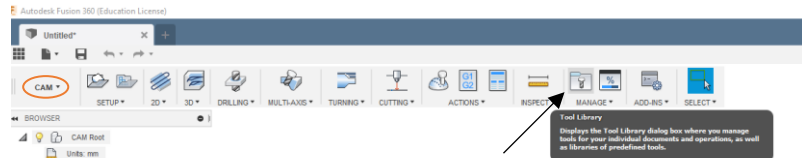TheCoolToolGmbH

TUTORIAL

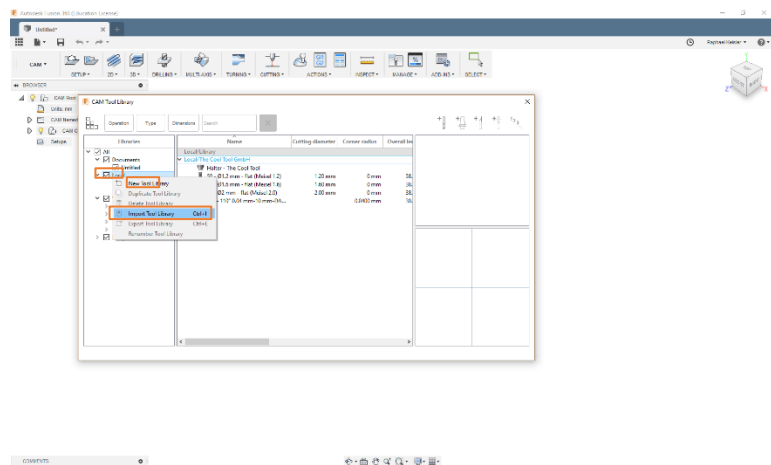Milling with 3 axes in Fusion 360

1.0 DOWNLOADS

Download the tool library and the post-processors.

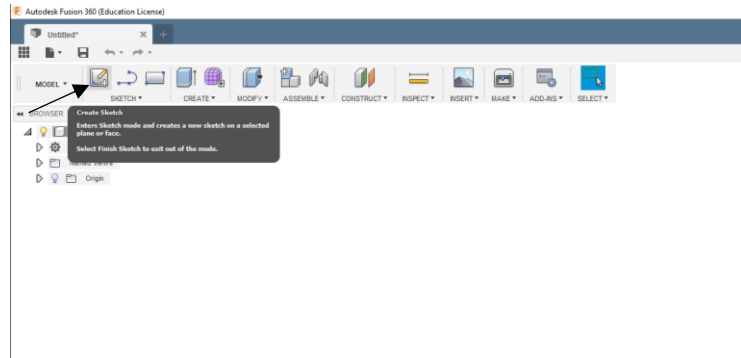In Fusion 360 switch to CAM mode and click on Tool library.

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.
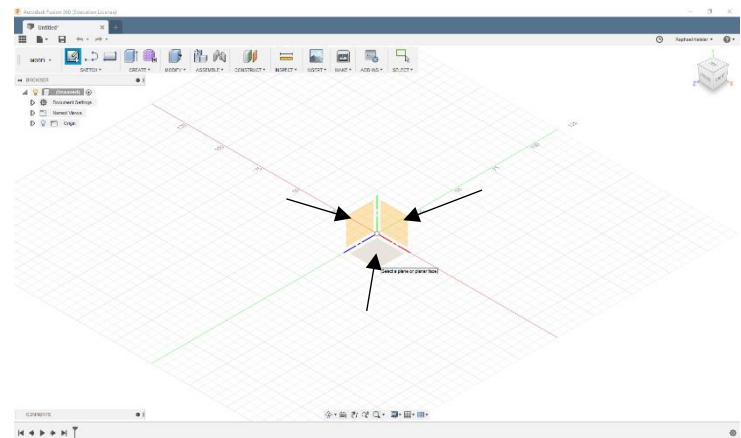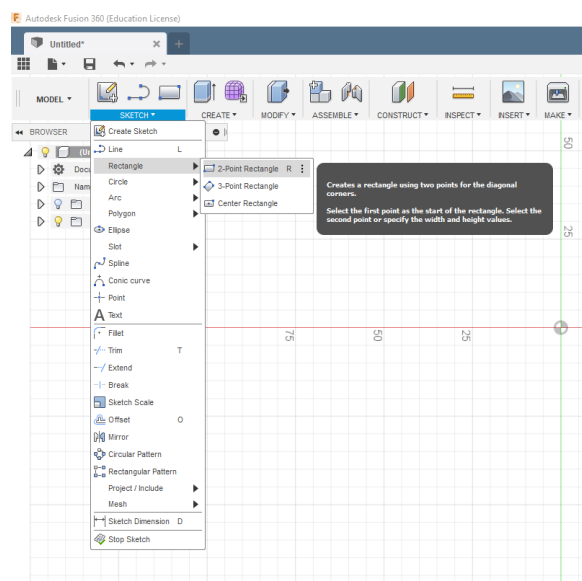
2.0 CREATE A BODY

