

TheCoolToolGmbH

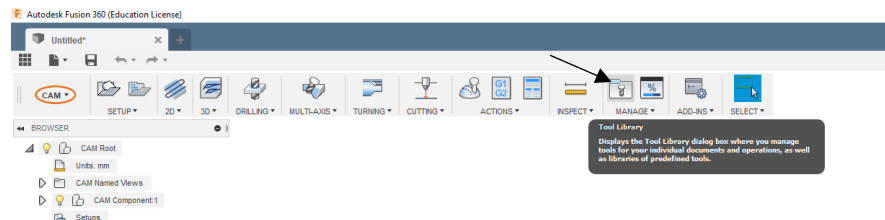
TUTORIAL

Turning with Fusion 360

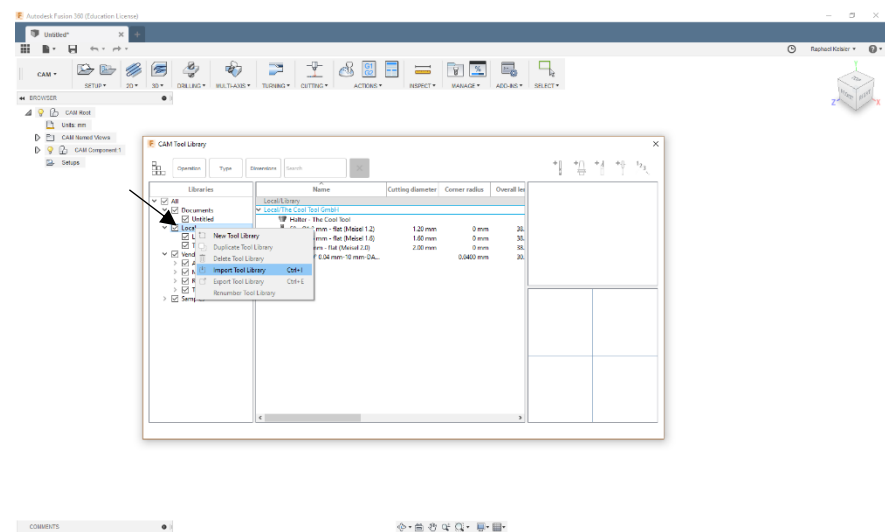
1.0 DOWNLOADS

Download the tool library and the post-processors.

In Fusion 360 switch to CAM mode and click on Tool library.

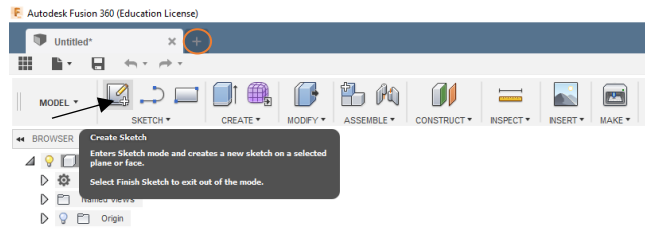


Right-click on Local and press Import Tool Library. Select the library from your Computer. If the library is imported go back to the model mode.

