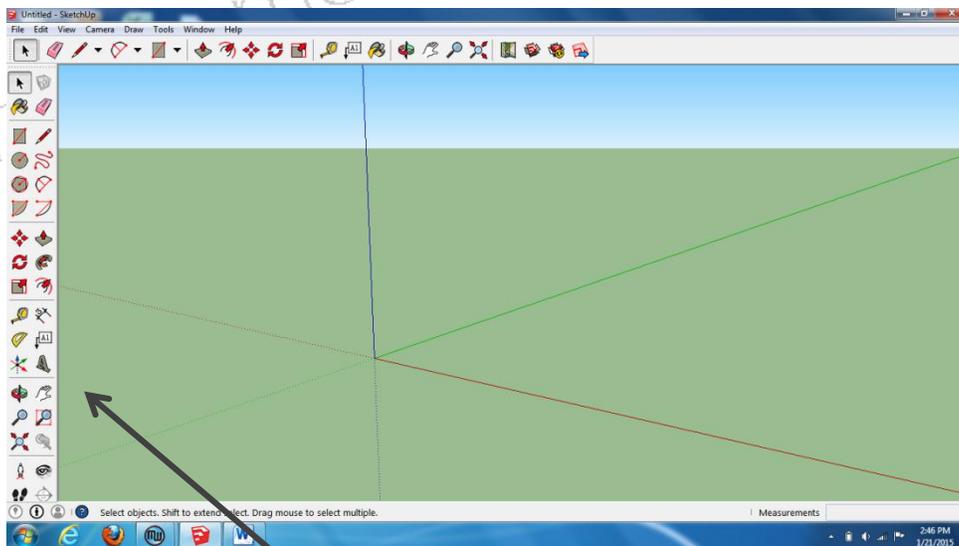
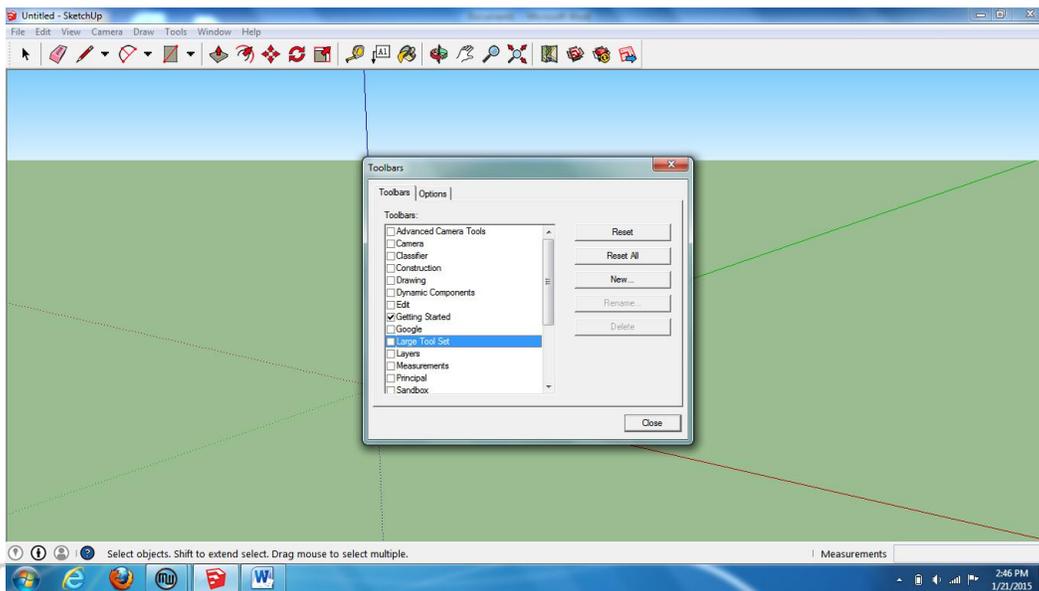


Creating 3D Letters in Sketch-Up

If you're looking to create custom nameplates, badges or just letters that will easily print in your 3D printer, look no further! Follow this step-by-step guide that will show you the tricks to getting letters designed and printed. Remember CTRL-Z : it erases your last step! You'll need it!

Instructions:

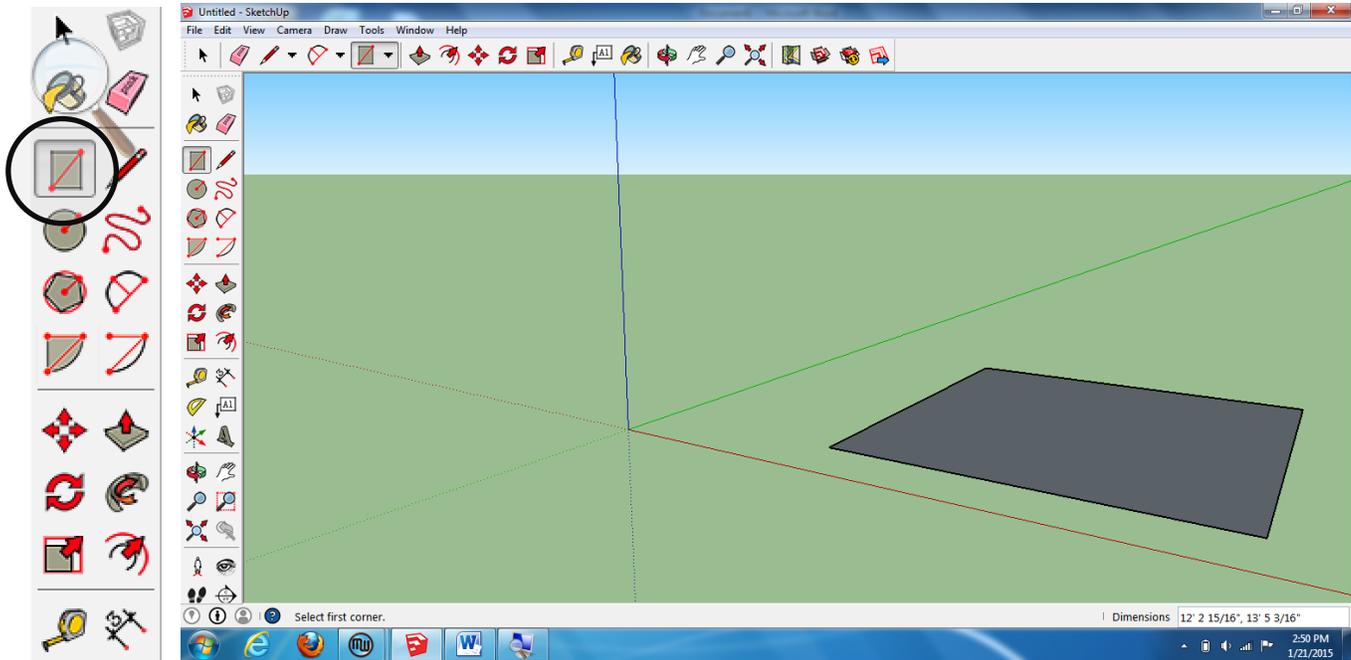
1. Open up Sketchup. Any version will work (this tutorial is using Sketchup 2014). If your window looks like this, you're in the basic window setup and are not seeing the full set of tools you could be using. You will want this full set of tools. To get these tools, click VIEW > TOOLBARS > LARGE TOOL SET.



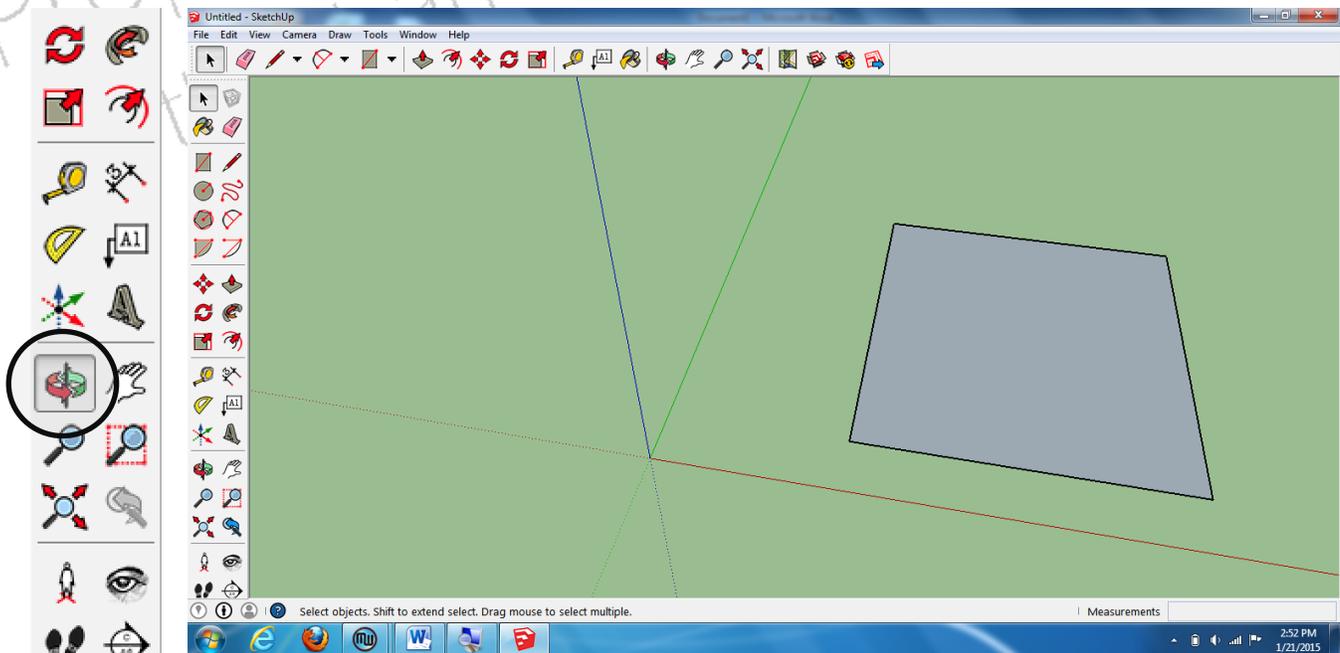
You should now have the **large tool set** in place (left side of the screen). This toolbar gives you access to a wider array of tools. You are now ready to begin designing.

