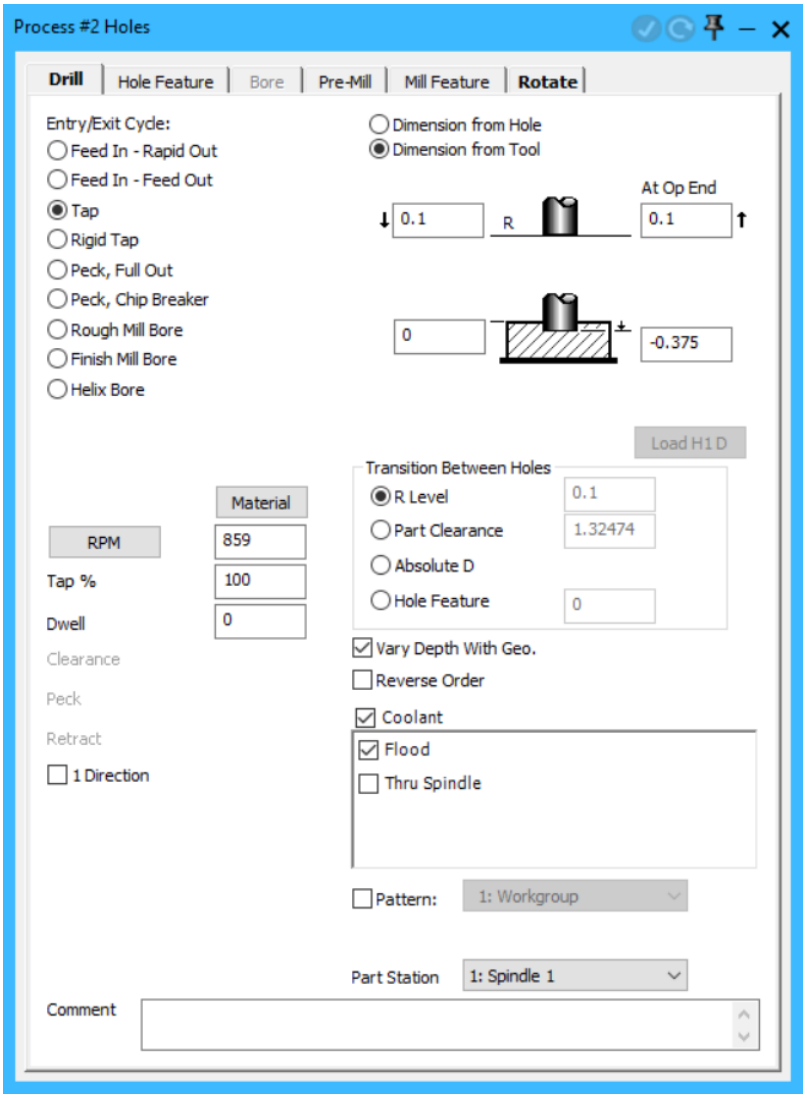
Drilling Ops TG2

We will now create drilling operations for Tool Group 2 using the processes from Tool Group 1.

1. Change the tool used in Process #1 to tool #20, the 0.05” Drill and modify the Process as shown.



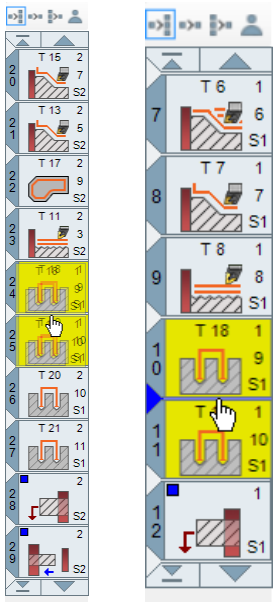
2. Change Process #2 to use the Tap tool (Tool #21) and modify the Process as shown.



3. Deselect the previous operations.
4. Create the toolpath.

   We now need to sync the drilling operations because otherwise we will have two milling operations trying to perform tasks on the same spindle at the same time. Additionally, we need to move the operations into the flows, meaning the operations need to be placed within the bounds of the utility operations.

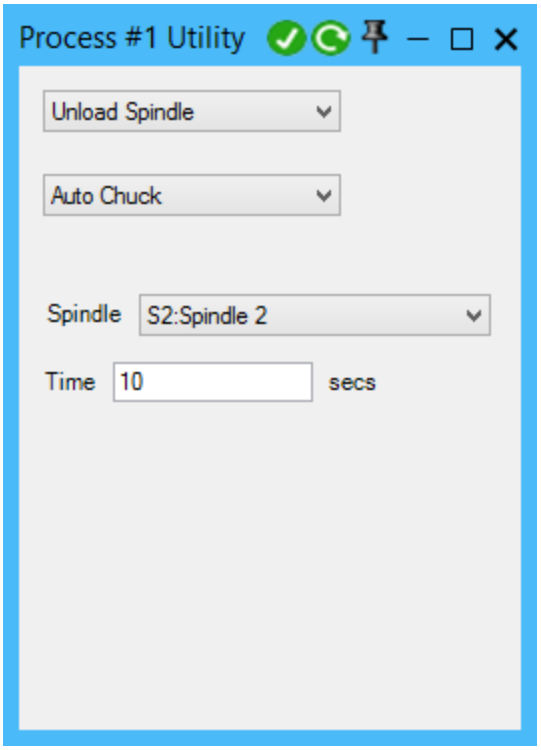
5. Drag the ops with tools 18 and 19 to the space between Ops 9 and 10.



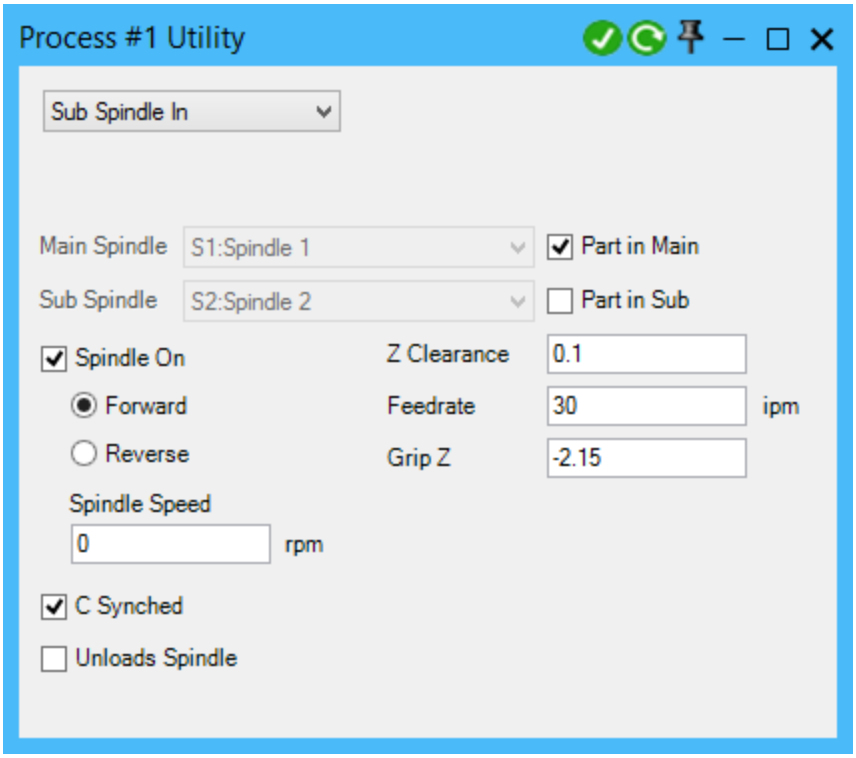
This will put the operations before the Utility operations begin.

