FabShops: Intro to 3D Printing
## SketchUp Make Tutorial
By Adrian Yao

SketchUp Make is a free 3D computer assisted drawing/modeling (CAD/CAM) tool that is open for anyone to download and use; it is the free version of SketchUp Pro—the paid version. While the capabilities of SketchUp Make are limited, an advantage that it supports software extensions written in the Ruby programming language. These extensions are free and available on Ruby script forums, and thus make SketchUp a dynamic, customizable, and constantly evolving tool for anyone.

In this tutorial, you will make an I-beam structure that has several features. This should teach you the most basic tools available and relevant to you in SketchUp and 3D printing. If you would like to learn more, there are a ton of tutorial videos online that are helpful.

Basic Tools:

| Icon	Tool (shortcut key)	Description
	Rectangle (r)	Draw rectangles on a plane. Start by clicking any point to be a starting corner and the diagonal of the rectangle will follow the cursor. Click again to release point. To draw exact dimensions, start by clicking on a start point, then type in "x , y" units for dimensions and press "enter." Hold down "shift" to draw perfect square.
	Circle (c)	Draw circles on a plane. Start by clicking any point to be the center of the circle and the radius will follow the cursor. Click again to release point. To draw exact dimensions, start by clicking on a start point, then type in "r" units for radius and press enter. (Note this is not an ellipse tool but a circle tool)
	Polygon	Draw regular polygons on a plane. Start by clicking any point to be the center of the polygon. Then type in the number of sides you wish for the polygon to have. The radius to the apex of the polygon will then follow your cursor. Then follow the same instructions to use the Circle tool.
	Arc (a)	Draw arcs on a plane. Start by clicking to designate the two end points of the line segment, then the amount of bend will follow your cursor. Click again to release.
	Push/Pull or Extrude (p)	Convert 2D surfaces into 3D shapes by "pushing" and "pulling" surfaces in a direction perpendicular to the planes. Start by clicking a surface. The extrusion will follow your cursor. Click to release and create solid. To push/pull exact dimensions, click a surface and take your cursor in the direction you would like to extrude, then type in "d" units.
	Move (m)	Move objects by clicking and objects will follow your cursor. To move objects to exact places, you can type in "d" units after clicking object and taking the cursor to the direction you would like to move. Alternatively, you can use the cursor to snap objects onto points of other objects.
	Rotate (q)	Rotate objects. Start by clicking the center of rotation and a click again to establish a zero degree line. The object will then rotate about the center as a second angle line follows your cursor. Click again to release. To rotate exact dimensions, type in "r" in degrees and press "enter."
	Scale (s)	Scale objects. Place your cursor over an object and drag "handles" will appear. Click and scale objects as necessary. Hold down "shift" to preserve proportions. Hold down "control" or "option" on mac to scale about the center of object. To scale objects to exact ratios, start by clicking a handle, and entering in "r" for a ratio, then press "enter."
	Tape Measure (t)	Use to draw guidelines to help with modeling. To draw an infinite construction line: Start by clicking an arbitrary point on a line (either from axes or from object) and an infinite construction line parallel to the line you clicked will follow your cursor. Click to release line or enter exact distance away from line. To draw a guide segment and point: Start by clicking an endpoint or corner and a guide line segment will follow your cursor. Click to release or enter exact distance away from point to draw guide segment. A guide point will then appear.
	Eraser (e)	Use to erase lines, guidelines, or objects. Click and drag over lines to erase and they will be deleted upon release.
	Orbit (o)	Use to orbit around object you are creating. Use only if you do not have a mouse. Otherwise, click the scroll wheel on the mouse to orbit.

Part Tutorial:
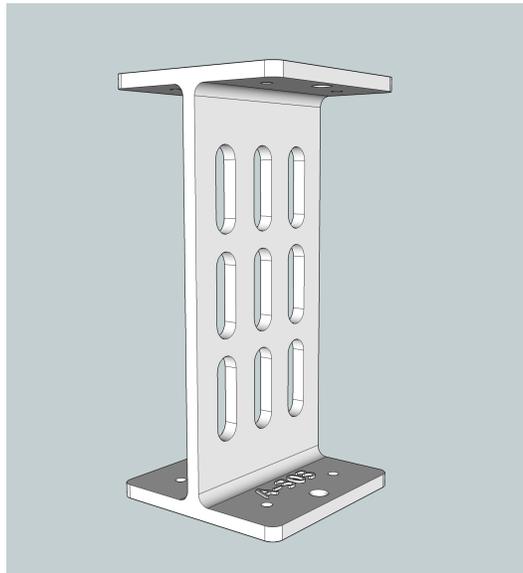
Starting

1. Open SketchUp Make and go to
    Preferences > Template > "Product
   Design and Woodworking – Millimeters"
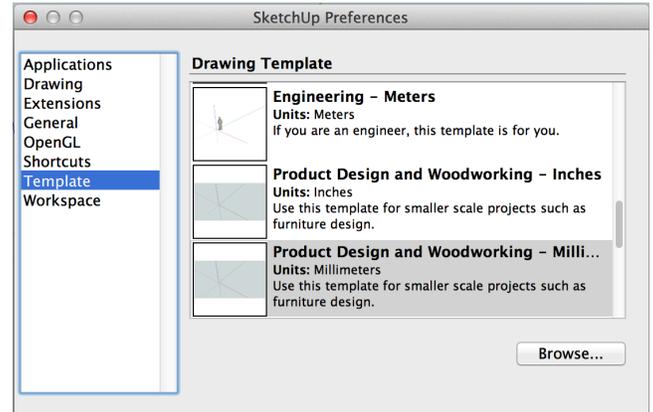
2. Close current workspace and reopen new file. Workspace should have an aqua-tinted
    background