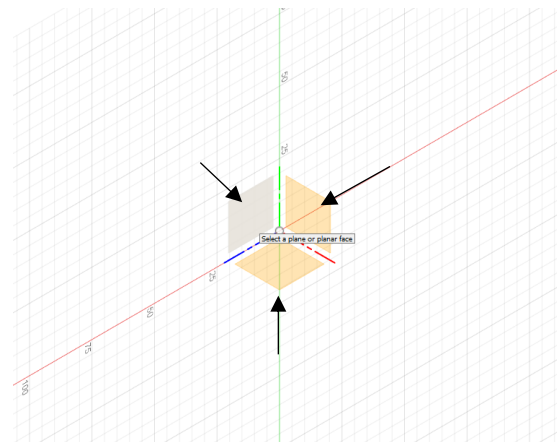


2.0 CREATE BODY

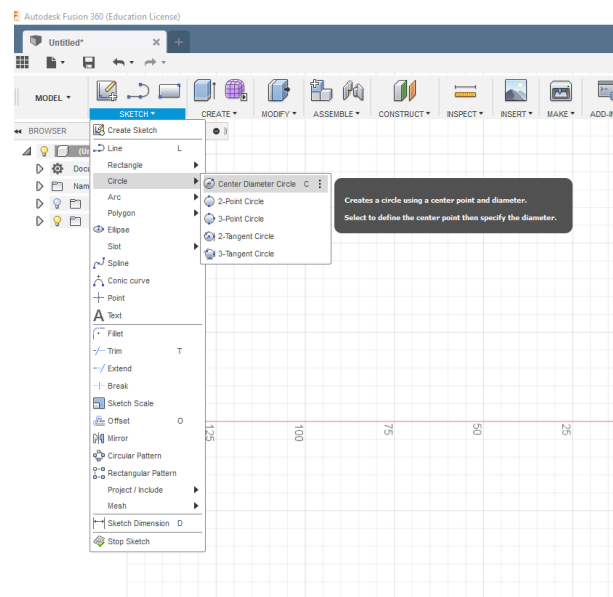
Open a new project and select Create Sketch.



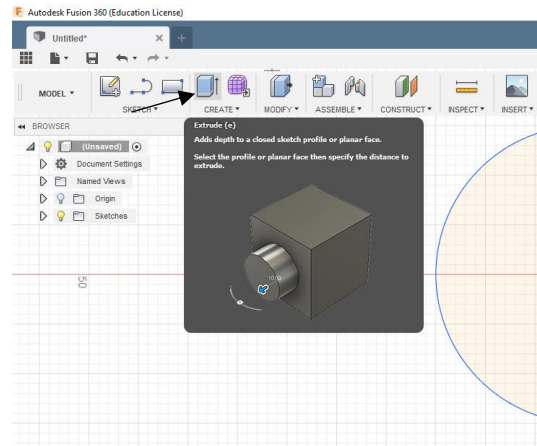
Select the layer on which you want to draw the body. If want to turn, the view from the left is recommended.



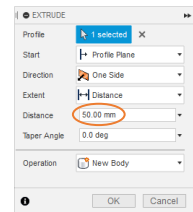
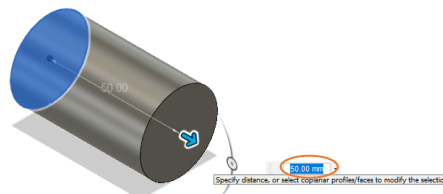
In our tutorial, we will create a simple cylinder. Click on Sketch -> Circle -> Center Diameter Circle. Define the center point and specify the diameter.



Click on Extrude and select the sketch you have created.

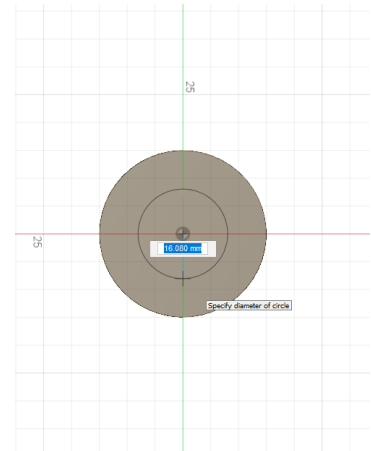


Enter the length of the cylinder in the box Distance (do not edit the other fields in the box). Confirm and your body will be generated.