Sub-Spindle Utility Ops

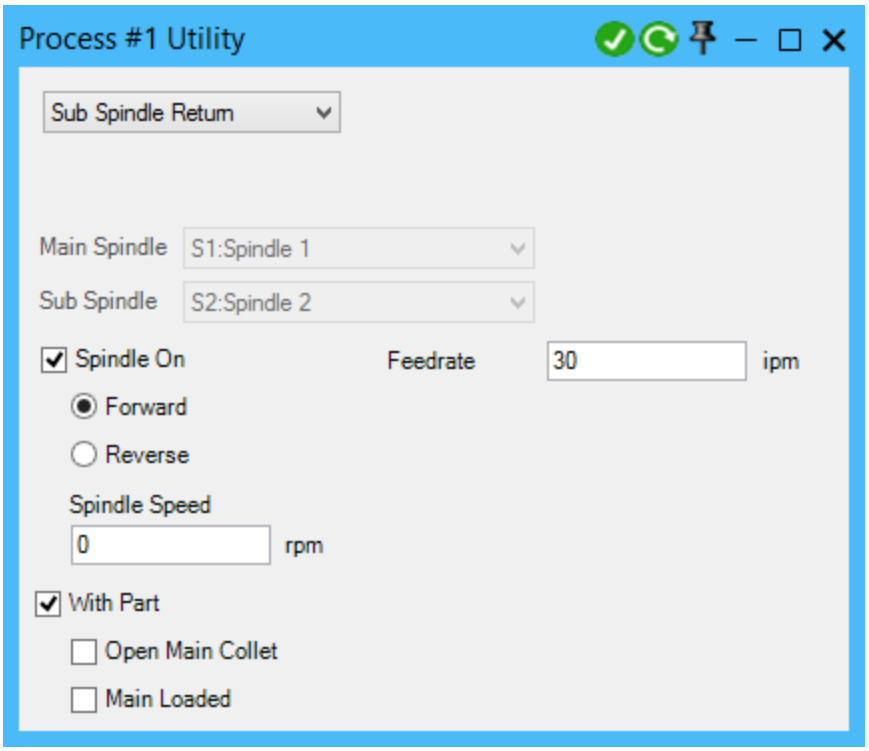
1. Create an Unload Spindle operation as shown.



2. Create a Sub Spindle In operation as shown.

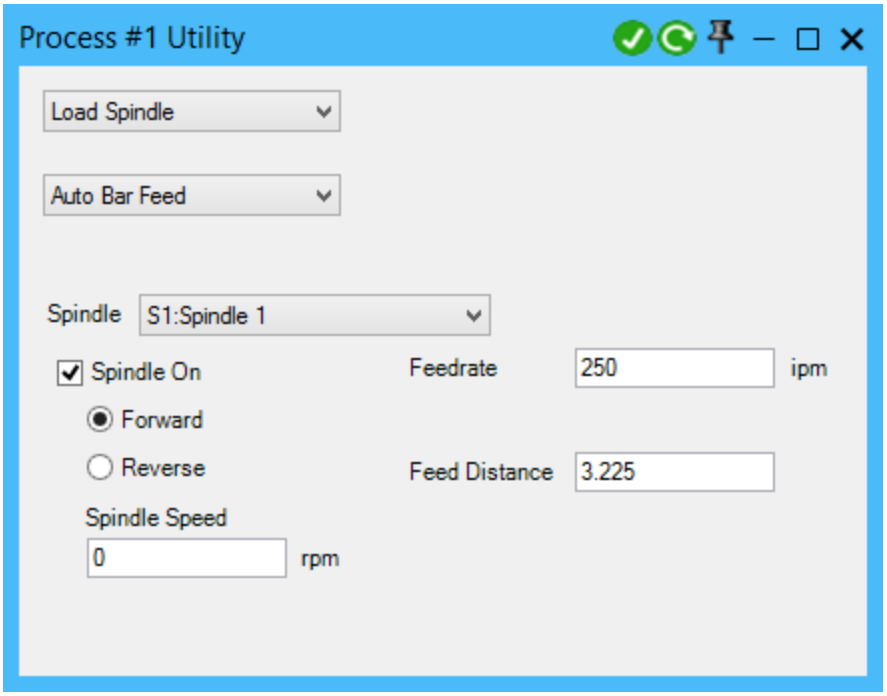


3. Create a Sub Spindle Return operation as shown.



Be sure that the With Part checkbox is checked.