2. To begin designing letters, you will first want to draw a one-dimensional plane or back-plate on which to draw your letters. This will be your canvas. To do this, choose the RECTANGLE tool.

Next, draw a rectangle. Make the rectangle larger than you think you'll need. You will be putting letters onto this shape.

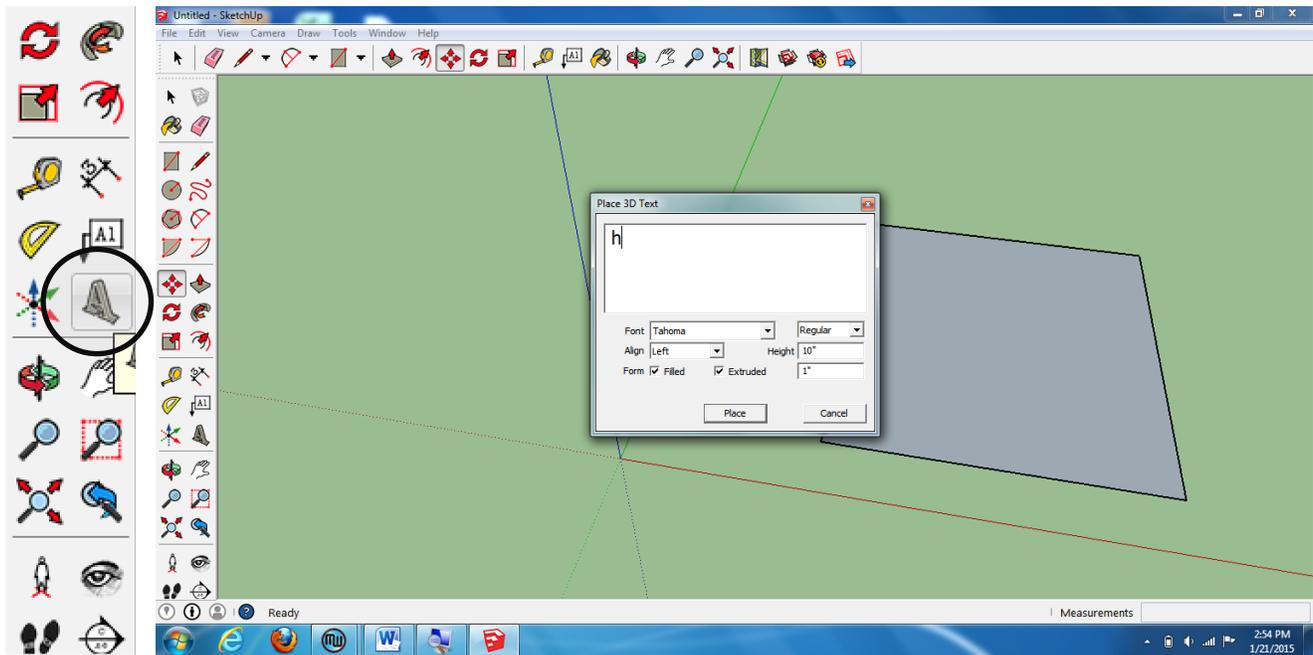


3. You'll notice we are looking at the rectangle from an odd angle. To change this, we're going to use one of the navigation tools. Choose the ORBIT tool, click anywhere in the frame and move your mouse up and down. This should tilt your screen up and down so that you are looking at your object from above or below.

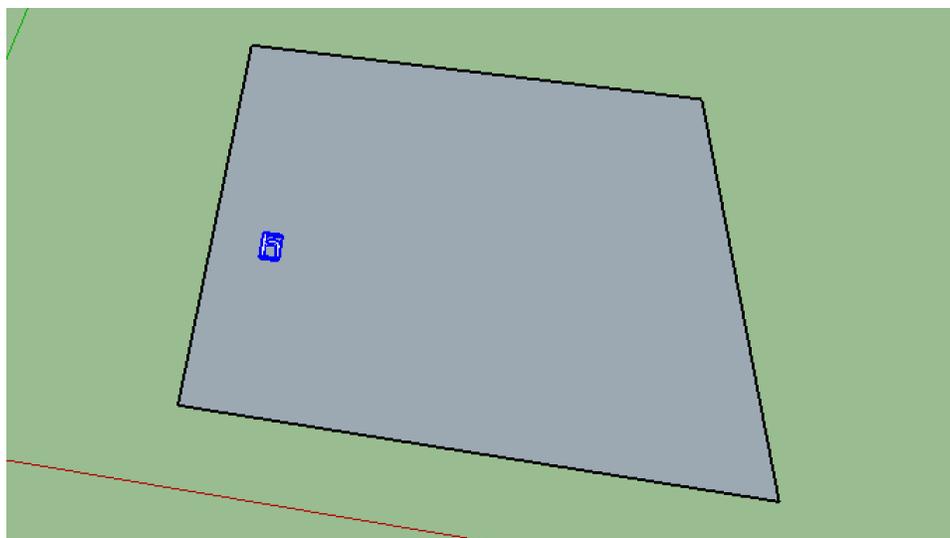


- Now that we've tilted the screen, we're ready to begin placing letters onto the rectangle. Choose the 3D TEXT TOOL (looks like a capital "A"). A box will appear giving you options to type in text. There will be options to change the font, alignment and height of the text. My recommendation is that you type in ONE LETTER at a time. This will save you a headache at the end. My other recommendation is to choose a boxy, thick font. If the font is too thin, it will be difficult to edit in the end and your print will be less structurally sound. **You will never have to change the alignment and height.**

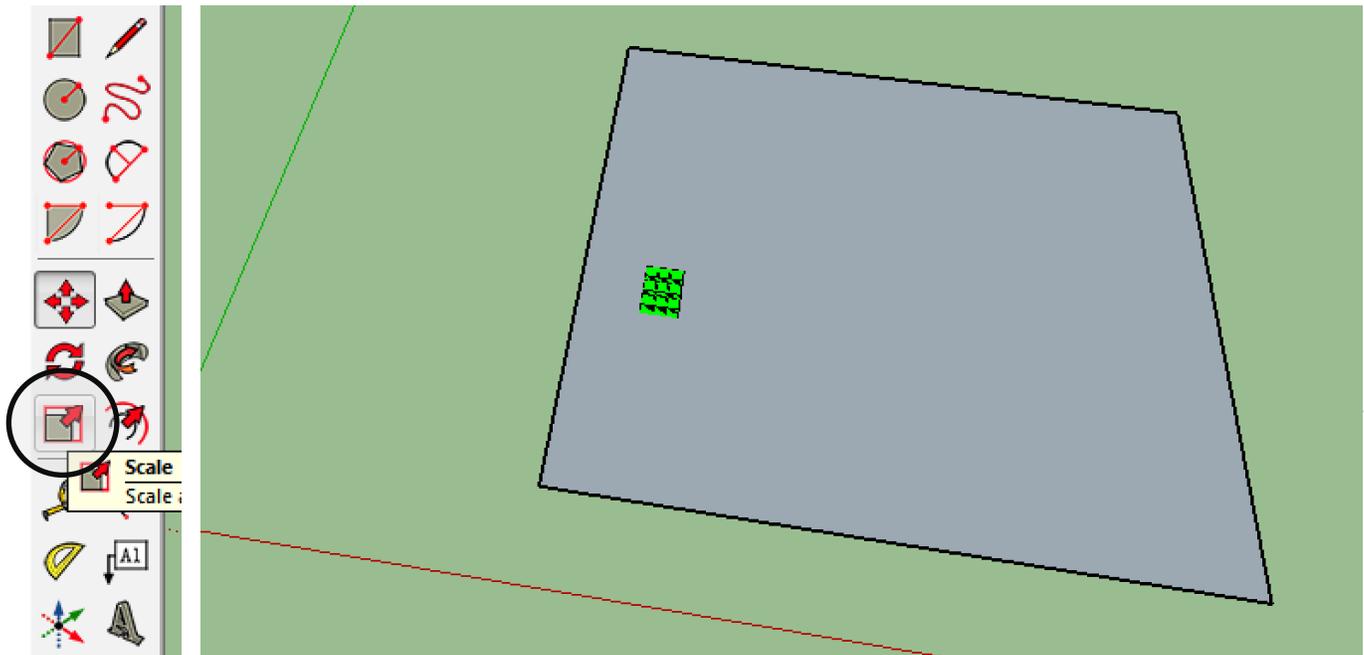
When you are ready to type your letter, type your letter and press PLACE.