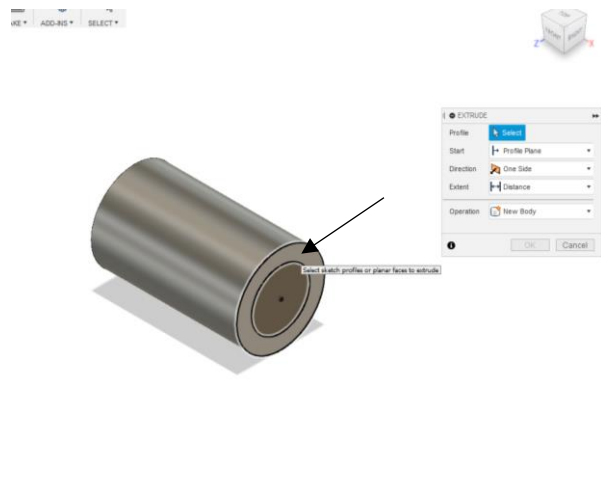


3.0 EDIT THE BODY

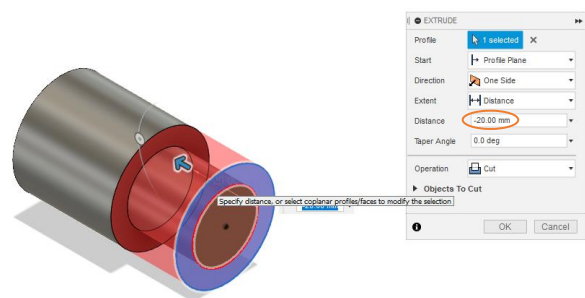
Autodesk Fusion 360 now allows you to edit the body. You can add or extrude a part. In our example we will extrude a hollow cylinder. Click on Sketch -> Circle -> Center Diameter Circle. Choose a side of the cylinder and enter the desired diameter of the hollow cylinder.



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

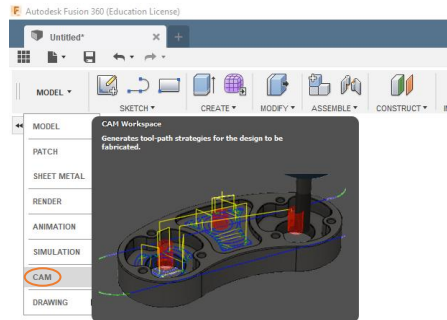


Enter the length of the cylinder as a negative value to cut it out. Confirm with ENTER and the hollow cylinder will be cut out.

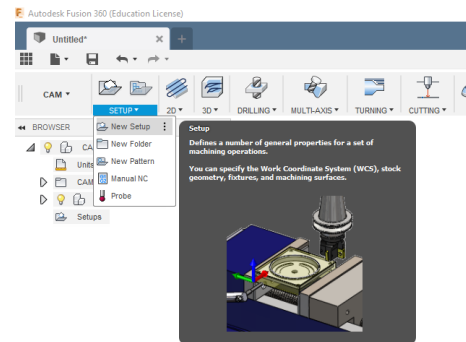


4.0 CREATE SETUP

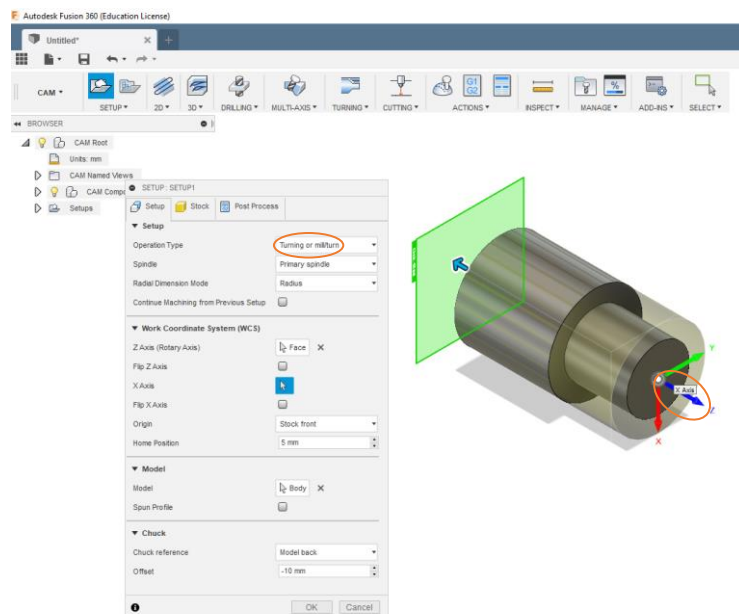
Switch to CAM mode.



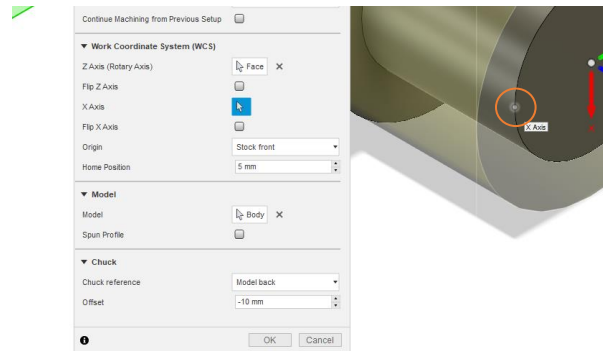
Press Setup -> New Setup.