4. Create the Utility operation shown.

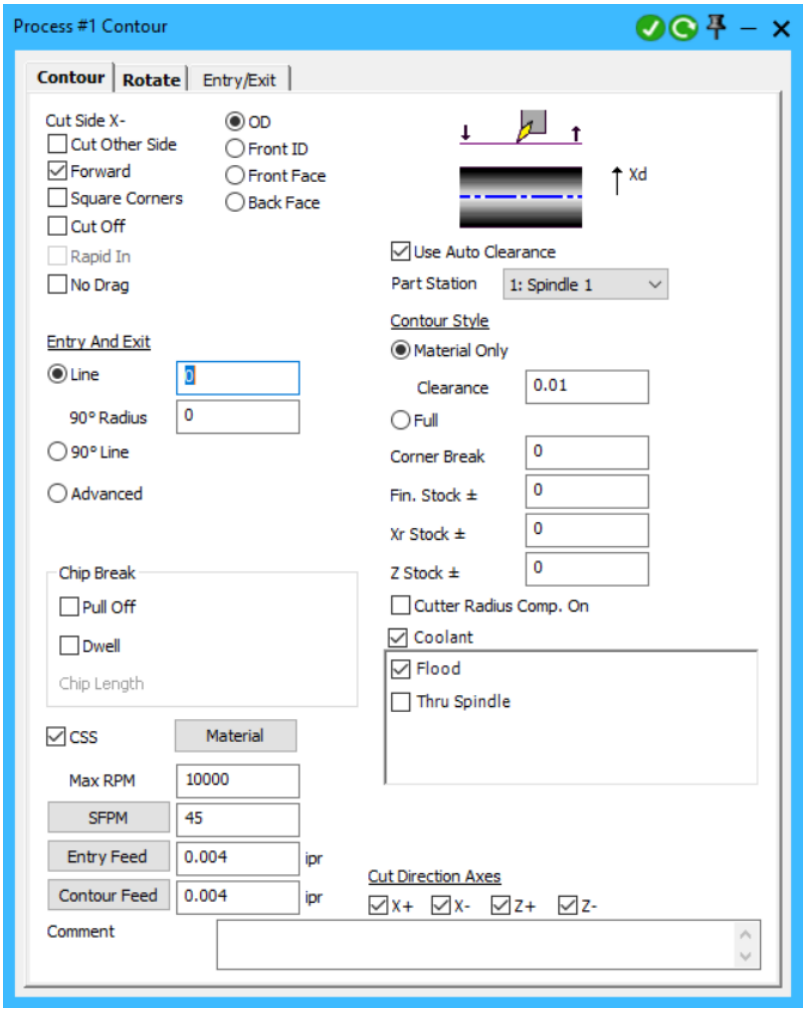


Click Do it (Ctrl+.).

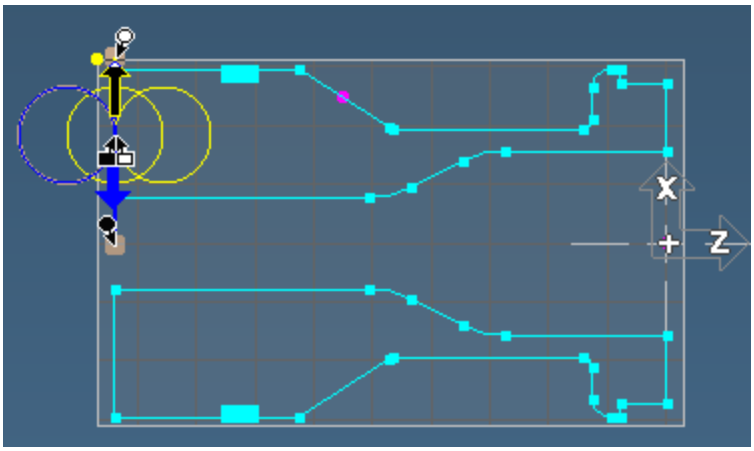
Cut Off

That completes the turning operations on the sub spindle. We now need to cut off the part.

1. Create a Lathe Contour process with the Cut Off tool (Tool #9) as shown.

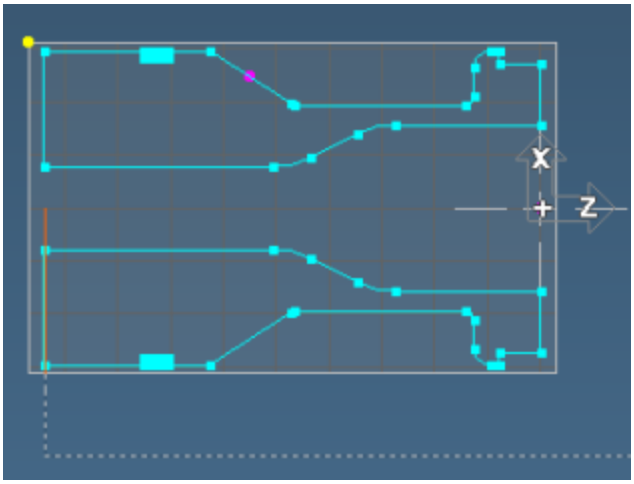


2. Set the machining markers as shown.



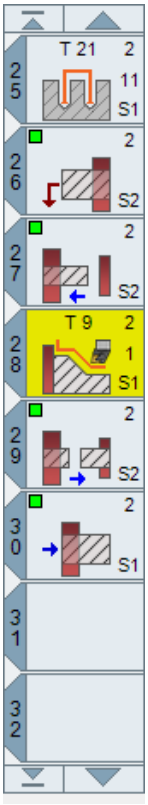
This will cut the back of the part.

3. Create the toolpath.



We will re-order the operations we have just created because we have created a sequence of operations that cuts off the part before the sub spindle actually grips it.

4. Drag the cutoff operation between Op #27 and Op #28.

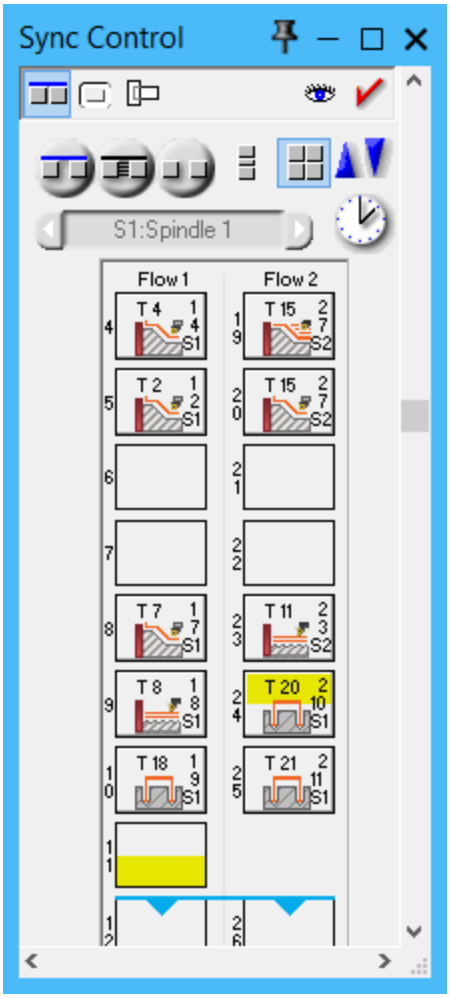


5. Open the Sync Control dialog.

An error message appears that says there is a spindle control conflict at Op 8. That is because we currently have turning and milling operations happening on Spindle 1 at the same time. When we moved the last two Holes operations we inadvertently caused this potential conflict.

To fix this we will sync the drilling operations. We will have the Tool Group 2 drill and tap wait for the Tool Group 1 tools to finish. That will fix the conflict and put our drilling operations in order.