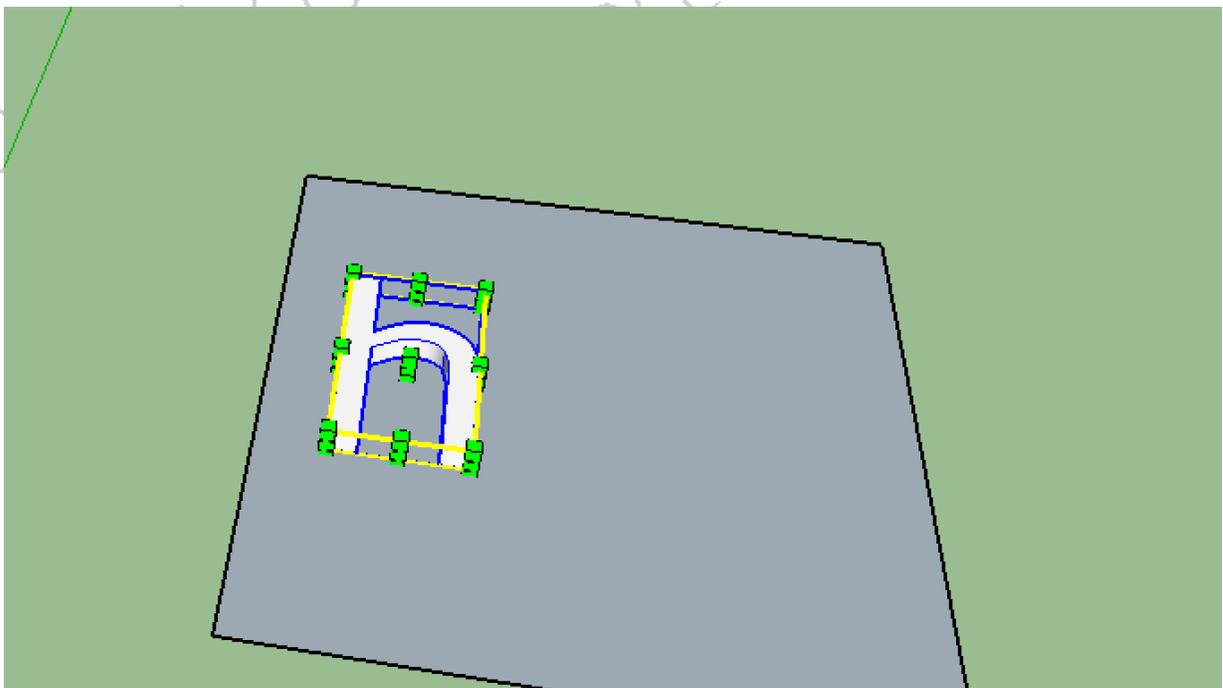
- Once you've hit PLACE, your letter, highlighted in blue, will appear—it will look like it is hovering. You can move it around anywhere you like on your rectangle and once you've picked an appropriate spot, click and release and your letter will be cemented to the face of your rectangle. When choosing a spot to put your letter, keep in mind that you are making a series of letters, so as far to the left as you can get is most likely best.



- Looking at this letter, it seems a little small, and it is. So, we're going to scale it up a bit. Choose the SCALE tool (looks like a square with a red arrow coming out of the right hand corner). Hit the SCALE button and select your letter. It should highlight itself in green boxes.



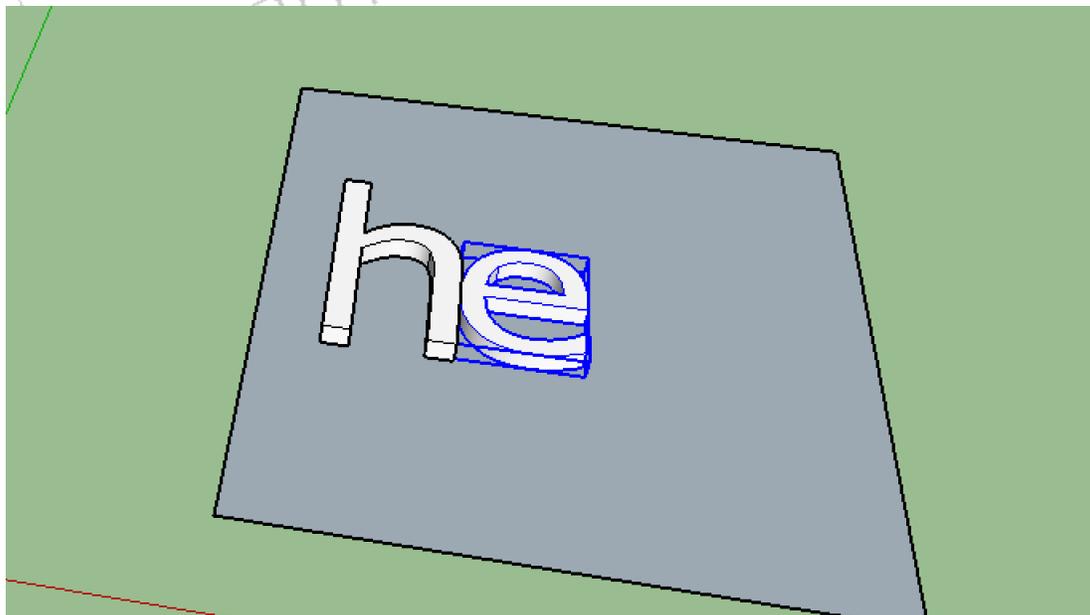
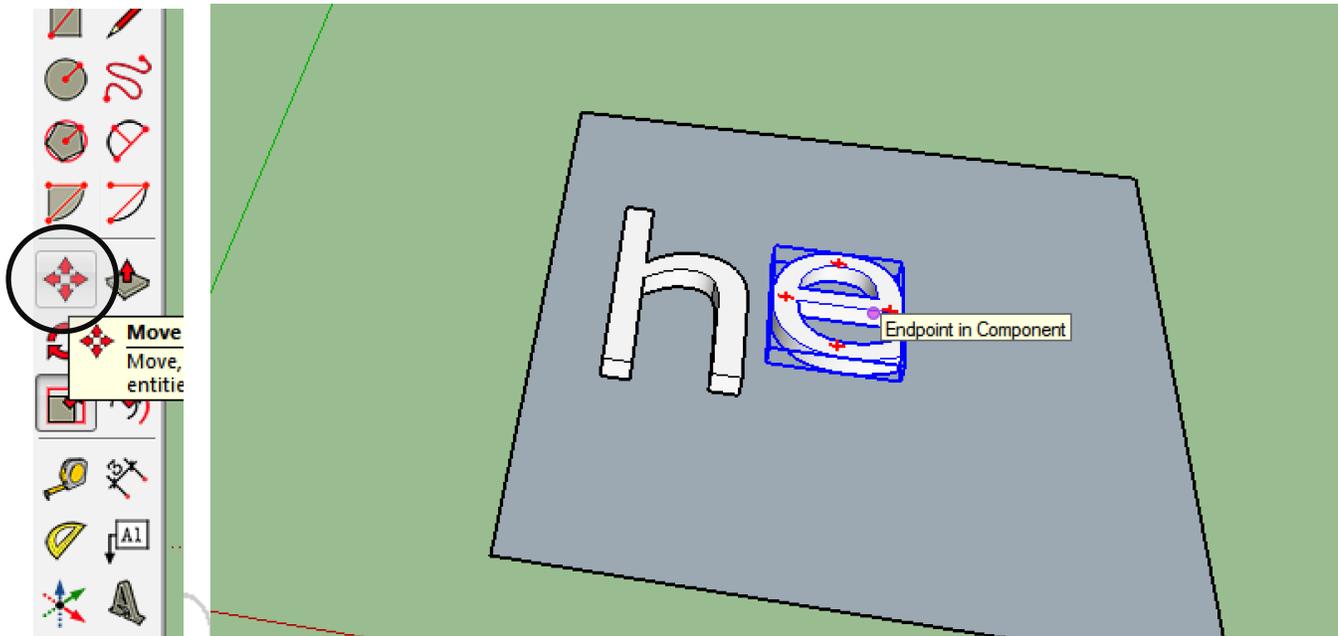
- Next, choose the right hand corner of your letter and pull and drag it until your letter is a more appropriate size (you judge what size is good for you).



- Repeat steps 4-7 with the remaining letters you want.

