Mr. Barbetta's ***Makerspace*** class

Laser Cutting/Engraving

Design a keychain

Today's Lesson is Sponsored by DIGGERLAND in West Berlin, NJ

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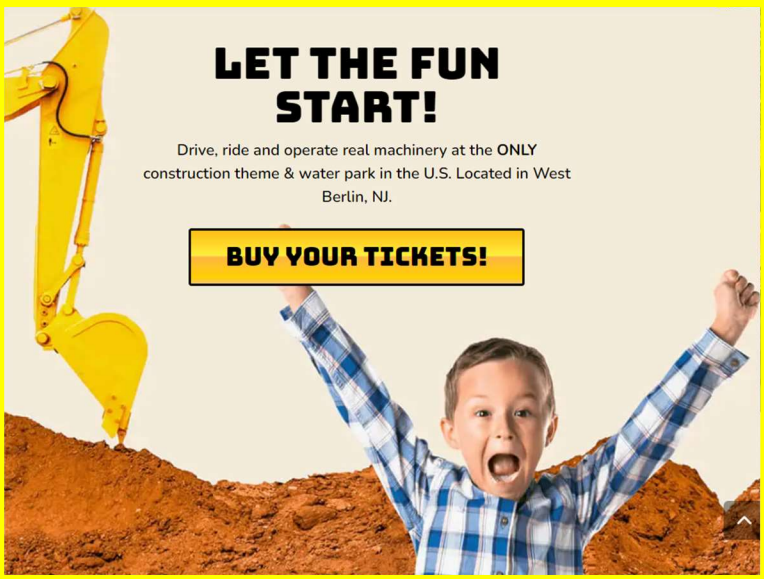

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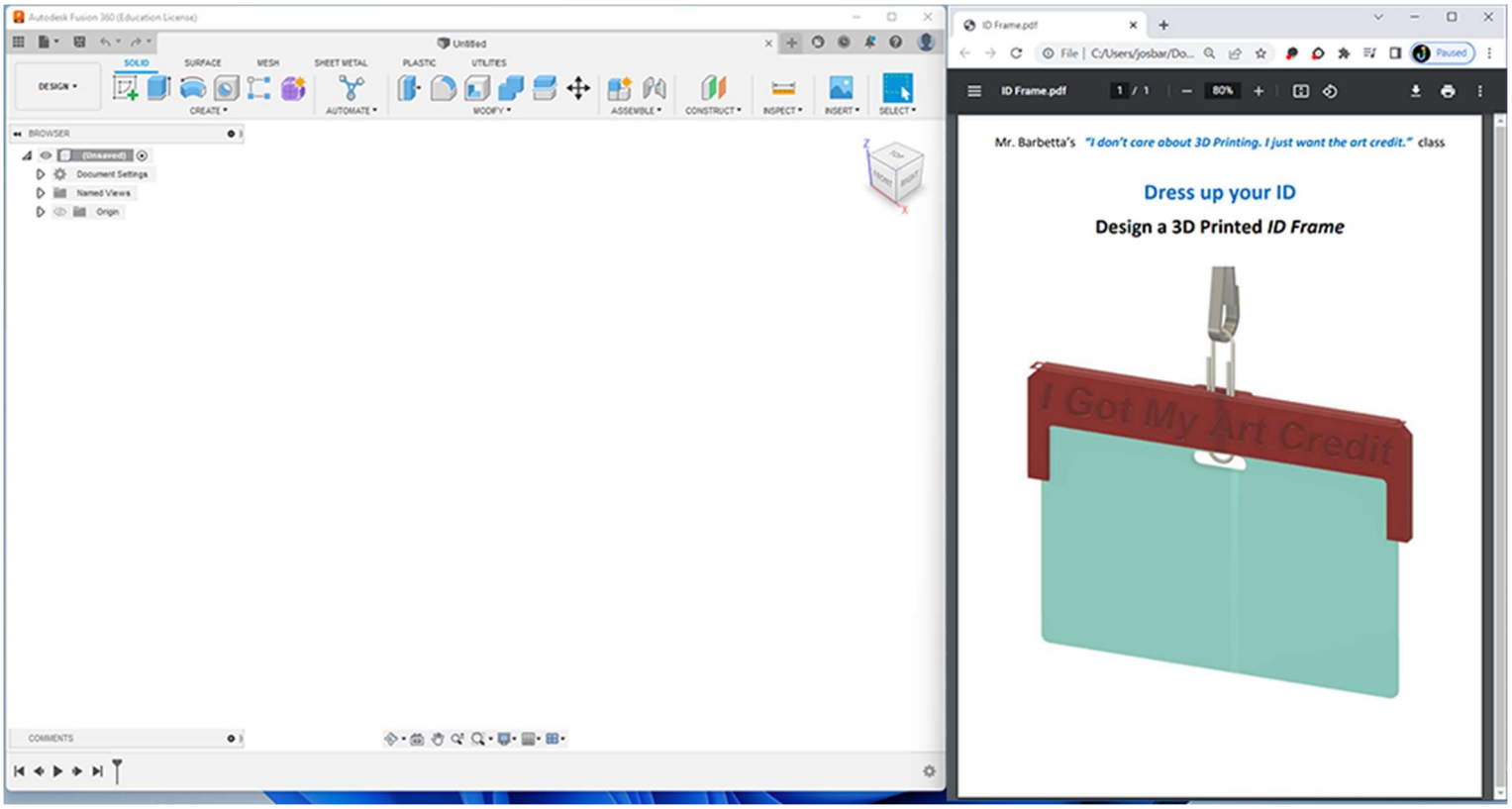
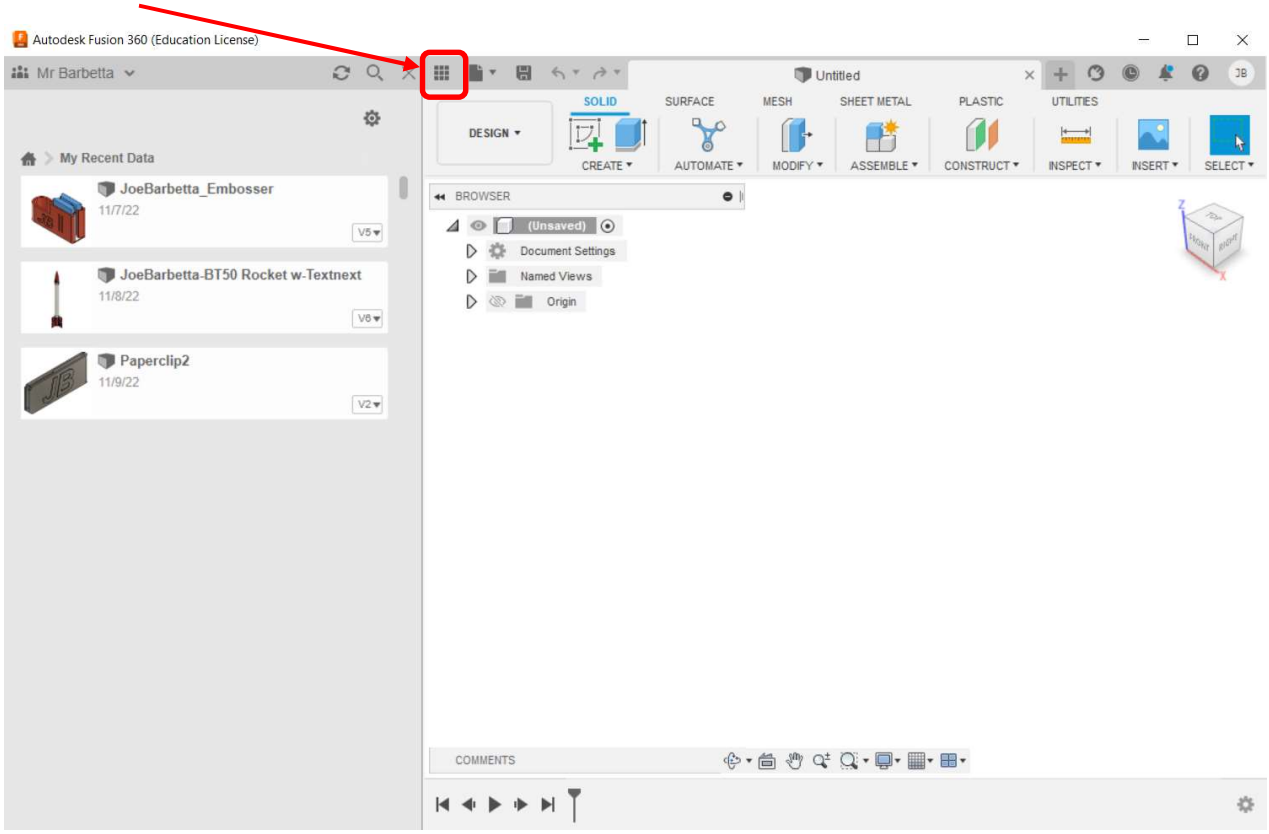
Using This Document

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.