3. Start by drawing a rectangle of 70mm,
    70mm and extrude it up by 5mm.
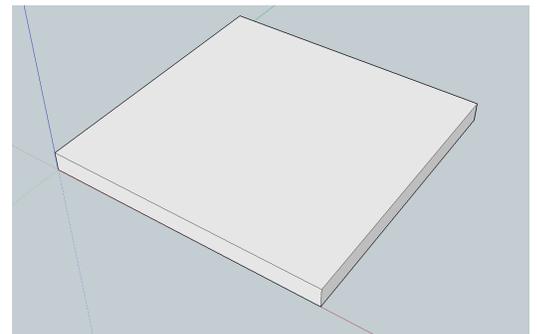
Creating Through-Holes

1. Use the tape measure tool to provide intersection points where through holes will be located.
    Draw infinite construction lines 15mm from each side onto the surface of the block.

2. Then click the circle tool and use the intersection points of the construction lines as the centers of
    the through-hole circles. Draw four circles of 2mm radius at each intersection.

3. Cut the circles out of the block by extruding them into the block. Use the base of the block as a reference point for your cursor.
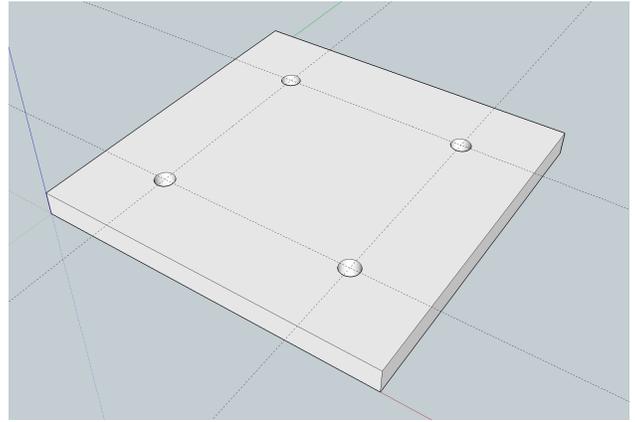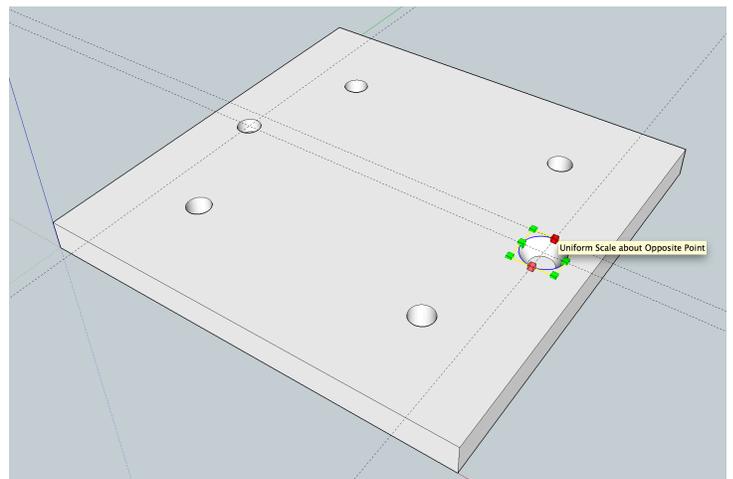
4   Erase construction lines as necessary. Voila! You have your through-holes!

Creating Countersunk Holes

1. Use the tape measure tool to draw a construction line across the middle of the block. Use the tape measure tool again to create an intersection point where the countersunk holes will be. These should be 7.5mm way from one edge.

2. Draw two circles of 3.5mm radius at those intersection points
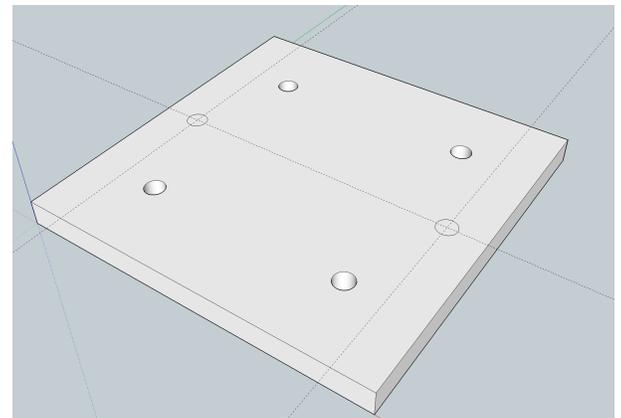
3. Cut the holes halfway into the block using the push/pull tool. Use a construction line if necessary or use the cursor to snap to the midpoint of the height dimension.

