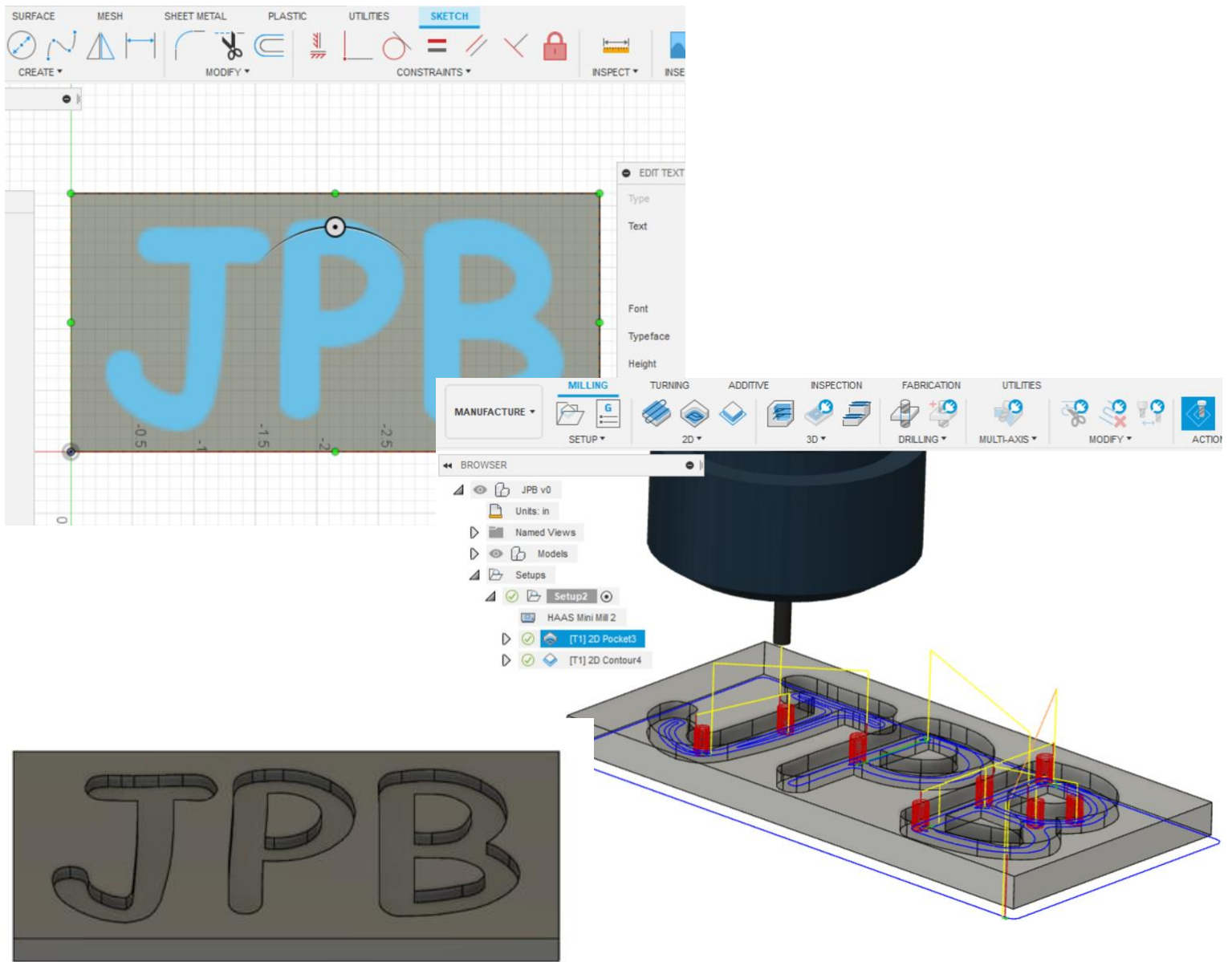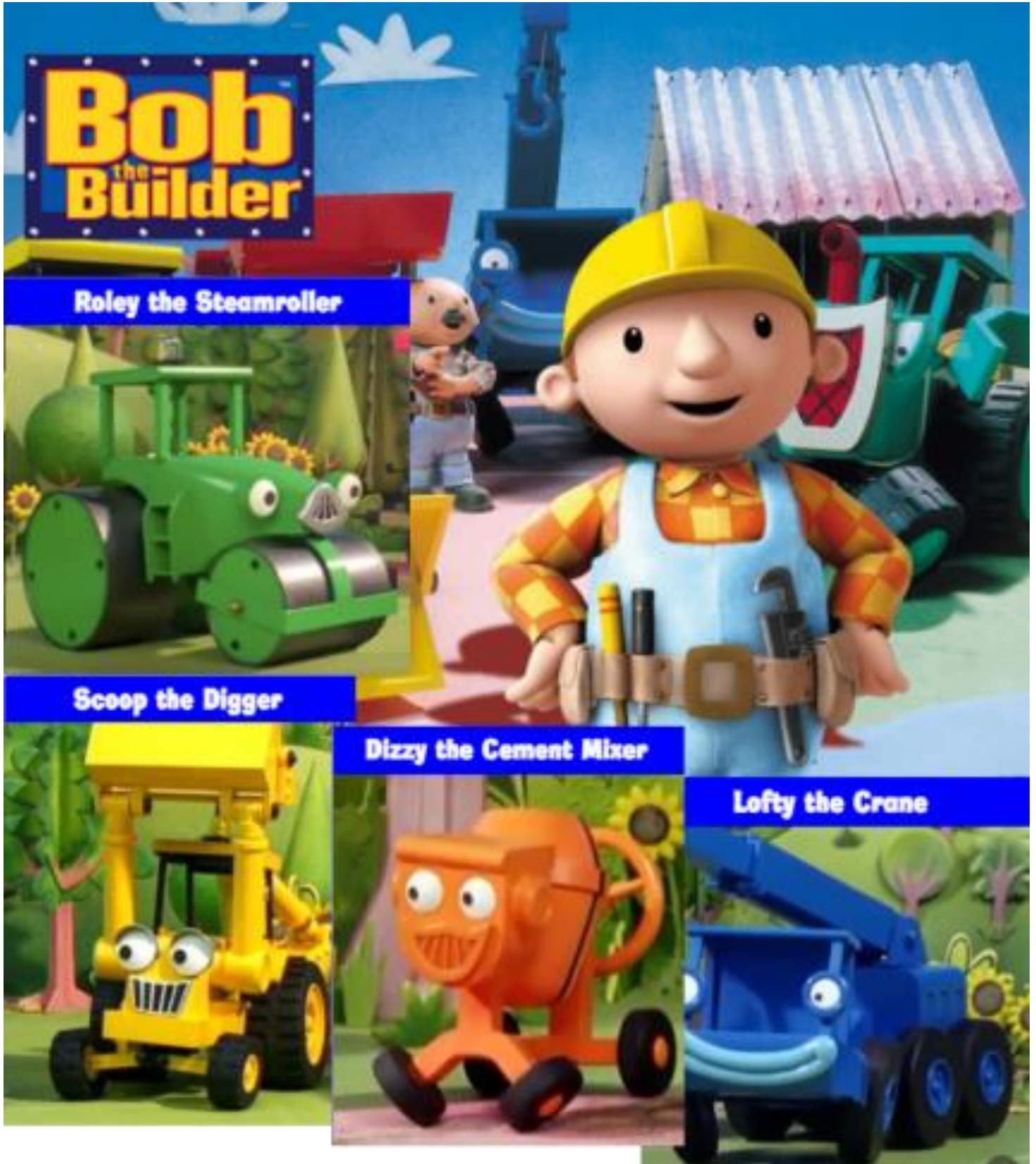# CNC Machining using Fusion by Mr. Barbetta

Today's lesson is sponsored by **Bob the Builder**

Contents

Using This Document

The best way to follow this document is to **reduce the width of the Fusion 360 window** and have this pdf document open in Chrome browser as shown below. This document can be **downloaded from Schoology and then dragged into Chrome** and scaled down to 80%.
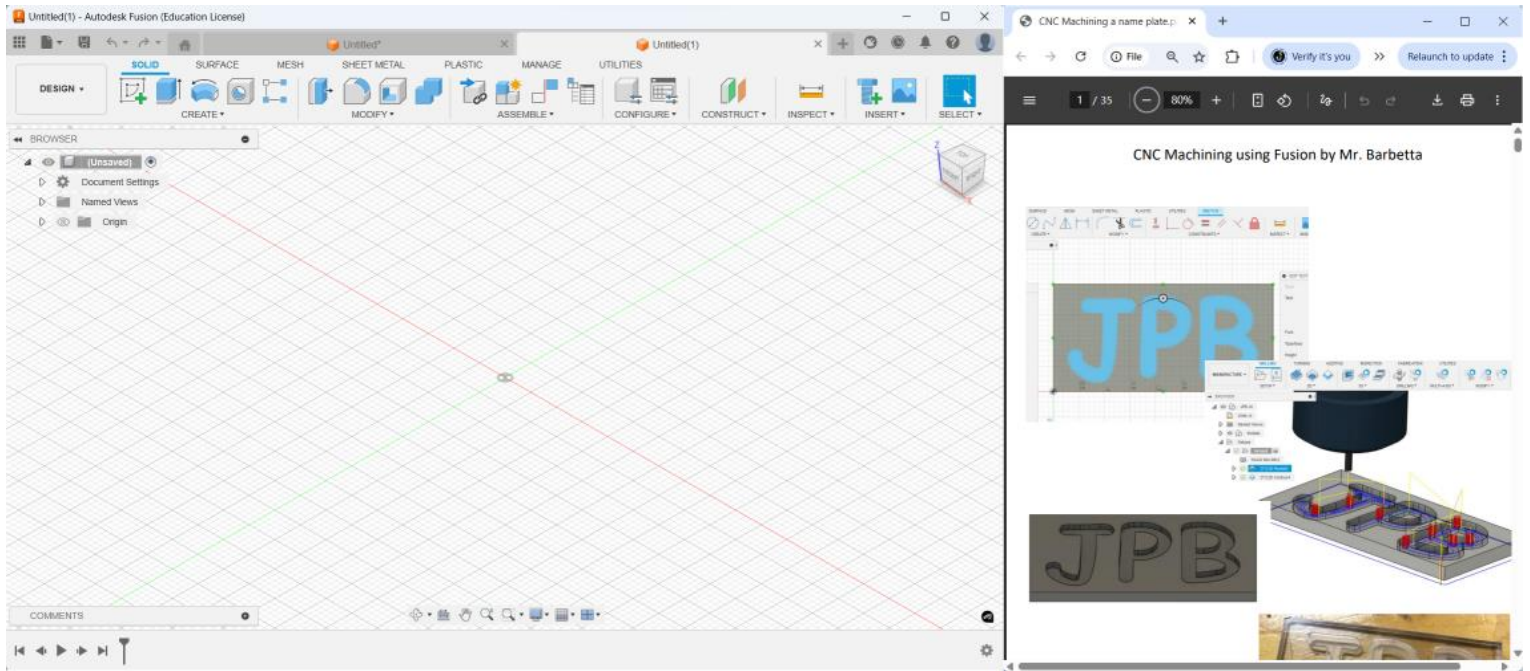
The Fusion 360 window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.
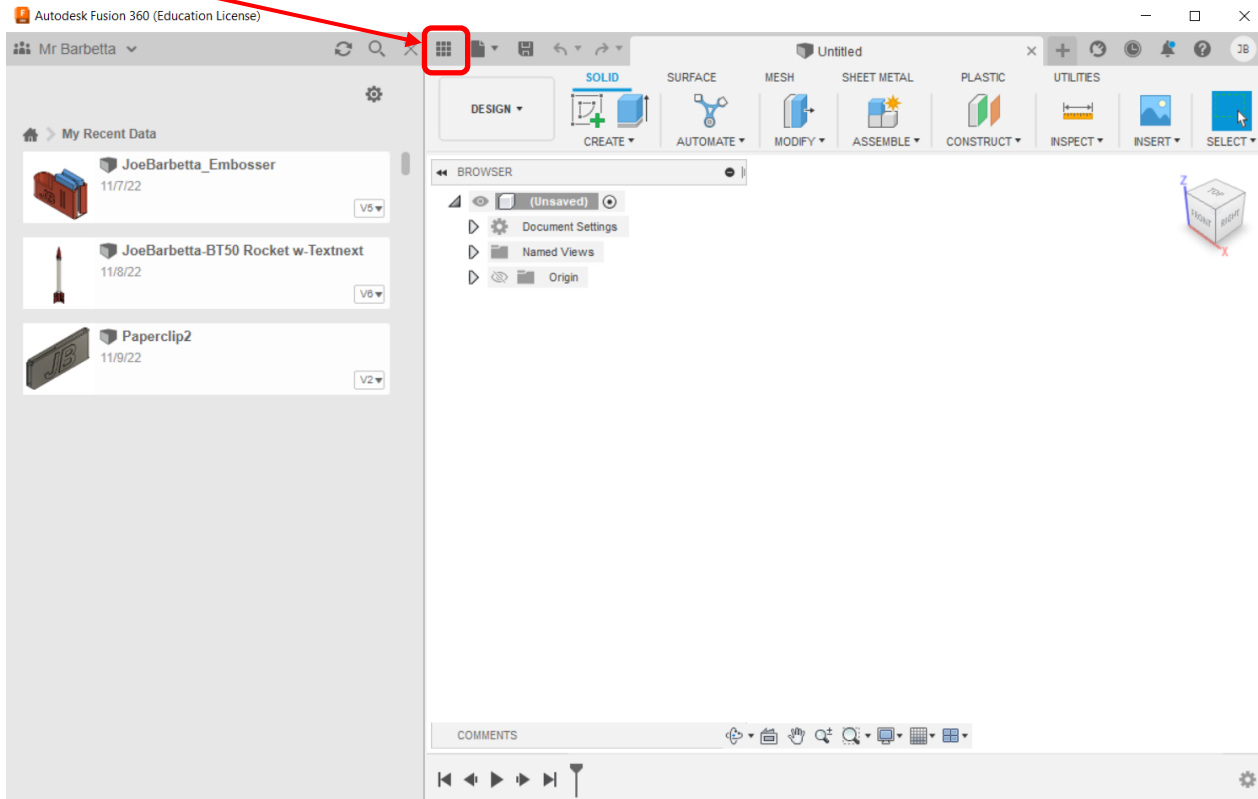
Setting Preferences

If this is the first time that you are using Fusion, set some Preferences that will be used with this and future projects.
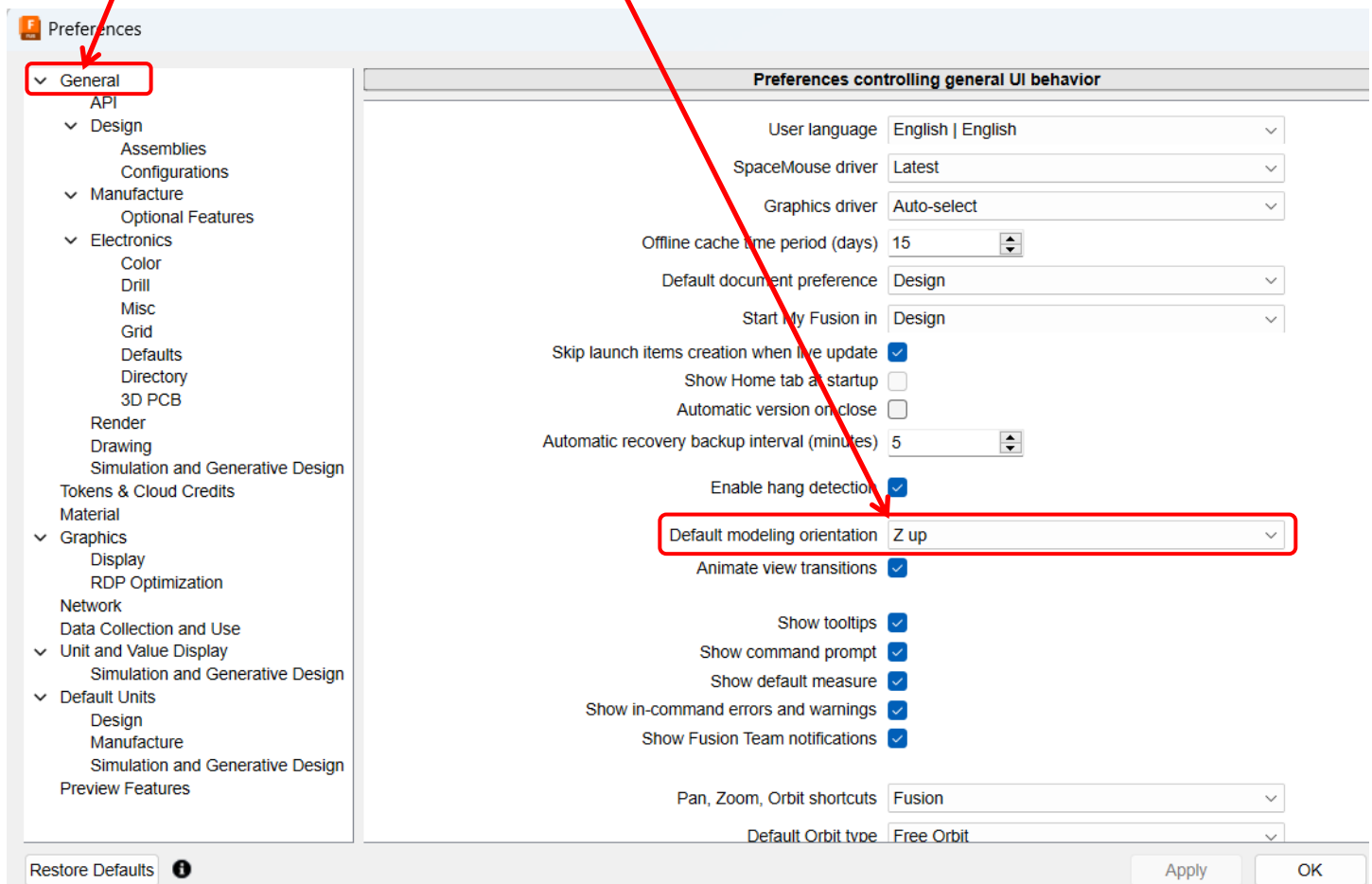
Setting preferences for MAUGA (Make American Units Great Again) Most manufacturing in the US uses inches for dimensions.

Right-click on your **profile icon at the top right** and select **Preferences**.
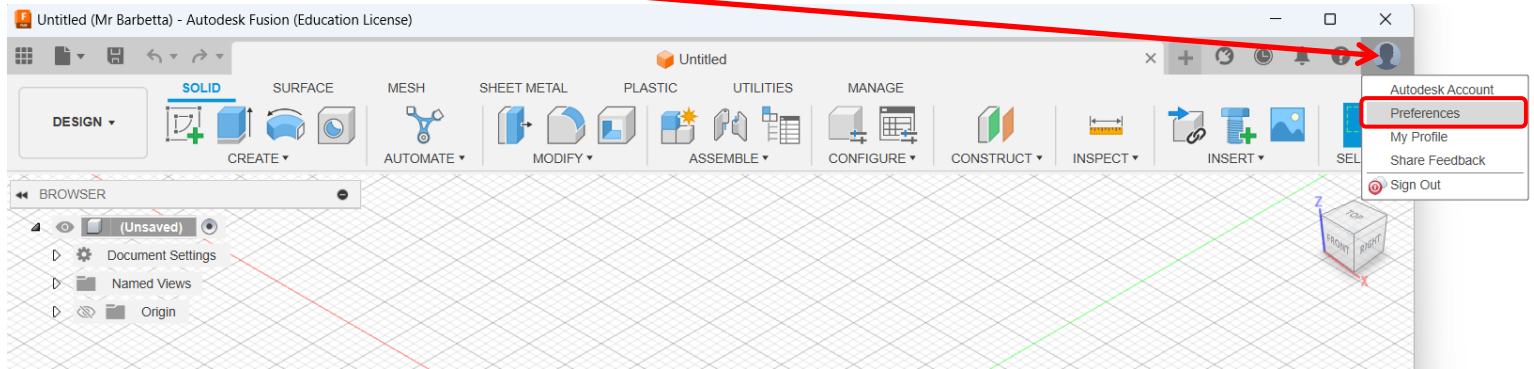
In the **General** category verify that "**Default modeling orientation**" is "**Z up**".

In the **Default Units** category (selected on the left):
   in **Design** change units to "**in**"
   in **Manufacture** change units to "**in**"
Click "**OK**".
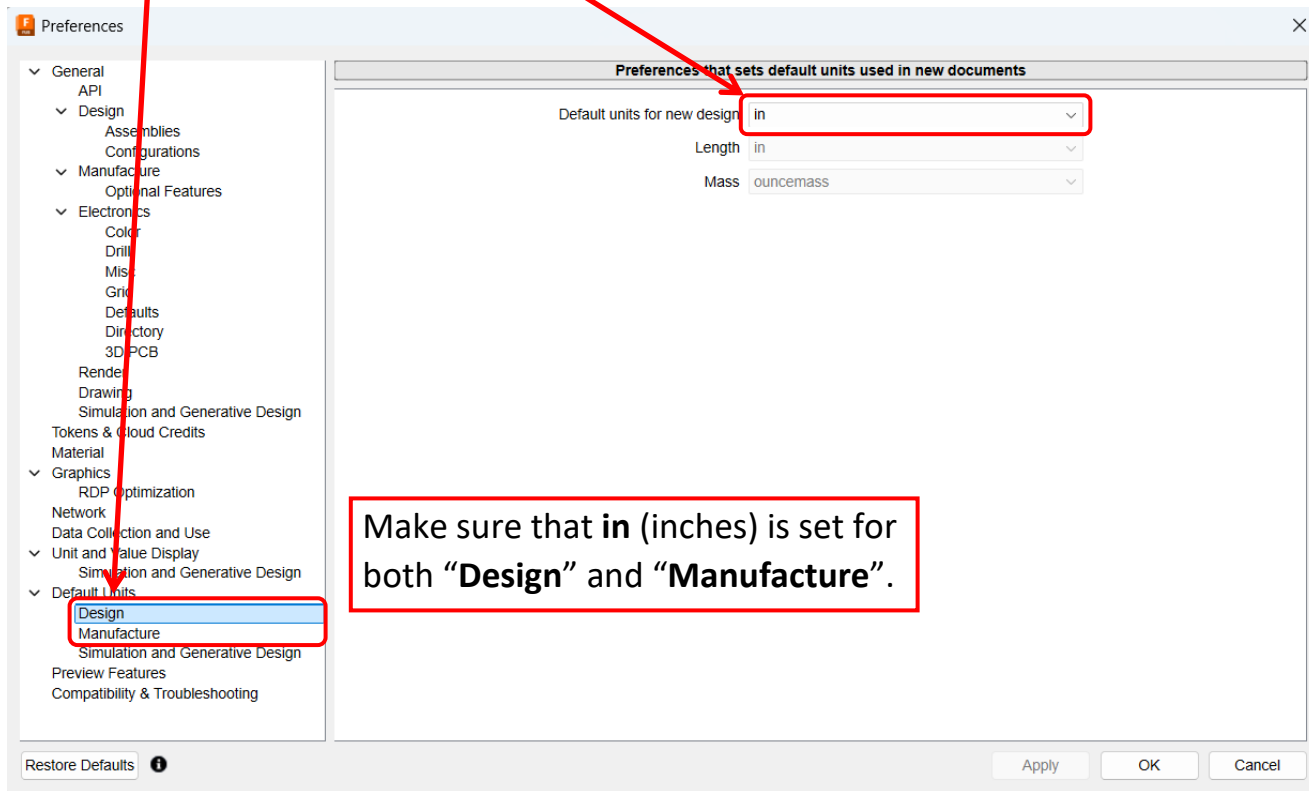
Make sure that **in** (inches) is set for both "**Design**" and "**Manufacture**".