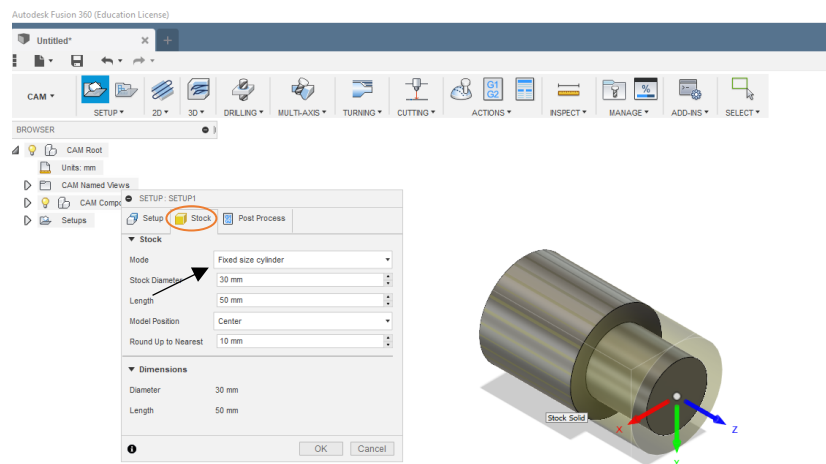
In this menu select the Operation Type: Turning or mill/turn. Click on the top surface of the body. The blue arrow now points forward.



Click on the point that appears left of the axis when you move over it. The red arrow now points to the left or to the right side.



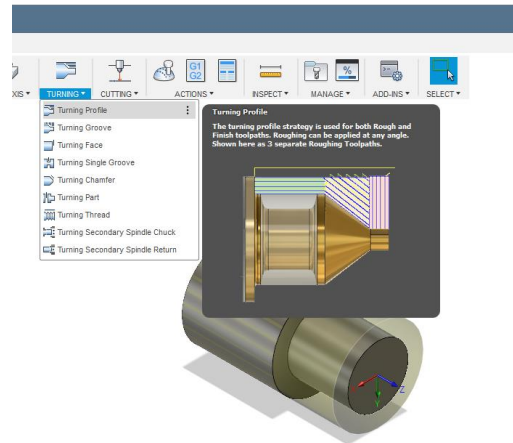
Switch to the tab Stock in the menu. Select the Mode: Fixed size cylinder. Enter the dimensions of the blank. In our example, the blank has the same size as the body we have designed.



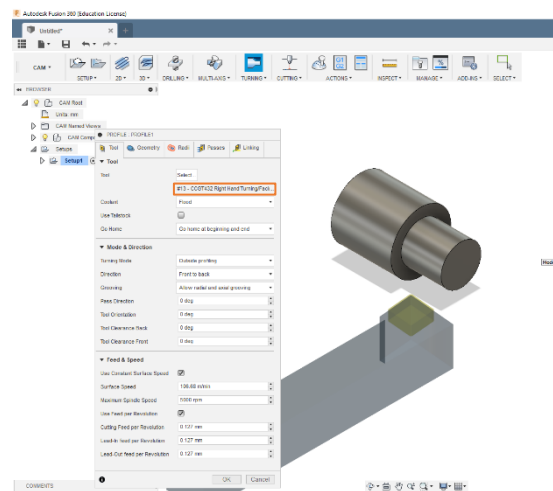
Confirm with Enter and the setup has been configured.

5.0 TURNING

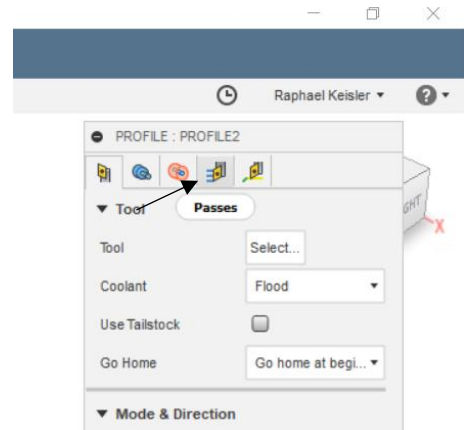
Once you have successfully configured your setup, you can start turning. To turn the hollow cylinder we have cut out, click on Turning -> Turning Profile.



