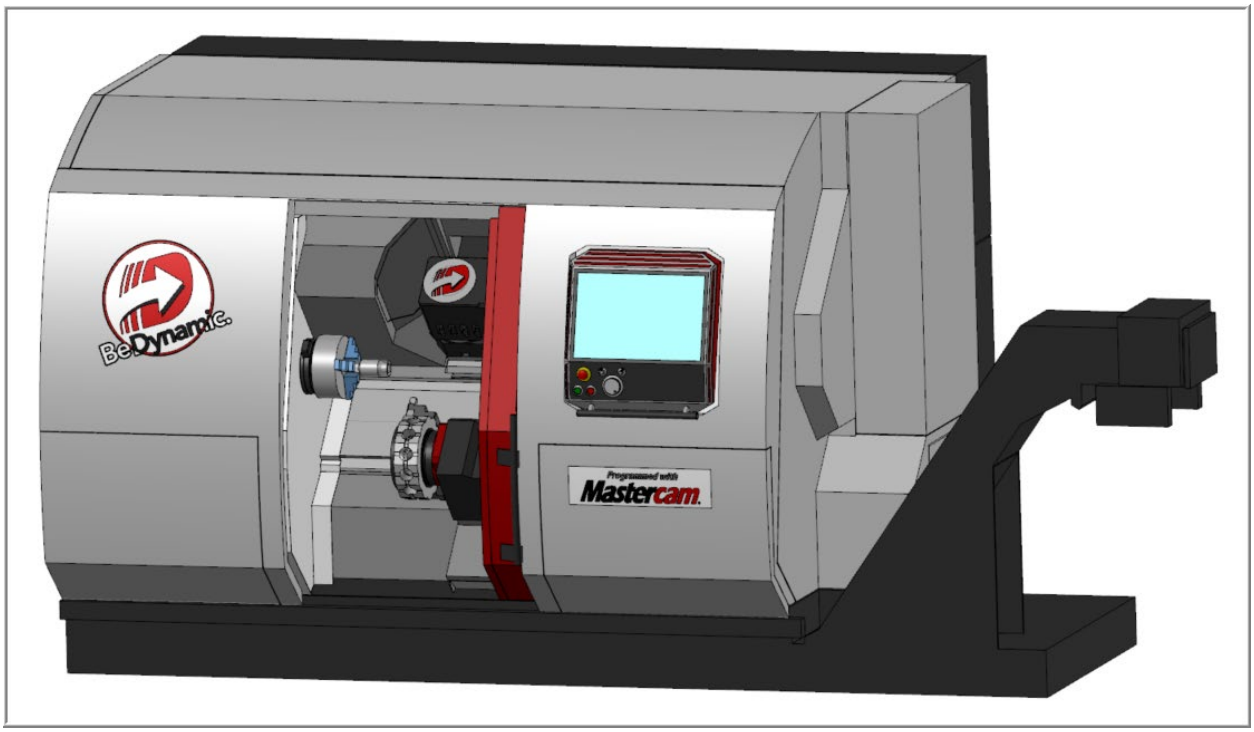
Mastercam 2023

Training Guide



Mill-Turn Lesson-2

camInstructor

Objectives

This lesson will add to what we learned in Lesson 1 by using some simple milling on both the main and sub spindles. We will still use some of the turning techniques from lesson 1 along with the POCO and syncing.

Rough net shape by turning:
Mastercam will create a turning profile of the net part shape we can use to rough the hex with a turning tool.

Job setup:
Make adjustments in the Job Setup for Mill-Turn.

Main spindle turning:
Face with lower turret.
OD rough with upper B axis head.
OD finish with lower turret.

Main spindle milling:
Machine the hex with C-Axis rotation.
Spot drill thru hole with milling spindle.
Drill thru hole with turning spindle.

POCO (Pick Off Cut Off):
Transfer the part to the sub spindle.

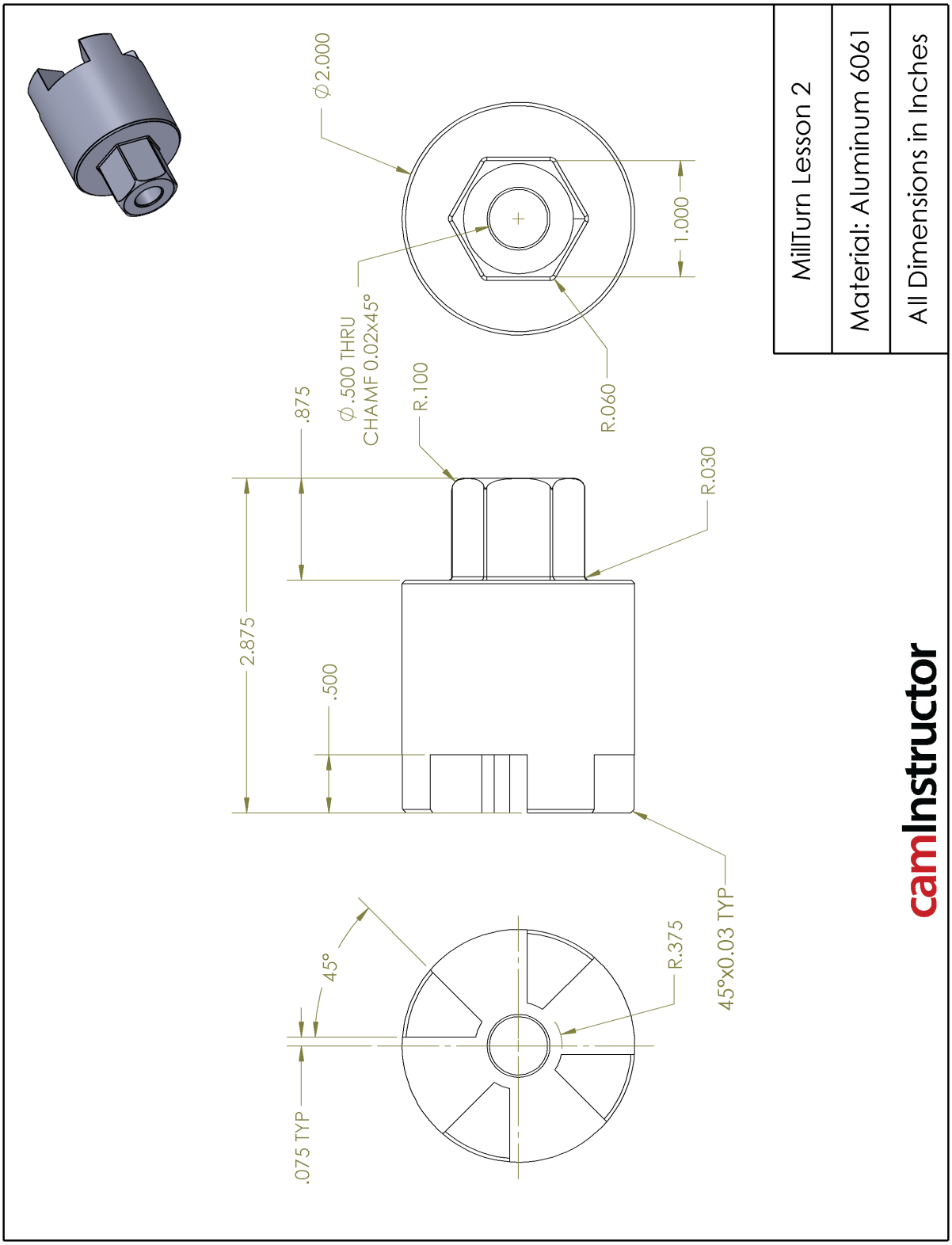
Sub spindle turning:
Face with lower turret.
Rough and finish the pocket with C-Axis rotation

Syncing:
Syncing ops to create efficient programs and avoid crashes.

Simulation:
Final verification of the machining process using complete machine verification.

Posting:
Additional settings before posting and view code in dual stream

MILL-TURN LESSON-2 DRAWING



MILL-TURN LESSON-2- THE PROCESS

TASK 1: Import the solid model and select the machine type

TASK 2: Job setup

TASK 3: Main spindle turning

TASK 4: Main spindle milling

TASK 5: POCO (**P**ick **O**ff **C**ut **O**ff)

TASK 6: Sub spindle turning and milling

TASK 7: Machine syncing

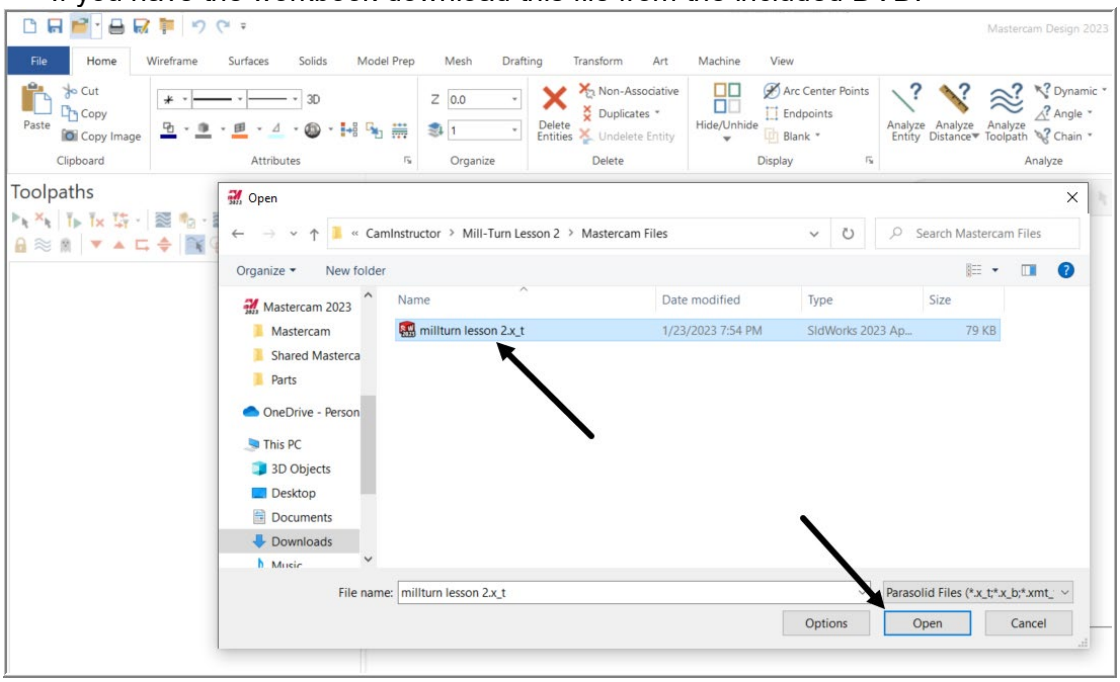
TASK 8: Machine simulation

TASK 9: Posting code

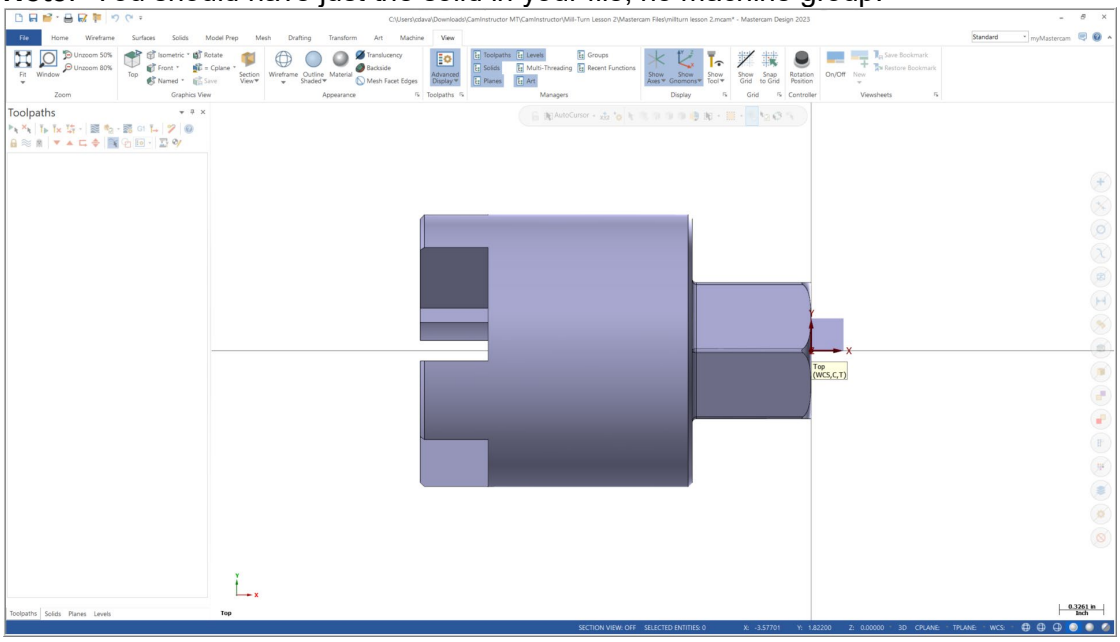
TASK 1:
IMPORT THE SOLID MODEL AND SELECT MACHINE TYPE

➲ We will start Mastercam in a **blank document** then open our solid model followed by selecting our Mill-Turn machine from the Machine Type menu.

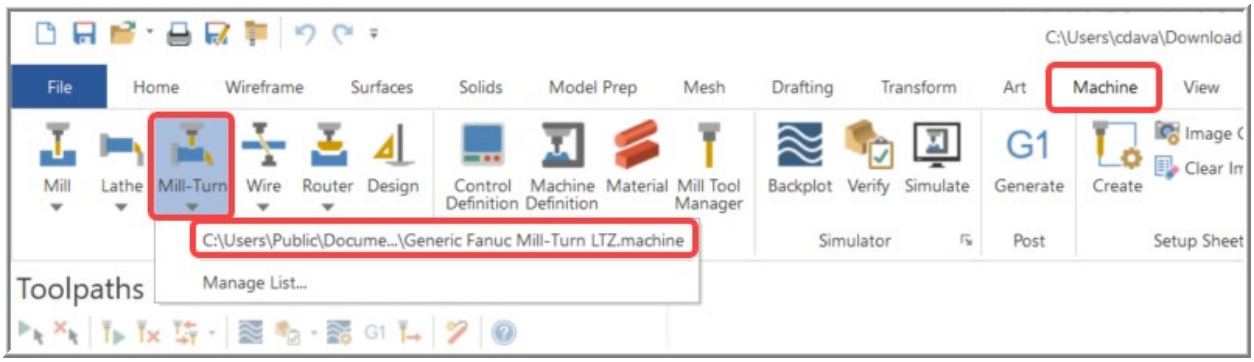
1. Launch Mastercam.
2. Open the solid for Lesson 2. - **millturn lesson2.x_t**
   If you have the online course download this file from the website.
   If you have the workbook download this file from the included DVD.



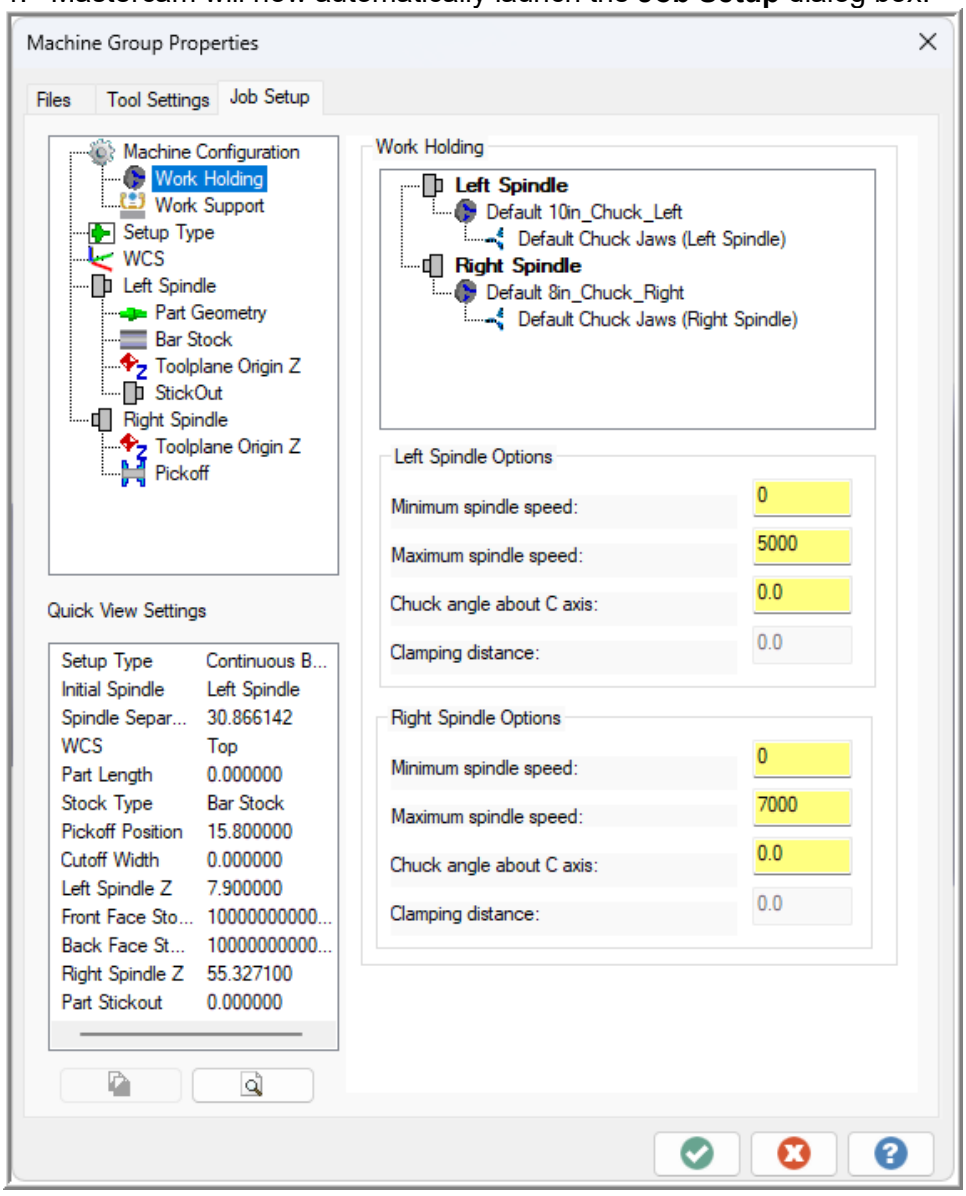
Note: You should have just the solid in your file, no machine group.



3. Select the Mill-Turn machine. **Machine > Mill-Turn > Generic Fanuc Mill-Turn LTZ.machine**. Mastercam will now launch Mastercam Code Expert which we will be using later.



4. Mastercam will now automatically launch the **Job Setup** dialog box.



TASK 2:
JOB SETUP

➲ In this task you will work through the Job Setup dialog.

To simplify job setup, Mastercam Mill-Turn uses a top to bottom series of steps. The order of these steps serves two main purposes.

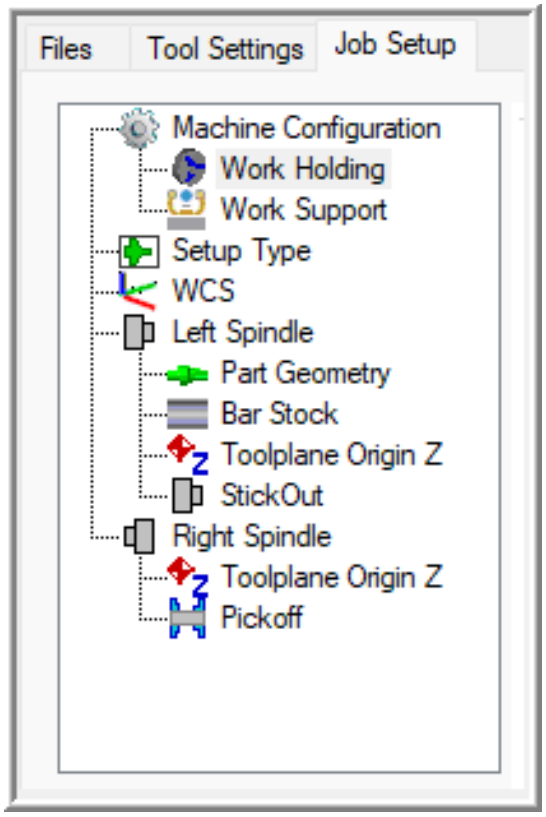
1. Makes it easier to for the end user and guide them through the process.
2. Help Mill-Turn automate the display of appropriate and logical parameters based on previous settings.

This interface workflow reduces the chance that a user will enter a parameter that does not make sense and limits instances of error.

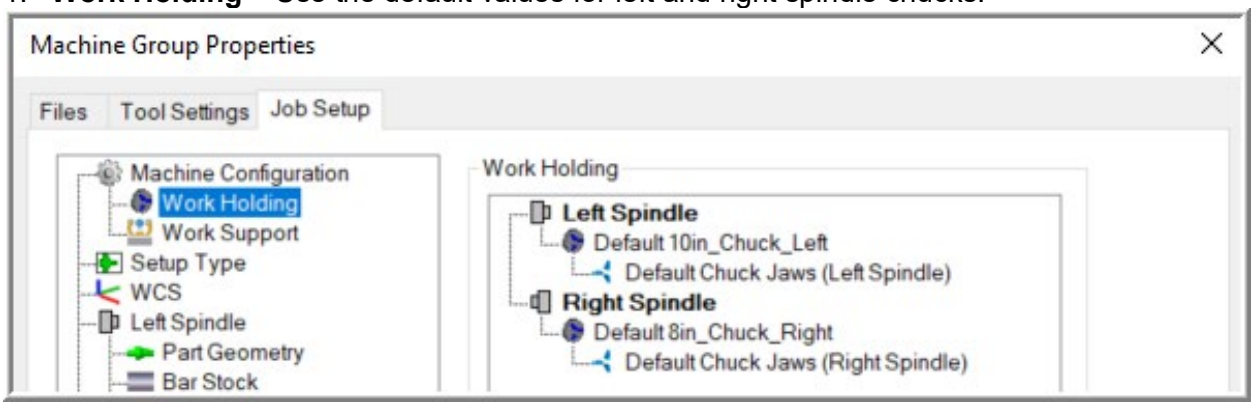
The general Job Setup workflow:

- ➲ Work Holding – Select chucks, jaws, and collets
- ➲ Work Support – Select steady rest when applicable
- ➲ Setup Type – Assign the spindle the work will start in and part handling
- ➲ WCS – Sets the Work Coordinate System
- ➲ Left Spindle
  - Part Geometry – Define your part
  - Bar Stock – Define the type of stock
  - Toolplane Origin Z – Set toolplane origin based on part geometry
  - Stick Out – Define bar stock projection from chuck face or jaws
- ➲ Right Spindle
  - Toolplane Origin Z – Set toolplane origin based on part geometry
- ➲ Pickoff – Set pickoff position and tool for cut off

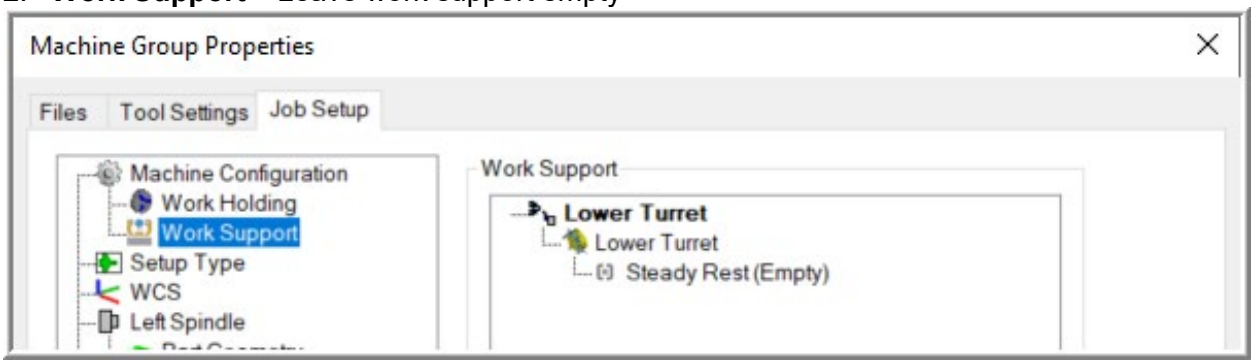
Note: Left spindle and right spindle order are determined by the initial spindle selection under Setup Type.



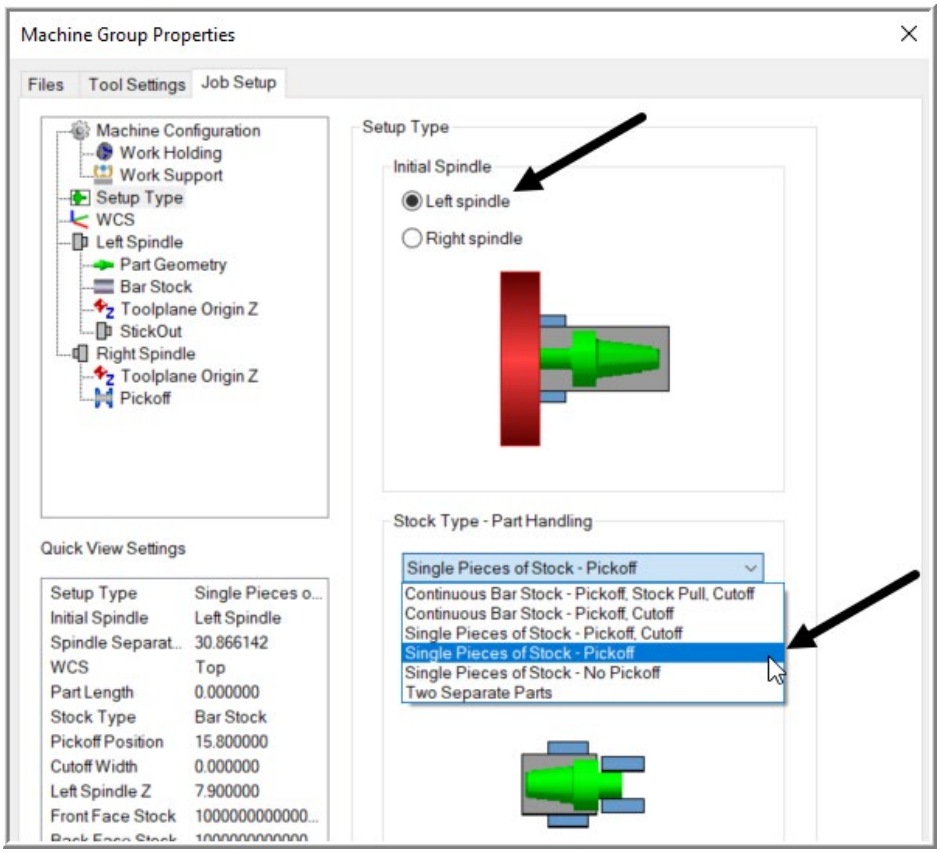
1. **Work Holding** – Use the default values for left and right spindle chucks.



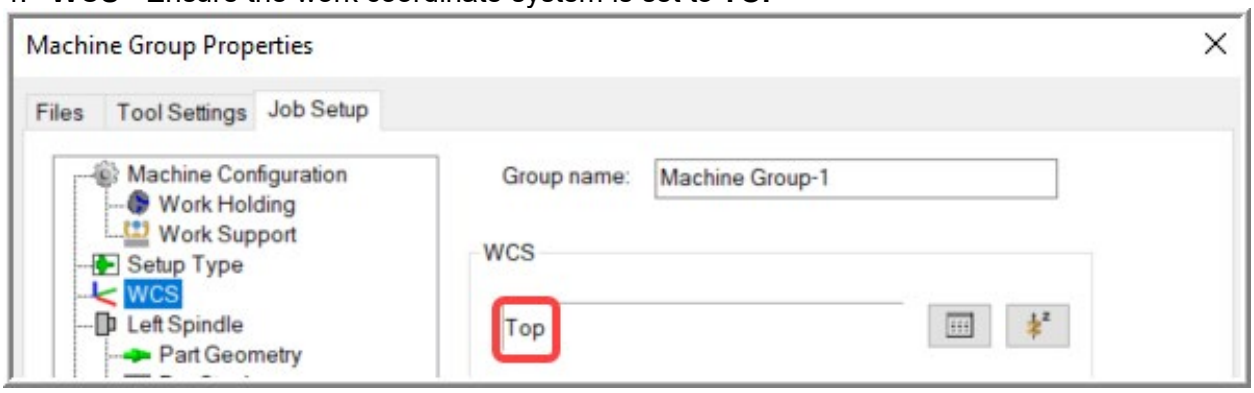
2. **Work Support** – Leave work support empty



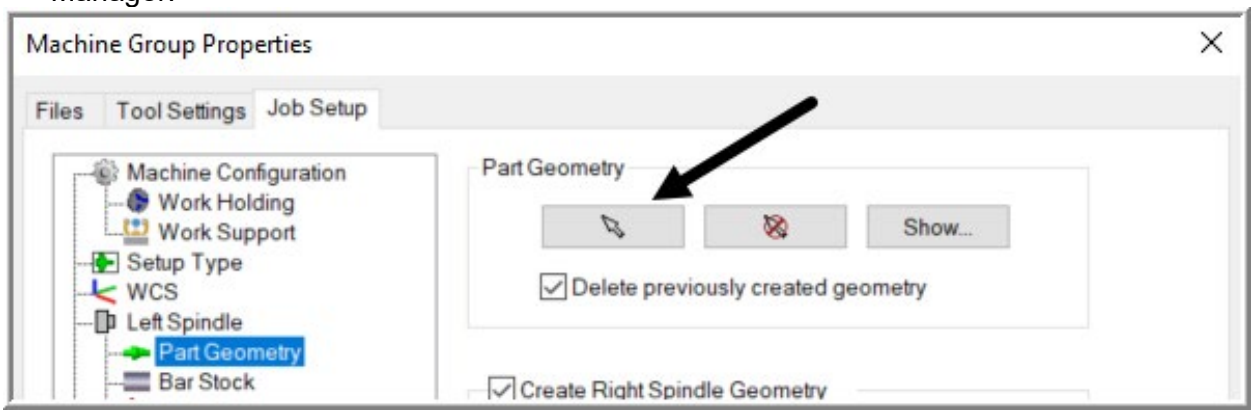
3. **Setup Type** – Initial Spindle set to **Left** and Stock Type – Part Handling set to **Single Pieces of Stock - Pickoff**.



4. **WCS** - Ensure the work coordinate system is set to **TOP**

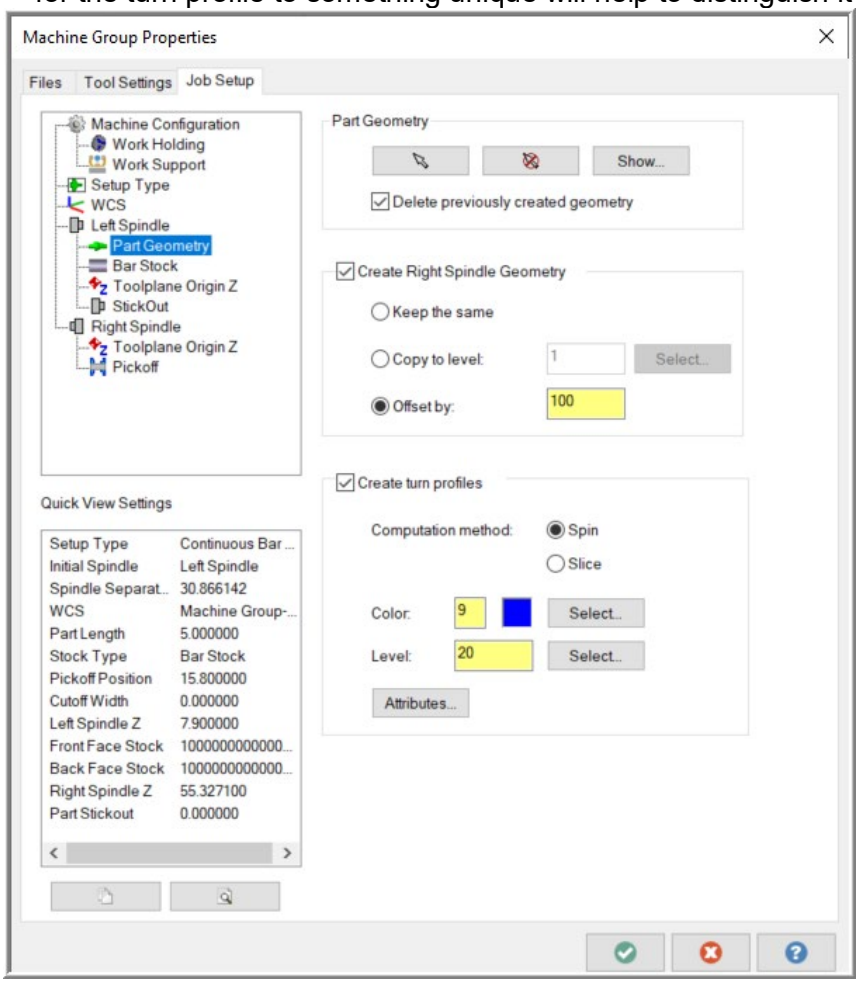


5. **Part Geometry** – Select your part model. Mastercam will pull dimensions from the model for other Job Setup settings. The model will also be used in simulation and in the Sync Manager.



6. **Select the solid** by clicking on it. Hit End Selection. 