Open a new project and select Create Sketch.

Select the layer on which you want to draw the body. If you are designing for three axes milling, the view from above is recommended.
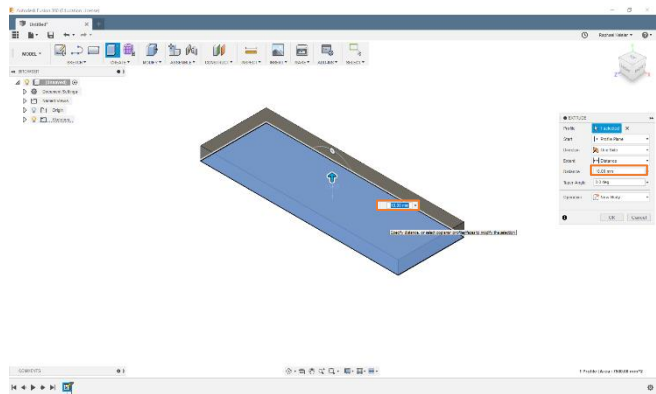
In our tutorial, we will create a simple cuboid. Click on Sketch -> Rectangle -> 2-Point Rectangle. Click on the zero point, drag the rectangle to the desired dimensions and confirm it.

Click on Extrude and select the sketch you have created.
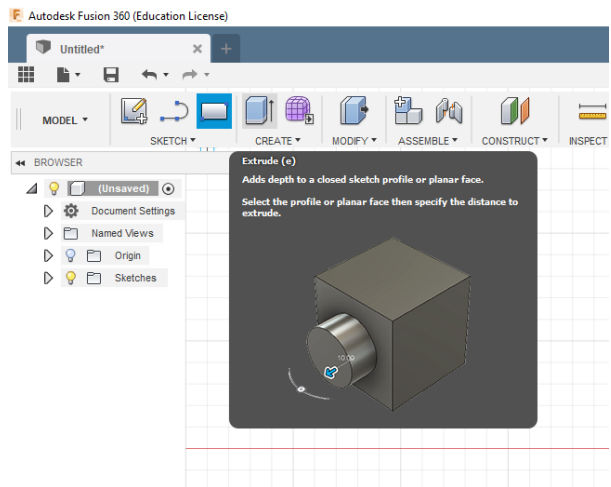
Enter the height of the cuboid in the box Distance (do not edit the other fields in the box). Confirm and your rectangle will be created.
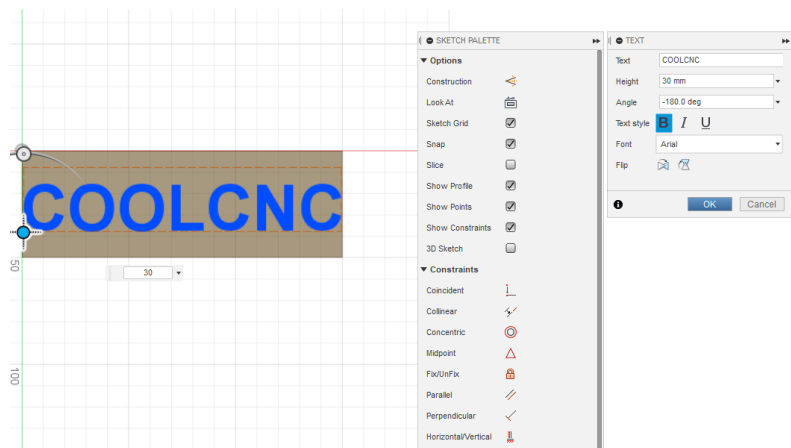
3.0 EDIT THE BODY