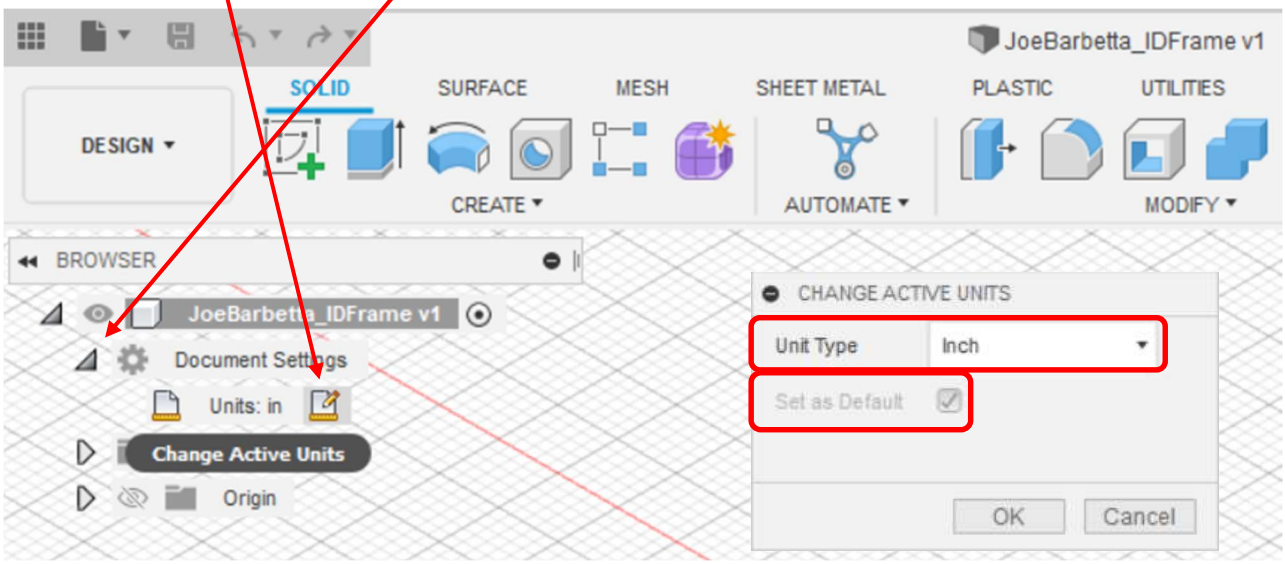
Starting a Design in Fusion 360

- open **Fusion 360**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fu
- from top **File** icon select **Save** and name the file.
  Use your name followed by **_Keychain** e.g. **JoeBarbetta_Keychain** (note the use of the underscore)

Note that by default Fusion 360 saves your project to "the cloud", which are the servers managed by AutoDesk. When you log into Fusion 360 on a different computer, your projects will be available.

As you work you may want to occasionally save your work in case Fusion 360 crashes or an asteroid hits Earth

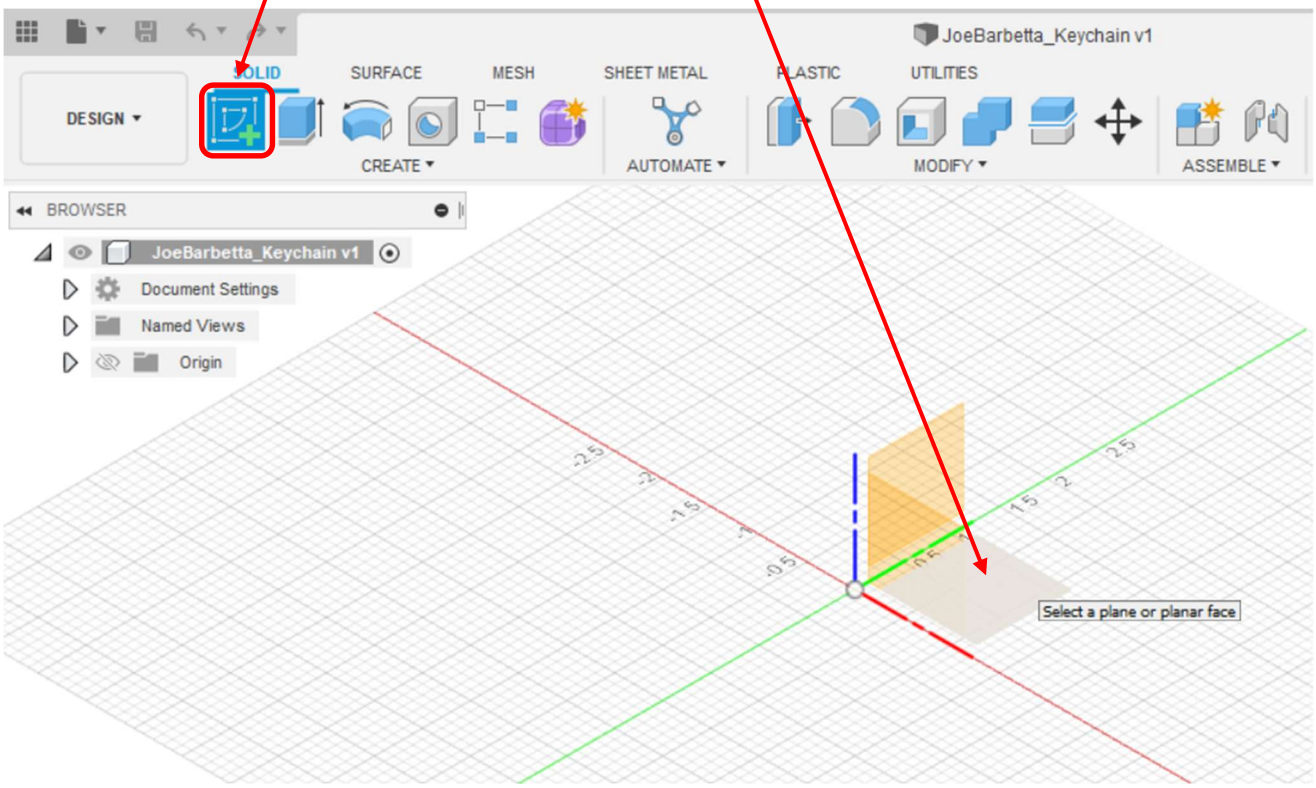
- in the left **"BROWSER"** click on the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over **Units**
- change **Unit Type** to **Inch** and click **OK**. You can also enable **Set as Default** if it is not grayed out.



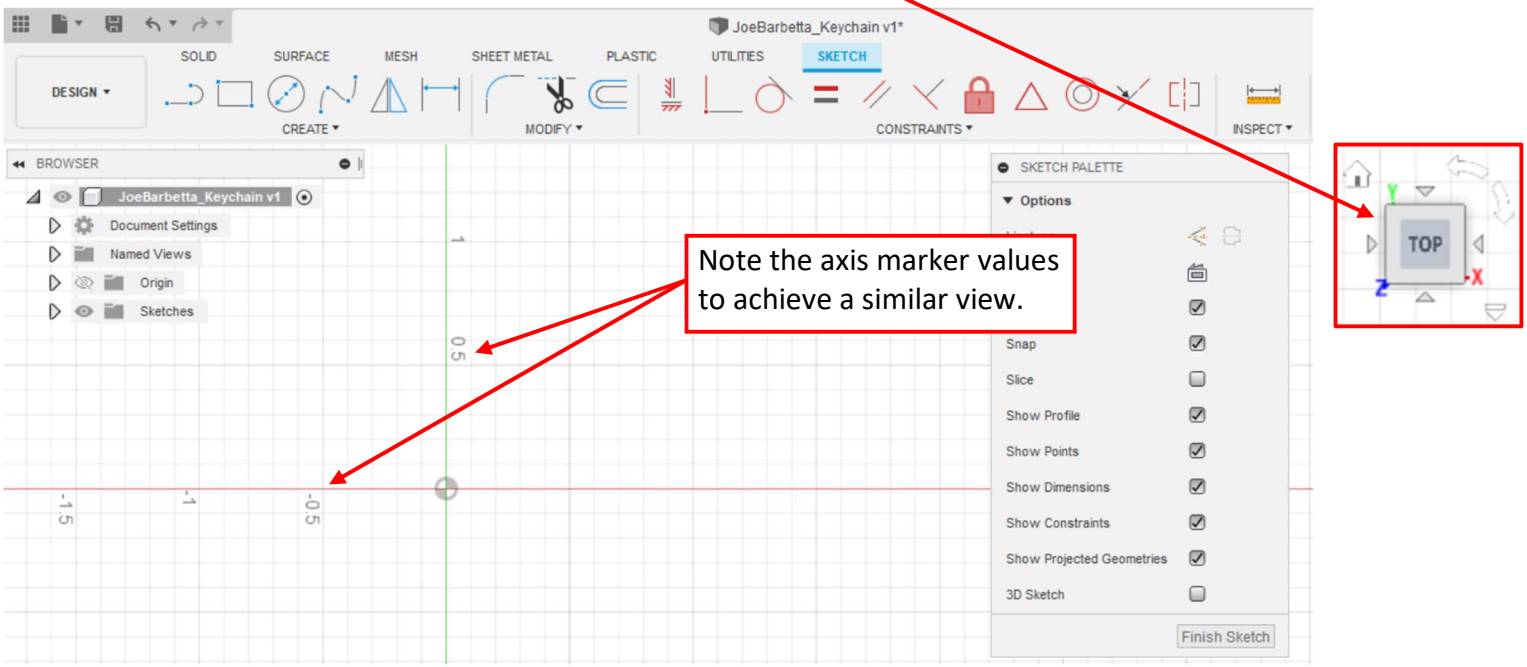
Creating the First Sketch

Note that a Fusion 360 expert would suggest first creating a *Component*, but you can say **"Dude, I'm just trying to make a keychain."**

