YETI
TOOL

AUTODESK®
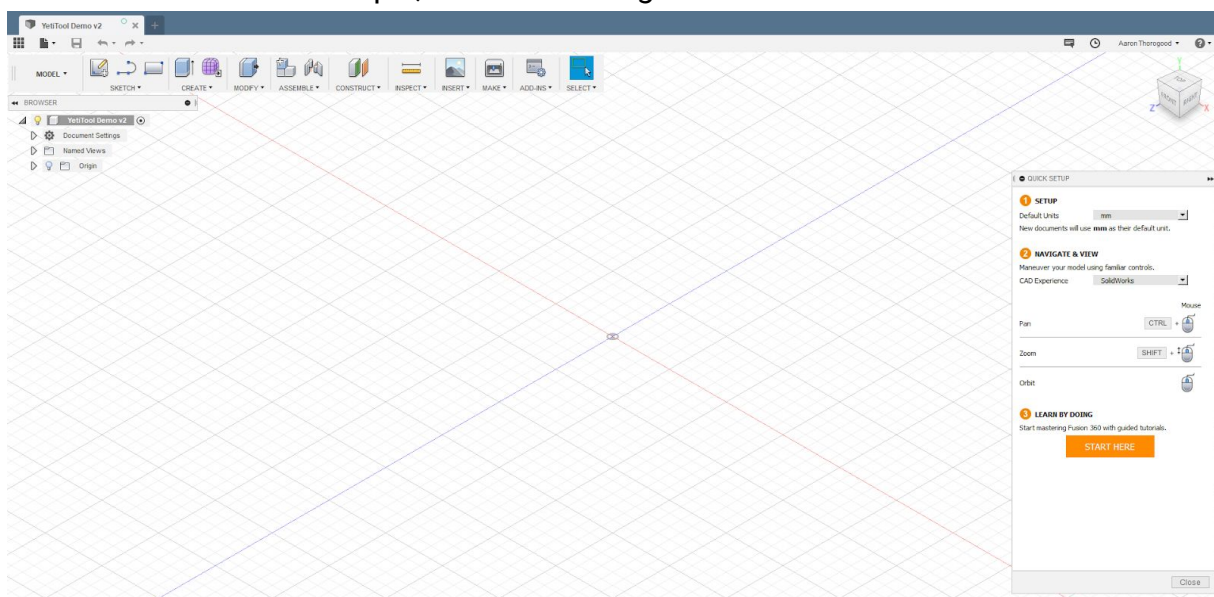# FUSION 360

Fusion 360

Table of Contents

1   Introduction

This guide will show you some of the key features in Fusion 360 in order to create a file for SmartBench.

2   Navigation

2.1 Setup

Using the help button in the top right corner, select Quick setup. Select the units you wish to use. For this example, we will be using mm.
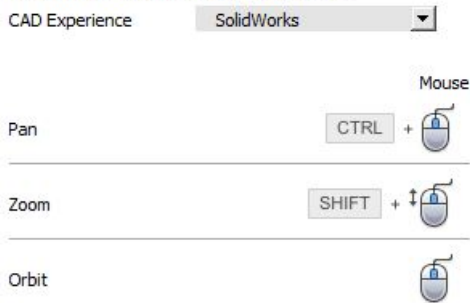
2.2 Mouse Controls

By select the CAD experience drop down, we can adjust the mouse controls. We have selected to use the mouse controls that relate to SolidWorks.
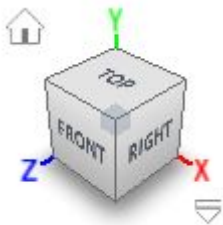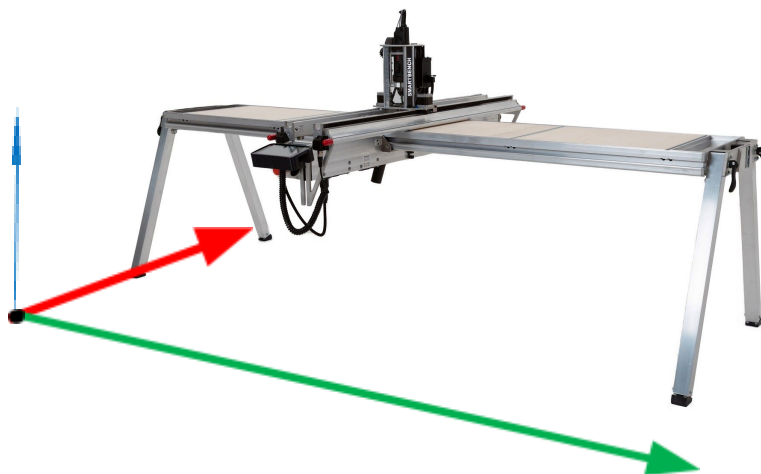

2.3 XYZ Axis

Fusion uses the standard XYZ axis setup. The navigation cube below can be used to quickly switch to your desired view.

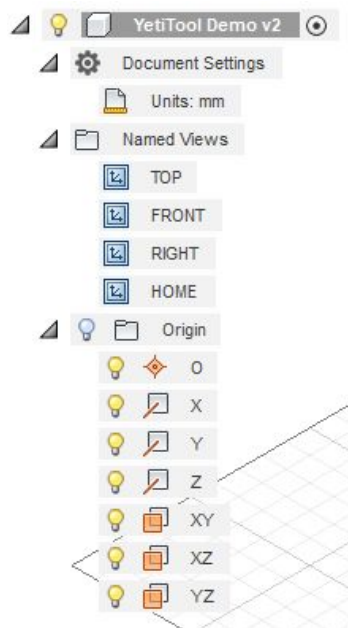We use the same XYZ axis on SmartBench detailed in the figure below

2.4 Keyboard Shortcuts

To find out about the keyboard shortcuts, please visit the link below

https://www.autodesk.com/shortcuts/fusion-360

2.5 Model Tree

The model tree enables you to view different features of your model. As sketches and bodies are created, they can be seen here.
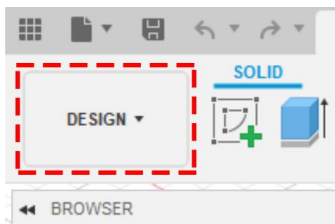

2.6 Timeline

The timeline will show you a workflow progression. We can use this to roll back to any feature during any time.