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 text. Click on Sketch -> Text and select the area you want to edit.
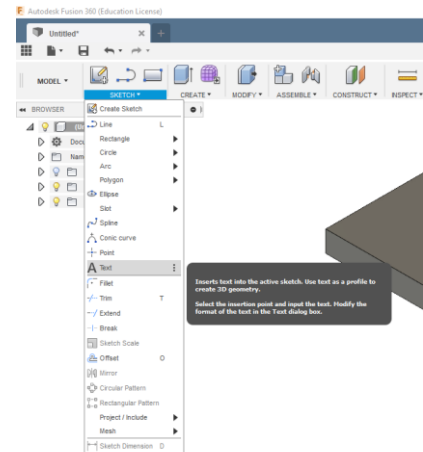
Click anywhere on the surface and enter the text and the height. Of course, you can also change the text style or the font. Move the text to the correct position and confirm.
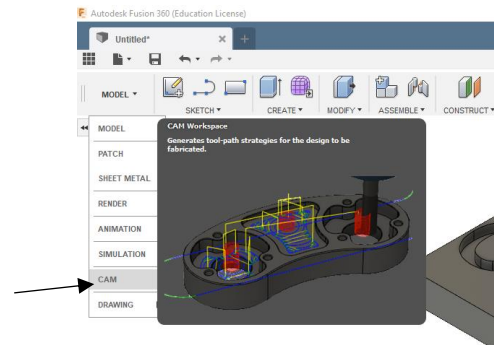

Select Extrude and click on the text. Enter the depth of the text as a negative value to cut it out. Confirm with enter and the text will be cut out.
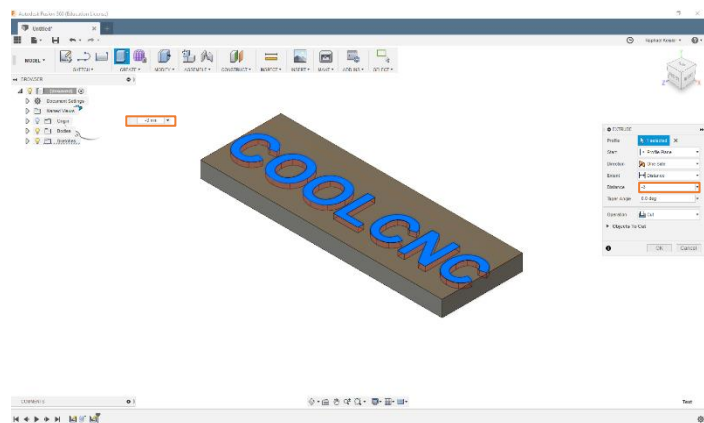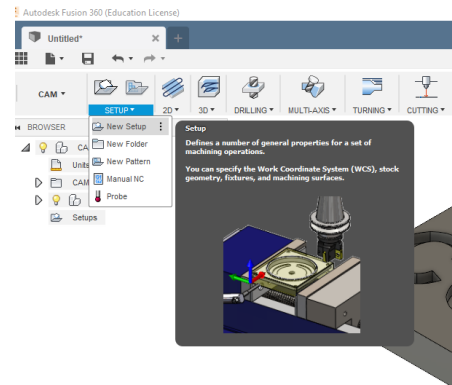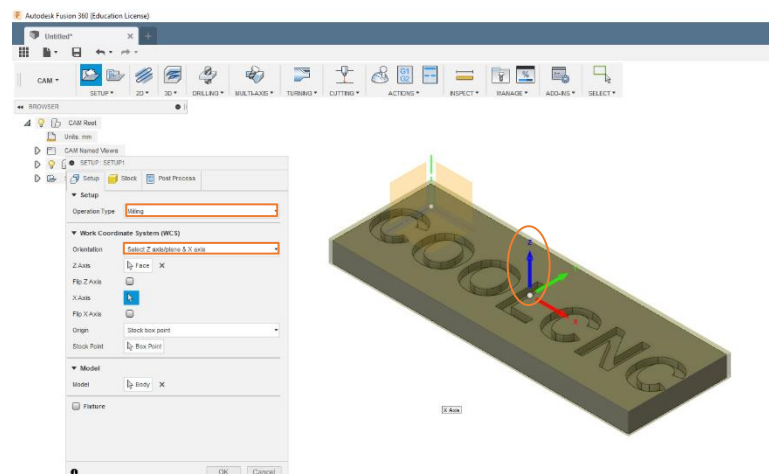
4.0 CREATE A SETUP