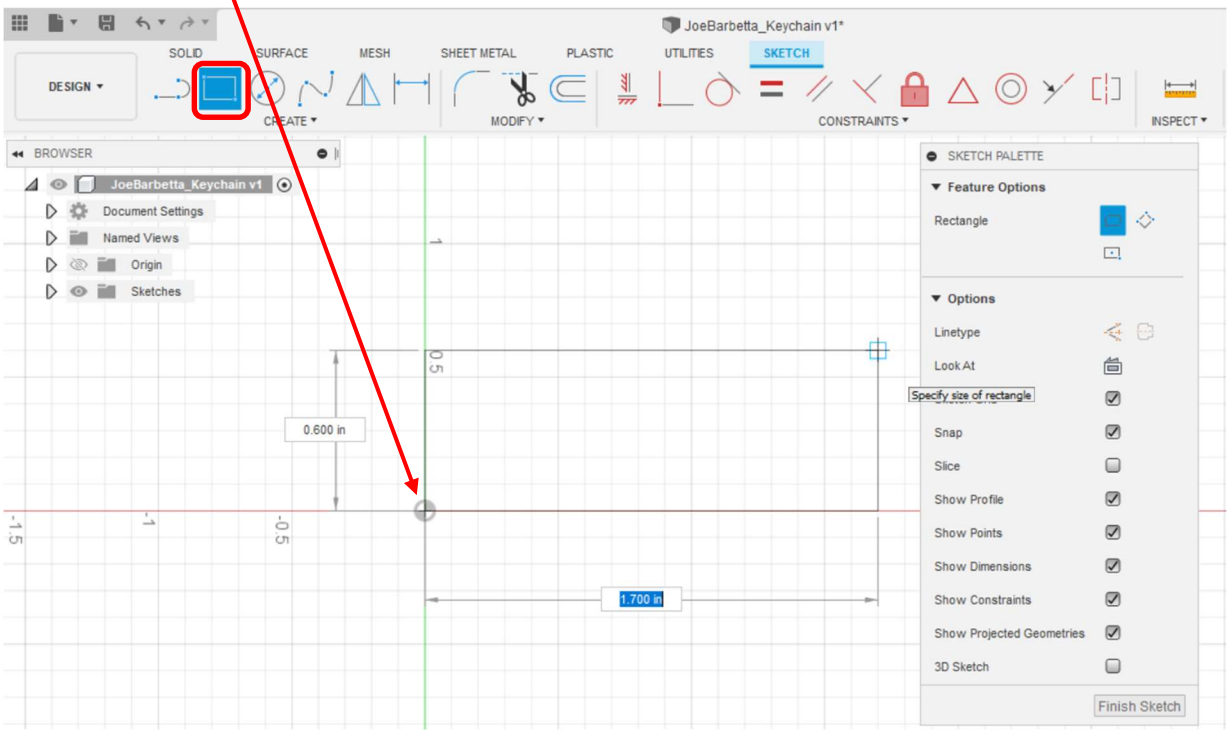
- select the top **Create Sketch** tool and click on the **bottom rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.



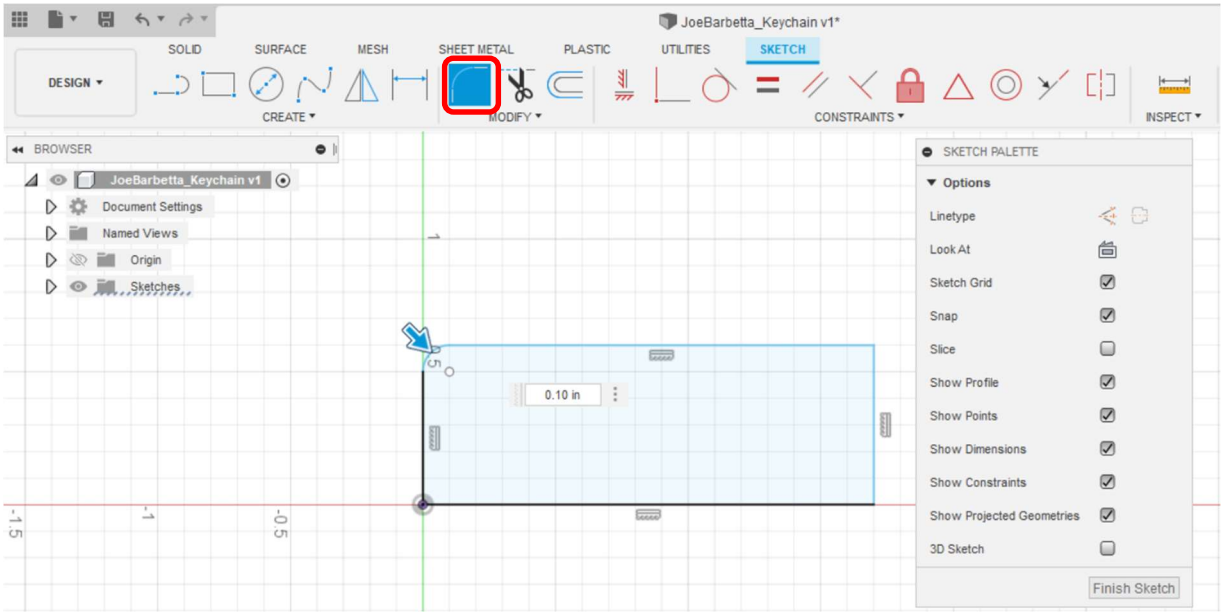
- ensure the upper right **View Cube** at the upper right displays **TOP** as shown. If not, return to the previous step.
- **zoom** and **pan** adjust the view to be similar to that below. To **pan** hold down the mouse wheel while moving.



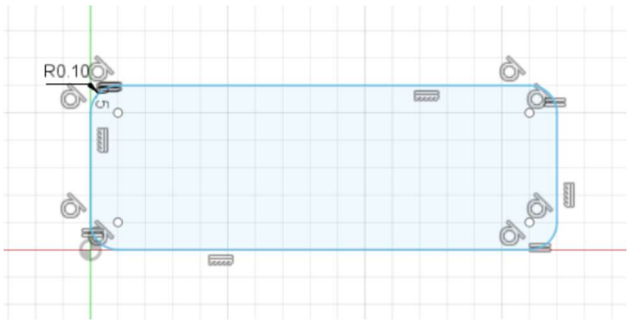
- select the **Rectangle** tool
- click on the **Origin** and extend the rectangle until it has a width of **1.7** and height of **0.6** and click near that point. Do Not click Finish Sketch.