-If this is the first time using Fusion, click on the **X** to start a new design that uses the new preferences that were just set.
-Click on the **arrow** for **Document Settings** to view the units to verify that they are set to **in, ouncemass**
-Verify that the **Z** axis of the **View Cube** is pointing up.

Creating a Sketch

A Fusion professional would suggest that a New Component should be created. You can say **"Dude, I'm just making a simple block with text."**
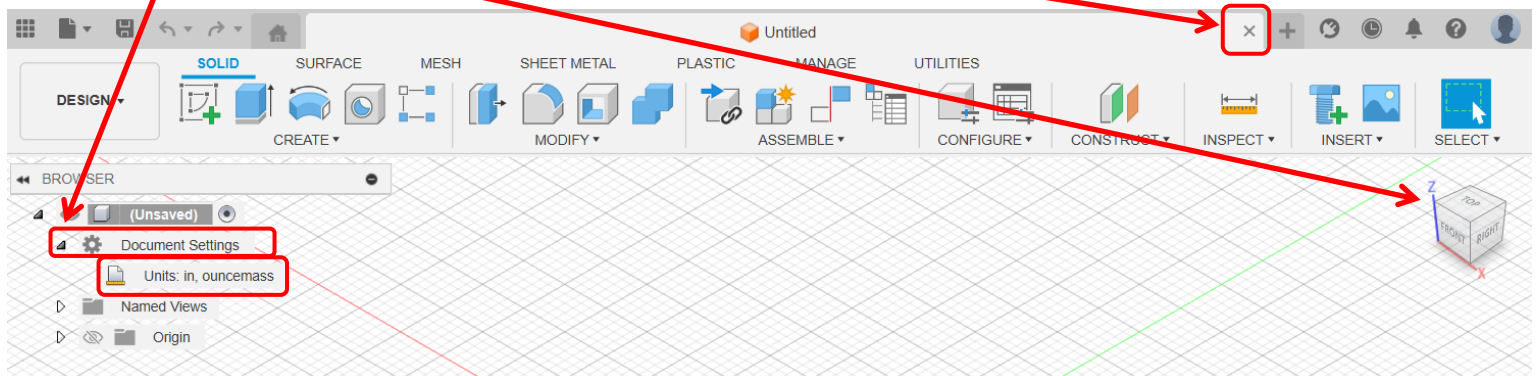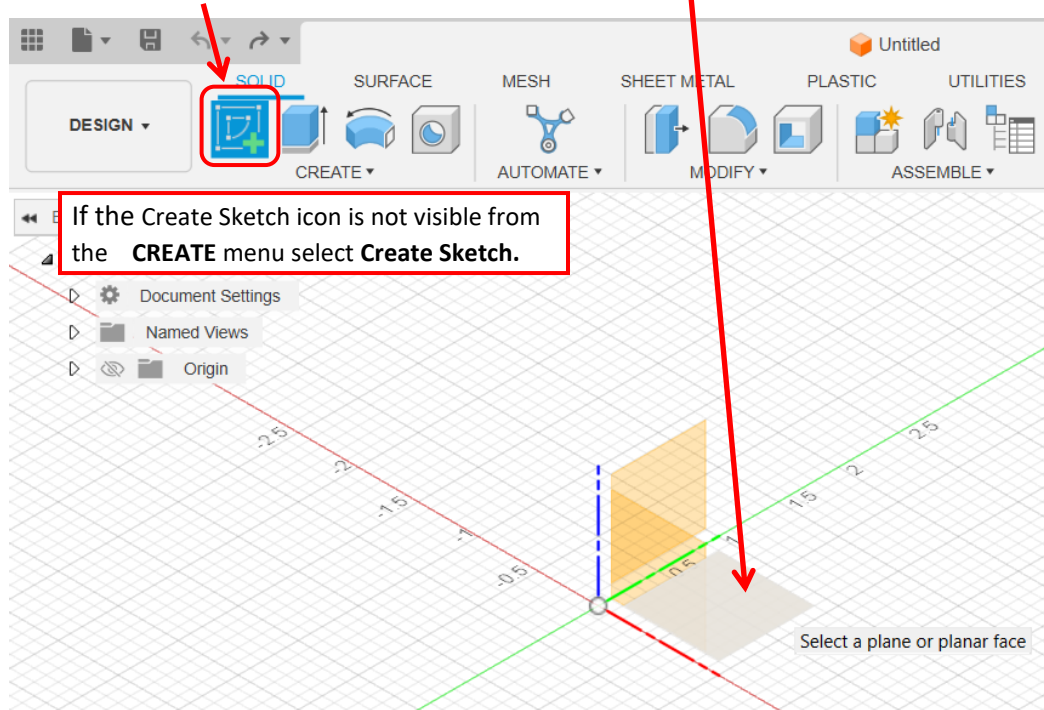
- select the **Create Sketch** tool and click on the bottom (X-Y) plane.

If the Create Sketch icon is not visible from the **CREATE** menu select **Create Sketch.**

Select a plane or planar face

- verify that the **View Cube** shows **TOP**
- if it doesn't use **Undo** and try selecting the bottom plane again. It may also not show top if the Preferences were not set properly.

Notice the green and red axes and the scales shown in inches.

Hold the **mouse wheel down to pan** (shift the view) and **rotate the mouse wheel to Zoom.**
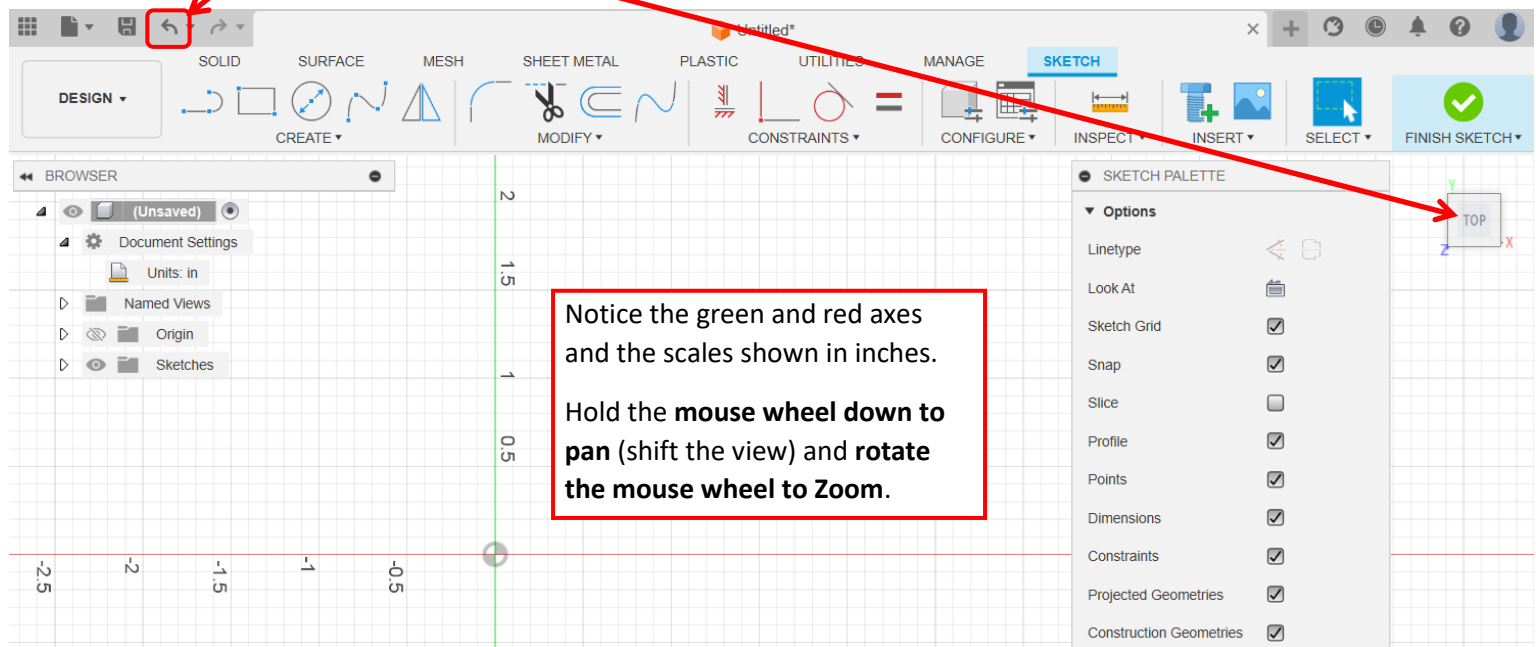
- select the **Rectangle** tool.
- click on the **origin to place the first corner**, then drag the **other corner up and to the right** until the dimensions sho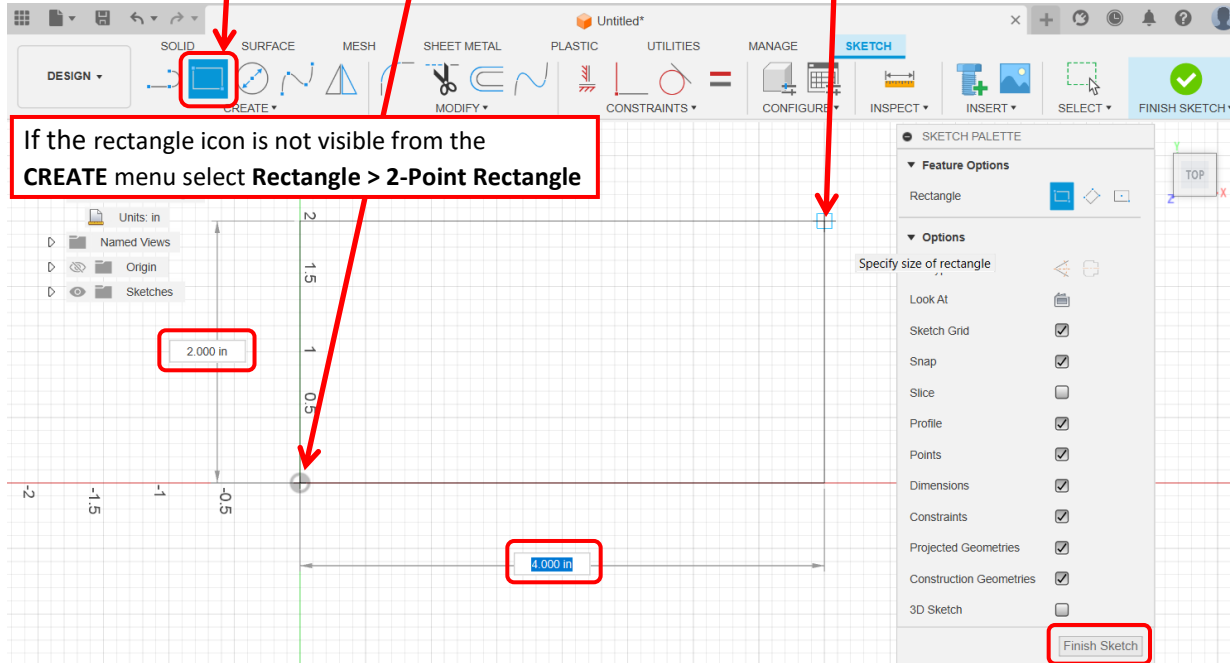w **4.000 in** for width and **2.000 in** for height. **Alternatively** one can also **enter the dimensions** and use the **Tab key** to switch between width and height.
- click **Finish Sketch**

If the rectangle icon is not visible from the **CREATE** menu select **Rectangle > 2-Point Rectangle**

- click on the **Home** icon at the **View Cube** to achieve a home view.
- Zoom using the mouse wheel to zoom out to a view similar to that below.
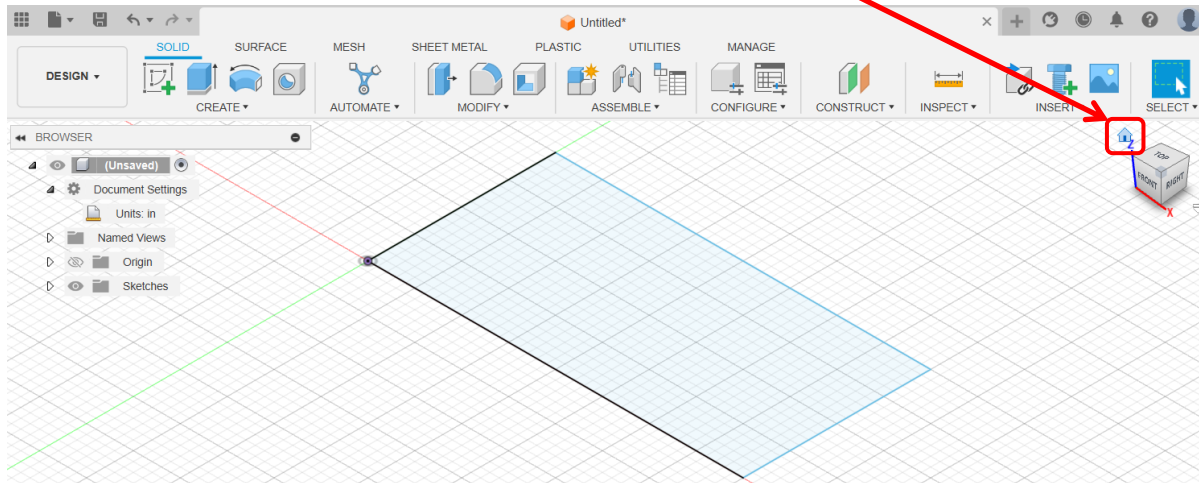
-Select the **Extrude** tool, enter a value of **0.25 for Distance**, and click **OK.**

If the Extrude icon is not visible from the **CREATE** menu select **Extrude**.
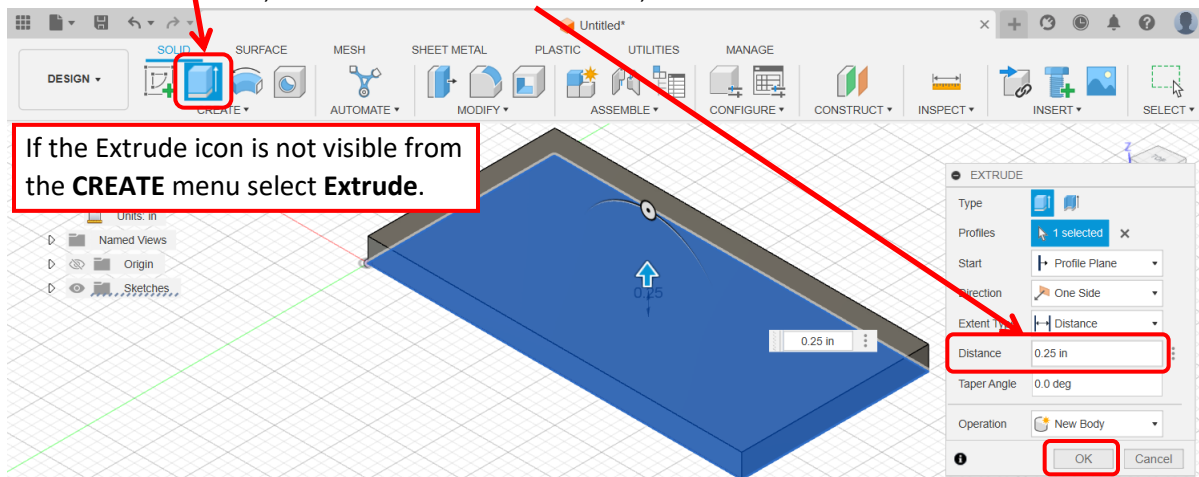
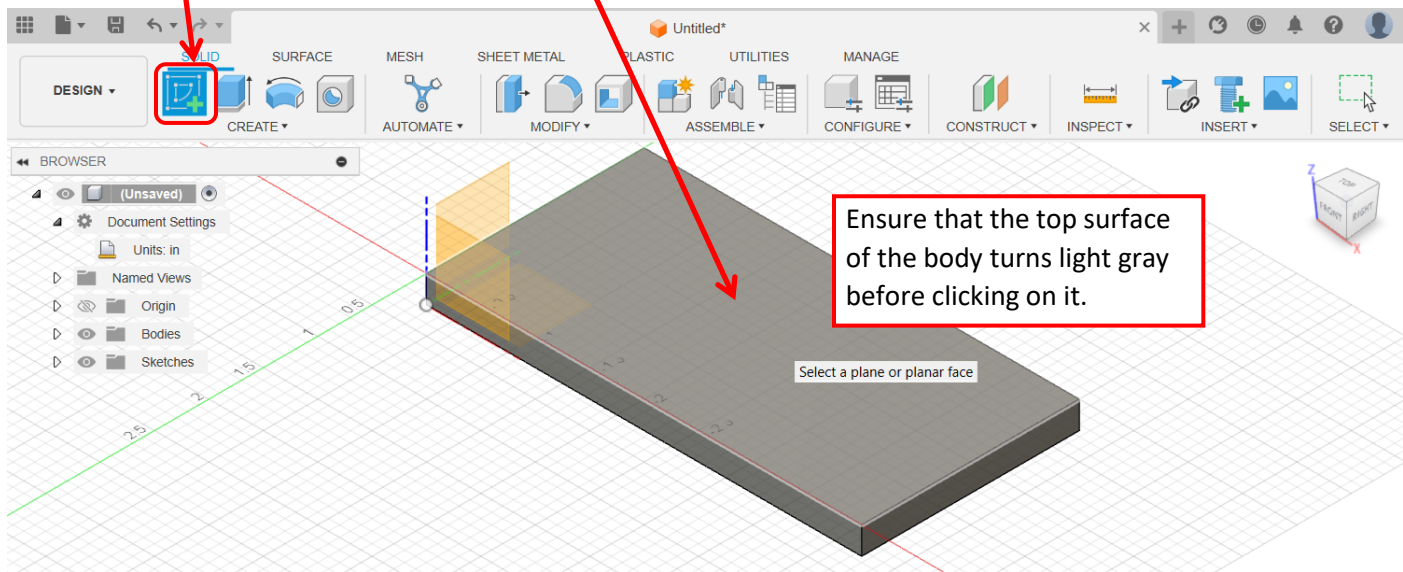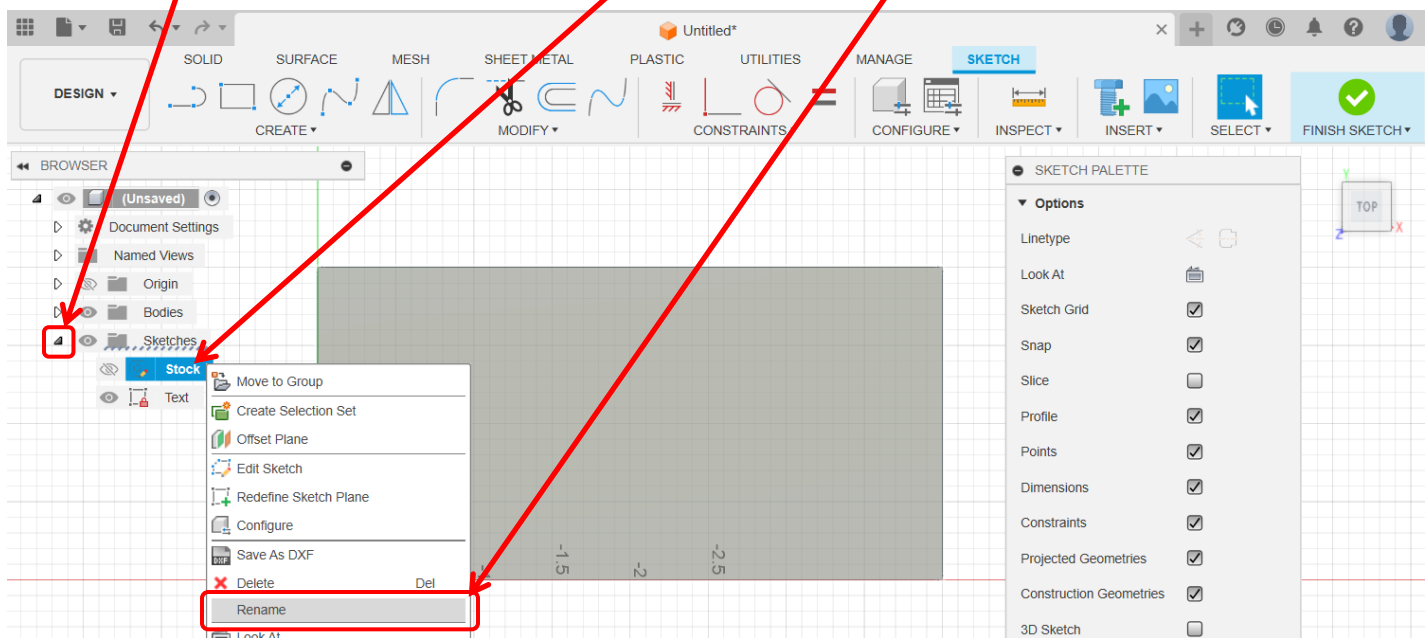# Creating Text

- select the **Create Sketch** tool and **click on the top surface** of the body.

Ensure that the top surface of the body turns light gray before clicking on it.

Select a plane or planar face

- click on the **arrow** next to **Sketches**, right-click on the **sketch name** and select **Rename**.
- change the first name to **Stock** and the second to **Text**.

- from the **CREATE** menu select **Text**

if a message about **Parametric Text** pops up, click **OK** to close it.

