

Introduction to Fusion 360

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Setting up Fusion 360

- Click on your initials in the upper right hand corner of Fusion 360, then select preferences.
- In the Default Units > Design panel, set the default units to inches.
- (Trackpad Macs) In the General panel, select Use Gesture-Based View Navigation. In the Preview Features panel, select “Native Trackpad” under General
- (optional) In the General panel, you can select your favorite 3D interface gestures. I prefer to use “Z up” modeling orientation, “Fusion 360” Pan, Zoom, Orbit shortcuts, and “Free Orbit” as the Default Orbit type.

Parametrizing the Design

Our initial design will be based on measurements taken from my cell phone and charging cable. After the design is finished, you can change these parameters to match your device. These parameters are defined in the sketches below and listed in the table.

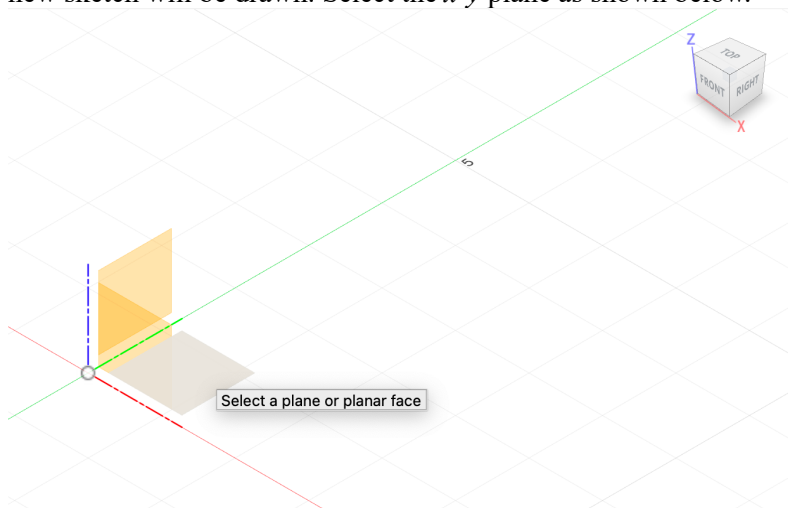


Parameter Name	Parameter	Measured	Extra Space
cableDiameter	0.12	0.10	0.02
cableGrommet	0.18	0.15	0.03
plugHeight	0.475	0.475	0
plugWidth	0.32	0.30	0.02
plugDepth	0.205	0.185	0.02
ledgeWidth	0.40	0.40	
phoneHeight	5.75	5.75	
standWidth	3.00		
standThickness	0.10		

- In the Modify menu of the toolbar, select “Change Parameters.” Click “+ User Parameter,” then define each of the 9 parameters in the table above. When you are finished, close the window using the “OK” button at the lower right.

Sketching the Stand Body

- In the Create menu of the toolbar, select “New Component.” Name your new component “standBody” and click OK.
- In the Create menu of the toolbar, select “New Sketch.” You need to select the plane in which the new sketch will be drawn. Select the x - y plane as shown below.

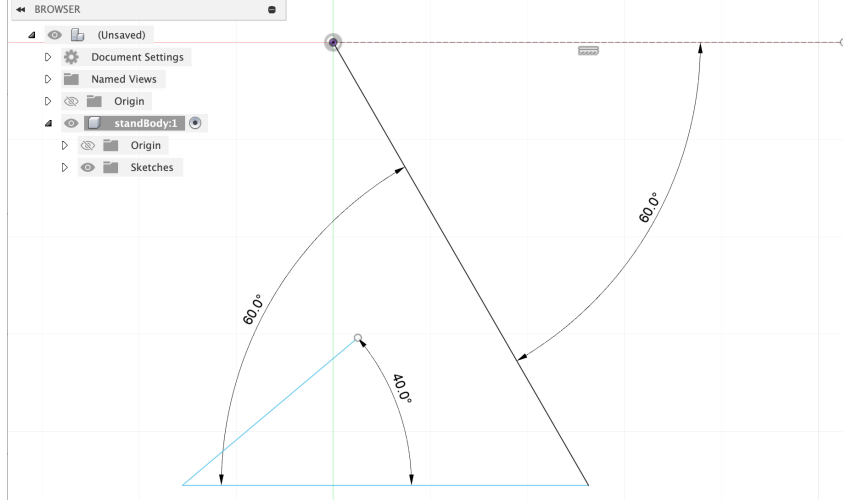


- Move the view so that the origin is in the upper left hand corner of your screen. You can either use gestures or the Pan and Zoom In / Zoom Out icons at the bottom of the screen.