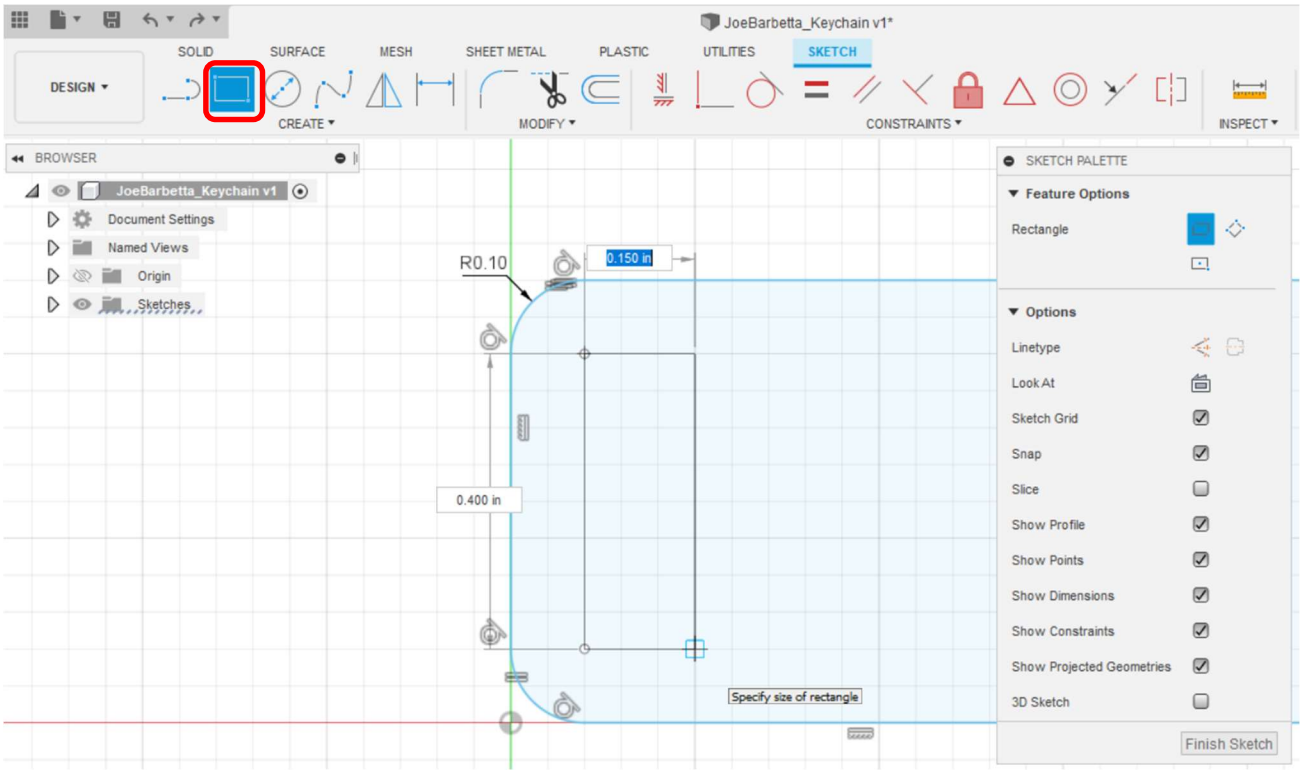
- select the **Fillet** tool, click on the upper left corner, type **0.1**. **Do Not** press the Enter key.
- click the **other 3 corners** and then press the **Enter key**. All 4 corners should now have a 0.1" fillet.



The result should appear as below.



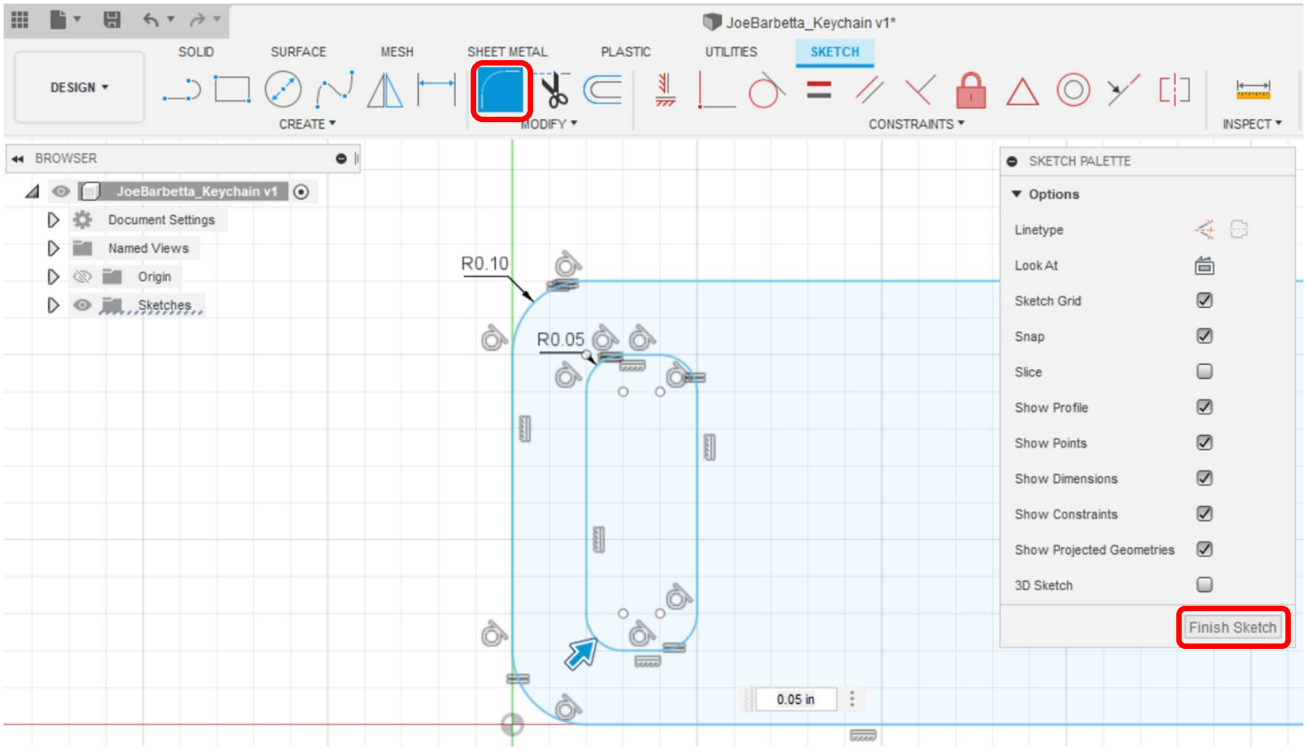
- zoom in as shown using the Mouse Wheel
- select the **Rectangle** tool and click on a starting point that is **0.1" from the side and top edges**. At this zoom amount, grid lines are spaced by **0.05"**.
- **extend the rectangle** until its height is **0.4** and width is **0.15**. This will be the slot in the keychain.



- select the **Fillet** tool and set the radius of each corner of this new rectangle to **0.05**.

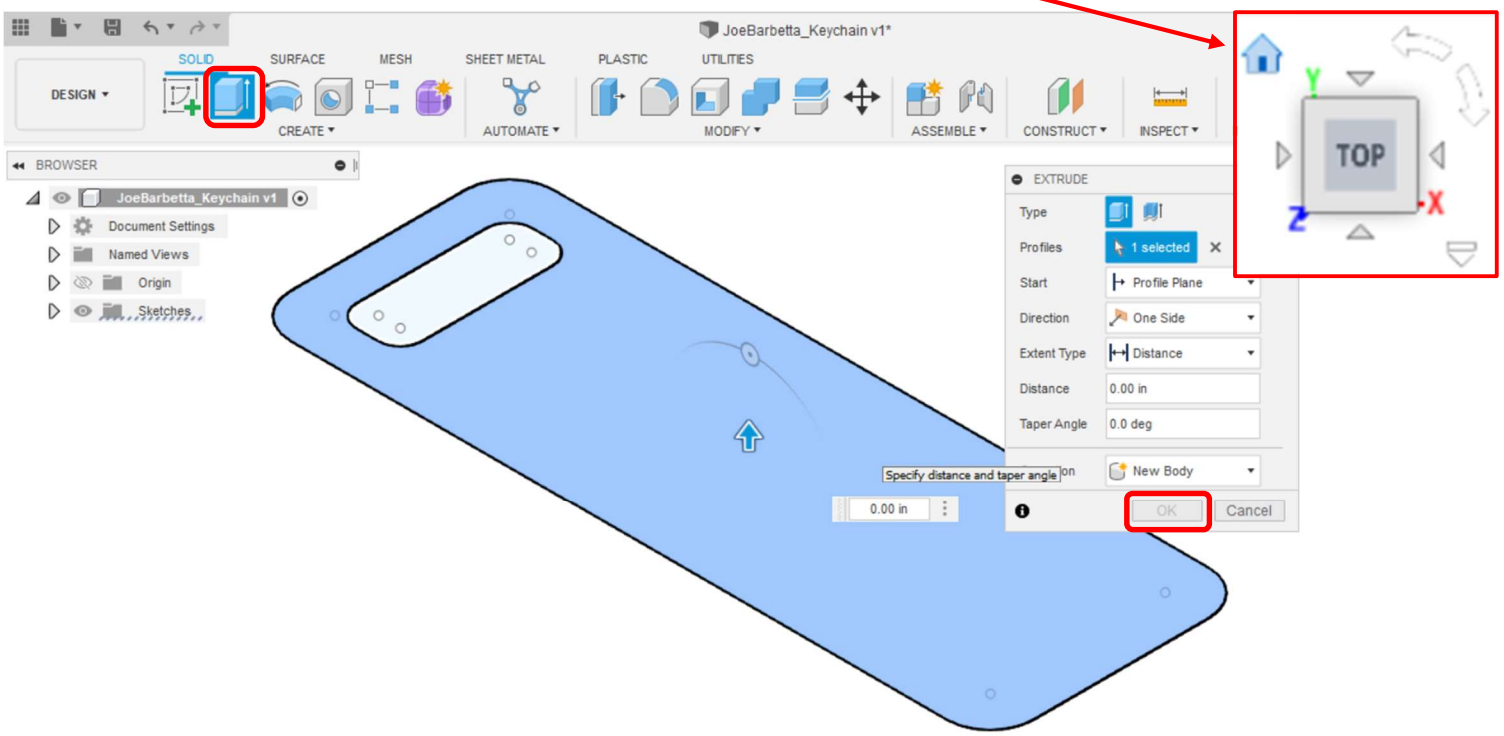
The first set of **outer fillets** was performed for aesthetic reasons and to avoid sharp edges. These **internal fillets** will reduce stress at sharp inner corners to prevent cracking.