3 Model

3.1 Create a Sketch

We are first going to create a sketch. To do this, we will need to ensure we are in the DESIGN workspace.
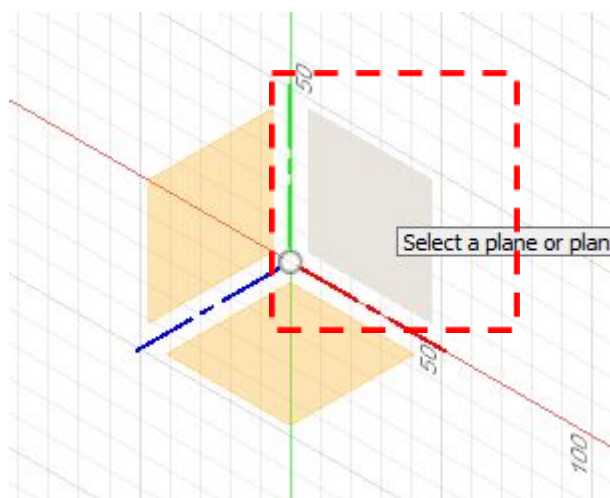
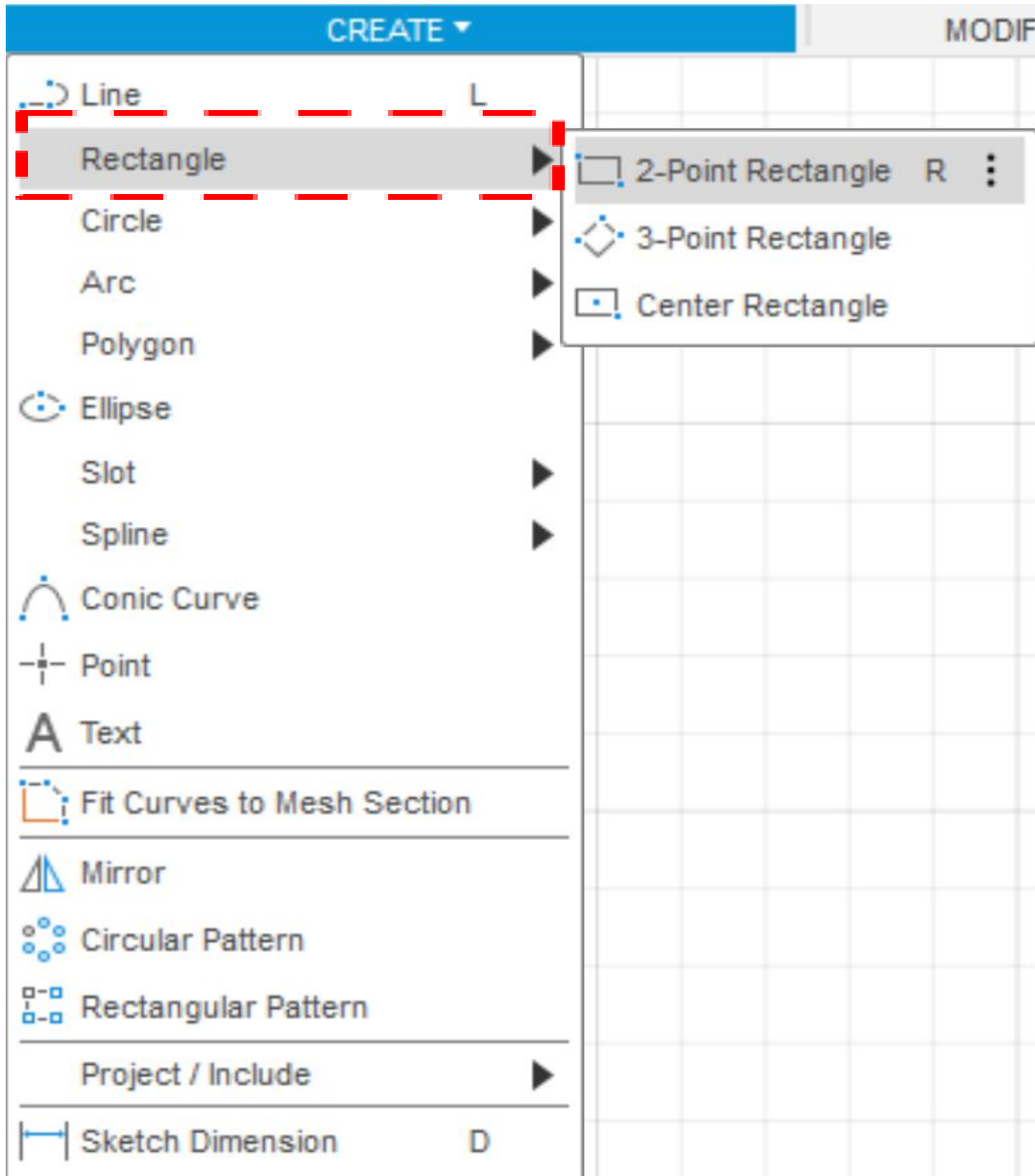Click the icon on the right in the navigation panel to create a sketch.

3.2 Select a Plane

Then Select a plane to sketch on. We would suggest using the XY plane as this will make it easier when we come to the CAM stage.
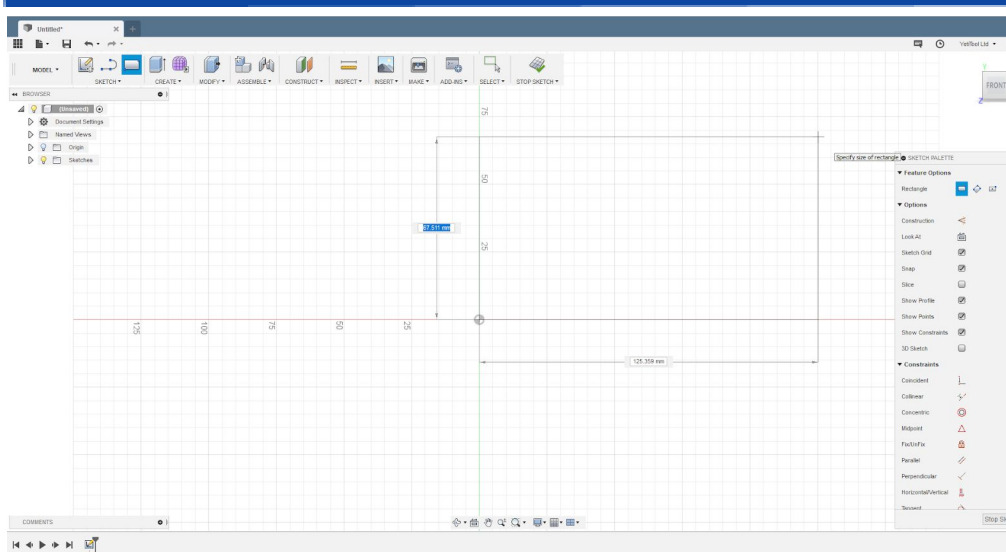
3.3 Drawing Tools

There are a number of different drawing tools available. For this example, we will use the rectangle tool.
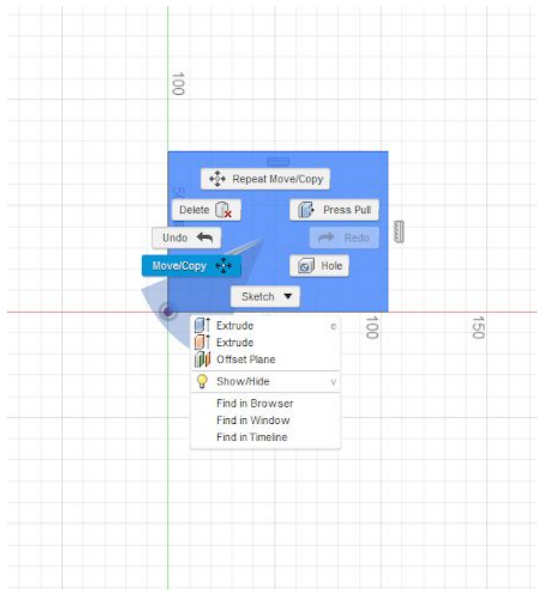

3.4 Drawing a sketch

After selecting the Rectangle tool, click on the sketch plane to create a start point, and then move the mouse to create your desired shape. Clicking again will create and end point and your desired sketch.
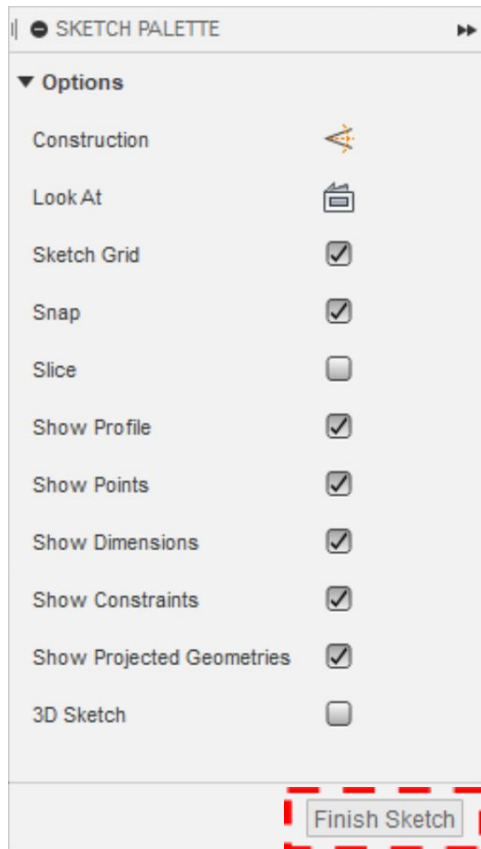
3.5 Moving a Sketch

You can move and reposition a sketch by selecting the sketch you wish to move,
Right click and selecting Move/Copy

3.6 Finishing a Sketch


4 Creating a body

This section will show you how to transform your 2D Sketch in a 3D Model called a body.

