

Class Handouts

Handouts for Protohaven classes

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2D (Inkscape) Design: Basic Wrench

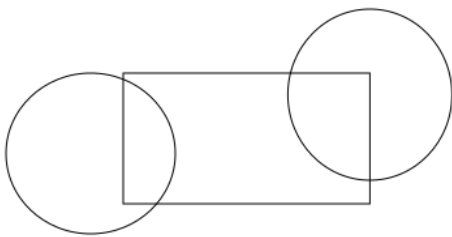
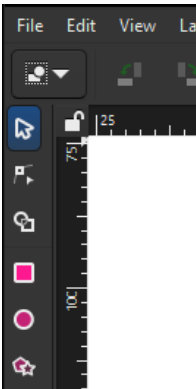
This simple tutorial walks you through designing a basic 10mm "box" wrench with accurate dimensions. You can modify the shapes in this tutorial to quickly create other designs which can then be created using Protohaven's vinyl cutter, CNC plasma cutter, CNC router, CNC embroidery machine, and other tools designed to cut 2D materials.

This tutorial is part of the [2D & 3D Design Bootcamp](#).

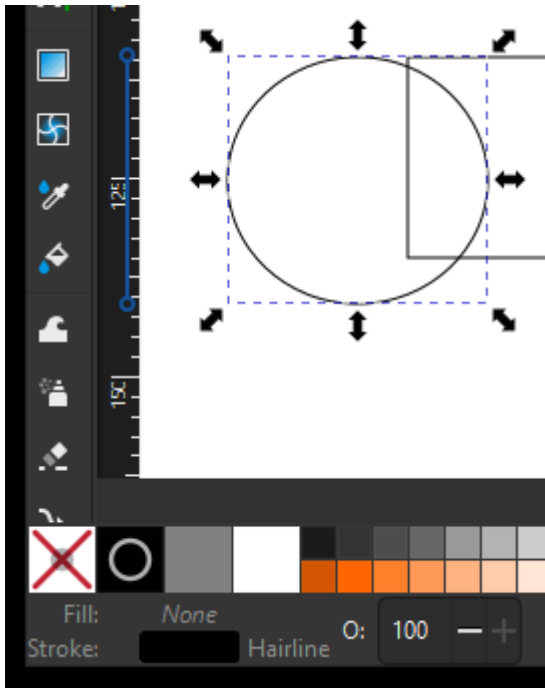
Setting up the primitive shapes

It's a good idea to save (Ctrl+S) periodically while designing in Inkscape - the program does sometimes crash and any unsaved data can be lost.

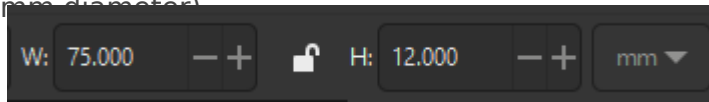
1. Using the circle and rectangle toolbar icons on the left side of the window, draw two circles and a rectangle of any size.



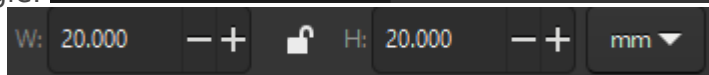
2. Select all objects and set the stroke to black, and the fill to transparent (the "X" color on the bottom left corner of the page)



- Set the dimensions of the rectangle to 75x15mm and the width/height of each circle to 20mm (i.e. 20mm diameter).

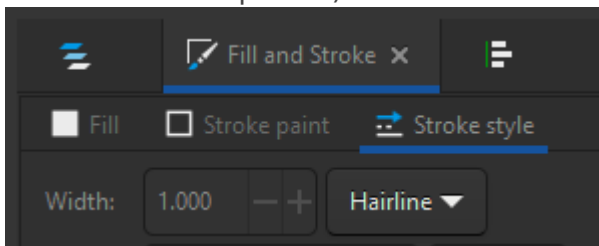


For rectangle:

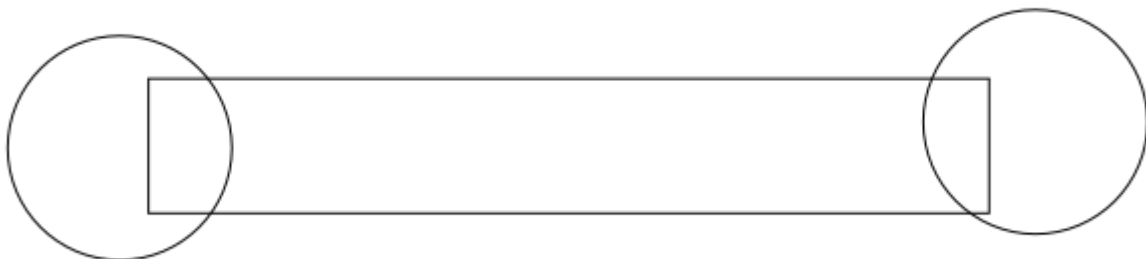


For circles:

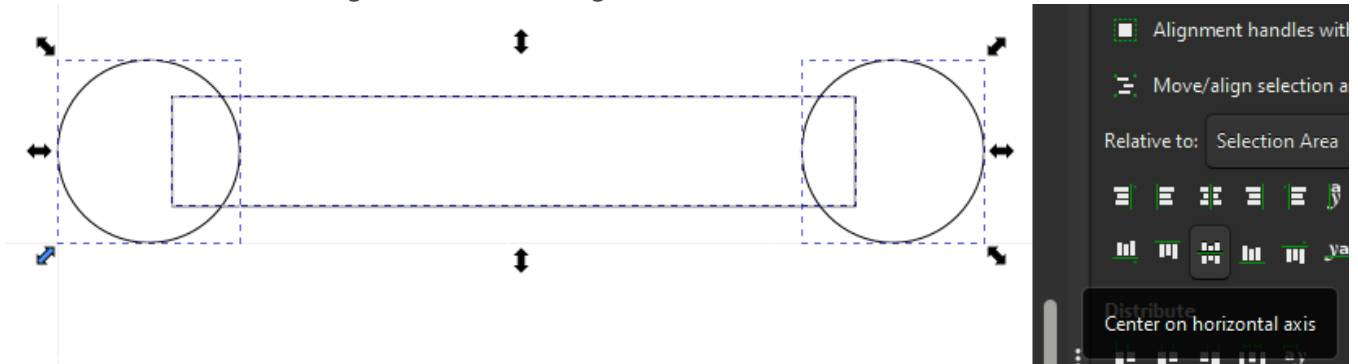
- Ensure the rectangle and circle stroke style are all set to "hairline" (Object > Fill & Stroke to open the right panel, then select the Stroke Style tab and select "hairline" from the width's units dropdown).



- Drag the circles so they hang over either side off the rectangle.

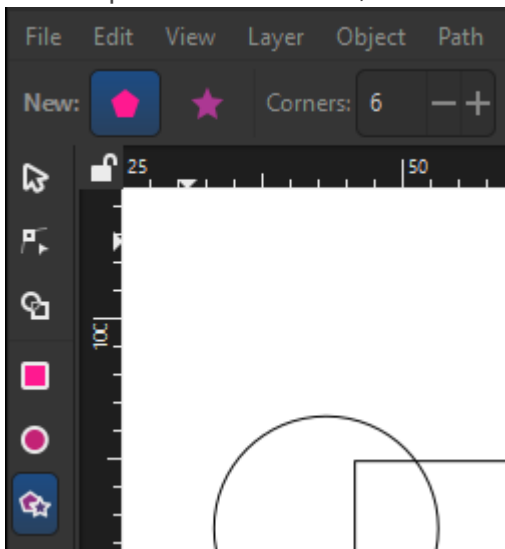


- Using the Align and Distribute panel (Accessible from the top menu - Object > Align & Distribute), select all three objects and click "Center on horizontal axis" (the icon in the bottom middle of the Align icon set) to align their vertical centers.