- click **Finish Sketch**

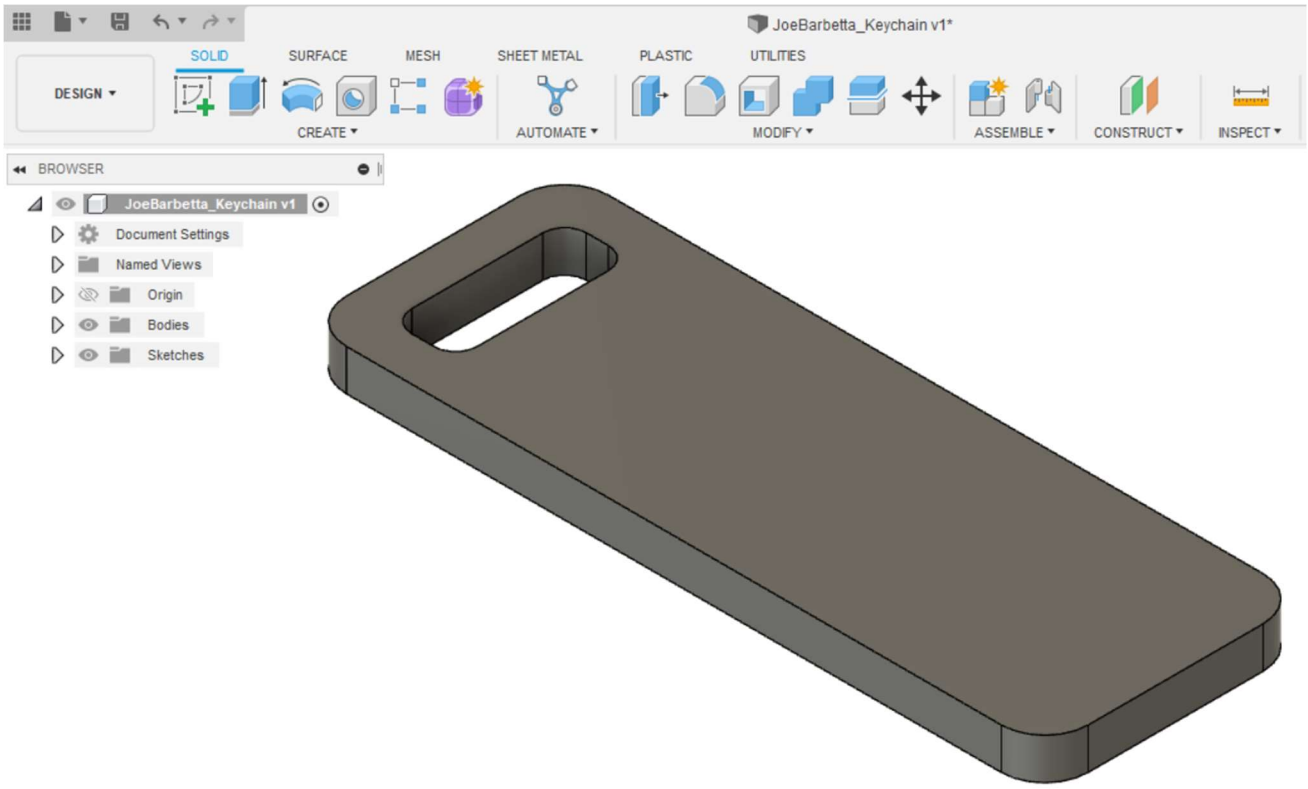


- click on the **Home** icon at the top left of the **View Cube** to return to the *Home View*
- select the **Extrude** tool, click on the top **Face**, type **0.094**, and click **OK**.

0.094 is used to produce a body with a thickness near **3/32"**, which is a thickness that **Acrylic** plastic is commonly sold in.



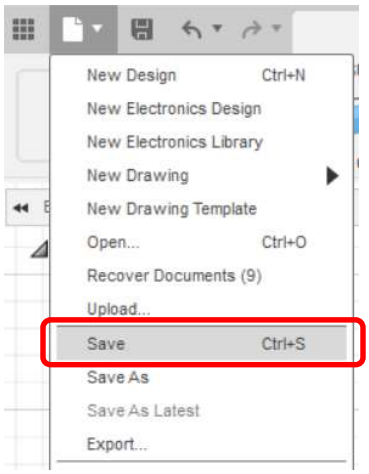
The Body of the keychain should look like that below. We just created a solid Body.

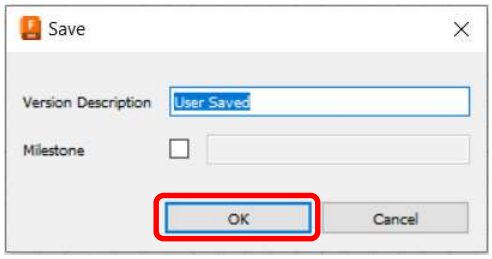


Saving a Project

There are some more steps, but It's a good idea to save your project occasionally. Fusion 360 does crash at times and it employs a sophisticated algorithm to time a crash at the worst time.

- click **OK** in the Save popup window. The Version Description can be kept at its default.