4. Create a construction line 3.5mm away from the center line

5. Highlight only the circle edge of the hole on the surface of the block. Then use the scale tool and hold down "shift" and "control" ("option" on Mac) while dragging the side handle toward the new construction line. This should expand the circle edge about its original center and create a cone-like countersunk structure.

6. Once you have the two countersunk cones created, you can then extrude the rest of the circle through the block.

7. Again, erase any construction lines as necessary and you have just created countersunk holes!
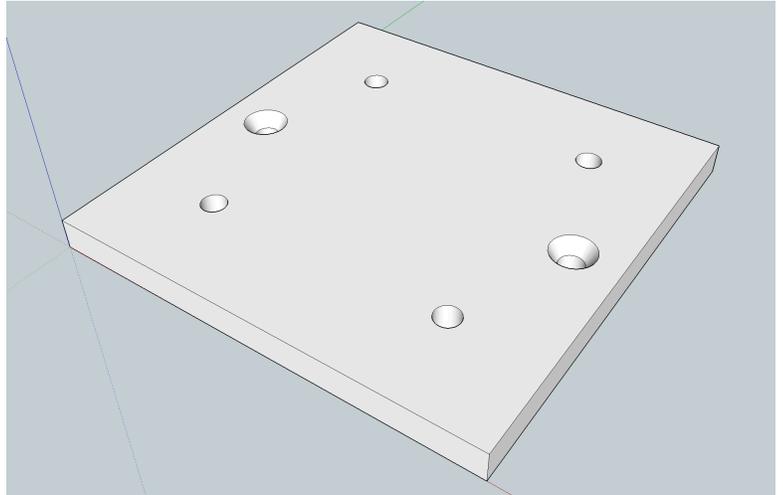

Creating the "T" Flange

1. Use the tape measure tool to draw a 5mm rectangle across the middle of the block.

2. Extrude the rectangle up by 70mm to create the "T" flange and erase any necessary guidelines.
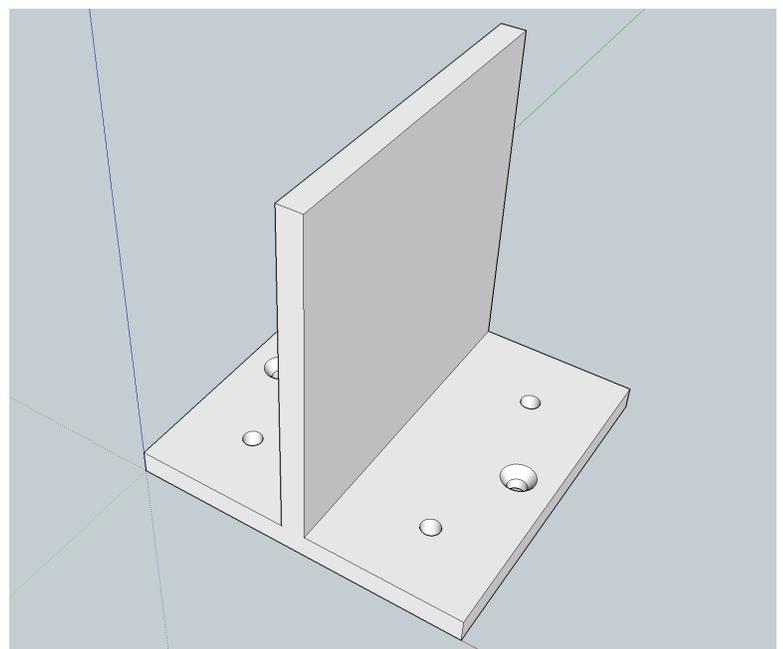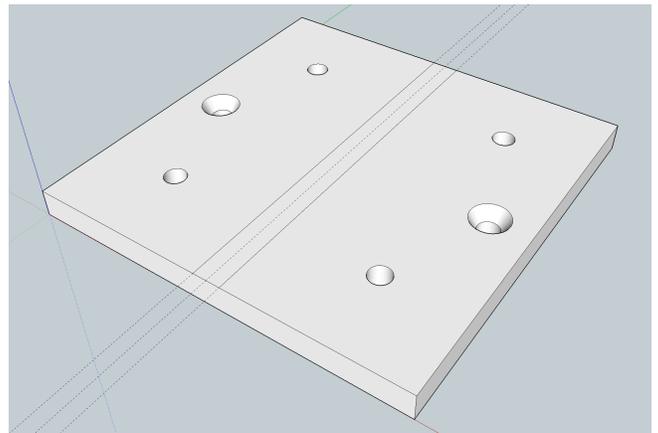
Filleting and Rounding Corners

1. First off we will round the corners of the base block. To do that, we will use the arc tool. Draw construction lines 4mm away from each edge on the base block.

