Autodesk Fusion Getting Started tutorial series

Video 15 – Manufacturing

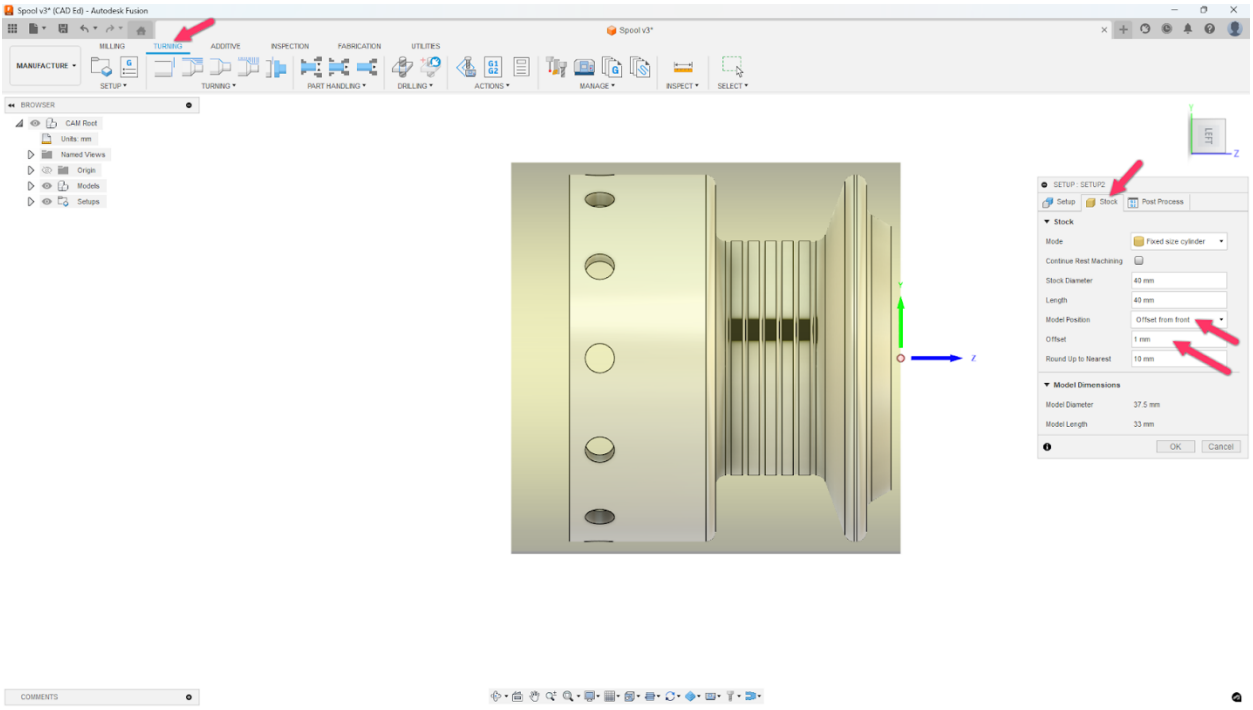

Exhibit 1

In this exhibit, we have created a new Setup under the Turning tab in Manufacturing. Click on the Stock tab and change the Model Position to Offset From Front. Then, change the Offset distance to 1mm and press OK to complete the Setup.