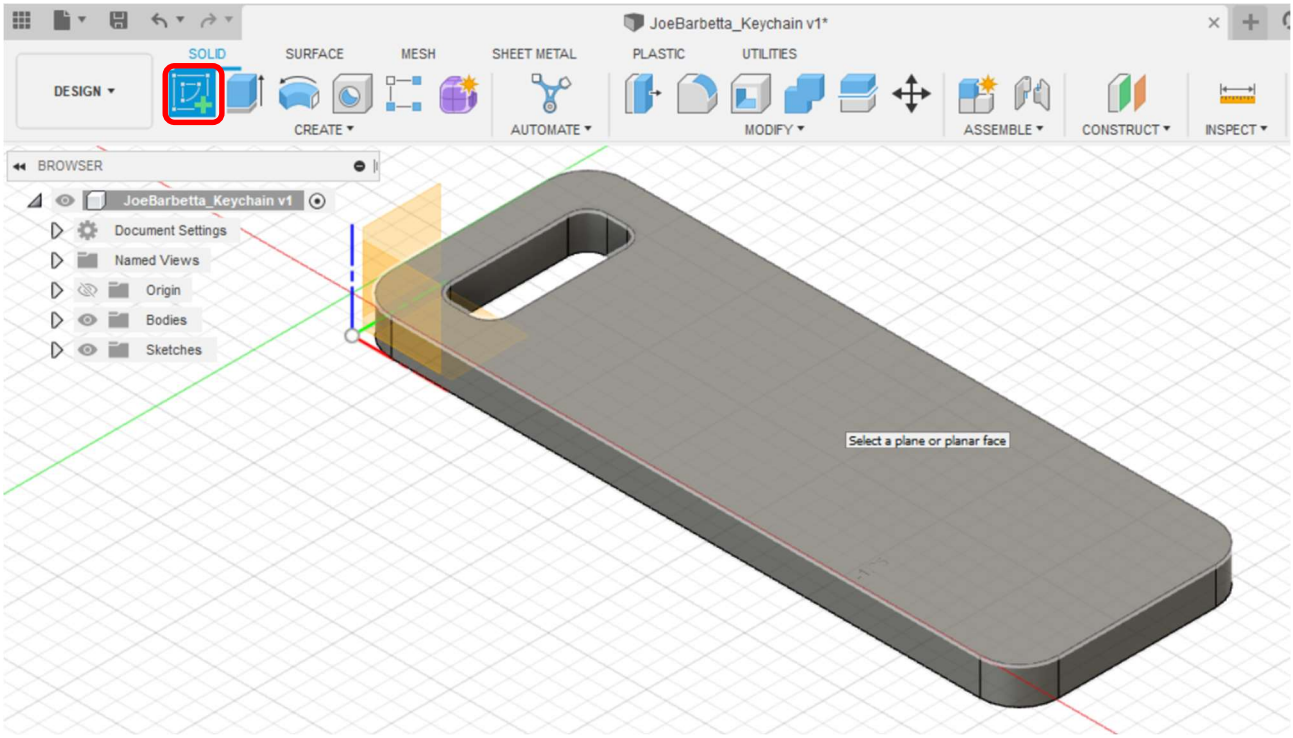




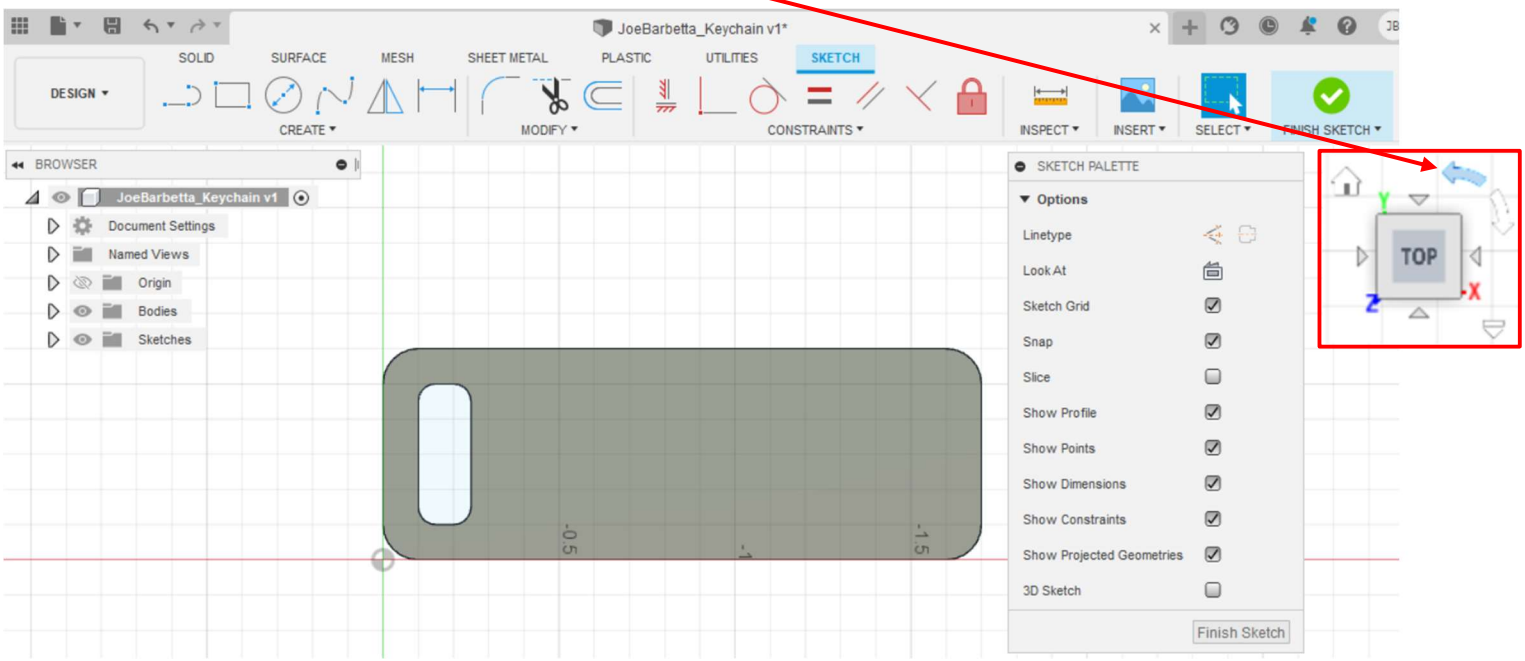
Adding Text

At the start of the project a Plane was selected to start a Sketch. When a design has one or more Bodies, any flat Face can also be selected to Sketch on.

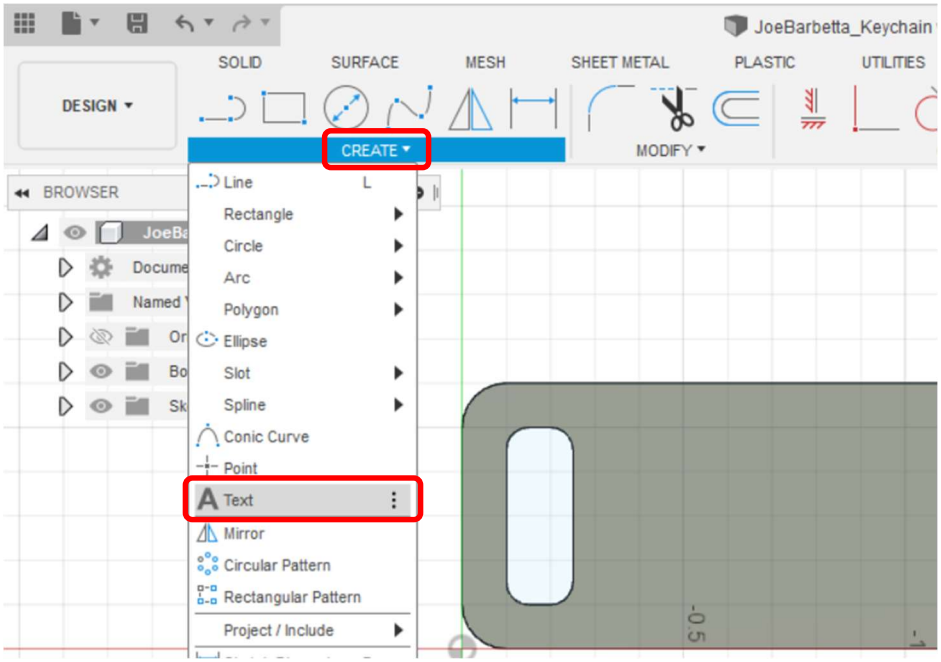
- select **Create Sketch** and click on the **top Face** of the keychain.



- zoom in to achieve a view as below. Click on the **Curved Arrows** to rotate the view if needed.

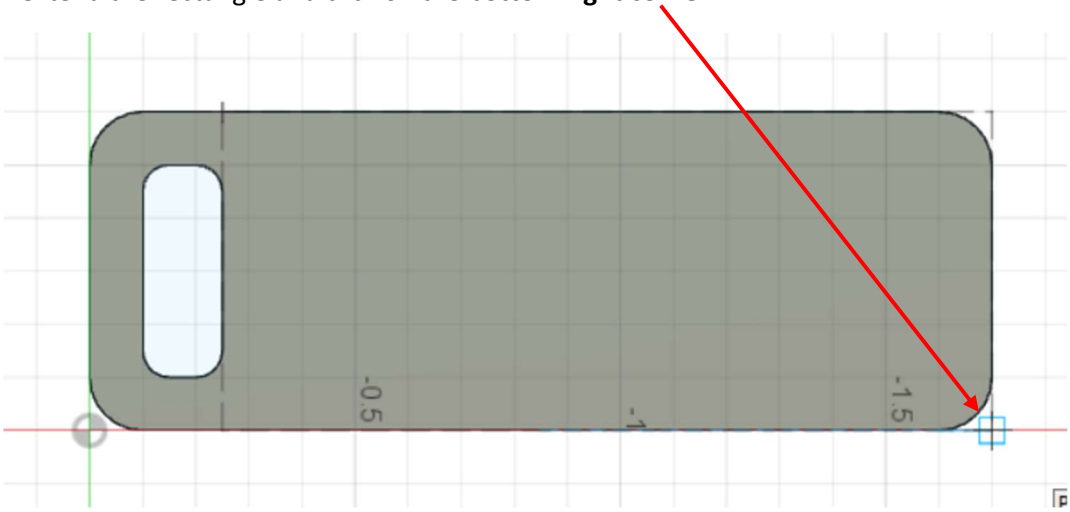


- from the **CREATE** pull-down menu select **Text**

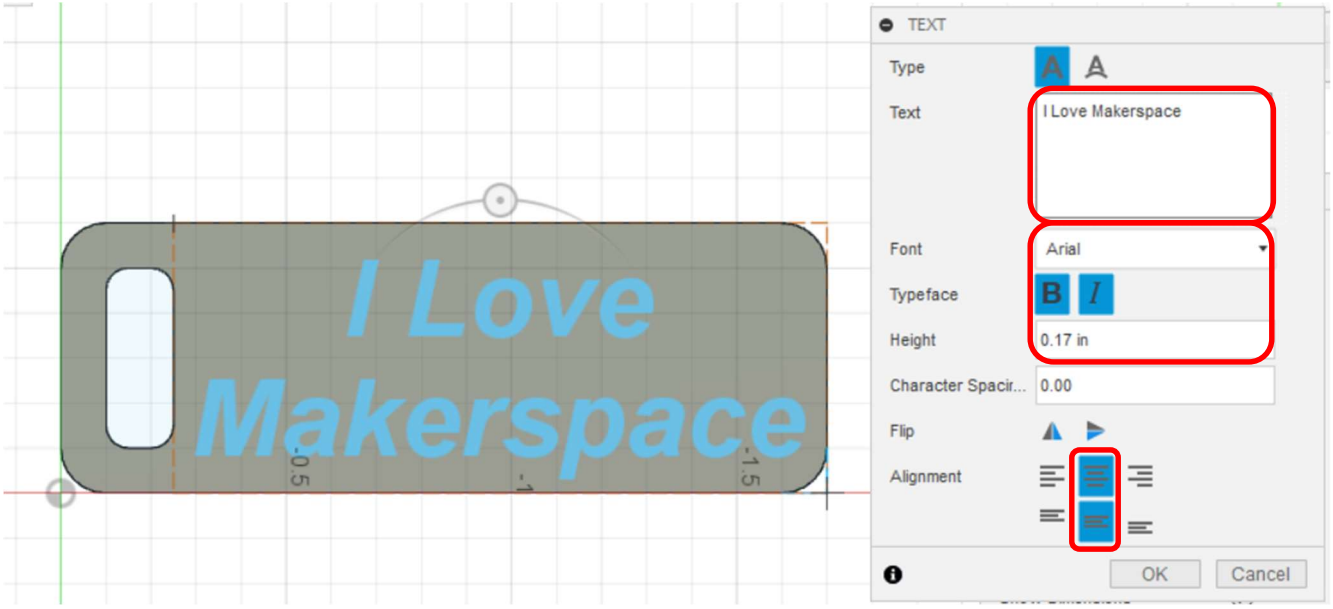


- define a rectangle for the Text Extents by clicking on the **edge above the slot**. Note that a blue X and a dashed line will appear when at the correct position.