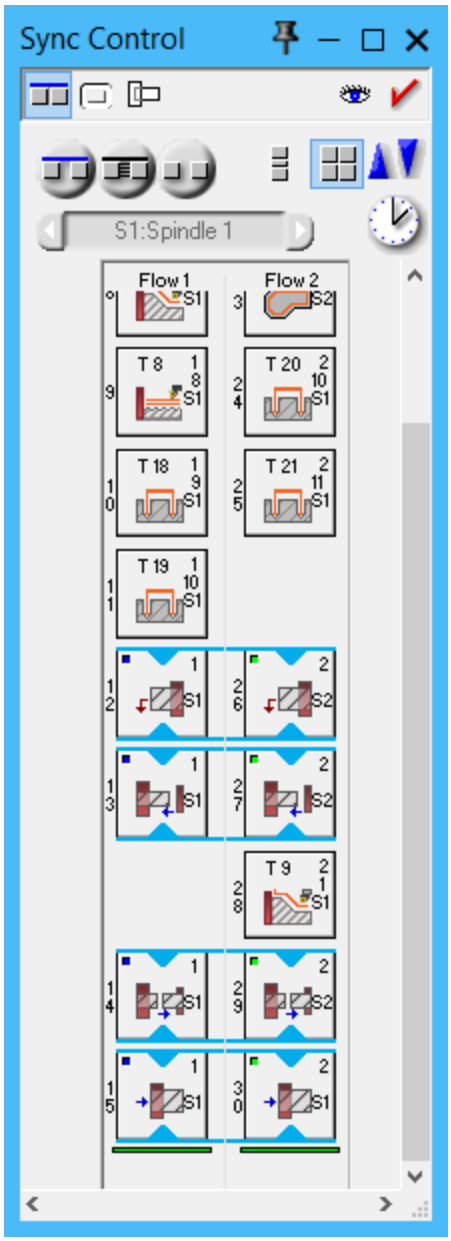
6. Switch to the uniform view.



This will allow you to select timeline scale so that you can see the operations at the end of both flows.

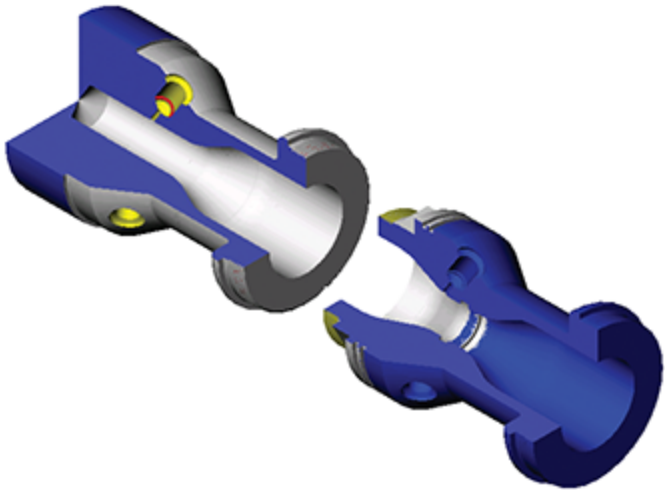
7. Select the Op end (bottom half) of Operation 11 and the Op Start (top half) of Operation 26.

8. `Click` the Operation Sync button.



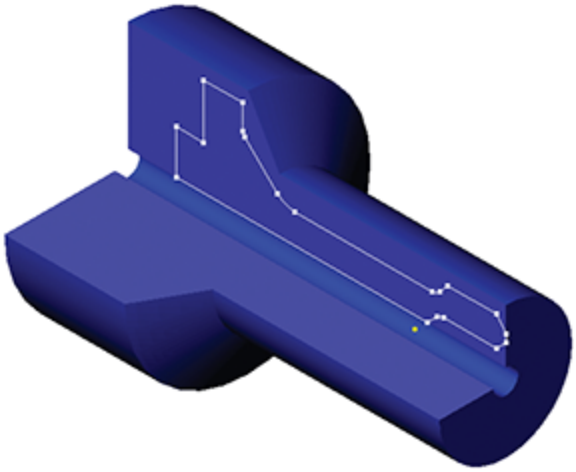
The TG2 drilling ops will now wait for the TG1 to finish.

9. Render the operations.



10. Save the part.

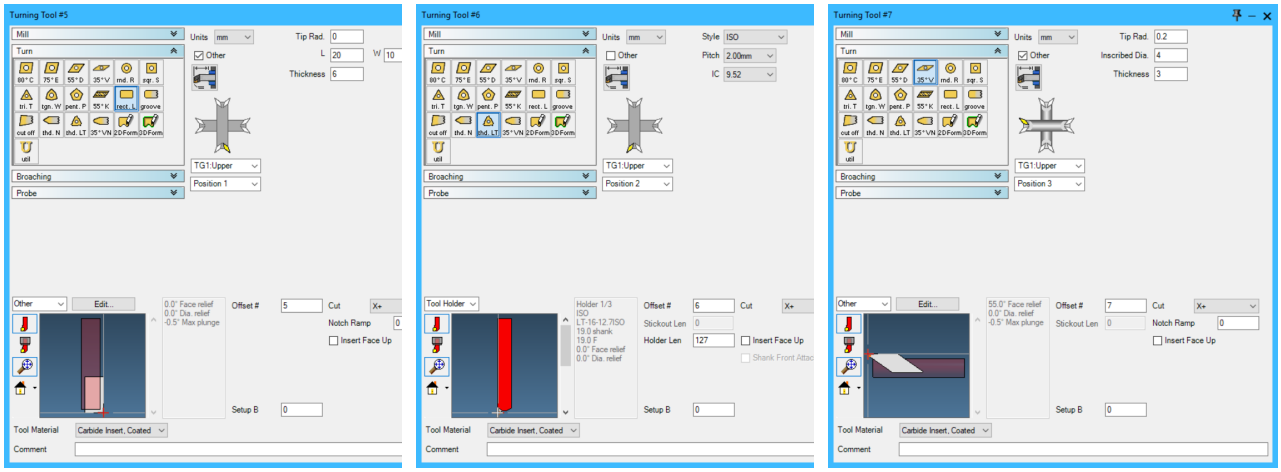
Auto-Flip



This exercise is intended to familiarize you with modifying tool information to automatically flip toolpath across X0. Some people find MTM to be easier or more familiar by setting up the part as if it were a regular lathe part, and then modify the operations. The exercise uses a pre-made part called Auto-Flip.vnc (For the part print for this exercise, see"Auto-Flip" on page 124). We will modify three of the tools to flip the existing toolpath from the +X side to the -X side.

We will modify the last three tools to flip the OD Plunge rough at the back of the part, the OD Thread and ID Contour operations. After we have flipped the operations, we will sync them.