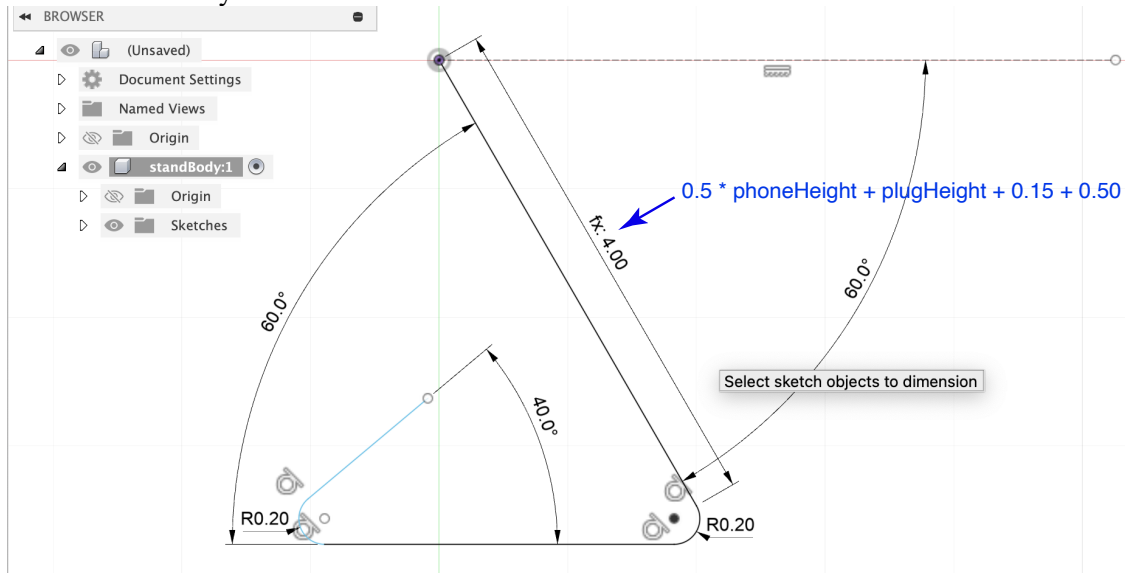


- Click the “Create Line” icon in the Create section of the toolbar.
- Mouse click on the origin then drag right and down. Hit the tab key to highlight the angle of the line. Change the angle to 60° then tab back to the line dimension. Continue dragging until the line is ~ 5 " long. Mouse click to put a break in the line. Drag to the left. Hit the tab key then change the angle of this line to 60° . Hit the tab key again to go back to the line dimension. Continue dragging until the line is well to the left of the origin. Mouse click to put another break in the line. Drag towards the upper right. Hit the tab key then change the angle of this line to 40° . Mouse click before you get to the original line. This will start a 4th line, which you do not want, so hit escape.