Parametric Text

You can reference *Text* as a parameter in sketches, features, and configurations. To create them:

Learn more

☐ Don't show this again

OK
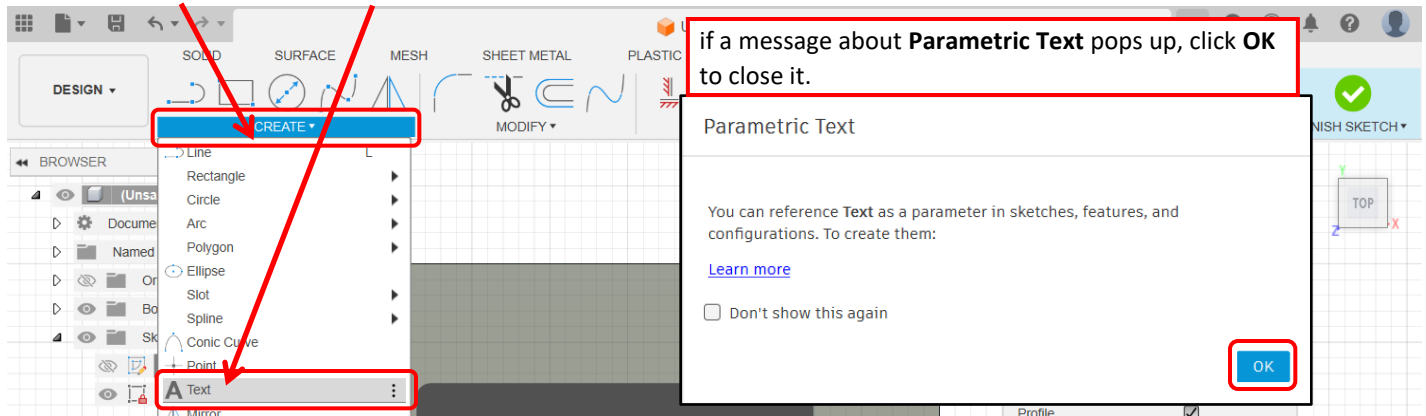
- click on the **top left corner** to start the creation of the dashed rectangle and then the **bottom right corner** to finish the rectangular area.
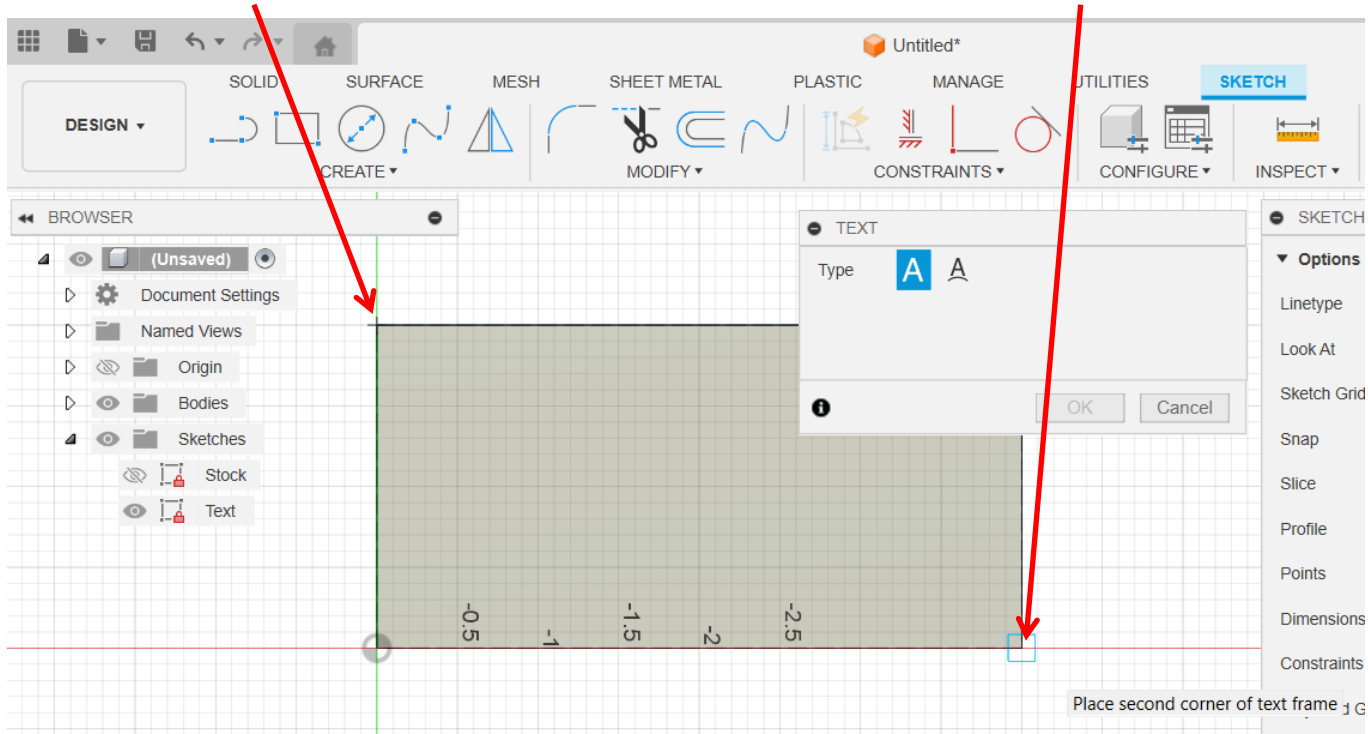

It is very important that the thickness of each letter segment is wider than 1/8". **Make the text as large as possible, choose a non-serif font with wide letter segments, and make it bold if possible.** Notes on the following page will help.

- enter **your initials in the Text box**
- set the Font, click on **B** for Bold, and set the Height to **1.5**
- click on the two **middle alignment options**
- **before clicking OK review the information on the following page**

Text entered has to be **surrounded by single quotes**. Fusion automatically adds them, but sometimes they can get deleted when entering text.

Note the the text has wide segments and is almost as large as the stock.
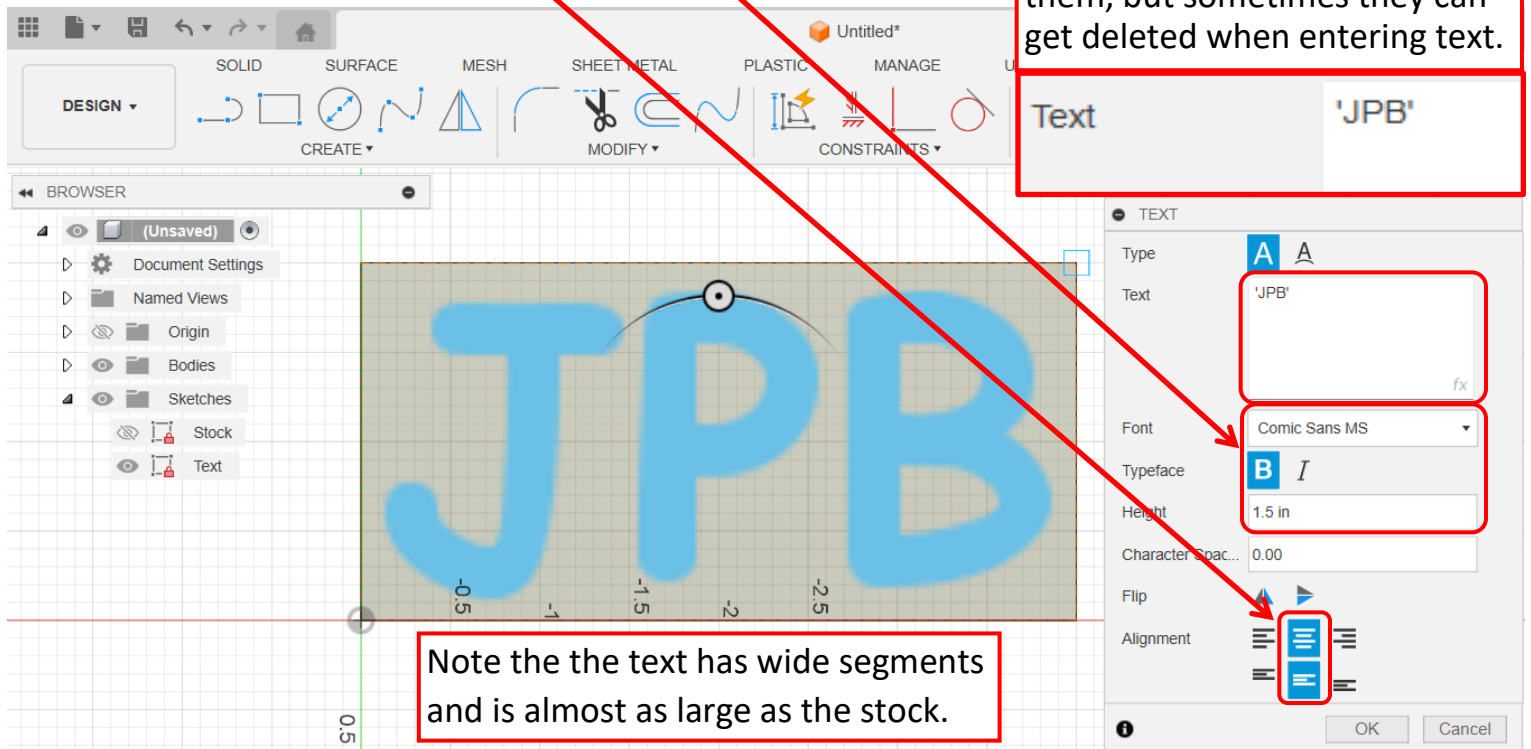
A key consideration is to have the text as large as possible while still maintaining about 0.1" between the edge of the stock and any text elements. It is also important to use text that doesn't have any narrow sections. All this will help ensure that the 1/8" endmill can mill out all the letters.

Use a Font that is best for CNC milling.

Because a round endmill will be used (we lost or square cross-section end mills) it is best to use a font with rounded ends. Ideally, we also want to use an endmill of a diameter of 1/8" or 3/16". A smaller diameter endmill is more likely to break and the cut will take longer. The below fonts can be good options:

-**Comic Sans MS**
-**Arial Rounded MT Bold**
-**GOST Common**
-**Bauhaus93**
-**MV Boli**

It also helps to change the **Typeface to Bold**.

A **serif** font has small strokes added to some endpoints of letters. This type of font will have elements that are too thin to accommodate the endmill.

A **sans serif** font does not have these small strokes and thus is much more likely to accommodate the endmill.

Some fonts are rounded and have no sharp corners, such as the last two below. These allow the machining operation to best create the font.

JPB   JPB   Times & Garamond   (serif fonts)

JPB   JPB   Helvetica & Arial   (sans serif fonts)

JPB   JPB   Arial Rounded MT Bold & Comic Sans (sans serif fonts)

The Height will default to 0.394 in. Increase this value so the text is large enough to fit on the material with at least 0.1 in space between text and the material edges. 0.1 in is not critical. There just needs to be some space between the text and edges of the stock.

- click **OK** and **Finish Sketch**
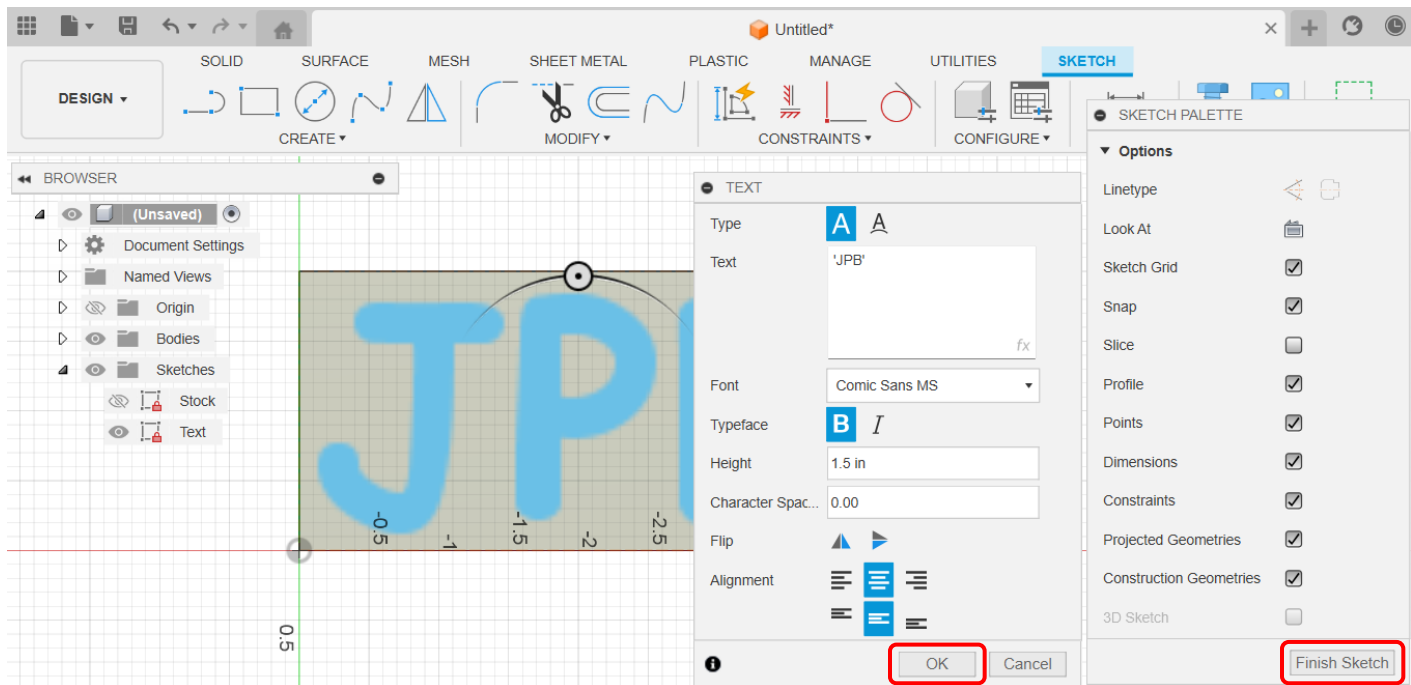
Extruding Text

- click on the **Home icon** at the **View Cube** and **Zoom** using the mouse wheel to achieve a view similar to that below
- click on a **letter** to select the text, which should make it a darker blue
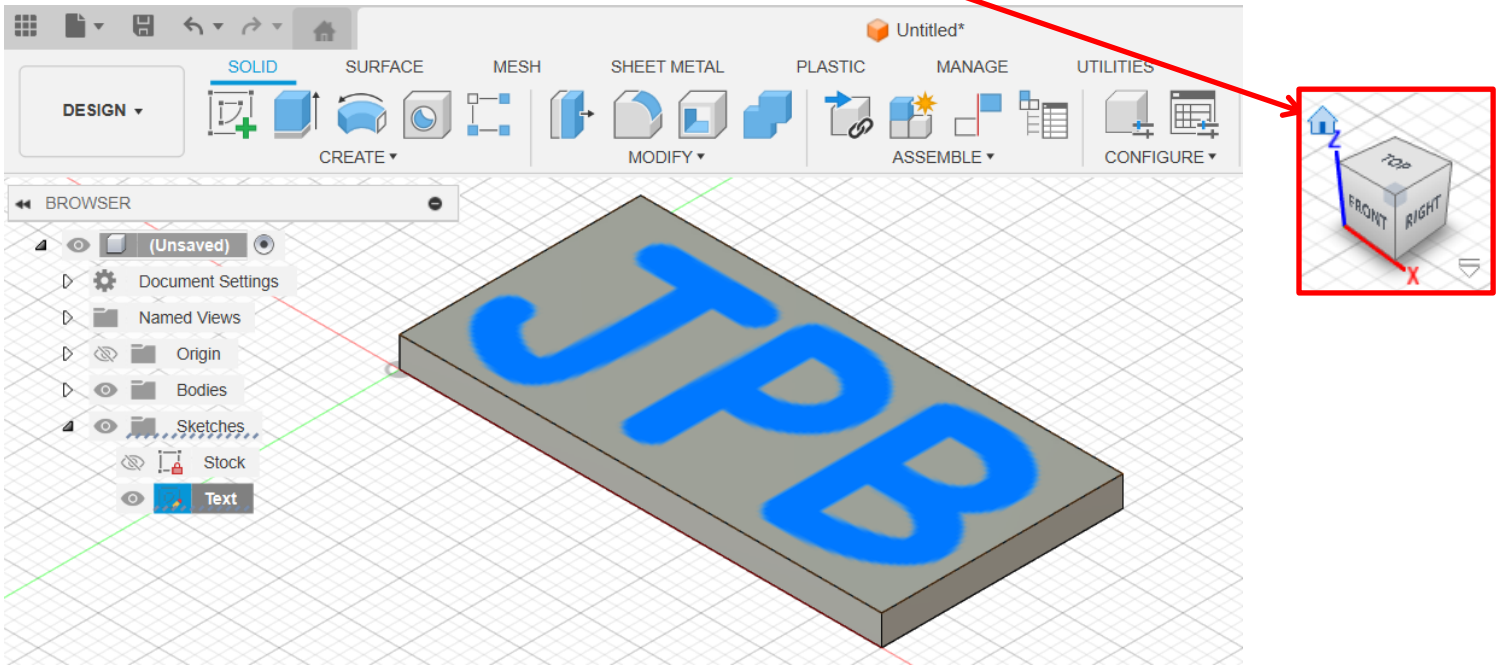
- select the **Extrude** tool, type **-0.125** (note the minus sign), ensure the Operation is **Cut**, and click **OK**

If the extrude fails, a new font may have to be selected. See Editing Text on the follow page to do so.
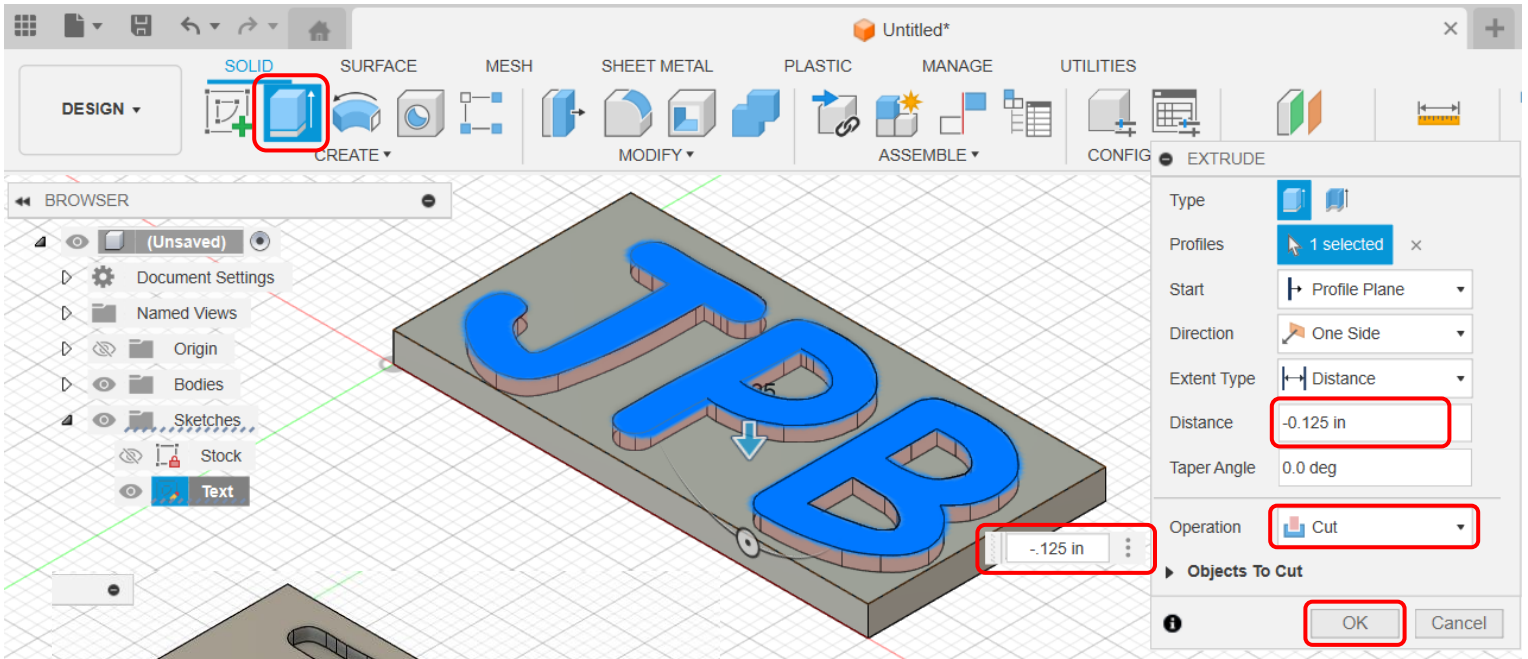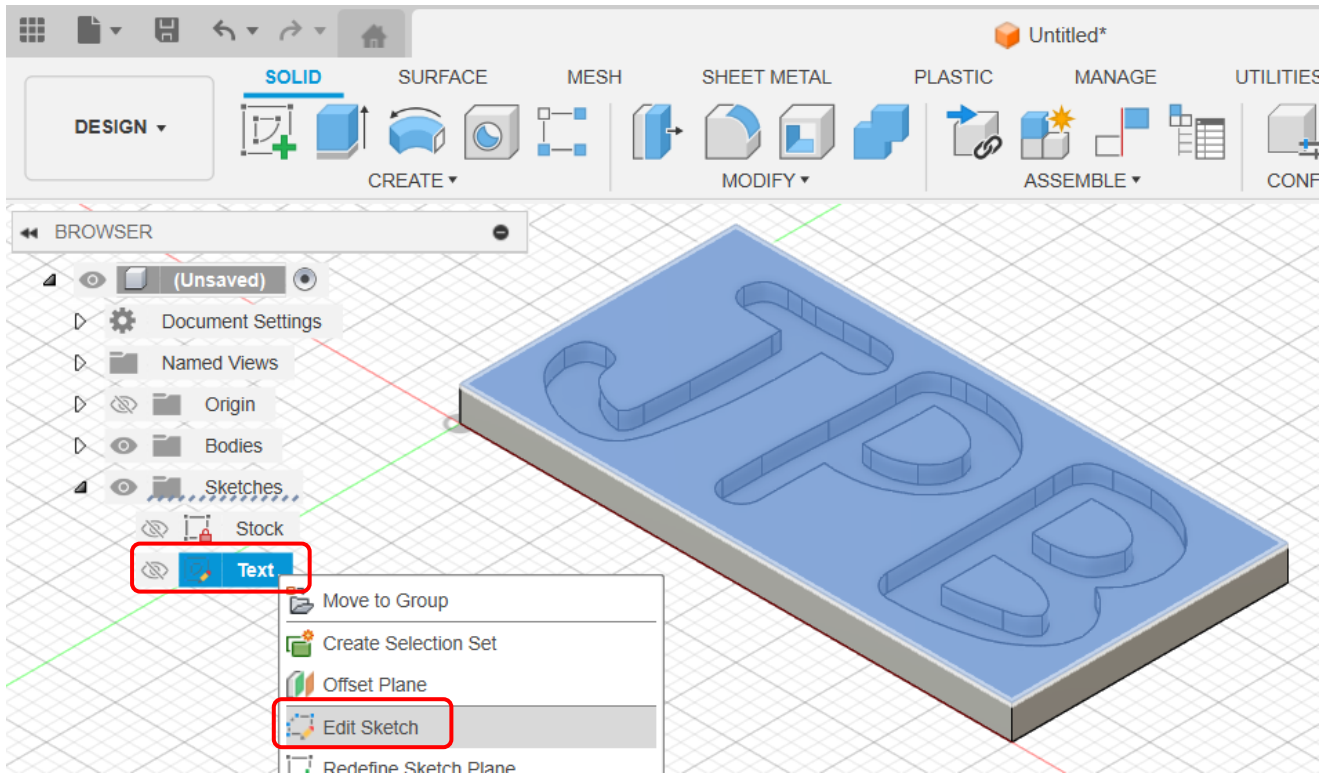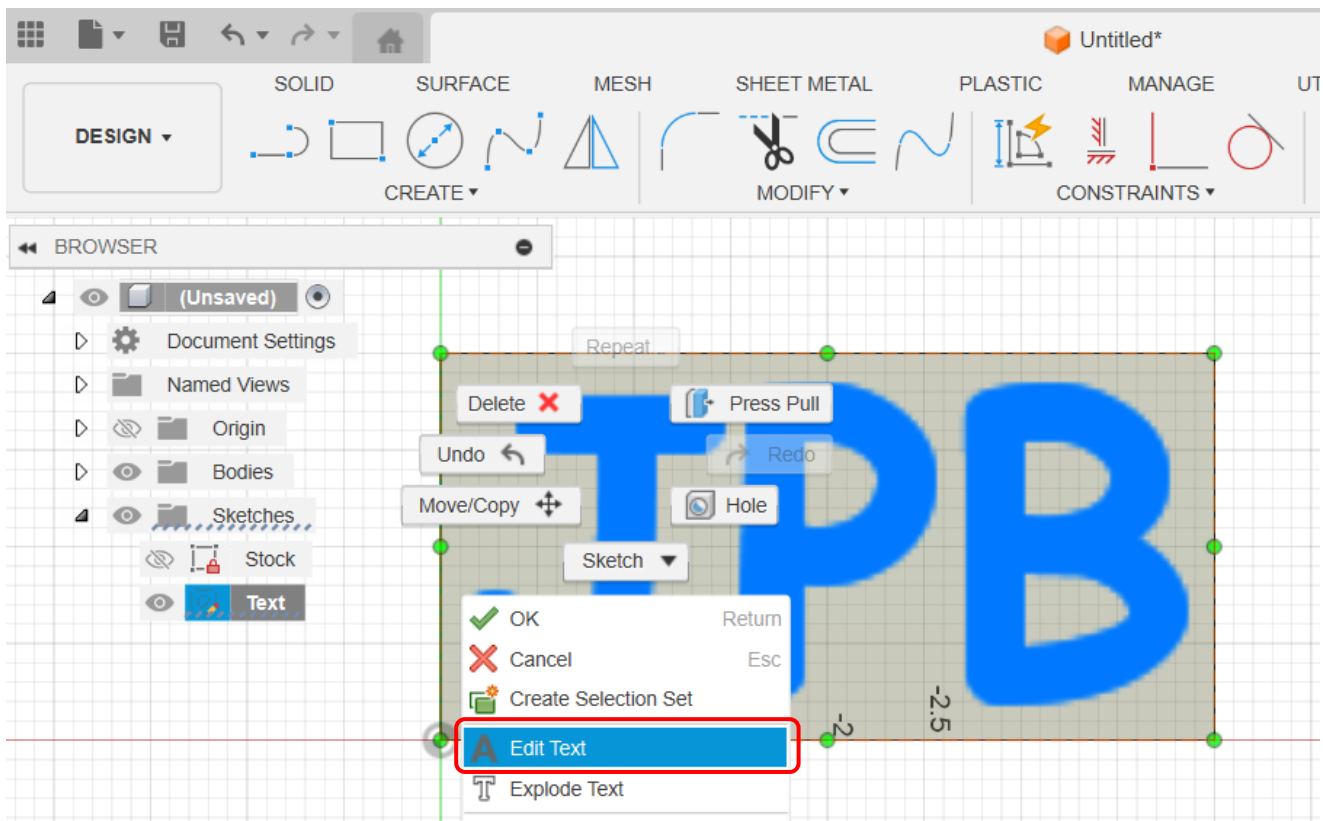