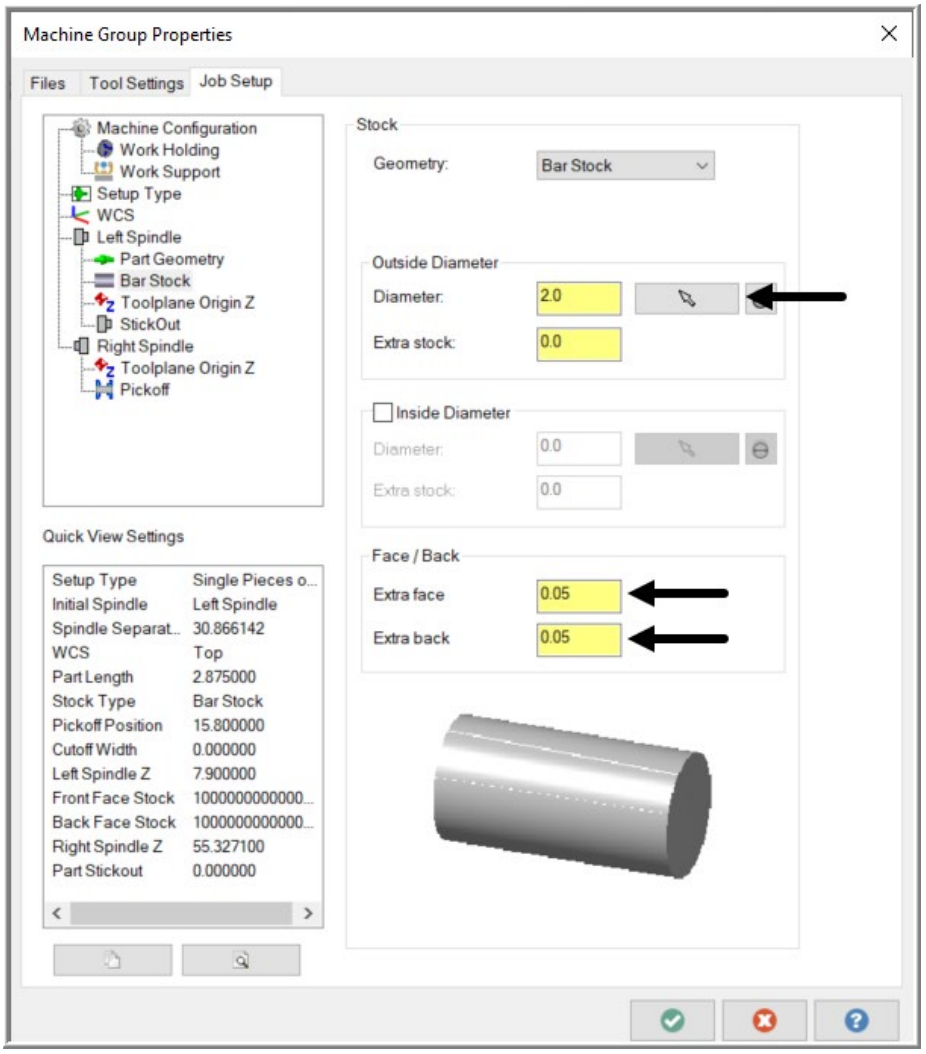
7. **Enable** the options to **Create Right Spindle Geometry** and **Create turn profiles.** For the geometry, select **Offset by** and enter **100**. This will put this model on the same level as the original model +100. So, if the original is on level 1 the created model will be on 101. For the **turn profile**, set computation method to **Spin** and set the **Level to 20**. Changing the color for the turn profile to something unique will help to distinguish it later.



Spin: The part geometry is rotated around an axis. The profile is generated from the furthest extents of the spin.

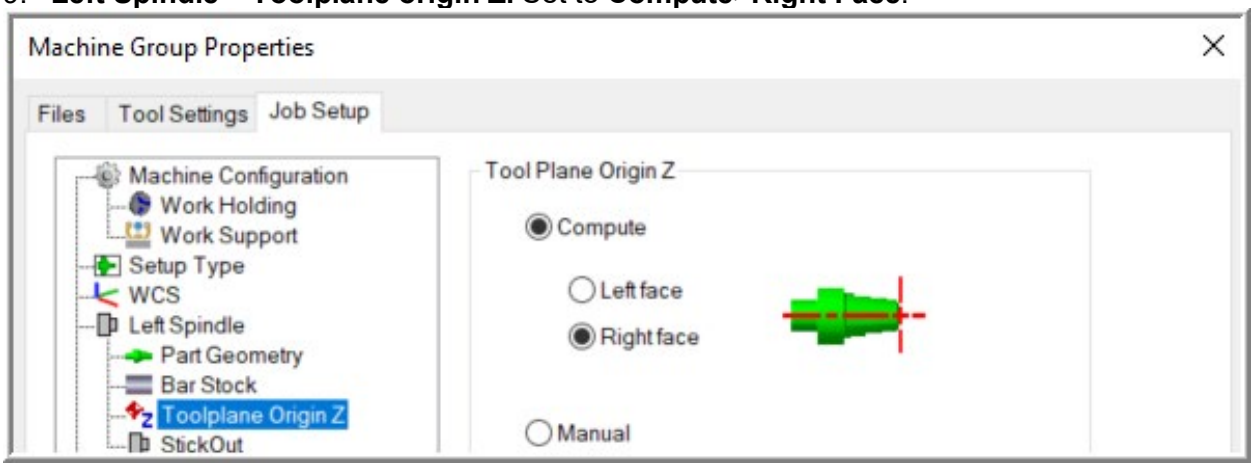
Slice: The part geometry is sliced in the lathe ZX (WCS XY) plane. The profile is limited to what the slice encounters and may not include the full extents of the part geometry.

8. **Bar Stock** – You can set your **stock type**, **number of parts**, **outside diameter**, **inside diameter** and amount of **extra stock** here. Mastercam will read the OD from the model and round up to the next closest stock size. We are not going to turn the **OD** so set it back to **2.00**. Add **0.05 extra stock** to the **face** and to the **back**.

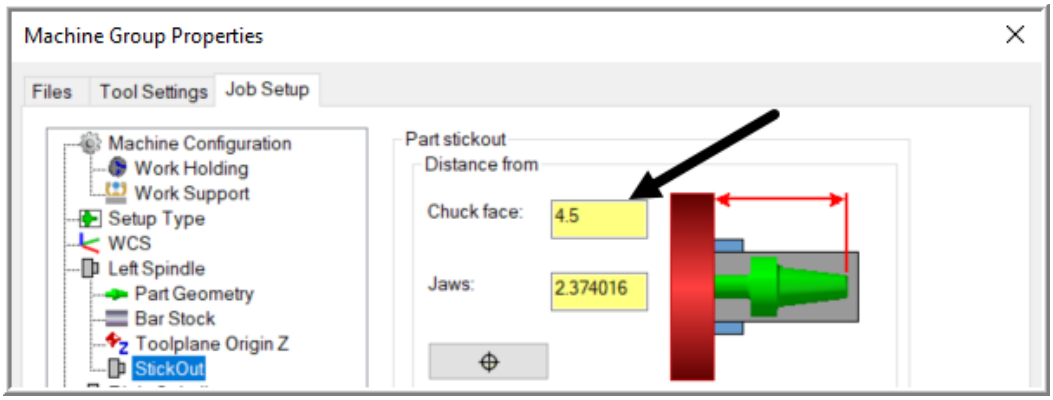


Geometry selection is based on previous choices made in Job Setup. Since we have selected Single Pieces of Stock – Pickoff in setup type we will use bar stock geometry.

9. **Left Spindle – Toolplane origin Z.** Set to **Compute>Right Face**.

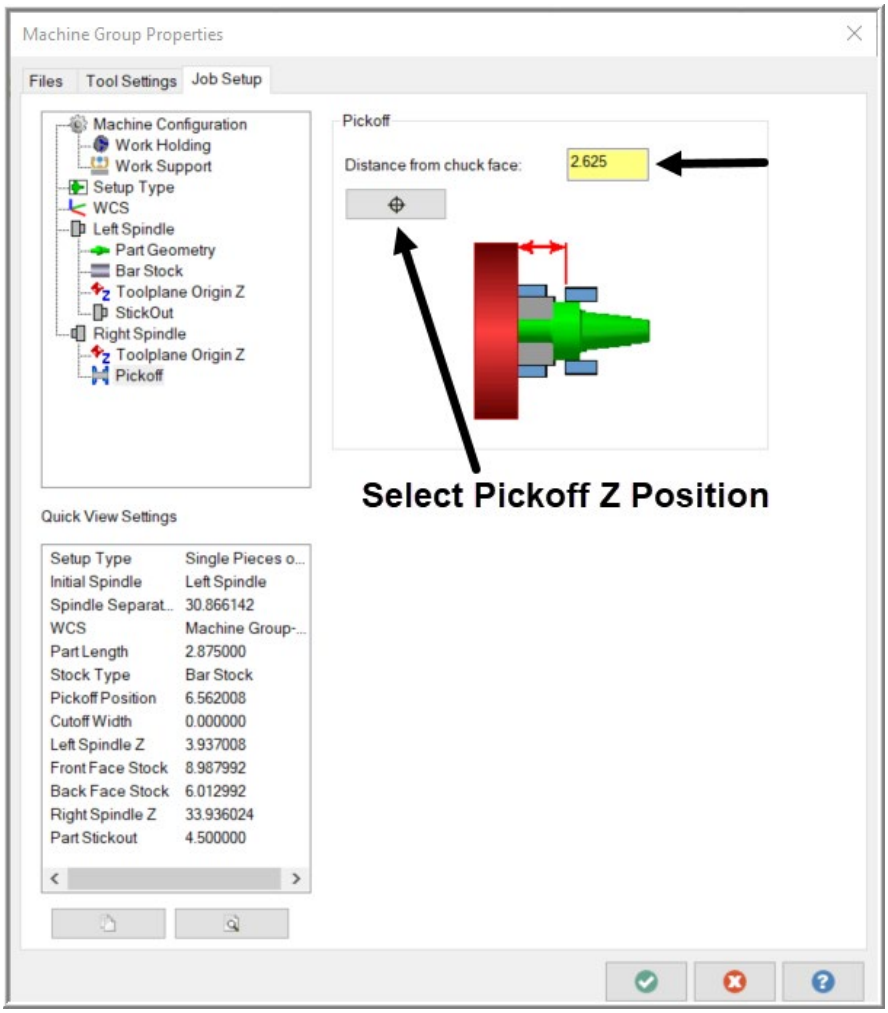


10. **Stickout** is calculated from the face of the chuck or jaw. We are using a single piece of stock so we need to leave enough material sticking out for the pickoff. Part length is 2.875 with 0.05 stock on front and back face. Use a part Stickout of **4.5**. Enter this value in the **Chuck Face:** box.

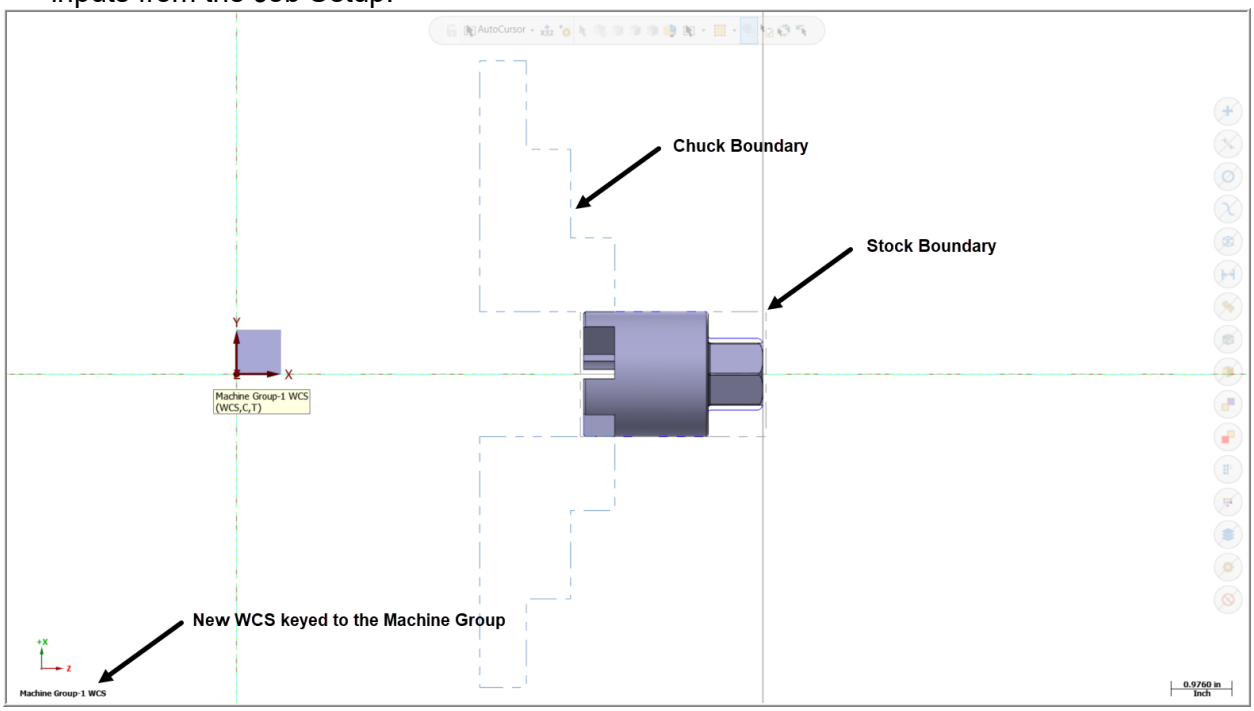


11. **Pickoff** – We need to calculate our pick off point. The hexagon on the right is 0.88 long, plus the chamfer of 0.030 so we need to be sure to stay clear of that. We should be safe grabbing the part close to the mid-point. Select the midpoint of the 2.00" diameter stock. **Distance from chuck face** should be **2.625**.

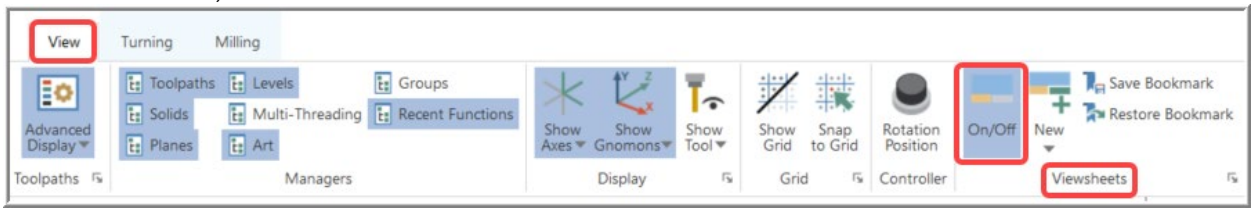
Hit OK 

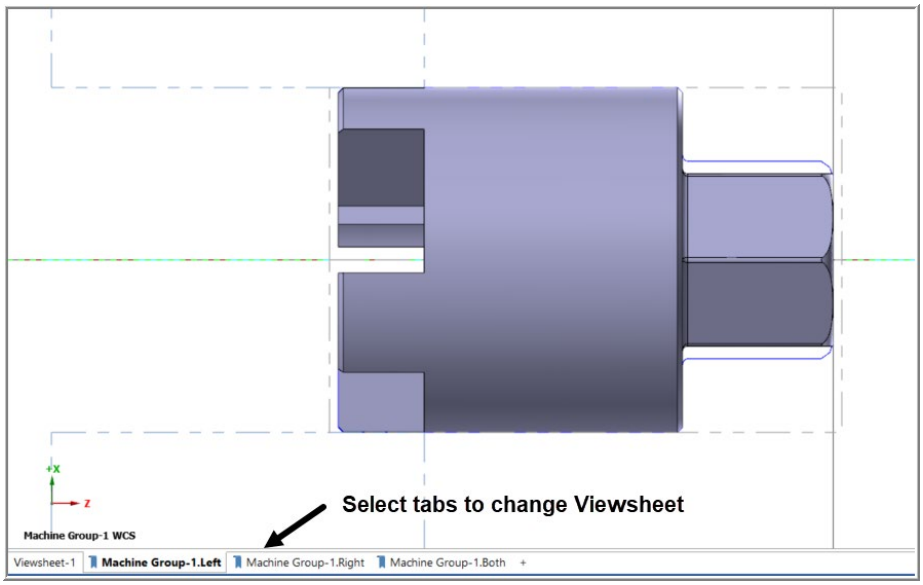


12. **Review of Job Setup Results.** Look at what Mastercam has done so far because of the Job Setup inputs. Mastercam creates a **new WCS** which is a copy of WCS-Top and has the origin set at machine zero as defined in your machine setup. You can also see **chuck** and **stock boundaries** on **both spindles** as well as the new solid. These are generated using inputs from the Job Setup.

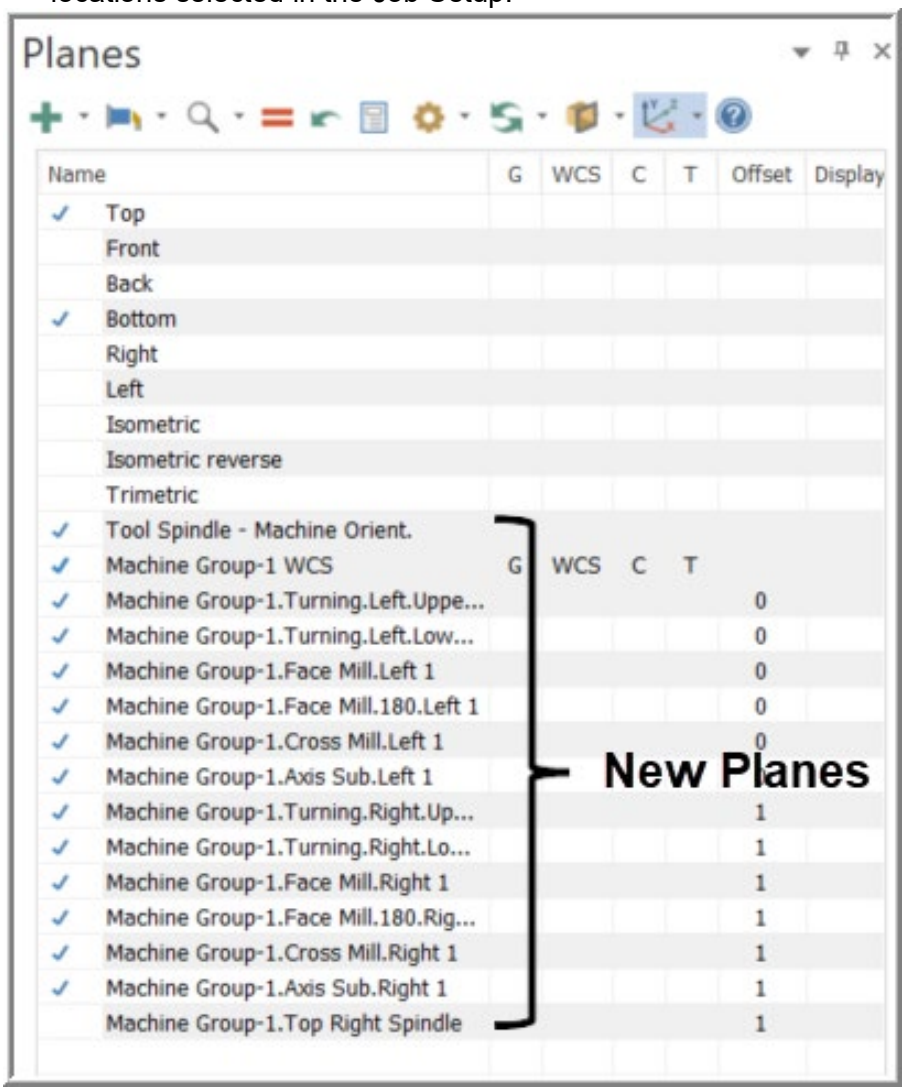


13. Mastercam will also **create view sheets** for each spindle. You may need to turn viewsheets on. To do so, **View > Viewsheets > On/Off**

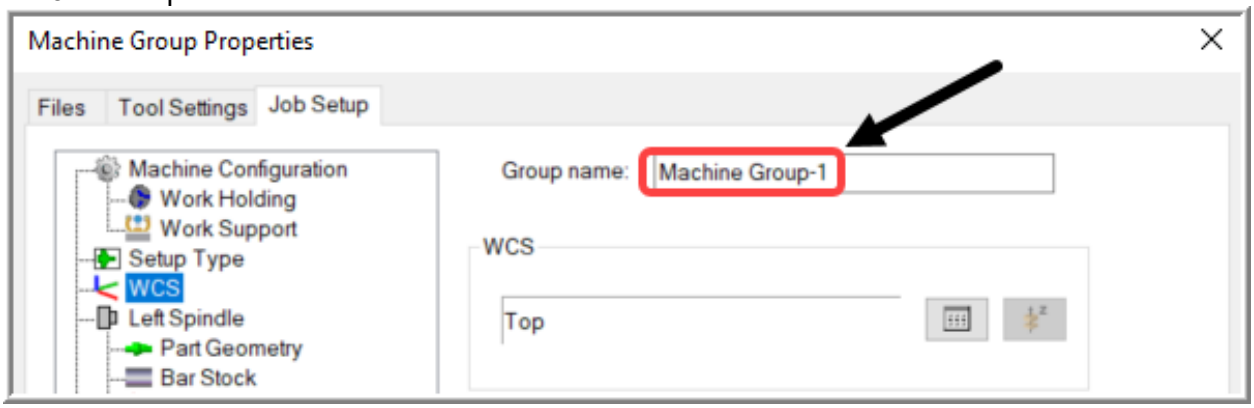




14. Mastercam also **creates planes** that relate to standard programming orientations for both the main (left) and sub (right) spindles. The origins of these planes are keyed to the locations selected in the Job Setup.



15. The **names** of the views, WCS and Viewsheets are **keyed** to the **Group name** entered in Job Setup.

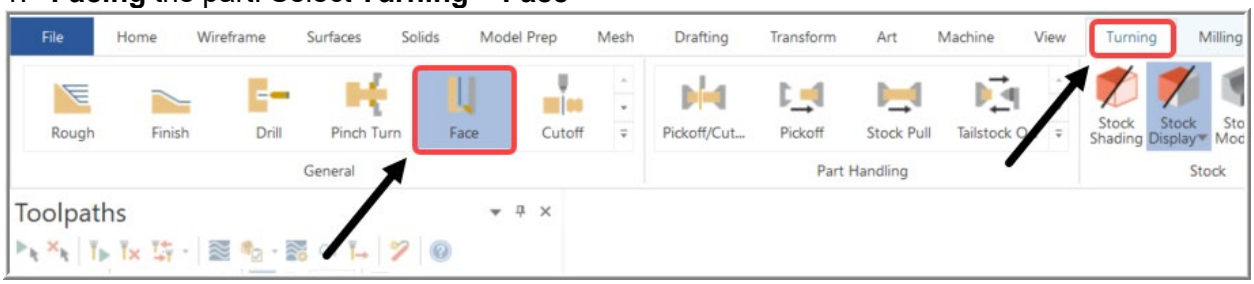


16. **Save** your file as MillTurnLesson2

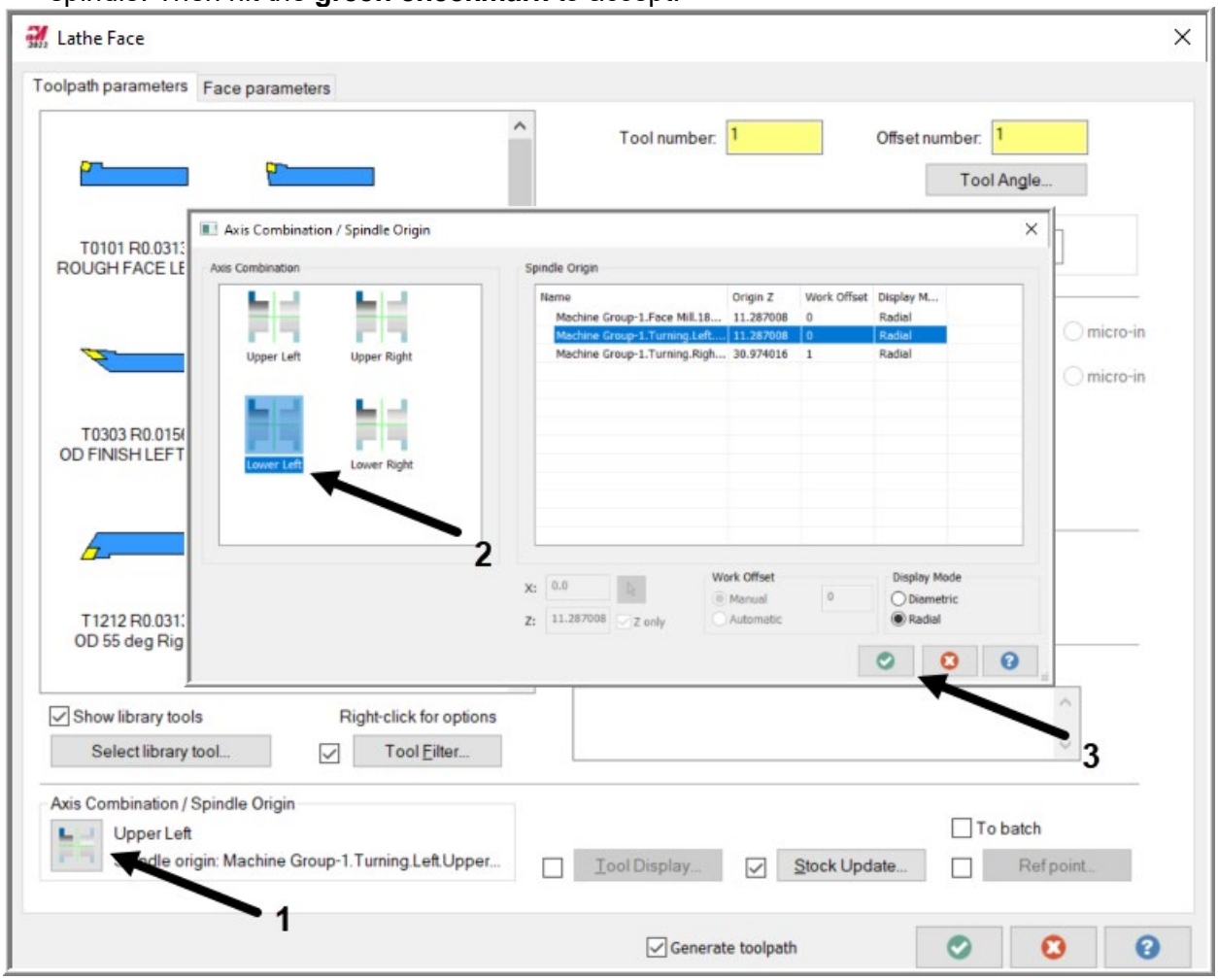
TASK 3:
MAIN SPINDLE TURNING AND THRU HOLE DRILLING

➲ In this task you will face the part and turn the OD profile in the main (left) spindle.

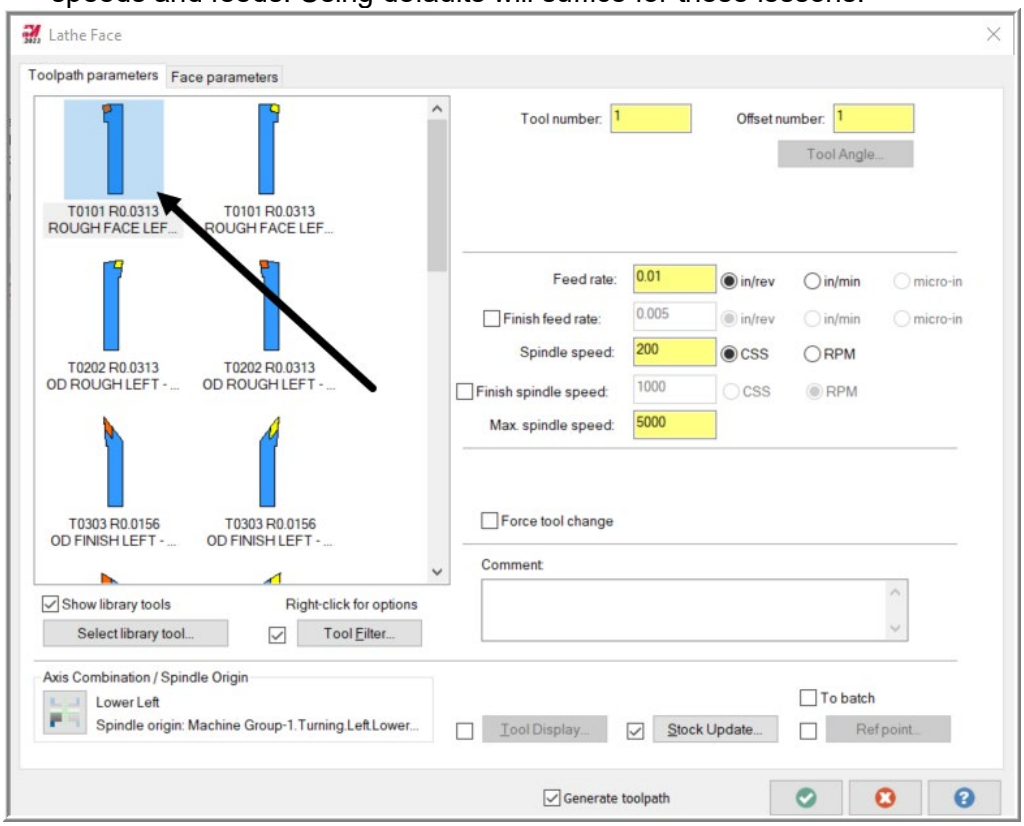
1. **Facing** the part. Select **Turning > Face**



2. When the **Lathe Face Properties** window opens, **select Axis Combination / Spindle Origin** by clicking the small graphic. This will open the Axis Combination / Spindle Origin window. **Select Lower Left** as we will be using the lower turret to turn on the main (left) spindle. Then hit the **green checkmark** to accept.



3. On the **Lathe Face Properties** window choose **T0101**. We will not spend time adjusting speeds and feeds. Using defaults will suffice for these lessons.

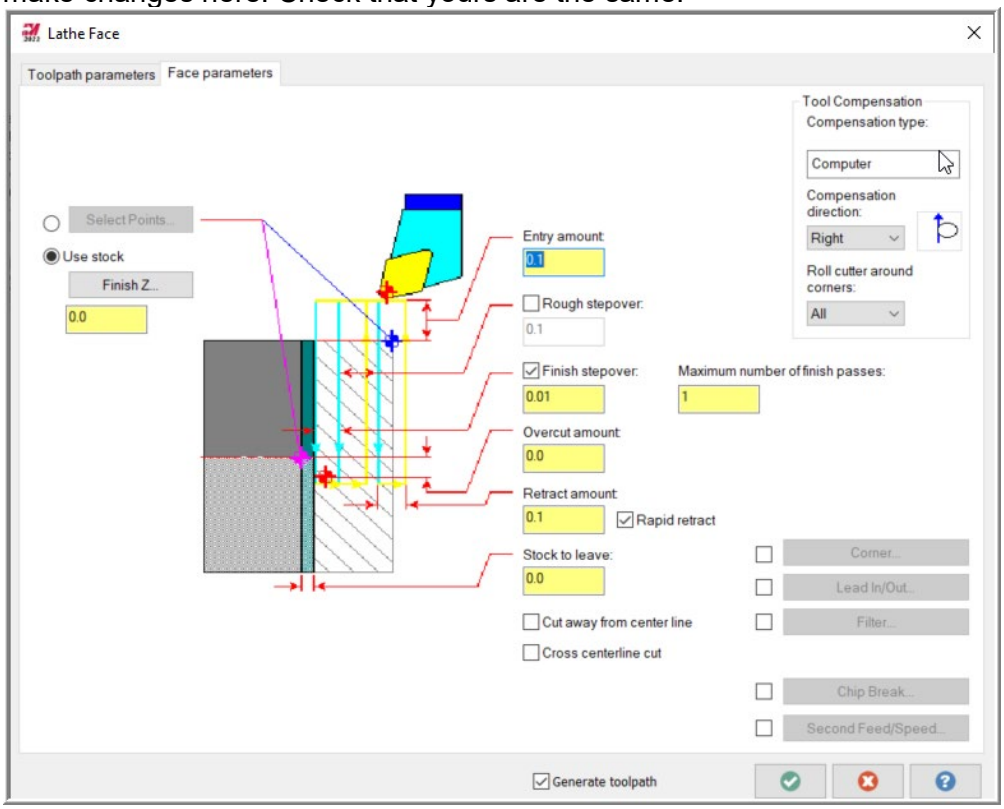


Insert colour

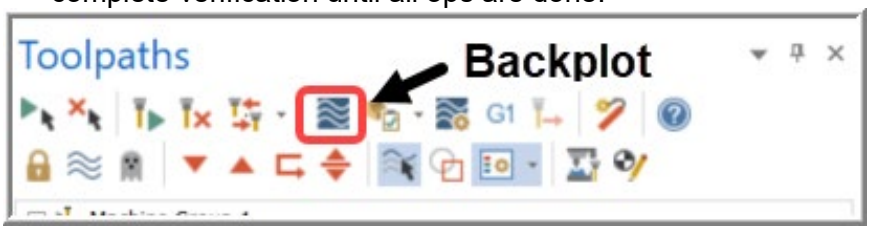
Orange – insert is facing down

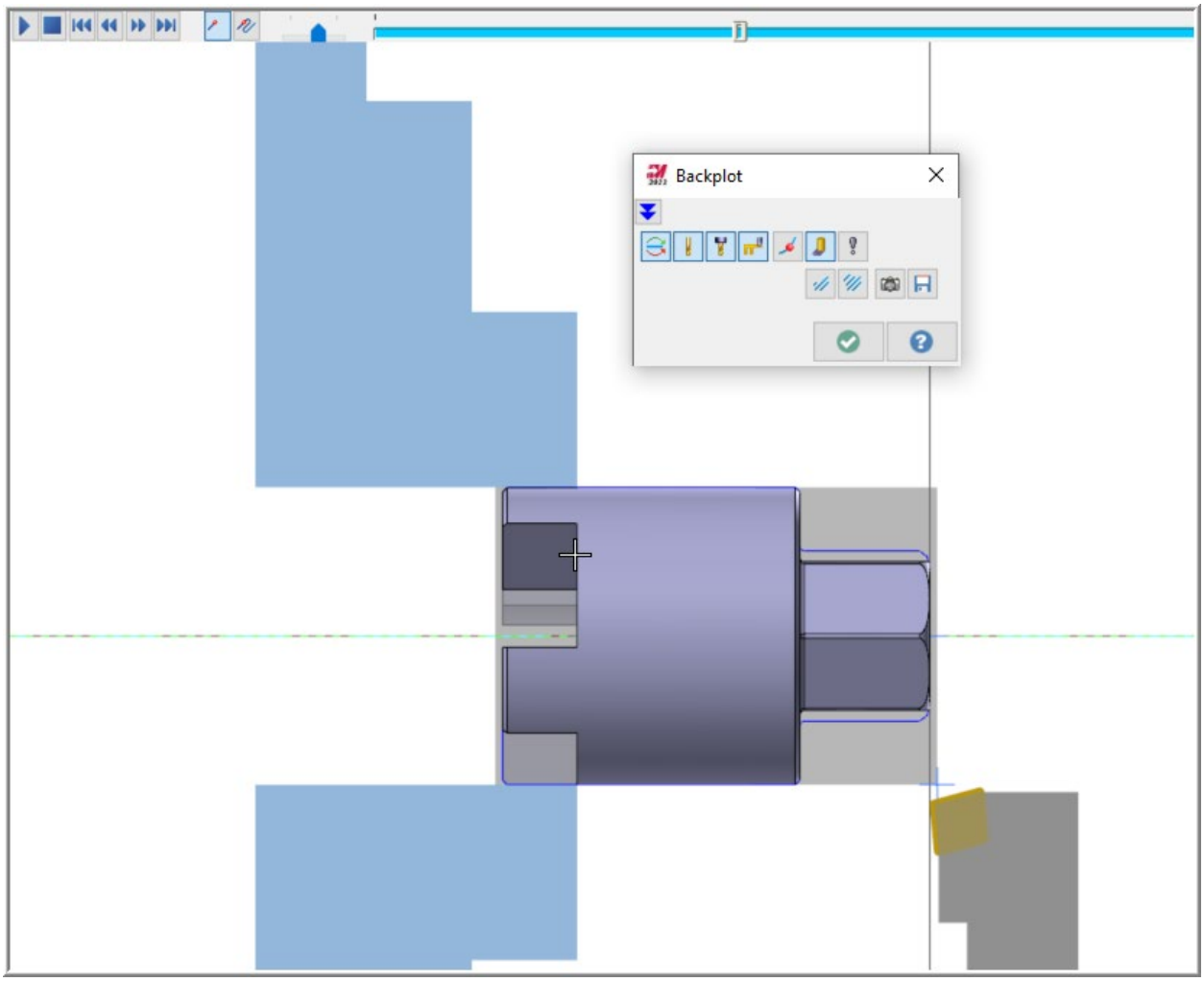
Yellow – insert is facing up

Note: The default settings on the Face Parameters page will work for our setup. No need to make changes here. Check that yours are the same.