At this point, your drawing should resemble

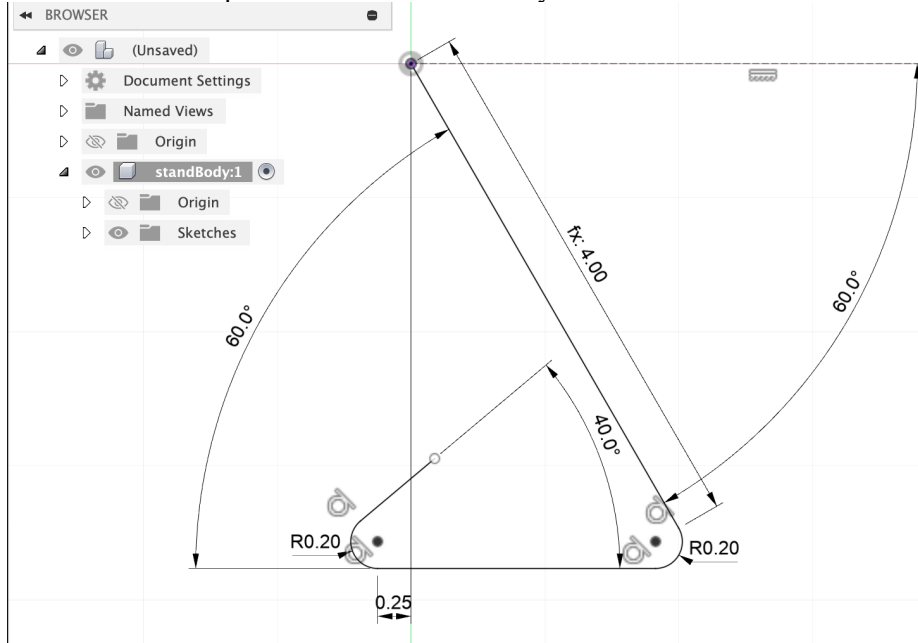


- This sketch would give your phone stand two sharp corners, which would be both ugly and prone to fracturing. To fix this, we need to put fillets (rounded corners) between the lines. Under the Modify menu in the toolbar, select “Fillet.” Drag fillets between each of the connected lines. The radius of the fillets are up to you. I used 0.20”.
- Now we need to start dimensioning our stand. Click the “Sketch Dimension” tool in the Create section of the toolbar, then click and drag a dimension for the phone rest. This distance will need to support some fraction of the phone back, the plugHeight, a little bit of plastic to support the plug, and a bit of space for the cable to feed back. To account for these dimensions, I clicked in the dimension entry box and entered “0.5 * phoneHeight + plugHeight + 0.15 + 0.50,” then hit enter. After this my sketch looked like

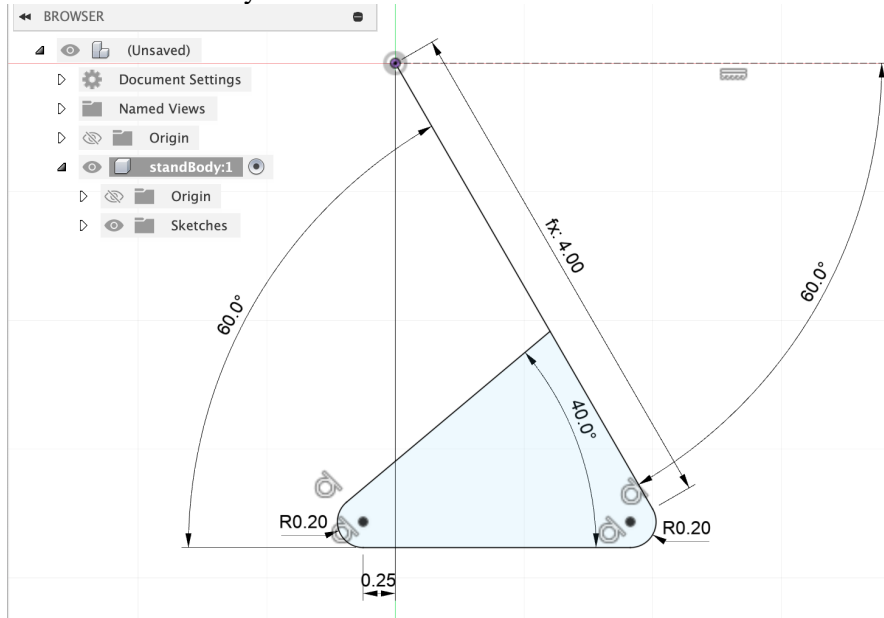


- Next we need to dimension the base. The base needs to extend past the center of mass of the phone. Since I made my back plate 50% of my phone height, the center of mass of my phone is approximately at the origin. I used the “Sketch Dimension” tool to set end of the base to be 0.25”

to the left of the top of the stand. After this my sketch looked like



- The last step is to make sure that the end of the stand meets the back of the support. We will do this using a *constraint*. Select the “Coincident” constraint (two perpendicular lines) in the Constraints section of the tool bar. Click on the end of the phone stand then click on the back of the stand. After this your sketch should look like

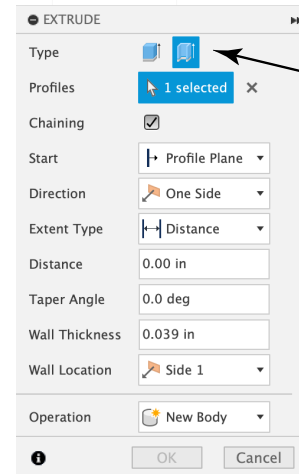


- Click “Finish Sketch” in the far right of the tool bar. At this stage, it is useful to double click on your sketch name (which is probably Sketch1) and change it to a descriptive name (such as bodySketch).

Extruding the Stand Body

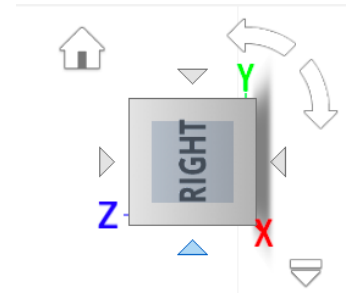
The next step is to *extrude* the stand body into the third dimension while also giving it some thickness. In doing this we want to make sure that the area enclosed by the base (shaded light blue in the sketch above) remains hollow. To do this, we will use a relatively new feature of Fusion 360 known as a *thin extrude*.

