

TheCoolToolGmbH

# TUTORIAL

Milling with 4 axes in Fusion 360

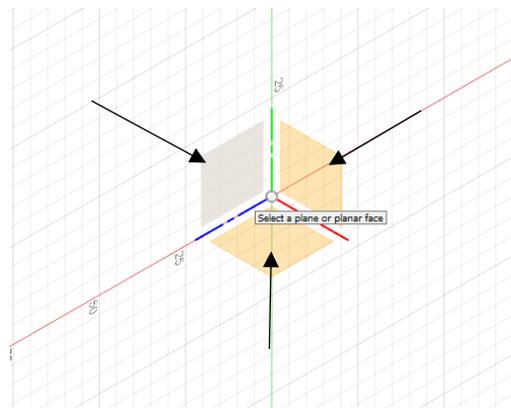


## 2.0 CREATE A BODY

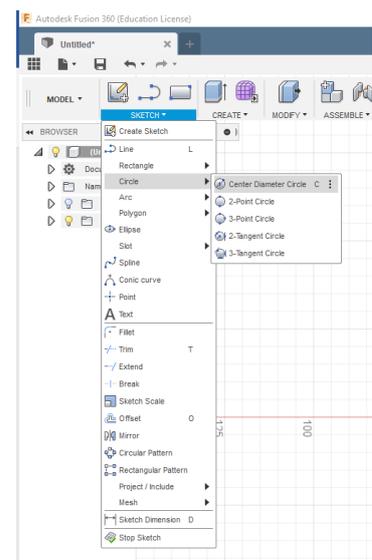
Open a new project and select Create Sketch.



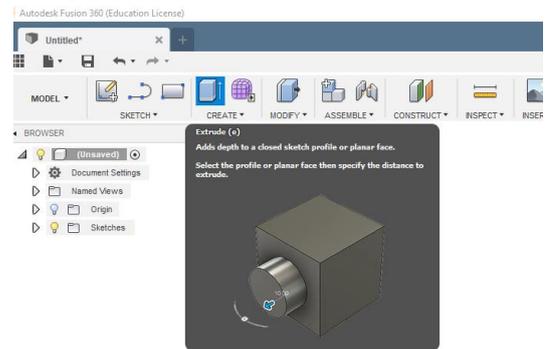
Select the layer on which you want to draw the body. If you are designing for four axes milling, the view from the left is recommended.



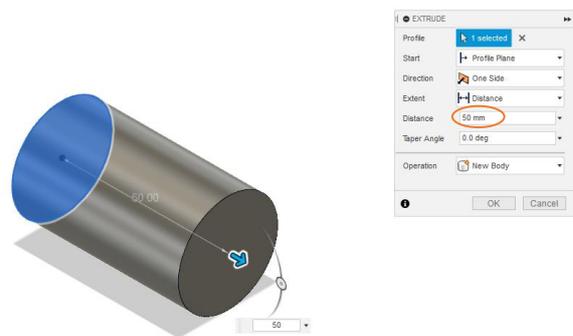
In our tutorial, we will create a simple cylinder. Click on Sketch -> Circle -> Center Diameter Circle. Define the center point and specify the diameter.



Click on Extrude and select the sketch you have created.

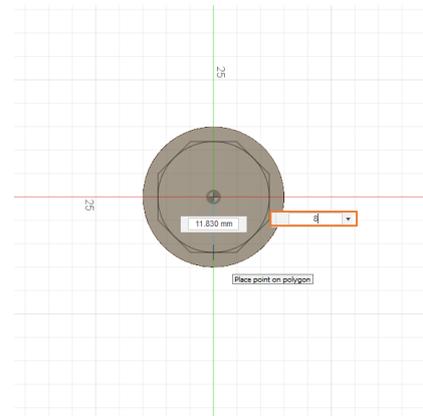


Enter the length of the cylinder in the box Distance (do not edit the other fields in the box). Confirm and your body will be generated.