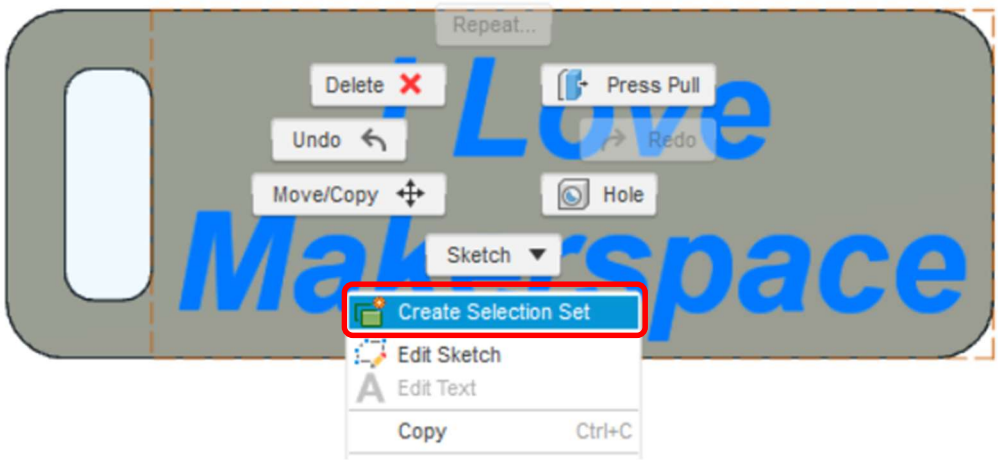
- extend the rectangle and click on the **bottom right corner**



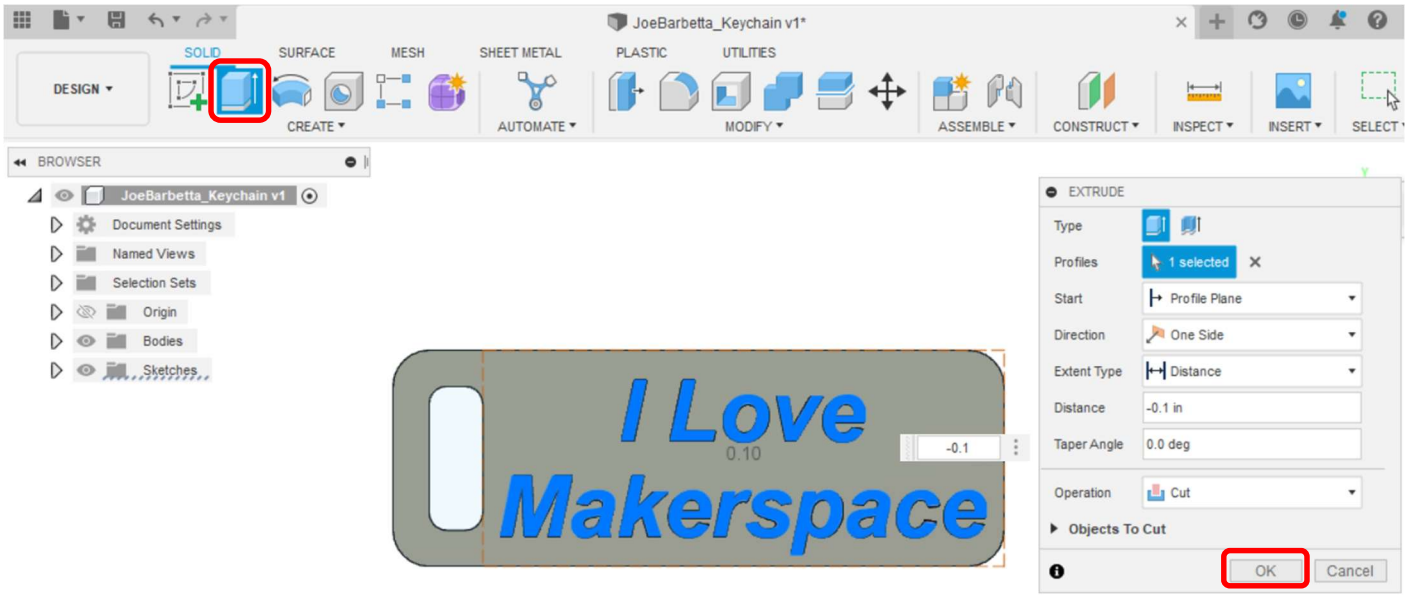
- in the **TEXT** window enter your text for **Text** and click on the **Center** and **Middle** icons for **Alignment**.
- try different values in the **Height** box to set the desired height. Here **0.17** is used. If the text doesn't fit try a smaller height.
- as per personal preference **Bold** and/or **Italics** can be selected for the **Typeface** and the **Font** can be changed from the default of **Arial**. Note that not all fonts can be extruded and if the later Extrude step fails, a new font must be selected.
- click **OK** when done and then **Finish Sketch**



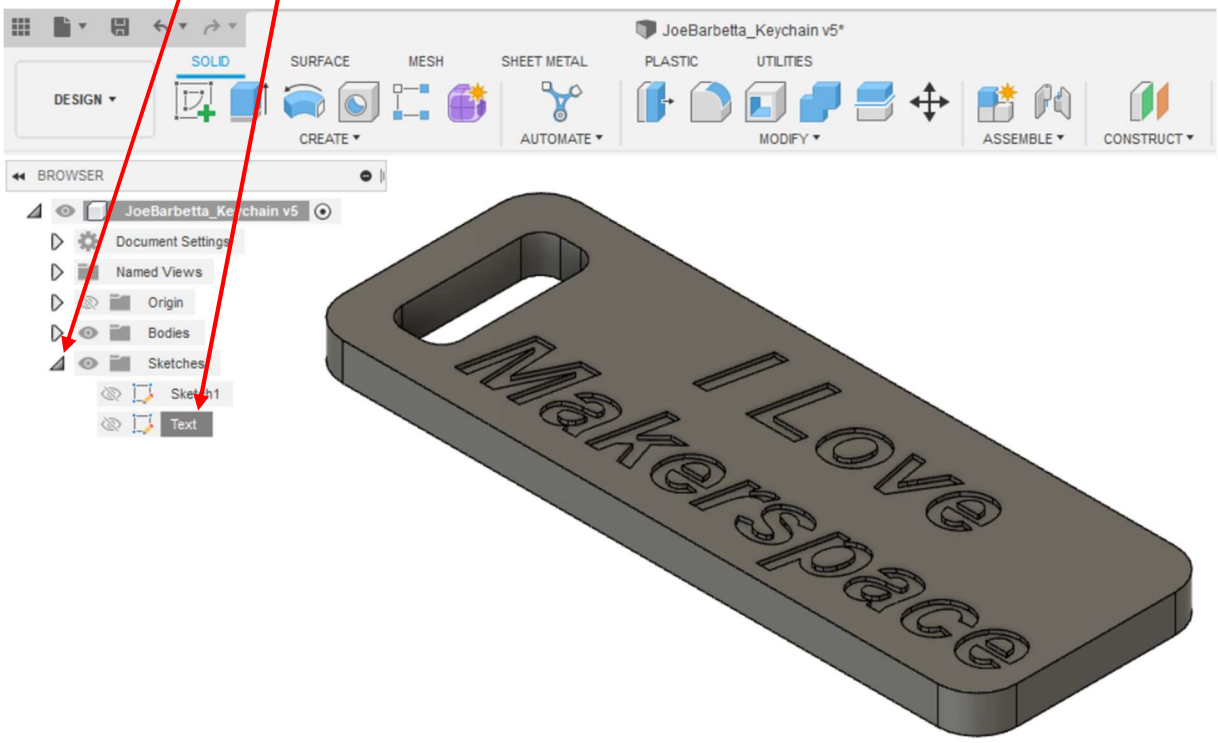
- right-click on the **text** and select **Create Selection Set**. Nothing will seem to happen, but this is needed for the next Extrude operation.



- select the **Extrude** tool, type **-0.02** (note the minus sign), and click **OK**



- click on the **Home** icon at the top left of the **View Cube** to return to the *Home View*
- click on the **Expand Arrow** next to **Sketches** to open the Sketches folder
- right-click on the **2nd Sketch** and select **Rename** and change the name to **Text**. If you wish to later edit the text or its properties, one can right-click on the Sketch name and select **Edit Sketch** and then right-click on the text and select **Edit Text**.