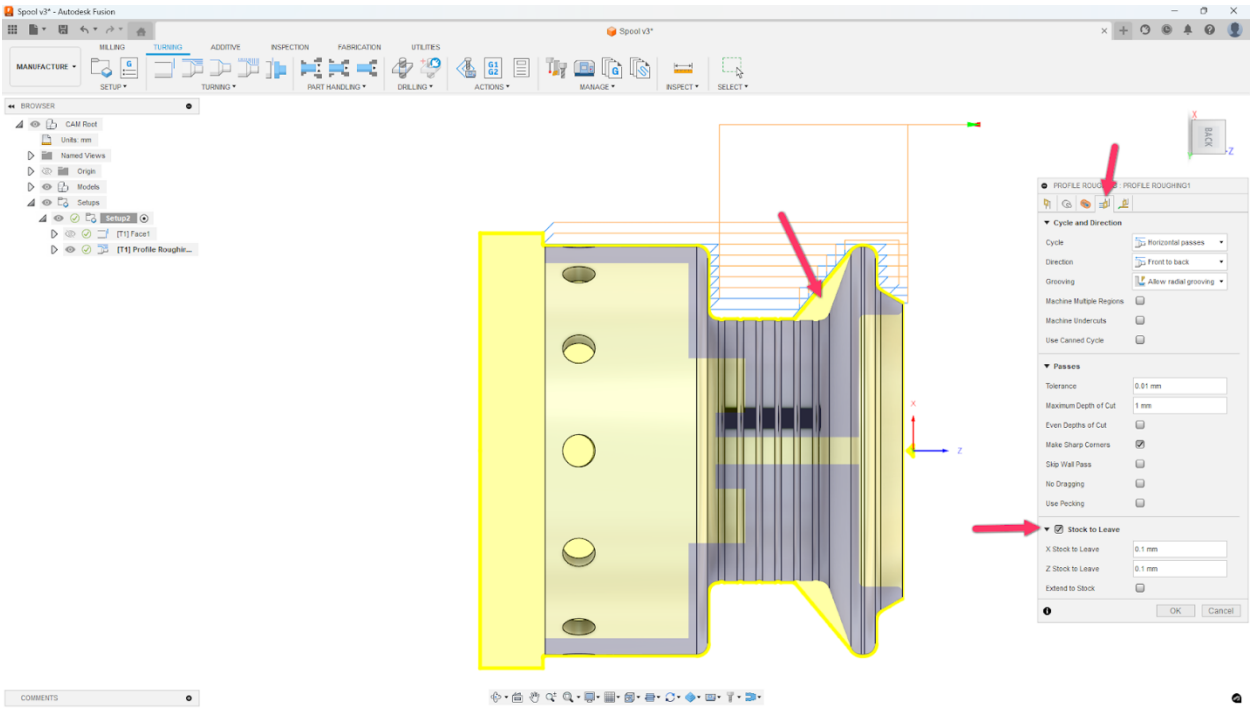

Exhibit 2

In this exhibit, we have created a Profile Roughing toolpath. In the Passes Tab, notice that Stock To Leave is turned on and it leaves a small amount of material to be left for a finishing pass to remove later. Also notice that the toolpath coil not remove all the material underneath the top of the Spool.