2. With the arc tool using the construction lines as a guide, draw an arc tangent to the edge of the base block. Notice how the arc color will change at significant radii. Once the arc color has changed and a label indicates that it is "Tangent to Edge," click to release.
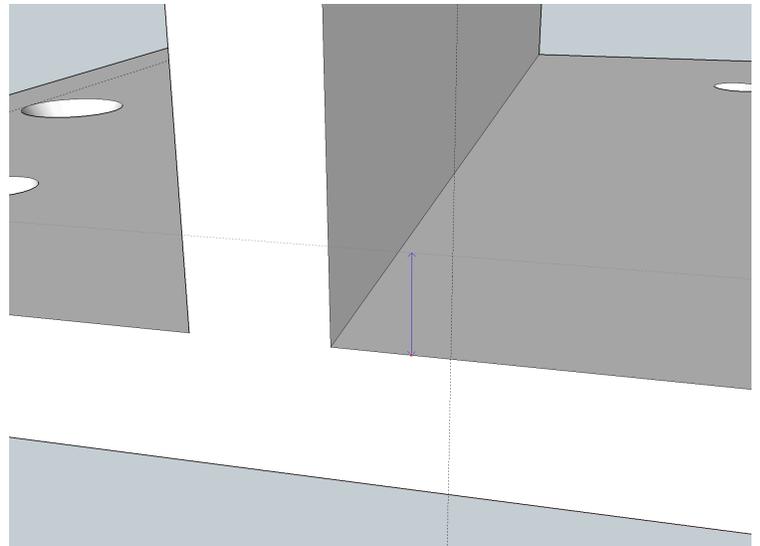
   Repeat this with all four corners.
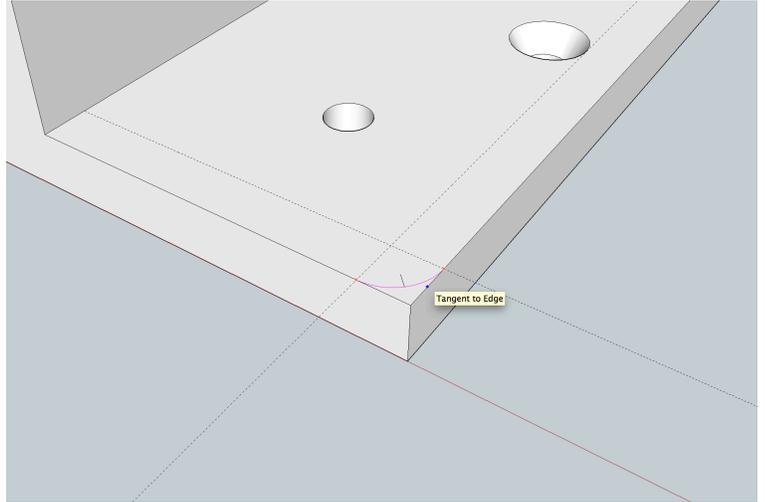
3. Extrude the unwanted corners away to round off corners.
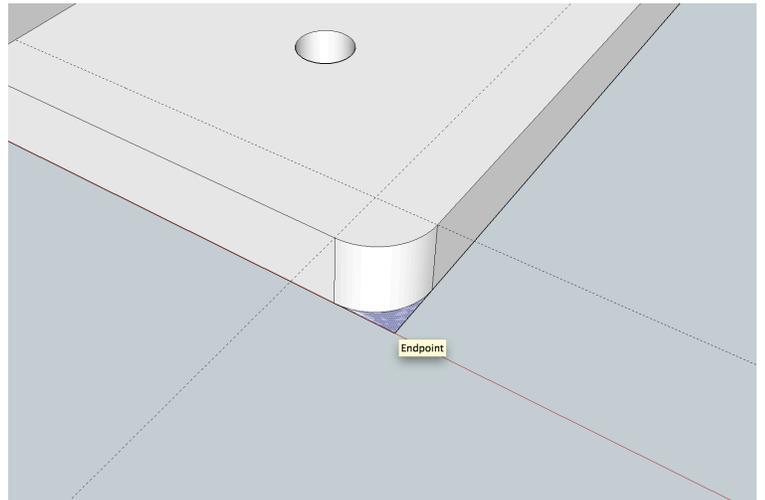
4. To fillet the flange, we will again draw an arc, but this time, we will draw an arc in free space.

5. Draw construction lines 4mm away from each edge of the flange. Take advantage of the colored axes to draw perpendicular to each edge.