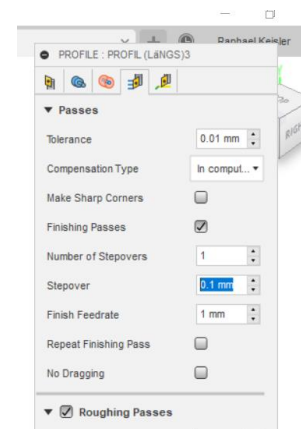
In the appearing menu select a suitable turning tool from the previously included library. The information regarding feed and speed are rather conservatively parameters. You can change them at your own risk.



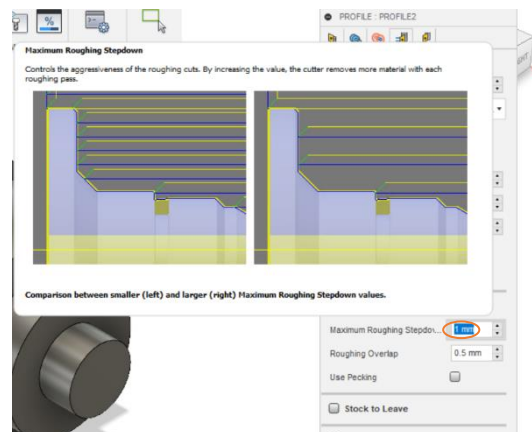
Change to the tab Passes.



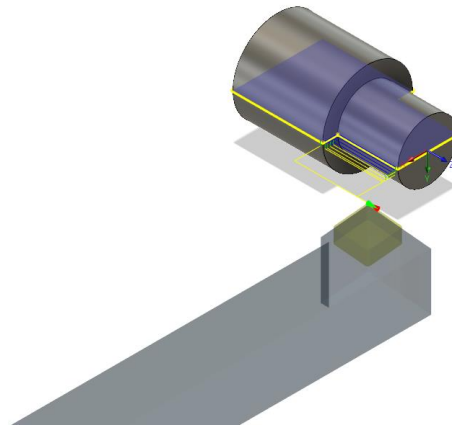
If "Finishing Passes" is activated, you can enter the delivery of the last pass in "Stepover". As default 1 mm is entered. We recommend 0,1 mm.



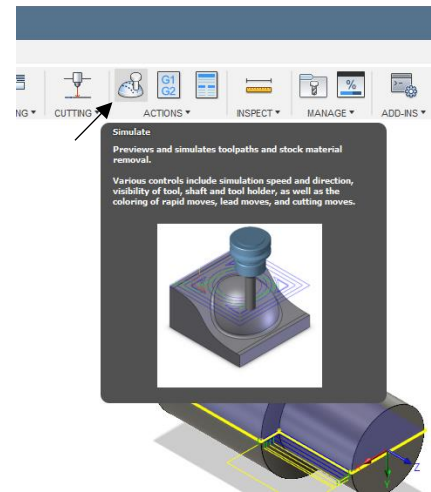
Activate Multiple Depths and enter the plunging depth at Maximum Roughing Stepdown. This parameter depends on the material and the tool. The default value for turning is 0.1 mm per roughing pass.



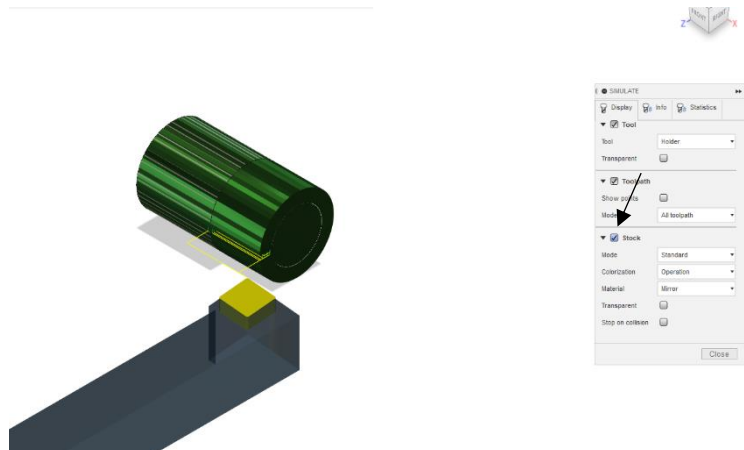
Confirm with Enter.