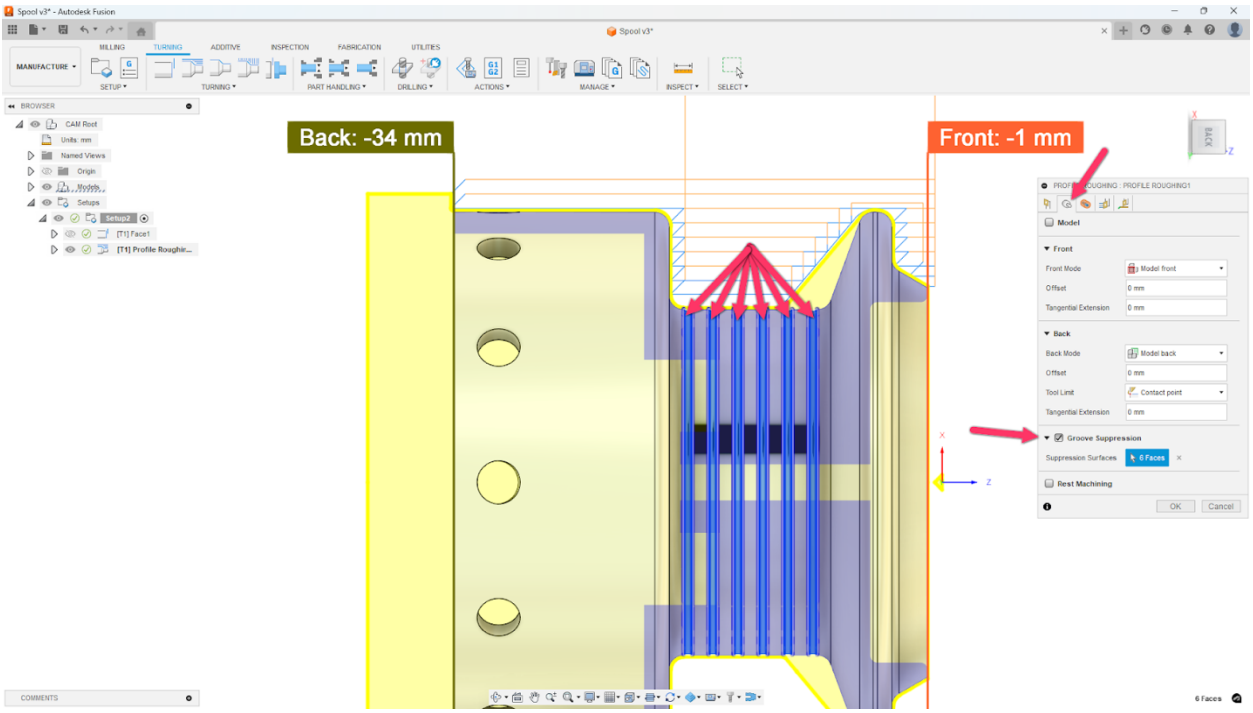

Exhibit 3

In this exhibit, we have edited the Profile Roughing toolpath and selected the Geometry Tab. Turn on Groove Suppression and then select the 6 faces of the small groves on the Spool, then press OK.