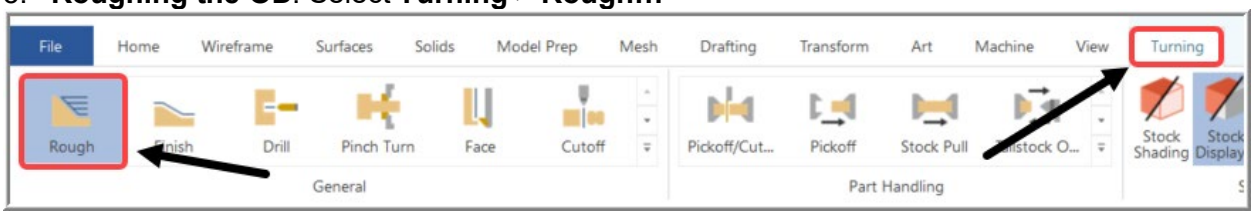


4. You can do a quick **backplot** to check your toolpath now. However, we will hold off on the complete verification until all ops are done.

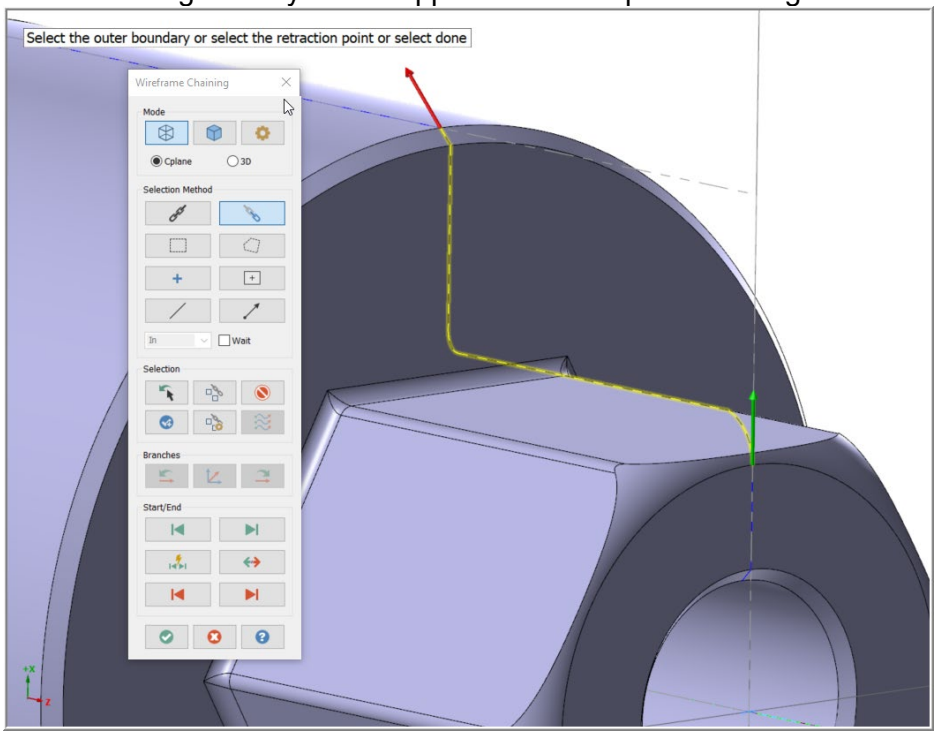




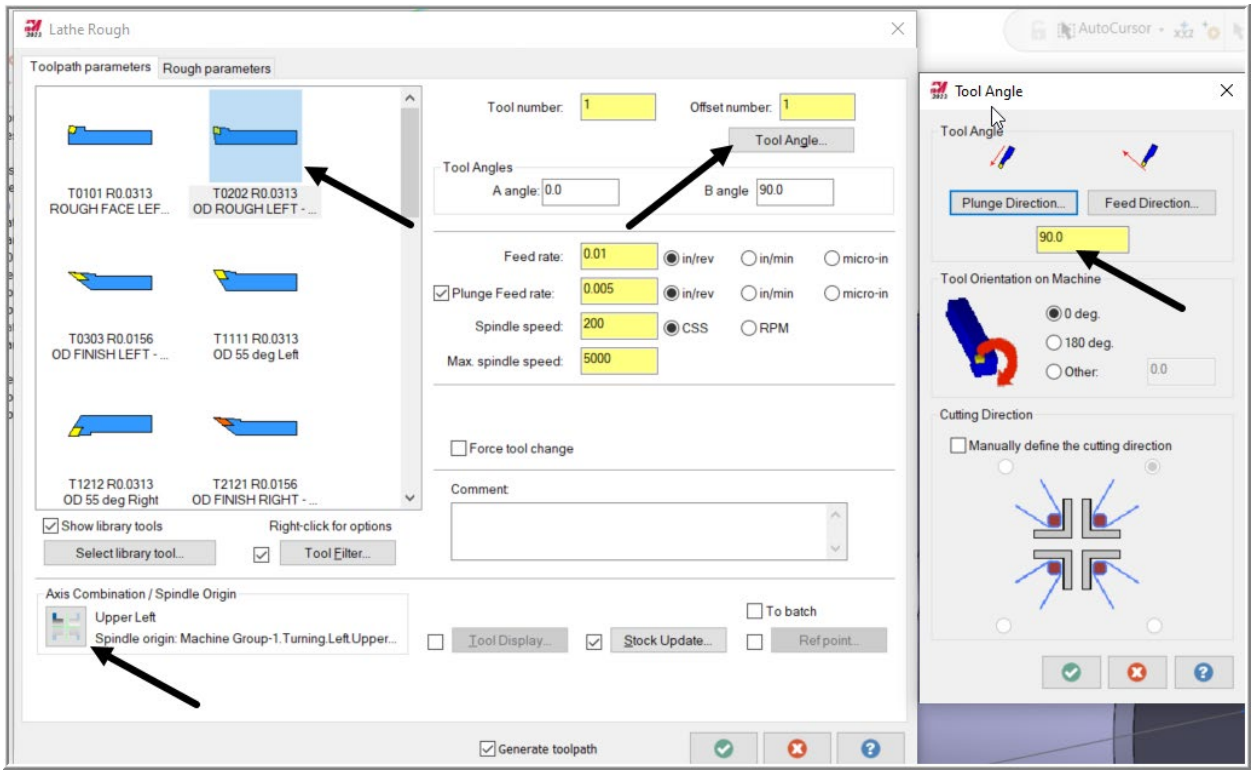
5. **Roughing the OD**. Select **Turning > Rough…**



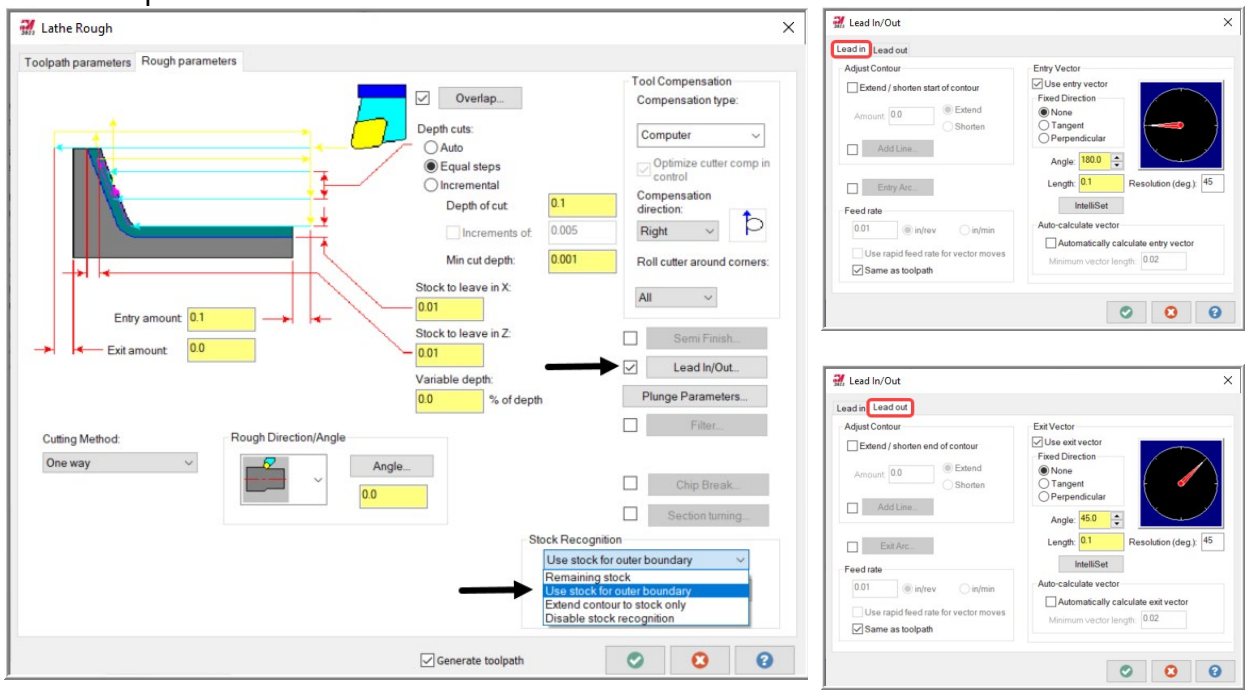
6. Chain from the 0.05 chamfer to the intersection of the taper and the OD. Make sure you chain the geometry on the upper side of the part. Hit the green check to accept.



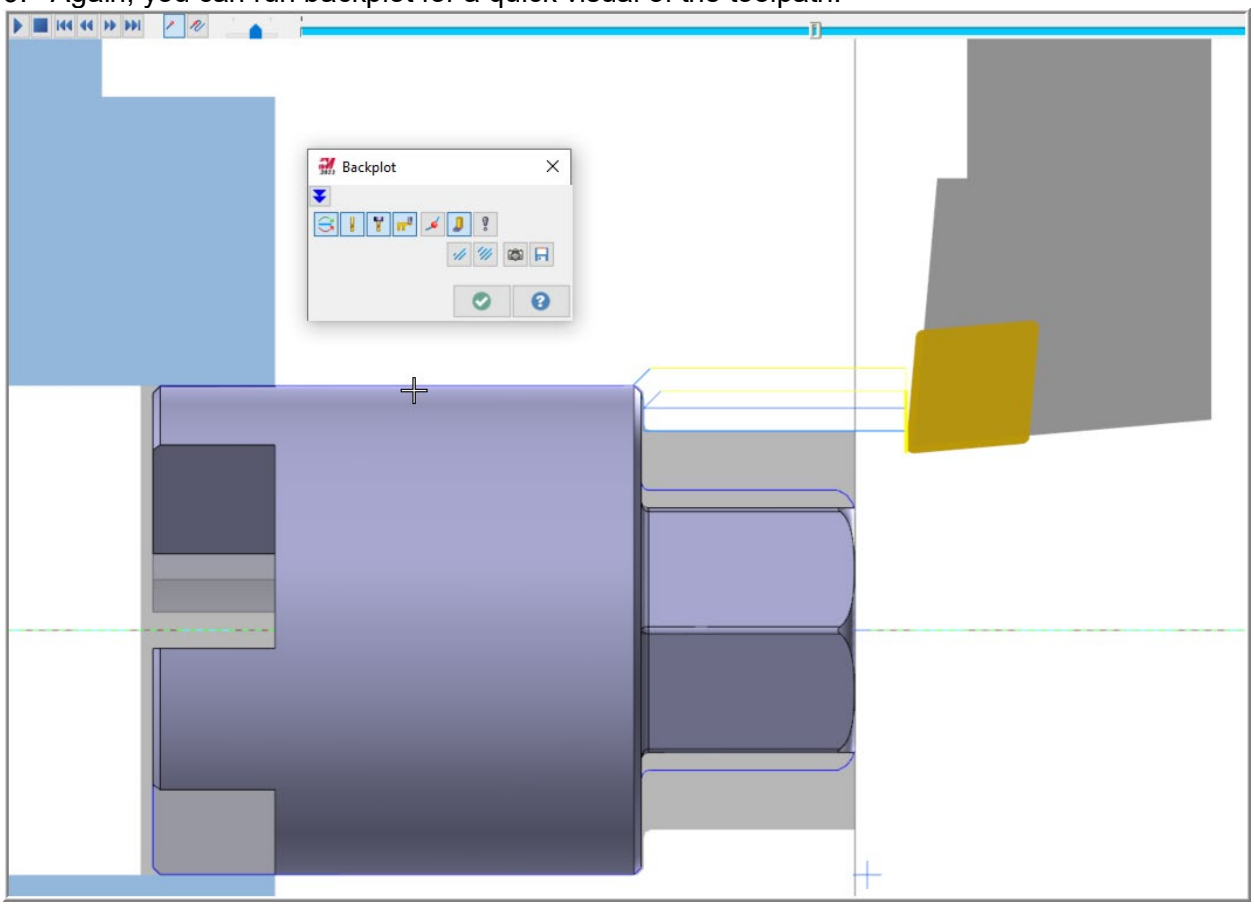
7. The roughing will take place with the Upper Turret (B-axis head). Select an appropriate tool for OD roughing. Check that **the Axis Combination/Spindle Origin** is set to **Left Spindle Upper Turret**. The tools in the library are shown in the toolchange position, for our machine this is 0 degrees. Since we are turning the OD, we set the **Tool B Angle at 90.0** for the correct orientation.



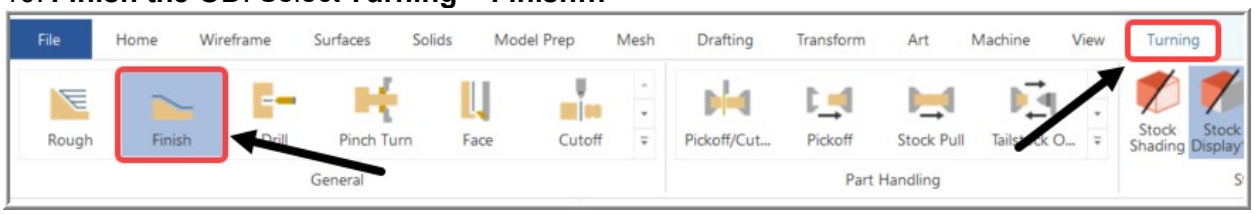
8. **Roughing Parameters**. Use the default settings. **Enable Lead In/Out** and set to appropriate values. Set Stock Recognition to **Use stock for outer boundary**. Green check to accept.



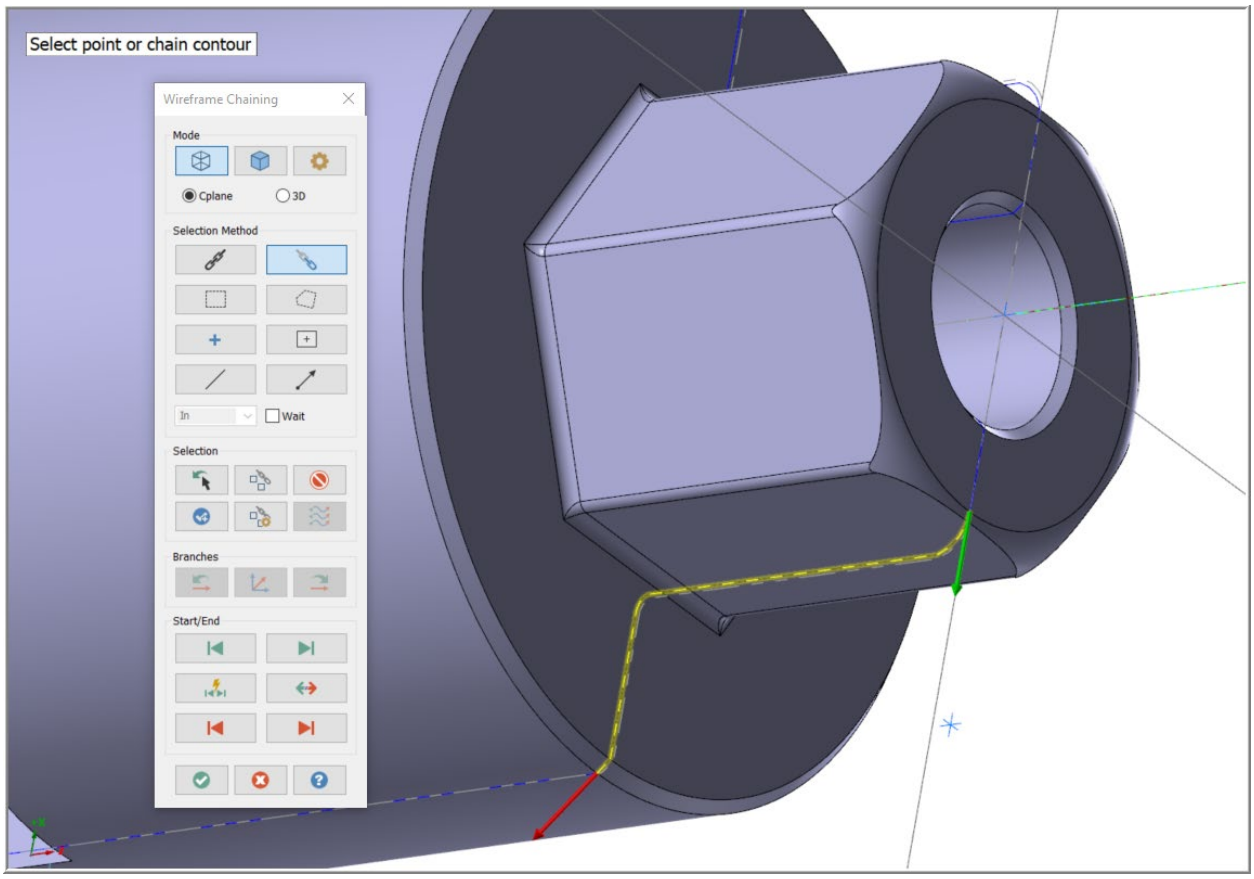
9. Again, you can run backplot for a quick visual of the toolpath.



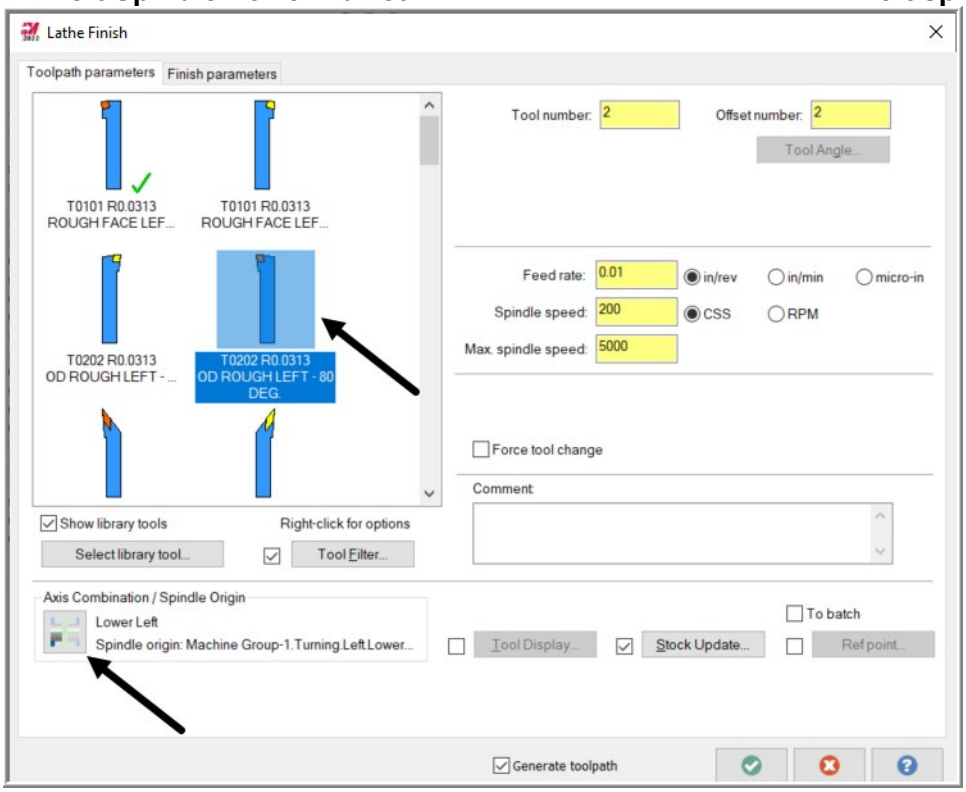
10. **Finish the OD**. Select **Turning > Finish…**



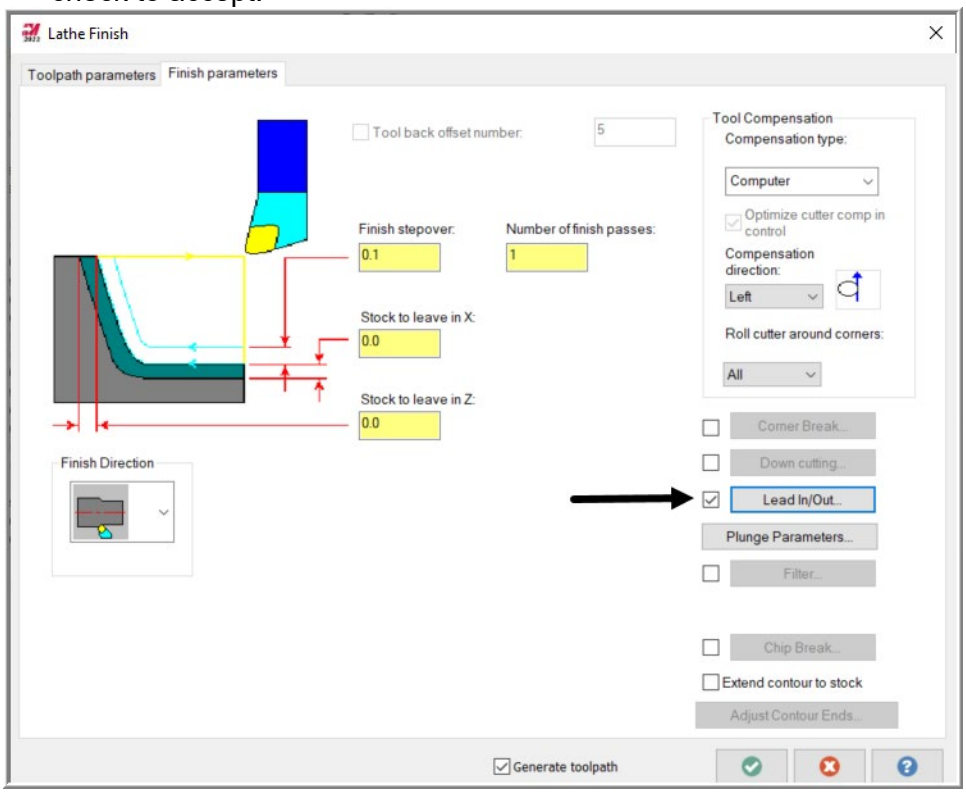
11. Since we will be using the lower turret to finish the OD, select geometry on the **lower side of the model**. Chain the same section as the Roughing op. Green check to accept the chain.



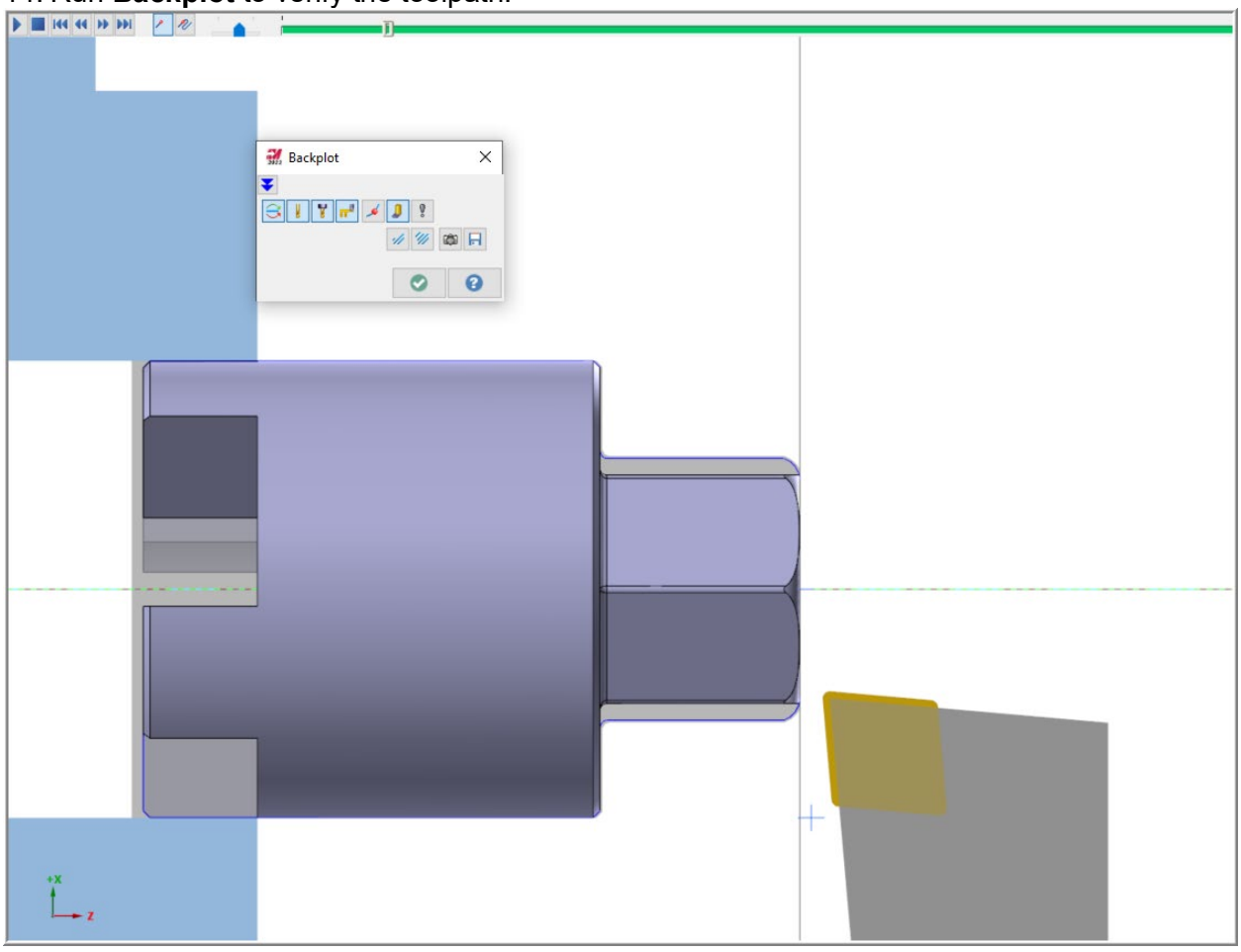
12. **Lathe Finish Properties**. Select a finish turning tool and check Axis Combination is set to **Left Spindle Lower Turret** and Turret Index Position is set to **Left spindle.**



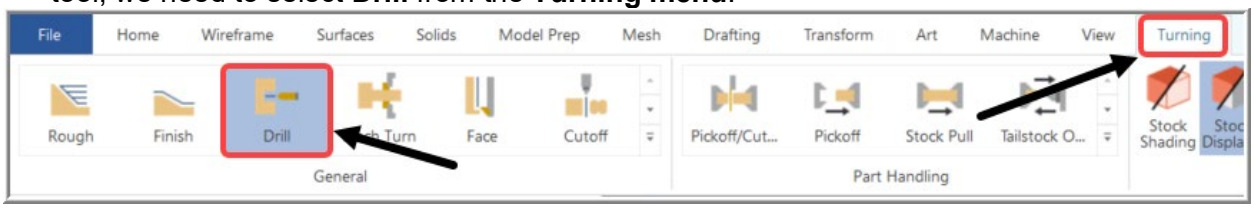
13. **Finish Parameters**. Default setting are good. Check that **Lead In/Out is enabled** and are set to appropriate values, keep in mind the tool orientation when setting your leads. Green check to accept.



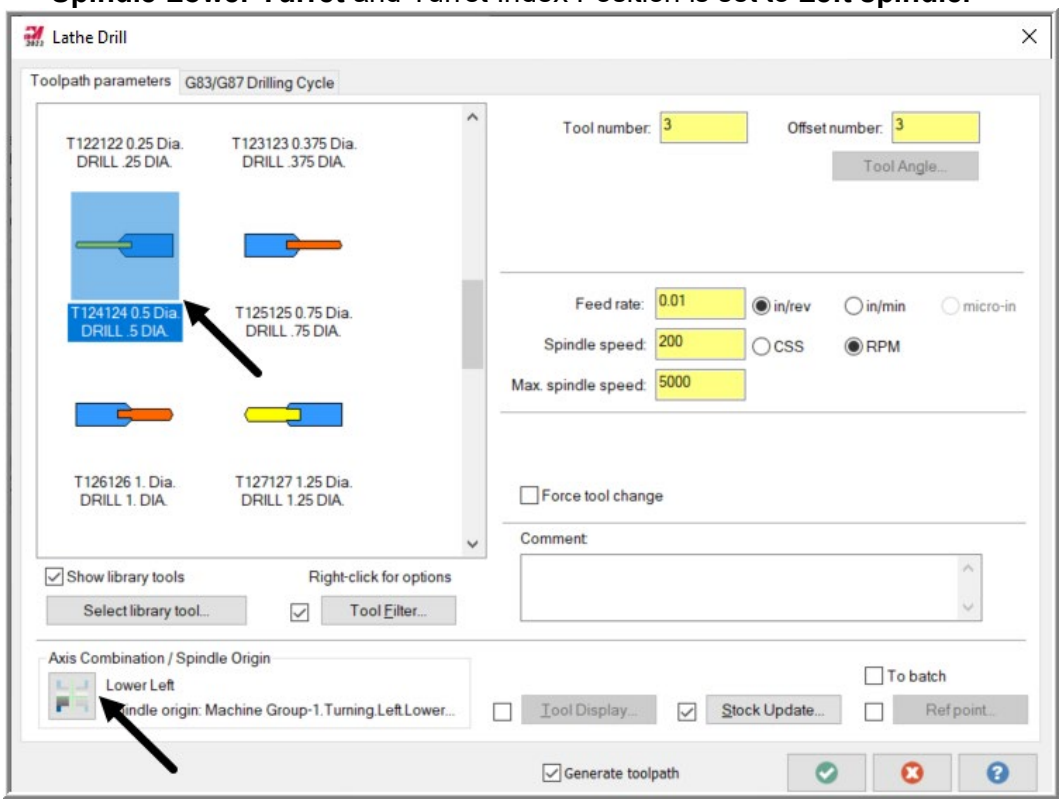
14. Run **Backplot** to verify the toolpath.



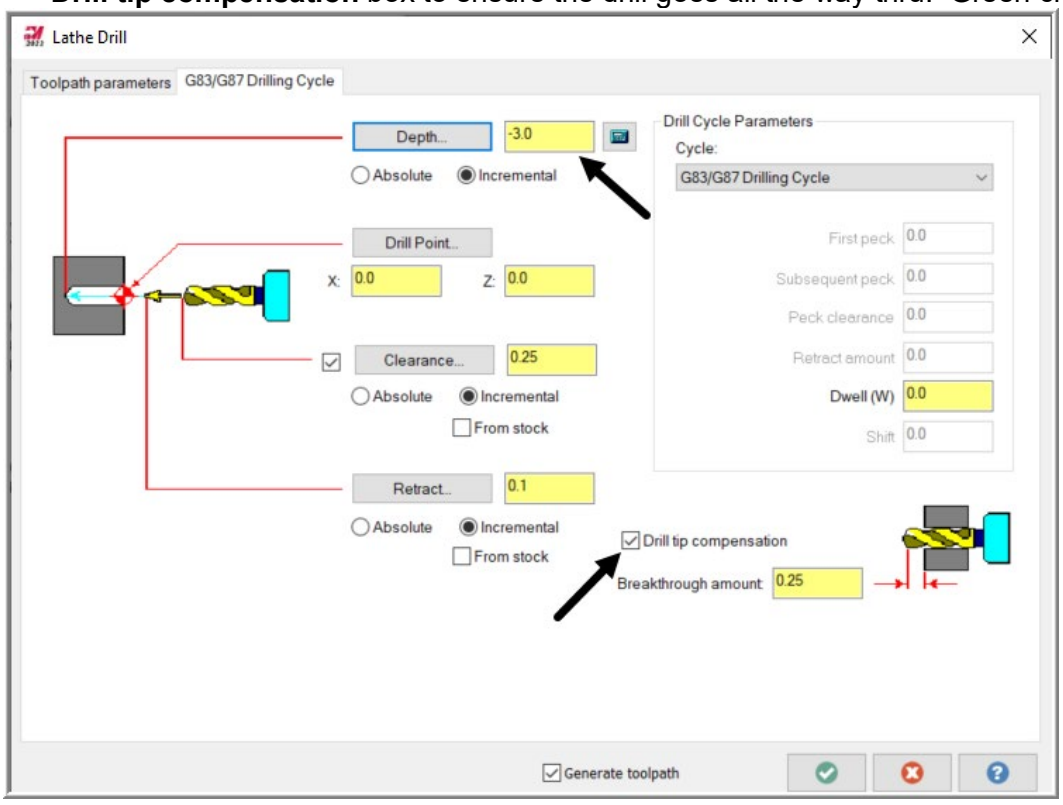
15. Next, we will drill the ½" diameter thru hole with the lower turret. Since this will be a static tool, we need to select **Drill** from the **Turning menu**.