4.1 Tools

We are going to use the extrusion tool. Select the tool, and then select the sketch that you wish to extrude. You can also use the shortcut key "E".
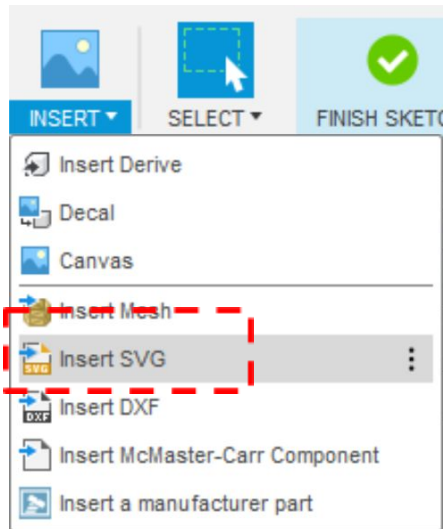
We can now select a distance to extrude the sketch too. We are going to extrude this by 5mm. Enter the amount and then click OK.
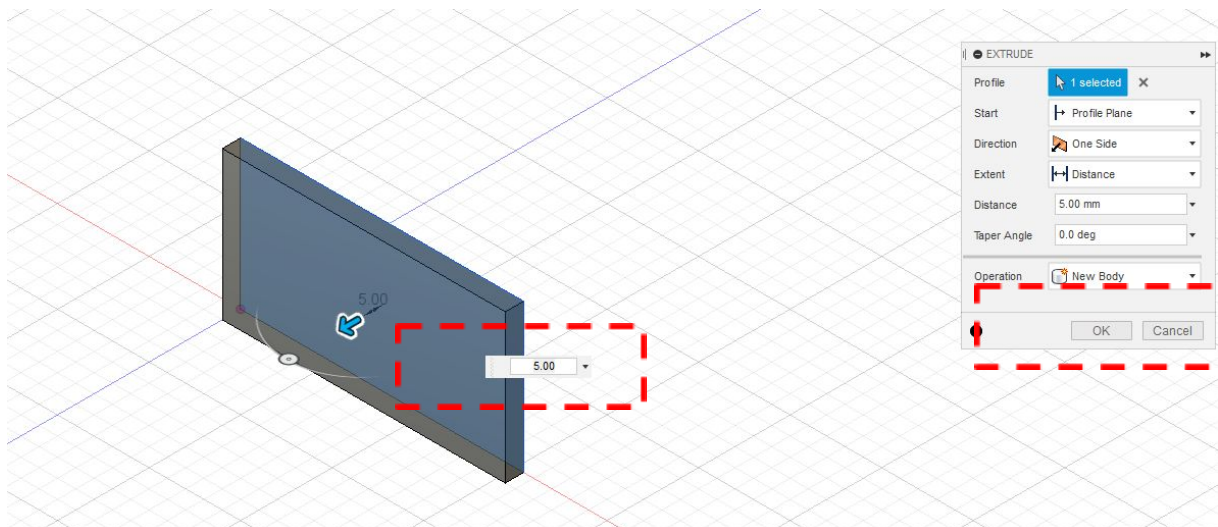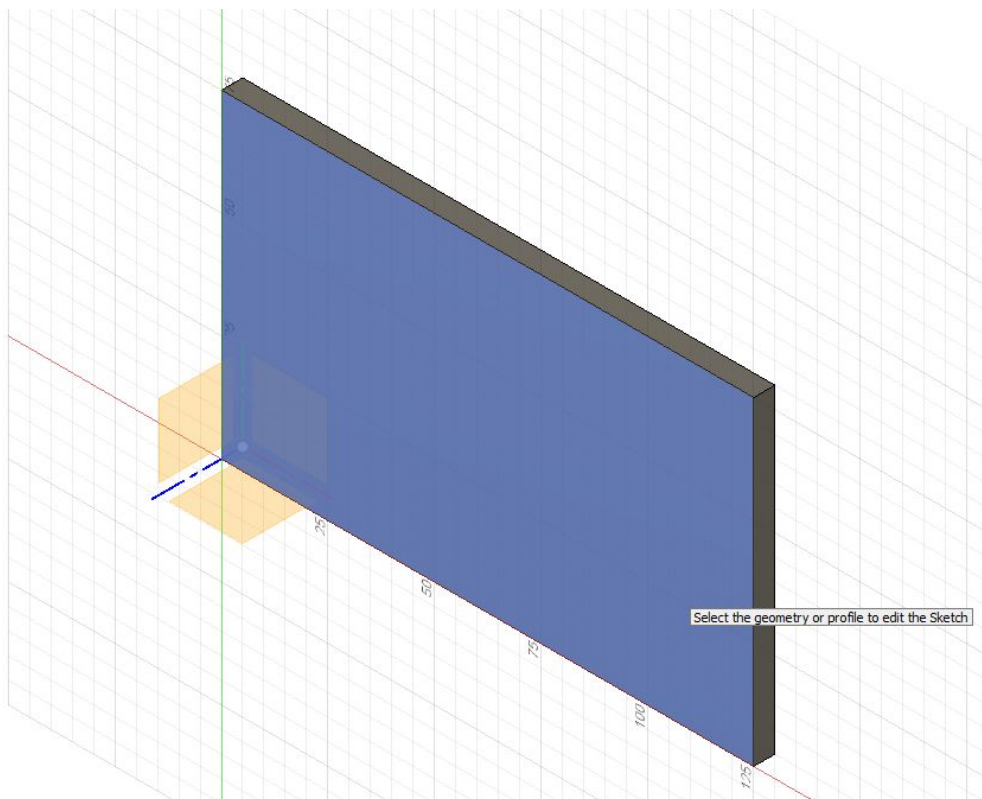
4.2 Inserting an SVG

We are now going to insert a logo onto our model, and then convert it to a model.
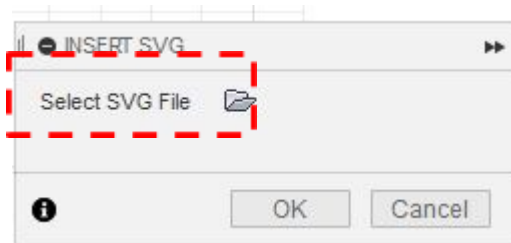
You can insert an SVG as a sketch using the Insert > Insert SVG
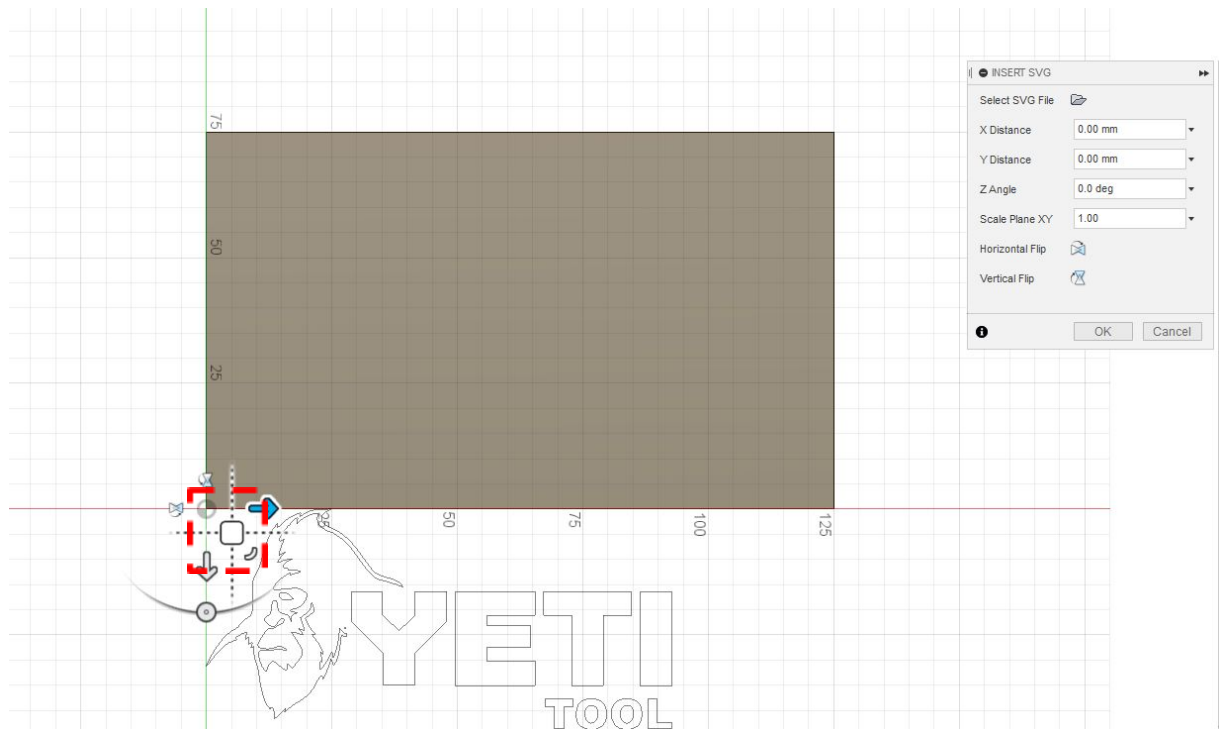