6. After drawing the construction lines, we can again draw an arc, but this time in free space.
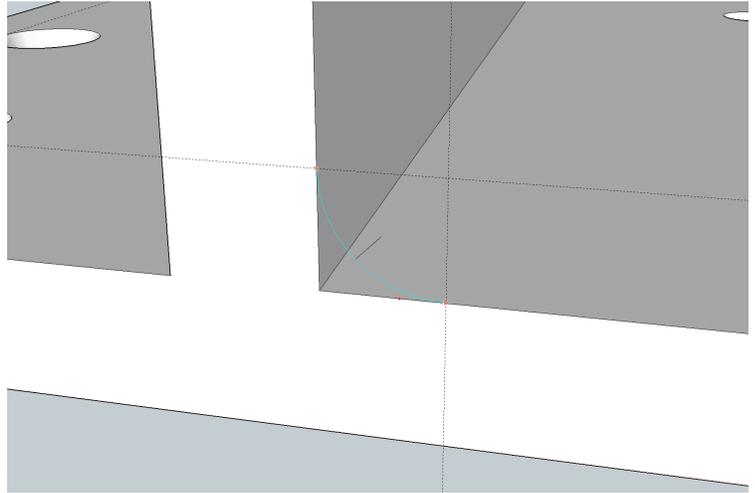
7. Extrude these new surfaces and erase any unwanted lines.
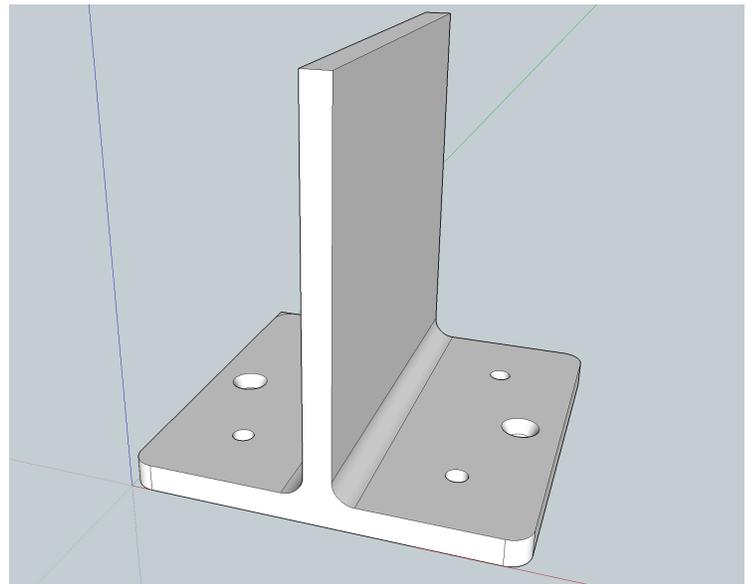

Mirroring the Flange

1. Now that we have half of the symmetrical structure built, we can mirror it about itself and join the two structures.

2. First triple-click or highlight the entire object and create a group by pressing "Control + g" ("command + g" on Mac).

3. Then, using the move tool, we will move the object aside. However, while doing so, press "control" ("option" on Mac) and this will create a duplicate object.
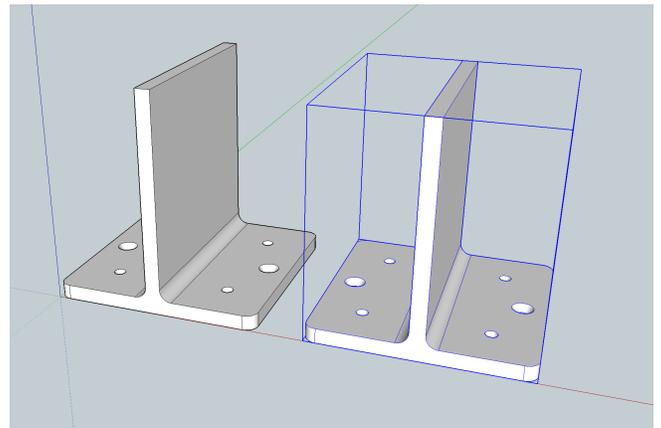
4. Since the object is symmetrical across two planes, we can either use the rotate tool to rotate one part upside down, or use the scale tool to mirror one part.

5. To rotate the part, use the rotate tool. Make sure the rotate tool is on the correct plane. In this case, it should be on the green plane.

6. To mirror the part, use the scale tool and drag the center handle on face into the negative region. Make sure it snaps to the ratio of -1.00.

Uniform Scale about Opposite Point
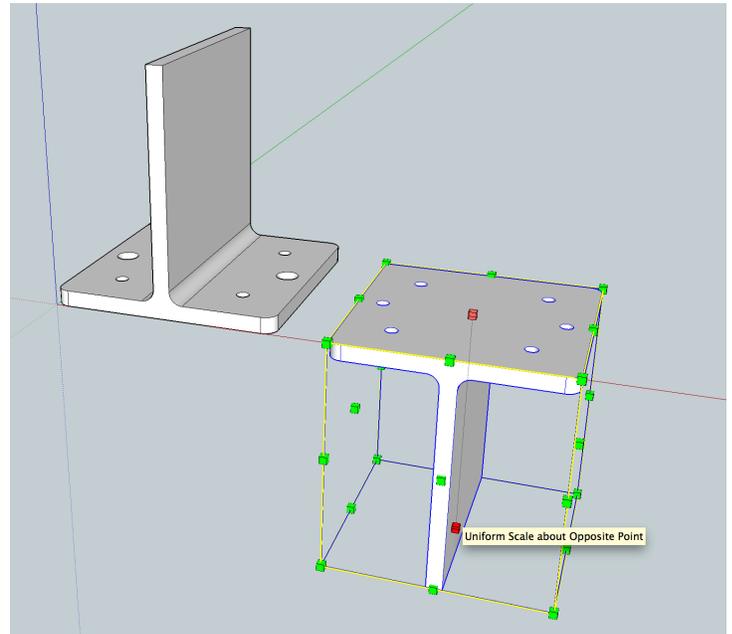
7. Now we can use the move tool to move the reflected part onto the original part. Using the move tool, click a corner on the surface of the "T" to be butt against the corresponding location on the original part.
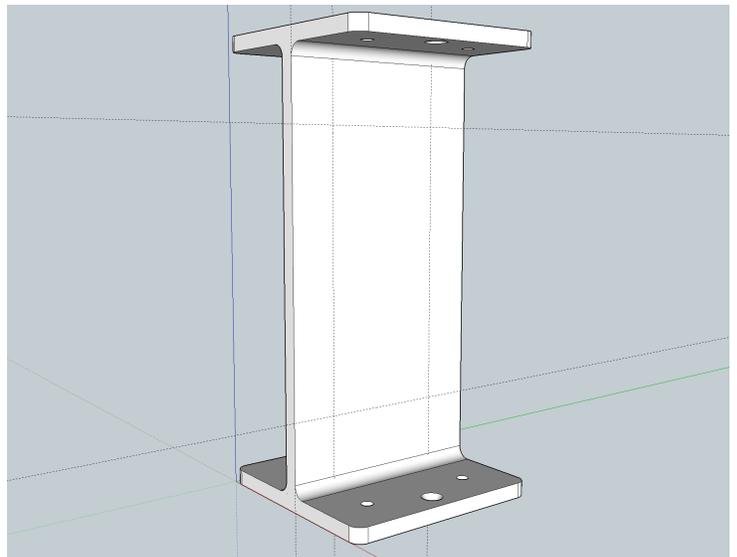