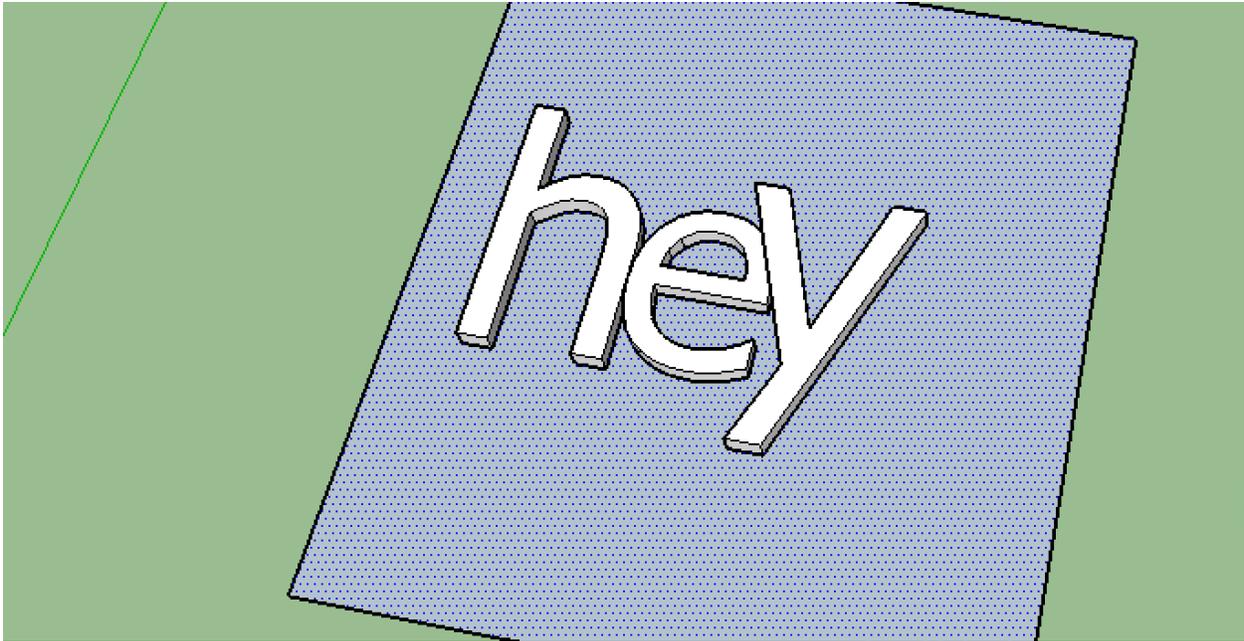
- Now that all of your letters are created, you will have to go back and arrange these letters so that they are right next to each other and overlapping a little bit. You can choose to do this one of two ways. Either you can create your letters one at a time (create a letter, scale it, move it, create a letter, scale it move it, etc.) or you can create them all at once, scale them individually and move them all separately. I prefer to them one at a time. Either way, you will eventually have to move them. This is what I mean by that.

To move a letter, chose the MOVE tool. Select the MOVE tool and then select your second letter. It should highlight itself in blue with four red arrows on all four sides. Now, with the letter selected, move it, so that it is right next to your first letter (almost but not quite overlapping).



You will repeat step 9 with every letter you have. Make sure to place them next to each other, almost but not quite overlapping (like the E and Y below).

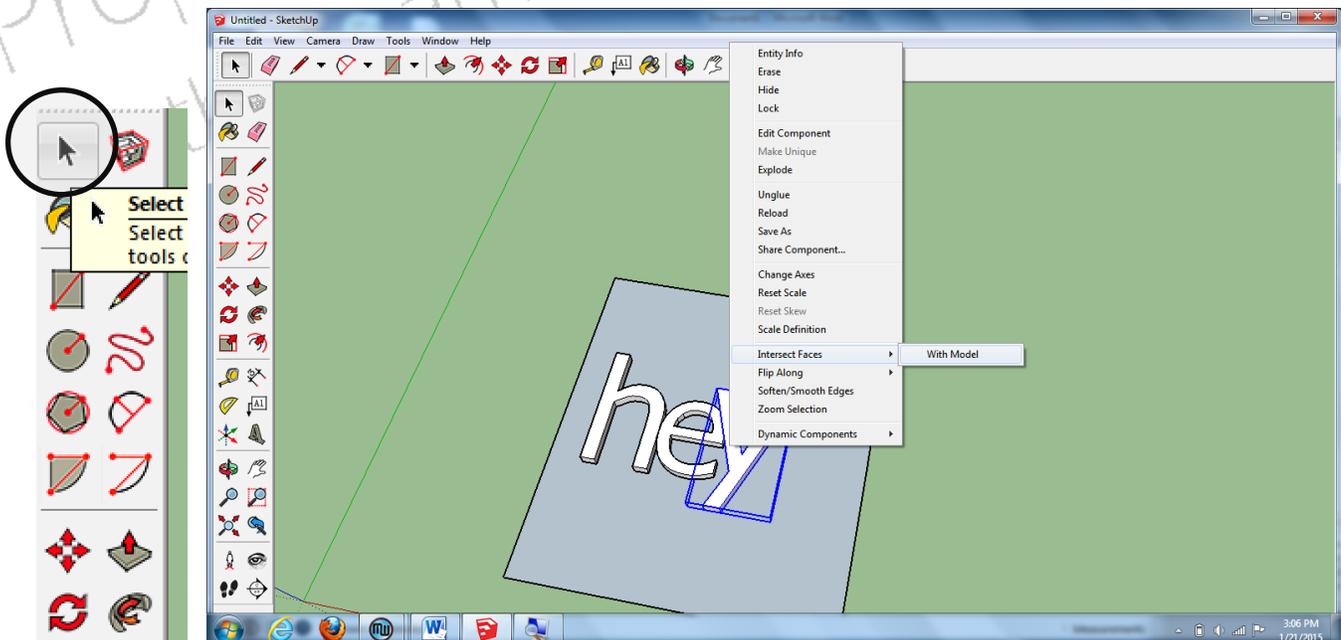
Your end result should look something like this (below).



10. Though it may look like your letters are ready to go, they aren't. They aren't actually connected in any way (not even to the rectangle below them) and if you were to print these, it would be unsuccessful.

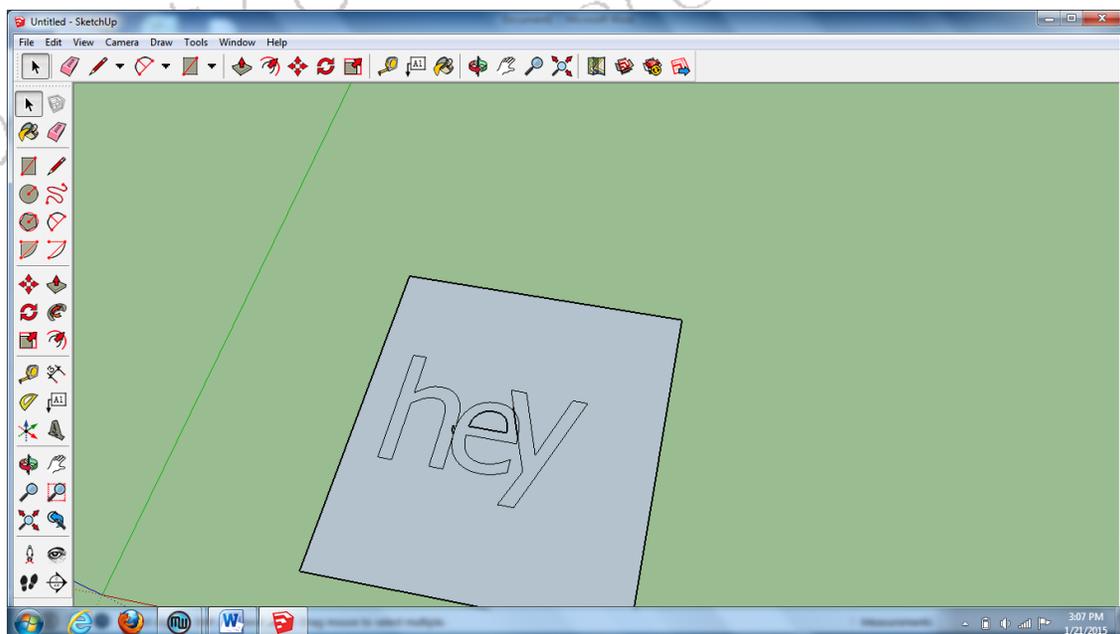
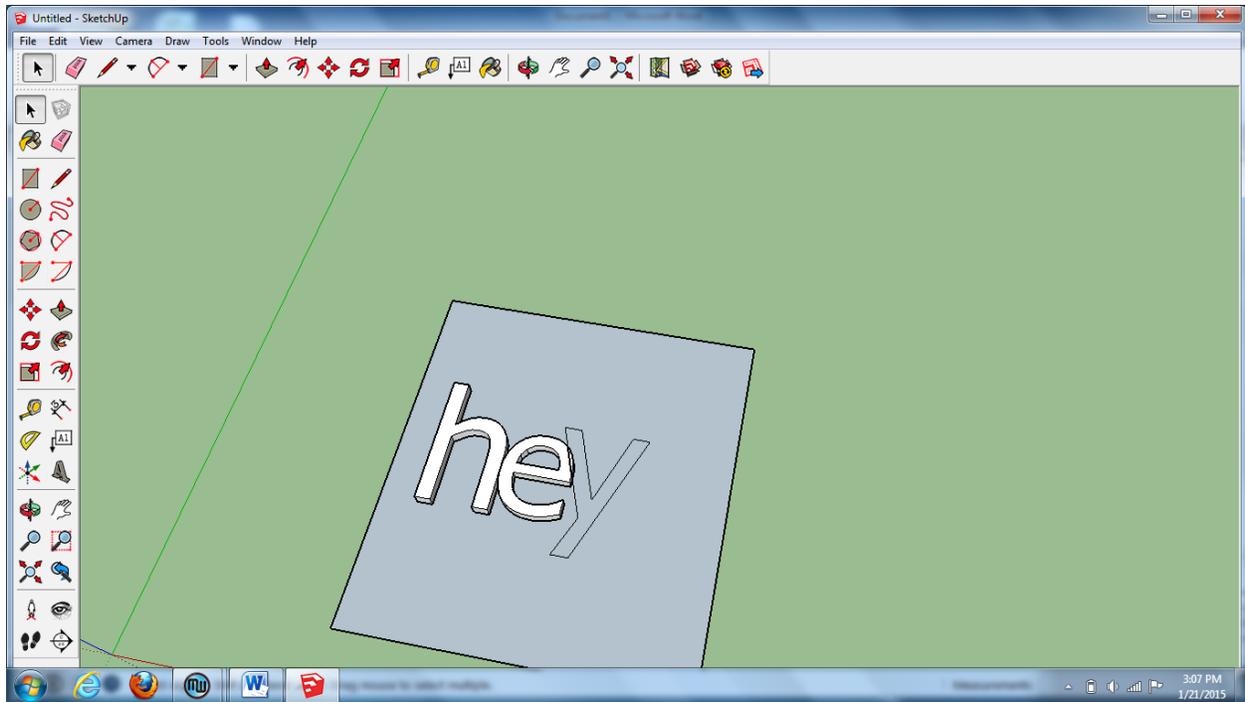
In order to get these letters connected, we need to intersect them with the backplate. To do this, we first need to select them. Choose the SELECT tool (the black arrow). Next, choose a letter. It should highlight in blue (a box will appear around it). Now, RIGHT CLICK > INTERSECT FACES > WITH MODEL.