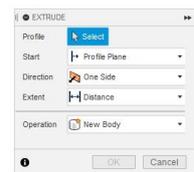
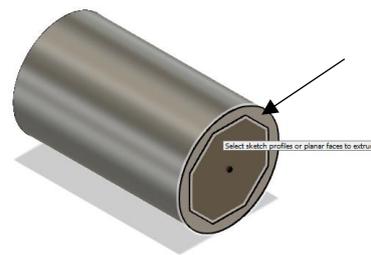


### 3.0 EDIT THE BODY

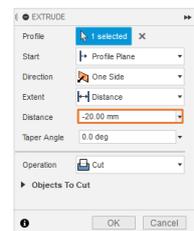
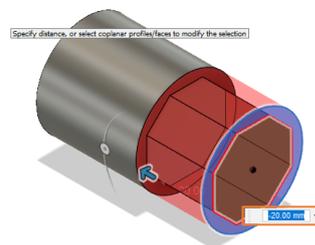
Autodesk Fusion 360 now allows you to edit the body. You can add or cut off a part. In our example we will cut out a Polygon. Click on Sketch -> Polygon -> Circumscribed Polygon. Click on the right side of the cylinder and drag the polygon to the right size, you can also set the number of edges.



Select Extrude and click on the area between the polygon and the outer circle.

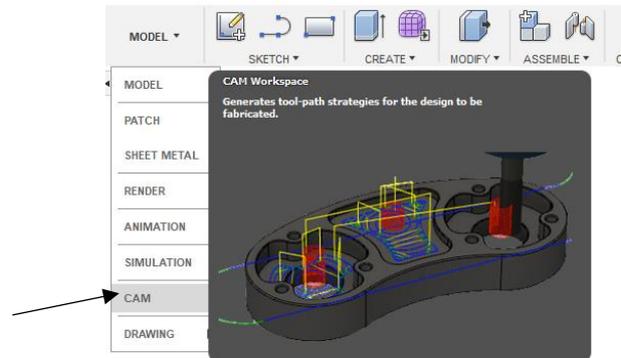


Enter the distance as a negative value to cut the part out. Confirm with enter and the area between polygon and circle will be removed.

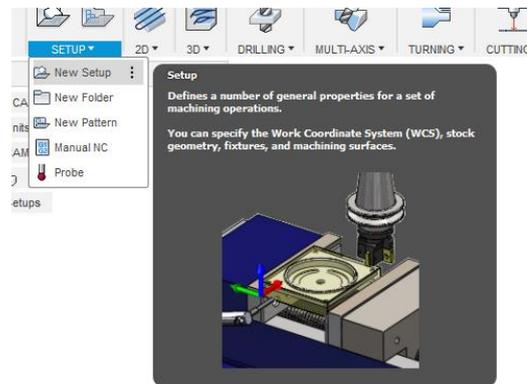


## 4.0 CREATE A SETUP

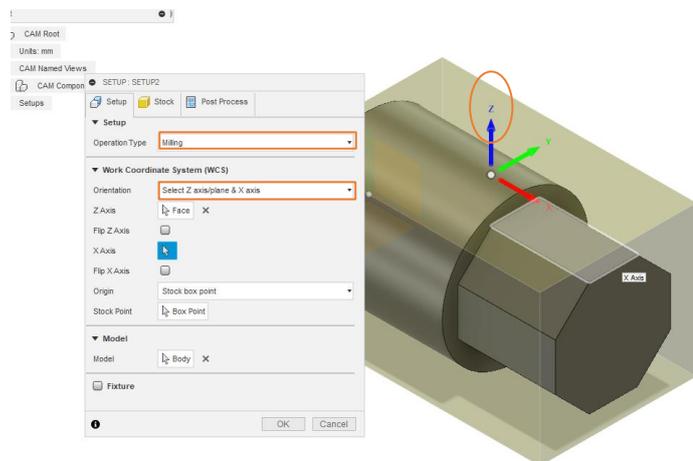
Switch to CAM mode.