Endpoint in Group
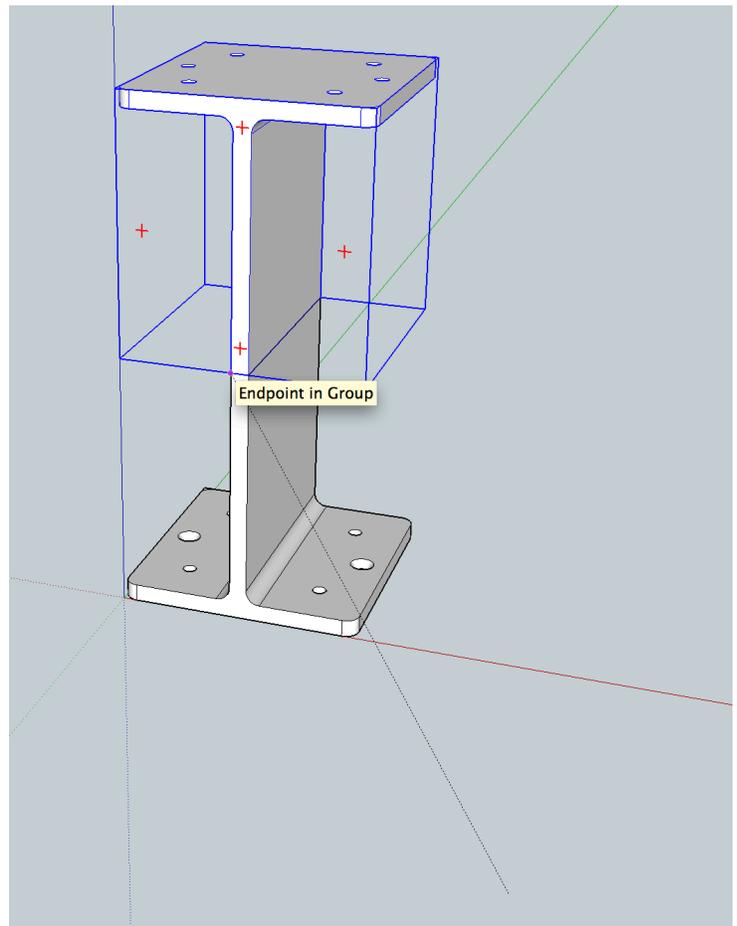
8. Once this is done, explode both groups and erase the dividing lines. This effectively joins the two parts into one.
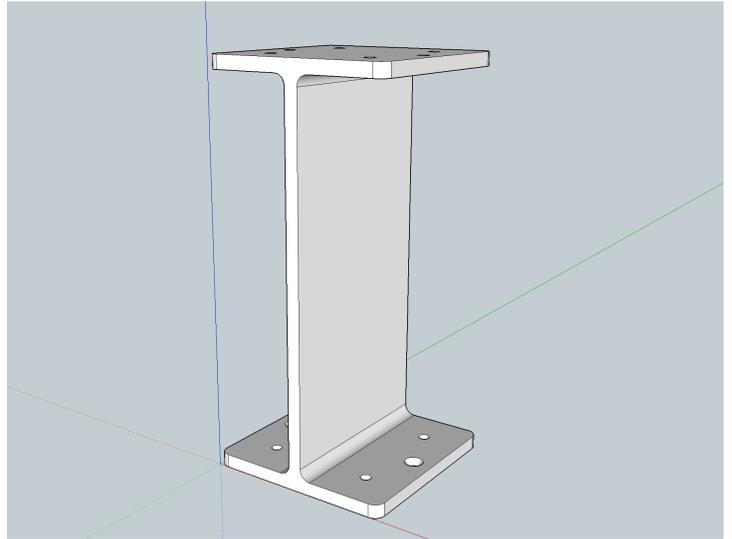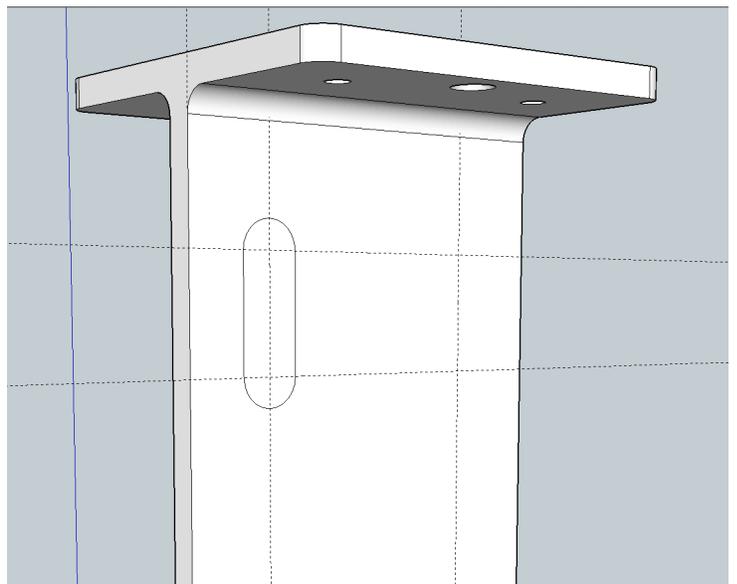