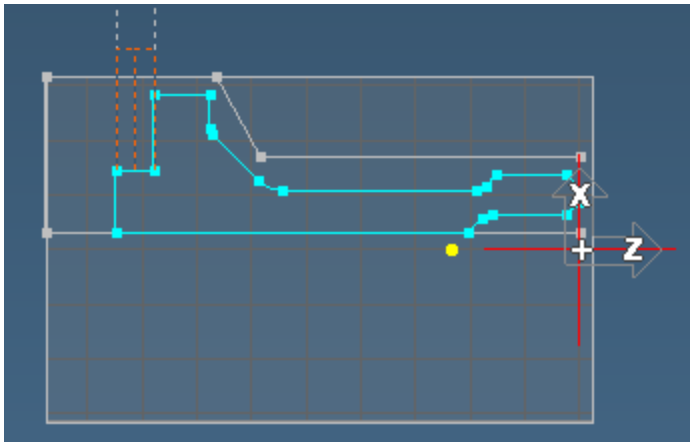
Looking closely at the last three tools in the list (tools 5, 6 and 7), you can see that each of the tools is in Tool Group 1. These tool positions also conflict other tools. Since these tools are not on a mini-gang of some sort, we have simply used these numbers for convenience since we will be changing the Tool Group these tools are assigned to.



Flipping Tool directions

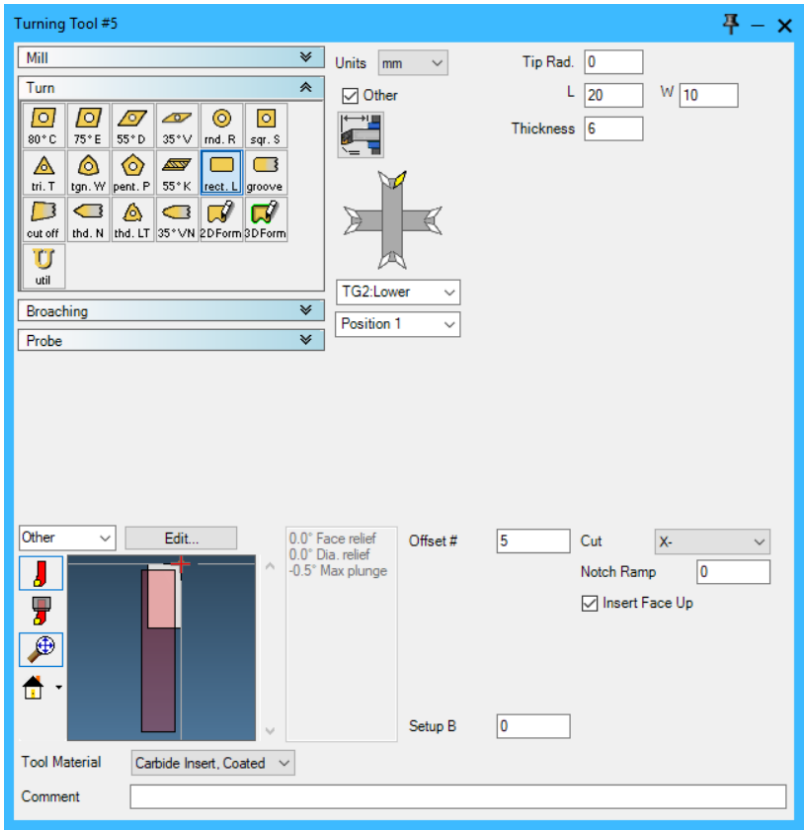
Upper to Lower

1. View the toolpath for Op #5.

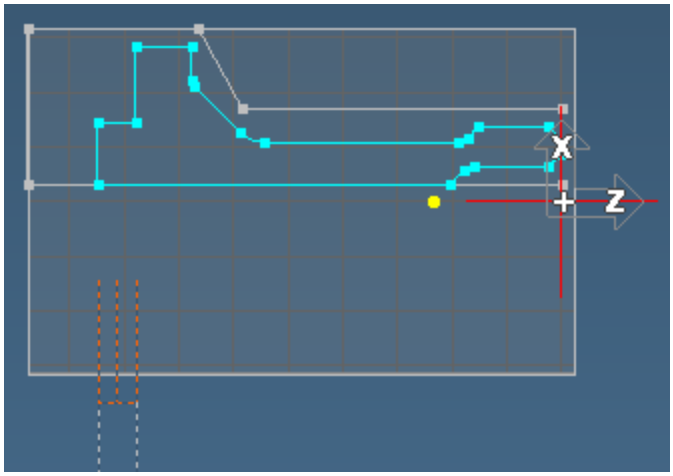


We are simply loading it so you can see the “before and after” results.

2. Modify tool #5 as shown.



3. Choose View > Redraw to redraw the toolpath. (Ctrl+R).

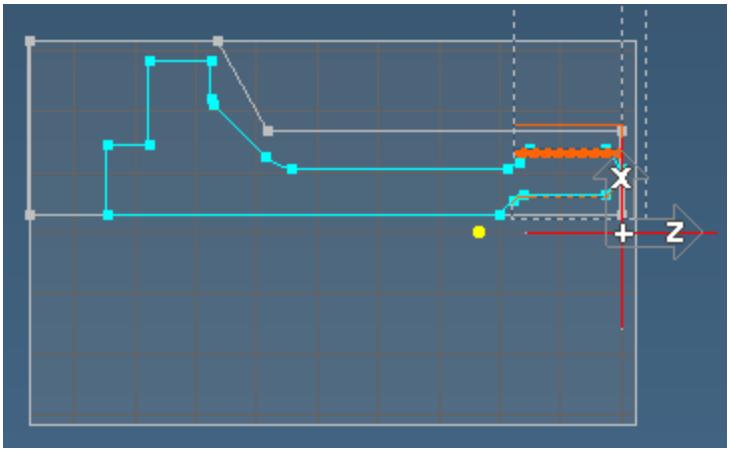


The toolpath is automatically updated to cut on the -X side.

Next we will update tool number 6 and 7.

4. Select Op #6 and Op #7.

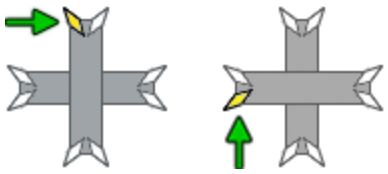
5. Choose View > Redraw. (Ctrl+R)



You can see that both operations cut on the +X side of the part.