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 surface of the body. This aligns the Z axis. The blue arrow now points upward.

Click on the right side of your body to align the X axis correctly. Select Box Point and click at the center point of your body.
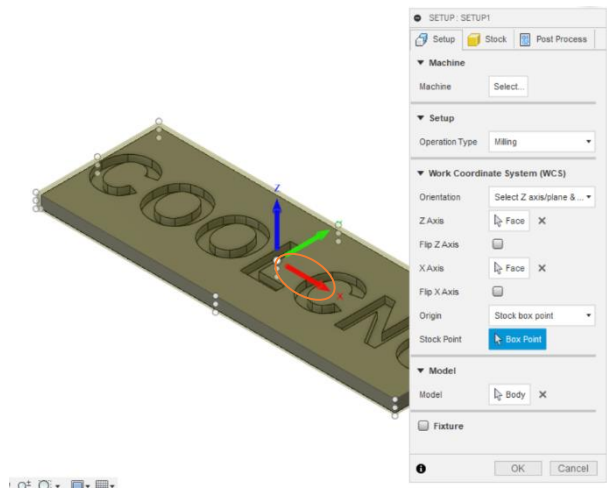
Switch to the tab Stock in the menu. Here you can enter the dimensions of the blank. In our example, the blank has the same size as the body we have created, so we enter zero in Stock Side Offset, Stock Top Offset and Stock Bottom Offset.
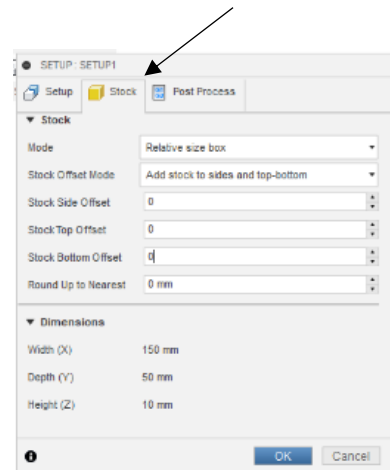
Confirm with Enter and the setup has been configured.

5.0 MILLING

Once you have successfully configured your setup, you can start creating the routing path. To mill away the text we cut out, click 2D -> 2D Adaptive Clearing.
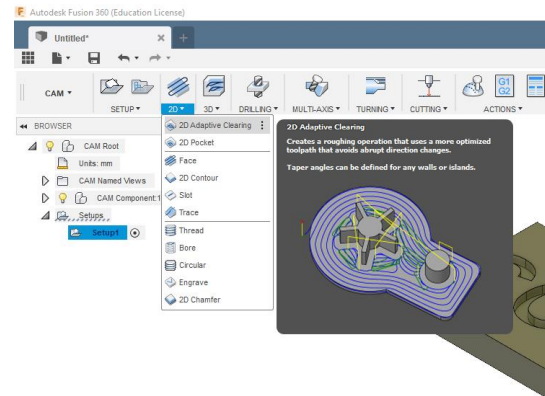
Choose a suitable milling bit from the previously included library. The information regarding feed and speed are rather conservatively parameters. You can change them at your own risk.
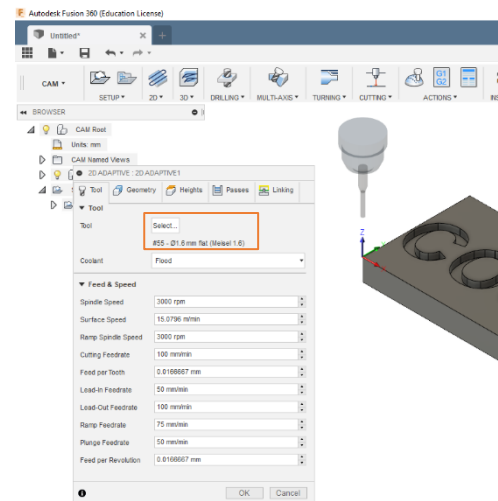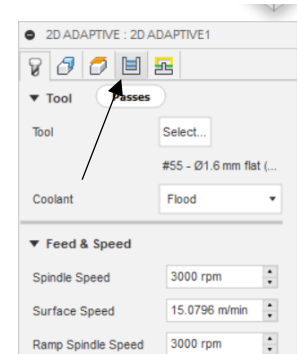