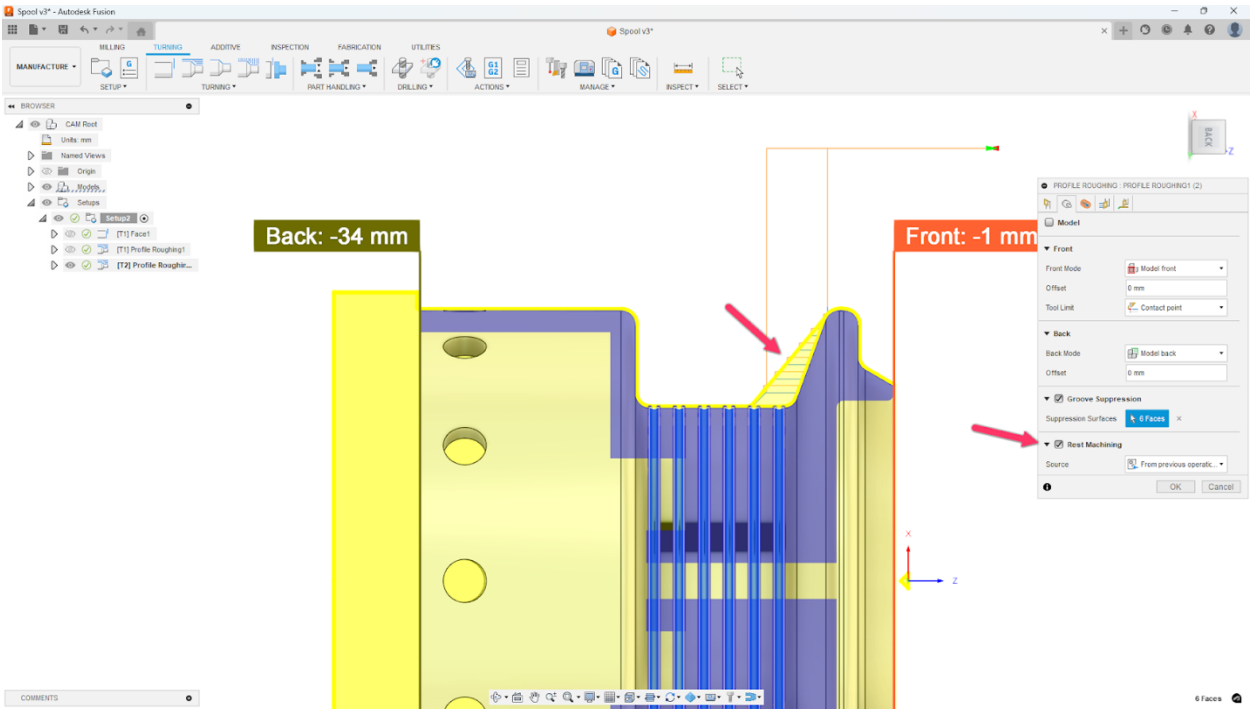

Exhibit 4

In this exhibit, we duplicated the first Profile Roughing toolpath and have then edited the new toolpath. Under the Geometry Tab, select Rest Machining to make the toolpath only machine the area left over from the previous machining operation.

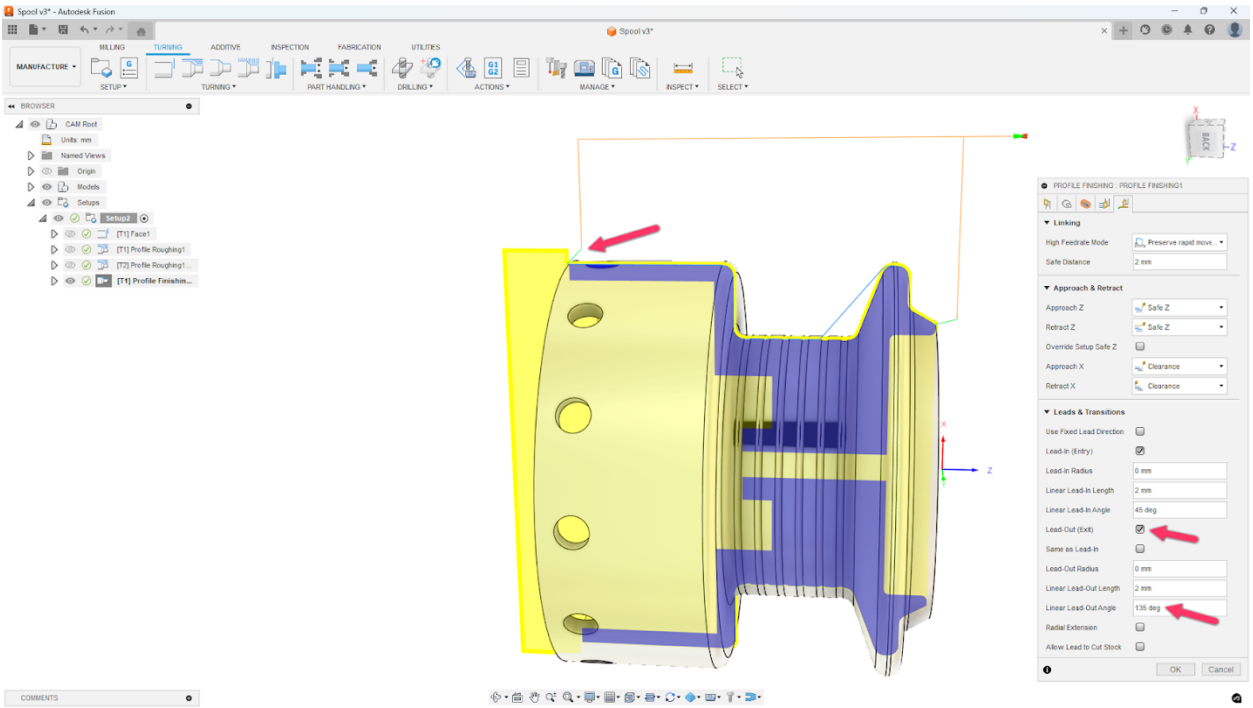

Exhibit 5

In this exhibit, we have created a Derived Finishing Toolpath from the original Roughing toolpath. It automatically turned off the Groove Suppression and Stock to Leave settings. Due to the warning in the Browser, we have clicked on the Linking tab and unchecked the Same As Lead-In option and changed the Linear Lead-Out Angle to 135 to make the tool not scrape the stock on exit.