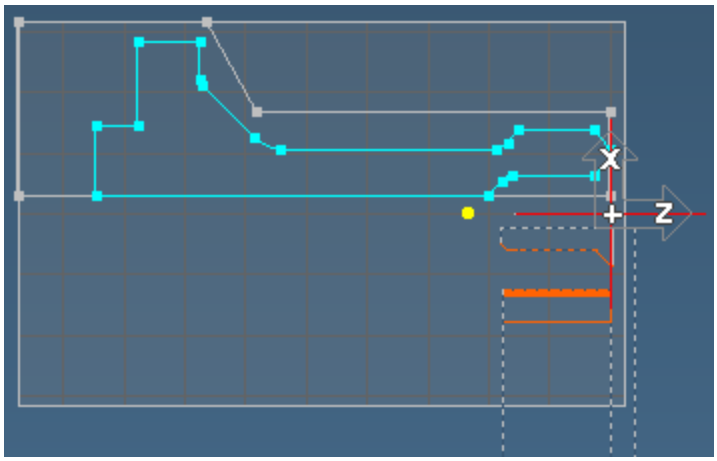
6. Modify the TG, cut side and insert settings of tool #6 and tool #7.

Make sure you choose Insert Face Up setting.



Tool #6	Tool #7

7. Redraw the workspace.



You can see that the toolpath is automatically updated to cut from the -X side.

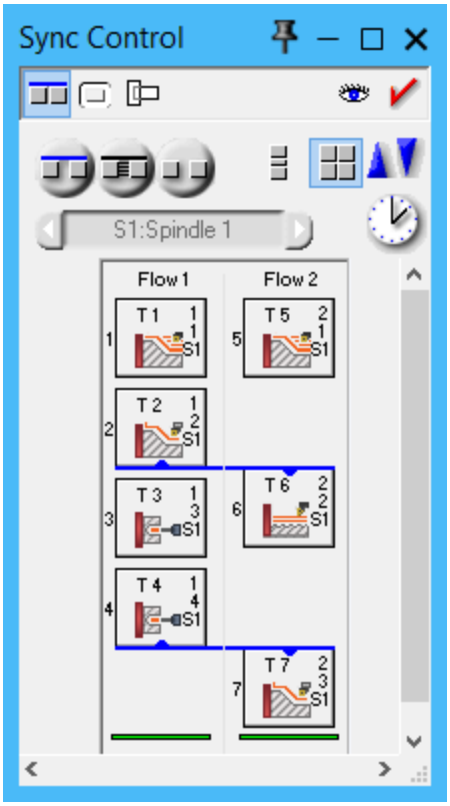
We now need to add two syncs to the part. The first will make Operation 6 start after Operation 2.

Syncing the new flow

1. Open the Sync dialog and `click` the Uniform Tile button. 
2. Create a sync at the end of Op #2 and the start of Op #6.

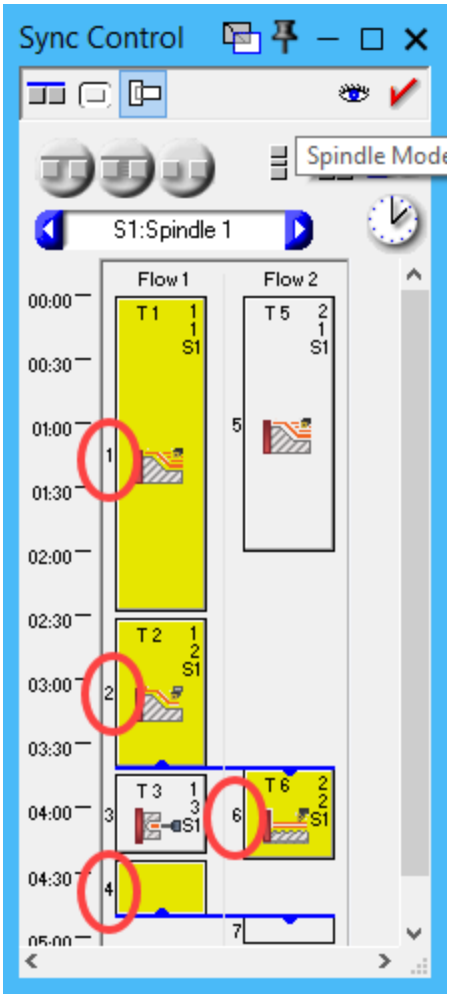
   The next sync will cause the ID Contour to wait for the drilling to finish.

3. Create a sync at the end of Op #4 and the start of Op #7.



Now that we have our syncs set we should check the run-time. Since we have different Tool Groups trying to control the spindle's RPM at the same time, we should specify which operations have control over the spindle speed.

4. Turn off Uniform View. 
5. Switch to the Spindle Mode. 



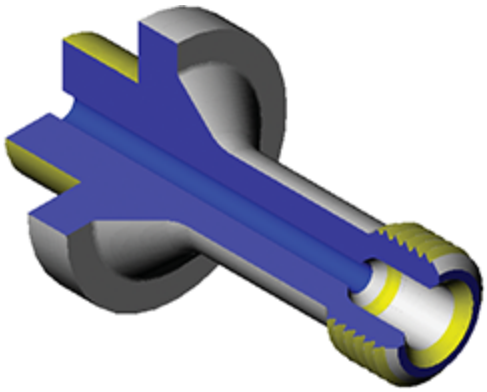
The highlighted operations currently have control over the spindle. We will now give Operations 1, 2, 4 and 6 control over the spindle speeds.

6. (`Ctrl+click`) Ops 1, 2, 4 and 6 and deselect any other Ops.