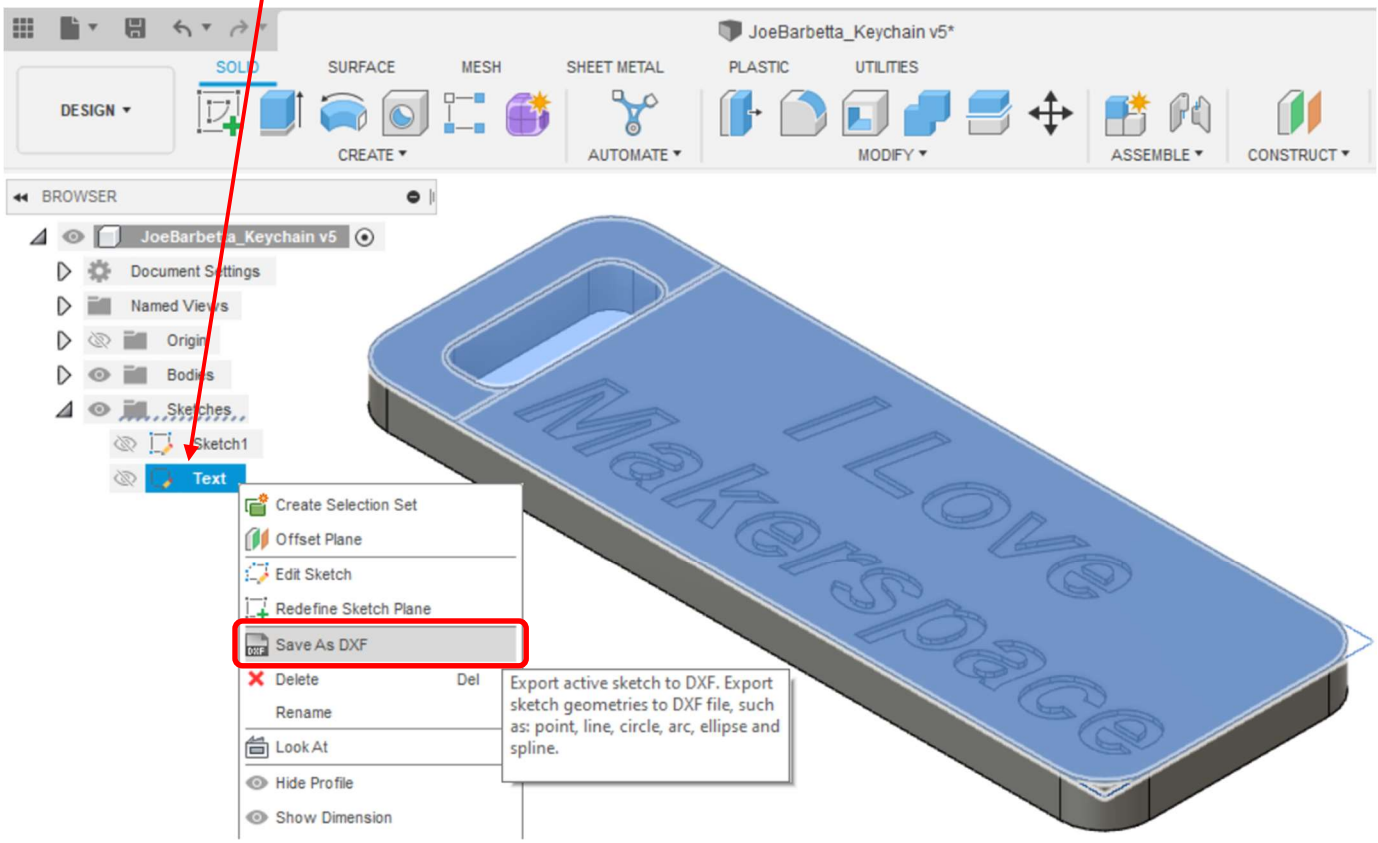


Exporting a DXF File

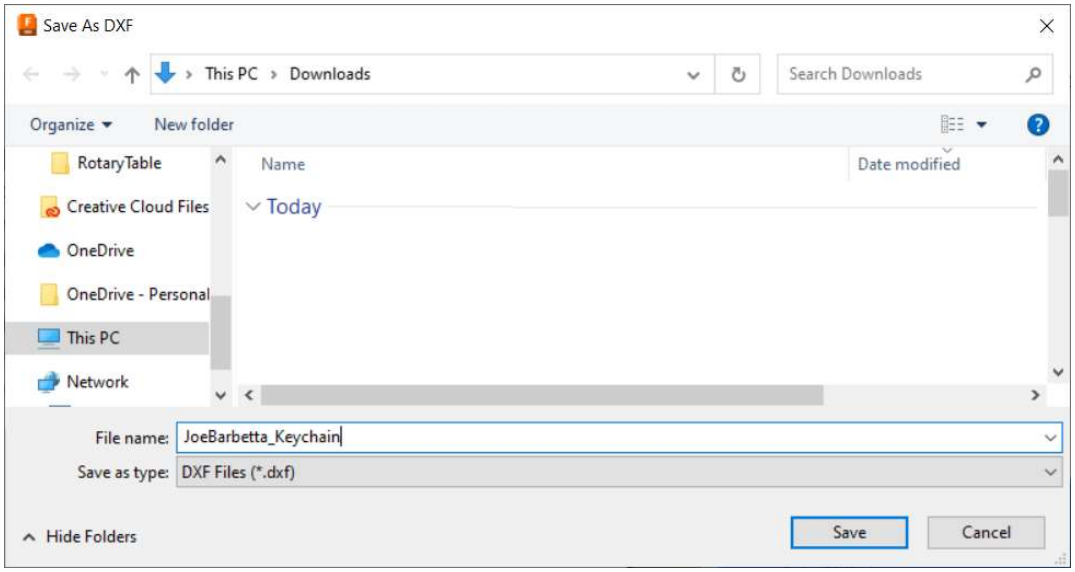
A DXF (Drawing Exchange File) is a file format first introduced in 1982 by Autodesk for their AutoCAD product. Autodesk also sells Fusion 360. Since then, it has been a common format for 2D drawings.

Some machines, such as a 3D Printer or CNC (Computer Numerical Control) milling machine or lathe, need 3 dimensional information for part fabrication. Most laser cutters/engravers only need 2 dimensional information and a DXF file can be used.

- right-click on the Sketch that contains the text and select **Save As DXF**



- enter **your name followed by an underscore and the project name**, e.g. **JoeBarbetta_Keychain**
- click **Save**. By default, Fusion 360 saves to the **Downloads** folder on the PC



Adobe Illustrator Instructions (**for teacher**)

Start **Illustrator**

Create a **New project** and select **8.5 x 11 inches, portrait**

Use top menu **Edit > Preferences > Units** to set **General** and **Stroke** to **Inches**

Use top menu **File > Place...** and **select dxf file**

Check **Show Import Options** near the bottom. This will show a window when a file is selected to allow changing the scale.

If the orginal work was done in inches it should show

Scale by: 100% selected

Scale: 1 Unit(s)= 1 Inches

In the future Show Import Options can be unchecked.

if needed: **Object > Transform > Rotate 180**

if needed: **Object > Transform > Scale** 2540% or 39.37% to correct for other units used

On the right side of Illustrator **Properties** should be selected as the top tab

Under **Appearance**, **Stroke** may show a default of **0.0098 in**

If the dxf file has only lines to be cut, select all objects using the selection rectangle and enter **0.001** in the **Stroke** field

If the dxf file has text and/or art, **select all objects** and set the **Stroke** to **0.015**. This will cause the Eplilog to engrave these items using its Raster mode. Then use the **Direct Selection tool (filled pointer icon)** and click on each object while holding the **Shift key** to select lines to be cut and set the **Stroke** to **0.001** It may be helpful to zoom in to select objects to be cut. Lines set to 0.001 will cause the Epilog to use its Vector cutting mode to cut these lines.

The Stroke setting of 0.015 works well for text that is about 0.25" high. The .dxf file generated from Fusion will have each letter outlined. If a thicked stroke is used, proper letter seperation may not result. If text will be larger, then thicker widths could be appropriate.

To fill in letters: (typically not needed)

select Selection tool (unfilled pointer) and click on imported object and change Stroke to 0.002

select the Direct Selection tool (filled pointer) and click on (and press the delete key) for unwanted line segments as some may have resulted from the text extents rectangle

use the selection rectangle to select all text

Object > Path > Join

Hold shift and click on unwanted segments and then press the delete key. Unfortunately, the selected segments will not indicate their selected state

One can also click on a segment and press delete, but then the background must be clicked on before the next delete

Hold shift and select the paths for any letter interiors, such as that in a, e, o, etc. right-click Arrange > Bring to Front

pull the selection rectangle to select all text Choose Fill black

use the Selection tool (unfilled pointer) to position the object