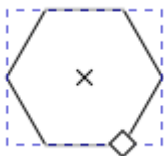


Creating the interior/flats

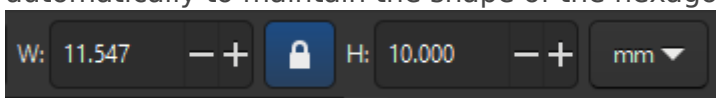
- Select the star/polygon tool from the left menu bar. The top toolbar will change to offer some options for the tool; ensure it is set to create a regular polygon with 6 corners.



- While holding **Ctrl** on your keyboard to snap rotation, draw a hexagon with horizontal top and bottom sides. Press escape a couple of times to get back to the selector tool.

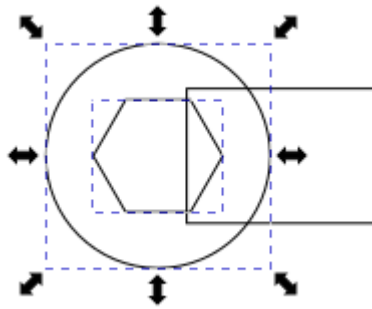


- Select the created hexagon, lock the aspect ratio using the padlock icon near the width/height text boxes, and set the height to 10mm. The width should adjust automatically to maintain the shape of the hexagon.

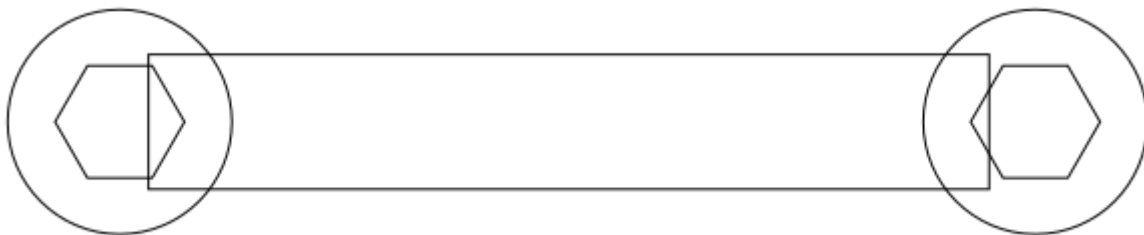


- With the hexagon selected, shift-select the left circle. Navigate back to the Align and Distribute panel, then set **Relative to** to **Last selected**. Center on horizontal axis and vertical axis using the two icons on the panel (you can hover over them to see the name of the

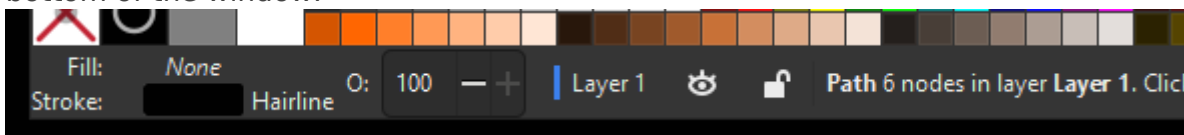
alignment operation).



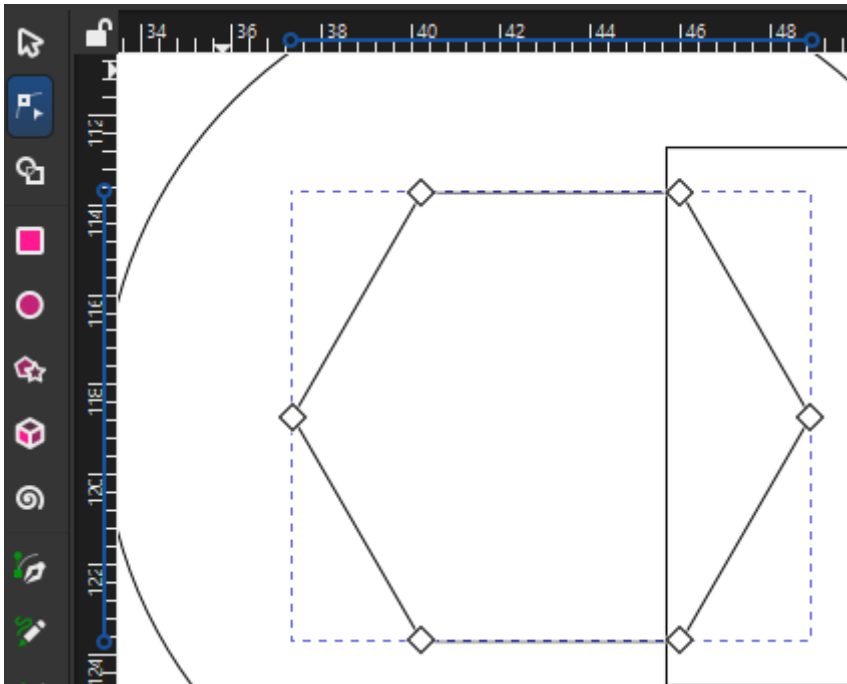
5. Deselect all objects by clicking elsewhere on the page. Select, copy (Ctrl+C) and paste (Ctrl+V) the hexagon near the circle on the right side, then perform the same alignment as in the previous step to align it to the right side circle. You should have something that looks like this:



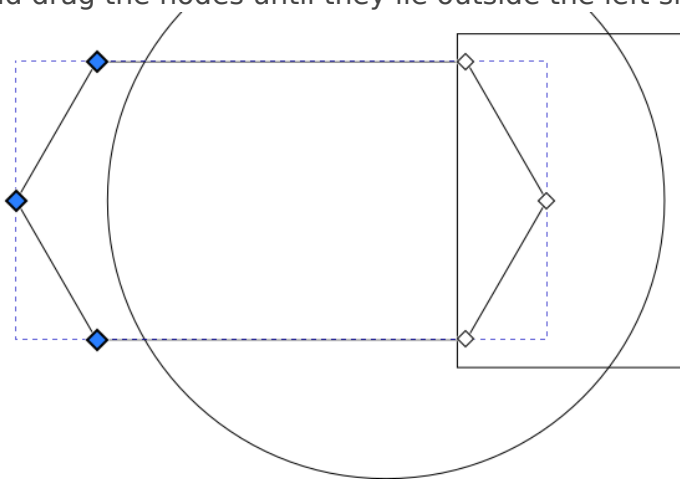
6. Select the left hexagon and convert it from a hexagon to a path via **Path** > **Object to Path**. Confirm you now have a path selected by looking for "Path 6 nodes in layer..." at the bottom of the window.



7. Double click the already-highlighted hexagon (or click the Node Tool in the left toolbar). The hexagon will now show drag handles on each of its corners.



8. Drag- or shift-select the left three nodes. While holding Ctrl to constrain movement, Click and drag the nodes until they lie outside the left side of the circle.



9. Press the Escape key a few times until the UI stops changing - nothing should be selected, and you should have the Selector Tool highlighted in the left toolbar.

Using Shape Builder to create the final design

1. Press Ctrl+A to select all objects, then click the Shape Builder tool on the left toolbar. Your design should now be filled in gray and you should see "Add", "Delete", and "Finish" icons in the top toolbar.



2. With the "Delete" icon selected, click on all of the parts of the wrench we do not want in the finished product. These will disappear from the page:



3. Now click on the Add icon in the top menu bar, then **click and drag** on all parts of the wrench to preserve. These will highlight in blue, and any edge you drag over will be eliminated:



4. Press the checkmark "Accept" icon next to the Finish label on the top menu bar to lock in your changes. Congratulations, you've made a wrench!