We are then going to select the front face of the model that we created before
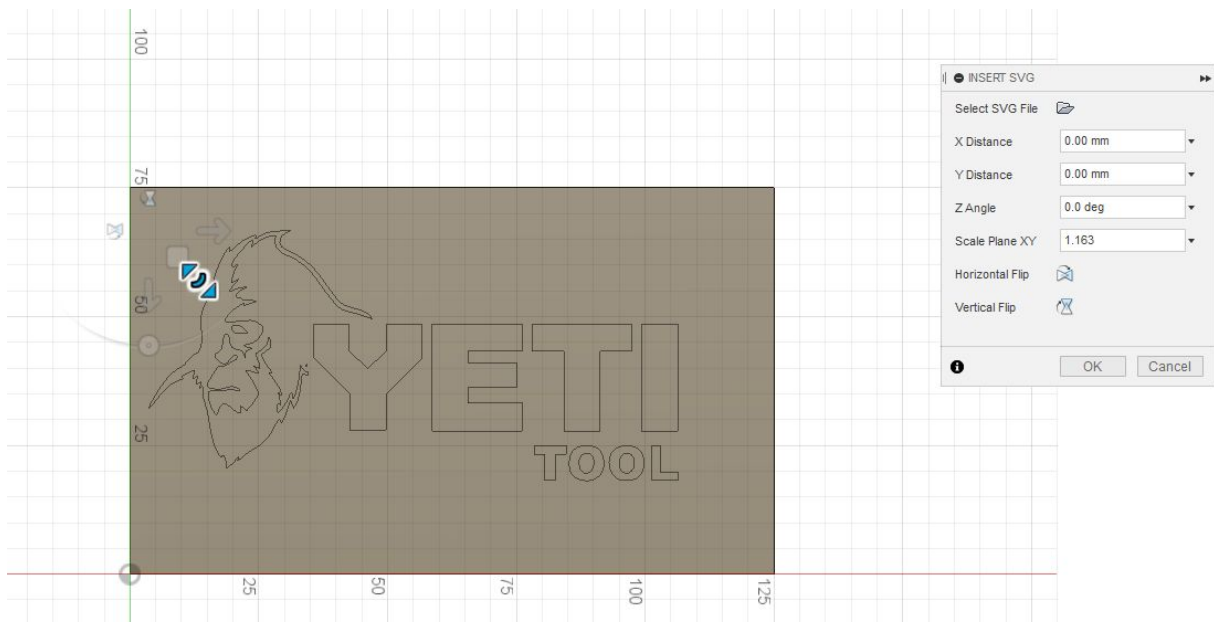
We then need to select and SVG file. *To find out more about converting images into SVG, please see the downloads page on our website here –*
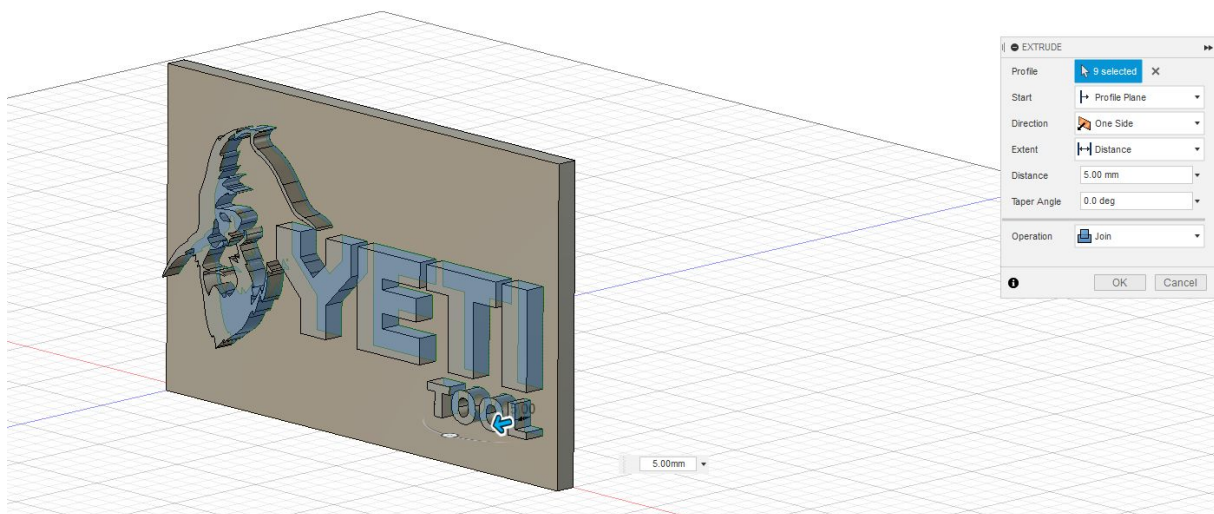*www.yetitool.com/support/downloads*

Now we need to move our sketch into place by clicking, holding and dragging the small square.

And then resize our SVG using the button below by clicking, holding and dragging the arc shaped button
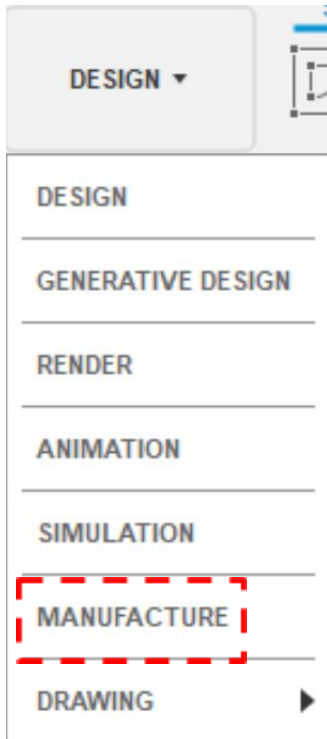
We can then click OK. We are then going to follow the steps that we used before to extrude the sketch of the logo
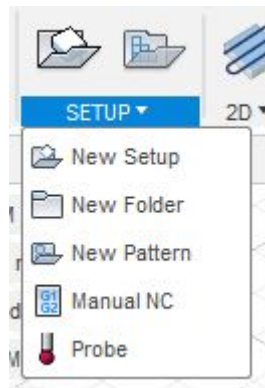

5 Manufacture

Now that you have created a 3D Model, we will need to switch to the manufacturing workspace within fusion. This can be found in the drop-down menu in the top left corner
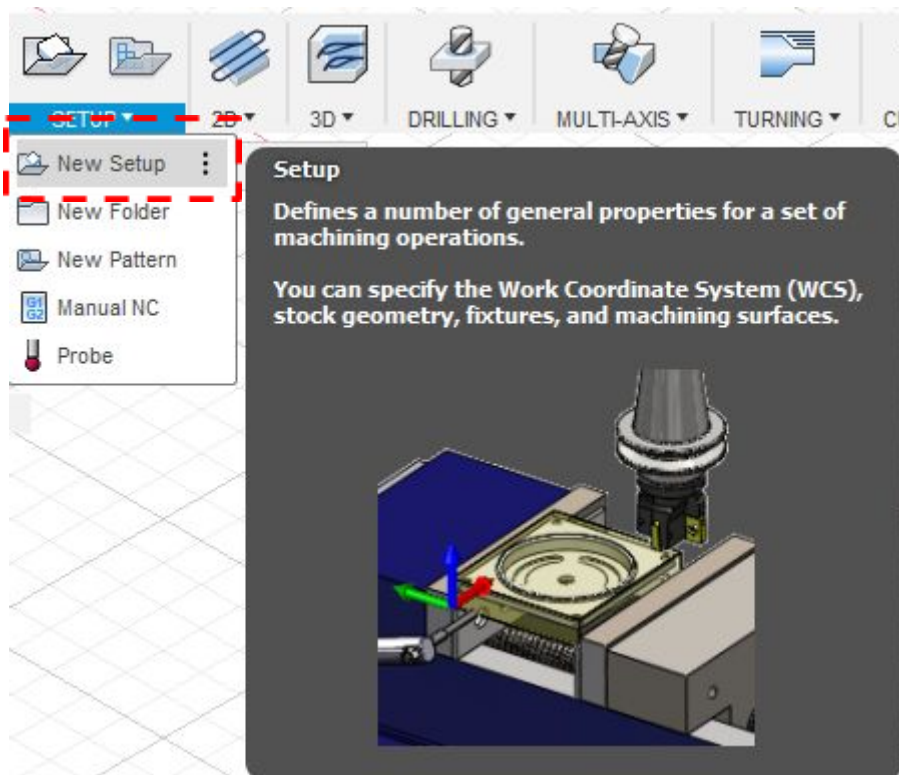
5.1 Creating a Setup

We will first start by creating a setup for our CAM programme.