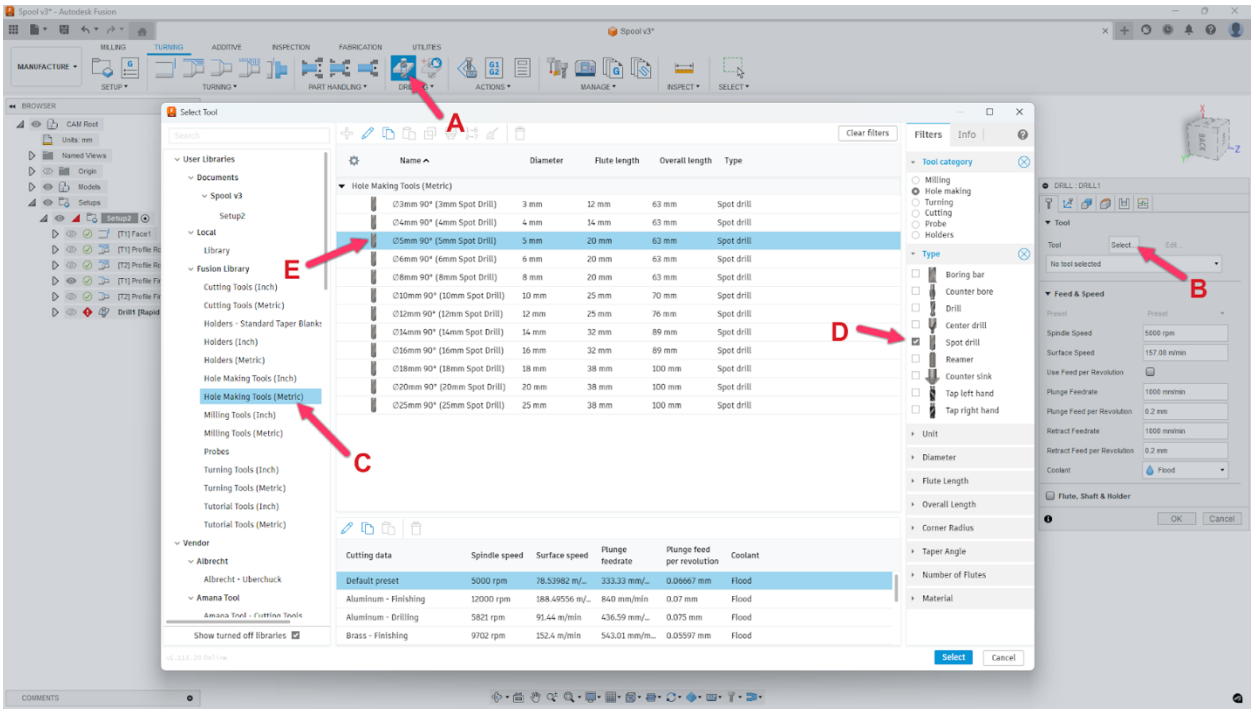

Exhibit 6

In this exhibit, we have clicked on the Drill command (A) in the main menu. In the Drill Dialog that appears, click on Select (B) next to Tool. The Select Tool dialog appears. Select Hole Making Tools (Metric) from the library section (C) and then filter the list by selecting Spot Drill (D) from the Type selection. Finally, select the 5mm Spot Drill (E) from the list.