IT will look like nothing happened but it did. The letter is actually now imprinted on the rectangle behind it. Repeat this process with all of your letters.



11. Once you have imprinted all of your letters onto the backplate, you will now delete them. Choose the SELECT tool again, choose a letter and hit the DELETE (not backspace) key on your keyboard. The letter should disappear and leave an outline on your rectangle.

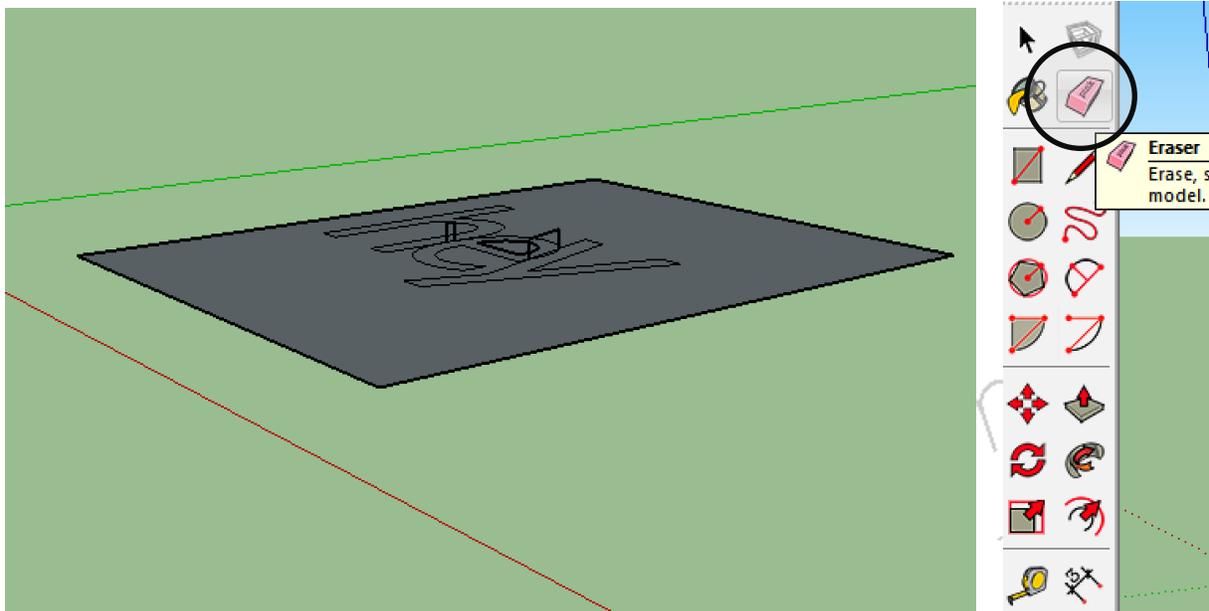
Repeat this step for all your letters.



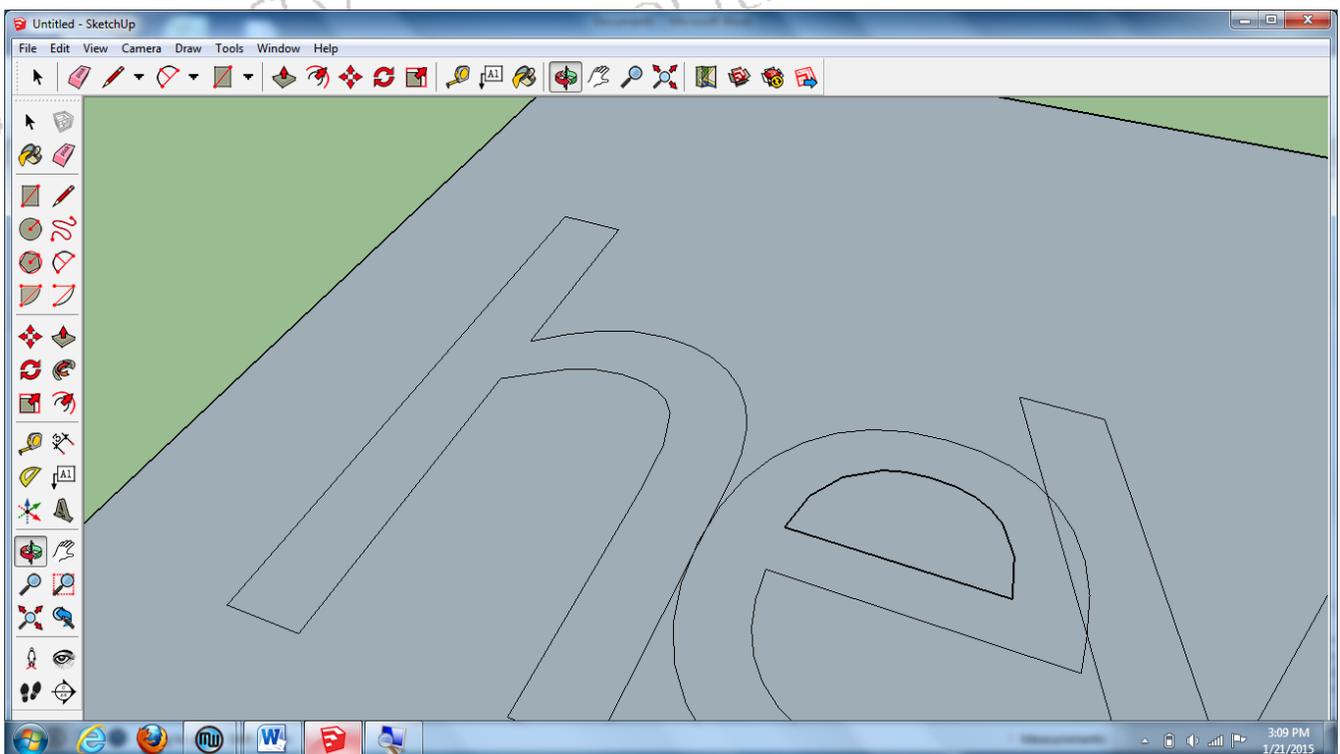
You should be left with an outline of your letters. You want your letters to be outlined in a light black, not bold. If things are outlined in bold or lines are sticking up into the air, there is a problem. It is a good problem to have because it means your letters have been properly meshed and just need to be scrubbed up a bit. The next few steps will show you how to fix these issues.

12. When letters overlap each other (as I instructed you to do), they will sometimes leave behind fragments that stick into the air. These need to be erased. To do this, choose the ERASER tool.

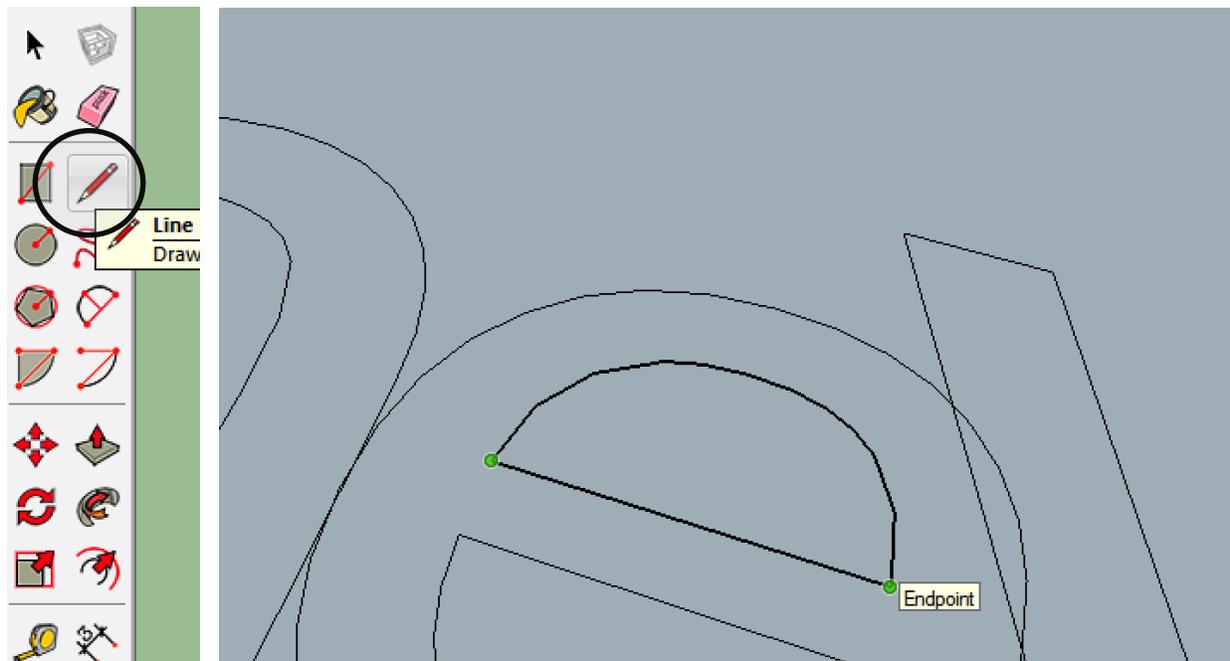
With the eraser tool selected, erase all the black lines sticking into the air. You can use the ORBIT tool to move around the frame and get to every spot.