16. **Lathe Drill Properties.** Select a ½" diameter drill and check Axis Combination is set to **Left Spindle Lower Turret** and Turret Index Position is set to **Left spindle.**

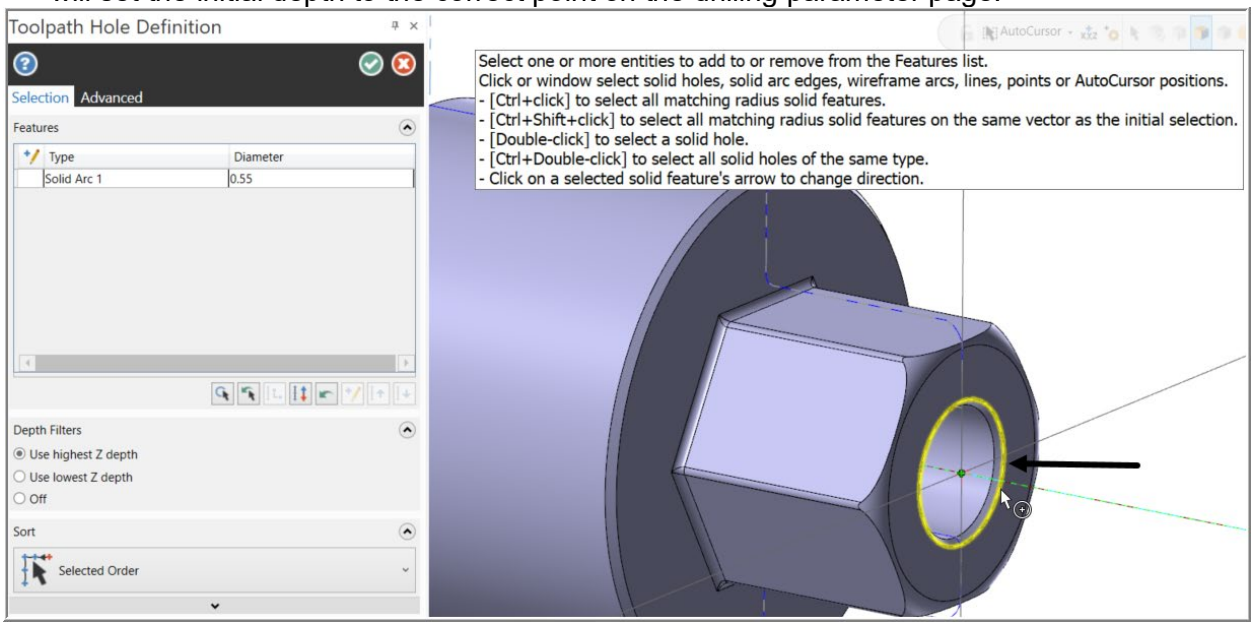


17. **Drilling Cycle Parameters.** Change the depth to -3.0 to drill thru the stock, then check the **Drill tip compensation** box to ensure the drill goes all the way thru. Green check to accept.

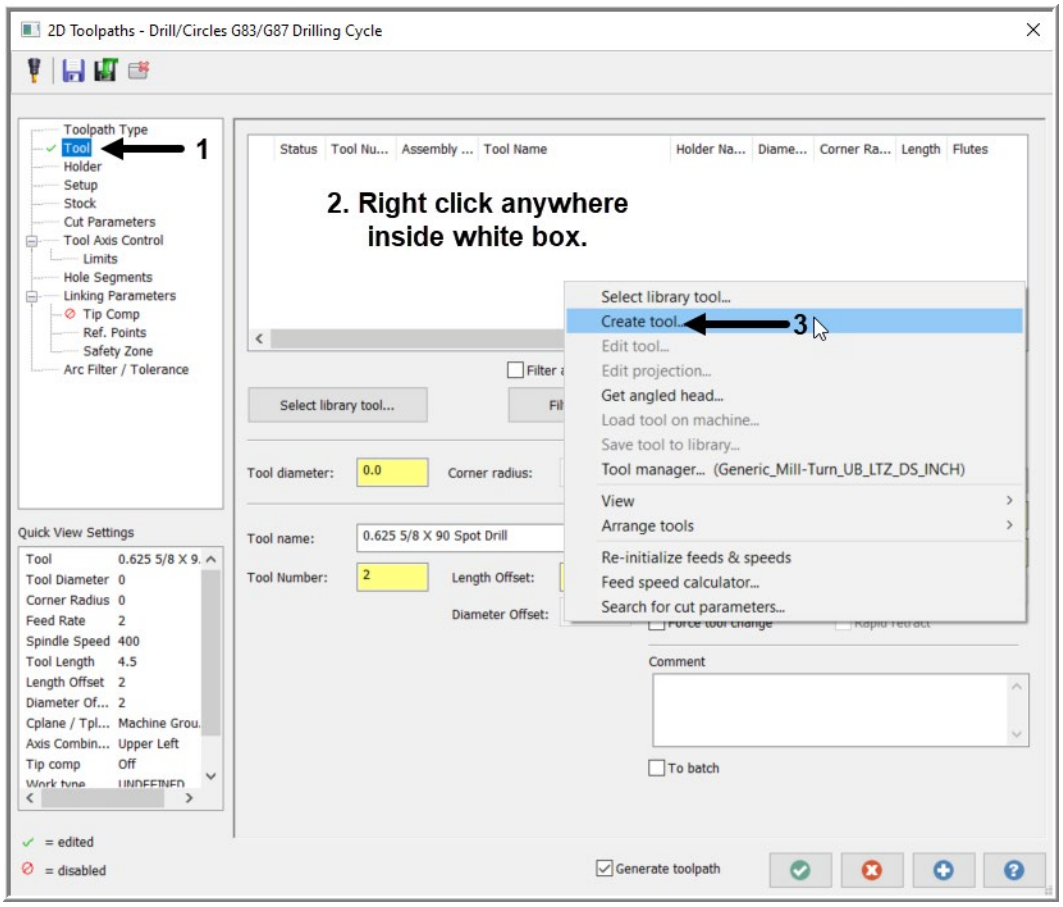


18. Next, we will chamfer the thru hole using the upper turret. Since this will be a rotating tool, we need to select **Drill** from the **Milling menu**.

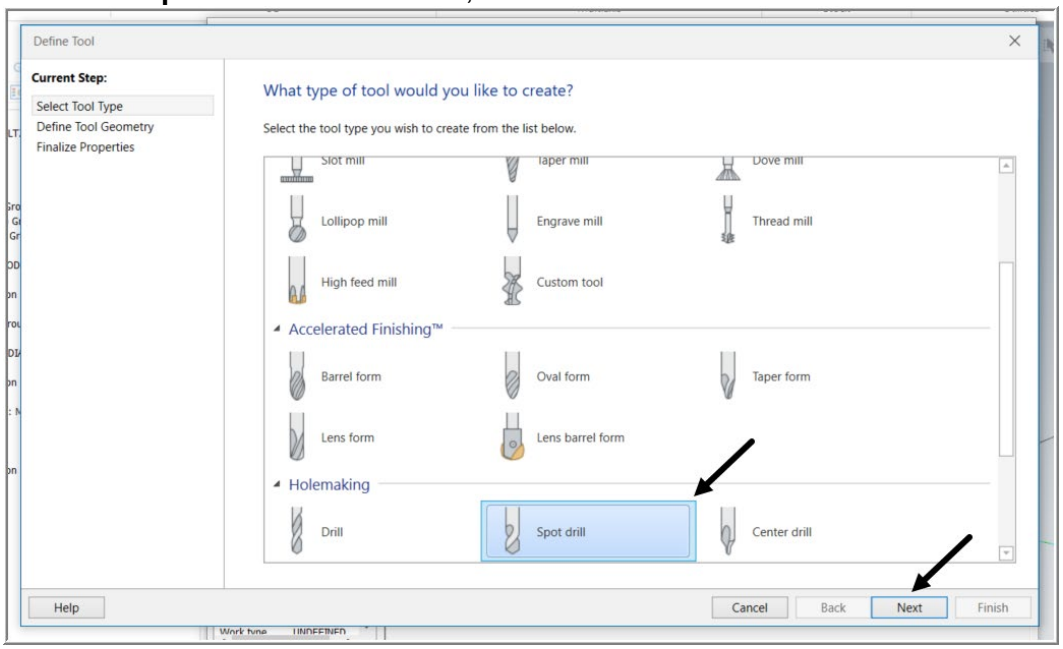
19. **Toolpath Hole Definition.** Select the outer diameter of the chamfer on the Z0 face. This will set the initial depth to the correct point on the drilling parameter page.



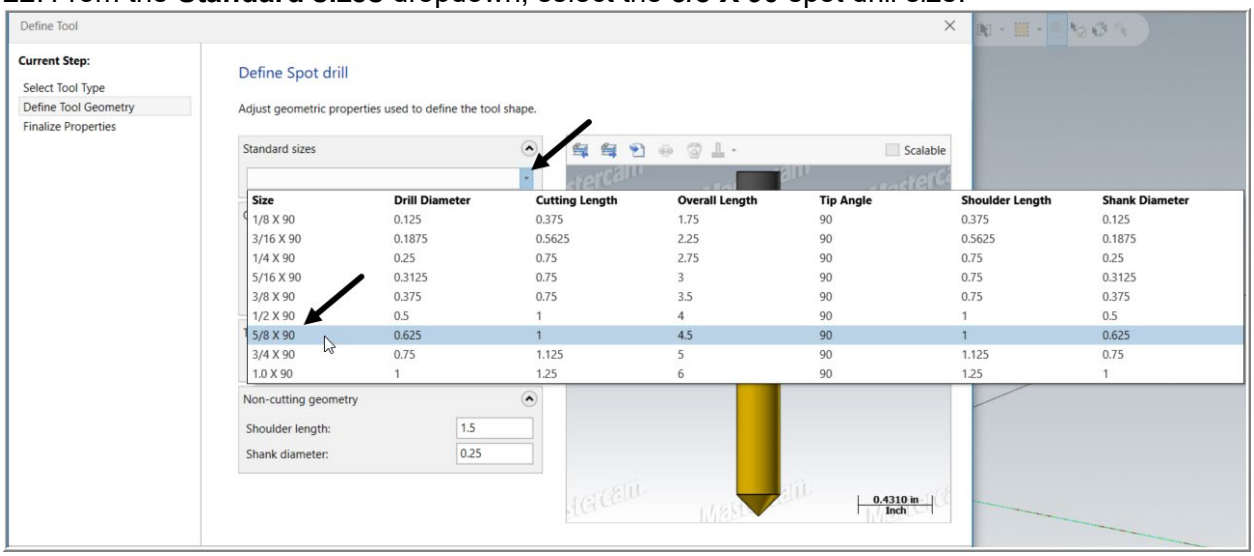
20. Next, we will create a tool. Highlight tool on the left side menu, then right click in the white box and select **Create tool**.



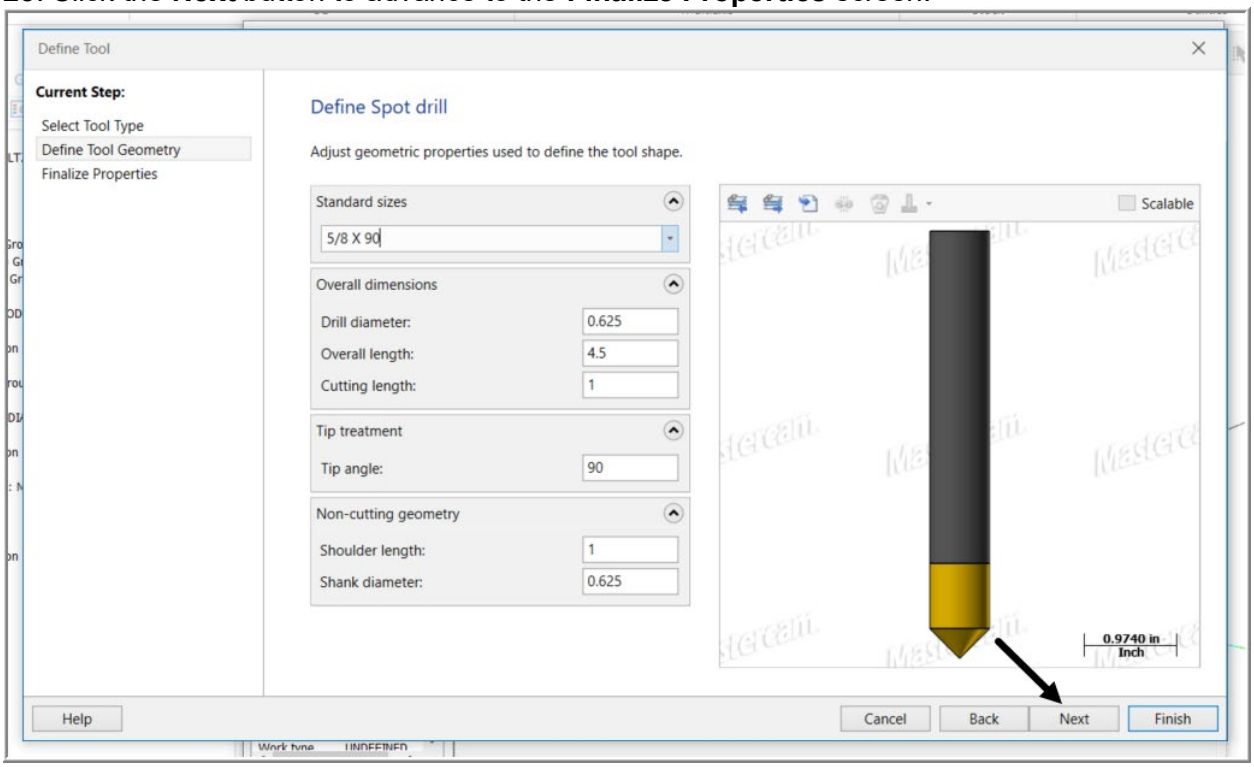
21. Select **Spot drill** from the menu, then click the **Next** button.



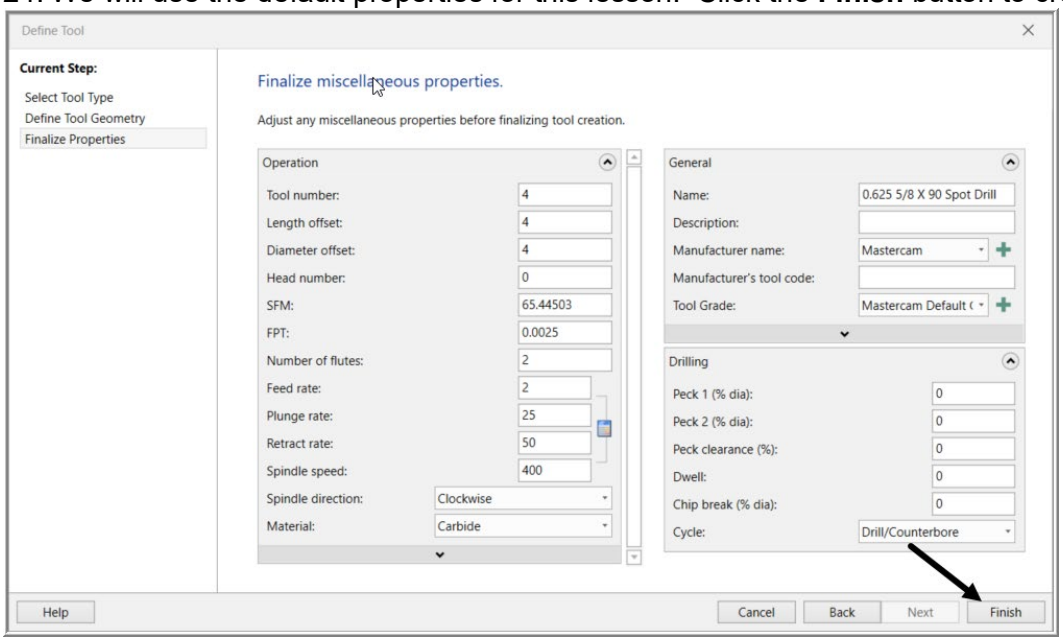
22. From the **Standard sizes** dropdown, select the **5/8 X 90** spot drill size.



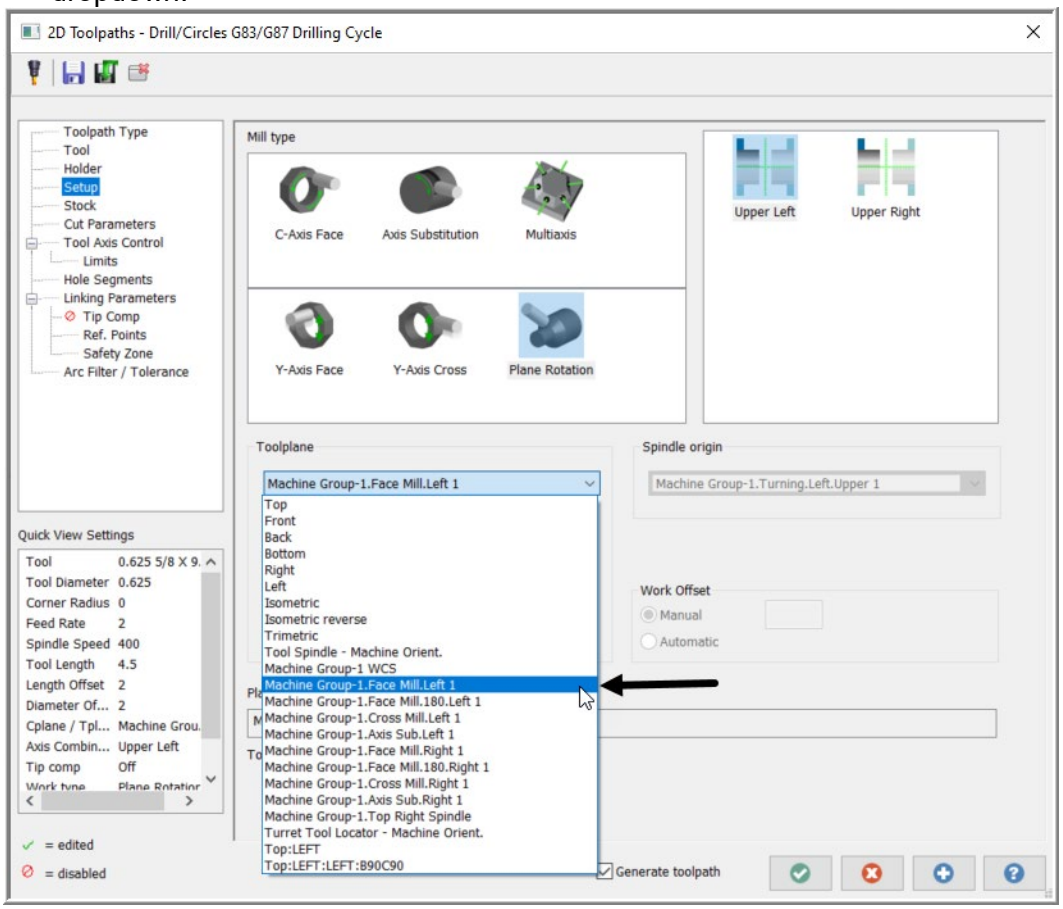
23. Click the **Next** button to advance to the **Finalize Properties** screen.



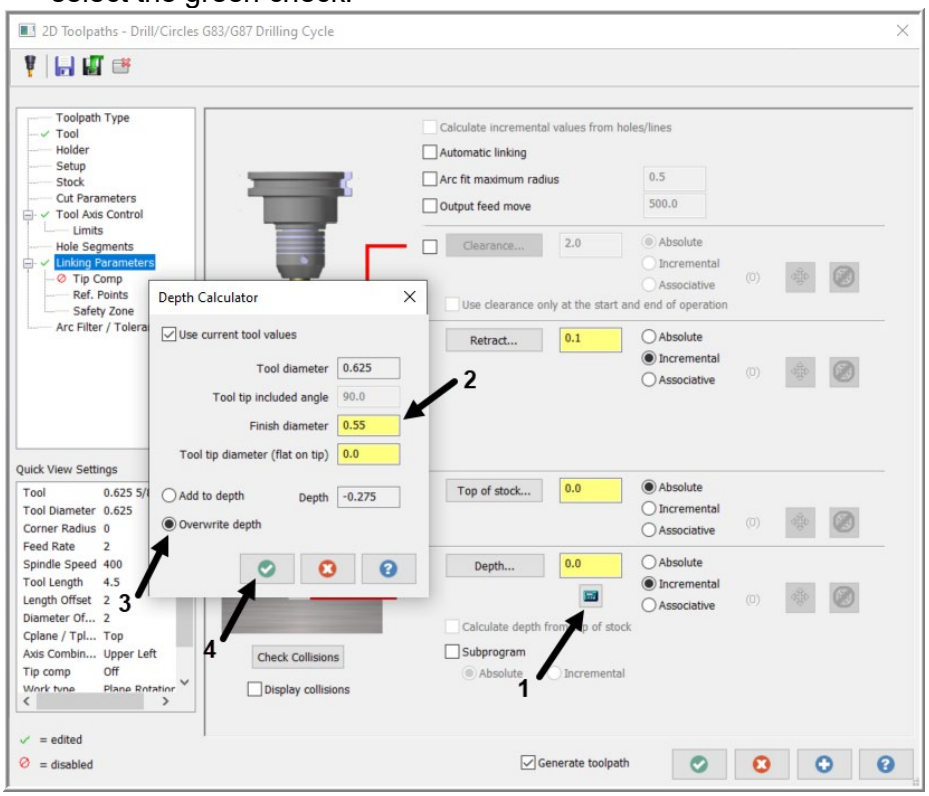
24. We will use the default properties for this lesson. Click the **Finish** button to create the tool.



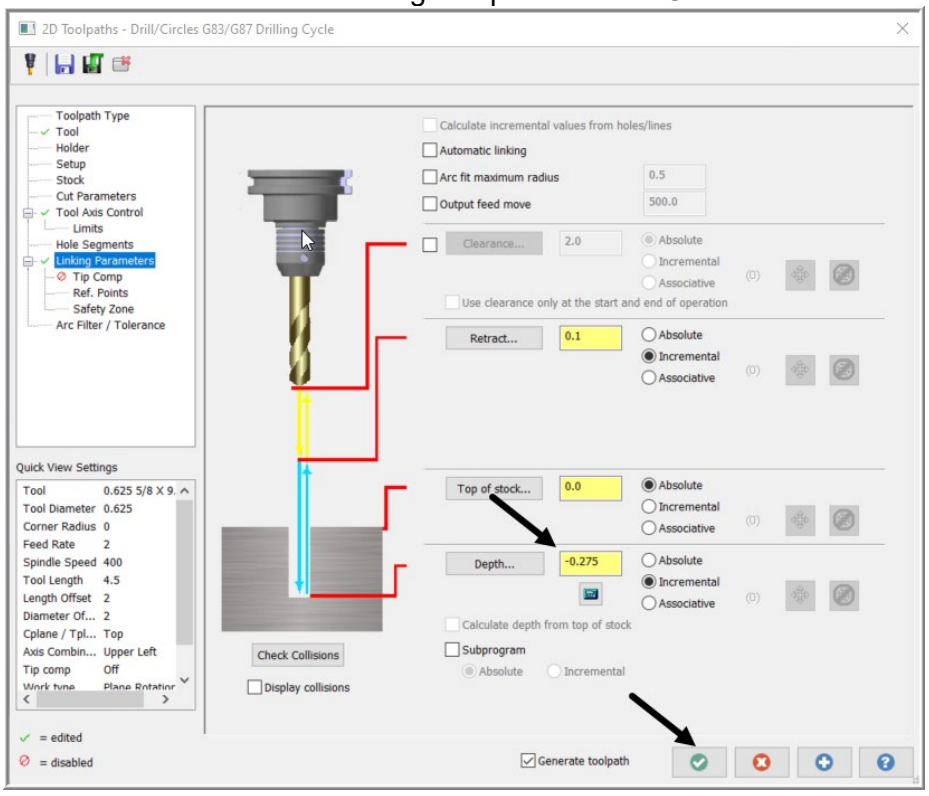
25. From the **Setup** menu on the left, select the **Machine Group-1.Face Mill.Left 1** from the dropdown.



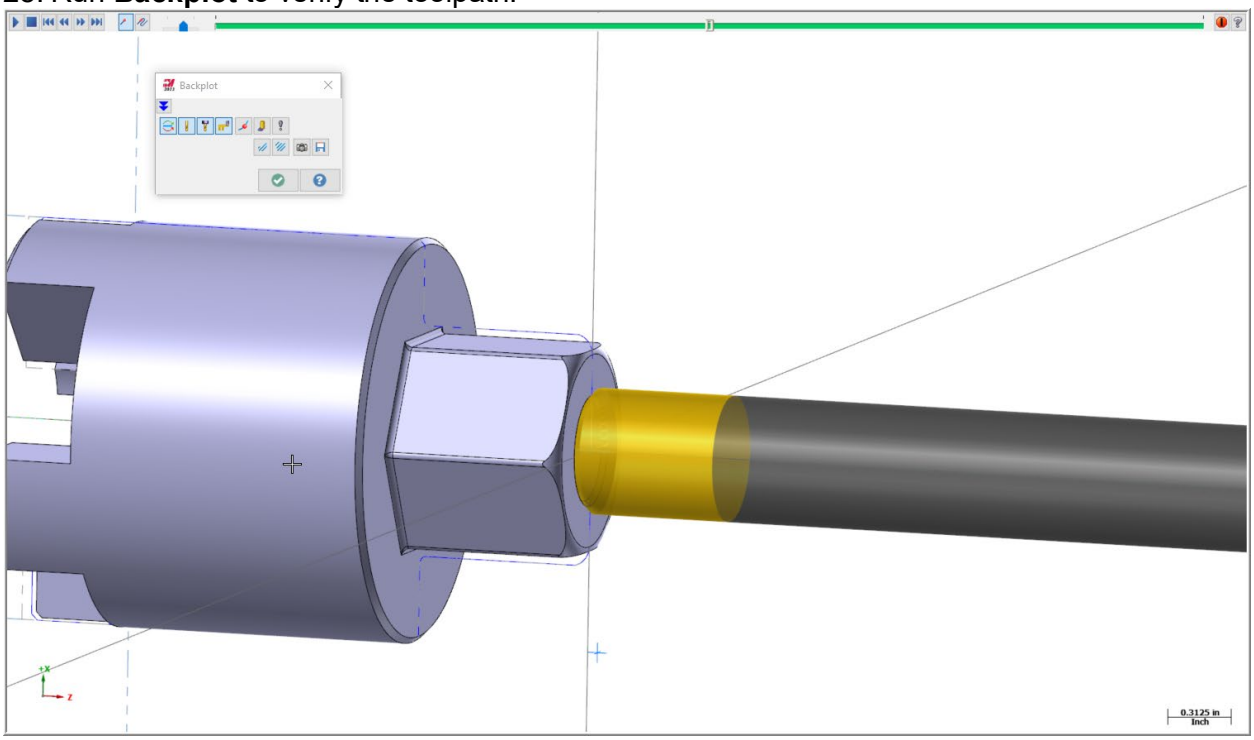
26. Jump down to **Linking Parameters** on the left side menu. Select the **Depth Calculator** button, set the **Finish diameter to 0.55**, select the **Overwrite depth** radio button, then select the green check.



27. Notice the **Incremental Depth** is now automatically calculated for you based on your finished diameter and cutting tool parameters. Green check to accept.



28. Run **Backplot** to verify the toolpath.

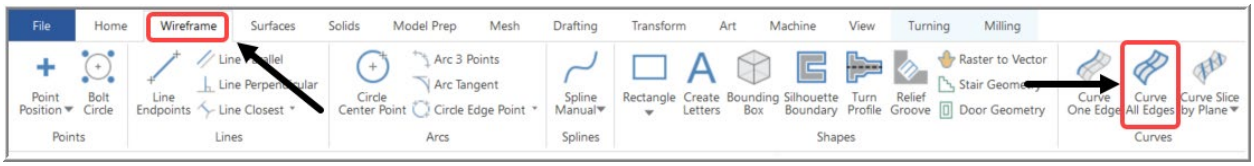


29. **Save** your file. 

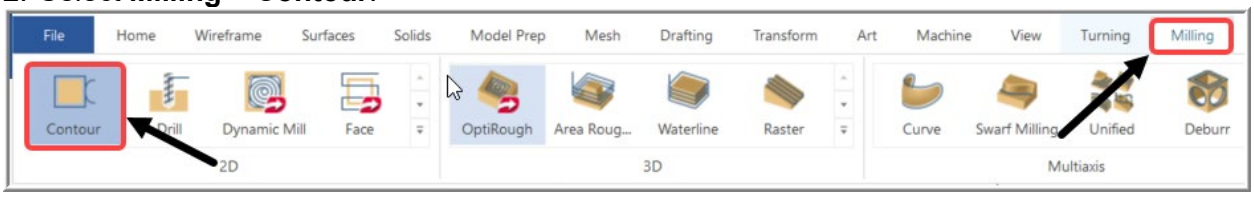
TASK 4:
MAIN SPINDLE MILLING

➲ In this task you will mill the hex on the main (left) spindle.

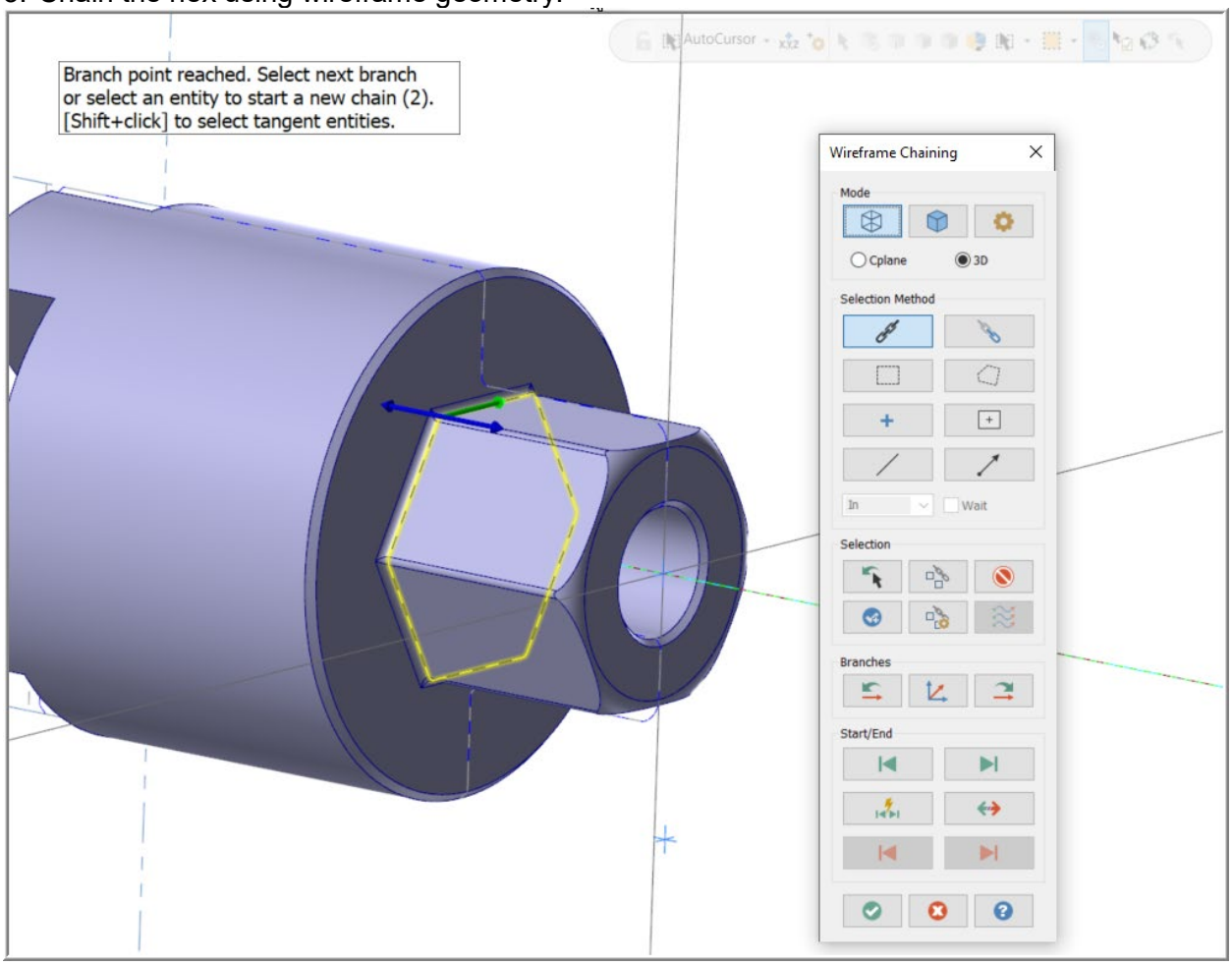
1. To mill the hex, we will use a simple contour path using wireframe geometry. If you do not have wireframe geometry, create it first by selecting **Curve All Edges**, then selecting the entire solid model.