You can compare your work against this reference file: [wrench.svg](#)

3D (Fusion 360) Design: Wrench Holder

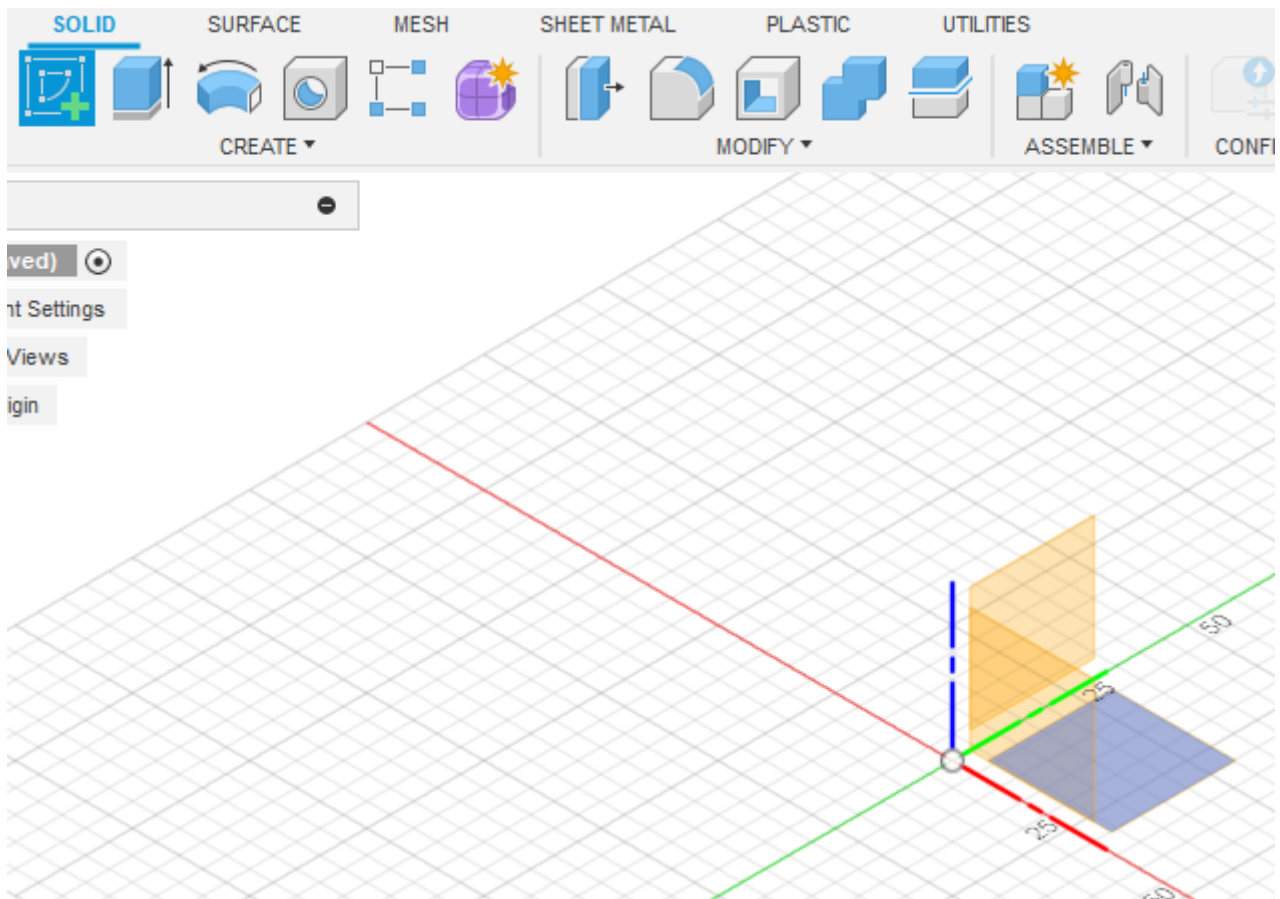
This simple tutorial walks you through designing a 10mm "box" wrench holder with accurate dimensions. You can modify the shapes in this tutorial to quickly create other designs which can then be created using Protohaven's 3D printers, Tormach CNC mill, and other tools that operate with STL or 3D design files.

This tutorial is part of the [2D & 3D Design Bootcamp](#), and relies on the file from [2D \(Inkscape\) Design: Basic Wrench](#). If you'd rather start with this tutorial, be sure to download [wrench.svg](#) so you can use it in the steps below.

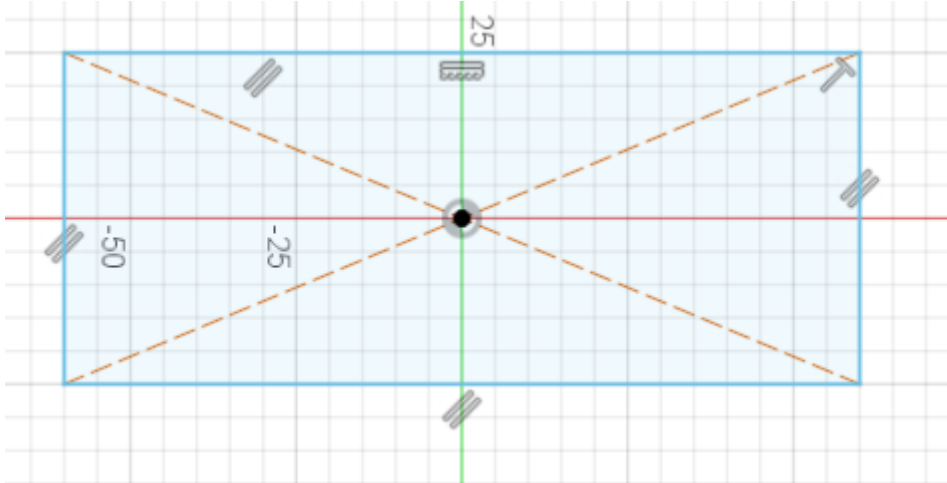
Fusion 360 requires a (free) license to operate - if you don't have a license already, go [here](#) to obtain one.

Setting up the main body

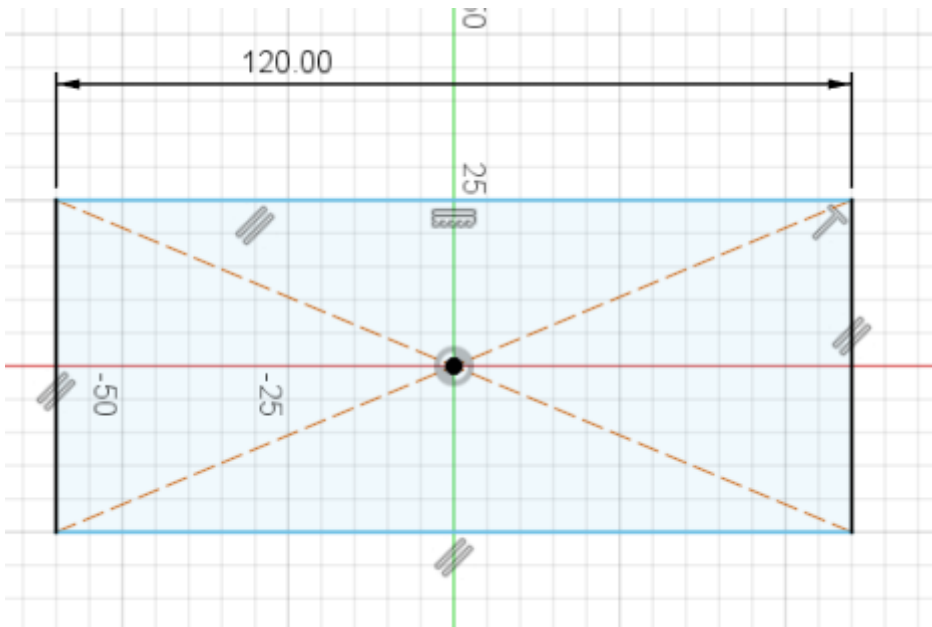
1. With Fusion 360 open to a new project, click on the "Create Sketch" tool in the top menu (under SOLID), then click on the bottom (XY) plane.