The result should look like this.

Editing Text

Some fonts may fail to extrude. If you need to change the font or any aspect of the text, you can follow these steps. Otherwise, continue to the next page.

- right click on the Sketch that was used to create the text and select **Edit Sketch**

- right click on the blue of a letter and select **Edit Text**, which will show the text settings

Generating Toolpaths and G-Code

In the past, one would use a different program to generate the G-Code for a design. This was referred to as CAM (Computer Aided Manufacturing) software to differentiate it from CAD (Computer Aided Design) software. This is still the case for some advanced CAM software. Fusion integrates CAM, which is called the **Manufacturing** worksapce.

Save your design with your name followed by an underscore and CNCNameplate, eg "JoeBarbetta_CNCNamplate".
You should always include your name in the project name. If your job ends up breaking the machine we will know who to blame.
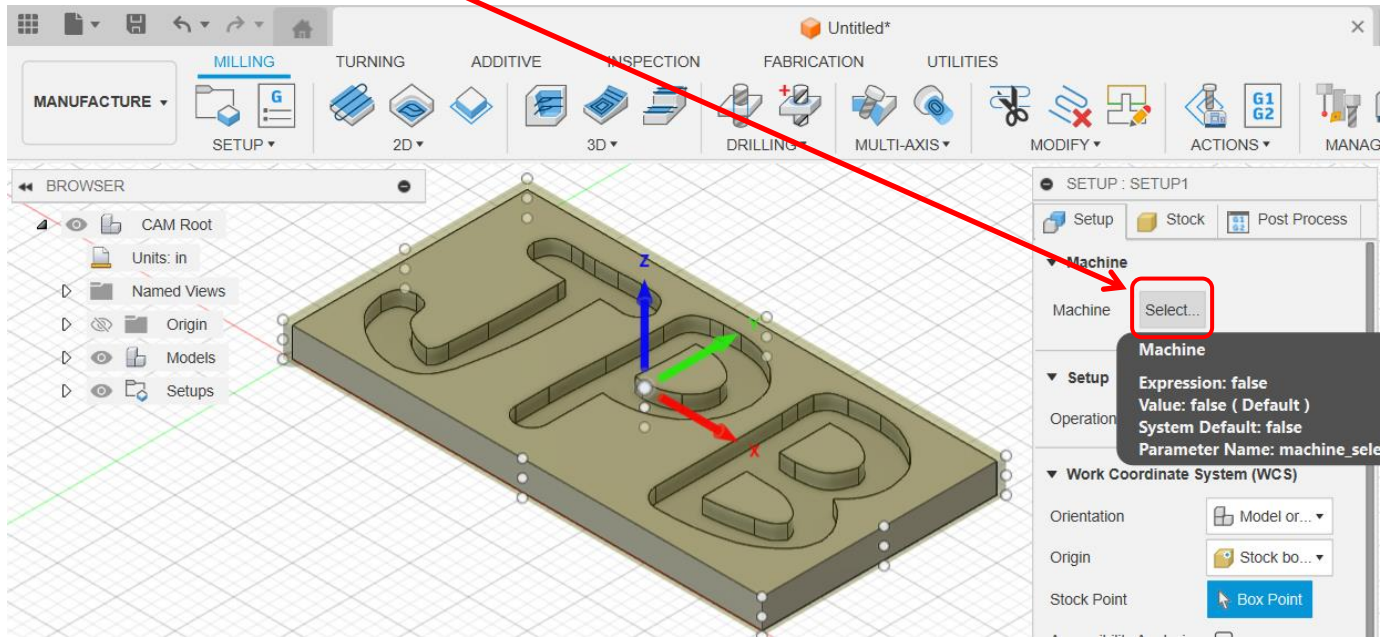
- click on the Worksapce button to swith from **DESIGN** to **MANUFACTURE**
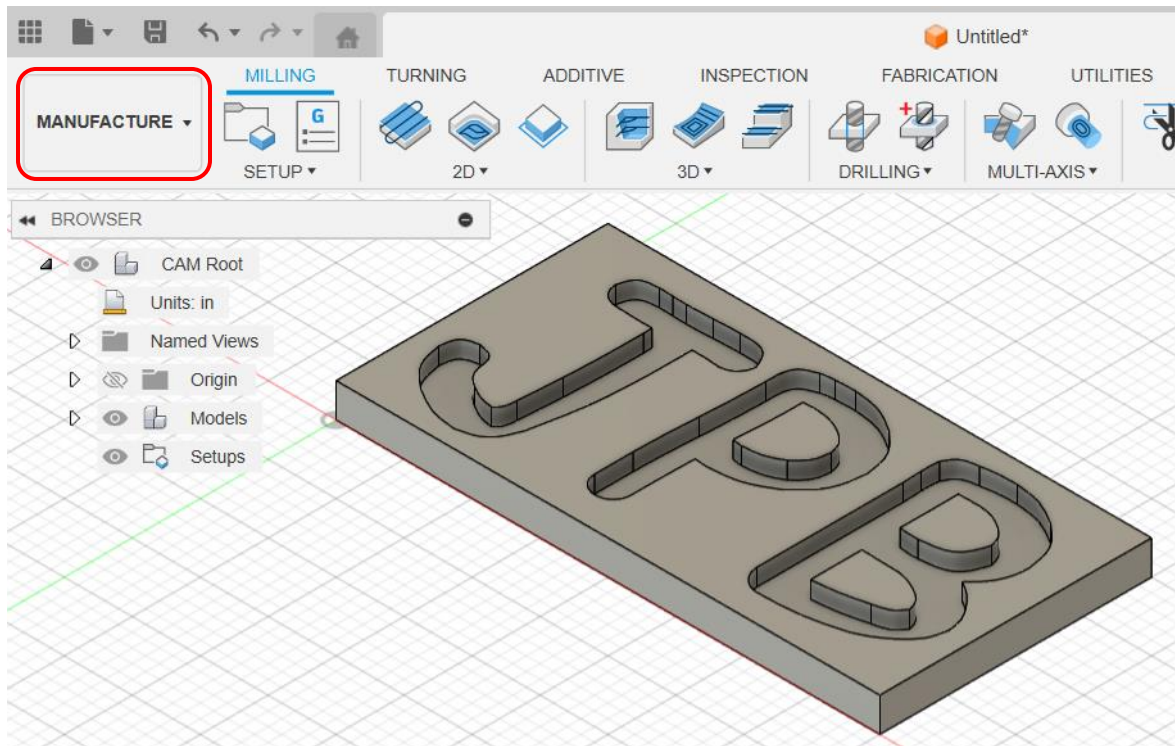
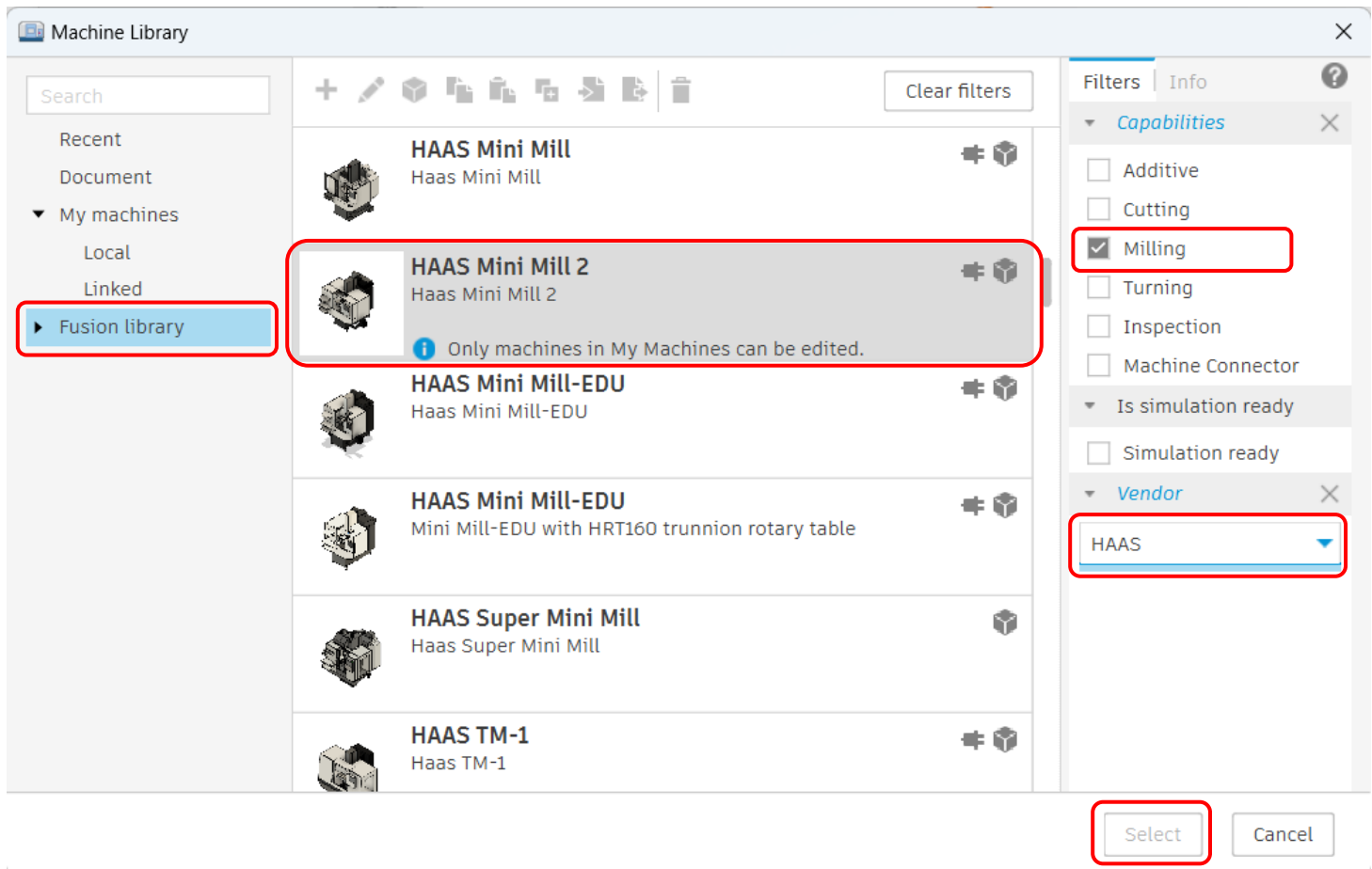- from the **SETUP** menu select **New Setup**
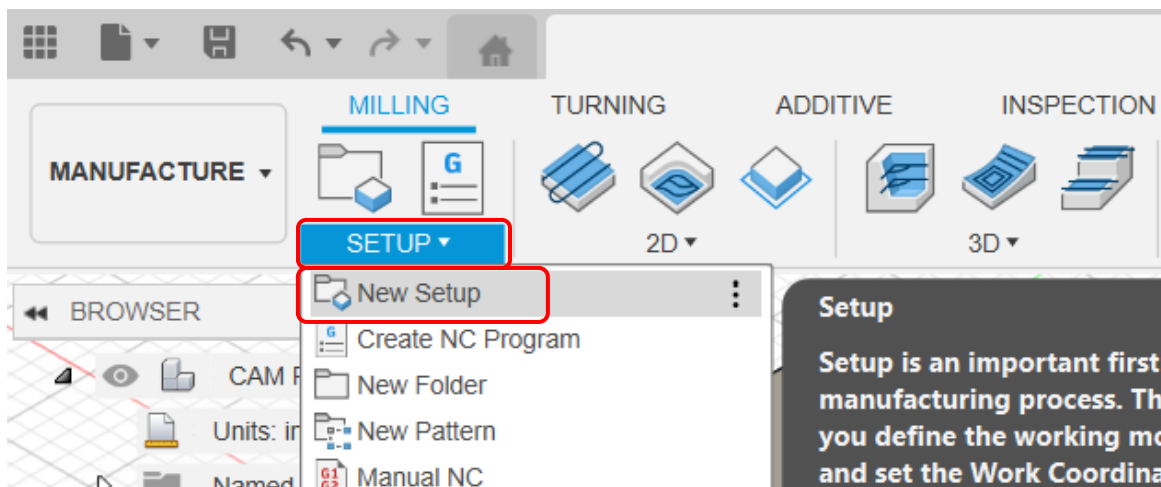
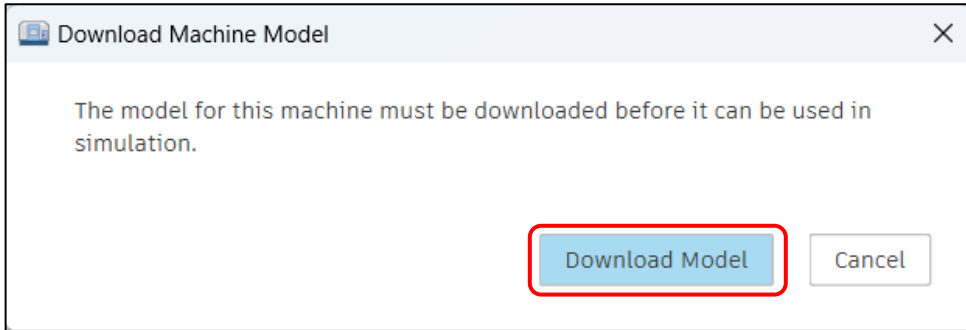- click on **Select** next to **Machine**

If this is the first time that you are using the MANUFACTURING Workspace on this computer, the next screen may say **"No Recent Machines Available"**.
- click on **Fusion Library**, ensure that **Milling** is checked, and select **HAAS** as the Vendor
- scroll to find **HAAS Mini Mill 2**, click on it, and then click on **Select**
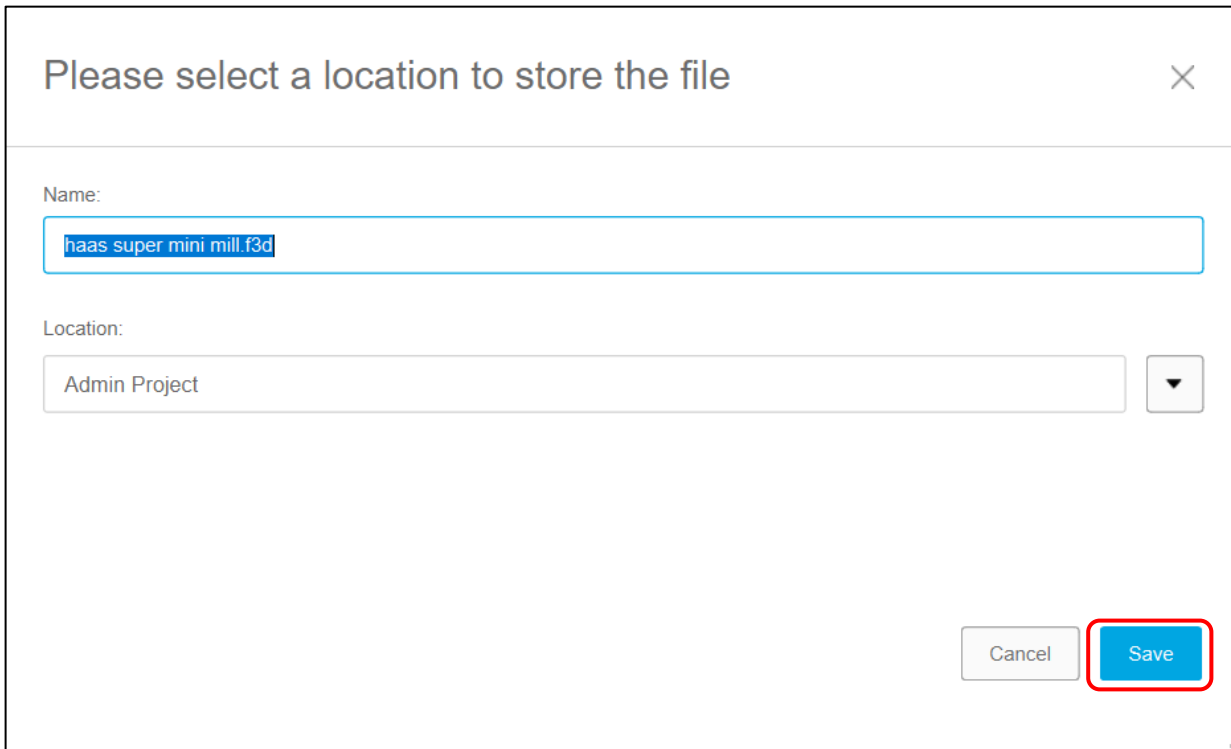
- click on **Download Model**
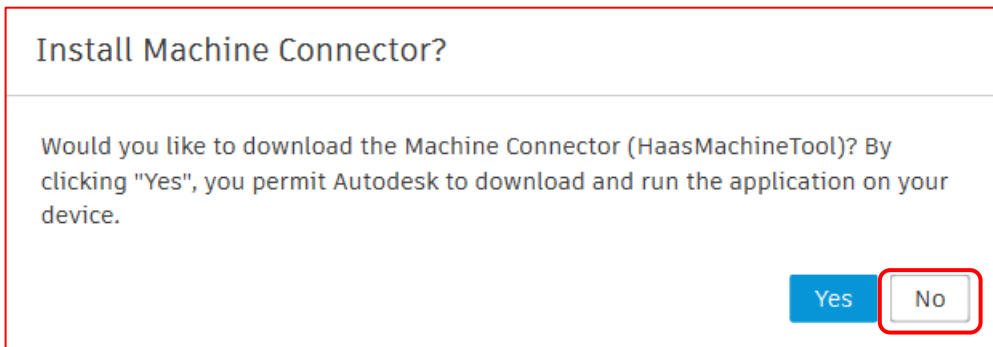

- keep the defaults and click on **Save**. The download process may take about 10 seconds.

If a small Upload complete message window pops up do not click Open Design. The X can be clicked to close it.

- if the below message shows, cick **No**.

After another delay period an error may show if the project has not been saved. If so save the project and redo the steps to select a machine.

Choosing a Datum point

The datum point is a location on a part that all cutting coordinates are referenced from.

Fusion sets a default in the center of the top surface. Common datum points are the the bottom left and top left corners. The top left corner is often used when the work is secured by a vise on the mill table. Because the rear jaw is fixed (top when looking down), using the top left corner makes sense.