5.1.1 Setup

Click "New Setup"

5.1.2 Stock

First, we need to set our stock box point. We can do this by setting the "Orientation" to "Model Orientation". We are going to set this to the corner of the model, ensuring that our Z Axis is the same orientation as the plunge direction of the spindle. We also need to set the longer side of the model as Y Axis.
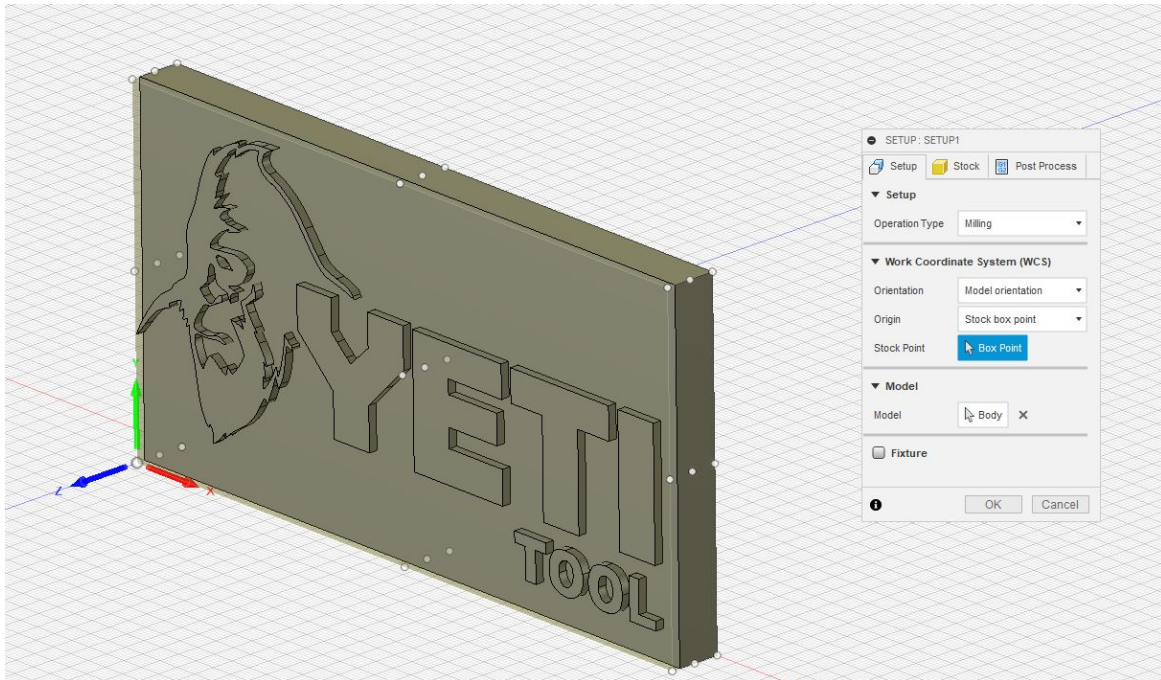
If this needs to be changed, we can change the "Orientation" to "Select Z Axis/ Plane and Y Axis" and select the edges of the model that we wish to represent Z and Y.
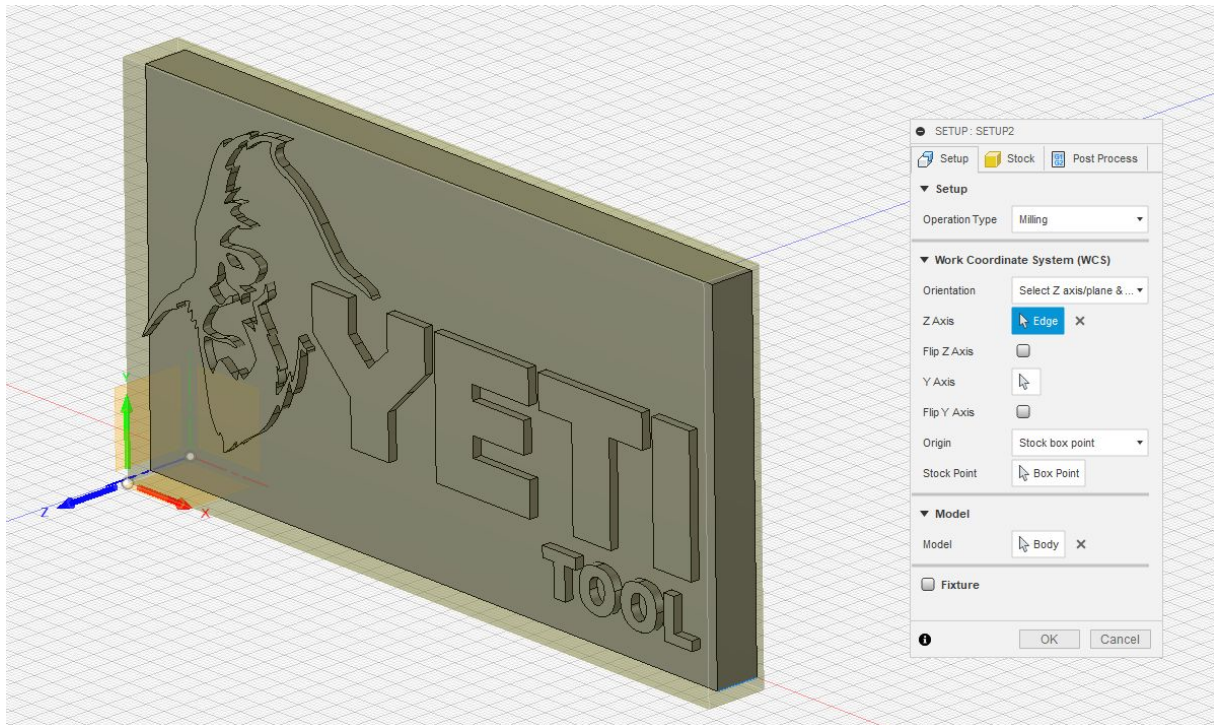
Next, we need to set the stock offsets to 0mm
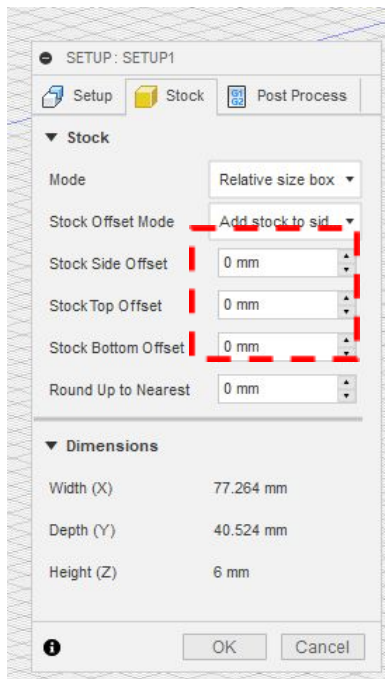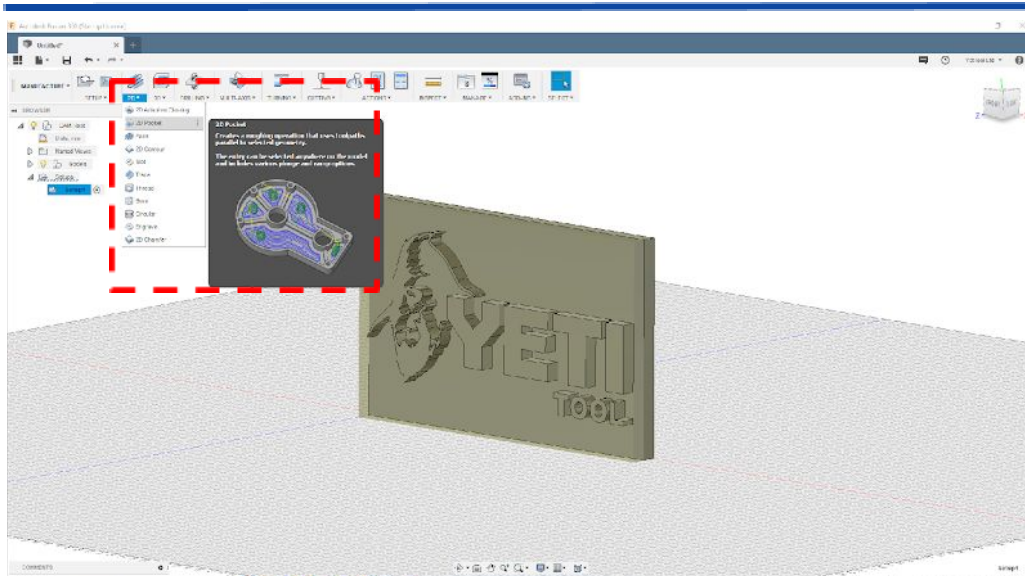
5.2 2D Operations

We are now going to select the correct 2D operation for the model we have created. In order to clear the material around our logo we are going to choose the "2D Pocket" operation.