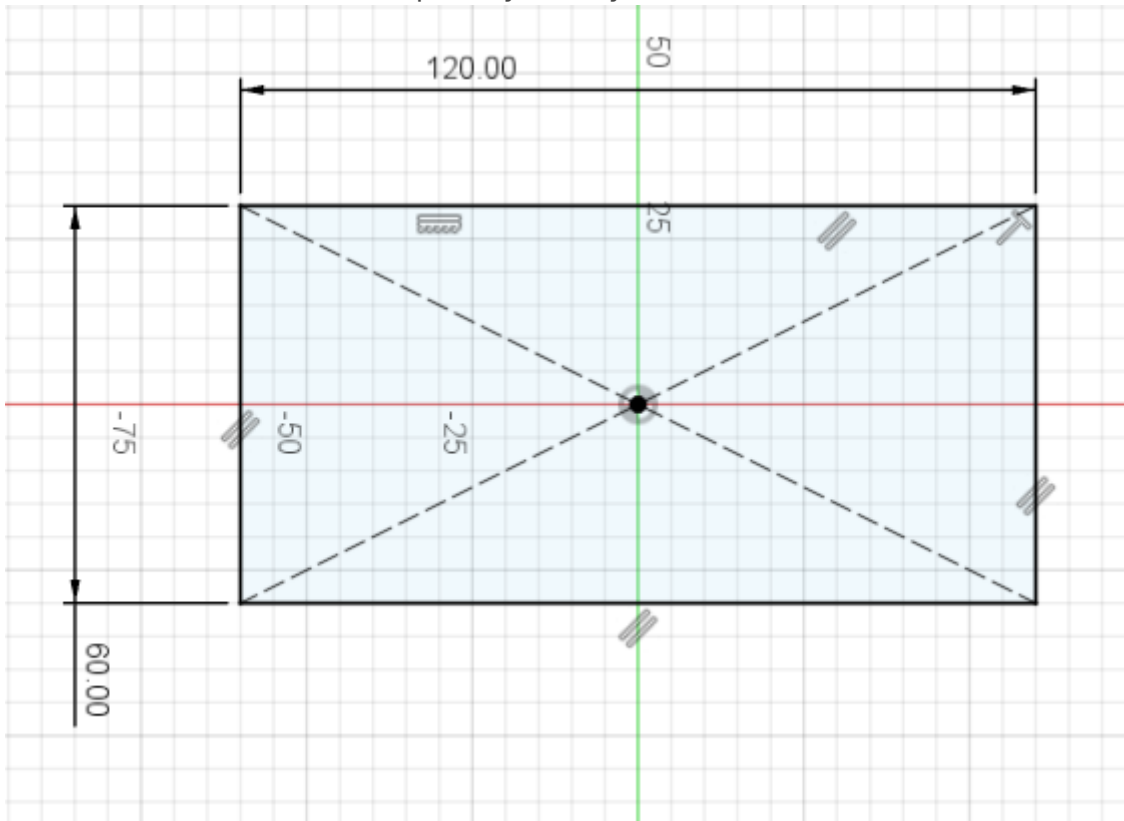
2. The view will change to a top-down view. click to expand the CREATE top toolbar menu, then navigate to Rectangle > Center Rectangle. Click the origin on the 3D view, then move your mouse and click again to create a rectangle centered on the origin.



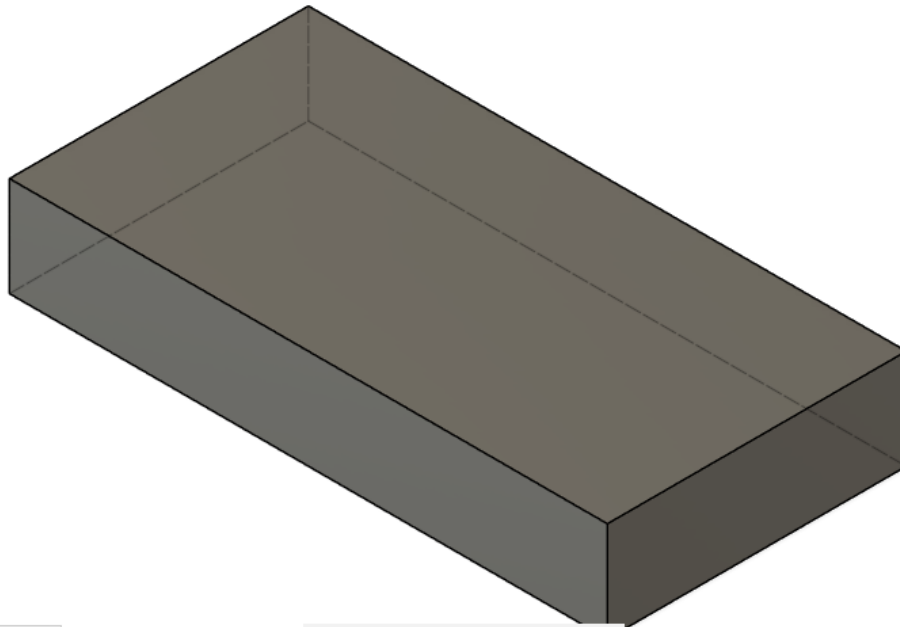
3. Expand CREATE again to select Sketch Dimension (or press the D key as a shortcut). Click the top edge of the rectangle, then move your mouse upwards a bit and click again to place the dimension. Type "120 mm" into the value box and hit Enter to set the dimension.



4. Use Sketch Dimension again, this time on the left edge of the rectangle. Make this dimension 60mm. Press escape on your keyboard to exit the Sketch Dimension tool.



5. At this point, the whole rectangle should be black. Clicking and dragging any part of the rectangle (other than the dimensions) should do nothing. This is called being **fully constrained**. Designing parts that are fully constrained is recommended when doing any kind of complex design work, as it saves time when adjusting things later.
6. Click the green "FINISH SKETCH" checkmark on the top right of the screen to finish the sketch. The view should change to an isomorphic (corner) view of your rectangle.
7. Click the "Extrude" tool in the top toolbar, then click on the interior of the rectangle. Enter -20mm, then press Enter to extrude a solid body downwards from the rectangle.



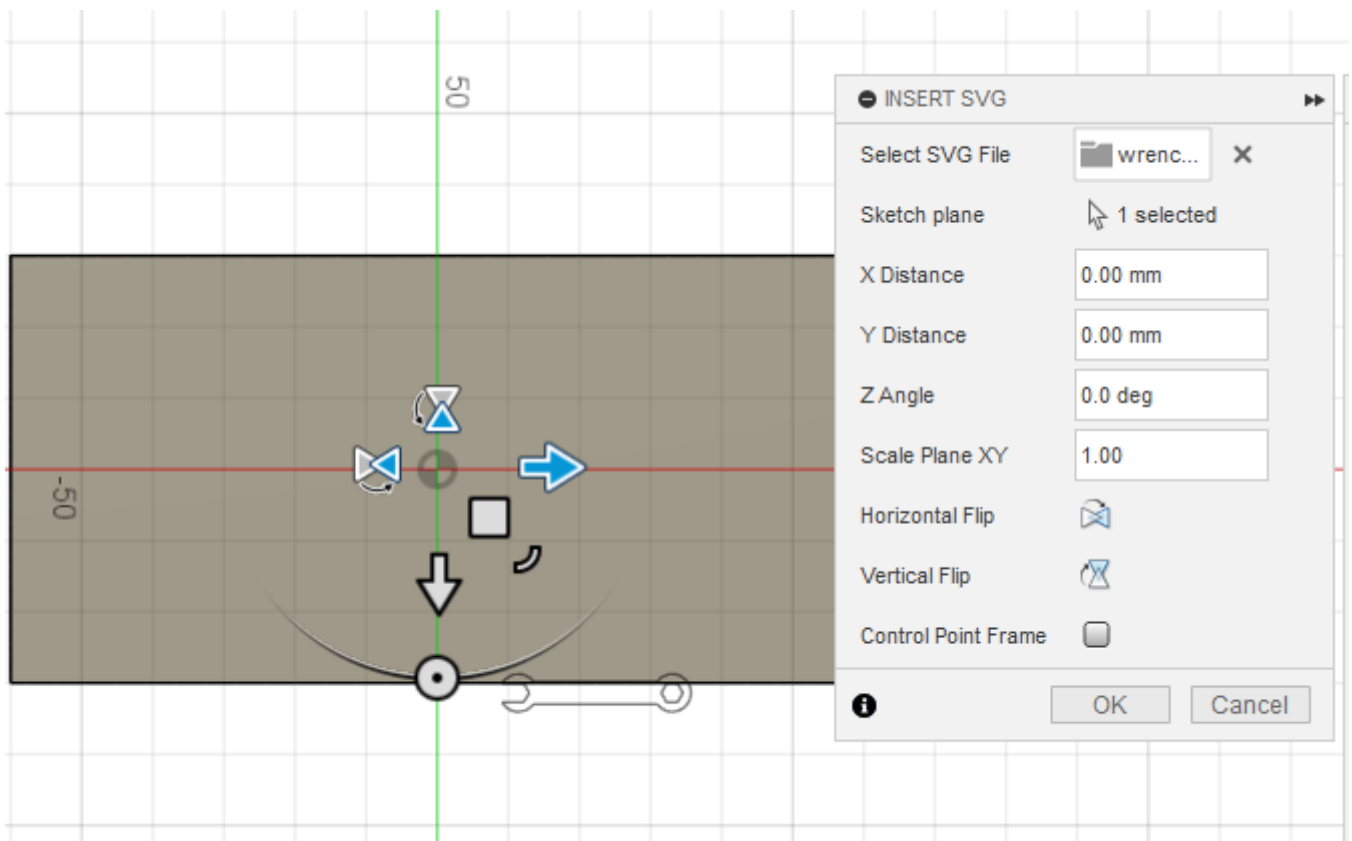
Now that we have a few operations, take a look at the timeline on the bottom of the application window. You can go back and edit any sketch or feature by double-clicking on one of the icons in the timeline.