13. The other issue we need to address is bold lines, as indicated here in the "E." In order to fix this, we simply draw over the line with the LINE tool. Select the LINE tool (looks like a pencil).



With the LINE tool selected, go to a point on your bold letter and draw from one endpoint to the other.



The letter should snap from being bold to being a plain black outline like all the other letters.

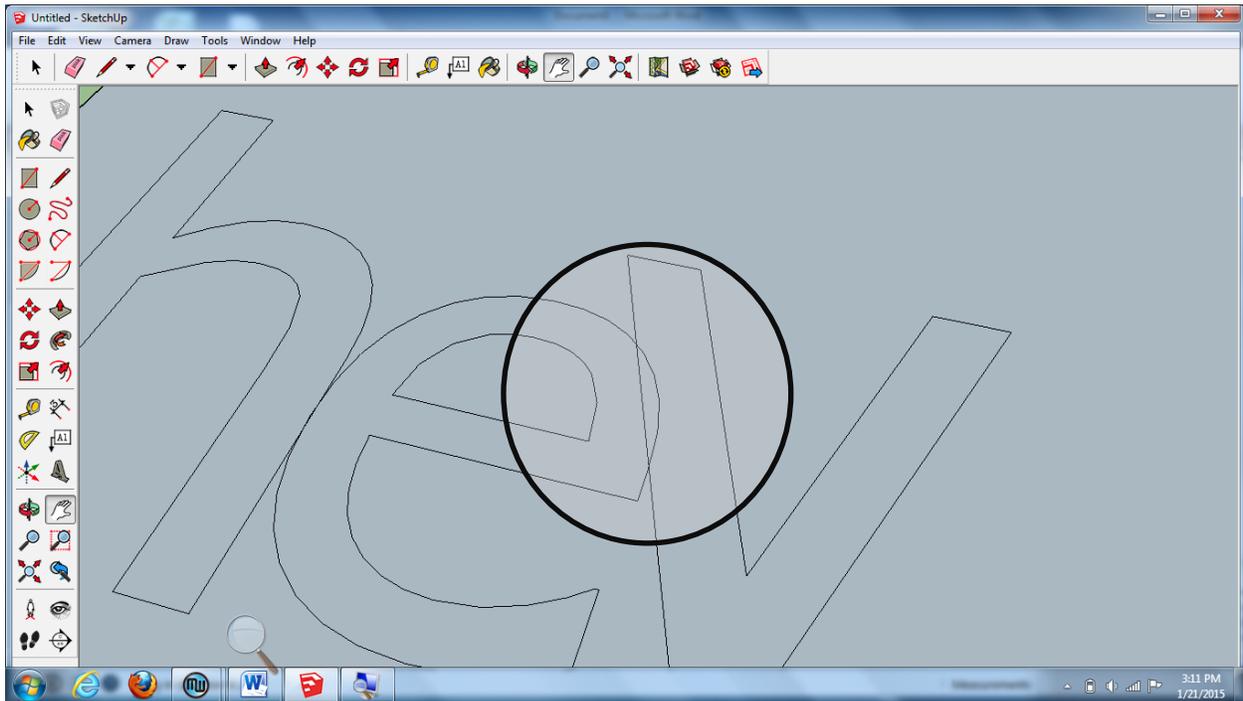


Repeat this process of drawing on all the bold lines until all the letters are uniform in color.

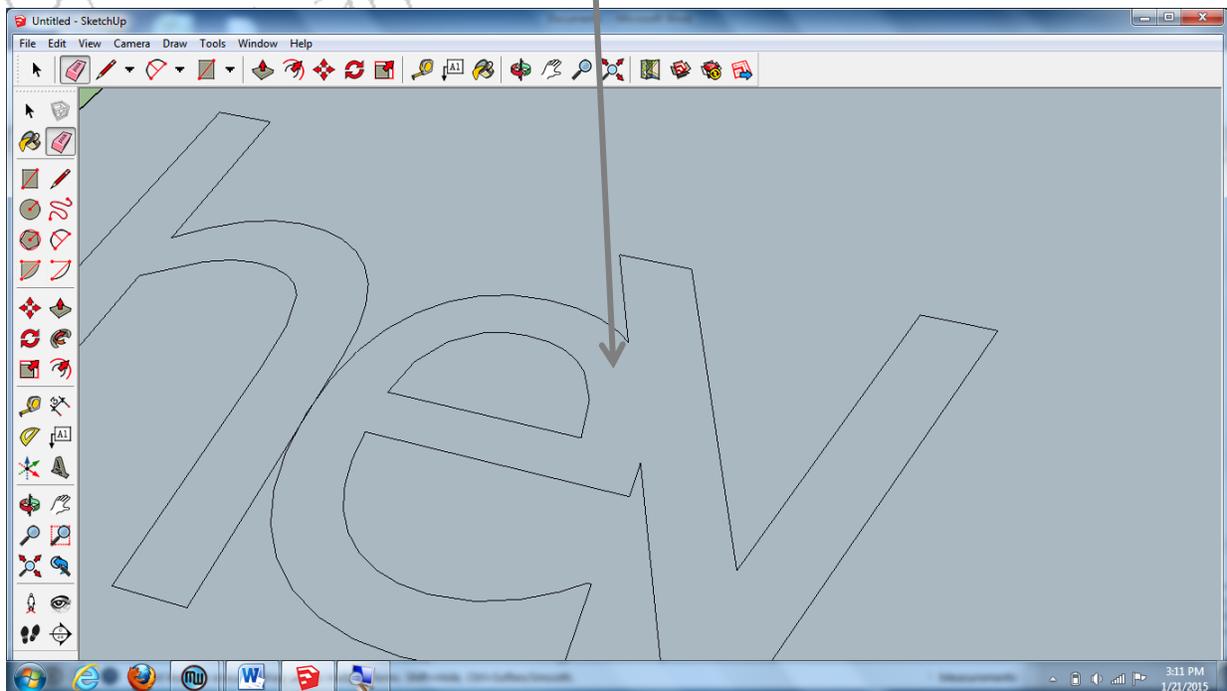
14. We now need to get the letters connected so that they will print as a unit and not as separate letters. Notice that the "E" and "Y" have an overlap. We need to delete this. Also, notice that the "H" and "E" are not connected in any way, they are just touching. In this instance, we need to build a connection.

When making nameplates or nametags or letters that will be printed together, this is a very important step. You are looking for places to build connections, deleting overlaps and strengthening weak connections if they exist. If, for instance, this were to hang from someone's nametag from the "H," I might consider building the "H" to be thicker so that it wouldn't break as easily.

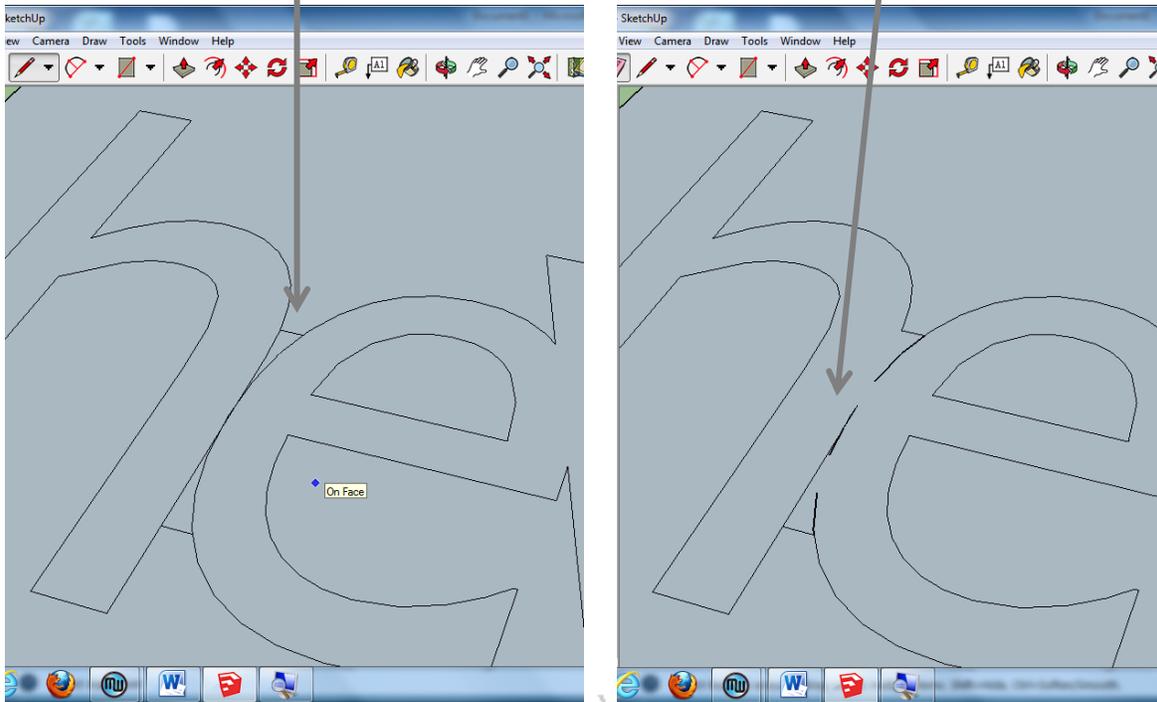
The tools you will be using here are the ERASER tool to erase overlaps and the LINE tool to create connections.