Select Simulate.

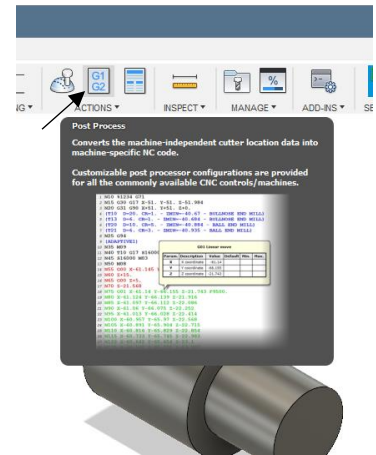


Check Stock and press the play button.

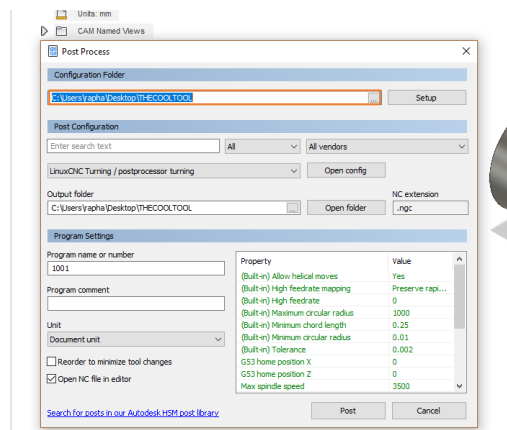


6.0 G-CODE AND POST PROCESSING

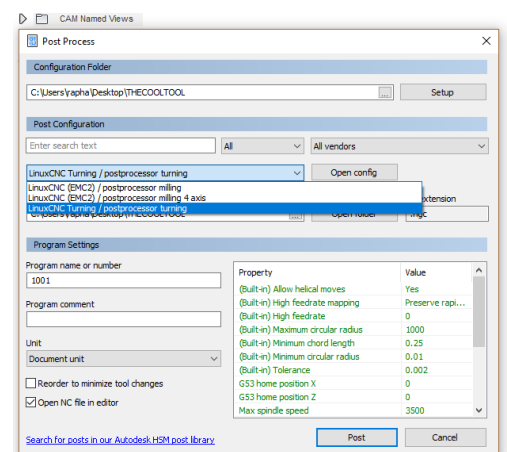
To generate the NGS file, click on Post Process.



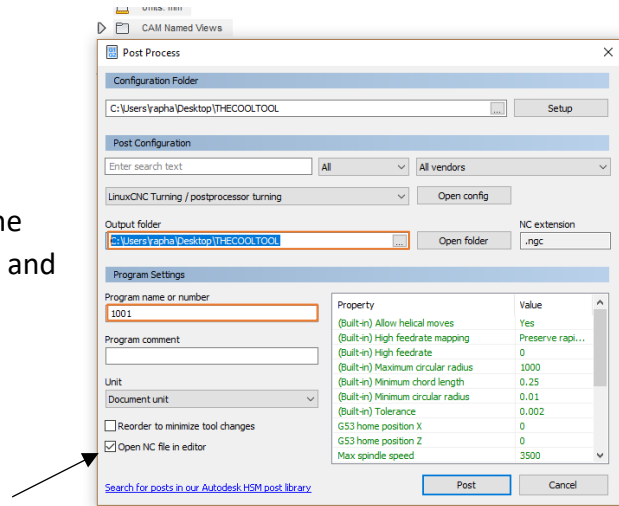
In the Post Process menu, select the location where the post processors have been stored to.



Select the correct post process from the dropdown list.



Select a storage location and a program name (xyz.NGC). Select the correct unit (mm/inch) and uncheck Open NC file in editor. Click Post.



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