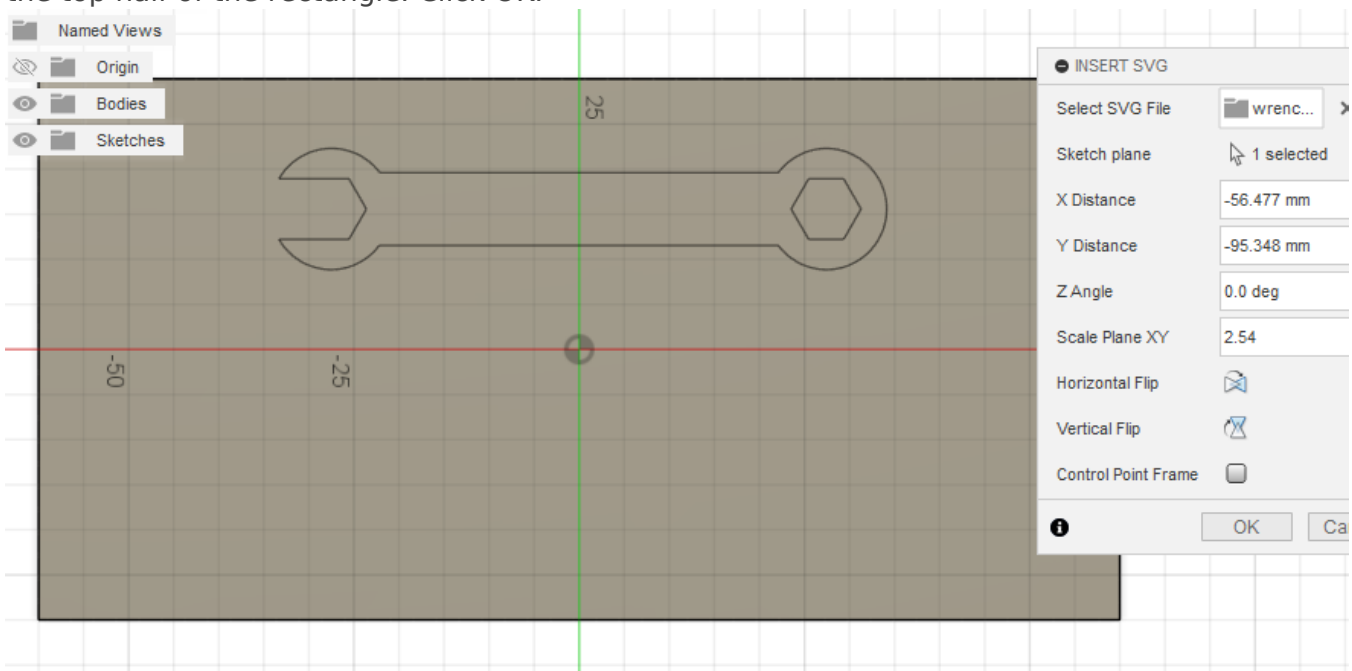
Creating the wrench caddy

1. Expand the "INSERT" menu on the top toolbar and select **Insert SVG**, then **Insert from my computer**. Navigate to the wrench.svg file and open it, then click on the XY plane (as with the first step when creating the rectangle sketch).

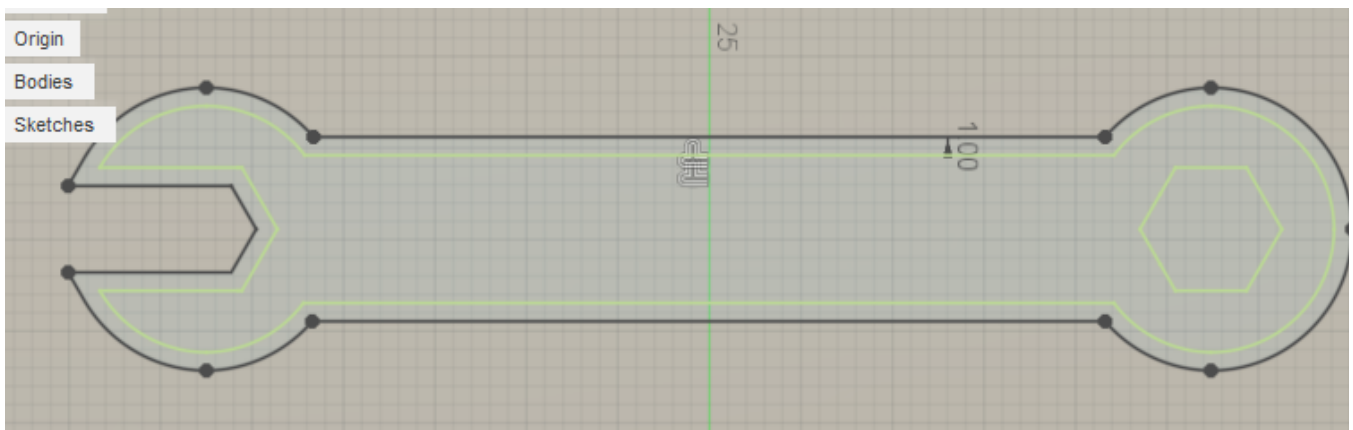
The wrench will appear, but it won't be the right size or in the right place. This is because SVG describes a **Scalable** Vector Graphic which does not usually include units (e.g. mm or inch) in the file data.