Change to the tab Geometry. Choose all surfaces on your body which should be milled (Make sure that you select the surfaces and not the edges)
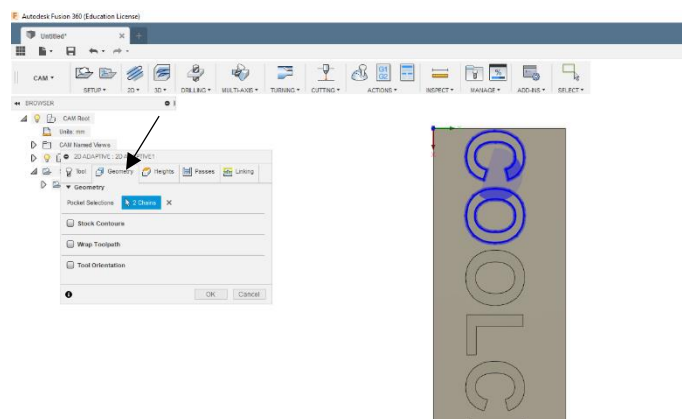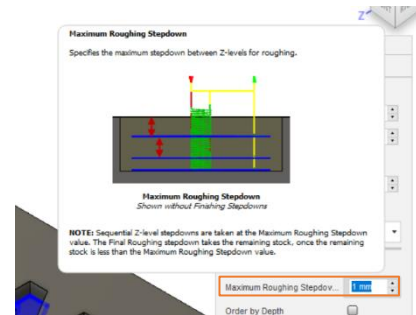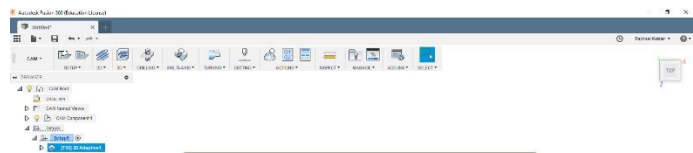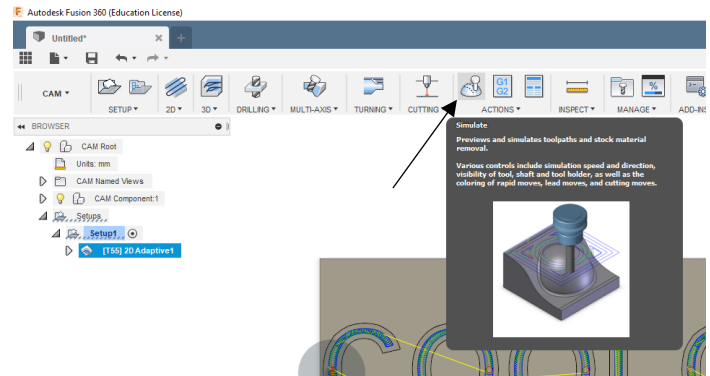
Change to the tab Passes.