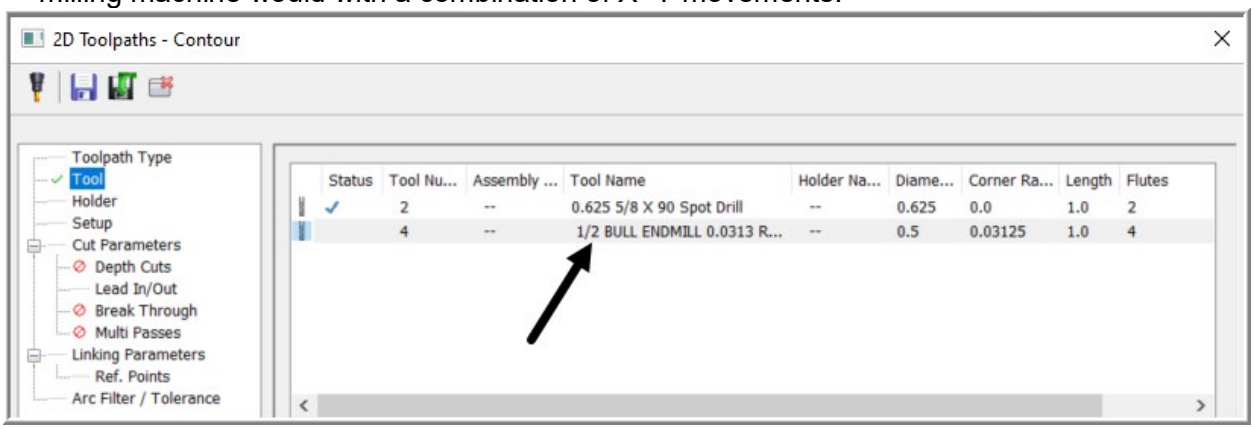
2. Select **Milling > Contour**.

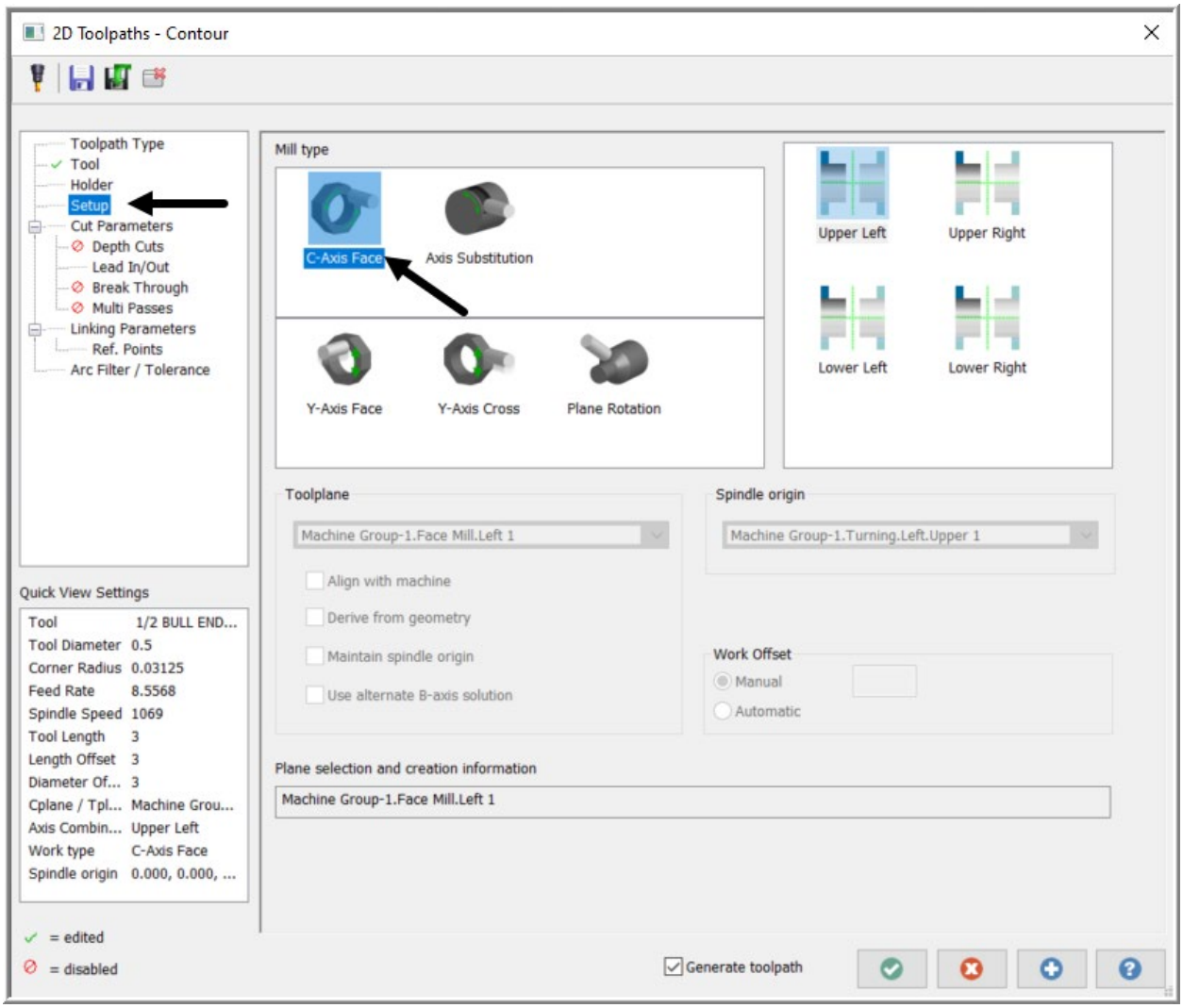


3. Chain the hex using wireframe geometry.

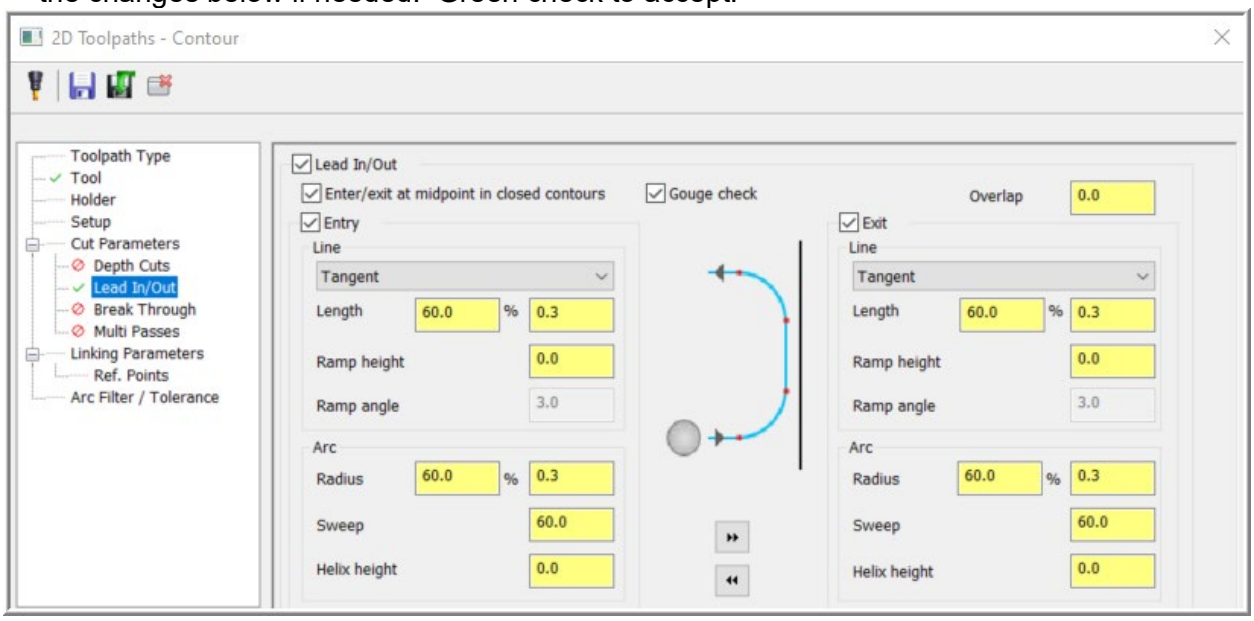


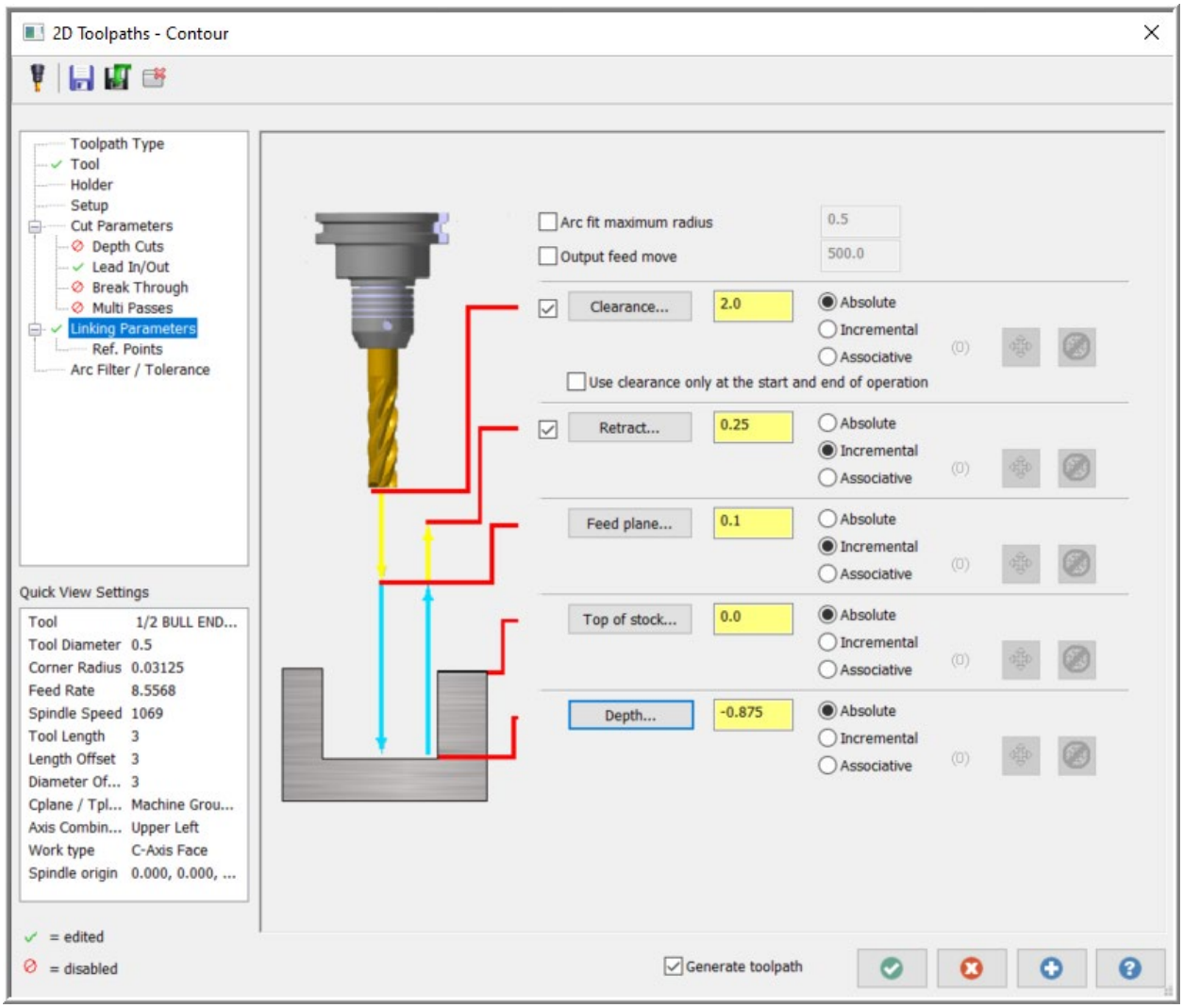
4. Select a **1/2 Bull Endmill** with a **1/32 rad** from the library. In the **Setup page** set the **Mill Type to C-Axis Face**. This will force the chuck to rotate and will also keep X above the centerline which is a must for some mill-turn machines. The output code will consist of C and X movements. If we had used Y-Axis Face the tool would profile the hex much the same as a milling machine would with a combination of X+Y movements.



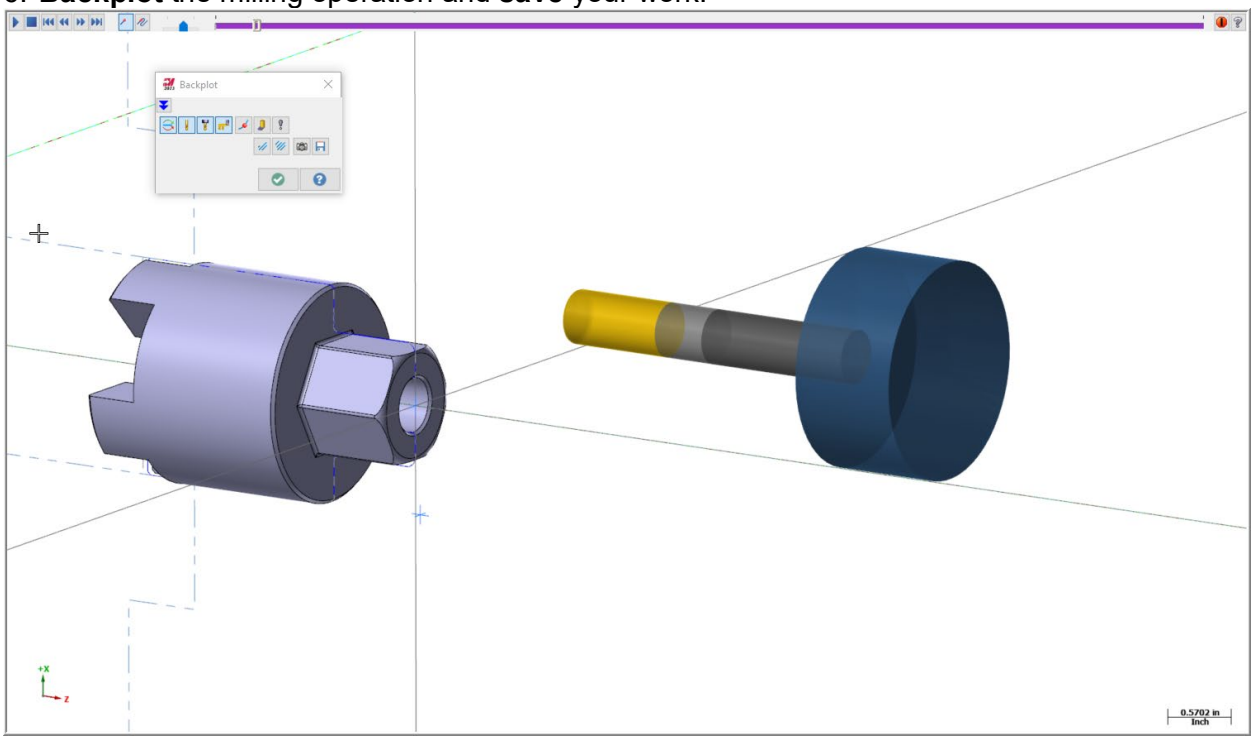


5. **Enable Lead In/Out** and set appropriate values. On the **Linking Parameters** page make the changes below if needed. Green check to accept.

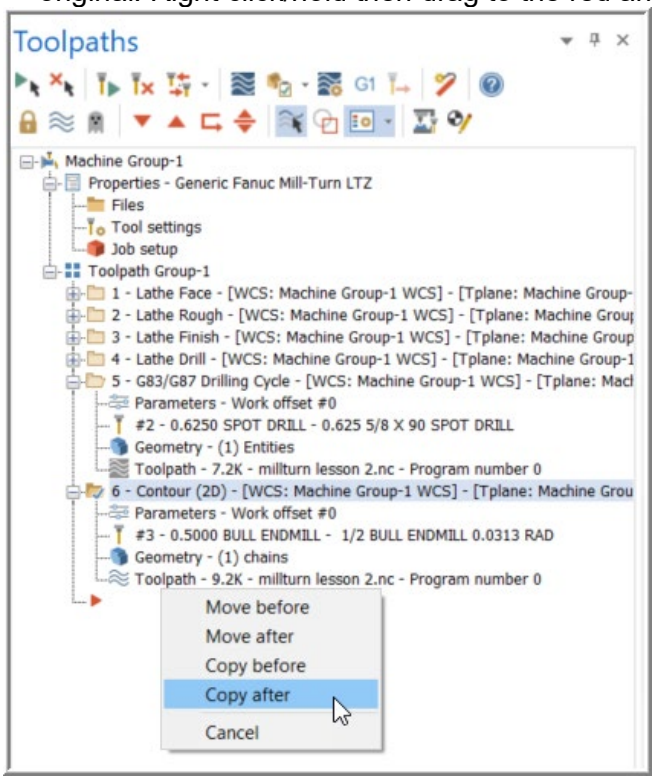


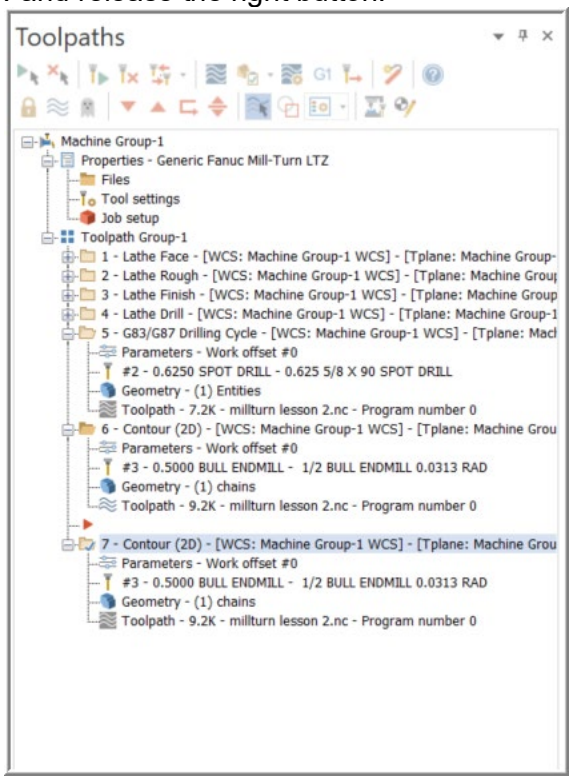


6. **Backplot** the milling operation and **save** your work.

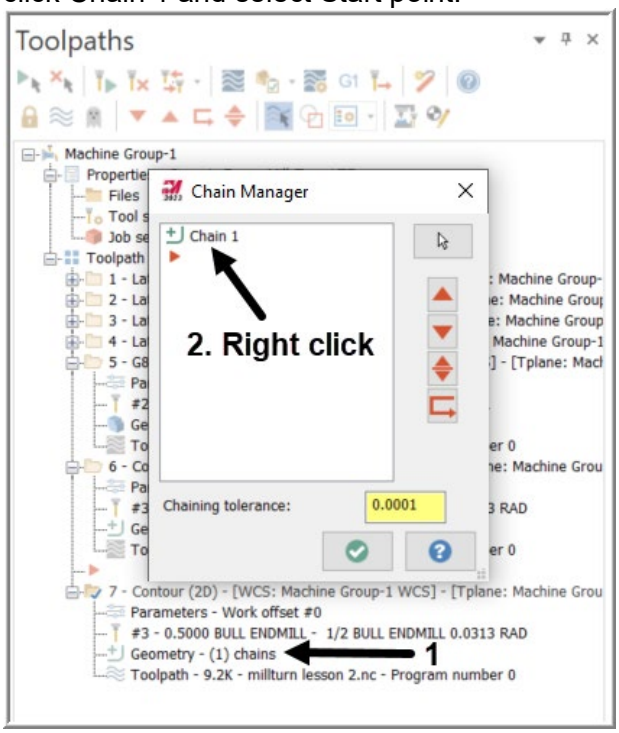


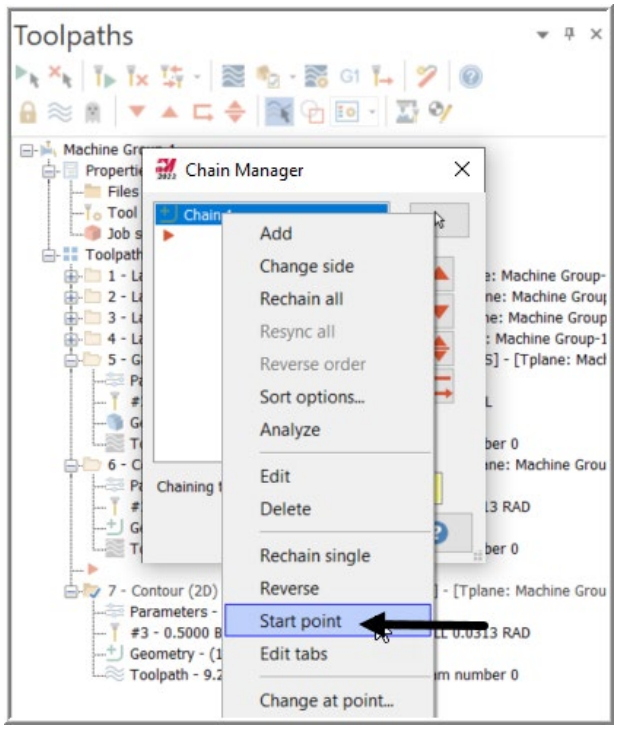
7. With the operation set to C-Axis face it is not important where the chain starts. The start of the cut will always be rotated to the top. For example, copy the op we just made after the original. Right click/hold then drag to the red arrow and release the right button.



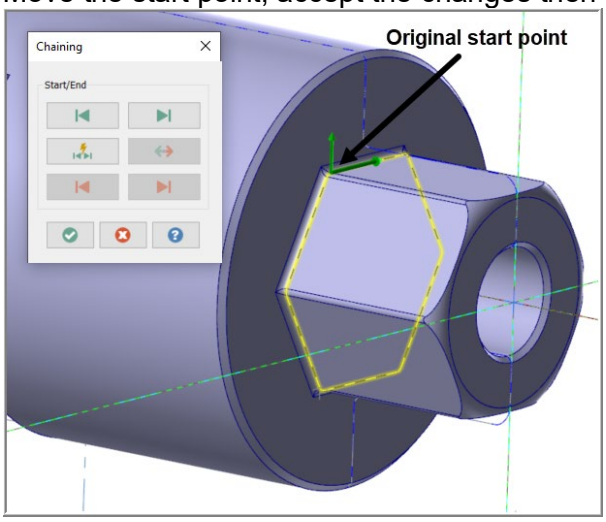


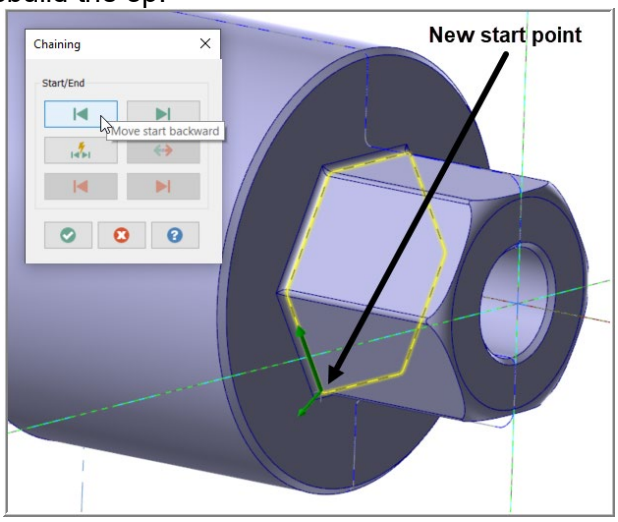
In this new operation (7-Contour (2D)) select the geometry to open the Chain Manager. Right click Chain 1 and select Start point.



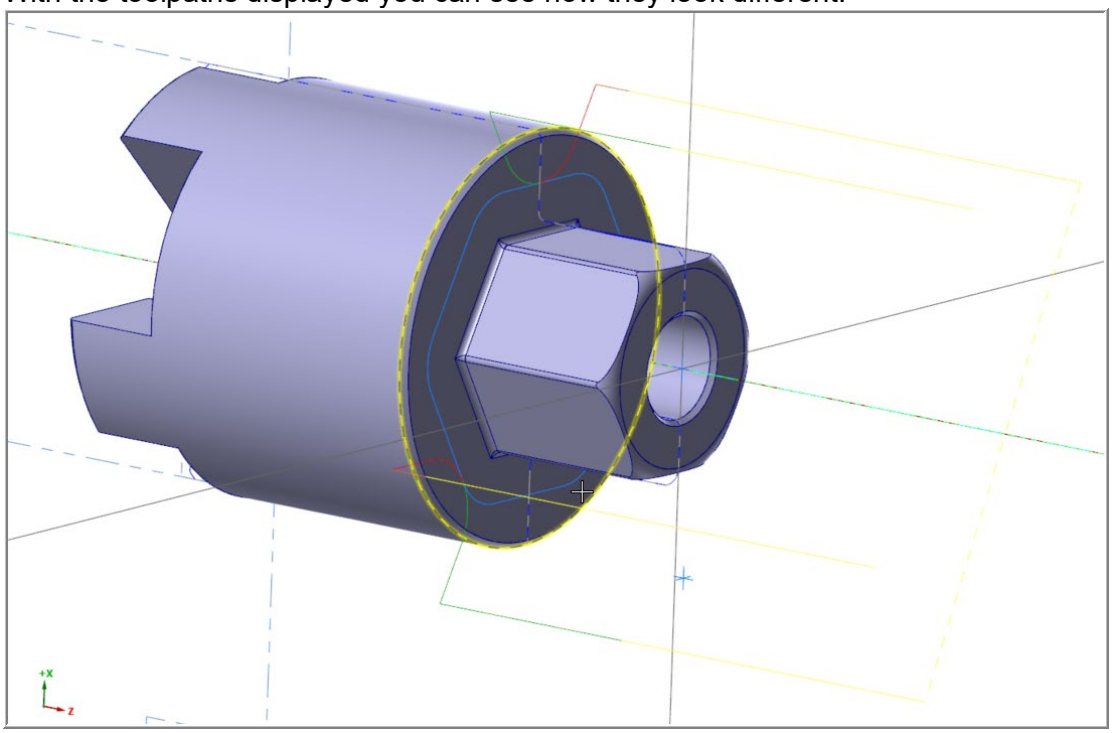


Move the start point, accept the changes then rebuild the op.

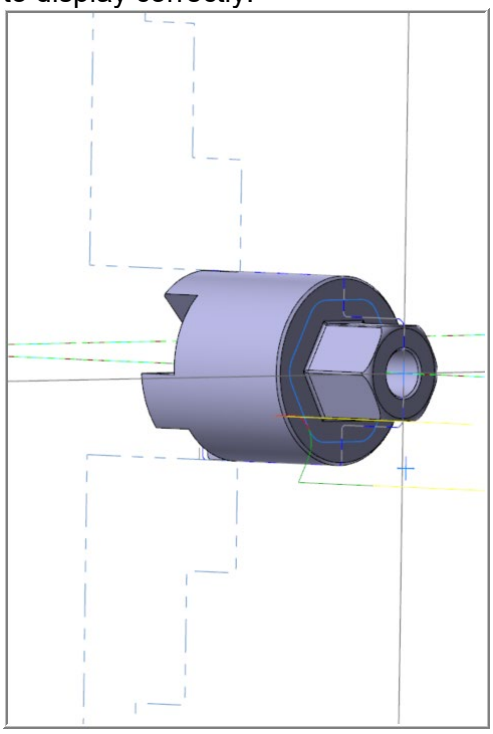


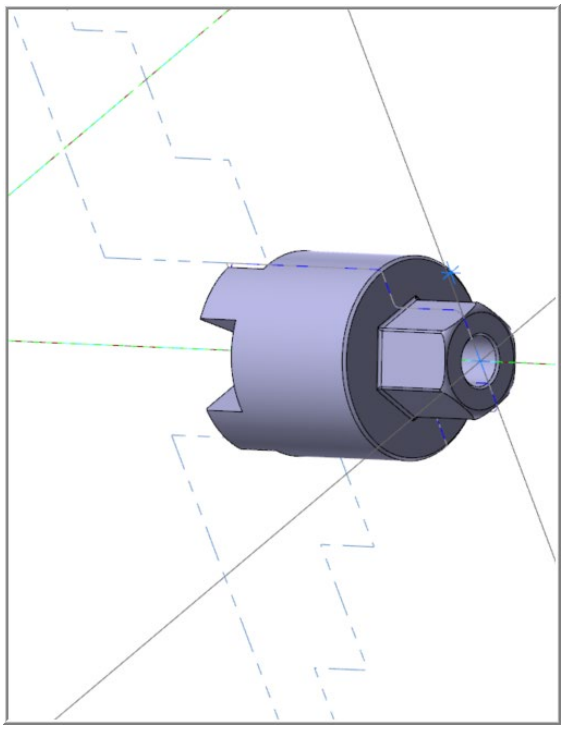


With the toolpaths displayed you can see how they look different.



Take note of the jaw position before starting a Backplot on this new op. When you start the Backplot you will notice the jaws have indexed to move this cut start point to the top.
Note: You may need to check the box for **Simulate Rotary Axis** in the Backplot Options for this to display correctly.



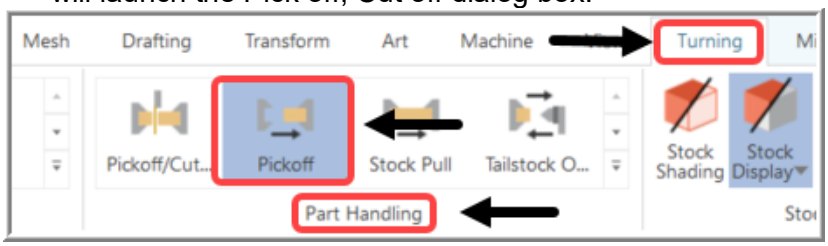


8. **Delete** this extra op (7-Contout (2D)). It was just for demonstration. Resave your file.

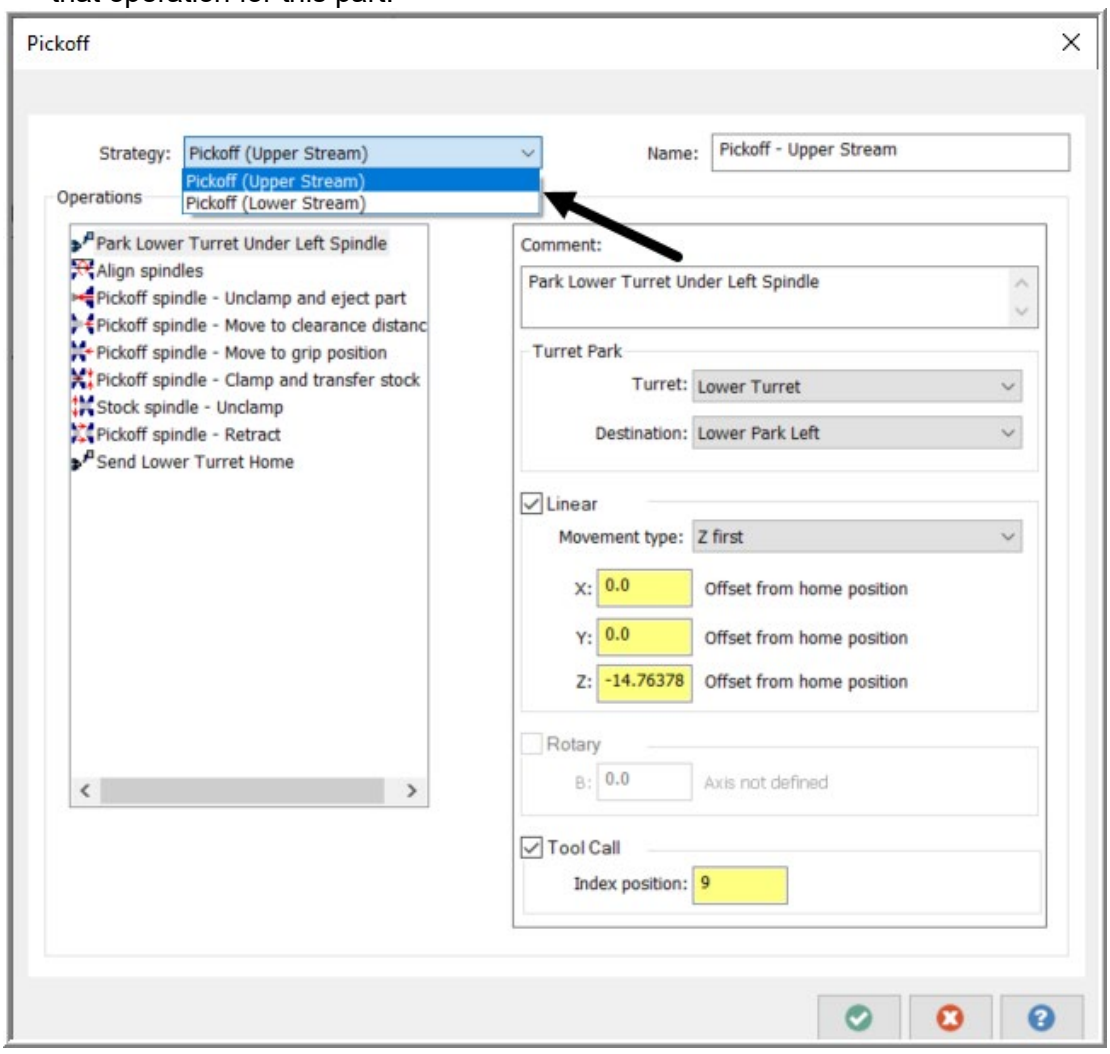
TASK 5:
POCO – PICK OFF CUT OFF

➲ In this task you will pick the part off and transfer it to the sub (right) spindle.