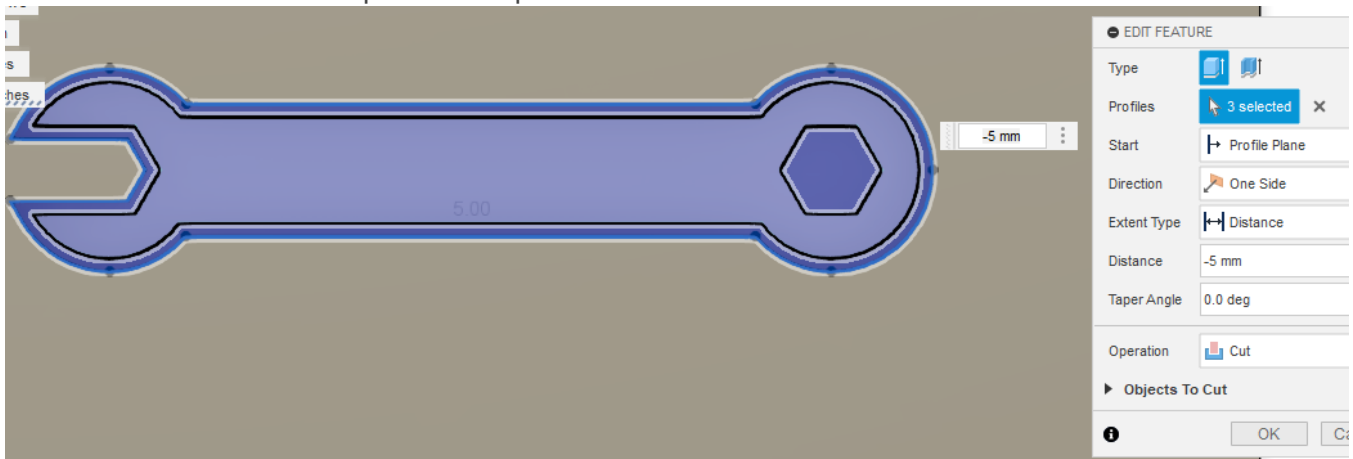
- Set Scale Plane XY to 2.54 to convert from cm to inches, then drag the wrench so it's on the top half of the rectangle. Click OK.



- Under the MODIFY top toolbar menu, click on Offset, then click the outer perimeter of your wrench. set distance to 1mm, then click OK to add a 1mm offset around the existing design. This prevents our wrench from being exactly the same size as our wrench slot, which would produce a "press fit" that makes our wrench difficult to remove and use.



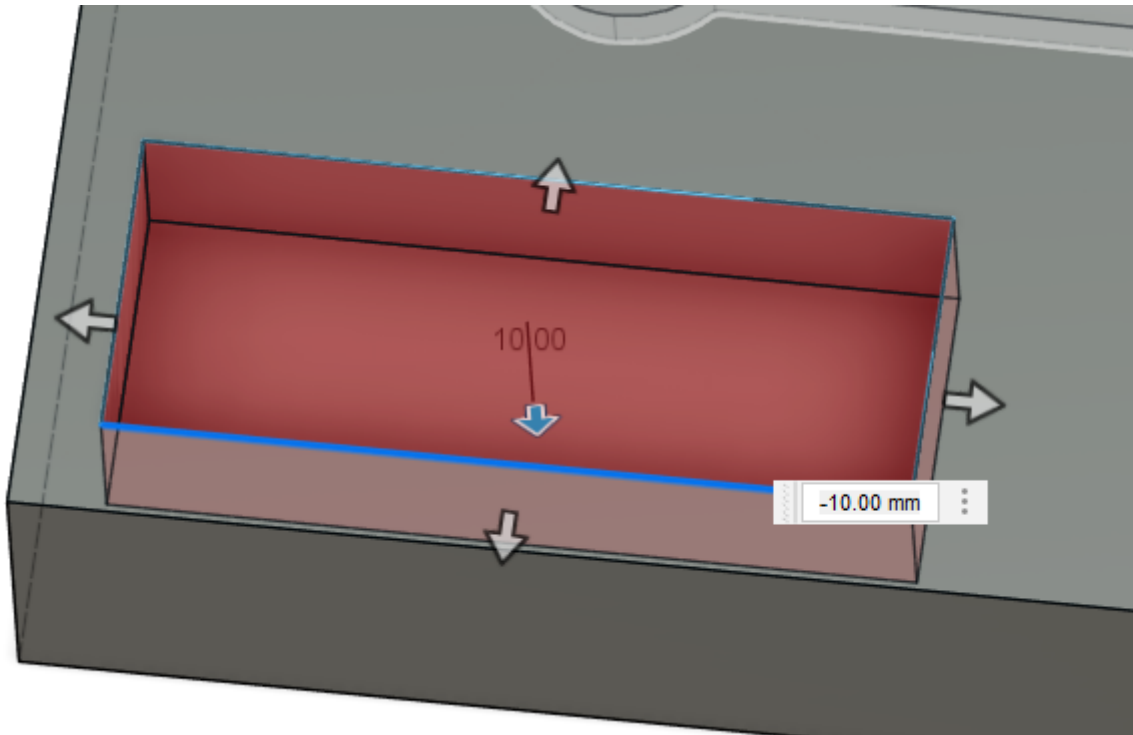
4. Click FINISH SKETCH to exit the sketch, then enter the Extrude feature tool just like we did before.
5. Left click within the area of the wrench sketch, the as well as the main body of the wrench and the closed hexagon side. Type **-3mm** to extrude it downwards into the rectangle body. Fusion 360 will automatically perform a Cut operation to remove this material. Click OK to complete the operation.



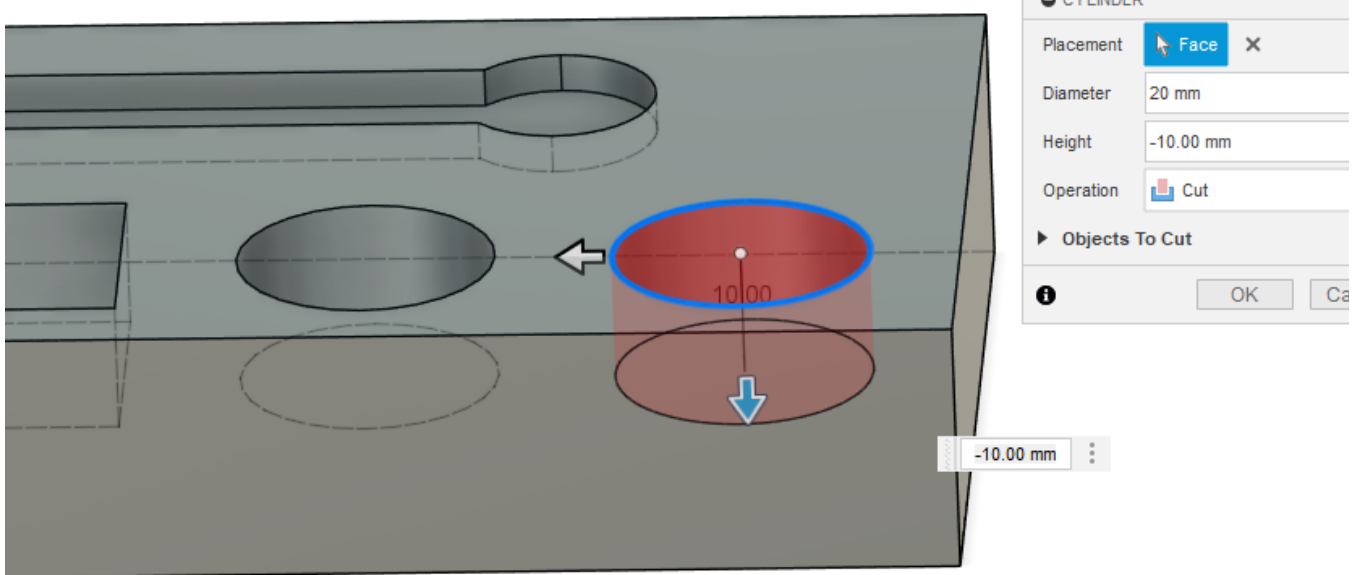
Adding machine screw and nut trays

Let's use some solid bodies to make trays for some nuts and bolts we can use with the wrench.