Explanations of which 2D operation to use can be found by hovering on the operation name. Further information is available here:
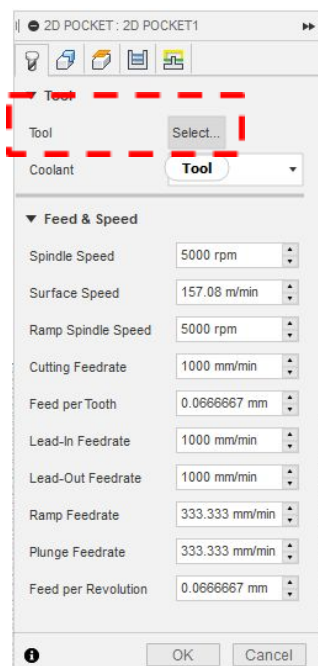http://help.autodesk.com/view/fusion360/ENU/?guid=GUID20E579E7-B808-47D1-BFB2-14056A2B1CFE
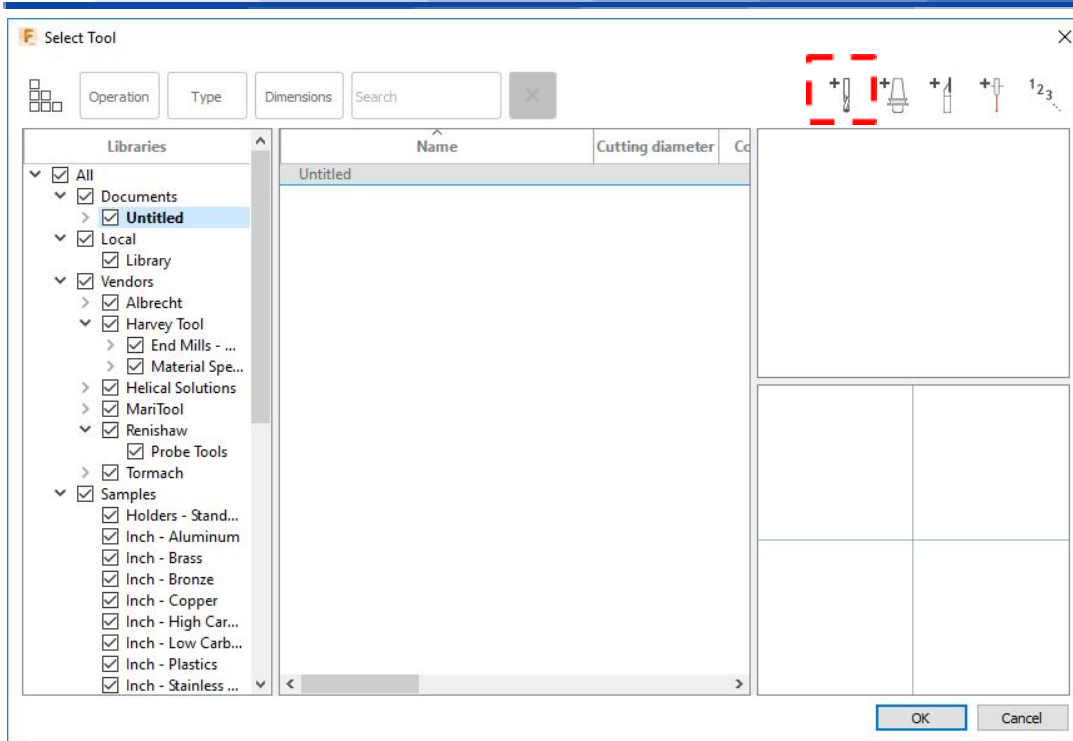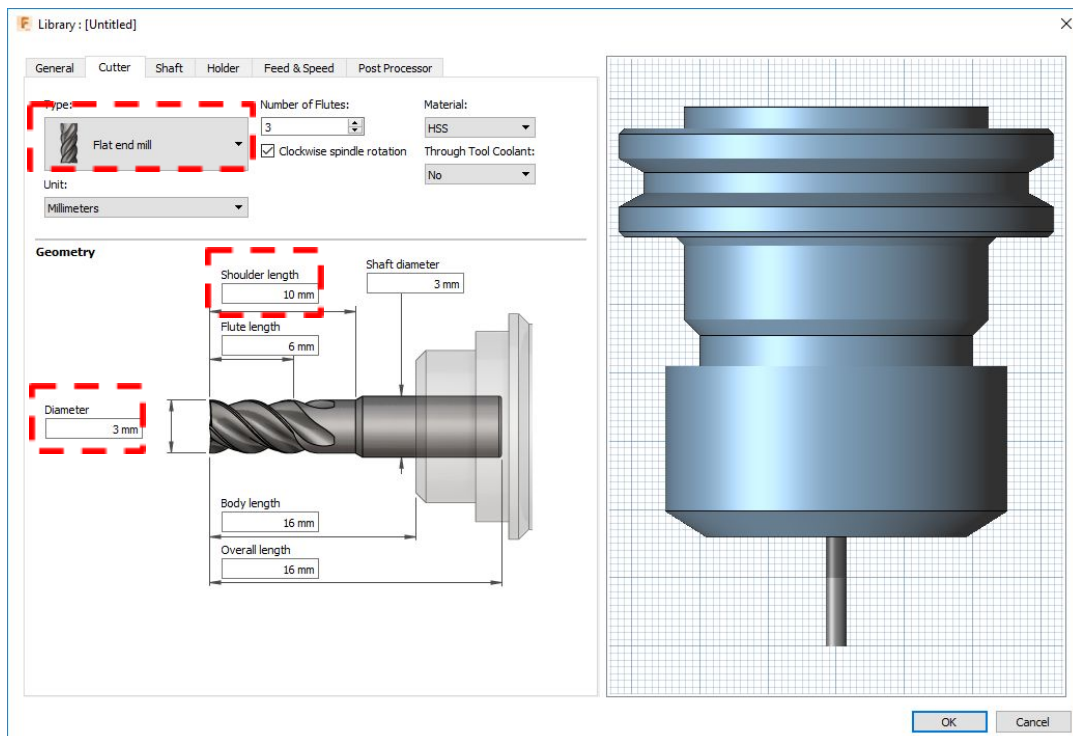
5.2.1 2D Pocket

Select 2D Pocket

We now need to select the correct tool. For this we are going to use a 3mm Straight Flute which has a shoulder length of 10mm
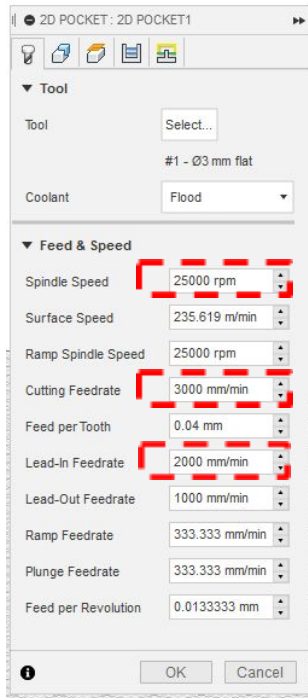

Now select "Add new tool"

Set the following options

Click OK, and then OK again.

We then need to set the correct feeds and speeds. These vary depending on material, and you may need to use an online feeds and speeds calculator to ensure these are set correctly.
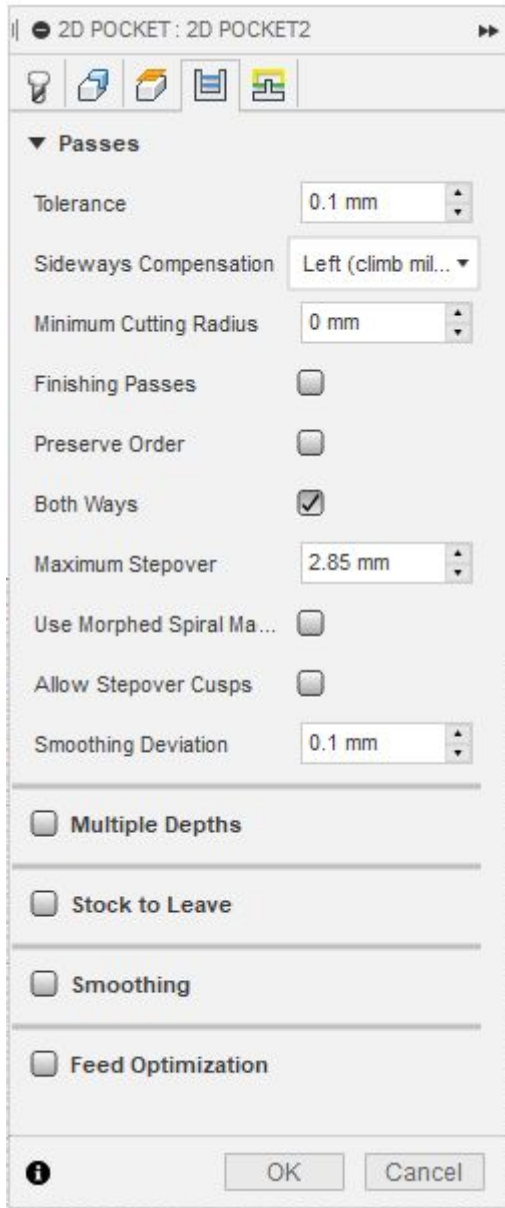
For this operation we will use the settings below