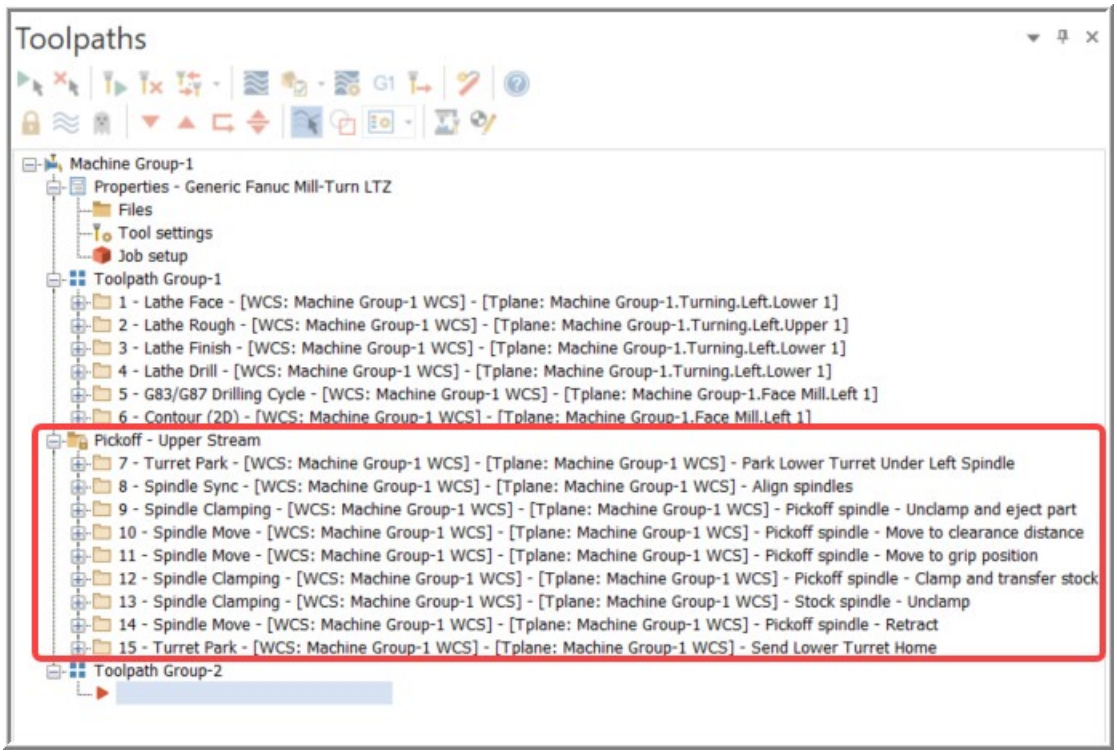
1. From the menu bar select **Turning > Pickoff** found under the **Part Handling** section**.** This will launch the Pick off, Cut off dialog box.



2. **Pickoff.** Since we are using a single piece of stock and not barstock, set the Strategy to Pickoff. Changes can be made to the operations parameters if needed, however the events and their order cannot be altered. Users can define their own strategies if specific part handoff sequences are needed. These new strategies will appear in the Strategy list. Green check to accept and create the POCO routine.
Note: The Pickoff/Cutoff routine is not available since our job setup settings do not support that operation for this part.



3. **Results of POCO** dialog input. Mastercam creates the handoff operations. None of these created ops can be deleted or moved. This safeguards against unintentional changes that could result in machine crashes.

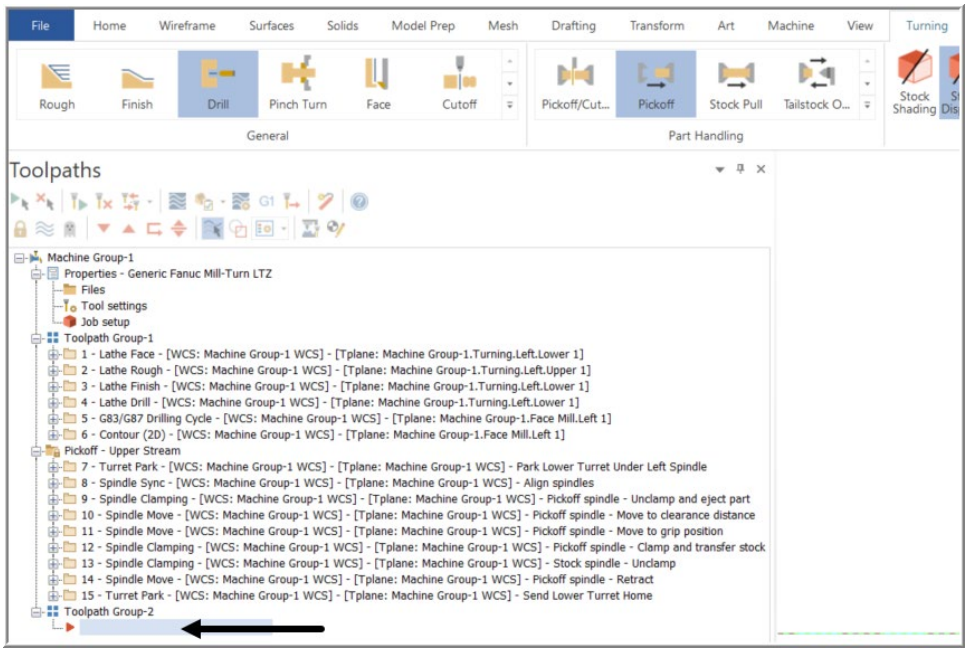


4. **Save** your file.

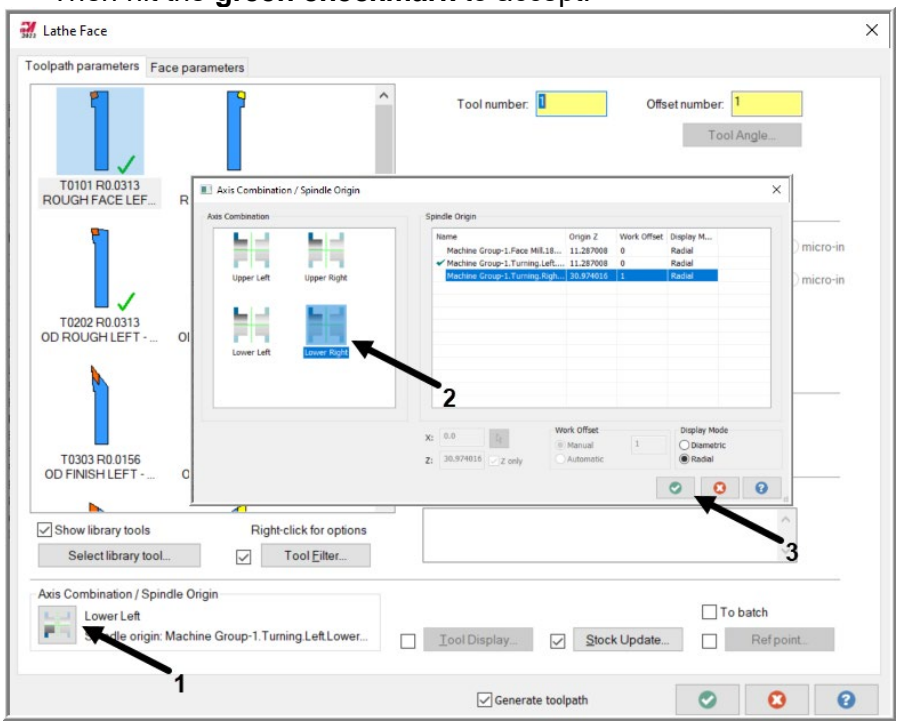
TASK 6:
SUB SPINDLE TURNING

➲ In this task you will face the part and turn the OD profile in the sub (right) spindle.

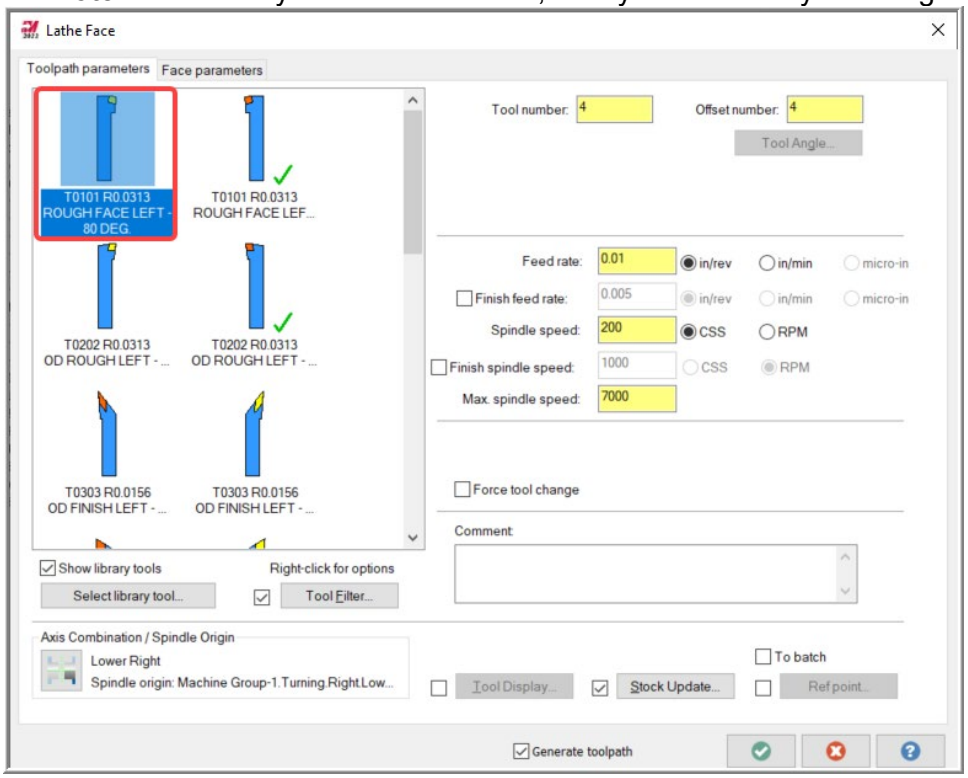
1. **Facing the part**. Before starting the sub spindle turning make sure the **red arrow** is in the correct position. It should be directly below **Toolpath Group-2.** Then select **Turning > Face…**



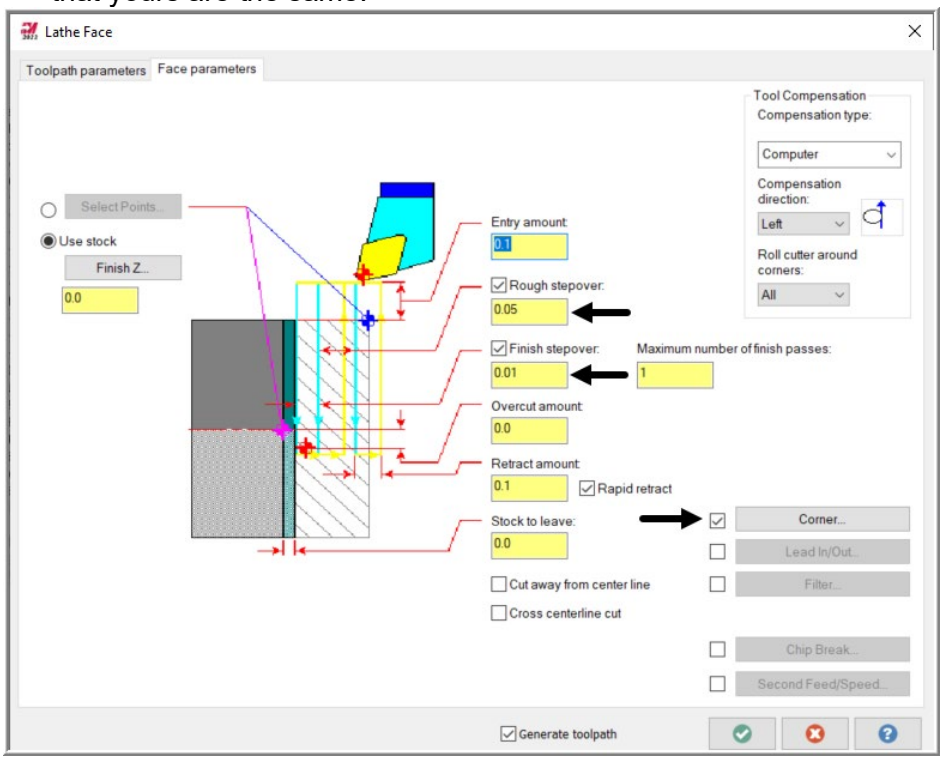
2. When the **Lathe Face Properties** window opens, **select Axis Combination/Spindle Origin** by clicking the small graphic. This will open the Axis Combination/Spindle Origin window. **Select Lower Right** as we will be using the lower turret to turn on the sub (right) spindle. Then hit the **green checkmark** to accept.



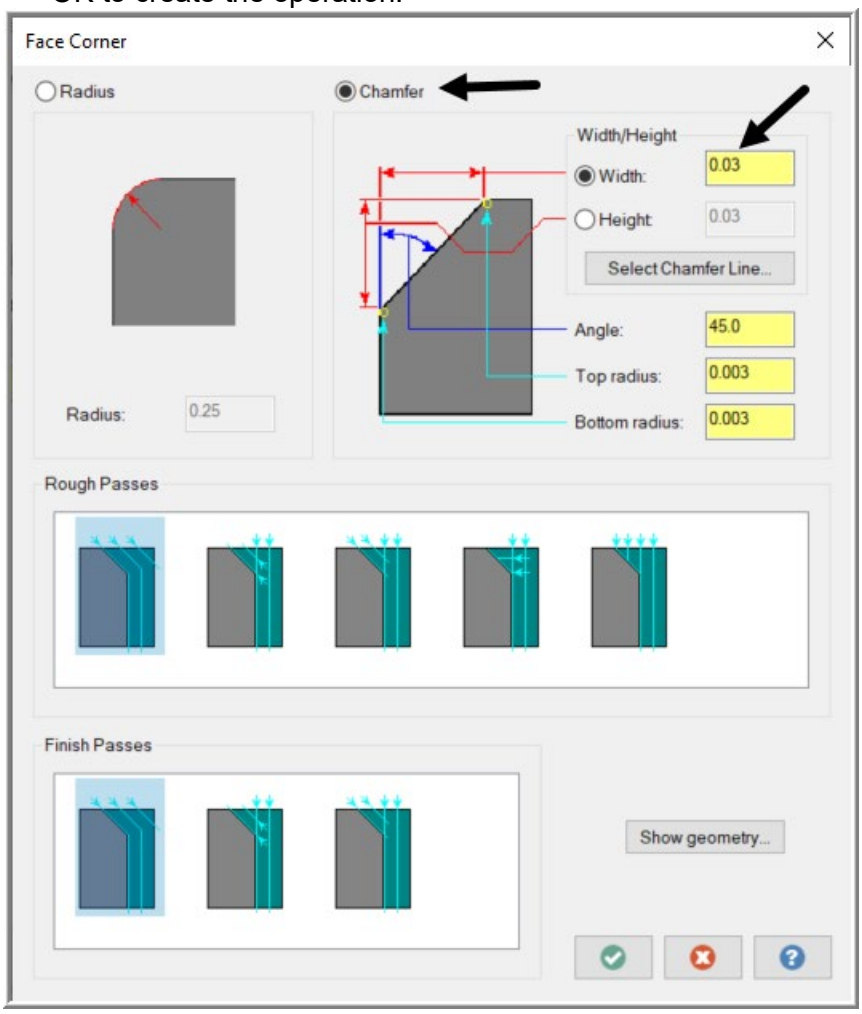
3. On the **Lathe Face Properties** window choose a tool that works on the right spindle. You must select a different tool than used on the lower turret left spindle as they will be facing opposite directions. The same lower turret tool cannot face on the left and right spindles. **Note:** Based on your actual machine, it may be necessary to change tool numbers.



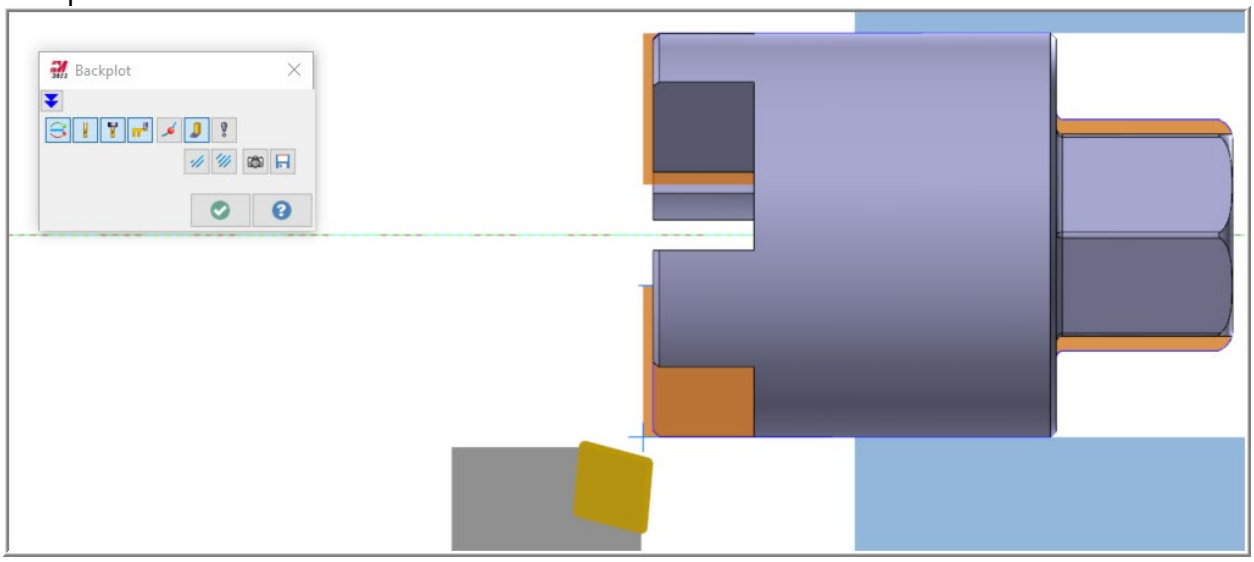
4. Make the following changes to the **Face Parameters** and enable the **Corner** option. Check that yours are the same.



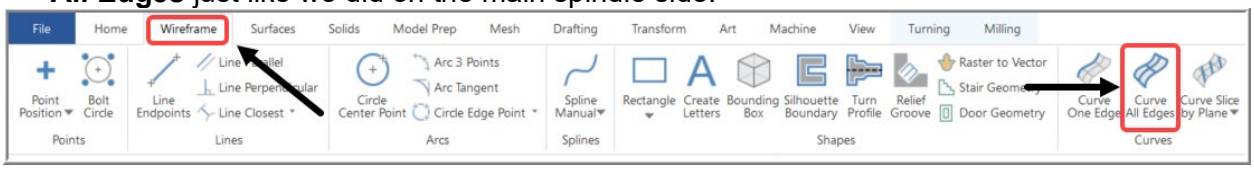
5. After enabling Corner, select it. Make the following changes on the Face Corner page. Hit OK to create the operation.



6. You can do a quick **backplot** again to check your toolpath. Select just the sub spindle facing op and run.

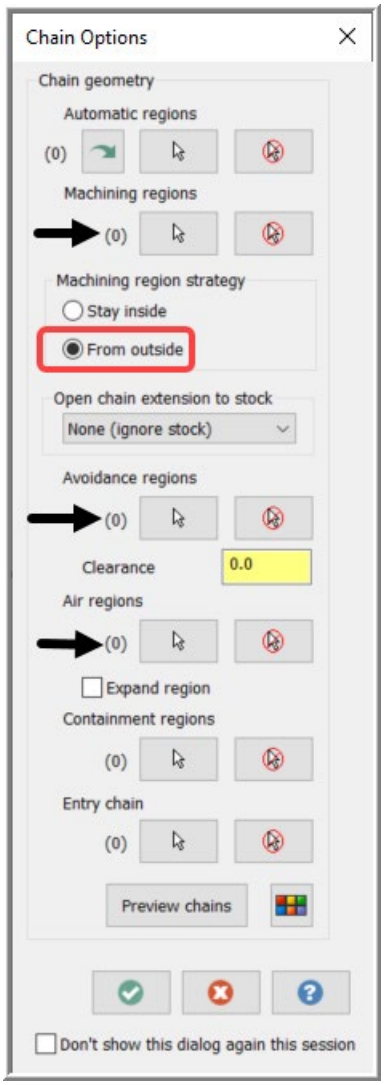


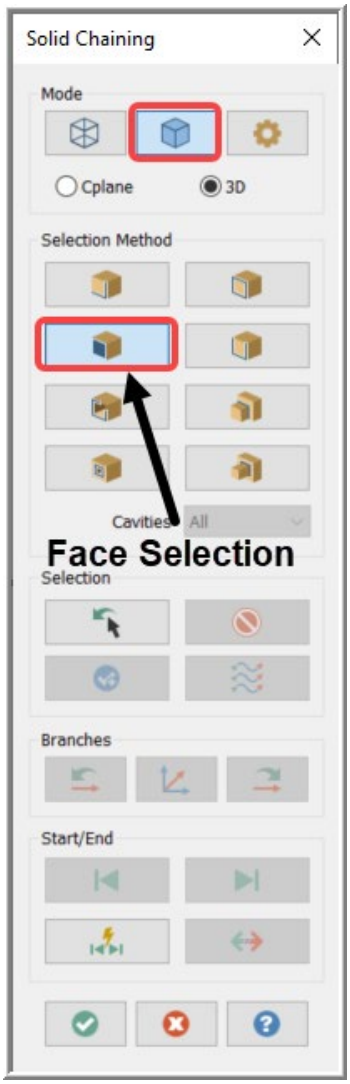
7. **Rough Milling**. We will rough this feature using a 2D High Speed Dynamic Milling operation. We will need some extra geometry for chaining in this operation. Create **Curves All Edges** just like we did on the main spindle side.



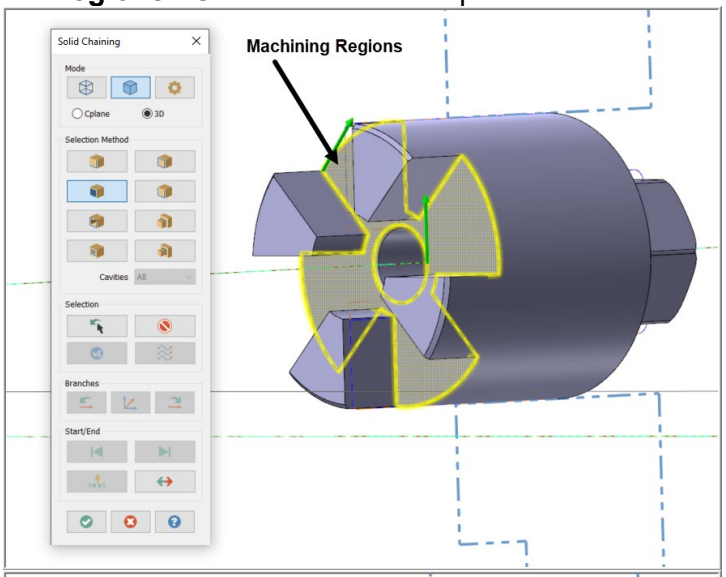
8. Select **Dynamic Mill** from the **Milling 2D** menu.

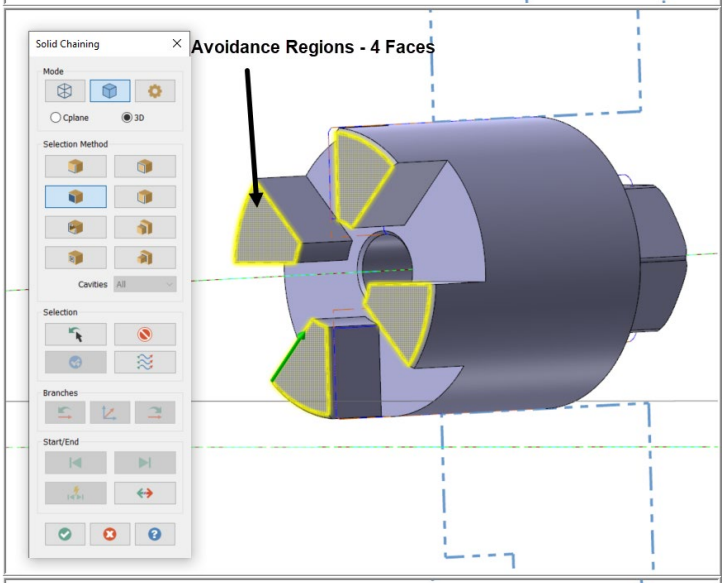
9. **Chain the geometry**. There are multiple ways to do this, but for this exercise we will use the following options shown with black arrows. Be sure to check the **From Outside** radio button. When selecting the regions, we will use the **Face Selection** option under the **Solids** mode.

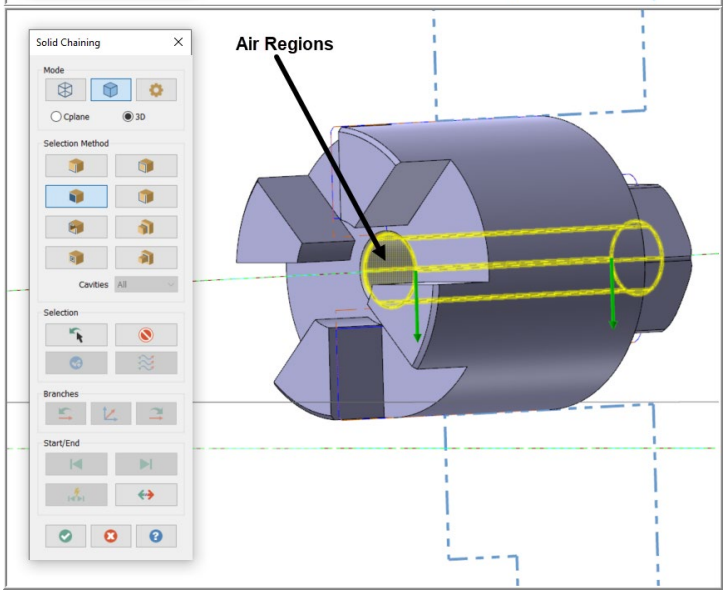




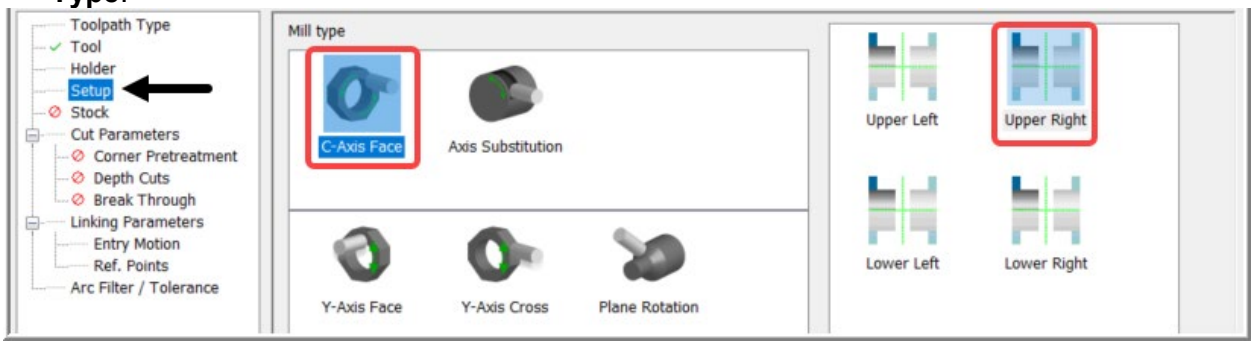
10. Make the following selections for **Machining Regions**, **Avoidance Regions**, and **Air Regions**. Green check to accept each set of selections and again when all selected.



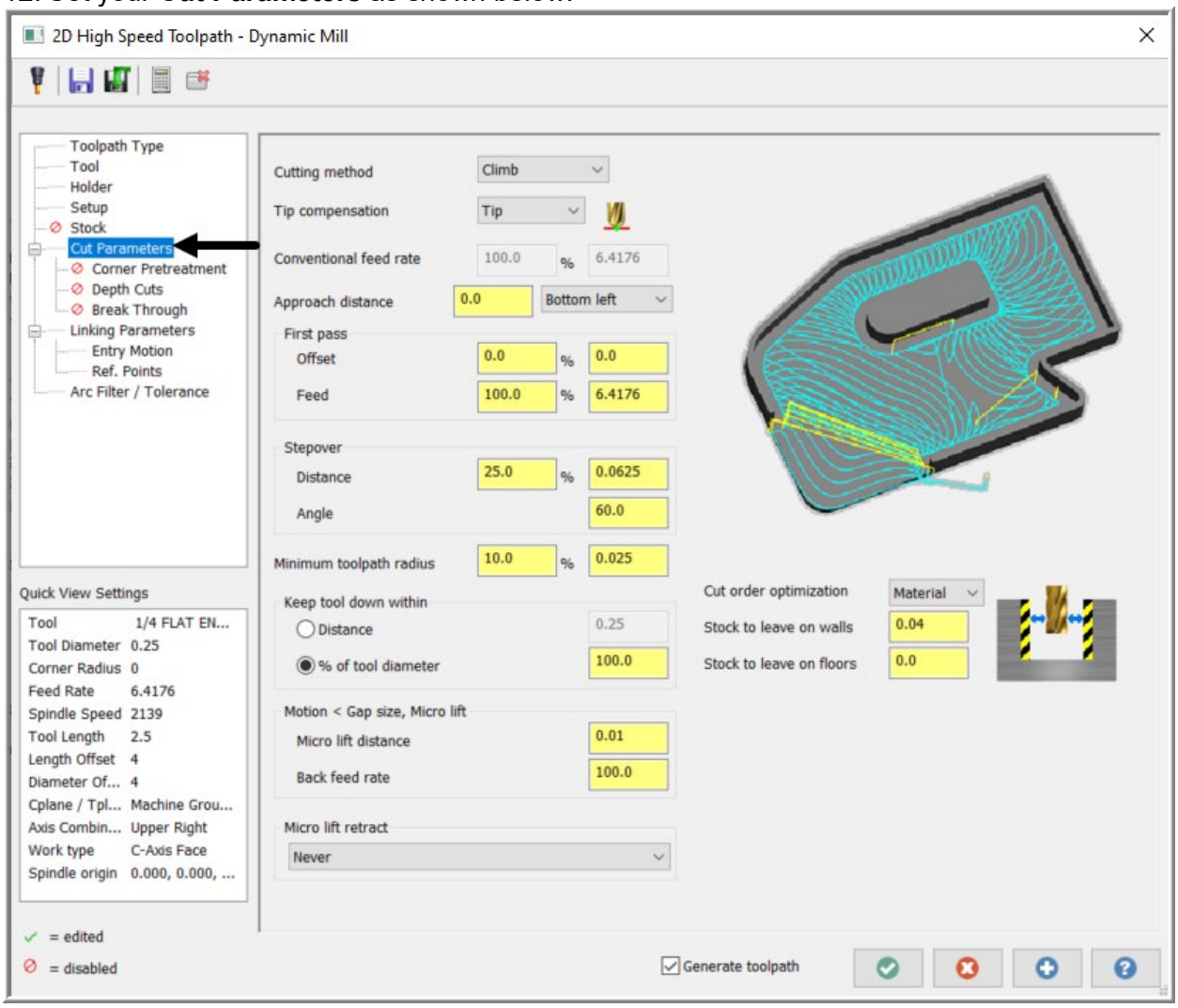




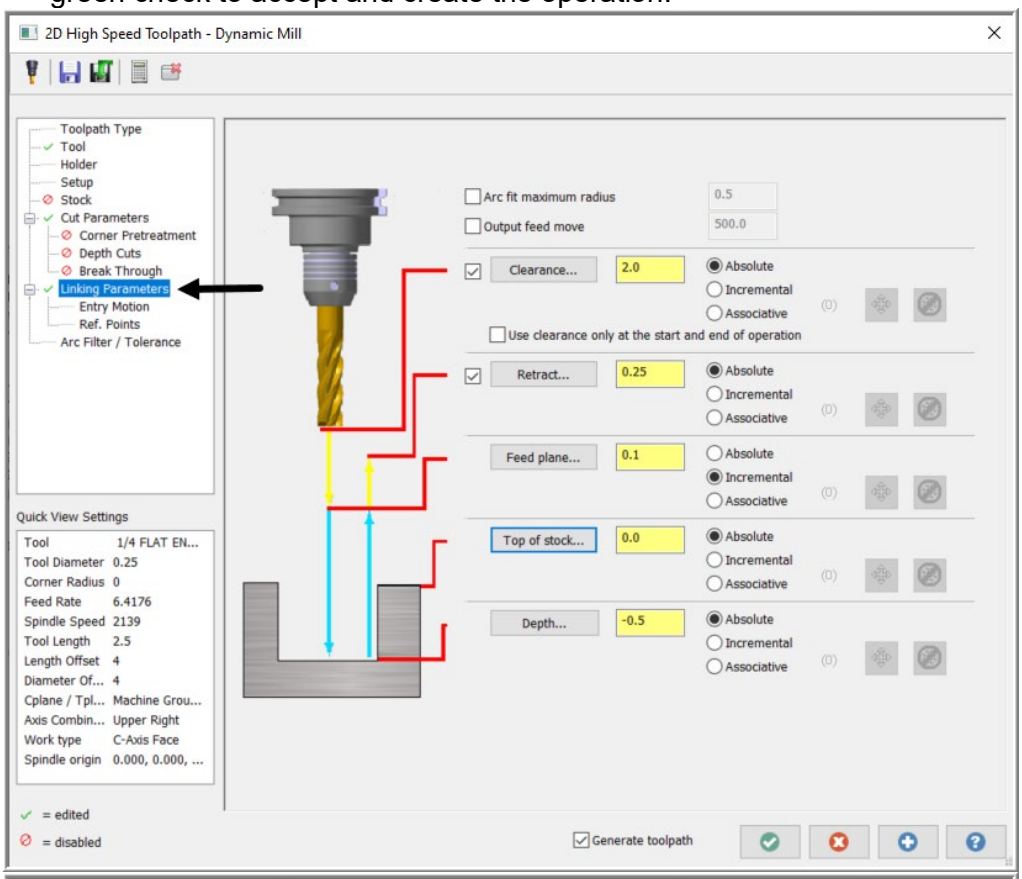
11. Select a **1/4" Flat Endmill** from the library and **change the cutting length** to **0.6".** From the **Setup** menu select **Upper Right** as your axis combination and **C-Axis Face** as your **Mill Type**.