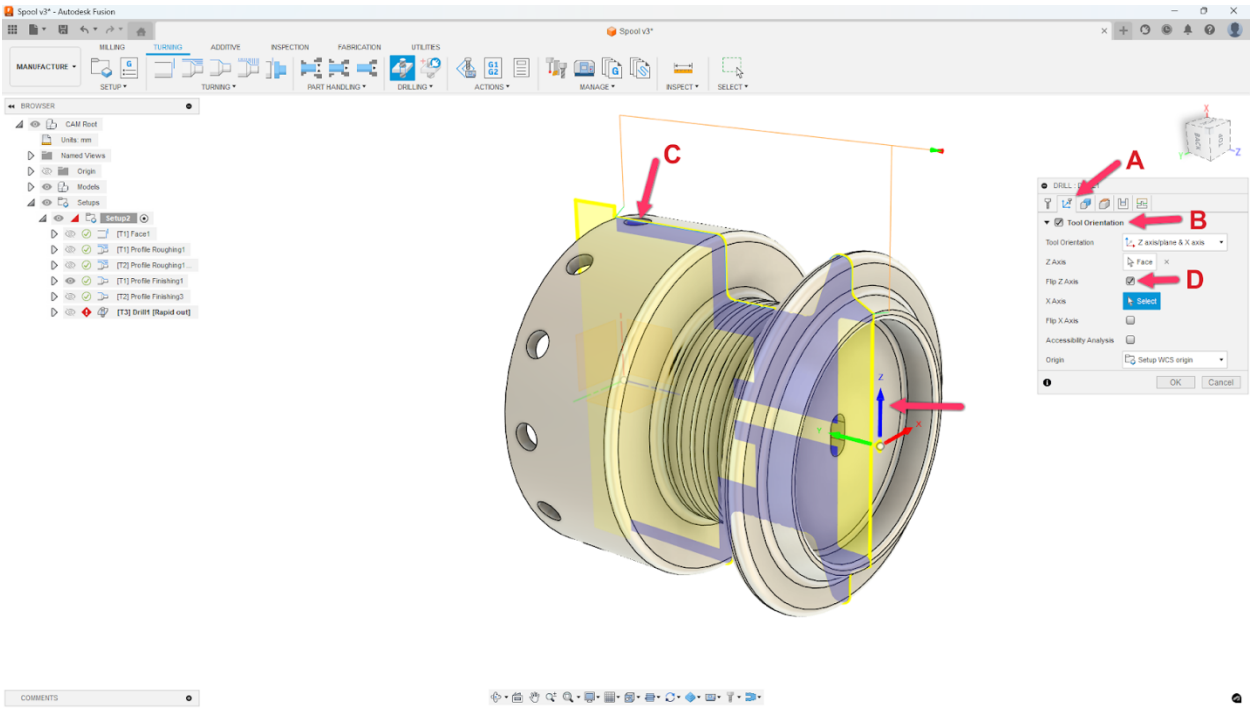

Exhibit 7

In this exhibit, we have clicked on the Multi-Axis tab (A) and selected Tool Orientation (B). Select the face of the 3mm hole (C) pointing in the X direction to specify the new Z direction. Click on Flip Z Axis (D) to flip the Z Axis to point upwards.