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.
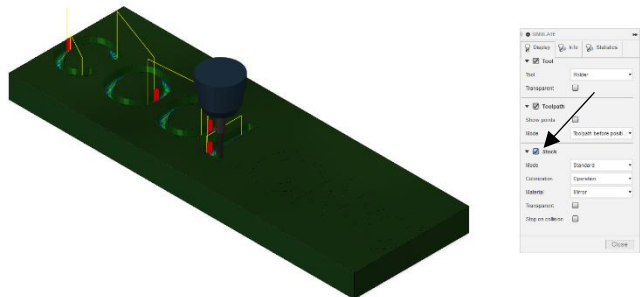

After confirming Adaptive Clearing
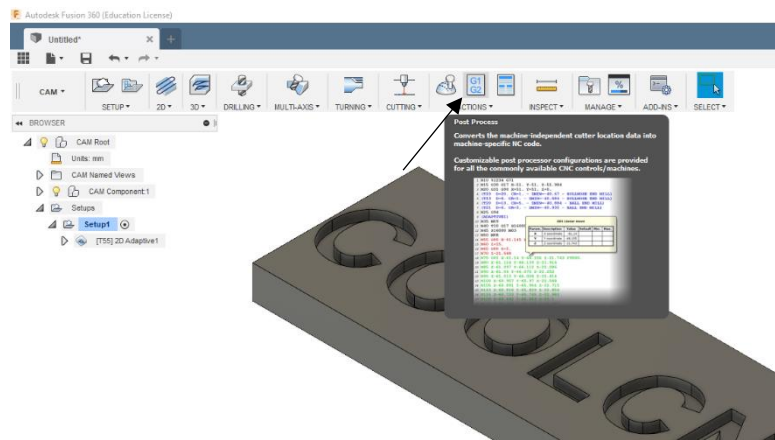
Select Simulate.

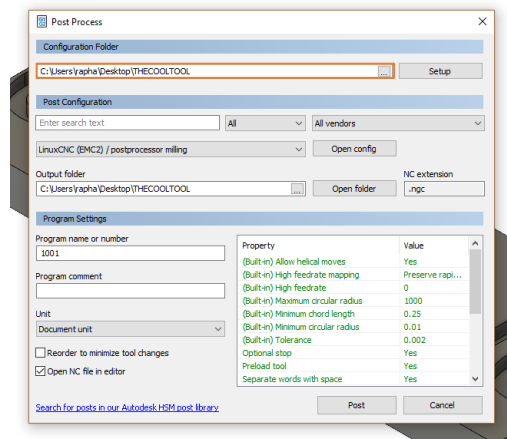Check Stock and press the play button.
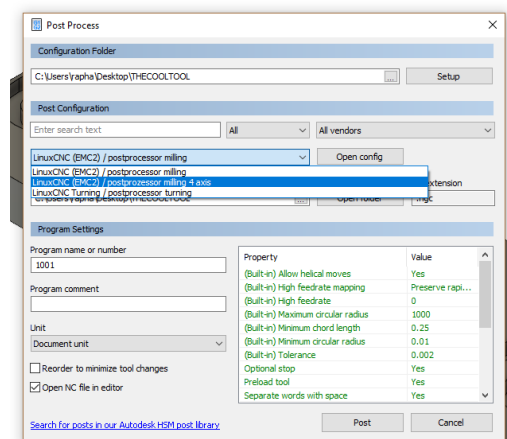
6.0 G-CODE UND POST PROCESSING

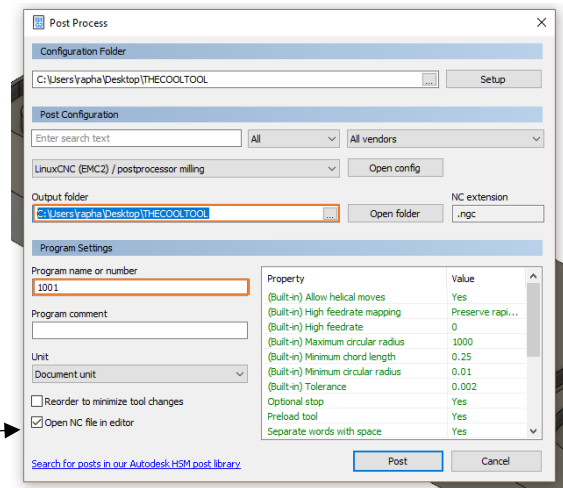To generate the NGS file, click on Post Process.

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 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.