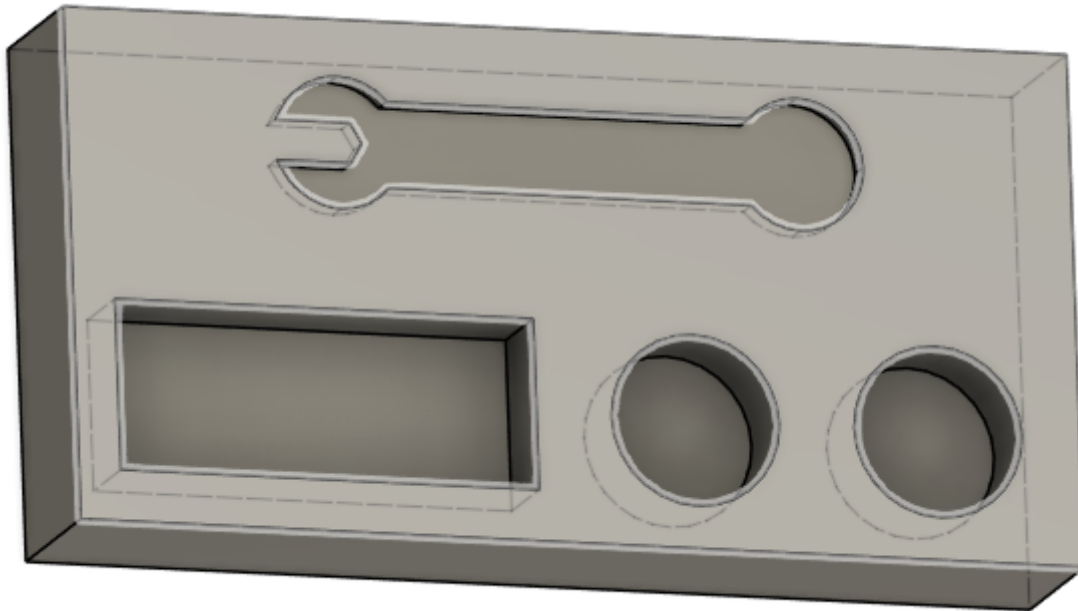
1. Under the CREATE menu of the top bar, select "Box", then click on the top face of the rectangular solid we've been working on.
2. Click nearby the bottom left corner, then click close to the origin to set the second point of the box. Click and drag the middle arrow that appears, until it's pointing 10mm down into the solid body. Click "OK" on the right-side popup menu to complete the operation.



- Repeat these steps two more times, but now using the Cylinder tool under the CREATE menu to create 20mm diameter cylinders that cut 10mm into the part. If you need to get a better look at what you're doing, you can orbit around your part by left clicking and dragging the axis cube with X/Y/Z/FRONT/TOP etc. markings on the top right of the screen.



- You should now have a wrench tray and a couple bins for nuts and bolts!

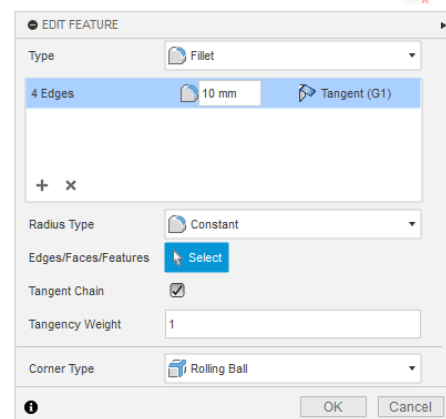
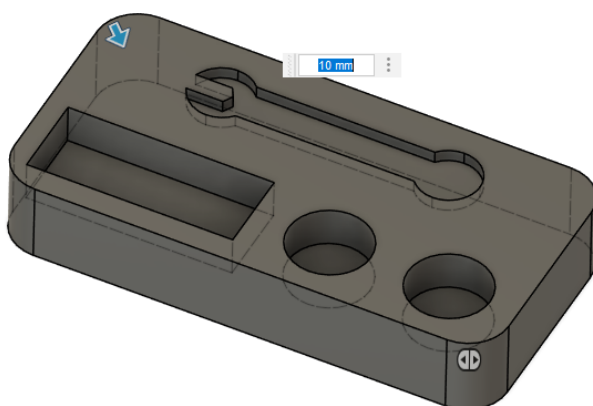


Using primitives instead of sketch/extrude is fast, but you may find it hard to precisely align the features. If you need a primitive exactly positioned somewhere, one way is to create a sketch *before* creating the primitive, and clicking during creation to reference it to a fully constrained point on the sketch. However, this does somewhat eliminate the benefits of using primitive solids vs just creating a fully constrained sketch and extruding it.

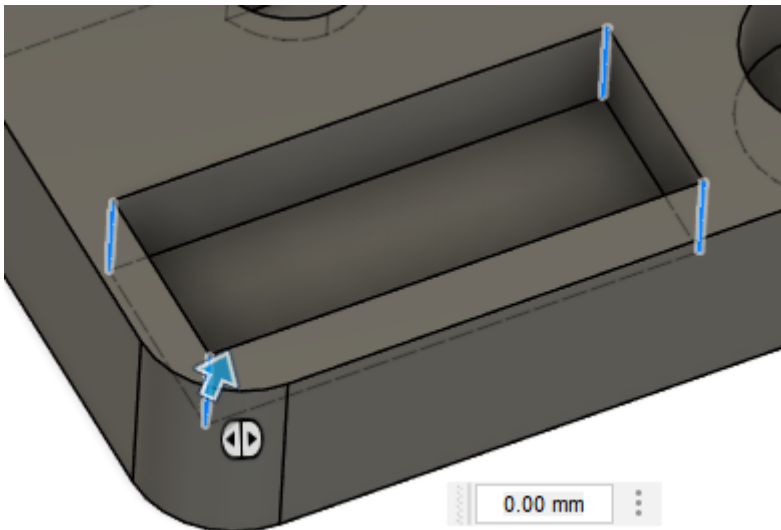
Rounding corners & finishing details

Let's knock off some of the sharp edges on our part to make it nice to handle.

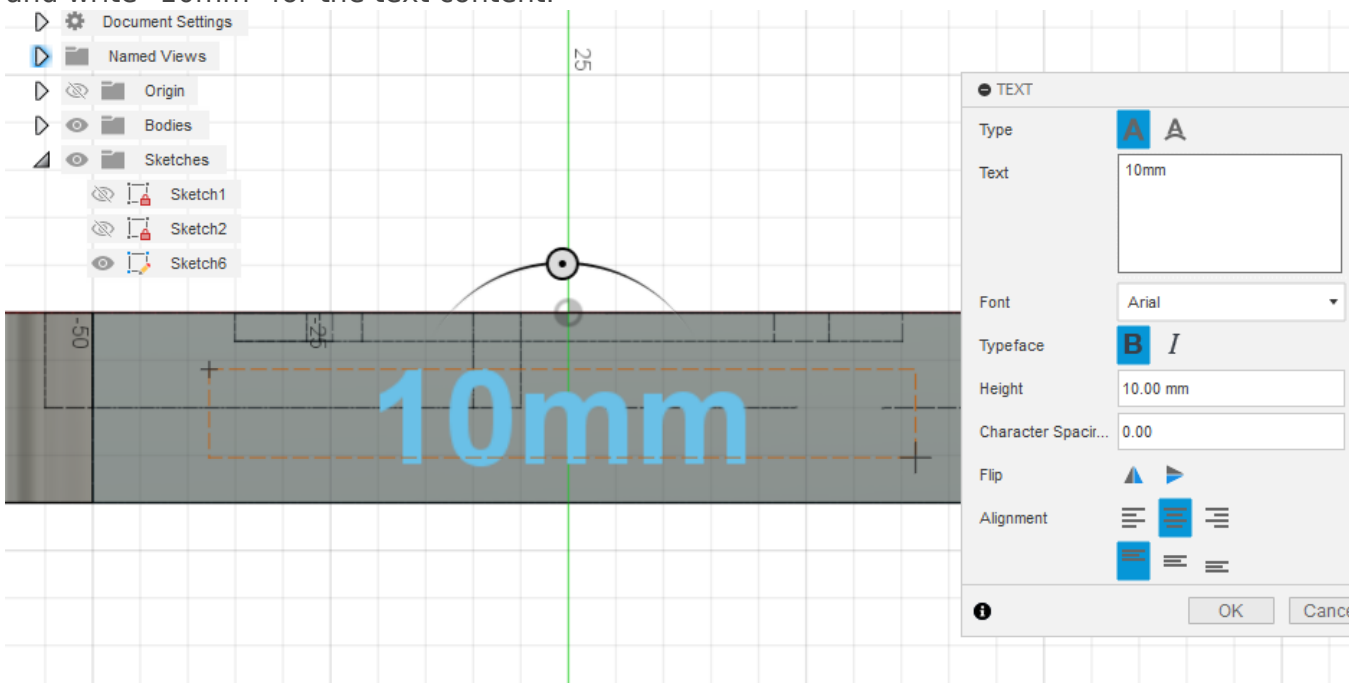
1. Under MODIFY, click on the Fillet tool. Orbit the view around your part and click to select each of the four vertical corners of the rectangle. Set the fillet dimension to 10mm, then click OK.