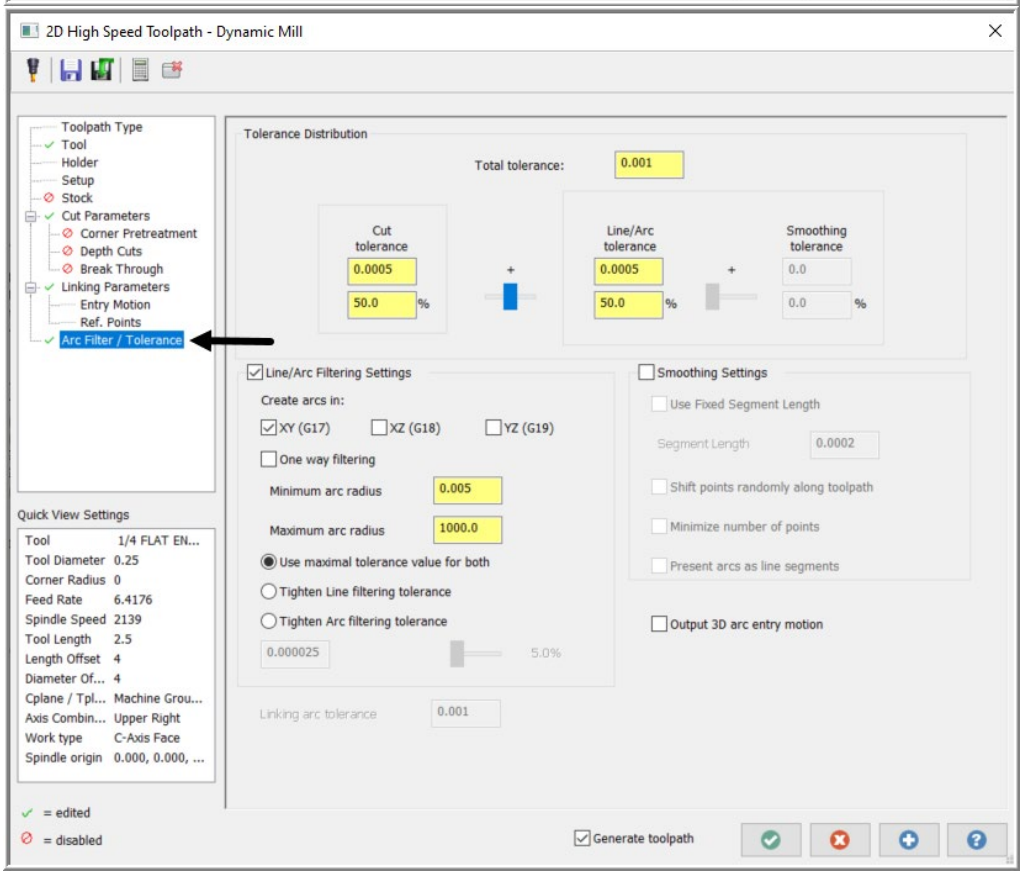


12. Set your **Cut Parameters** as shown below.

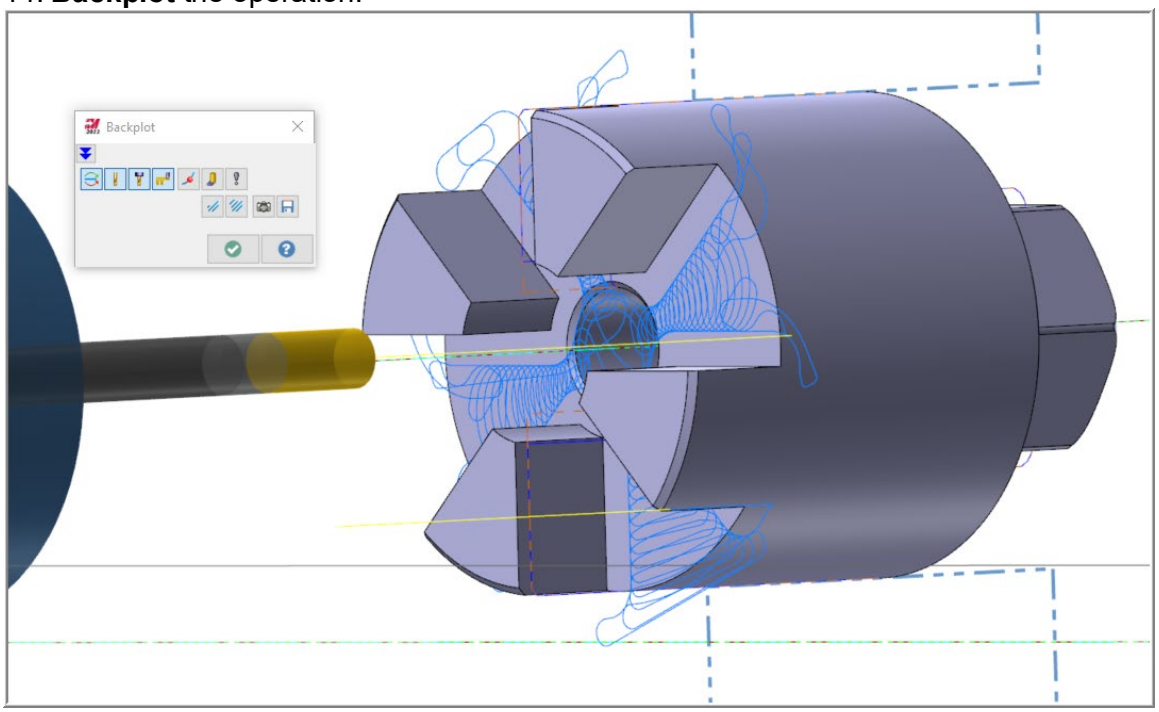


13. Set you **Linking Parameters** and **Arc Filter/Tolerance** parameters as shown below, then green check to accept and create the operation.



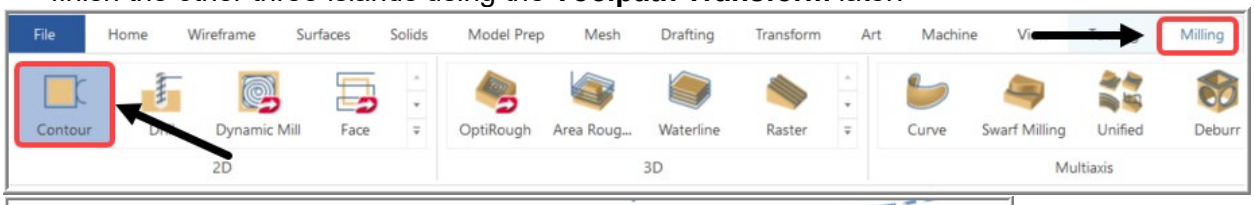


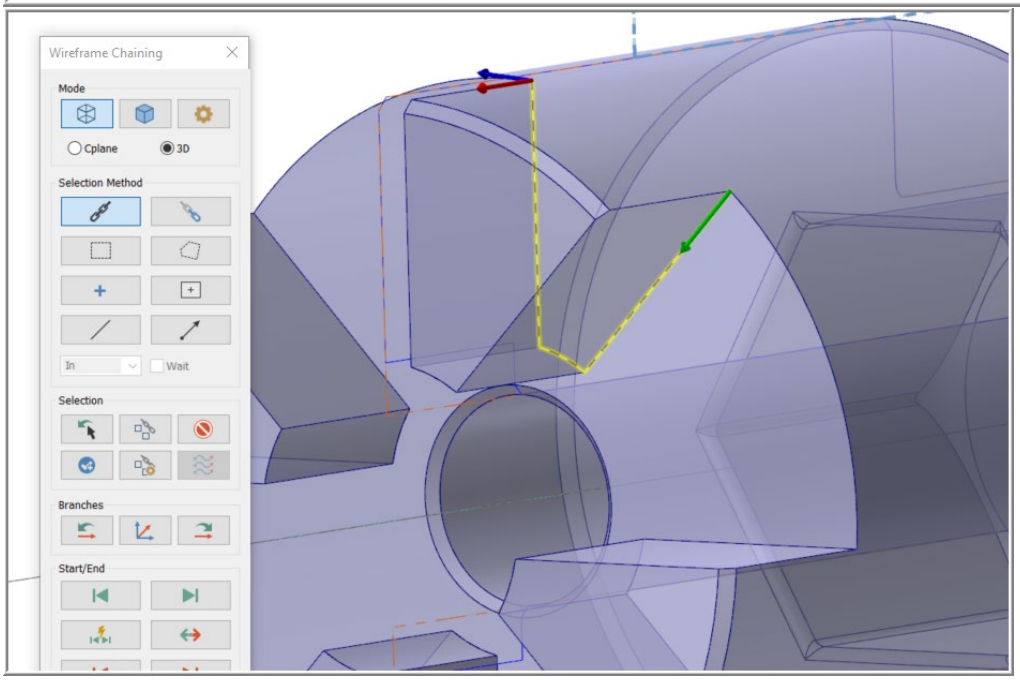
14. **Backplot** the operation.



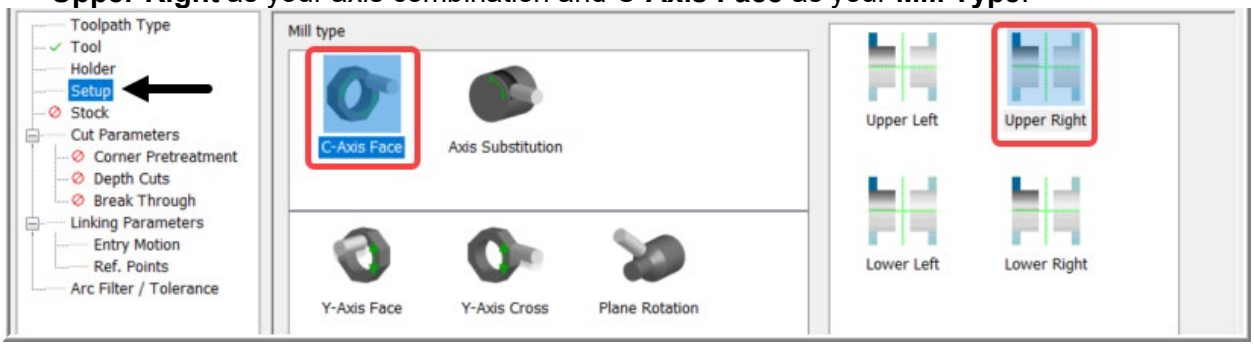
15. **Save** your file.

16. **Finish Mill** the pocket. Select **Milling > Contour**, then chain one of the islands. We will finish the other three islands using the **Toolpath Transform** later.

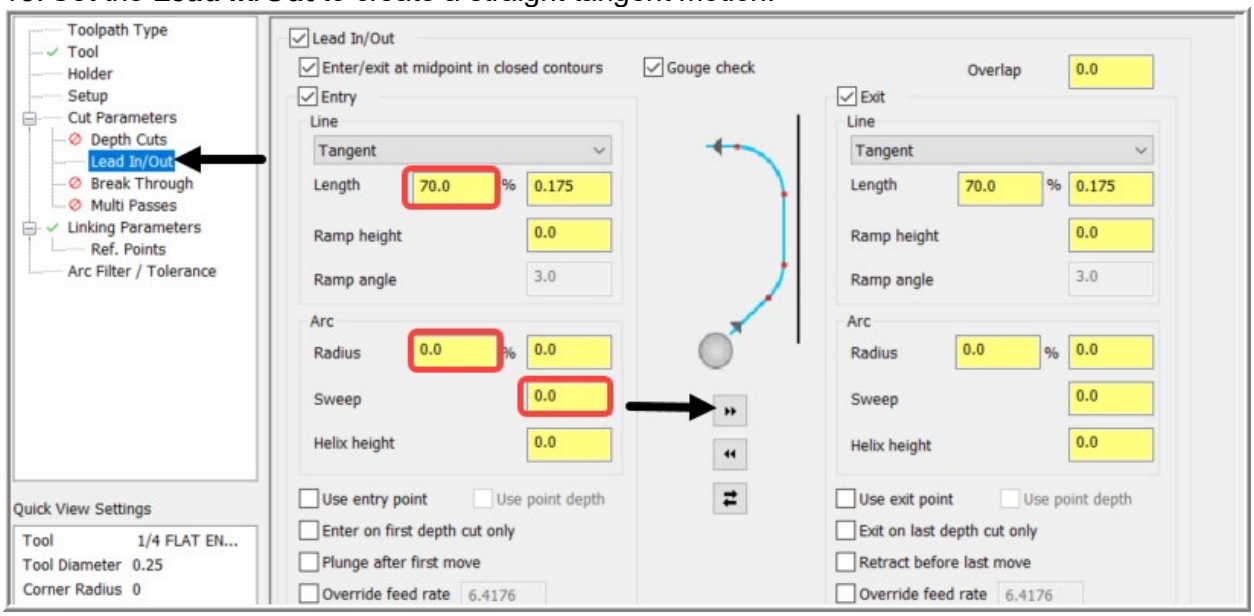




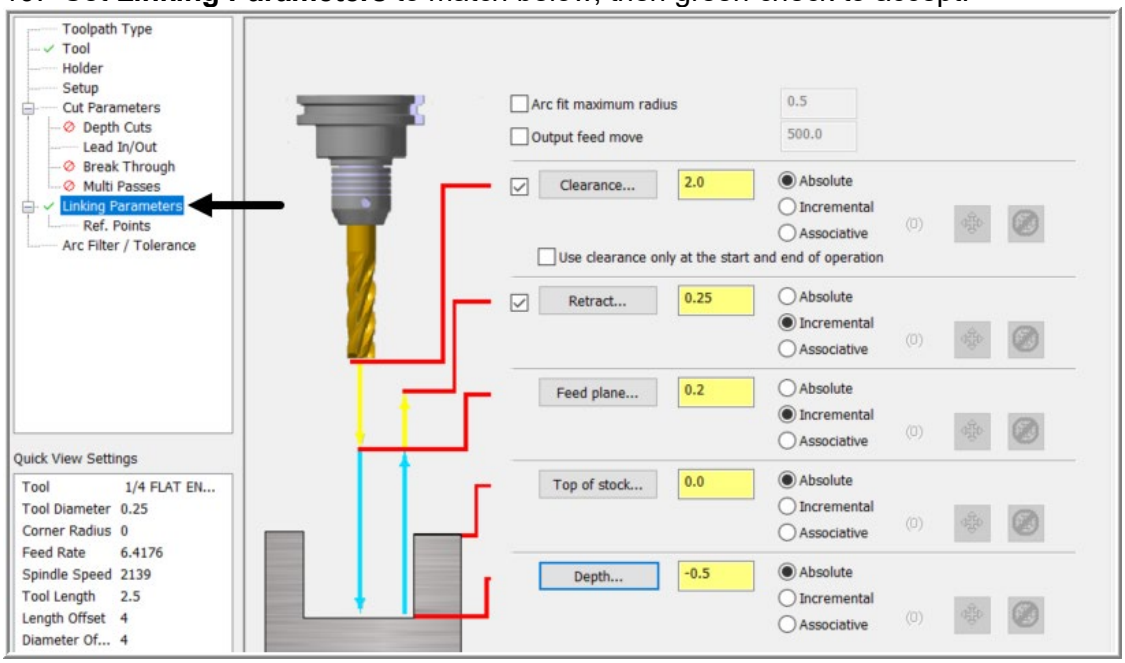
17. Use the same **1/4" Flat Endmill** for the finishing operation. From the **Setup** menu select **Upper Right** as your axis combination and **C-Axis Face** as your **Mill Type**.



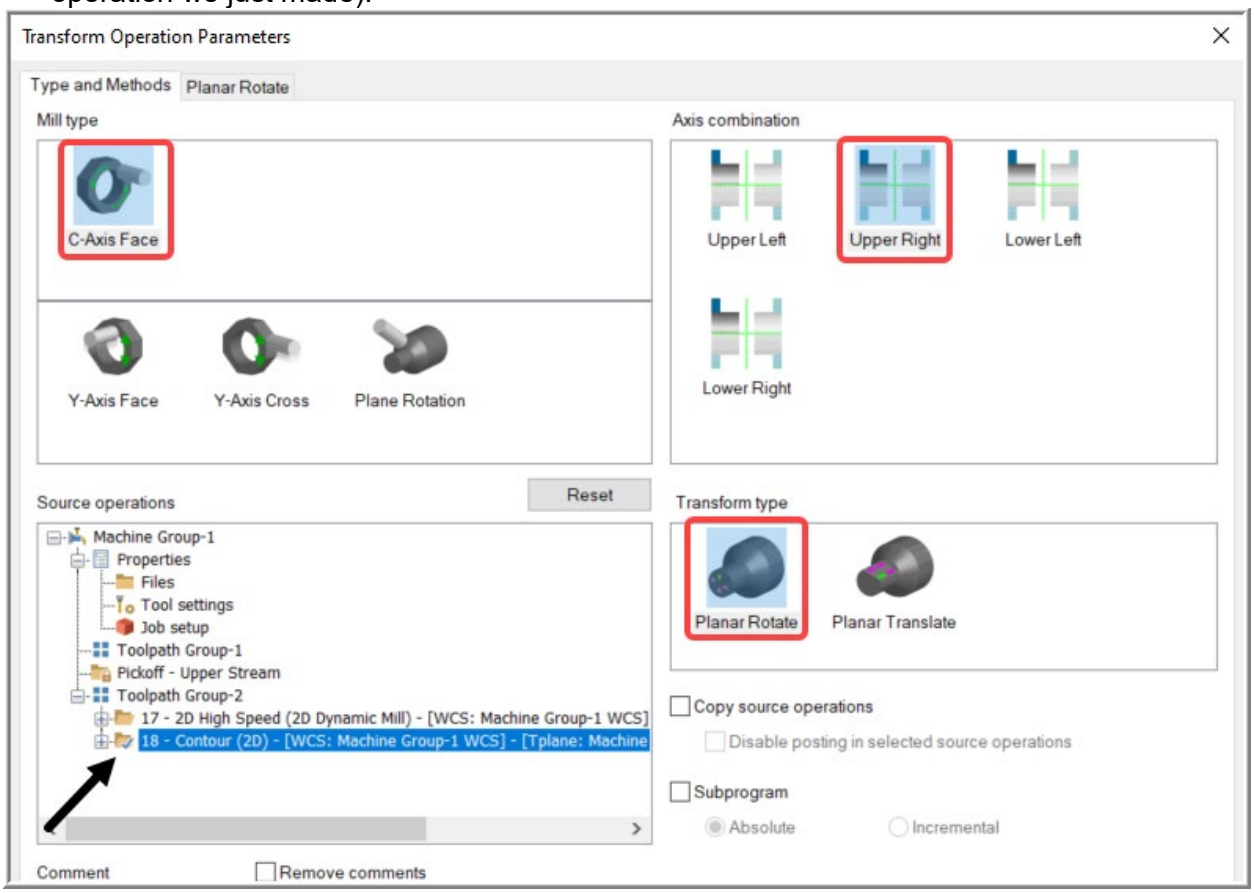
18. Set the **Lead In/Out** to create a straight tangent motion.



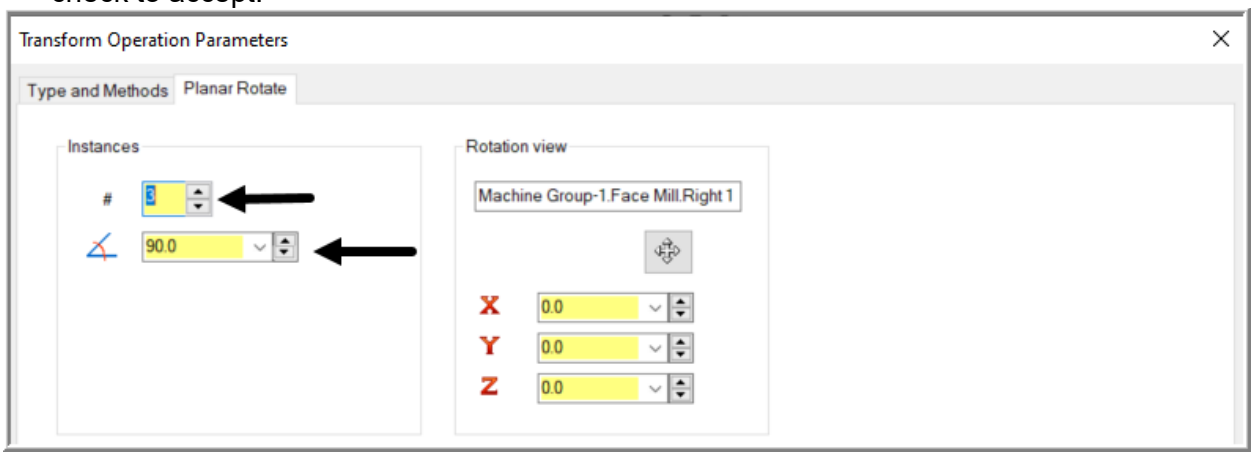
19. Set **Linking Parameters** to match below, then green check to accept.



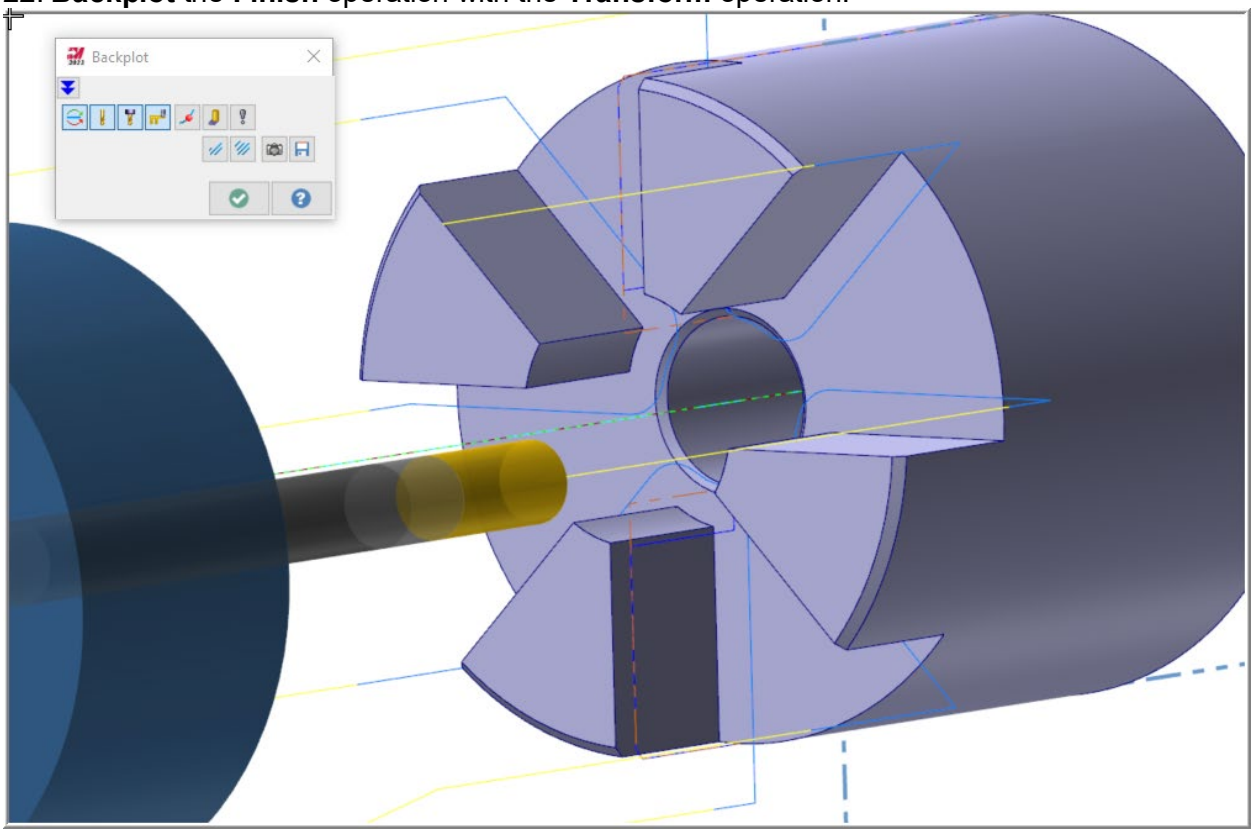
20. **Transform** the finish operation to machine the three remaining islands. Select **Milling > Toolpath Transform**, set **Mill Type** to **C-Axis Face**, **Axis Combination** to **Upper Right**, **Transform Type** to **Planar Rotate**, and **Source Operations** to **Operation 18** (the finish operation we just made).



21. On the **Planar Rotate** tab set the number of instances to 3 and set the angle to 90.0. Green check to accept.

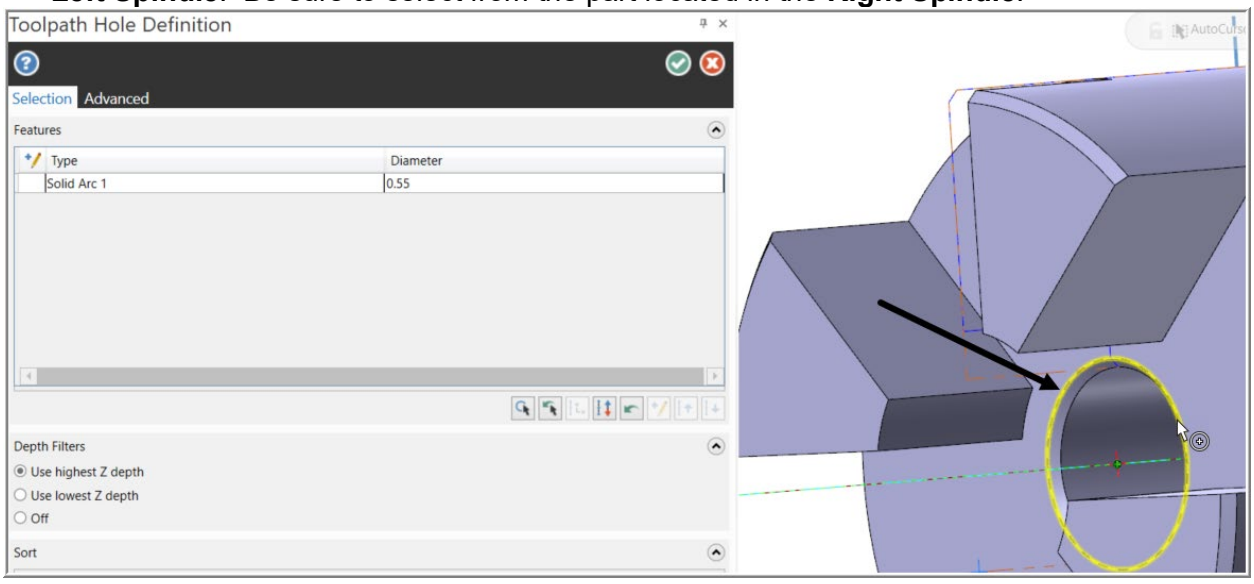


22. **Backplot** the **Finish** operation with the **Transform** operation.

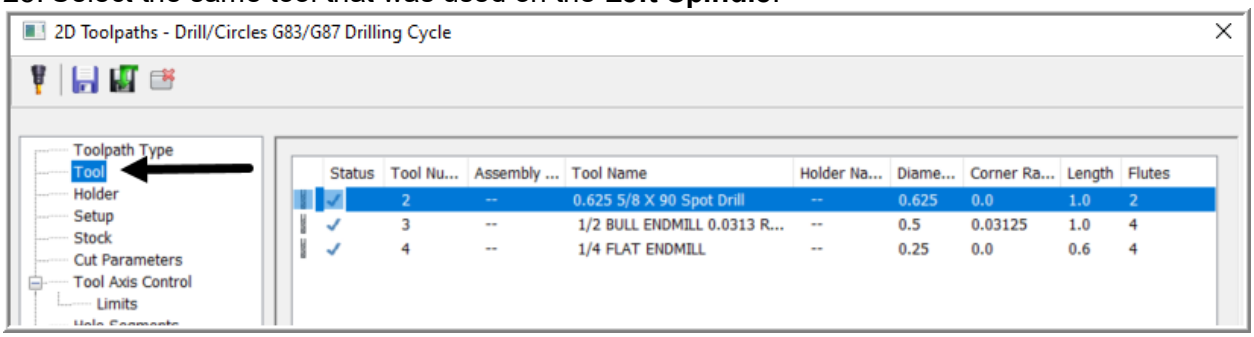


23. **Drill** toolpath to countersink backside of thru hole.

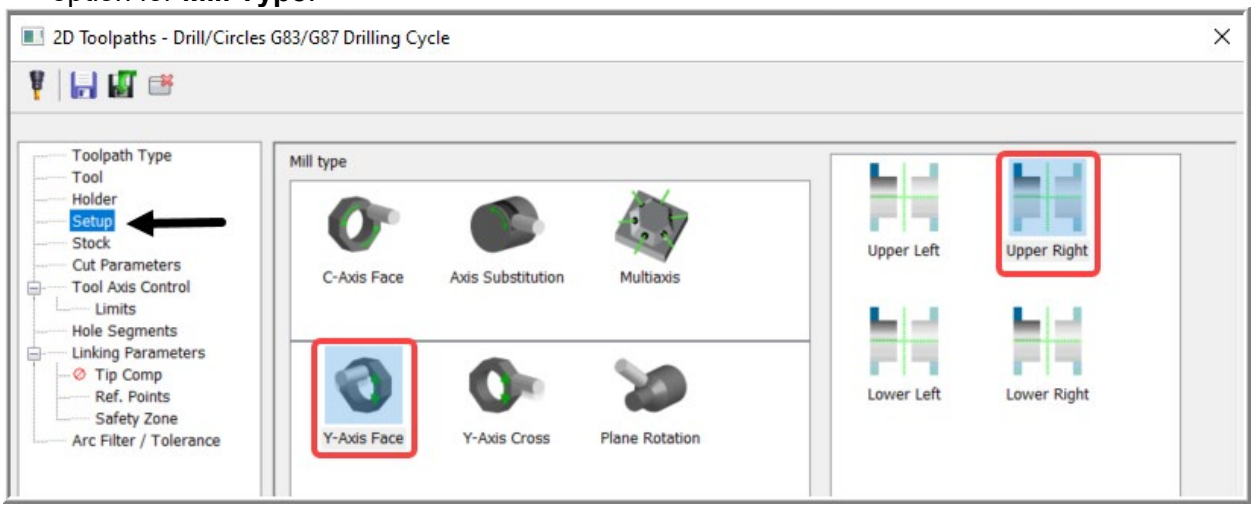
24. Select the outer diameter of the chamfered hole, similar to what was already done on the **Left Spindle**. Be sure to select from the part located in the **Right Spindle**.



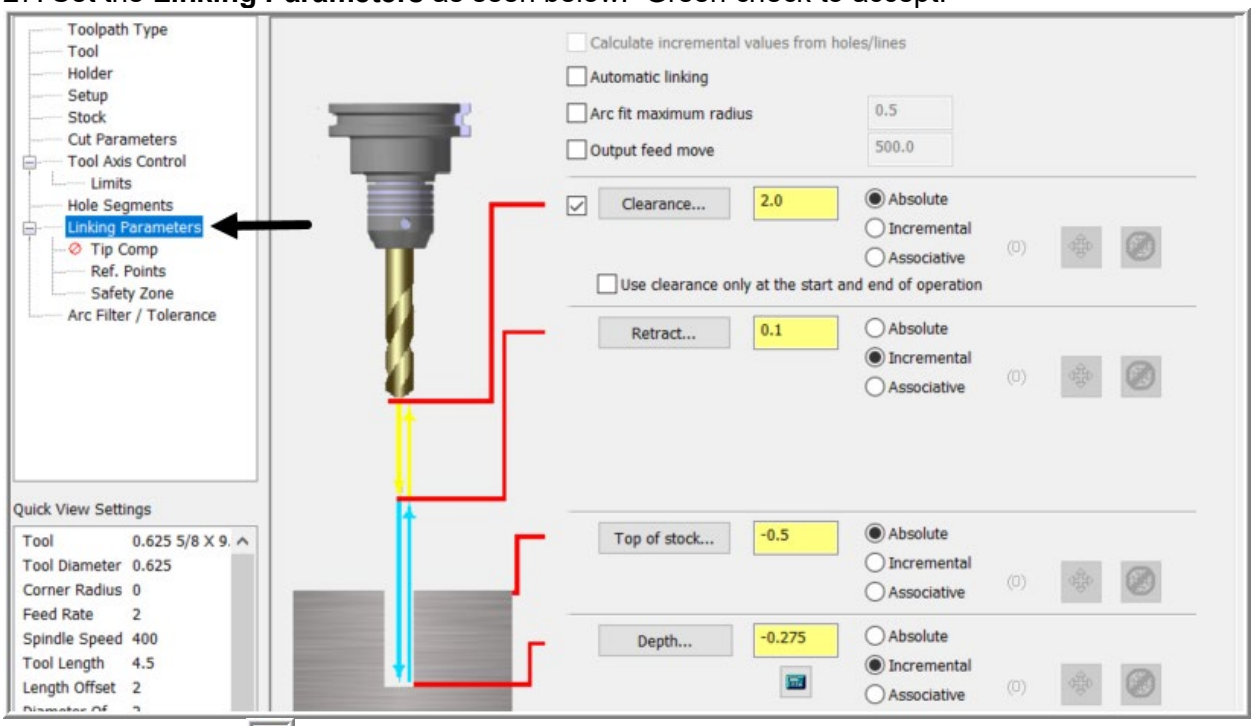
25. Select the same tool that was used on the **Left Spindle**.



26. From the **Setup** menu select the **Upper Right** axis configuration and the **Y-Axis Face** option for **Mill Type**.