With regard to the z-axis the top surface is common because the tool can be slowly brought down to the surface to "touch-off".

bottom-left

Top-left

Top-left
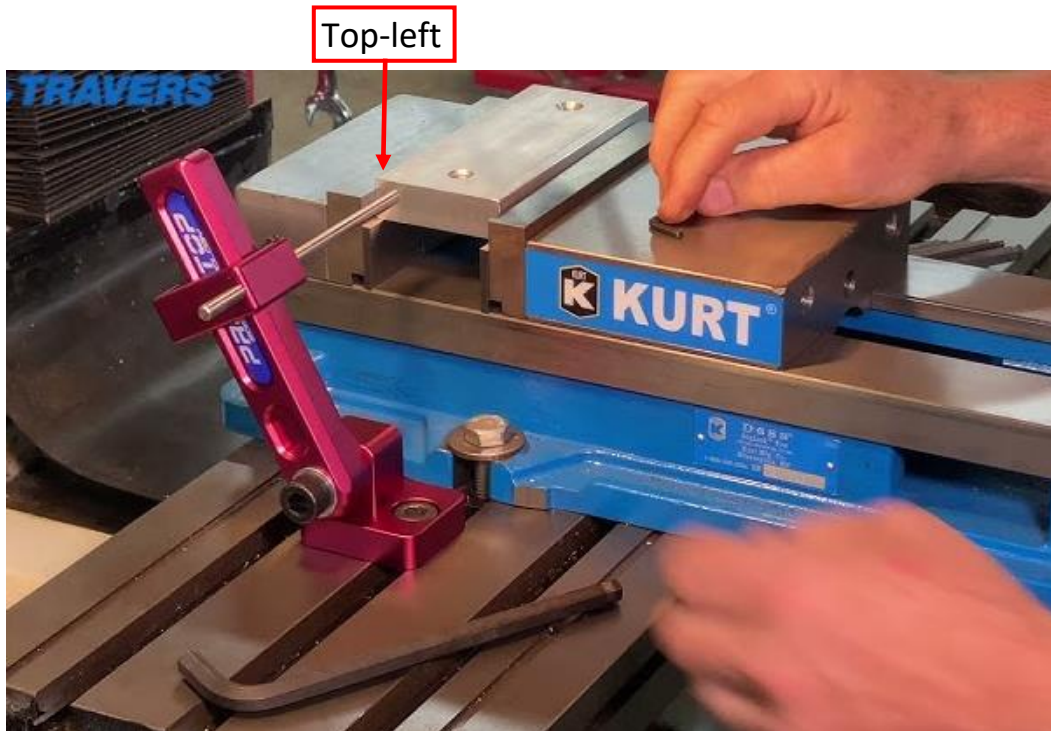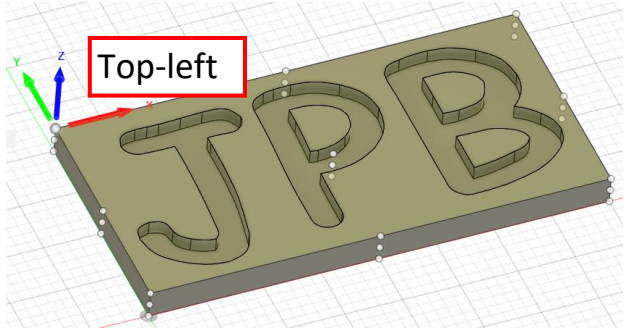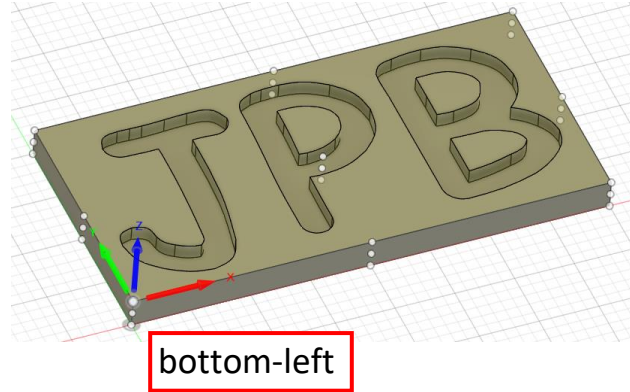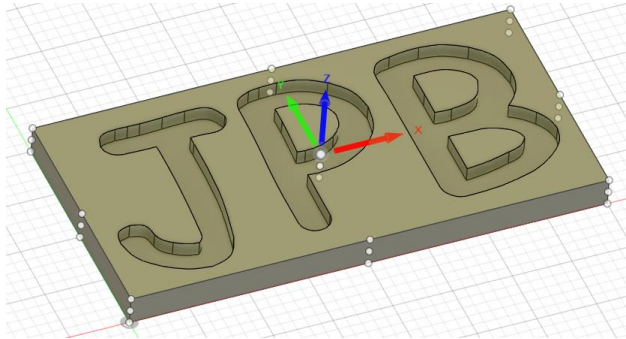
- click on the **top dot**
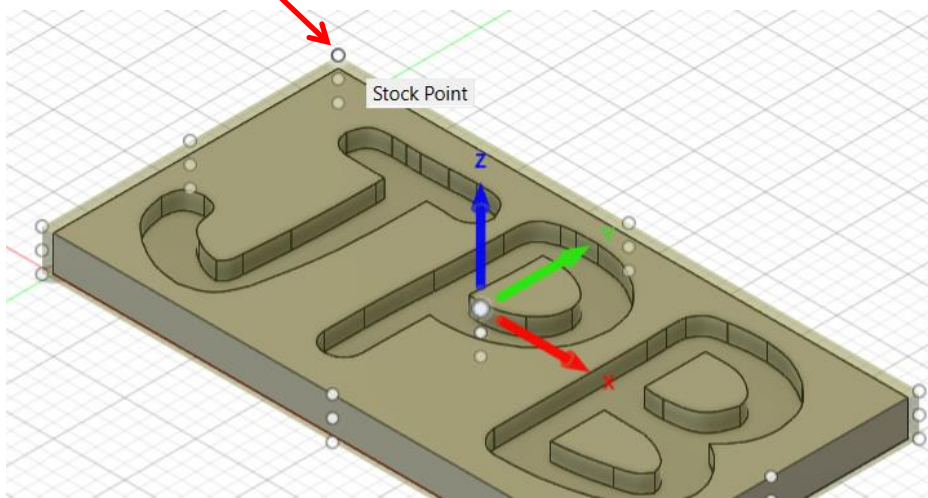
The axis arrows should now appear at that corner
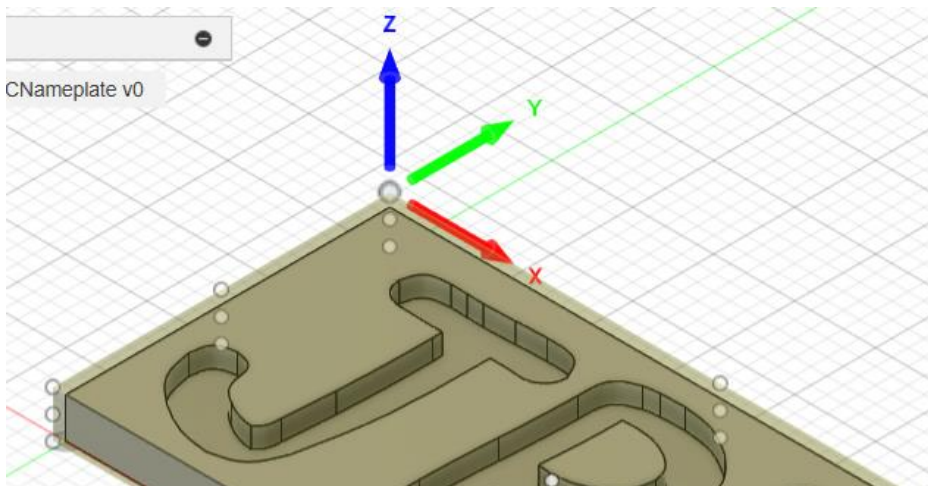
- select the **Stock** tab
- change Mode to **Relative size box** and Stock Offset Mode to **No additional stock**
- click **OK**
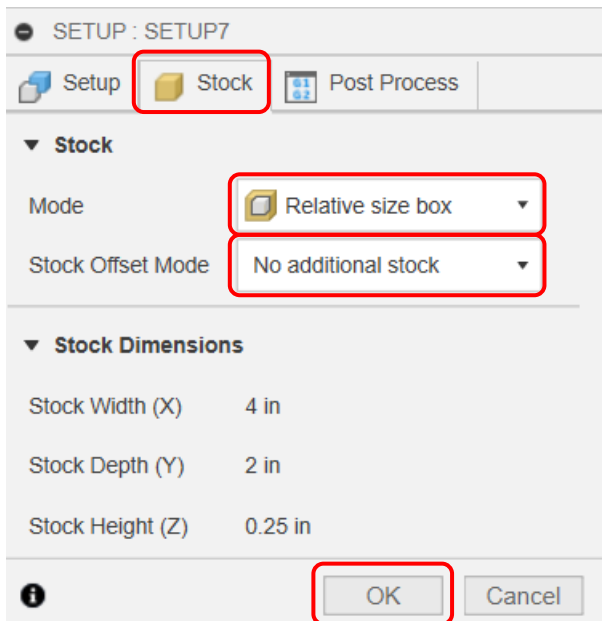
- use the mouse wheel to Zoom out
- yell **"Dude, where did this come from!"**
- click on the **eye icon** to hide the mill and Zoom in to your part
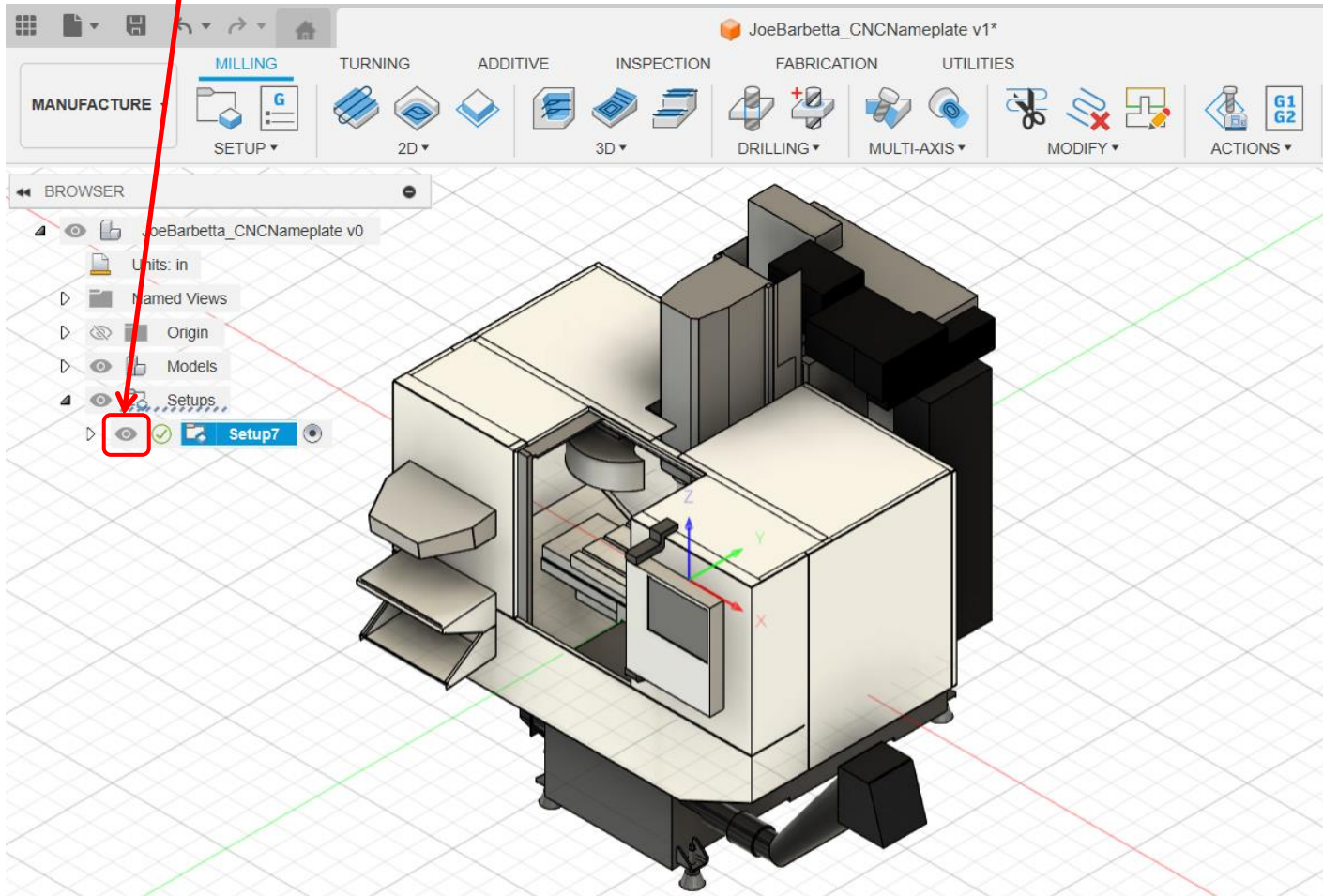
Your part should look like this now
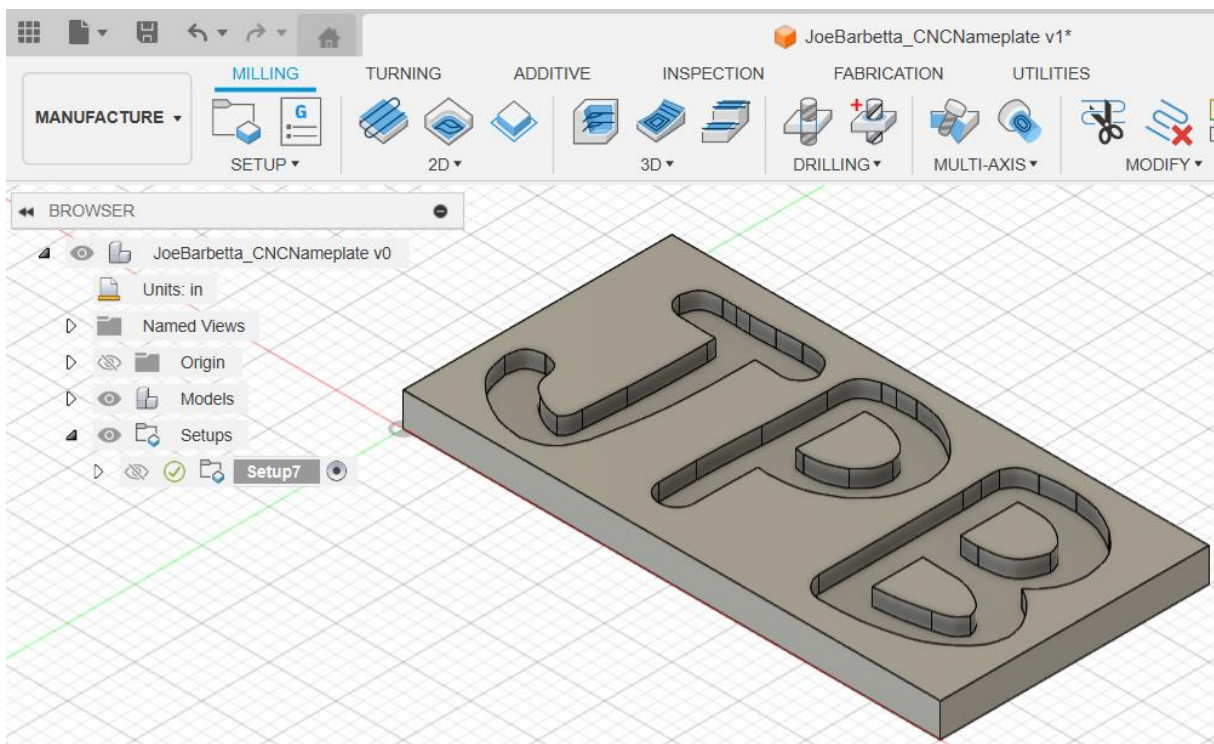
Setting up the Pocket operation

The Pocket operation will cut the letters into the stock. The endmill will first move over the stock to the position of a letter, plunge down into the stock, cut sideaways through the material, then return back up above the stock. It will then continue this for the other letters.

- from the **2D** menu select **2D Pocket**
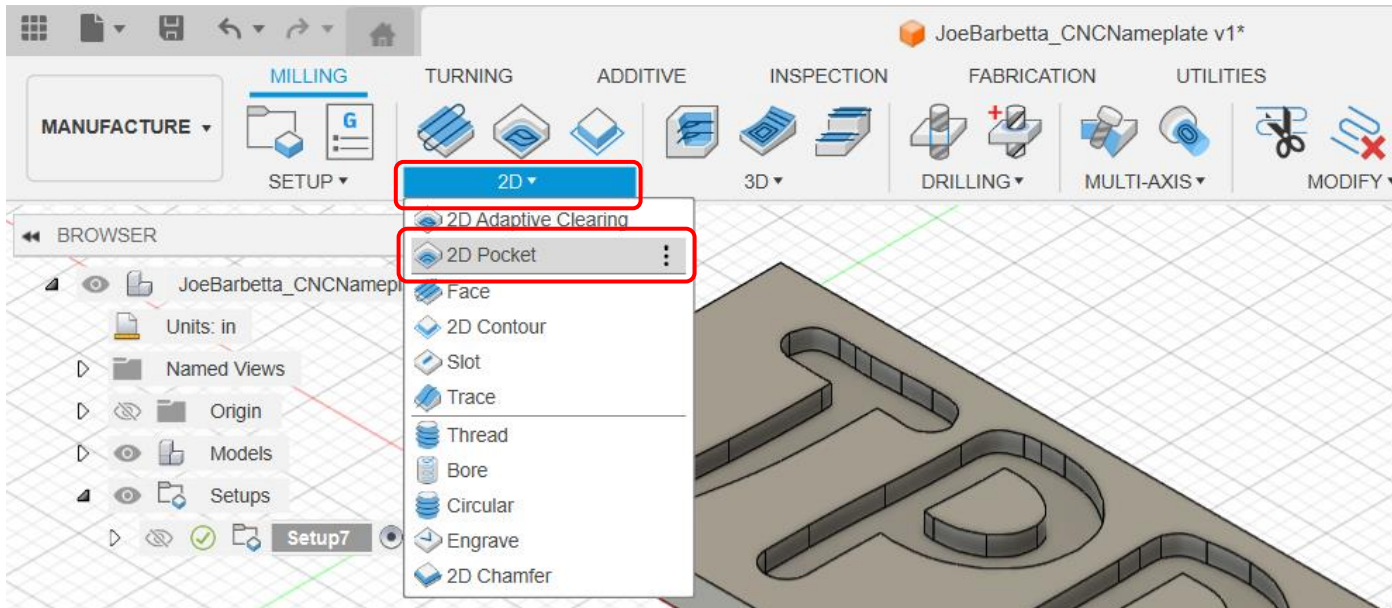
- click on the **Tool** tab and then **Select...**
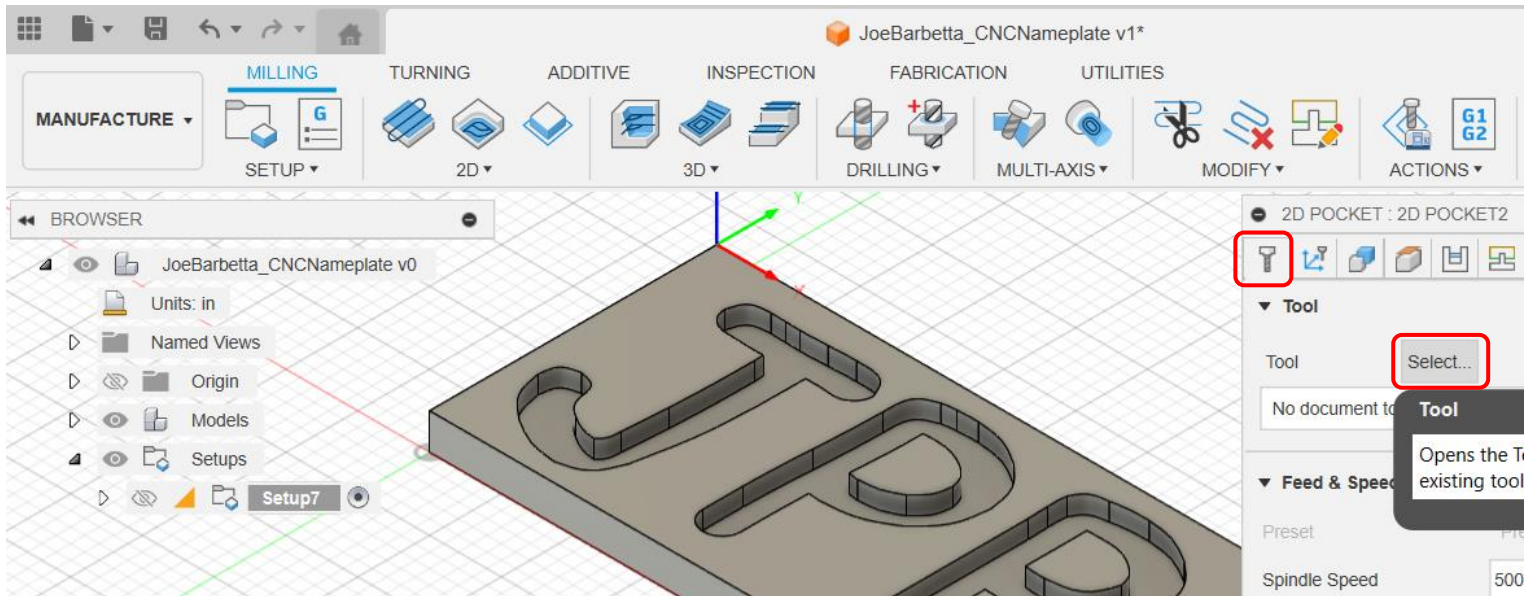
The tool typically used for milling is an end mill. It's flutes make it resemble a drill bit with a flat end and the flutes are formed so that it can cut sideways. A drill bit can only drill down into material, but it can't cut sideways. Most endmills are flat (no point at its end), but some do have a point, ball end, or other styles.