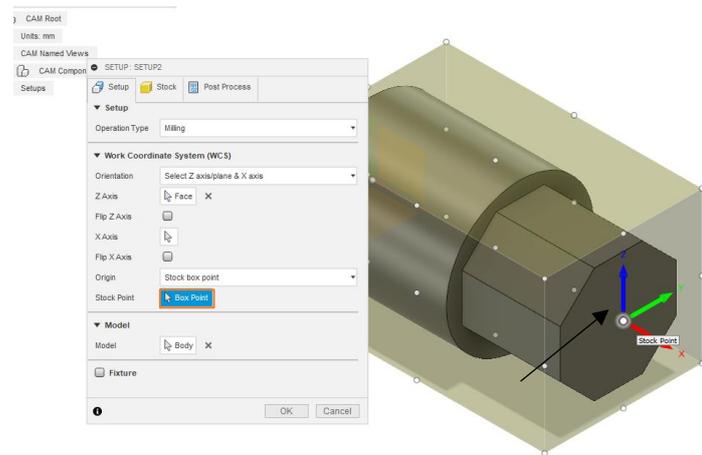
Press Setup -> New Setup



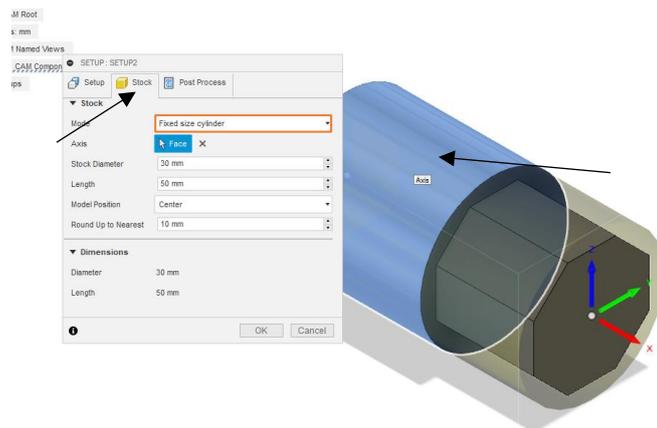
In this menu select the Operation Type: Milling. For Orientation set: Select Z-Axis/Plane & X-Axis. Click on the upper polygon surface. This aligns the Z axis. The blue arrow now points upward.



Select Box Point and click on the center point of the body.



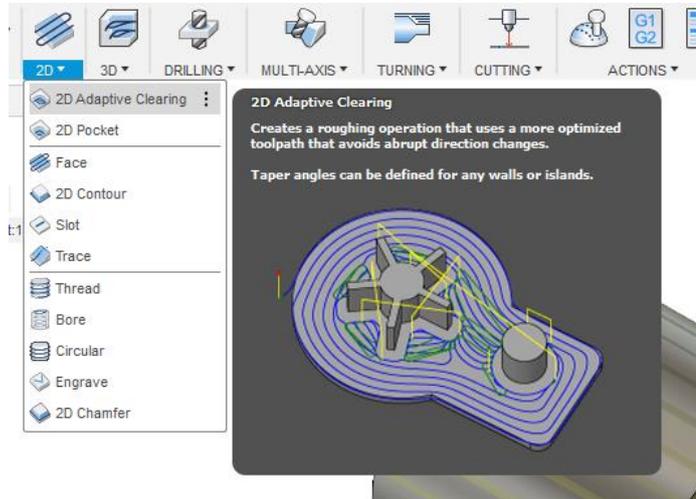
Switch to the tab Stock in the menu. Select the Mode: Fixed size cylinder. Click on the back part of your cylinder and enter the dimensions of the blank. In our example, the blank has the same size as the body we have created.



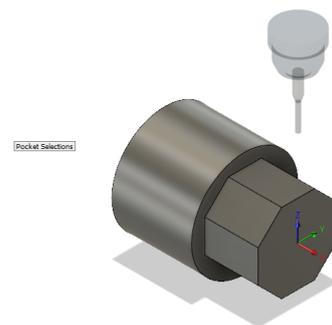
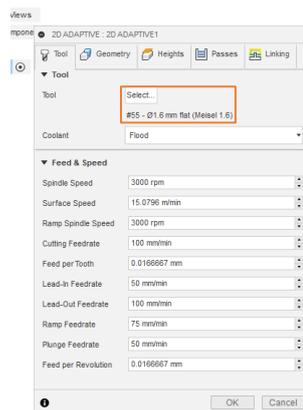
Confirm with Enter and the setup has been configured.