7. Recalculate the run-time. 

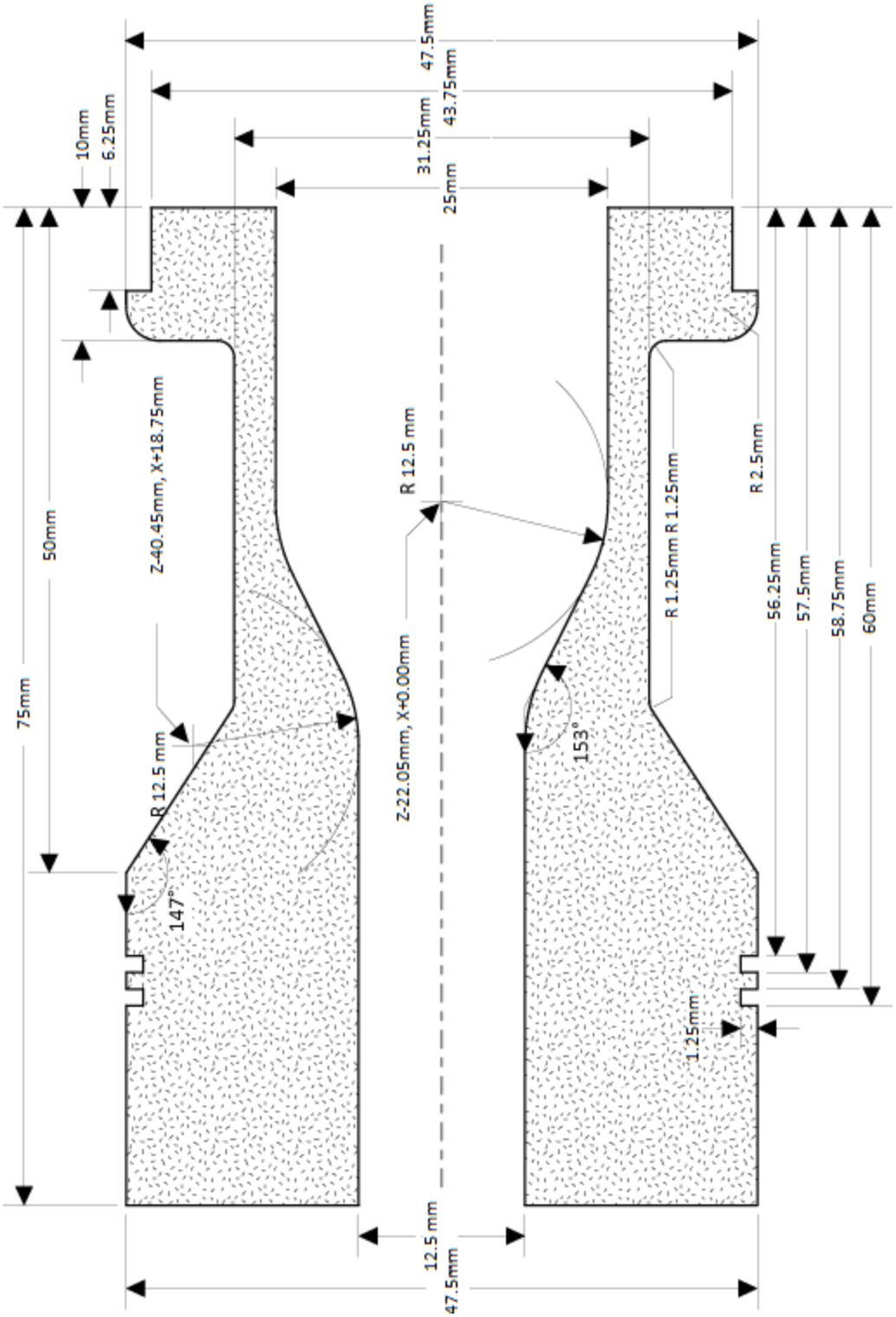
   The run-time should now take a little over five minutes.