Creating an Array of Holes

1. On one face of the midsection of the I-Beam, create construction lines 15mm away from the sides and 20mm away from the edges of the fillet.


2. Create the shape seen on the right. The radius of curvature is 5mm, and the centers are 20mm apart

3. To create an array, we will use the move tool, coupled with the "control" ("option" on Mac) key and a simple keystroke.

4. Highlight the boundary of the shape you just created and create a duplicate to be moved to the intersection point to its right.
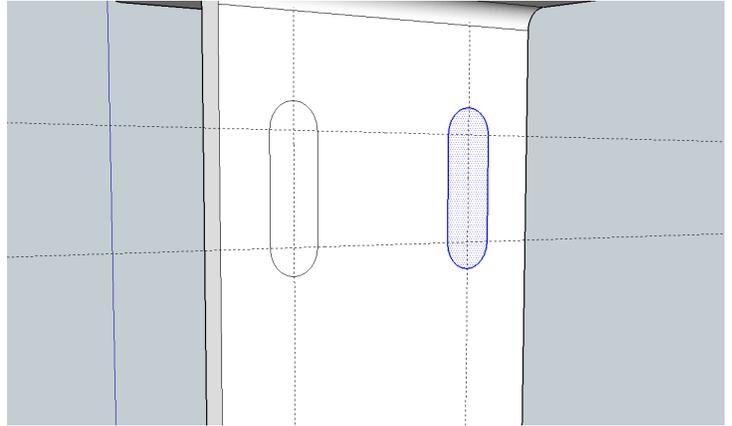
5. Then right after clicking the mouse to release the object, type in "/2." What this does is it creates two copies of the object you just duplicated and it positions them evenly spaced across the distance the object was moved.
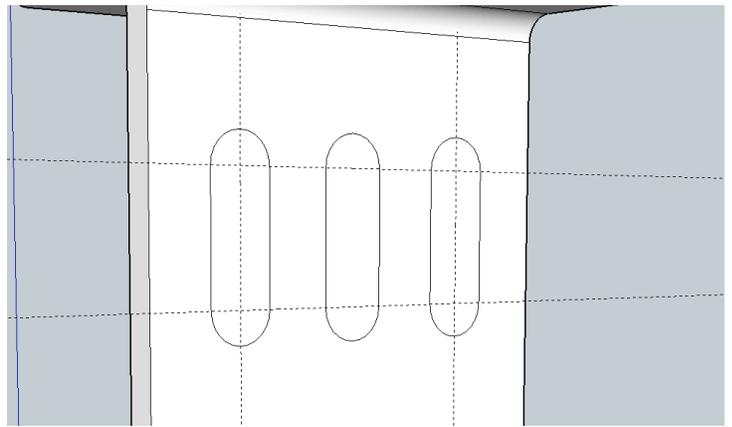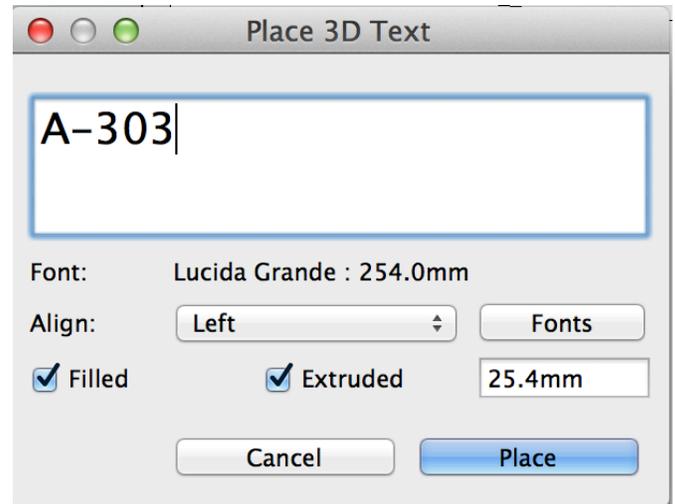
6. To make a 2D array, we will highlight all three shapes that were just created and move and duplicate them vertically in a similar fashion to create an array of 9 shapes.