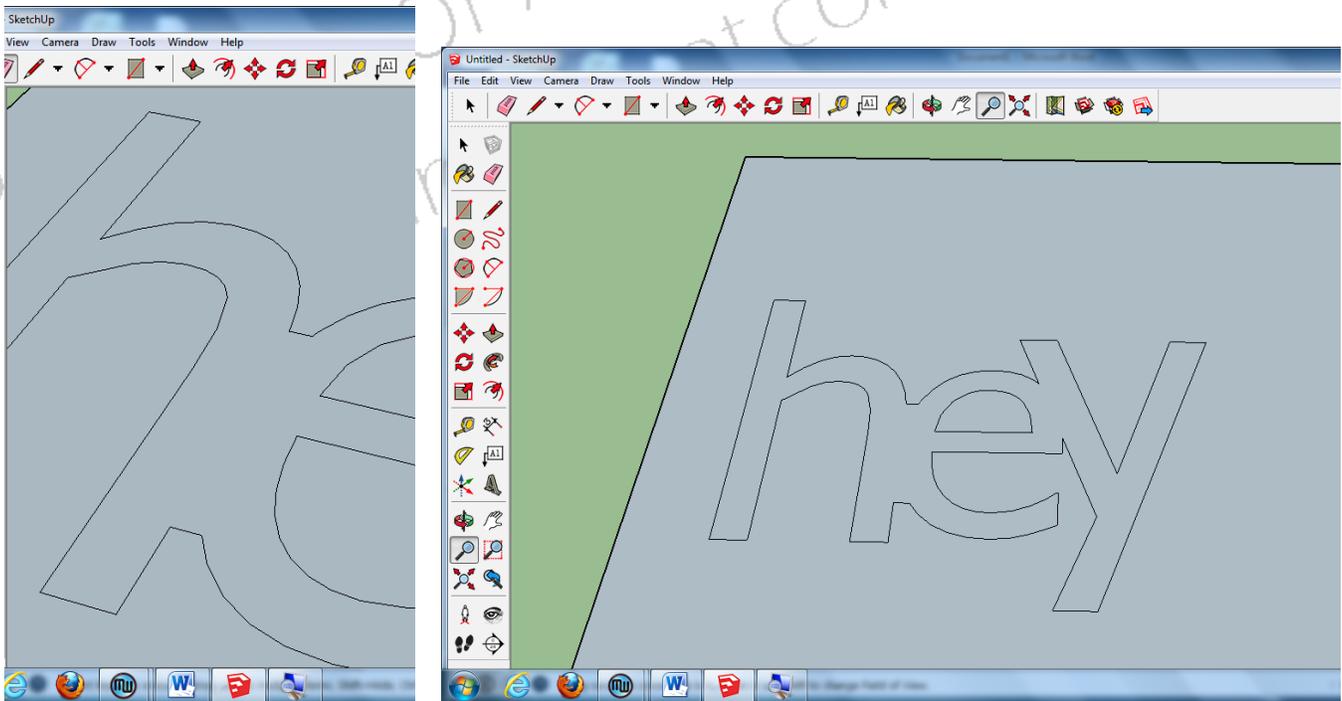
No more overlaps!



A connection or two goes in between the "H" and the "E." There are now weird overlaps I need to get rid of.

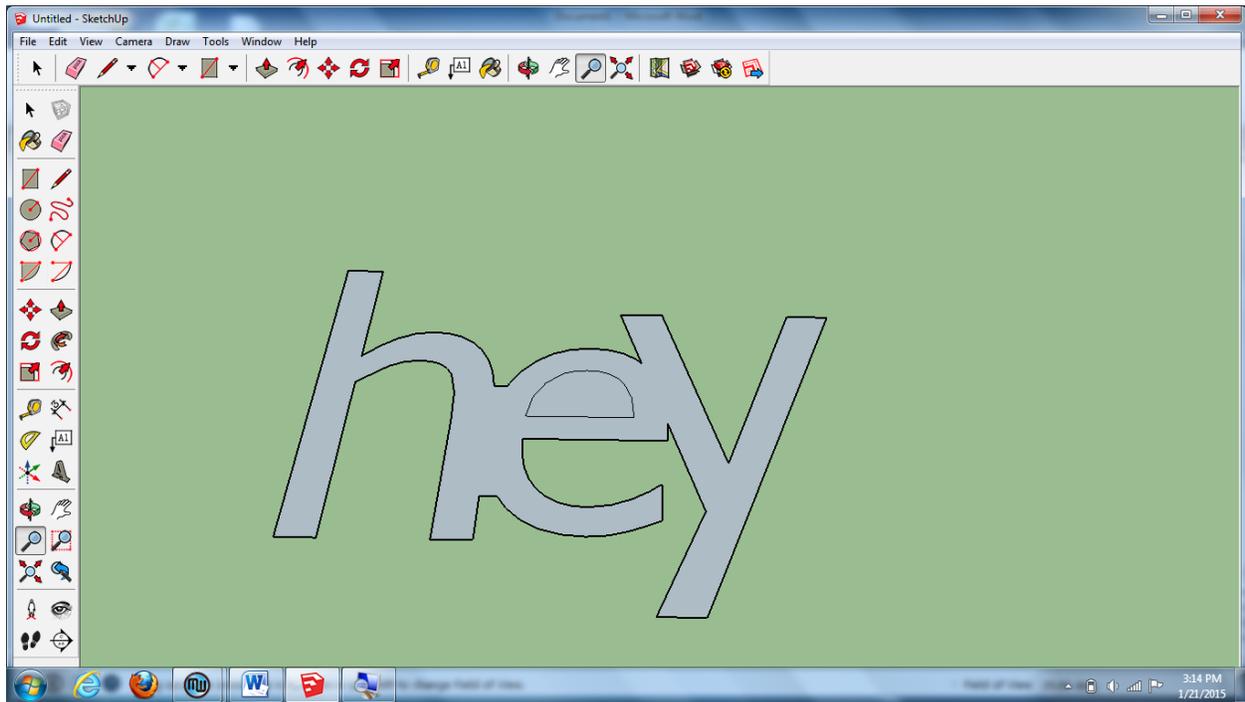


When I begin deleting those overlaps, I get the bold lines. That's because Sketchup sees this as a missing connection and wants me to fix it. If I keep deleting and complete the loop, it will all be okay.

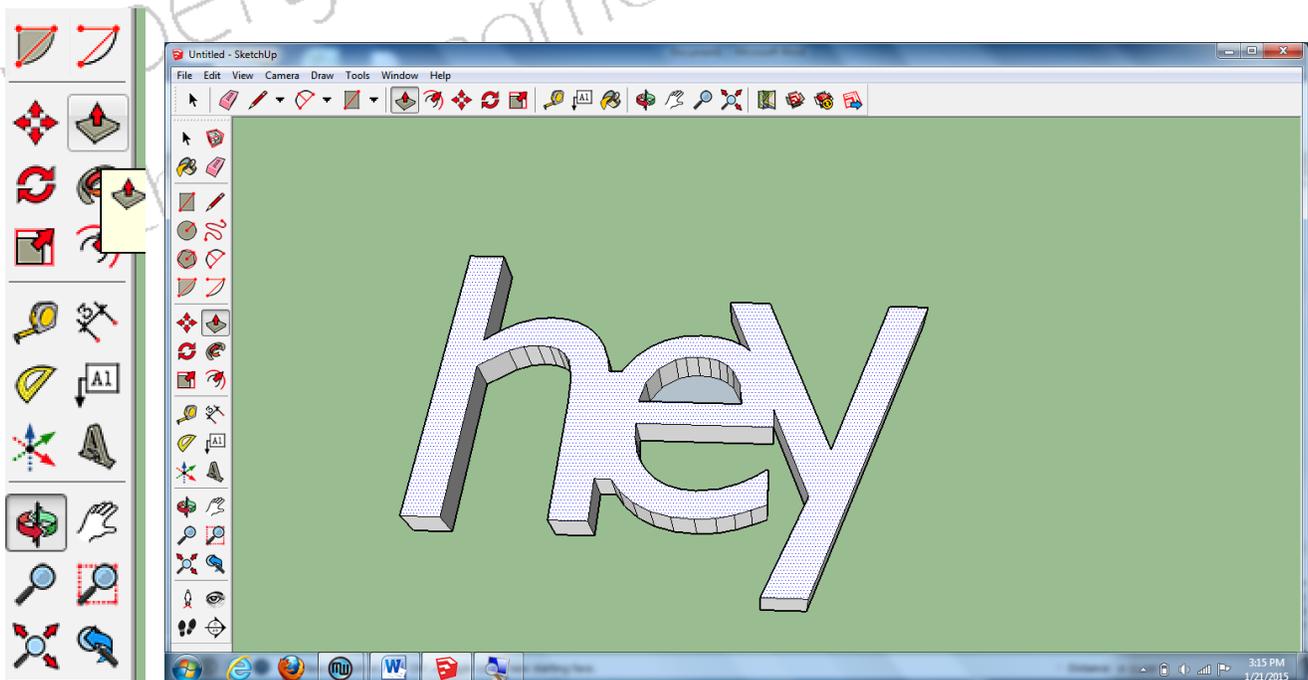


I should end up with letters that connected and outlined like this (above).