Now we need to select the Geometry to cut. We are going to select the pocket around our logo.
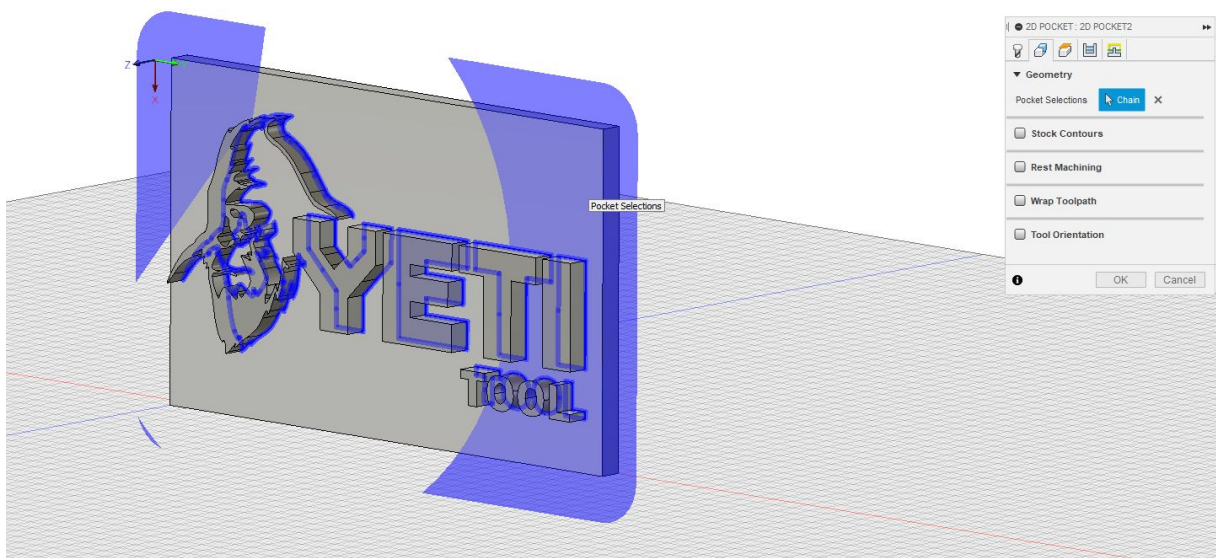
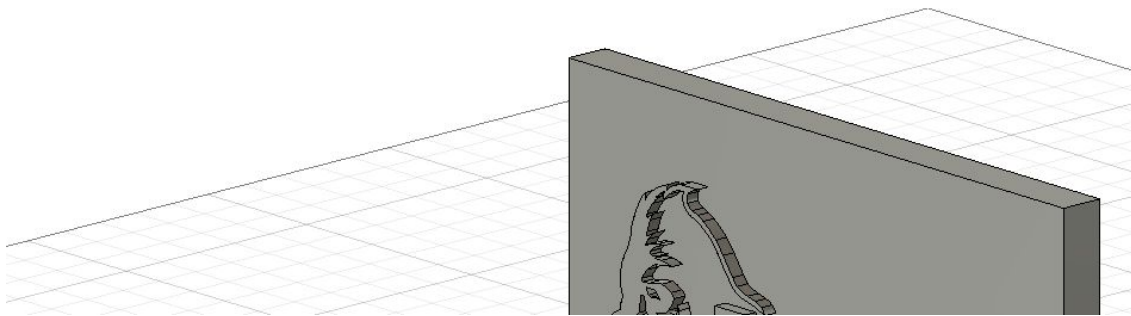Next we need to ensure that we select both ways, and that we uncheck the box which says "Stock to leave"

Click OK
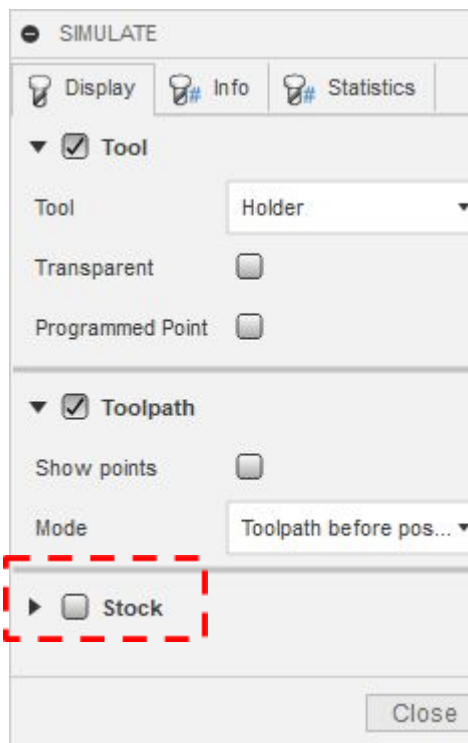
Our toolpath will now be generated

5.3 Tool Path Simulation

We can now simulate our toolpath within fusion to ensure there are no errors
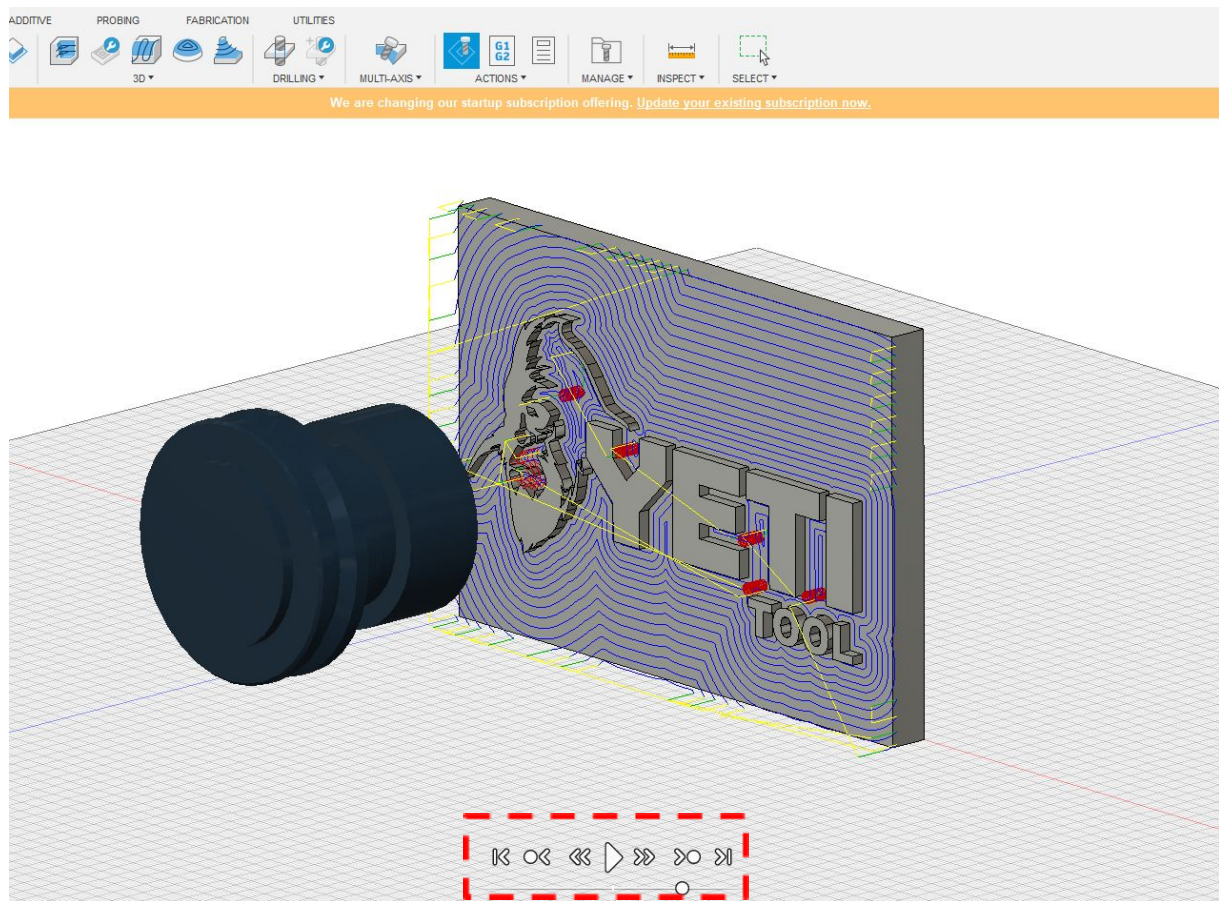

You will now be able to view a simulation of the toolpaths that you have generated.