8. Render the new operations.

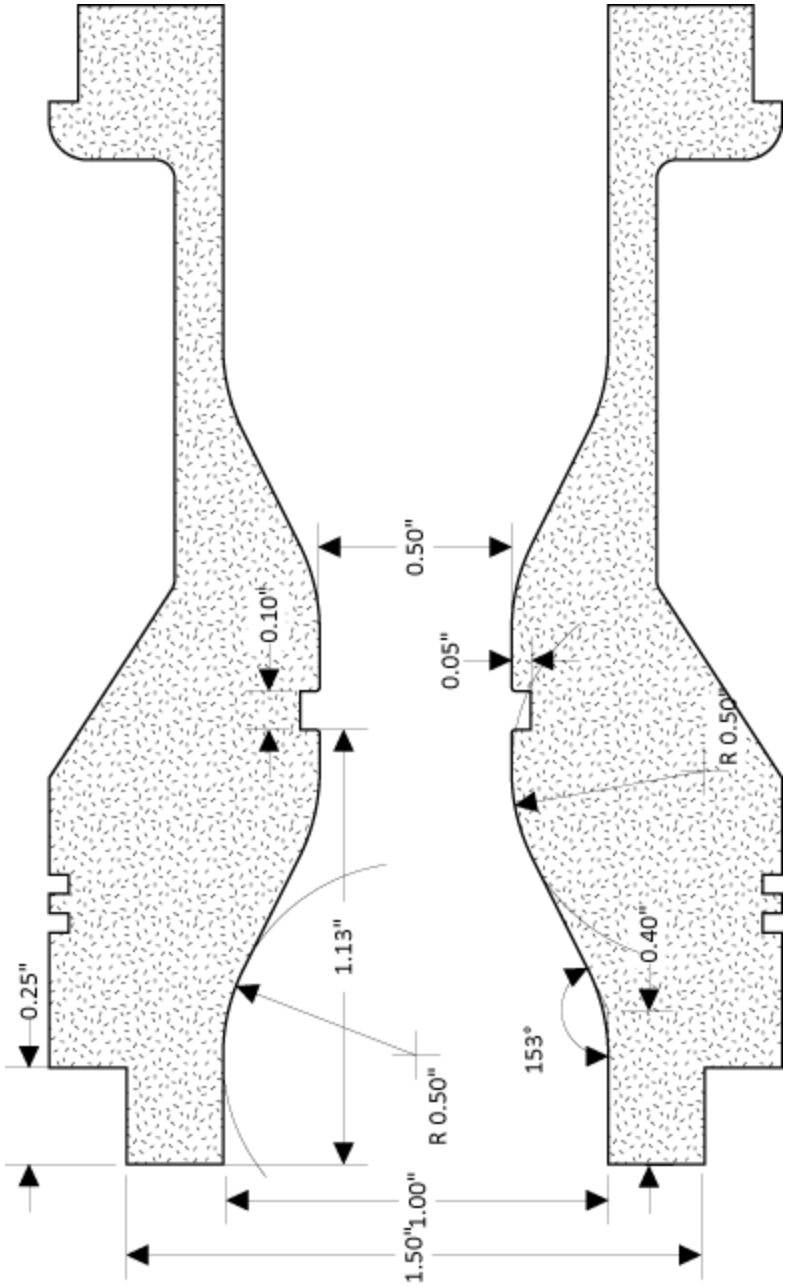


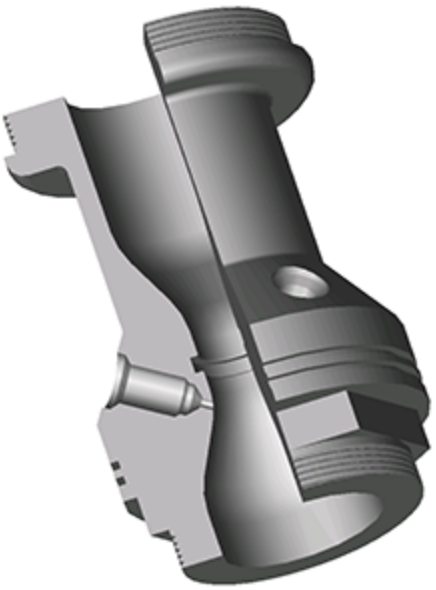
PART PRINTS

Main Spindle

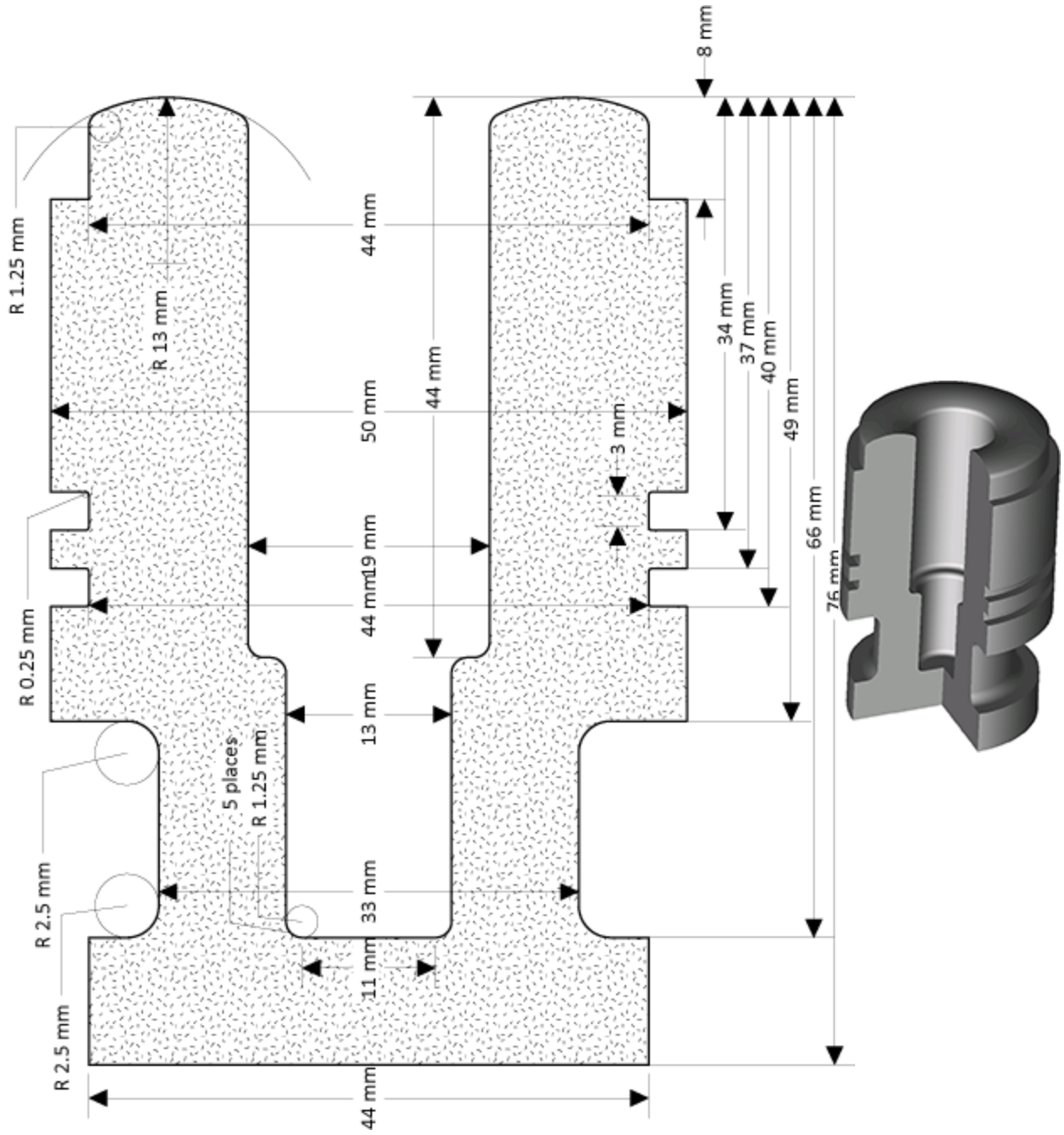


Sub Spindle

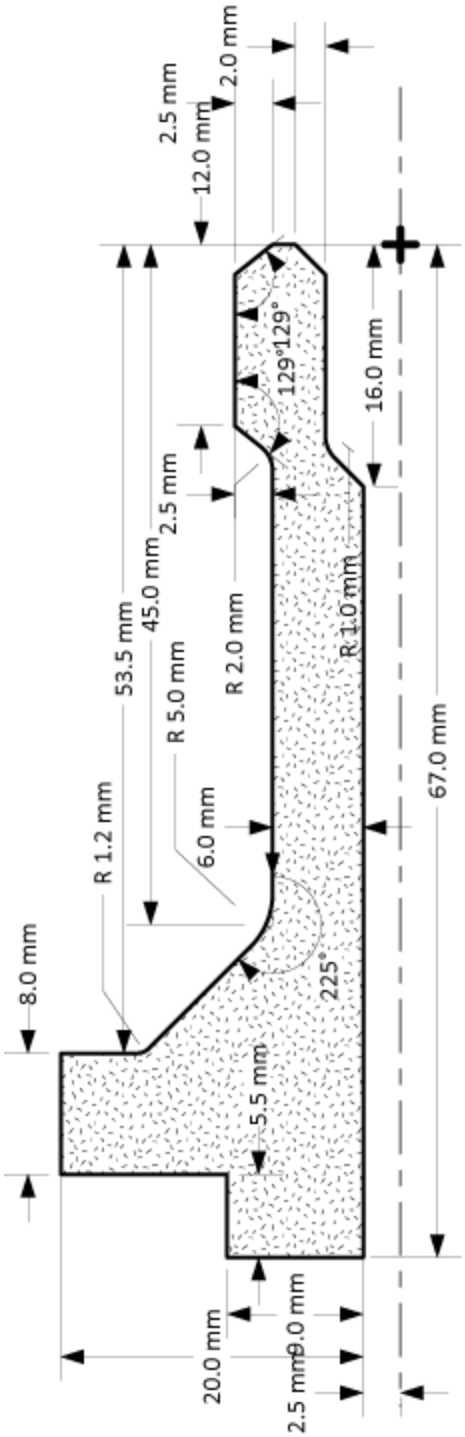




Single Spindle Part



Auto-Flip


2.0 mm
2.5 mm
12.0 mm
129°
16.0 mm
2.5 mm
45.0 mm
53.5 mm
R 5.0 mm
R 2.0 mm
R 1.0 mm
67.0 mm
R 1.2 mm
6.0 mm
8.0 mm
225°
5.5 mm
20.0 mm
9.0 mm
2.5 mm