   Finally, extrude all shapes to create holes and erase all necessary lines.
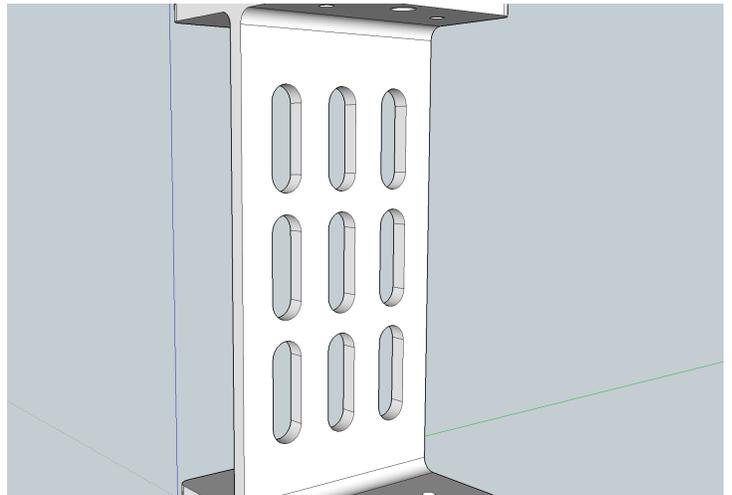
Creating 3D Text

1. Find the 3D Text tool under Tools > 3D Text. A dialogue window will pop up. Type in what you want, and specify the font.

Place 3D Text

A-303

Font: Lucida Grande : 254.0mm
Align: Left | Fonts
☑ Filled ☑ Extruded 25.4mm

Cancel | Place

2. Once you press "place," an object appears that is the word that you just typed in 3D. Move and scale the label and place it on your model. Use guidelines if you would like an exact placement.

3. Double click into the group to edit the 3D shapes. You can adjust the height of the extrusion here.
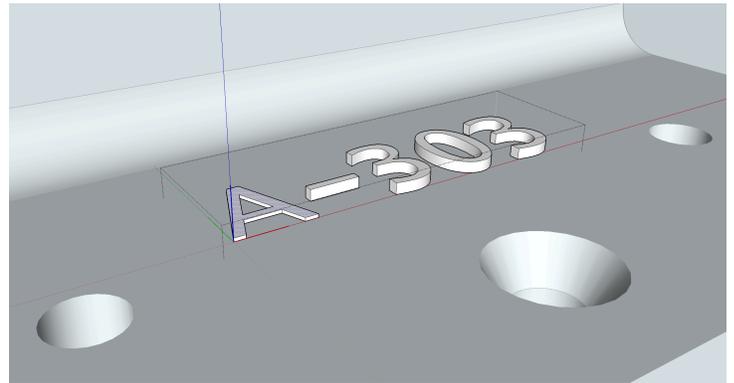
Finishing Up

1. Since the 3D text is in itself a solid part that is grouped, it is important to combine both parts to create one single solid part that is 3D printable.

2. First, highlight everything except the 3D text, and group them together. To check that it is indeed a solid part without any errors, right click it and choose "entity info." This should bring up a window.

   Notice the Volume. If your "entity info" window does not have a Volume, then your part is not a complete solid. You will need to find any extraneous lines or any incomplete surfaces to fix this.

Entity Info

Solid Group (1 in model)

Layer: Layer0
Name:
Volume: 85511.4 Millimete
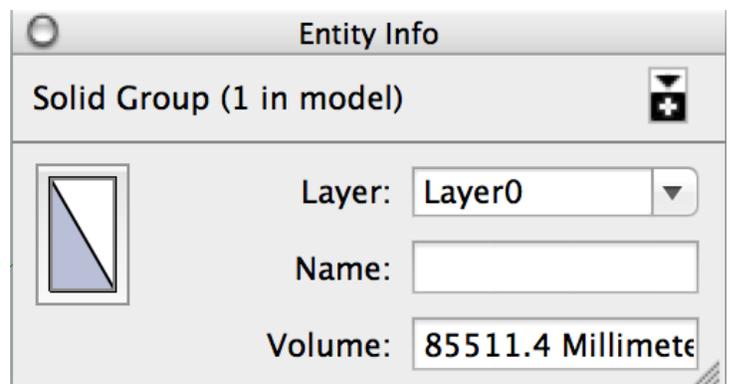
3. Once you are sure your part is in fact a solid, go to View > Tool Palettes > Solid Tools. This should bring up a tool box.

4. Click the top left option (the union tool) and highlight both the I-Beam and the 3D text. This will unify both groups into one solid. To check that it is a solid, check the entity info and check for the volume.

Voila! You have your part complete!

Tips on making 3D Printable/Feasible files:

- Keep in mind that if you have overhangs in your parts, the quality may not be as good.
- Most 3D printers have a maximum overhang angle of 45º that is printable without the need for support structures.
- Unless you have a dual extruder head that can print support structures in a different material, the final finish may not be as good.
- Think about infill for parts that need to bear load
- You can use acetone to smooth out parts printed with ABS plastic, and THF to smooth out parts printed with PLA.

To export a SketchUp file into a 3D printable .stl file, download and install the Ruby extension from the SketchUp Extension Warehouse: http://extensions.sketchup.com/en/content/sketchup-stl