15. Now, I need to actually make them 3D. First, let's get rid of the backplate because we don't need it anymore. Select the ERASER tool and delete all four sides of the rectangle. Once it's gone, your letters should look something like this.



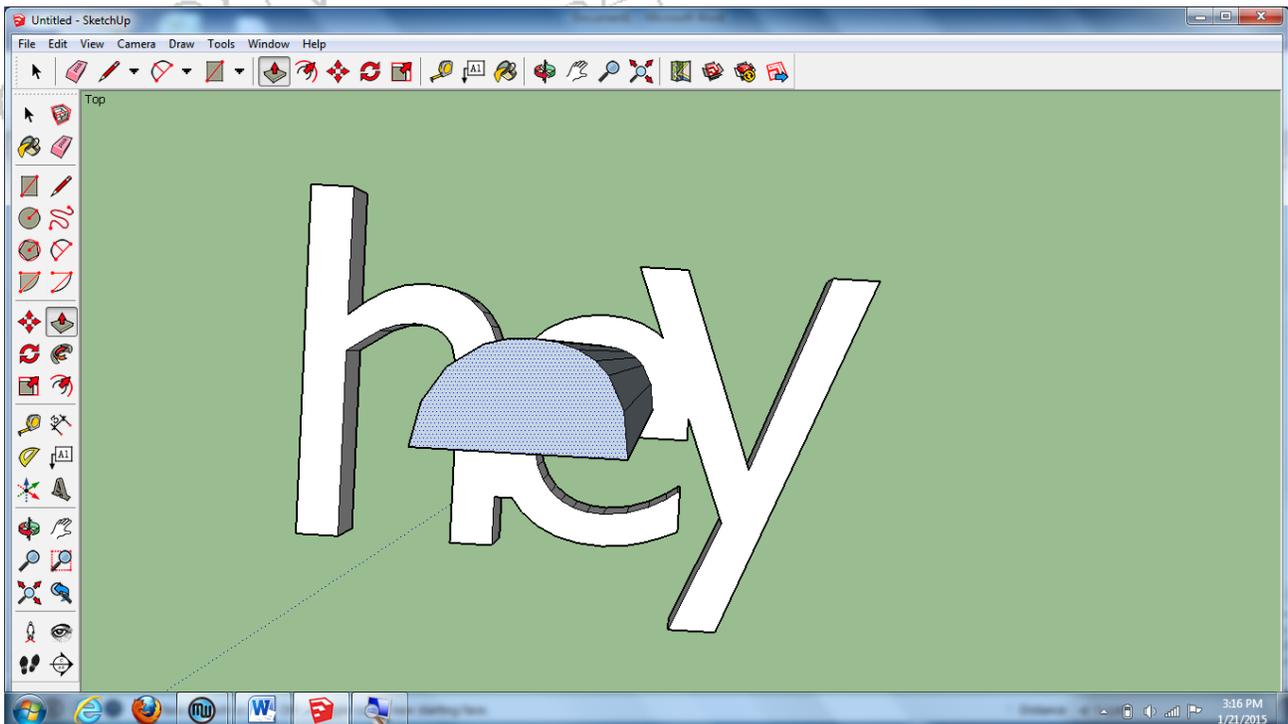
16. Next, we will make them 3D. Select the PUSH/PULL tool. With the PUSH/PULL tool selected, hover over your letters (they should highlight with polka-dots) and pull towards you. Your letters should appear 3-Dimensional. You can make them as thick or thin as you would like.



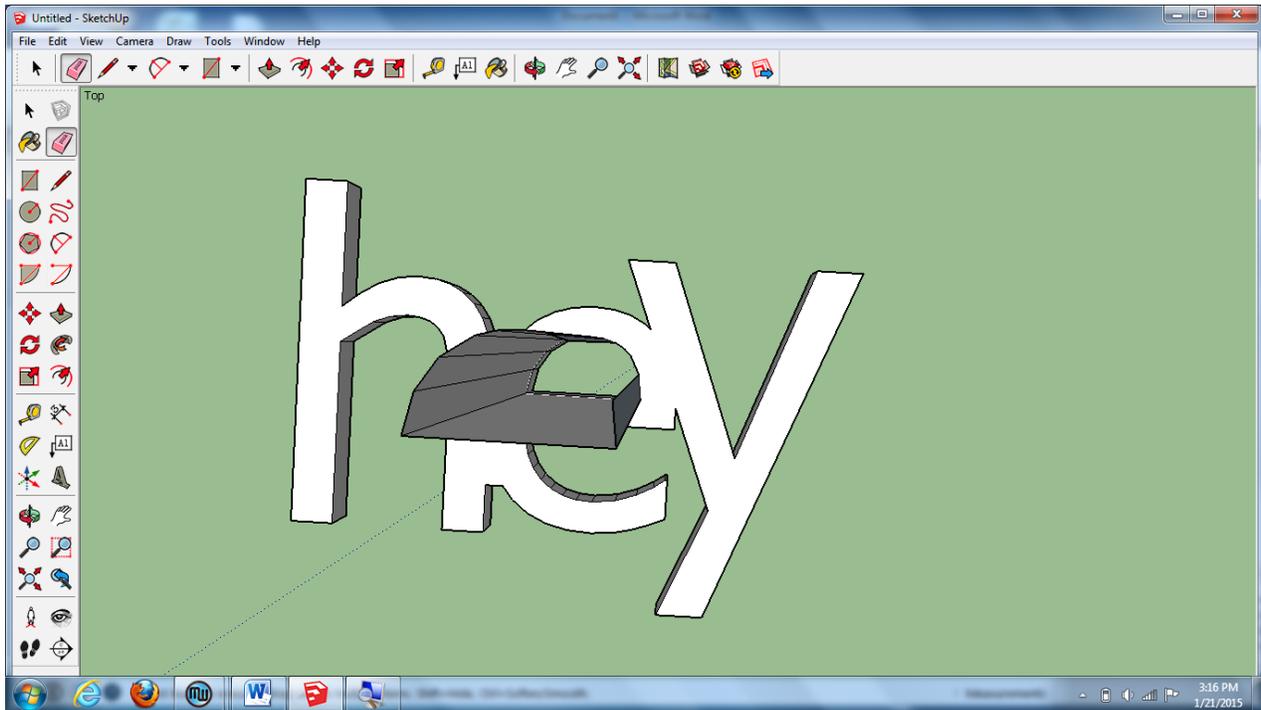
17. Notice, that there is a blue spot in the middle of the “E.” That is because it is not cut-out yet (a left-over from the backplate) and we need to get rid of it. This will happen with letters like “O,” “A,” “P” or any letter with a hole in it. If this happens to you, select the ORBIT tool and rotate your letters so that we are looking at your letters from above.



Now, choose the PUSH/PULL tool and hover over the blue space in the middle of your letter.” Pull it toward you.



Next, choose the ERASER tool and erase the blue space until it is gone and you are left with a space in the middle of your letter that is normal.



18. Final Steps: If you are using this as a name tag, you can now add embellishments like shapes or cut-outs to your letters. To make a circle (to attach to a clip), simply, select the CIRCLE tool, place your circle where you want your cut-out to be and follow what you did in step 17.



Congratulations! Letters accomplished!

Additional: Printing Your Letter Creation

1. When you want to print your creation, there are a couple of things you will want to remember. First, your 3D file will need to be saved in a particular format—a format that the 3D printer can read. This file type is an STL file. To save in this format, choose, FILE > EXPORT STL.

In this window, you have the option to save your file to wherever destination you choose. I would suggest saving to a flashdrive, unless your computer is connected directed to the printer. In this instance, it would be advisable to save to the desktop (this would make it the easiest to find your file). Once you have picked the “EXPORT STL” option, choose your destination, and then name your file. After the name of your file, you will add the extension: .STL. For example, if my file name was ashley3dcar, I would add the extension .stl to make it: ashley3dcar.stl. Adding .stl at the end is VERY IMPORTANT. If you don’t add it, your creation won’t print.

Once your file is saved to the flashdrive, you are good to print your object!

Property of Ashley Hamernik
thisahamoment.com © 2015