- select **Milling Tools (Inch)**
- check **Flat end mill**
- select **1/8" (1/8" Flat Endmill)**
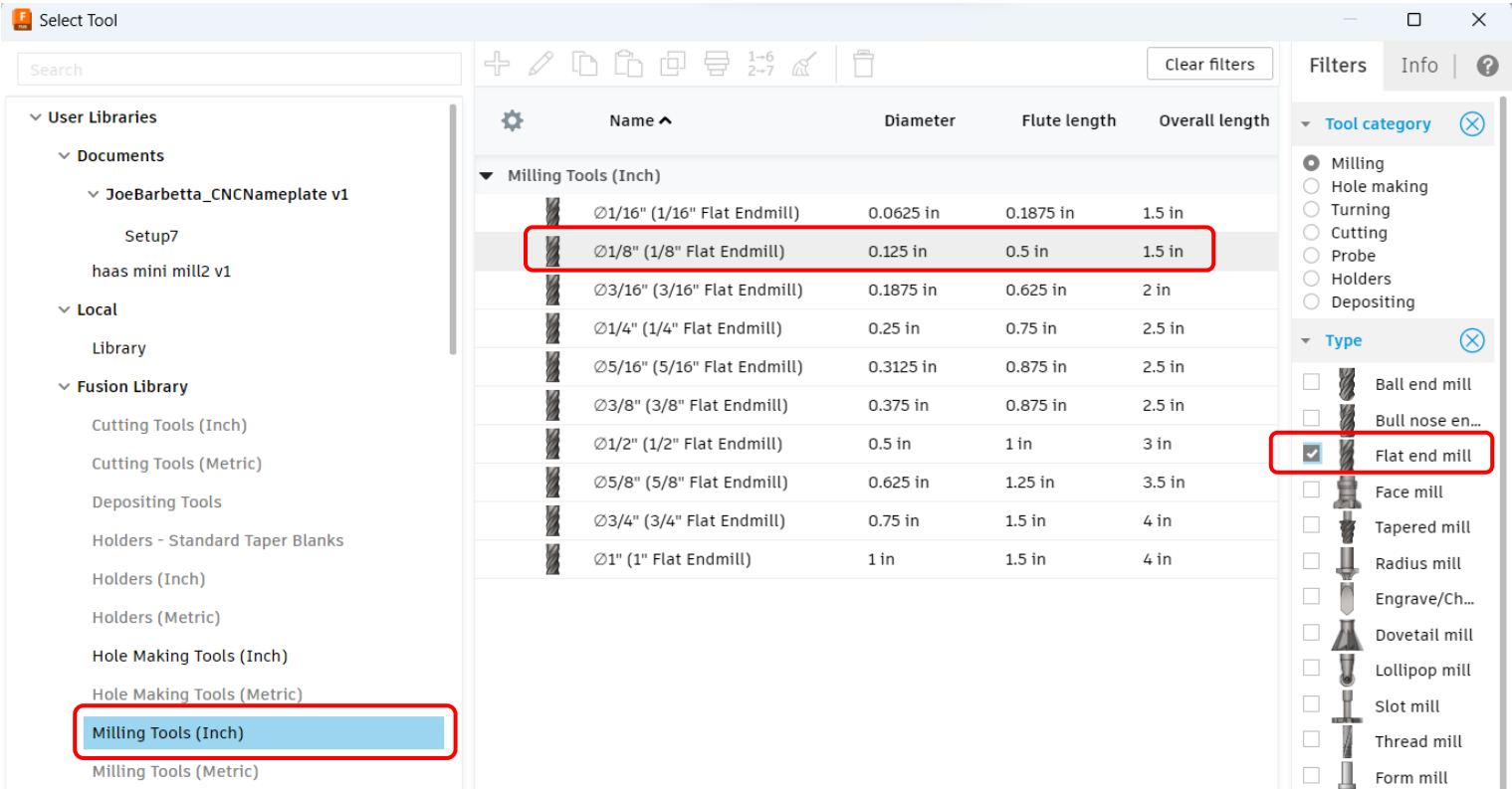
- at the bottom scroll and click on the row for **Titanium – Slotting** and then click **Select**

Are we actually cutting a titanium name plate? See next page.
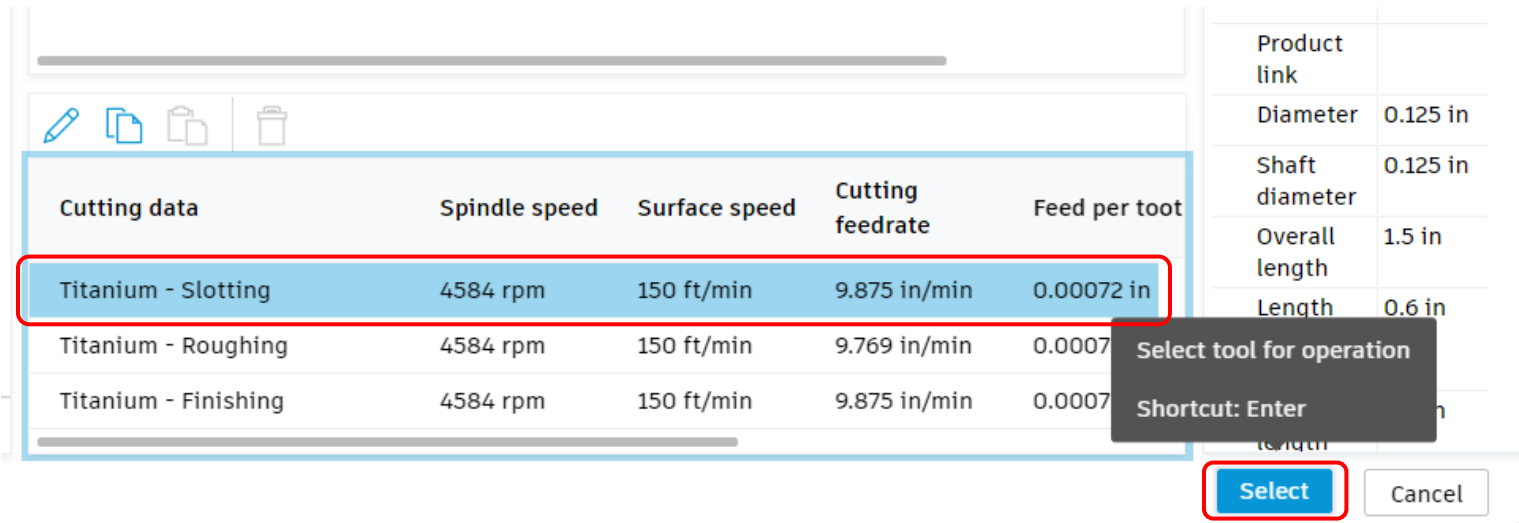
Feeds and Speeds

It would be nice to have titanium to cut, but we will just be cutting a common plastic named **polycarbonate**, also known by the trade name Lexan.

There are guidelines on milling and drilling for various materials (metals and plastics), which specify the optimum speed at which to cut a material. A value for a material may be specified as **surface feet per minute**, which can then be translated to **Speed** in **RPM (Revolutions Per Minute)** and a **Feed** in **feet per minute** or **inches per minute** for a particular diameter tool.

One will find that for milling polycarbonate with a 1/8" endmill RPMs in excess of 10,000 are recommended. Fusion has a *plastic* cutting option that specifies a **Speed of 12,000 RPM**. The "SS" series of Haas mills have a top speed of 12,000 RPM and thus can achieve this speed. Our **"microwave oven" MiniMill 2 has a top speed of only 6,000 RPM**. Because the plastic option as well as all the metals have cutting speeds of greater than 6,000, were are selecting Titanium for its slower speed, which will be embedded in the G-Code file.

- click on the **Geometry** tab
- click on the **bottom surface** of each letter and then click **OK**
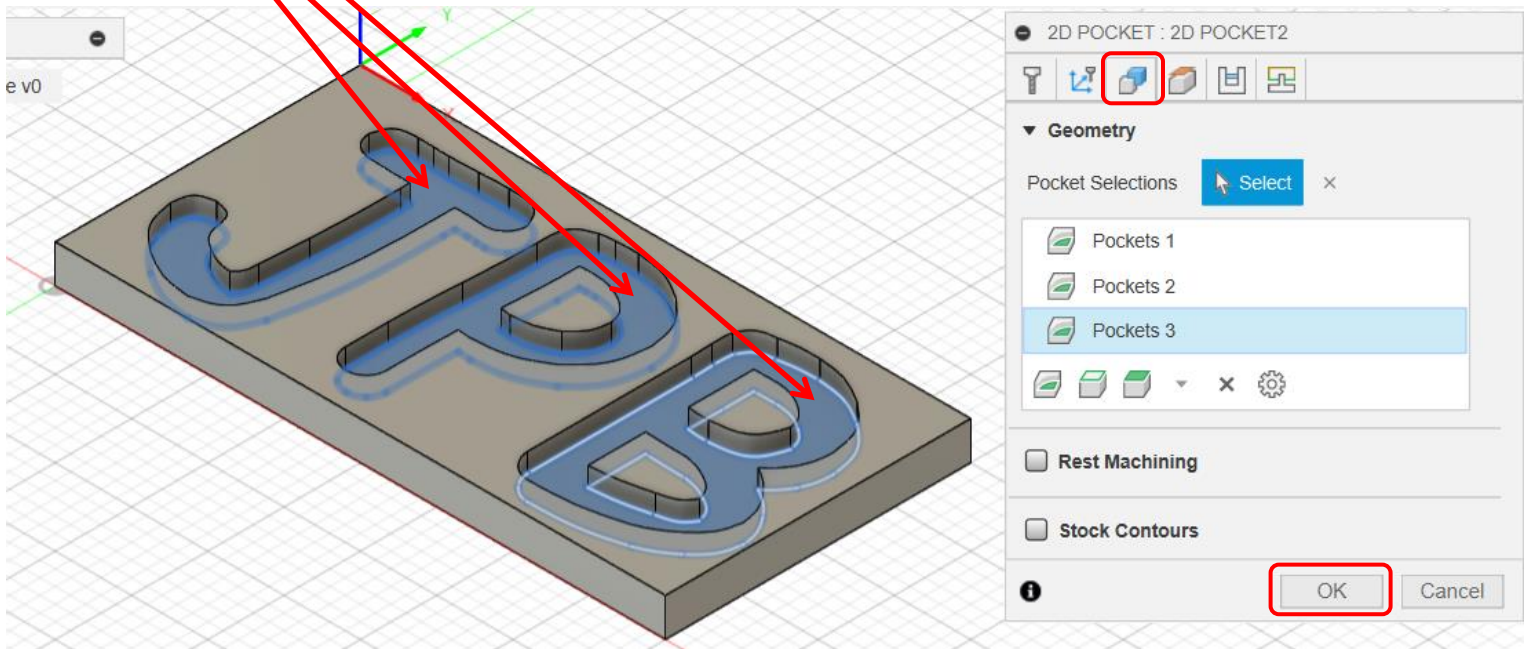
You **do not** want to click on the letter edges as shown.
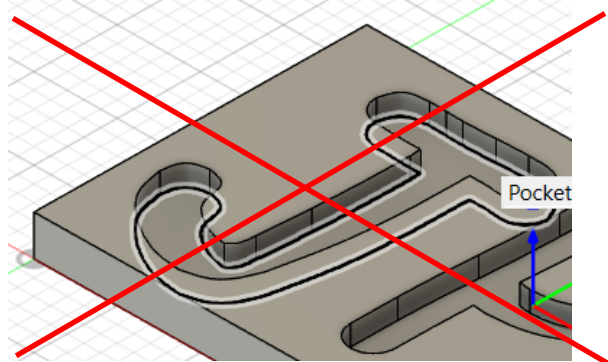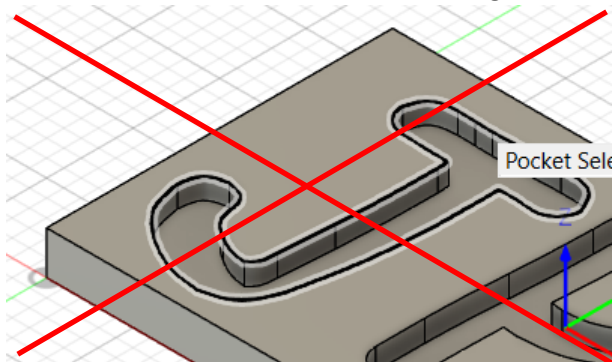
The screen should look like that below. The 2D Pocket operation can always be clicked on later to return to this view.

The yellow line show the fast (non-cutting) movements, the red helixes show the downward cutting, and the blue lines show the "sideward" cutting.
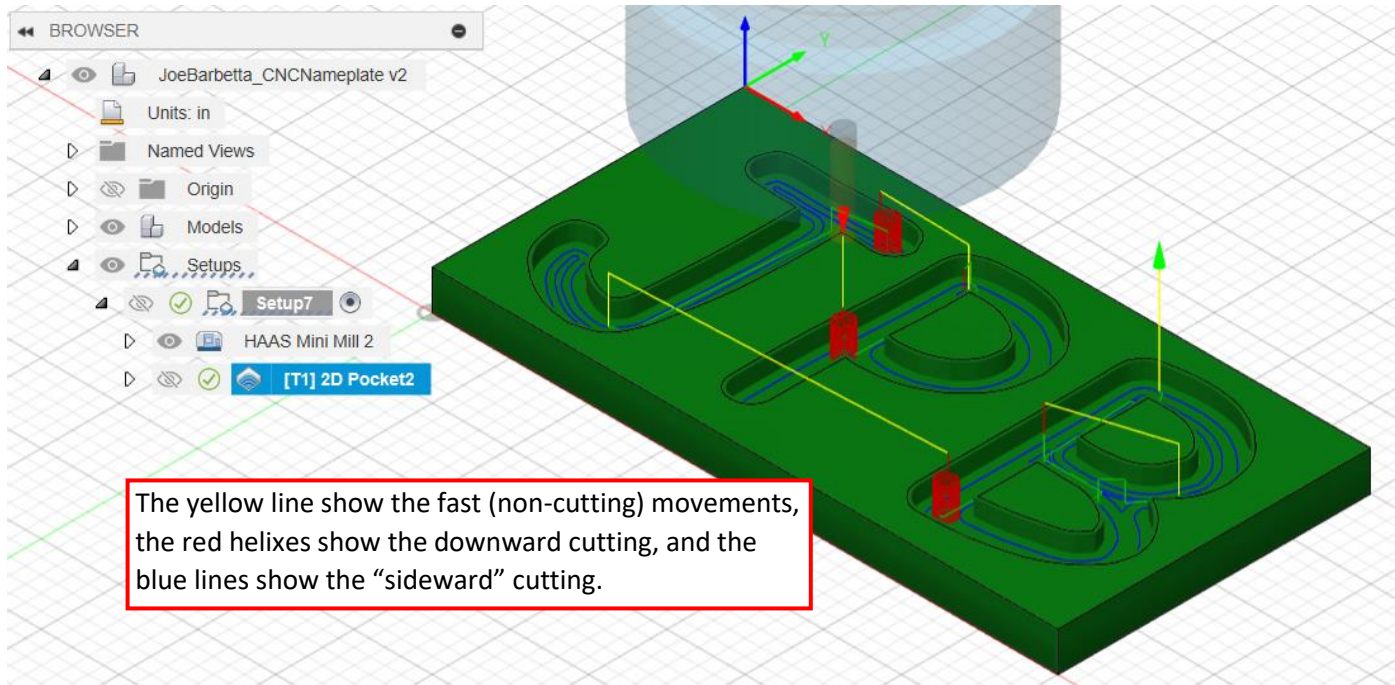
Note that there should be **blue lines inside each letter segment**.
If there is a letter segment without blue lines, that may indicate that a letter segment is not wide enough to be cut and a new font may need to be selected or the present text made larger.

Switch from the **MANUFACTURE** Workspace back to the **DESIGN** Workspace and see the earlier section "Editing Text".
Then switch back the the **MANUFACTURE**. Note the orange icons in a few places. This indicates that the tool paths need to be updated. Right-click on **Setup** and select **Generate**. This should restore the green checkmark icons.

If the NCProgram was created already, click on it and select Edit and then Post at the bottom right of the screen that opens.
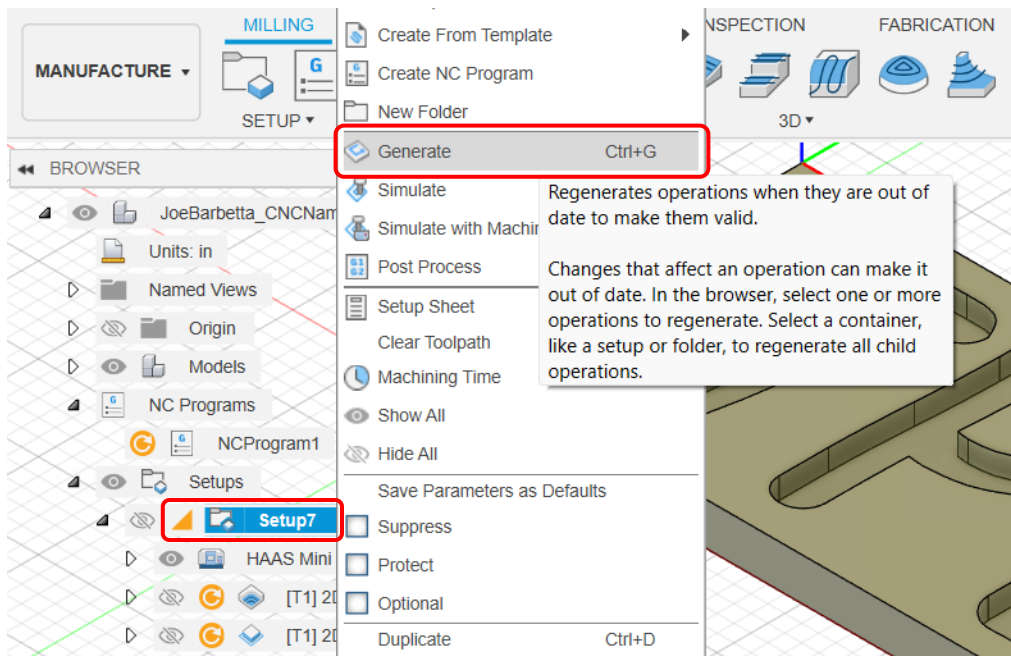
Setting up the Contour operation

The Contour operation will cut the nameplate out of the stock material.

If we had pieces precut to 4" x 2" would could secure them in a fixture or vise and just mill the letters out using the Pocket operation. However, by adding a 2nd contour operation we can use rough cut material or scrap and the contour operation will cut out or part with perfect dimensions and a good surface finish around the edges of the part.

But doesn't this method waste material?

From a business perspective, **reduced labor is often a much greater concern than material**. The extra work needed for an additional operation to prepare the stock will make the part more costly.

Note that one could also load up multiple nameplates for the job and cut them all out of one piece.
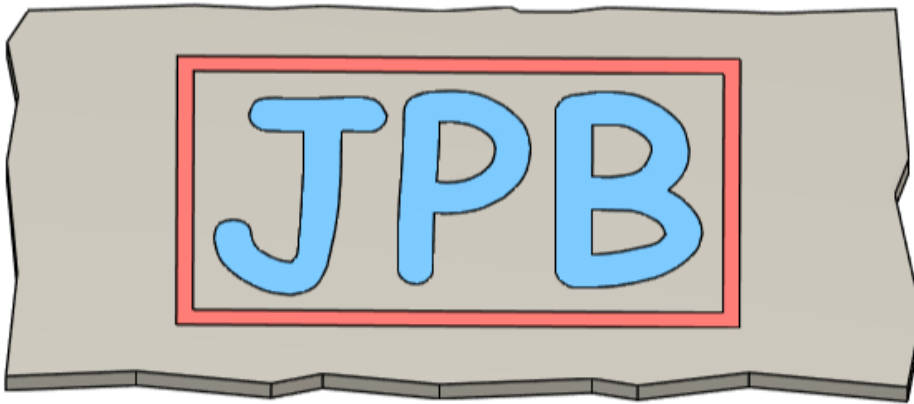
The "roughness" of the outer edge of the material is exagerated here to illsutrate the point.

The blue regions represents the material cut for the letters and the red is the material cut with the 1/8" end mill.

- from the **2D** menu select **2D Contour**
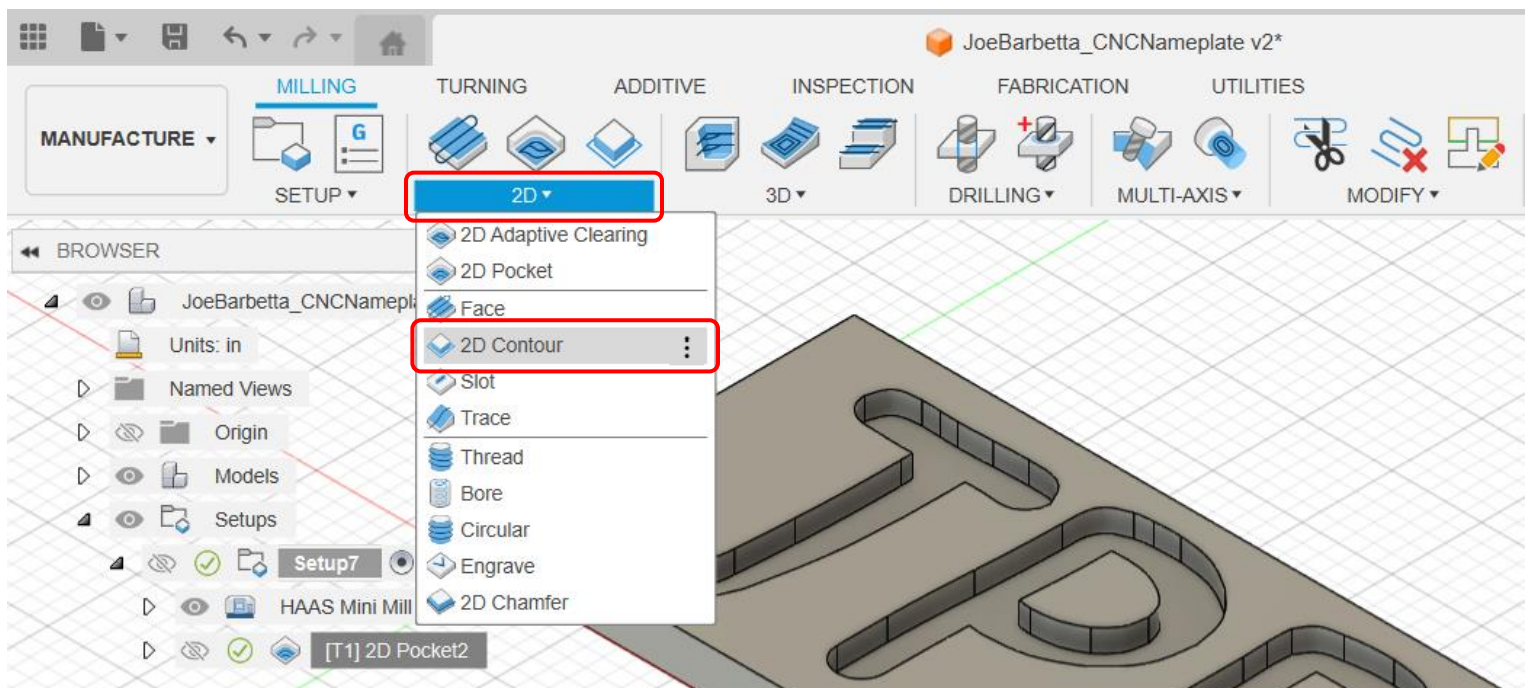
- select the **Geometry** tab
- click on any **top outer edge**

Note that the red arrow is on the outside of the stock. If it is on the inside, click on it to move it to the outside.