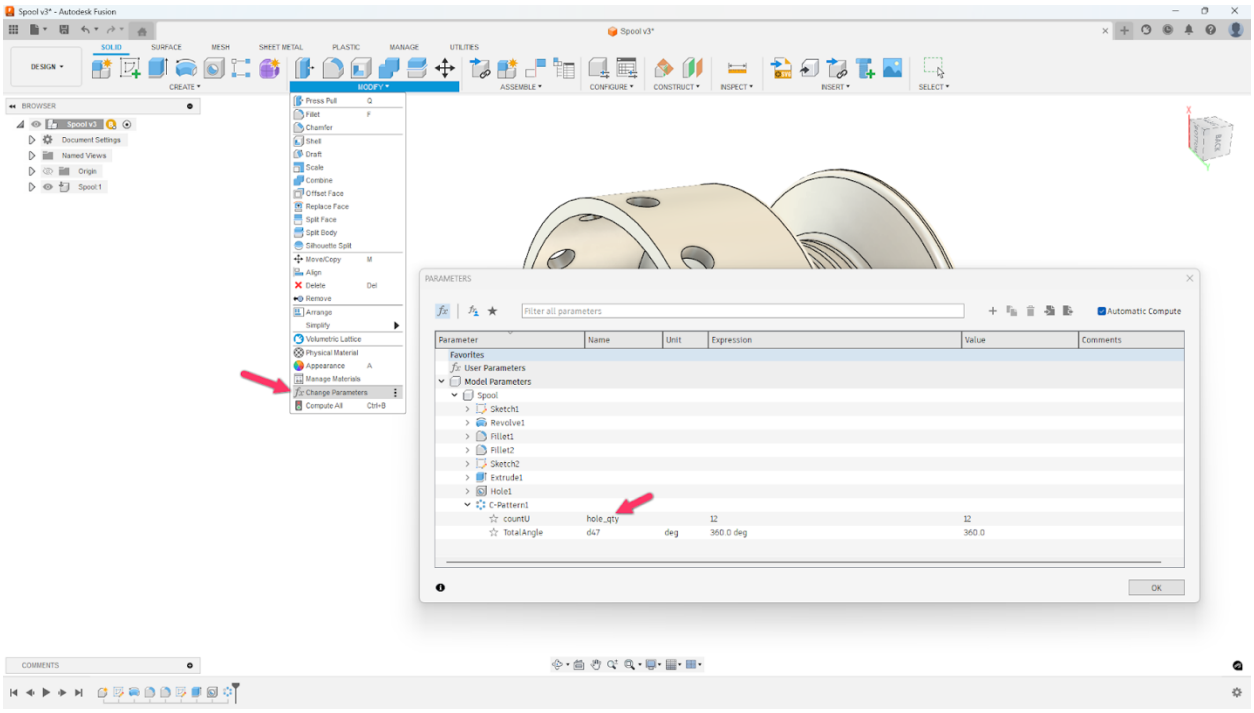

Exhibit 8

In this Exhibit, we have jumped back to the Design Workspace and selected Change Parameters from the Modify menu. Then, expand open Spool, then C-Pattern1. Change the name next to CountU to hole_qty and then press OK. Then, switch back to the Manufacture Workspace.