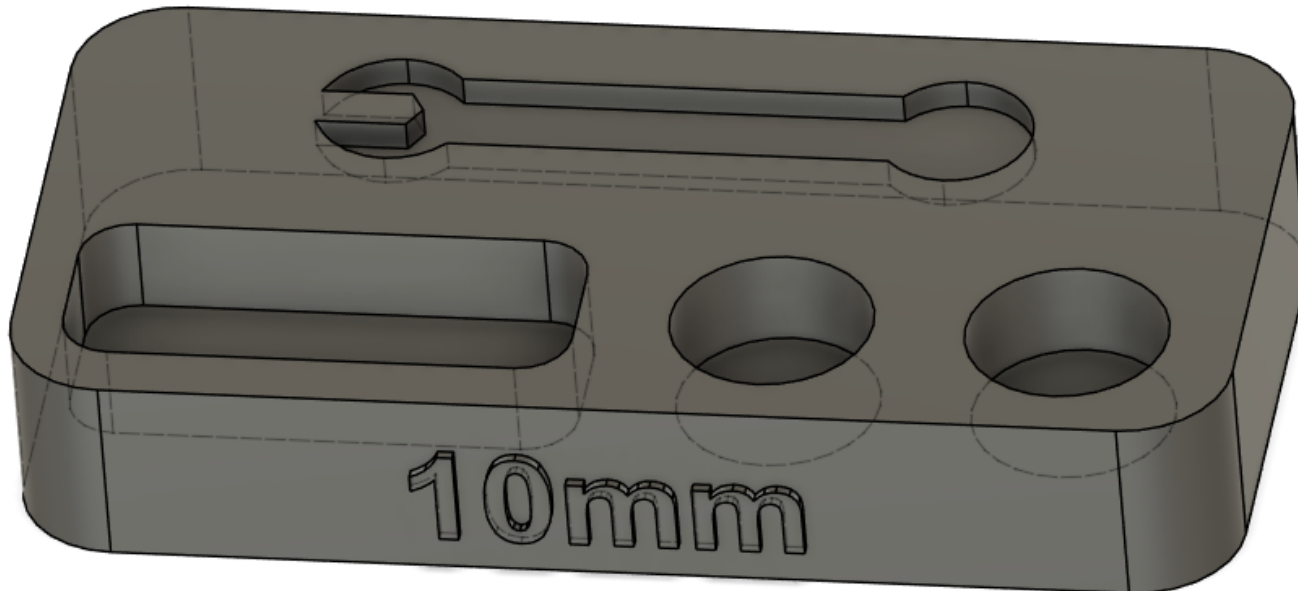
2. Do this again to create a 5mm fillet on the interior edges of the rectangular tray on the front left side of the part. If you have trouble selecting individual edges, click and hold over the edge to open a menu that allows you to "select through" the body of the part.



3. Create a new sketch on the front face of the part, using the Text tool (under the CREATE menu) to place some custom text. Select center alignment, 10mm Arial Bold for the font, and write "10mm" for the text content.

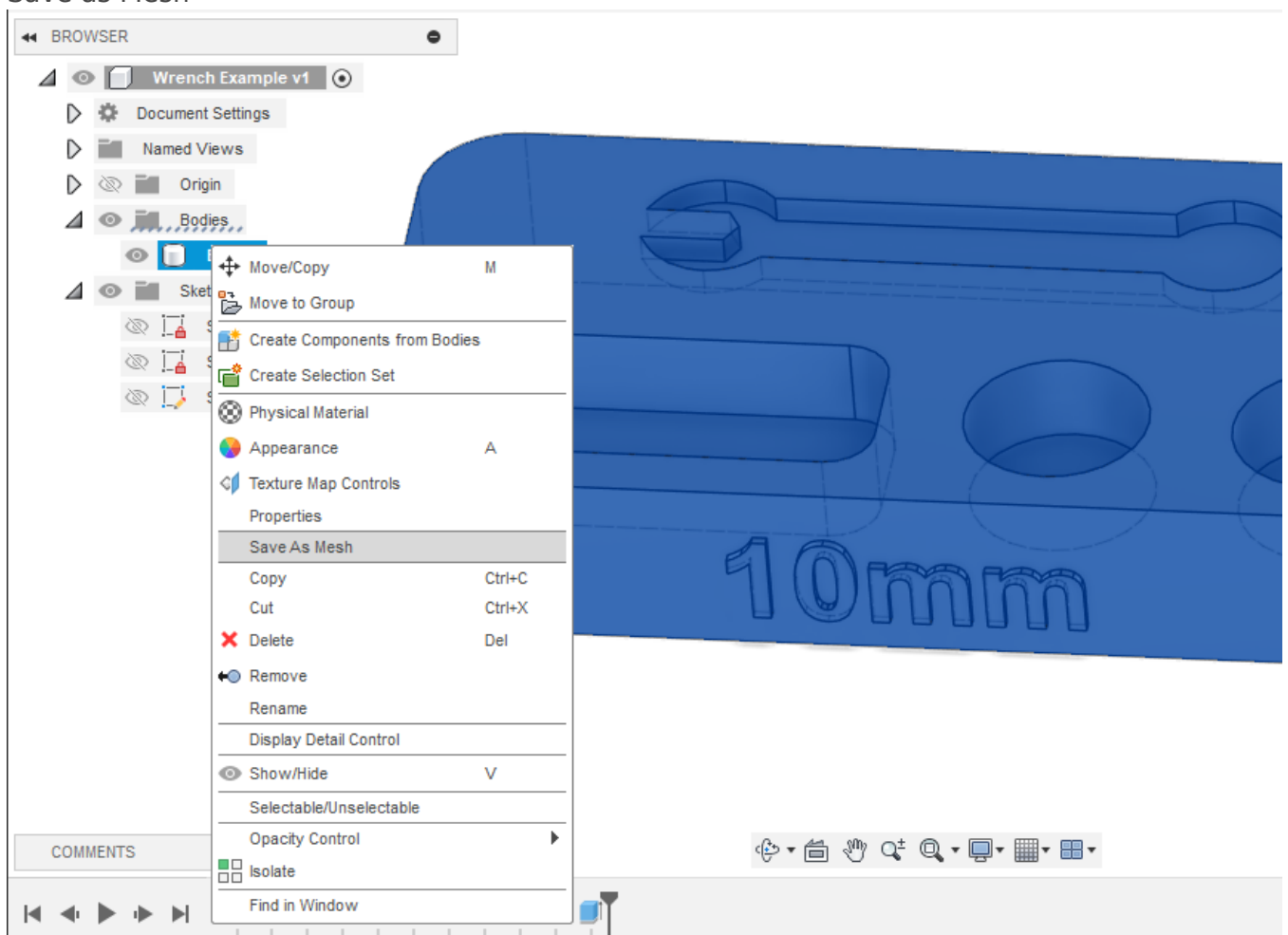


4. Press OK, then FINISH SKETCH, then select the text and Extrude it 1mm away from the front surface of the part. You should now have a finished wrench and nut/bolt holder!



Exporting to SVG

1. Expand the Bodies folder in the browser overlay, then right click on Body 1 and select Save as Mesh



2. Under Format, select STL (Binary), then click OK and choose where to save your file. You should end up with something like Body1.stl saved onto your computer.