2D CONTOUR : 2D CONTOUR1

▼ **Geometry**

Contour Selection   ▸ Select   ✕

📦 Closed Chain 1

▢ **Tabs**

▢ **Rest Machining**

▢ **Stock Contours**

OK   Cancel

- select the **Heights** tab
- in the lower section **Bottom Height**, change From to **Stock bottom** and the Offset to **-0.05** (note the minus sign)
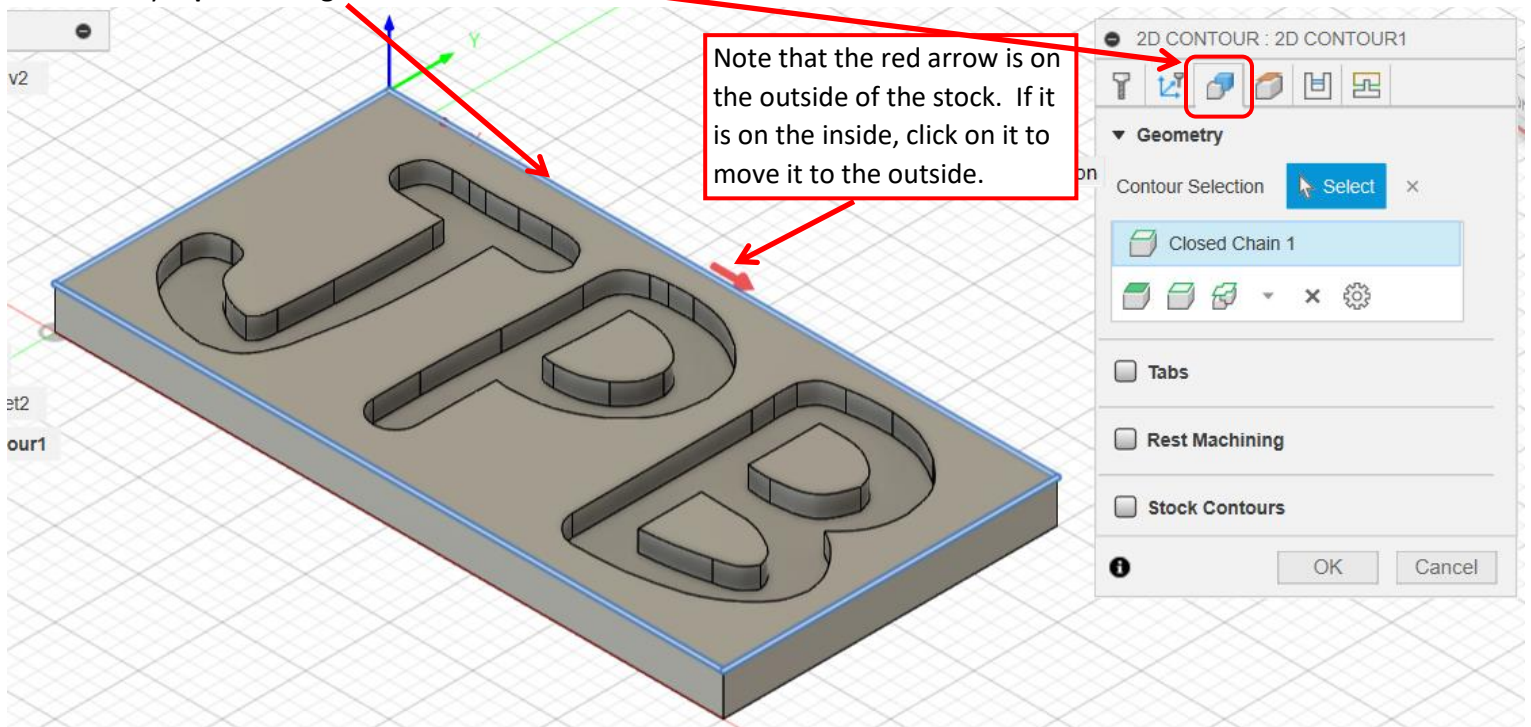The -0.05 value is chosen to ensure the final cut is all the way through the stock to free the part.

JoeBarbetta_CNCNameplate v2*

TURNING   ADDITIVE   INSPECTION   FABRICATION   UTILITIES

2D ▾   3D ▾   DRILLING ▾ MULTI-AXIS ▾ MODIFY ▾

2D CONTOUR : 2D CONTOUR1

▼ **Clearance Height**

From   ▭ Retract height   ▾

Offset   0.4 in

▼ **Retract Height**

From   📦 Stock top   ▾

Offset   0.2 in

▼ **Feed Height**

From   ▯ Top height   ▾

Offset   0.2 in

▼ **Top Height**

From   📦 Stock top   ▾

Offset   0 in

▼ **Bottom Height**

From   📦 Stock bottom   ▾

Offset   -.05 in

OK   Cancel

- click on the **FRONT** of the **View Cube** and the view should look similar to that below.
The blue line should be slightly below the stock indicating that it will be cut fully through.
- click **OK**
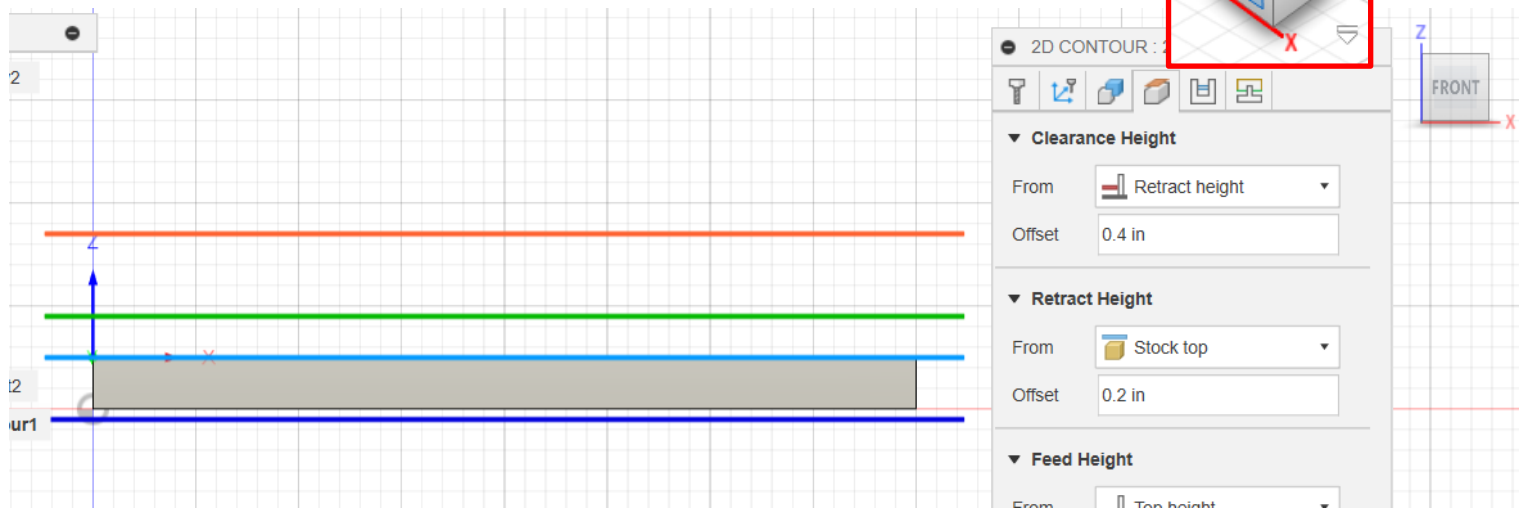
- click on the **Home icon** at the **View Cube** and the view should look similar to that below.
A blue line should surround the bottom of the stock.
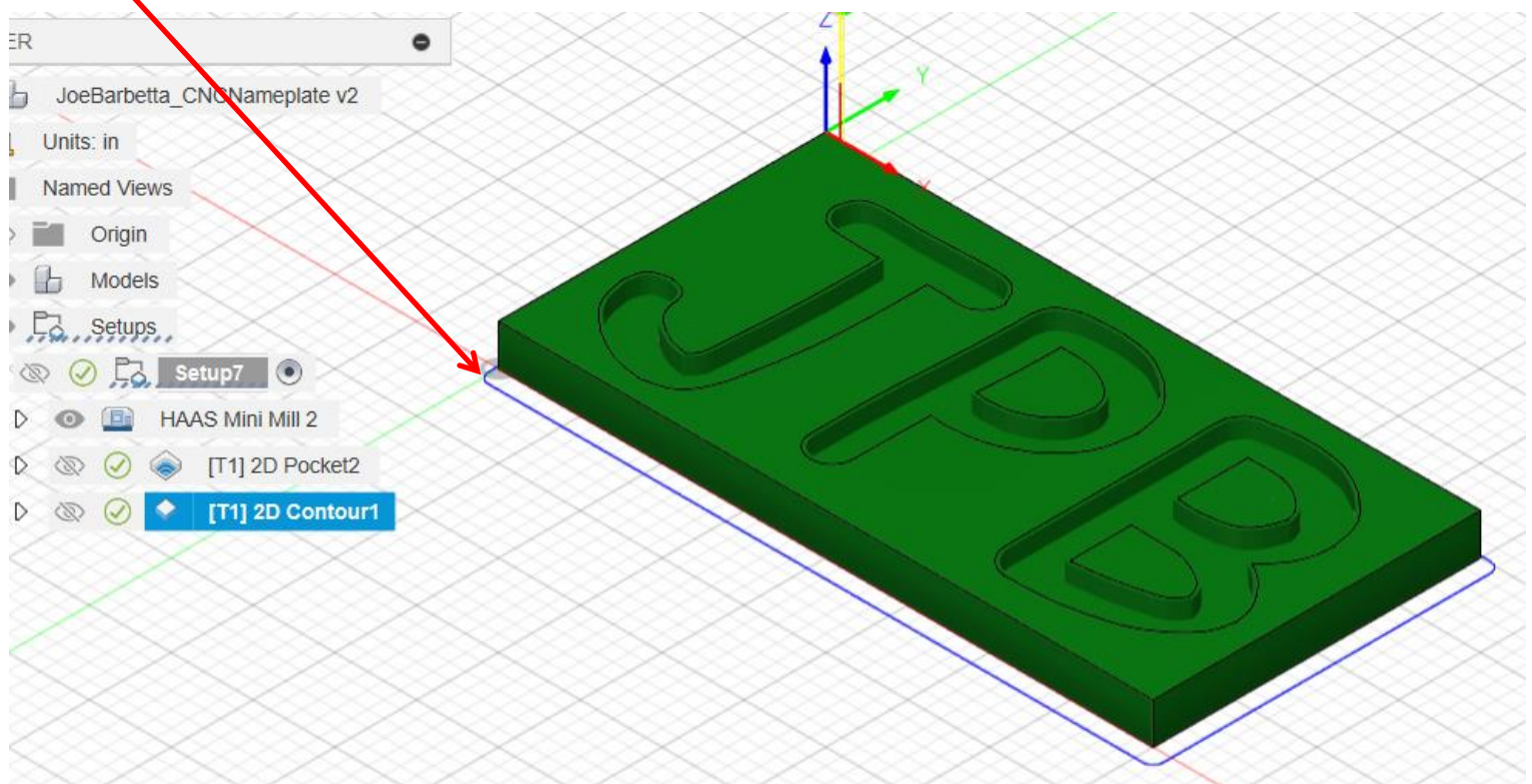
If any changes are needed later on, know that you can **right-click** on either the 2D Pocket or 2D Contour operation ans select **Edit** at the top of the menu.
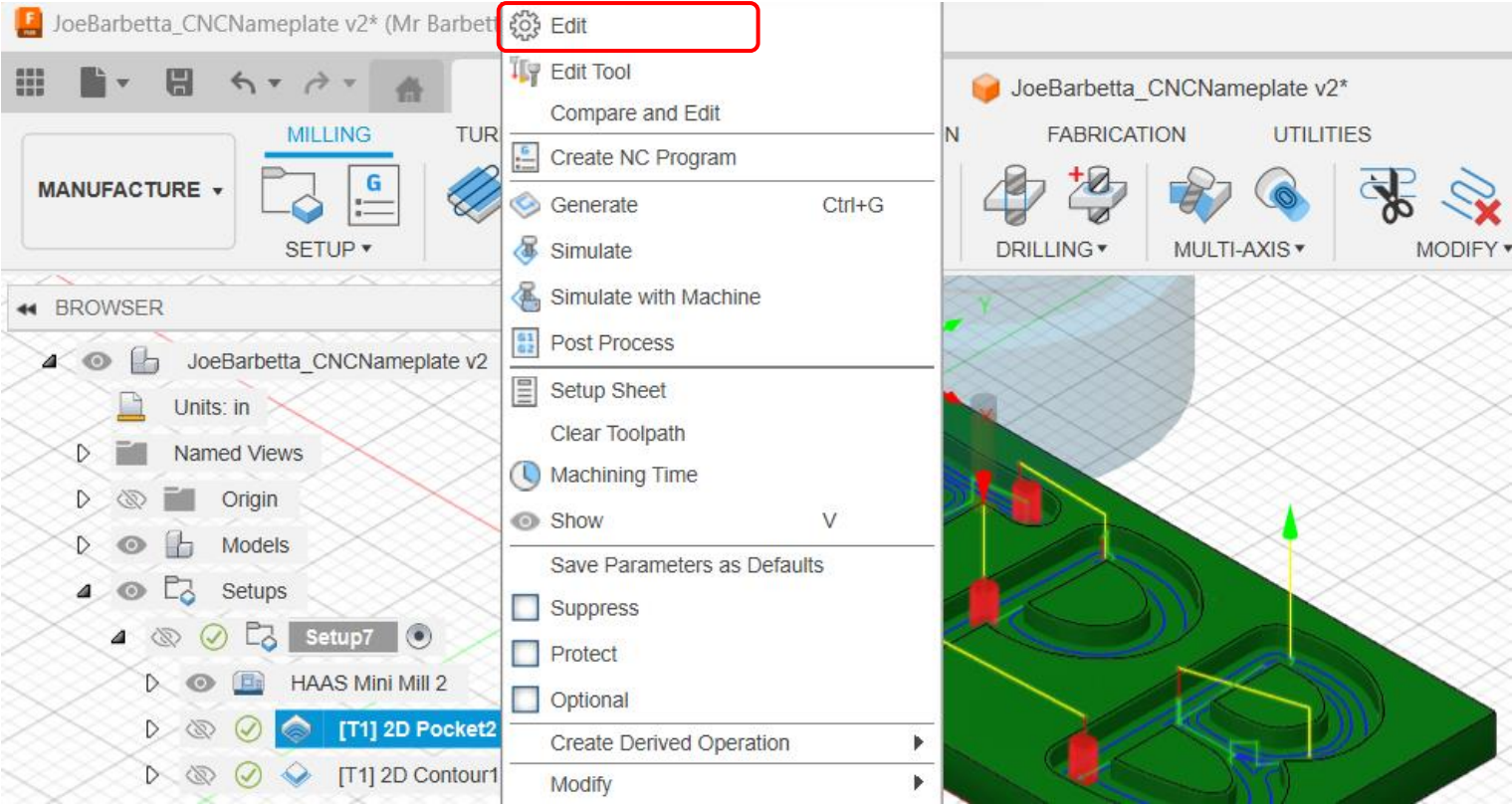
Running a simulation

- right click on **Setup** (note that we don't care about the number after it) and select **Simulate**
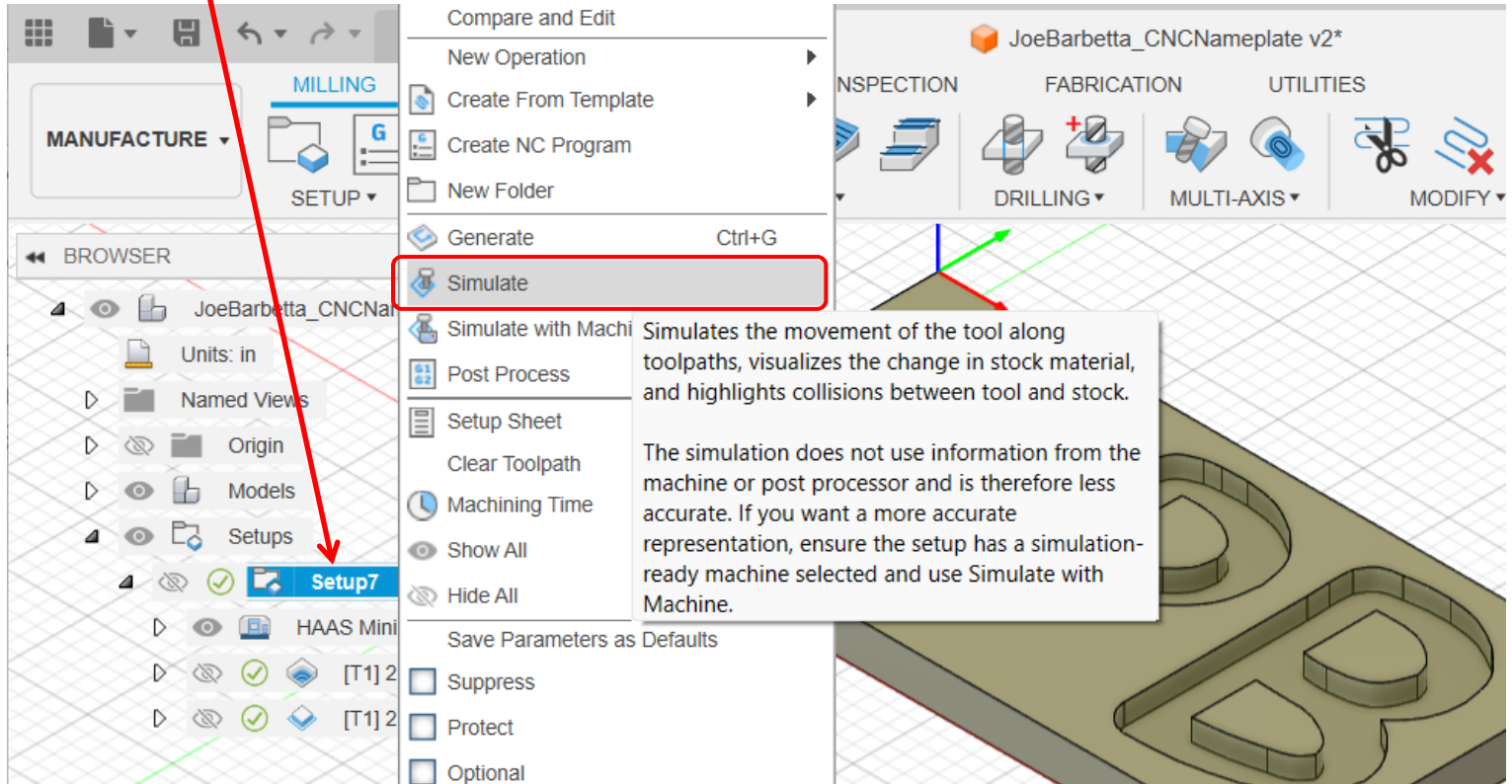
- click on the **Play** icon at the bottom of the screen
- the **speed slider** can be moved left or right to slow down or speed up the simulation
- click on **Exit Simulation** when you are done marveling at the simulation

The simulation should show the cutting of the letters **before** the outside cutting.

Generating G-Code
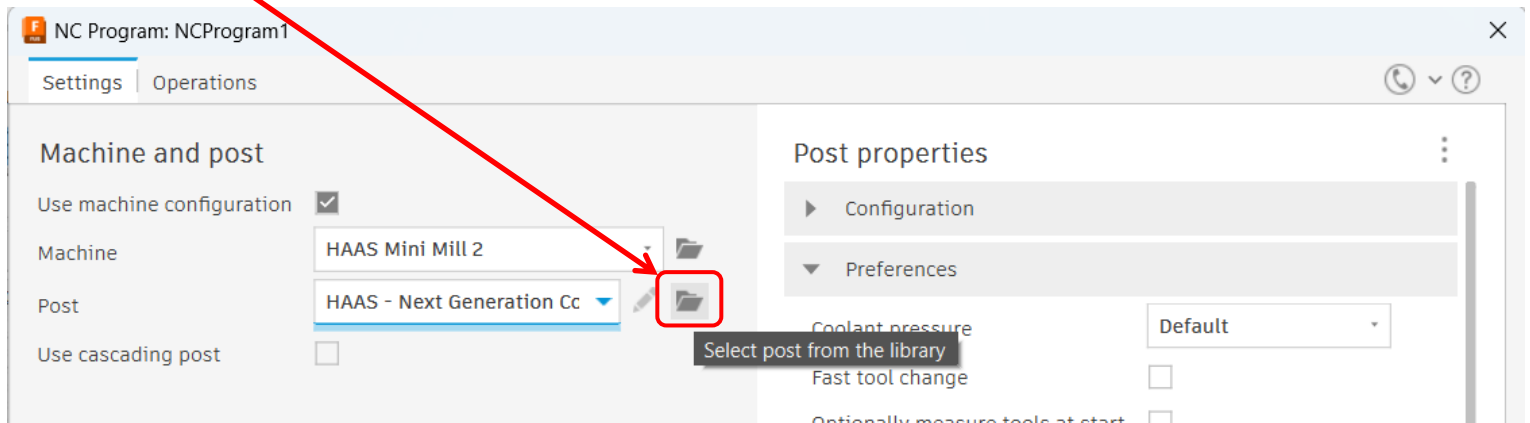
- right click on **Setup** again and select **Post Process**
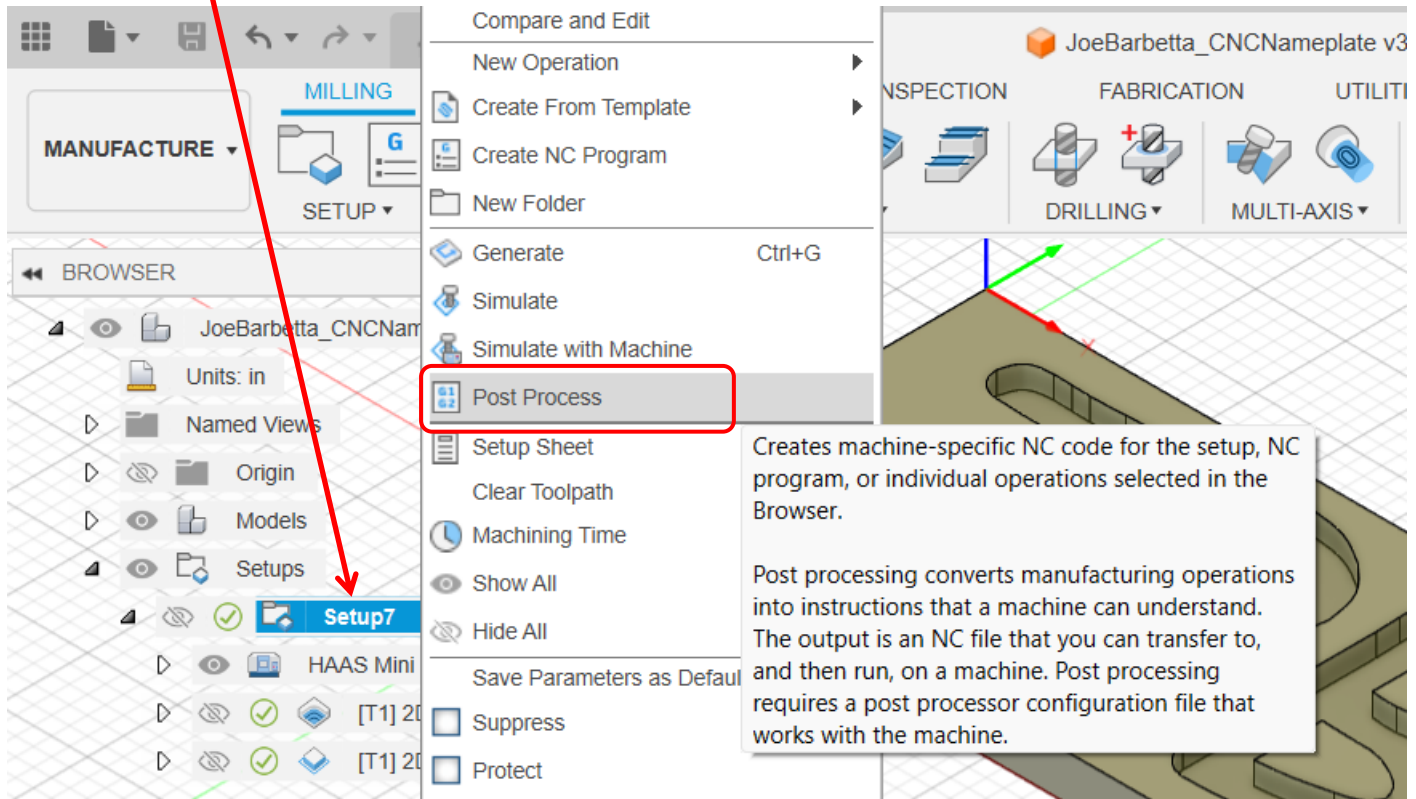
- click on the **folder icon** for the **Post** selection box

- scroll up a few lines to select **HAAS (pre-NGC)** and click **Select**

- when this window pops up click on **Copy to My Posts**

If Local doesn't show in the location box, click on the down arrow and select the Local location.