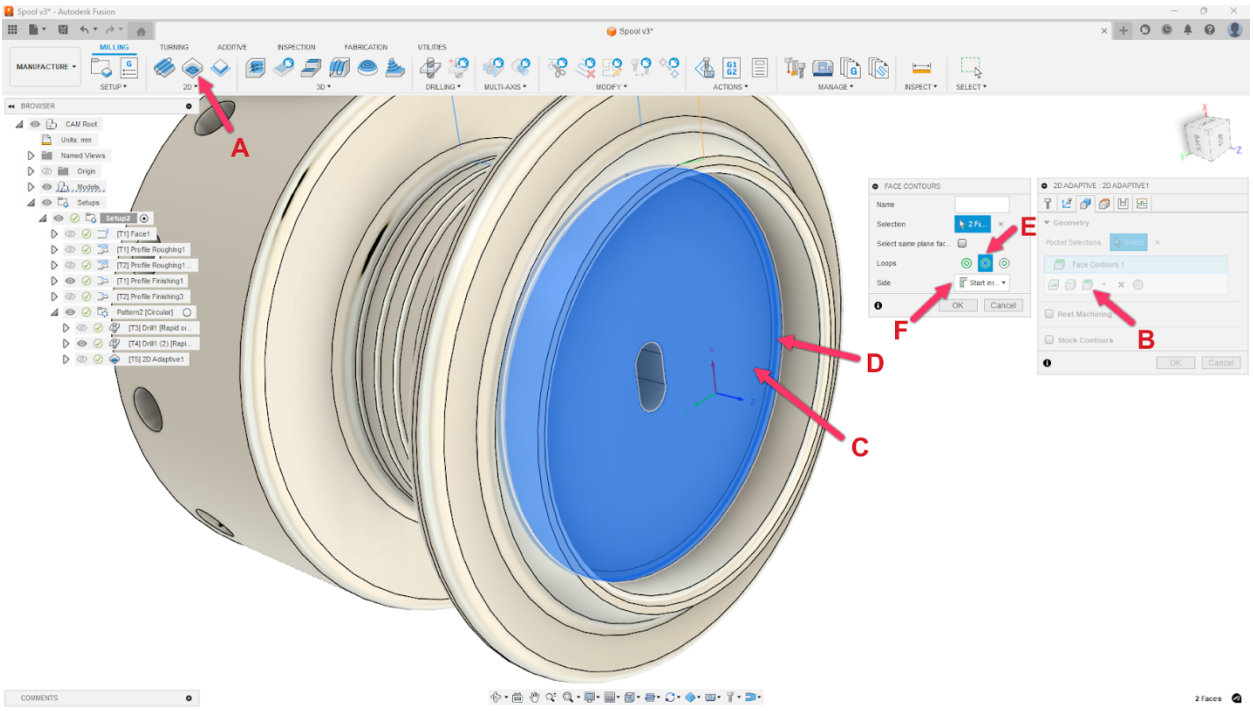

Exhibit 9

In this exhibit, we have created a 2D Adaptive toolpath (A). In the Geometry tab of the dialog, select Face Contours (B) and then select faces C & D. In the Face Contours dialog, select Outer Loops (E) and Start Inside (F). Finally, in the Passes Tab, turn off “Stock To Leave” (not shown in exhibit).

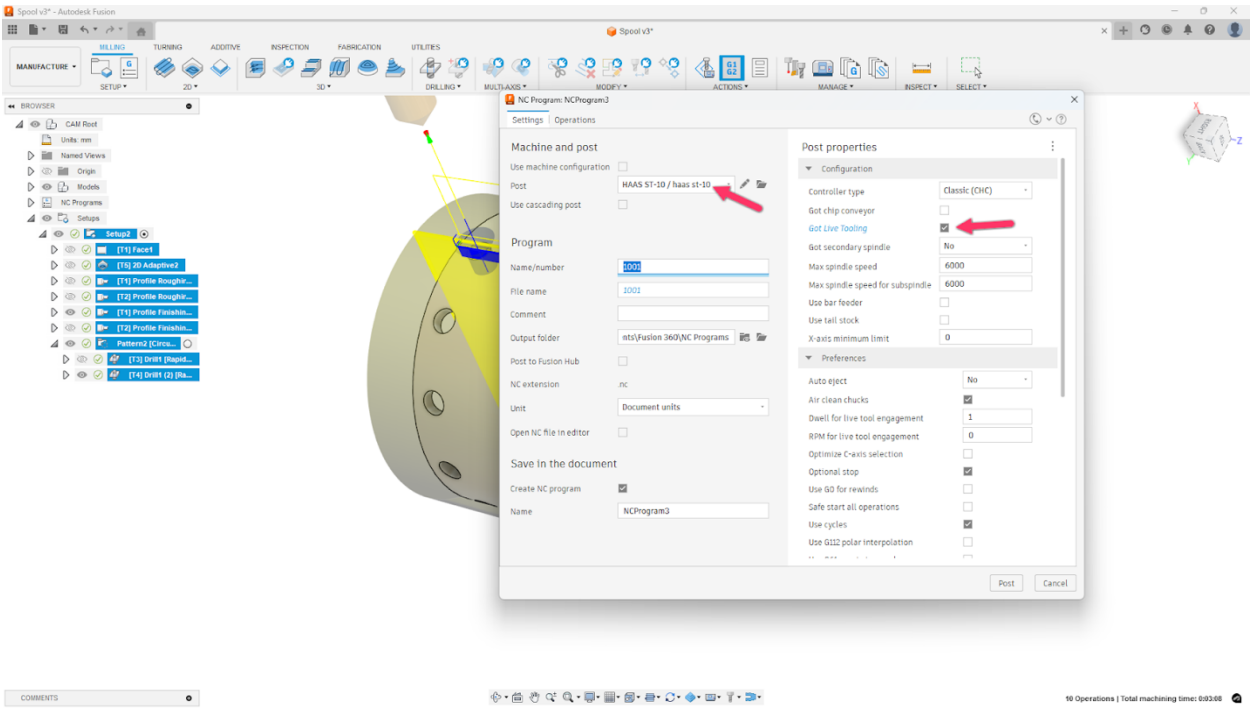

Exhibit 10

In this exhibit, we have selected the Haas ST-10 as our Post processor and made sure to turn on Got Live Tooling to signify that we are using the milling spindle. Then press Post to create the G-Code.