27. Set the **Linking Parameters** as seen below. Green check to accept.

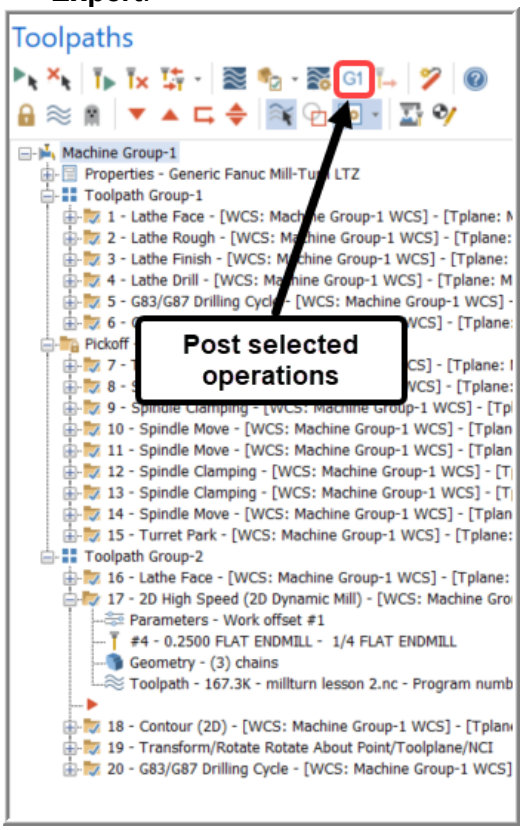


28. **Save** your file.

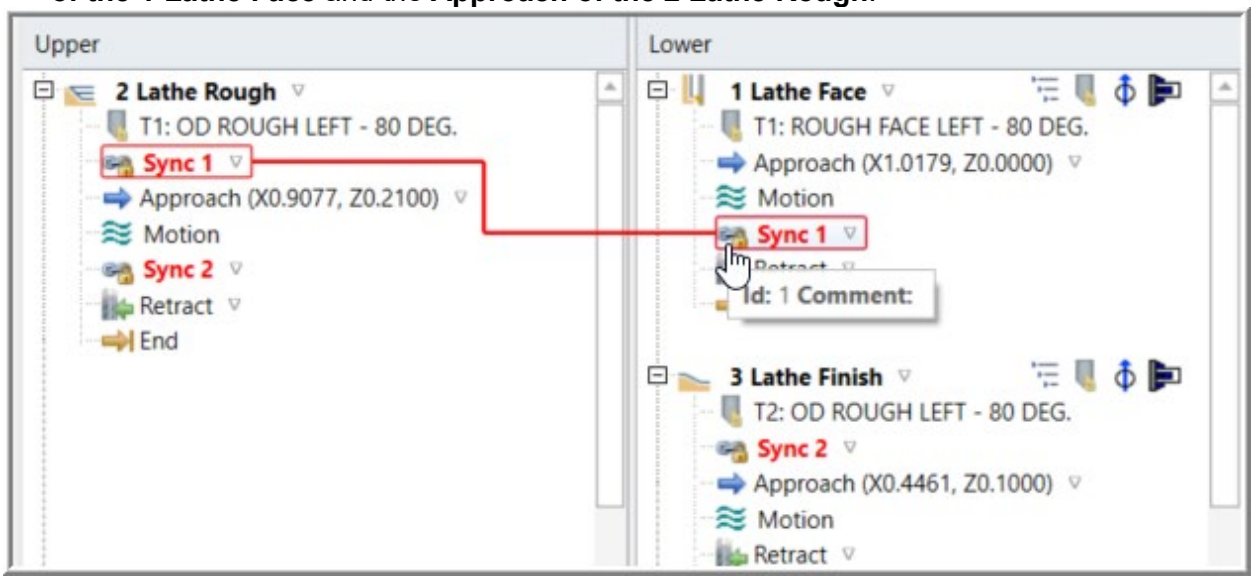
TASK 7:
MACHINE SYNCING

➲ In this task you will work in Mastercam Code Expert to set the machine syncs.

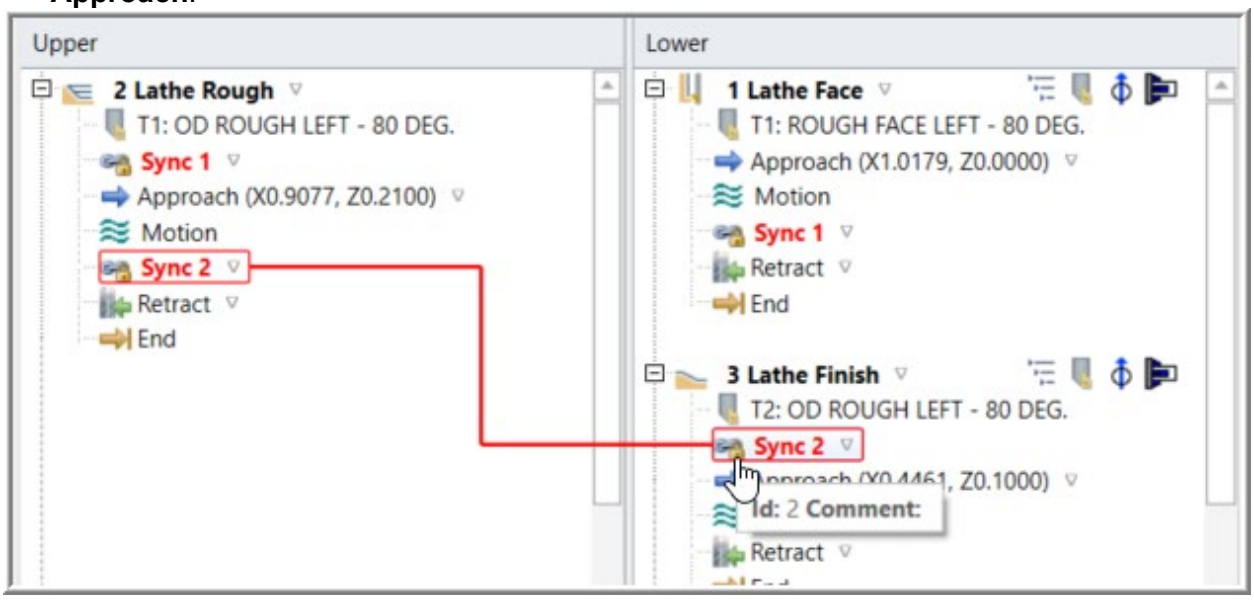
1. Select the **Main Viewsheet-1** to display both spindles. In the **Operations Manager** left click Machine Group -1 to **select all operations**. Select the **Post Selected Operations** button. This will generate the IOF file and open a Sync Manager window in **Mastercam Code Expert**.



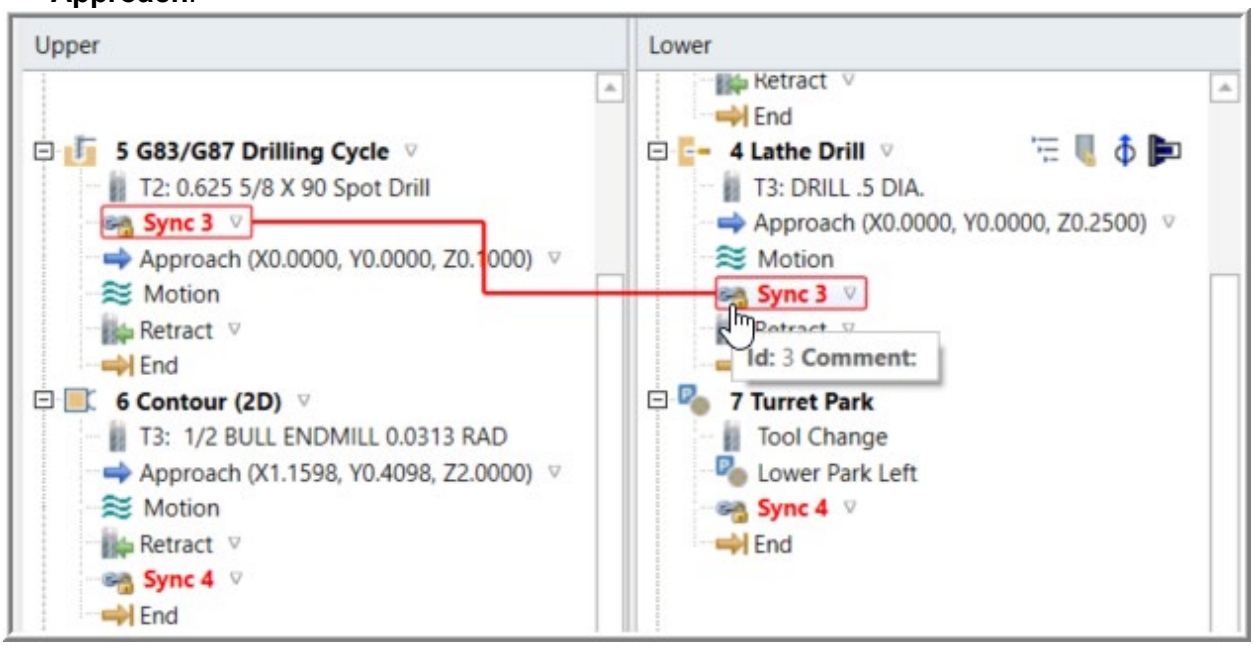
2. **Code Expert**. Next, we will create all the sync points. Create **Sync 1** between the **Retract of the 1 Lathe Face** and the **Approach of the 2 Lathe Rough**.



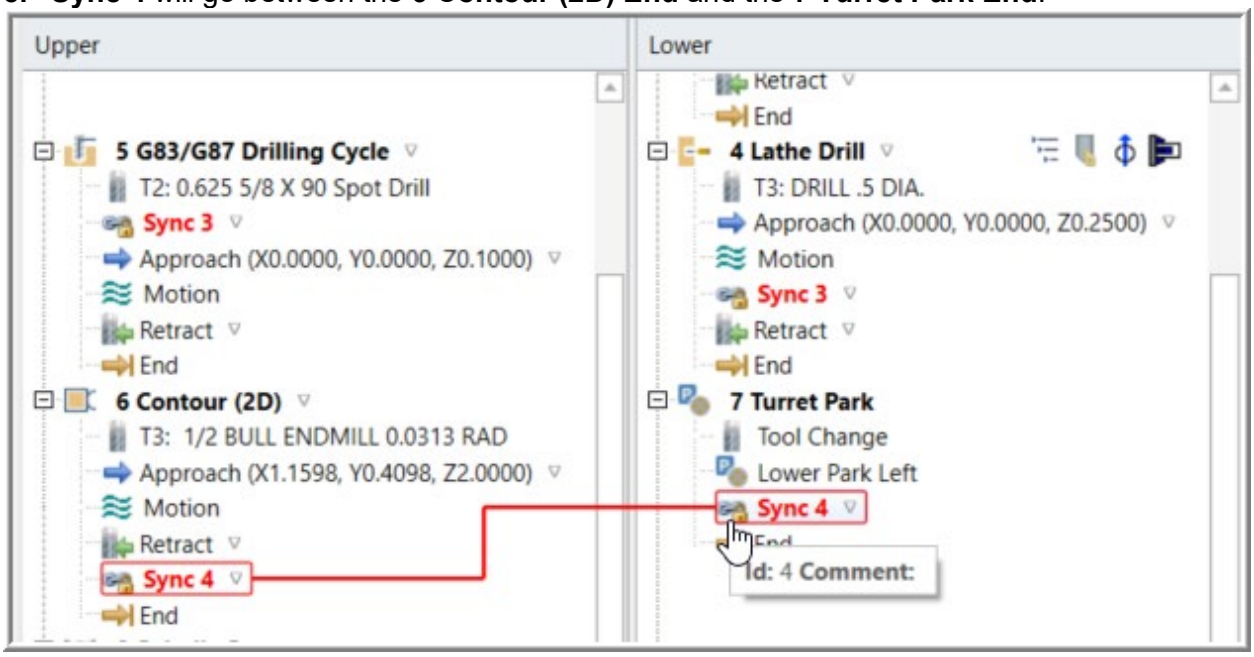
3. Next, create **Sync 2** between the **2 Lathe Rough Retract** and the **3 Lathe Finish Approach**.



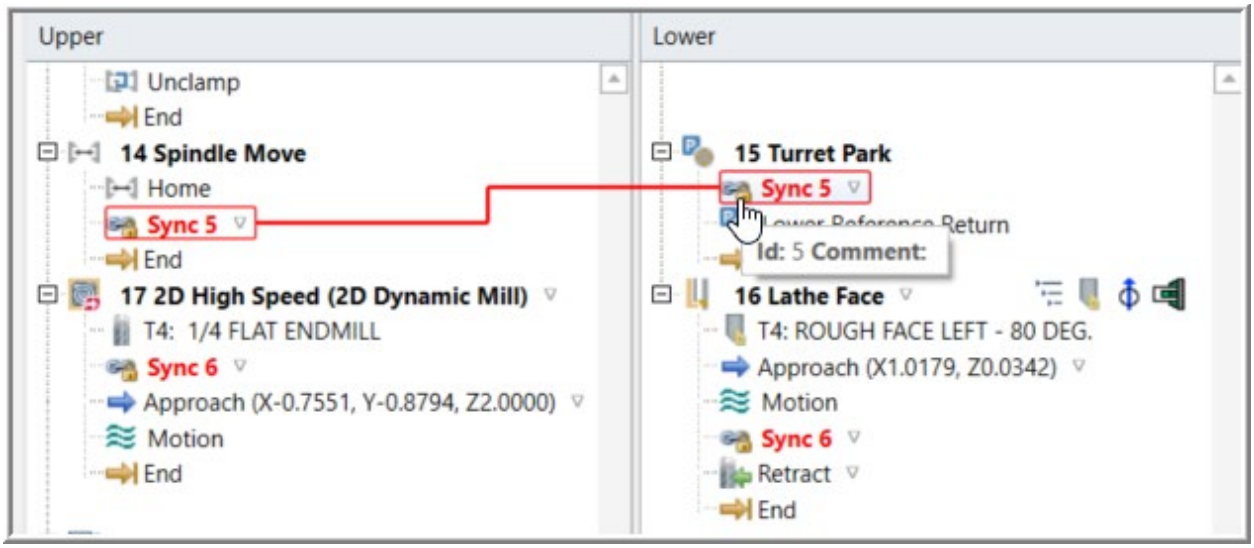
4. **Sync 3** will be between the **4 Lathe Drill Retract** and the **5 G83/G87 Drilling Cycle Approach**.



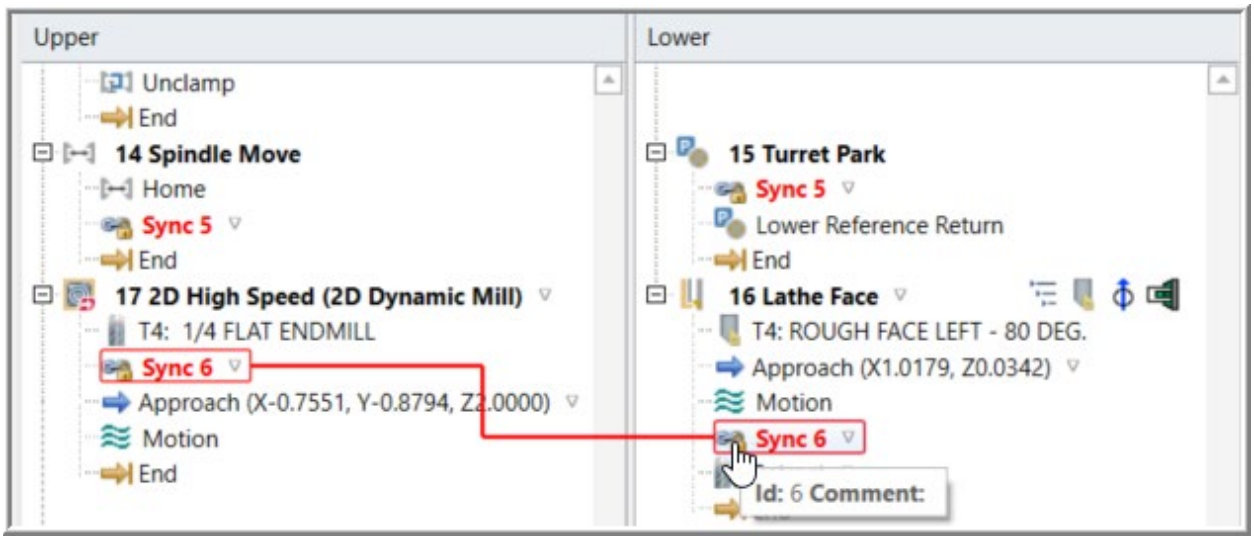
5. **Sync 4** will go between the **6 Contour (2D) End** and the **7 Turret Park End**.



6. **Sync 5** will be between the **14 Spindle Move End** and the **15 Turret Park Lower Reference Return**.



7. **Sync 6** will be the final sync located between the **16 Lathe Face** performed by the lower turret and the **17 2D High Speed (2D Dynamic Mill)** which will be machine by the upper turret.



8. Save you work.  Saving from the Code Expert Sync Manager also writes back to the Mastercam file. The colored light at the top of the Operations Manager will indicate whether the file has been saved or not.

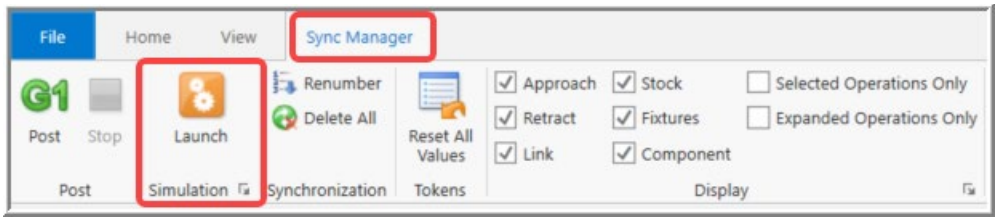
Not saved.

Saved.

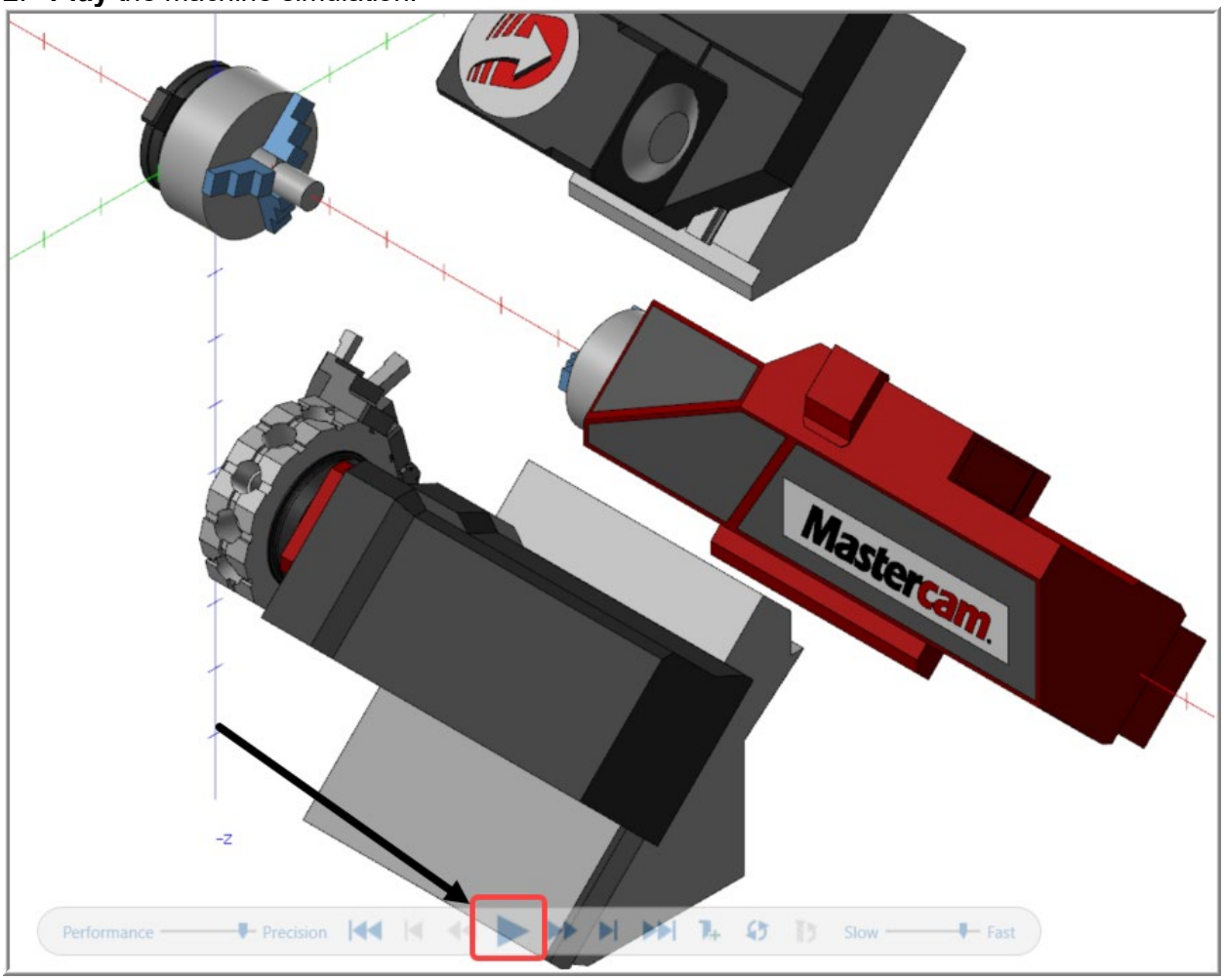
TASK 8:
MACHINE SIMULATION:

➲ In this task you will run a complete machine simulation to check for potential problems.

1. In Code Expert, from the **Sync Manager** tab click on the **Launch Icon** to launch machine simulation.



2. **Play** the machine simulation.

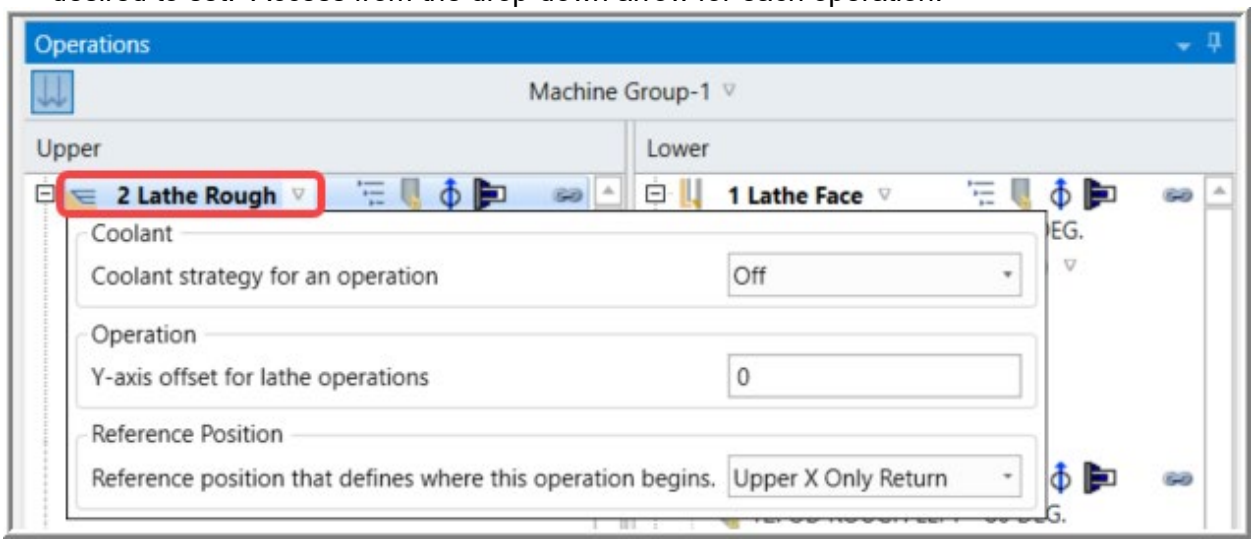


3. Adjust the simulation speed as needed. When finished, **Exit** and return to Code Expert to post the NC code.

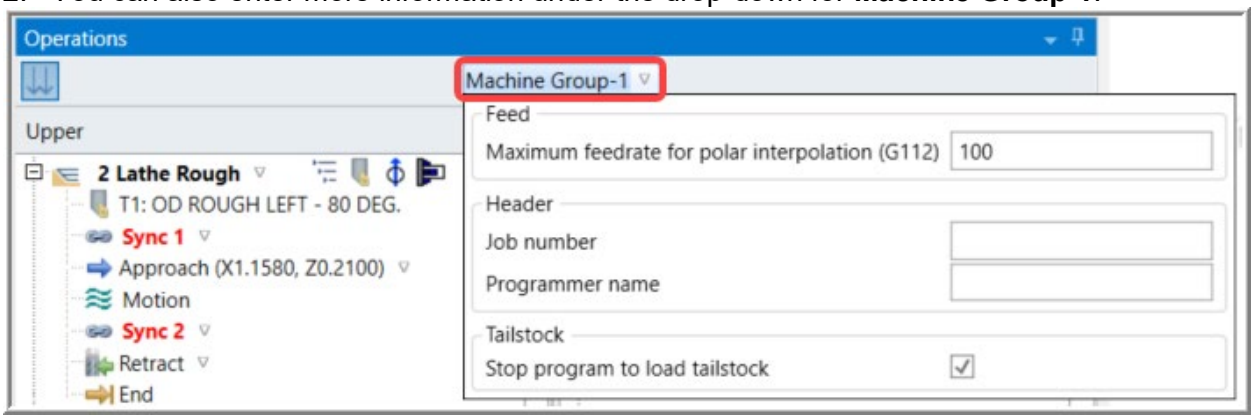
TASK 9:
POSTING THE CODE:

➲ In this task you use code expert to post the NC program.

1. Before we post the code, we have a few extra settings to consider. Each operation has an independent **Coolant Strategy** selection as well as Y-axis offsets and Reference Positions if desired to set. Access from the drop-down arrow for each operation.

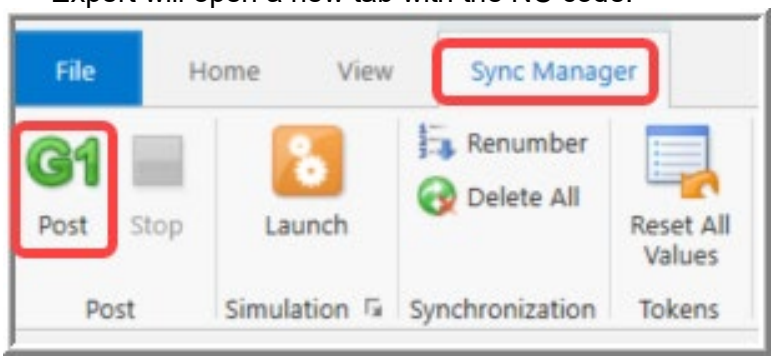


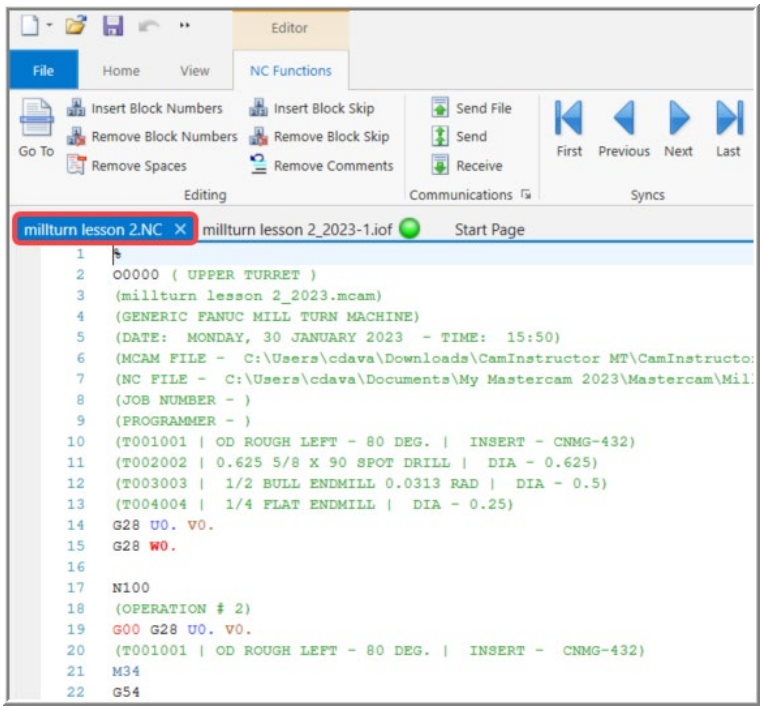
2. You can also enter more information under the drop-down for **Machine Group-1**.



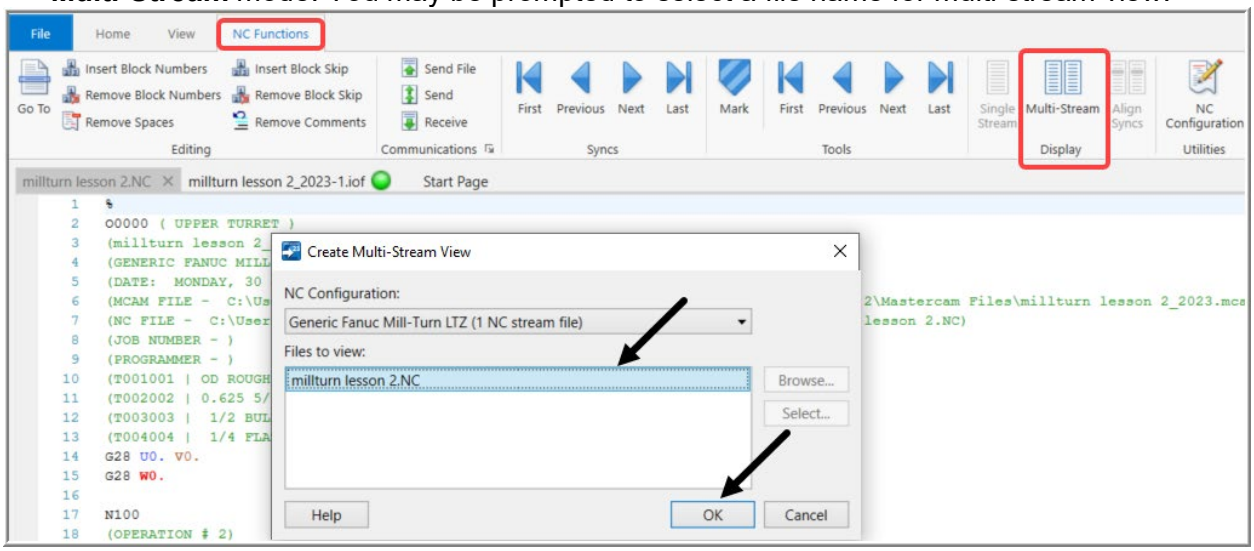
3. **Save** your file.

4. **Post** the code by pressing the **G1 button** located under the **Sync Manager** tab. Code Expert will open a new tab with the NC code.

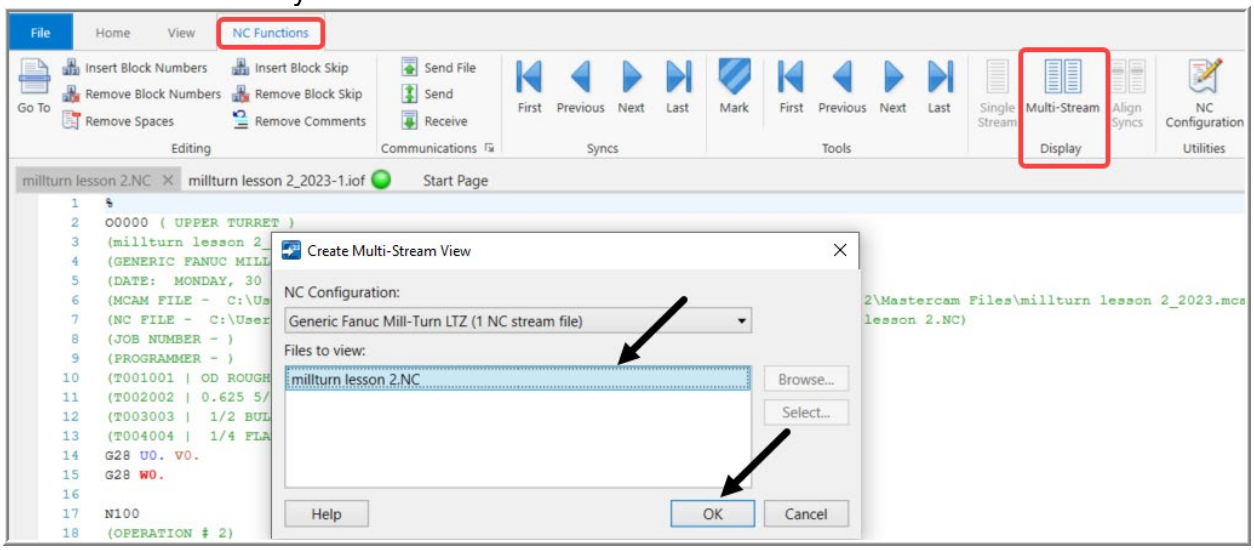




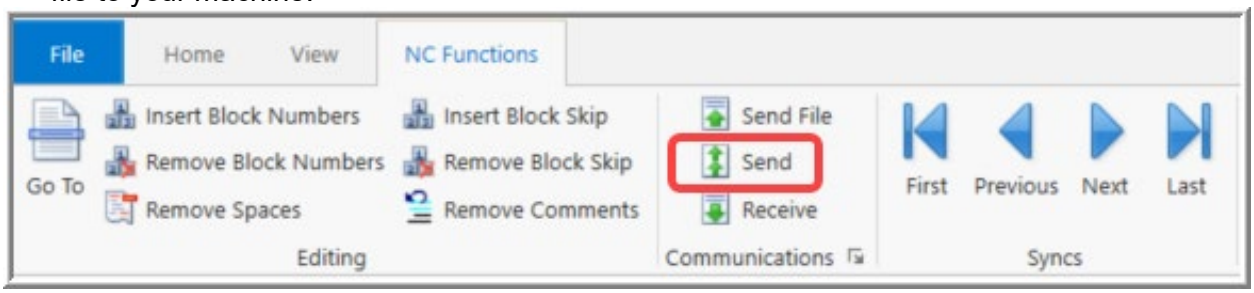
5. By default, the code is displayed in single stream. Under the **NC Functions** tab, to switch to **Multi-Stream** mode. You may be prompted to select a file name for multi-stream view.



6. You will see all the sync points you created. Hovering over one sync point will highlight it in both streams if they are both visible on-screen.



7. Your code is now ready to run. If you have communications set up, hit send to send the NC file to your machine.



8. This concludes Mill-Turn Lesson 2.