- change File name to your **first name and last initial**
- click on the **Output folder icon** to open the Post Directory window and select **Downloads**
- click the **Post** button at the bottom

If you are prompted to download Visual Studio you can decline.
If this window ever pops up **do not click** View NC Code. This window disappears on its own

NC code successfully posted

The following NC code **successfully** posted:

Fusion design: JoeBarbetta_CNCNameplate
NC program name/number: NCProgram1
NC file location: C:\Users\josbar\Downloads\JoeB.nc

View NC Code

- go to the **Downloads** folder on your computer
- from the **View** menu select **Show** and ensure **File name extensions** is checked
- you should see your **.nc** file that was just created
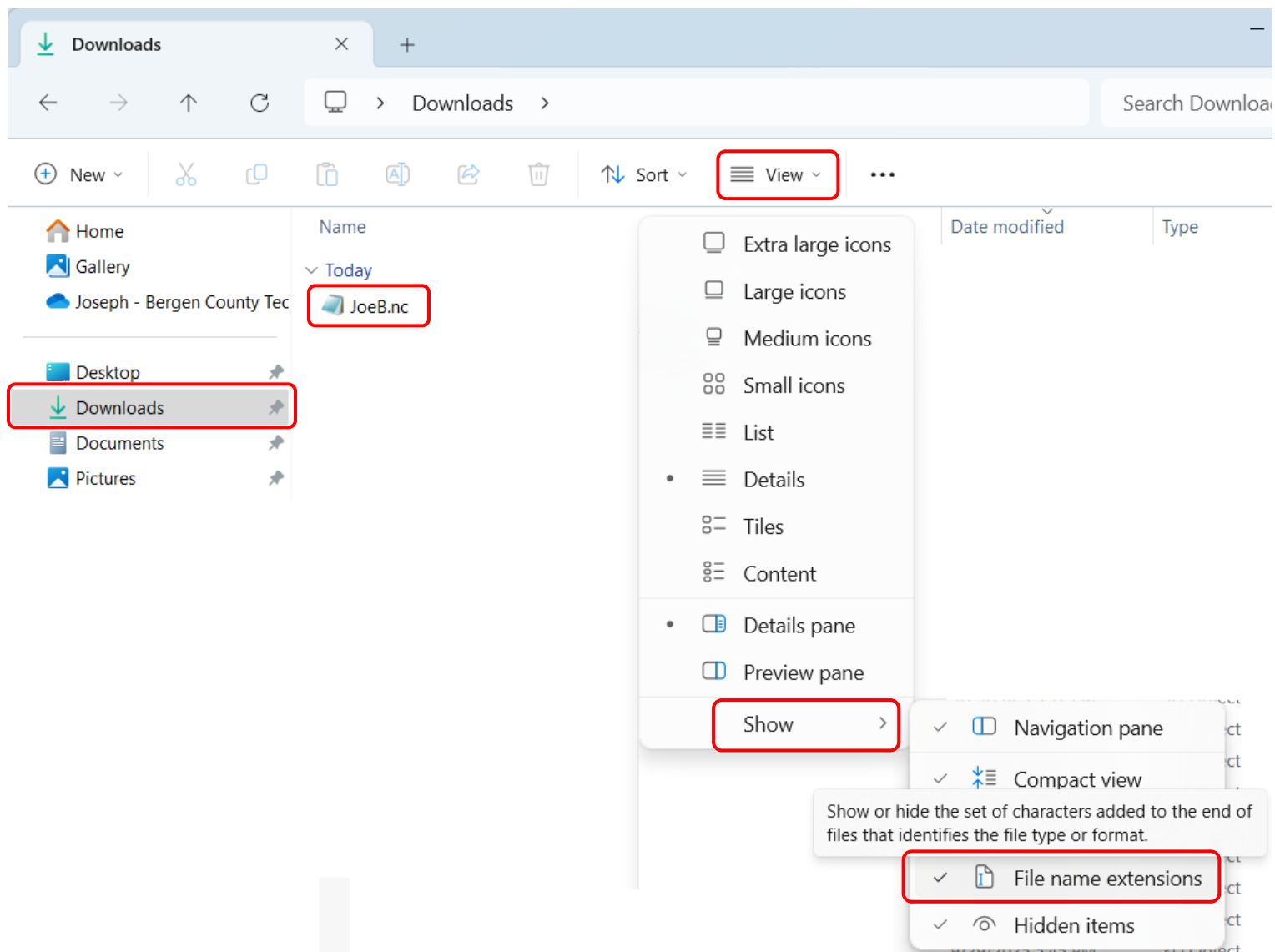
Viewing your G-Code

Windows NotePad is a simple text editor built into Windows. Macs have TextEdit as their text editor.

- using the bottom Windows search features enter **NotePad** and run the program.
- from the **File** menu select **Open**
- select the **Downloads** folder
- change **Text Documents (*.txt)** to **All files (*.*)**
-select the **.nc file** and click **Open**
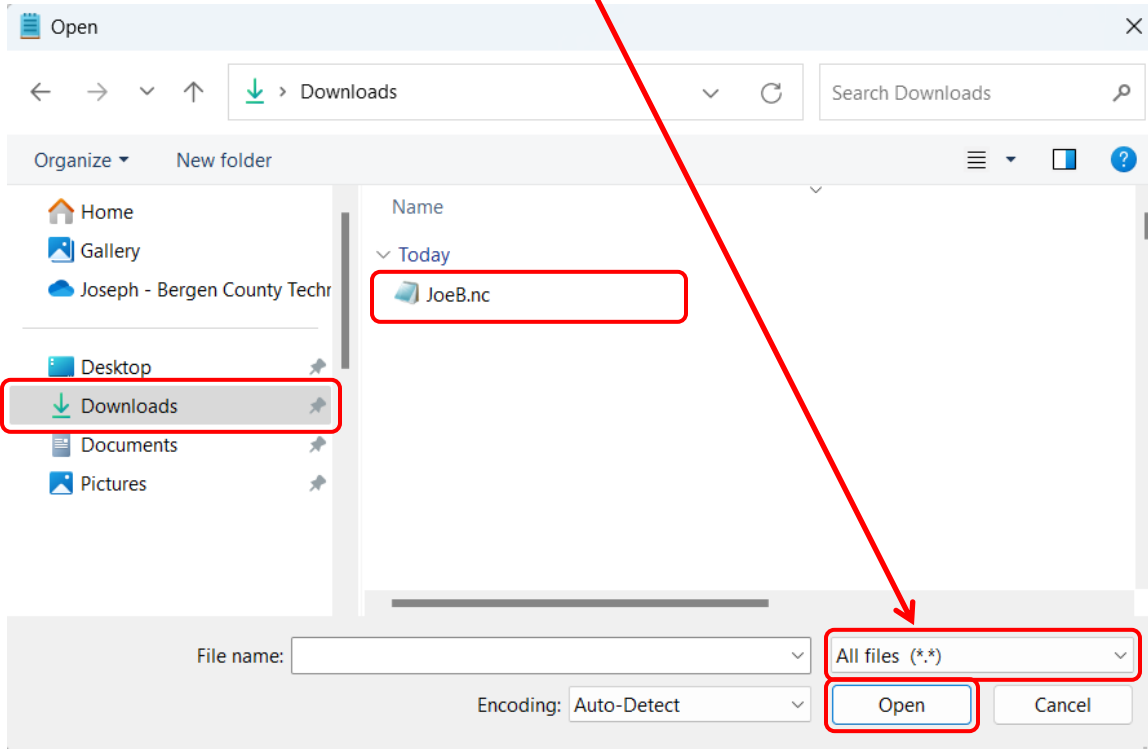
Note that the G-code can be long. This example has almost 1000 lines. This is mostly due to the font that is being used because many X Y movements are needed to mill out the letters. The next page shows and annotates what the first lines of the file can look like.

Here are the initial lines of G-code file.   Note that there may be variations in the lines

As with programming languages, G-Code has a syntax for comments. A **comment** can be specified by **sorounding it with parentheses**, as used here, or a comment can be preceded by a **semicolon, ":"**.

My comments are in blue.

```
%
O01001 (JPB)
(Using high feed G1 F500. instead of G0.)
(Machine)
(  vendor: HAAS)
(  model: Mini Mill 2)
(  description: Haas Mini Mill 2)
(T1 D=0.125 CR=0. - ZMIN=-0.3 - flat end mill)
N10 G90 G17       (G90=Distance Absolute, G17=Select X-Y Plane, Note1)
N15 G20           (G20=Length Unit Inches)
N20 G53 G0 Z0.    (G53=use Absolute Coordinates, Rapid move to Z0)

(2D Pocket2)
N25 T1 M6         (T1=Select Tool 1, M6=Change Tool)
N30 S4584 M3      (S=set Spindle Speed, M3=start Spindle Clockwise, Note2)
N35 G54           (G54=Coordinate System 1, our Work Coordinate System)
N40 M8            (M8=turn flood coolant on, Note3)
N45 G1 X-1.044 Y-0.0203 F500.  (Rapid move to position, Note4)
N50 G0 G43 Z0.6 H1 (G43=Use tool length offset from tool table for H1, Note5)
N55 G0 Z0.2       (Rapid move to 0.2" above the stock surface)
N60 G1 Z0.1125 F9.876
 a lot more follows
```

Because we are using a single tool for this project, it is a good idea to delete this line so the mill doesn't choose a tool that we don't want to use.

As stated in the notes, this line should be removed if the Tool Table on the Haas control panel has not been set.

For this project the plan is to use a single tool (1/8" endmill) and on the Haas we can zero or Work origin with the end mill touching the top surface of the stock.

Note1: One may also see G94 in this first line to ensure that Feed Rates are interpreted as Inch/Min.

Note2: The Haas Mini Mill 2 has a maximum spindle speed of 6000 RPM.

Note3: "Coolant" is a mix of cutting fluid and water that is sprayed onto the tool, ie end mill, as it is cutting. This removes heat from the cutting tool, helps remove chips, and, especially for aluminum, prevents the metal from building up on the tool surfaces. Presently, we don't have any coolant in the mill to avoid the mess.

Note4: G1 is typically used for slow cutting moves. However, here it is used with a high feed rate. This is sometimes used instead of a G0 to move to this position on a straight line. Using a G0 results in a "dog leg" move, wherein both axes may not be always moving. Using G1 results in a more predictable movement.

Note5: If one is unsure about the machine's tool table settings and if a single tool is used for the job, this line can be removed.

Final lines of G-code file. My comments are in blue.

```
N4730 M5                (M5=Stop Spindle)
N4735 M9                (M9=Flood Coolant Off)
N4740 G53 G0 Z0.        (G53=use Machine Coordinates, Rapid move to Z0)
N4745 X0.               (Rapid move to X0)
N4750 G53 G0 Y0.        (G53=use Machine Coordinates, Rapid move to Y0)
N4755 M30               (M30=Program Done)
%
```

Using a G-Code Simulator

Using an online G-code simulator

One can Google   online G-code simulator   and one that works well is   **ncviewer.com**   and it is shown below.

- click on the **folder** icon to open the . nc file that Fusion created.

Notice the location on the origin at the corner. If it shows at the center, a step was skipped earlier in the document.

Note the thicker gray lines. There is one every inch and thus there should be four as shown. If there are more than there was a mistake in the Fusion steps.

The letters look thin because the blue line shows the path of the bottom center of the end mill and doesn't account for its diameter.

What look like springs is the "helical plunging" path of the end mill. Instead of plunging the end mill straight into the stock like a drill bit, this movement eases it in and is the default for Fusion's pocket milling.

```
1   %
2   O01001
3   (Using high feed G1 F196.85
4   (Machine)
5   ( vendor: HAAS)
6   ( model: Mini Mill 2)
7   ( description: Haas Mini Mill
8   (T1 D=0.125 CR=0. - ZMIN=-0.3 -
9   N10 G90 G17
10  N15 G20
11  N20 G53 G0 Z0.
12
13  (2D Pocket2)
14  N25 T1 M6
15  N30 S5000 M3
16  N35 G17 G90
17  N40 G54
18  N45 M8
19  N50 G1 X1.7391 Y-1.1118 F196.85
20  N55 G0 G43 Z0.6 H1
21  N60 G0 Z0.2
22  N65 G1 Z0.1125 F13.123
23  N70 X1.7397 Y-1.1112 Z0.1078
24  N75 X1.7416 Y-1.1094 Z0.1038
25  N80 X1.7443 Y-1.1065 Z0.1011
26  N85 X1.7472 Y-1.1029 Z0.1
27  N90 G3 X1.7439 Y-1.1069 Z0.0872
28  N95 X1.7403 Y-1.1106 Z0.0743 I-
29  N100 X1.7364 Y-1.114 Z0.0615 I-
30  N105 X1.7322 Y-1.1171 Z0.0486 I
31  N110 X1.7278 Y-1.1198 Z0.0358 I
32  N115 X1.7231 Y-1.122 Z0.0229 I-
33  N120 X1.7183 Y-1.1239 Z0.0101 I
34  N125 X1.7133 Y-1.1253 Z-0.0028
35  N130 X1.     Y-1.      Z-  156
36
```

JoeB.nc

Digital Read Out

Machine Options

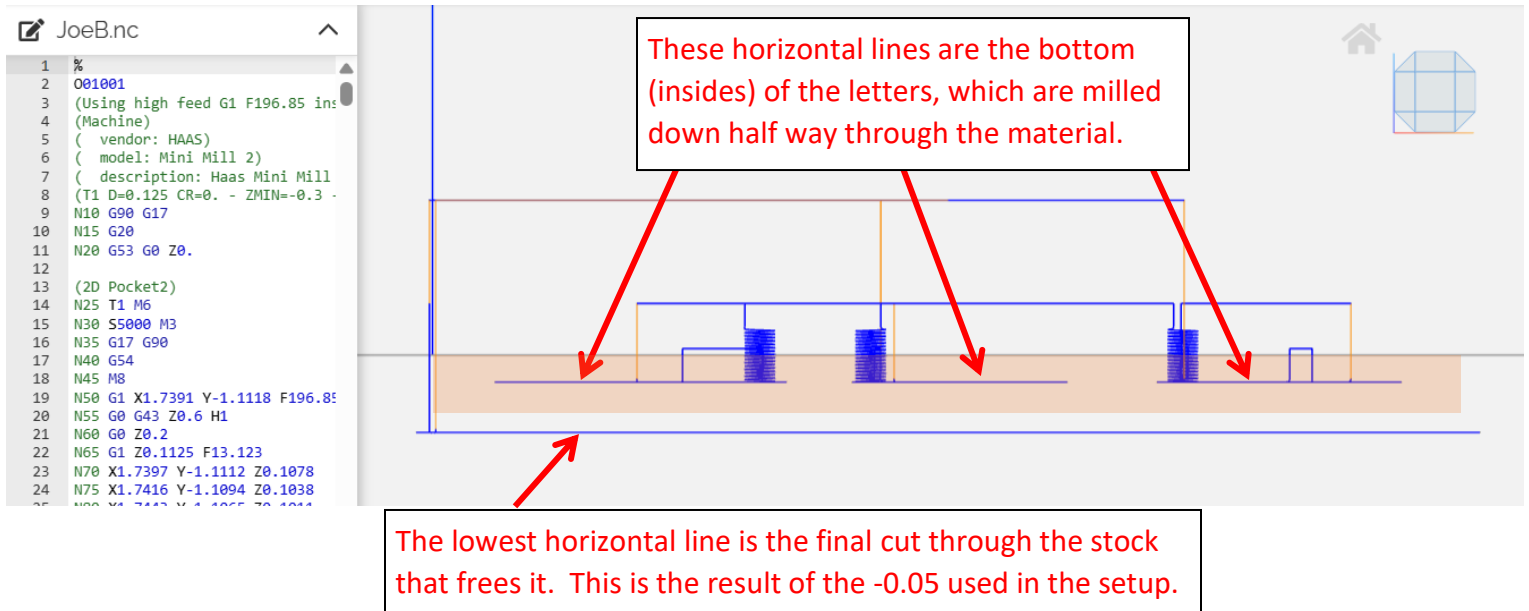- click on the **Run** button at the bottom of the NC Viewer screen.
This should show the **letters being cut first** and finaly the part outline.
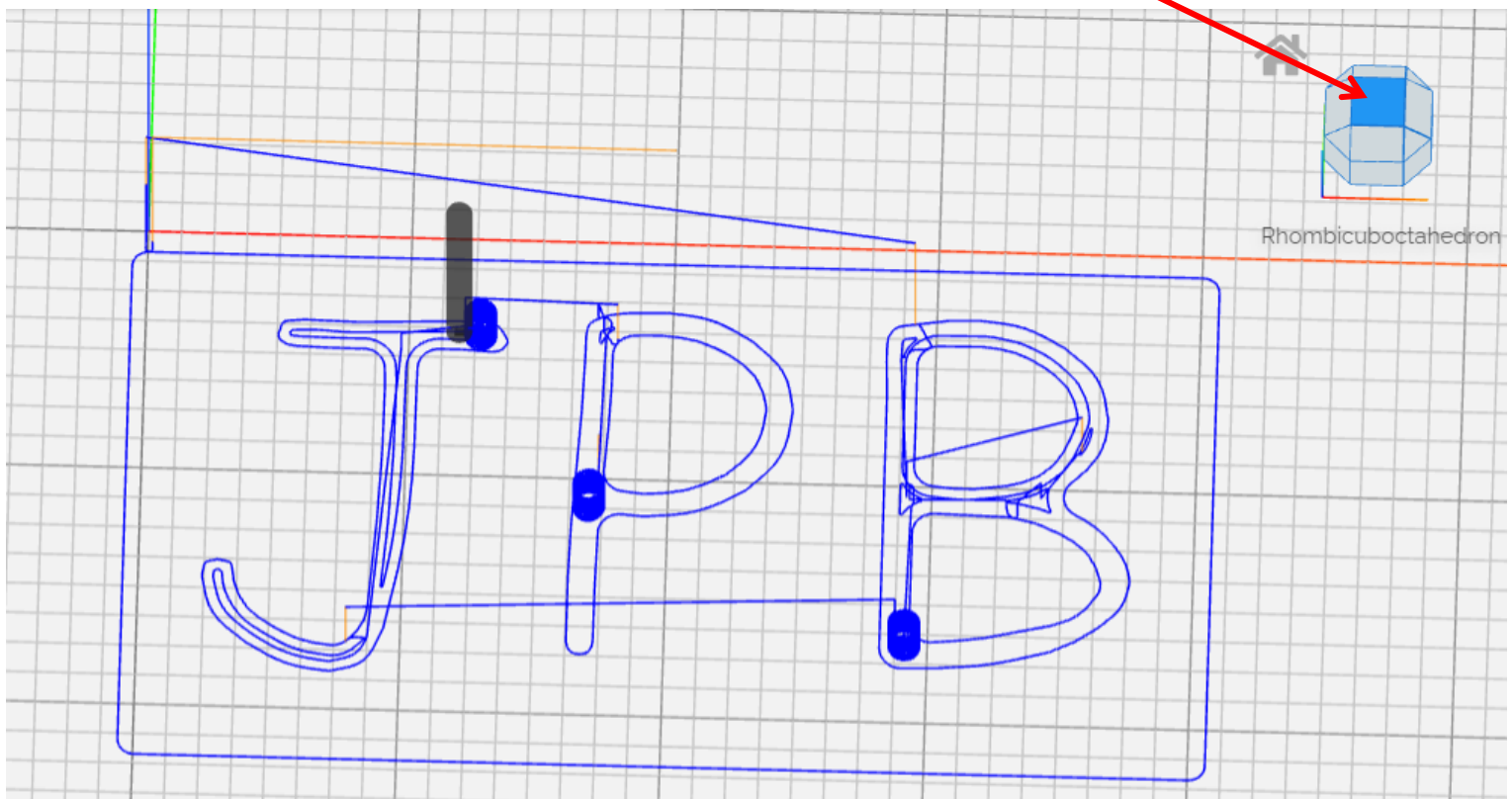
- click on the **face shown in blue** on the **Rhomicubooctahedron** View Cube at the top right of the NC Viewer screen.
Can you believe there is such a thing?

Rhombicubooctahedron
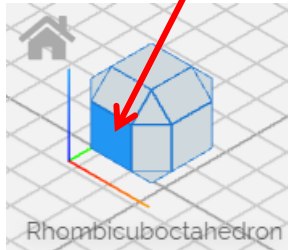
- after Zooming the view should look like this. **The orange region will Not be present. It was added to illustrate the location of the stock.**

These horizontal lines are the bottom (insides) of the letters, which are milled down half way through the material.

The lowest horizontal line is the final cut through the stock that frees it. This is the result of the -0.05 used in the setup.

- right-click on the NC Viewer screen and adjust the view similar to this and then click the **top** of the **Rhomicubooctahedron**.

The view should now look like this.

Because we choose the origin to be at the corner of our finished part, you can see that the tool path is 1/16" (the radius of the 1/8" end mill) from the origin.