## 5.0 MILLING

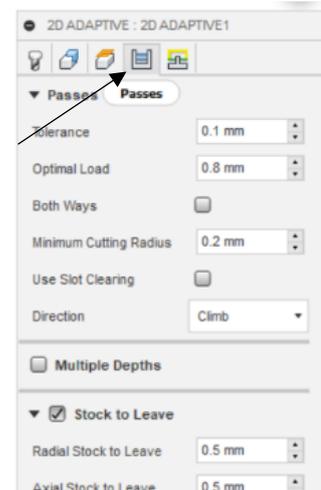
Once you have successfully configured your setup, you can start creating the routing path. To mill away the part we cut out, click 2D -> 2D Adaptive Clearing.



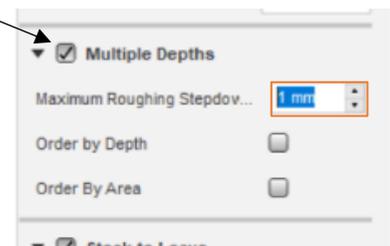
Choose a suitable milling bit from the previously included library. The information regarding feed and speed are rather conservatively parameters. You can change them at your own risk.



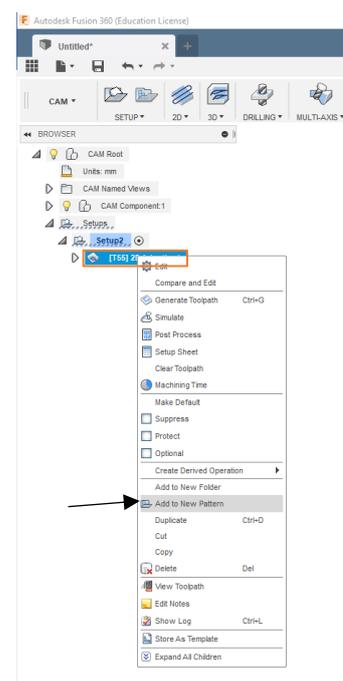
Change to the tab Passes.



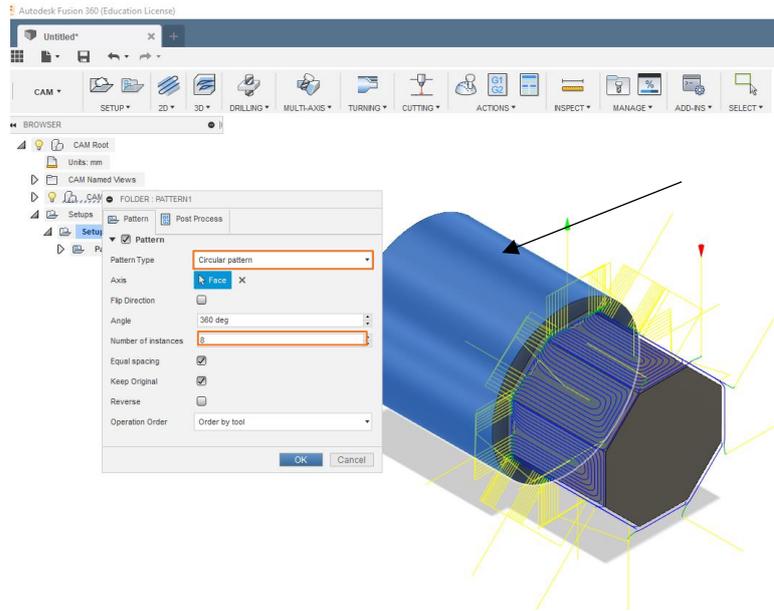
Activate Multiple Depths and enter the plunging depth at Maximum Roughing Stepdown. This parameter depends on the material and the tool. The default value for milling is 1mm per roughing pass.