You can toggle to view or hide stock

You can use the button along the bottom to play the simulation and adjust viewing speed
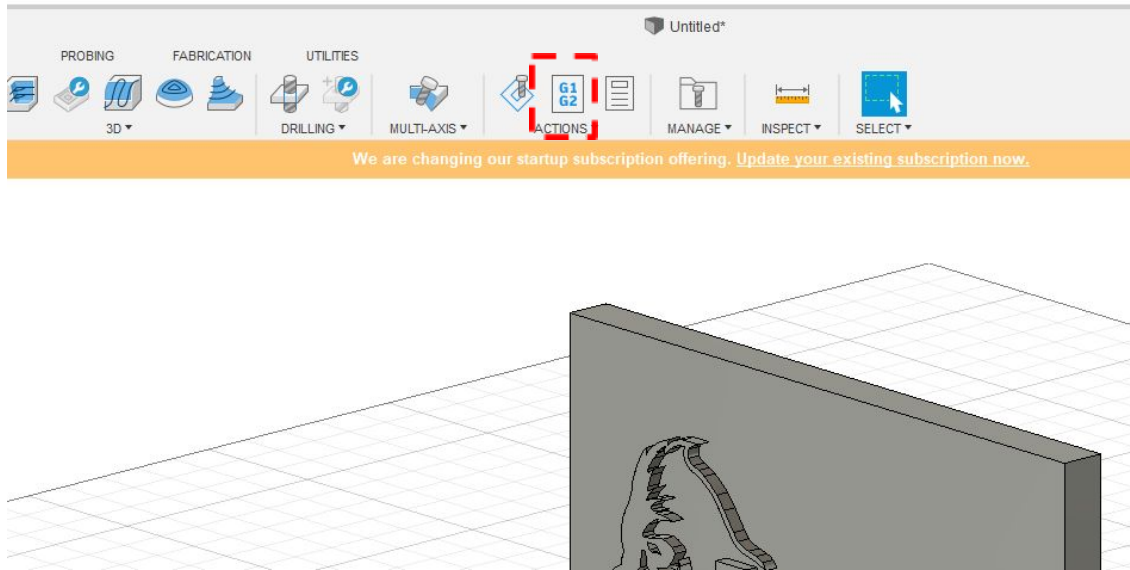
If there are any collisions, they will be shown in red in the progress bar at the bottom
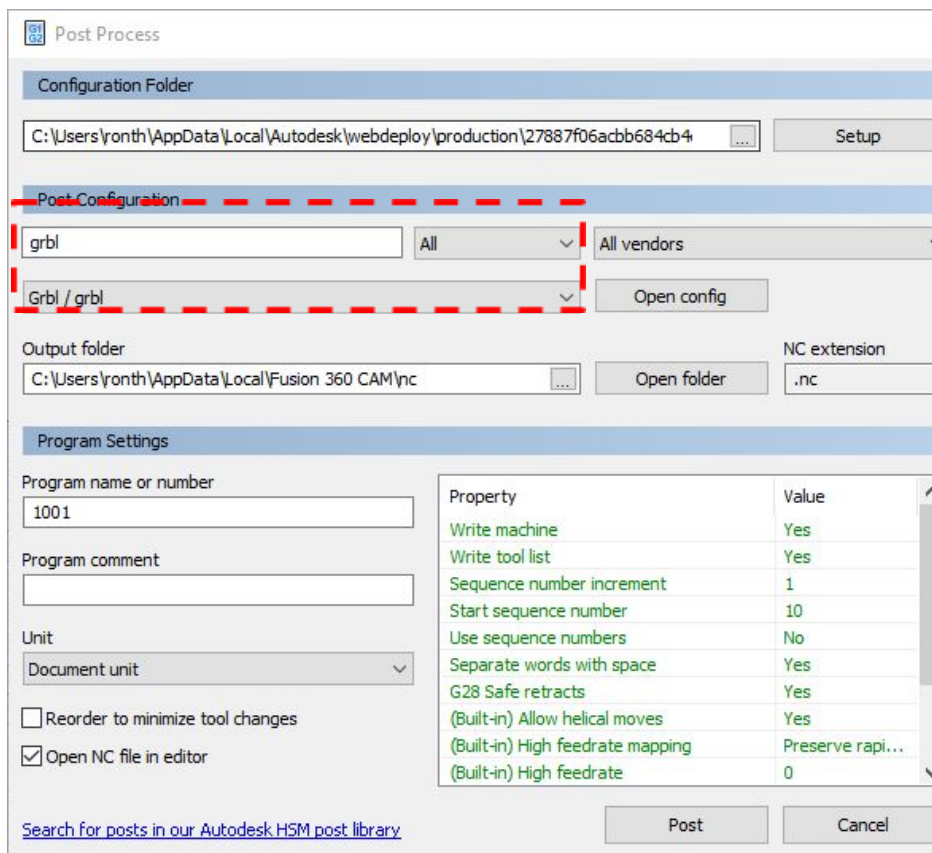
6 Post Process

We can now post process our toolpath ready to send to SmartBench

6.1 Post Configuration

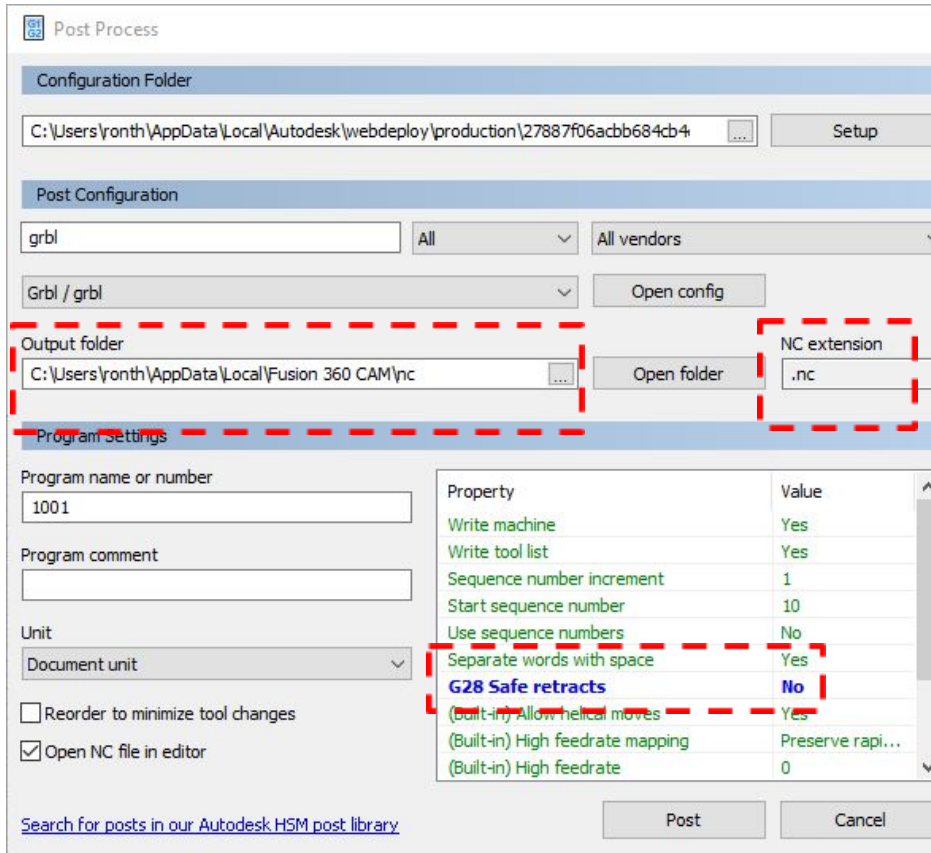We need to post process our file using the GRBL post processor. Select Post process

We then need to search for GRBL

6.2 Saving a File

We can then select an output folder to save to as well as the filename extension. We need to use .nc or .gcode

We also need to ensure G28 safe retracts is set to no

Now click Post

7 File Transfer

We can now transfer our file to SmartBench using either USB or send the file over Wi-Fi. Please refer to the Easycut manual