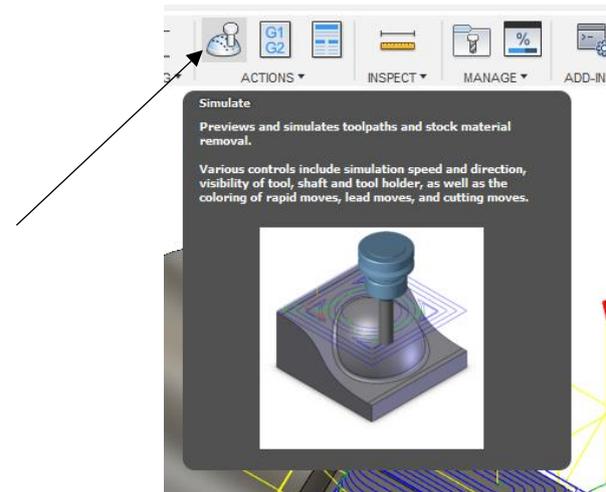
Click on the upper side of your polygon and confirm with ENTER. As you can see, the milling machine would only process one side of the polygon. To change this, right-click on 2DAdaptive1 and click on Add to new pattern.



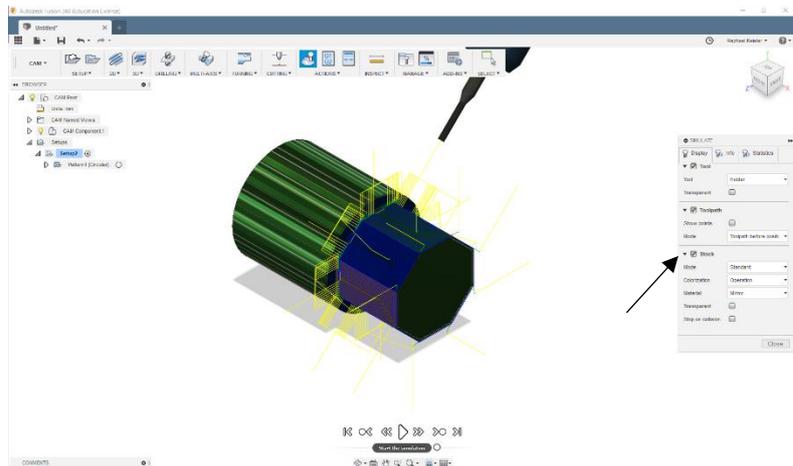
In this menu select Pattern Type: Circular pattern. Click on the rear cylinder. Before you confirm the box with ENTER, set the number of corners of your polygon under Number of Instances.



Select Simulate.

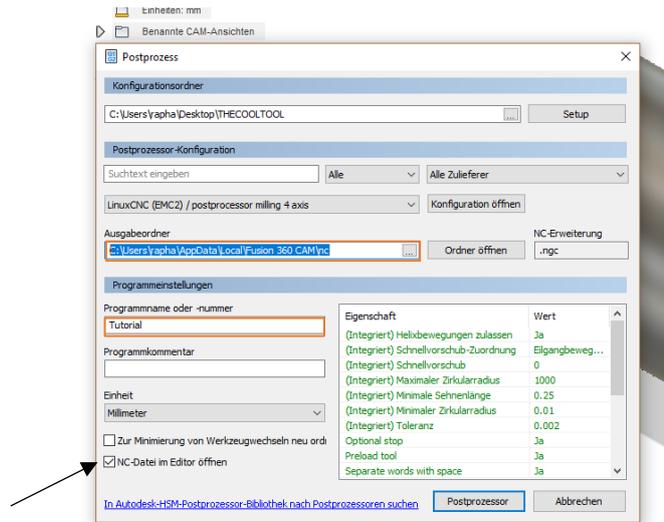


Check Stock and press the play button. The simulation does not rotate the work piece but only the tool.





Select a storage location and a program name (xyz.NGC). Select the correct units (mm/inch) and Uncheck Open NC file in editor. To complete, click Post.



Load the NGC-file in Machinekit to see a preview of the paths generated and run a simulation.