- Select your stand body by clicking and dragging.
- Under the Create menu of the tool bar, choose “Extrude.” This will open the Extrude menu on the right of your screen. At this point, the interior of the base (shaded light blue above) turns a darker blue, indicating that whole section will be extruded. To prevent this, click the Thin Extrude icon in the Extrude menu. Click the interior of the base to deselect it, then click the line that define the stand body to select it. The line should now be dark blue and the interior light blue.
- In the Extrude menu, set the Distance of the extrusion to standWidth. Set the Wall Thickness to standThickness. Click OK.
- You should now have a 3D stand body. You can check this by rotating the view. You should also have a new Body entry in your standBody component. I would change the name from Body1 to standBody before proceeding.



Navigating in 3D

There are a number of different ways of changing your view in Fusion 360. On a Mac with a trackpad, you can use the standard “pinch” command to zoom. Rotation is activated by holding down option-shift and dragging on the trackpad. Alternatively, the row of icons at the bottom of the screen allow you to orbit, look at, pan, and zoom. Hit escape to exit these commands. Finally, there is a widget (shown at right) in the upper right hand corner of your screen that you can use for navigation. The arrows appear when you hover over the widget.

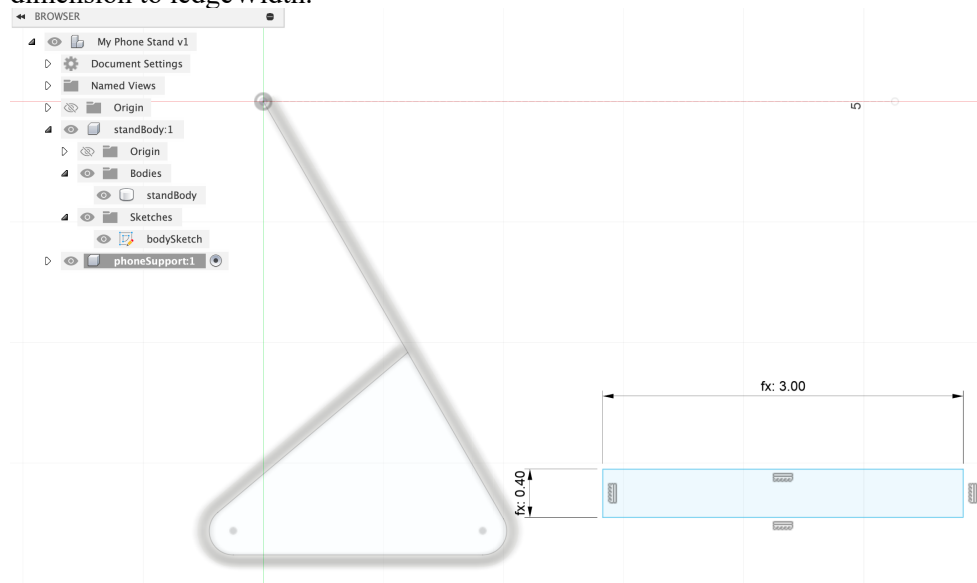


Sketching the Phone Support

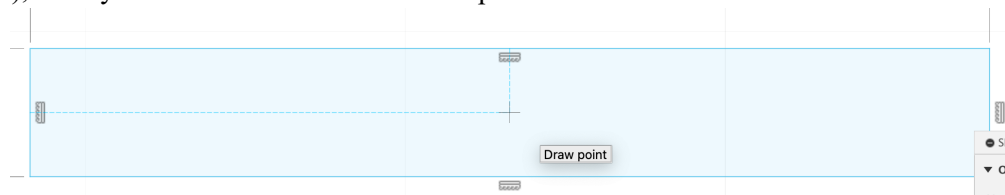
The next step is to create the support that will hold the charging cable and the phone. In principle, we could just draw and extrude this from the phone support, but it is very awkward to do so. Instead we will create a new component. After this component is complete, we will attach it to the stand body.

- At the very top of your browser list, you will see the name of your drawing. If you have not saved your drawing yet, it will be named “(Unsaved).” Now would be a good time to save it. By default Fusion 360 saves everything to the cloud, which is fine for now.
- Click on the little dot to the right of your drawing name to activate the drawing. Create a new component from the Create menu. Give the component a name and click OK.
- Create a new sketch in your component. Put the sketch on the same plane as the stand body.
- Draw a rectangle that is approximately 3" wide and 0.5" tall. It doesn't matter where you do this, but I sketched it a little to the right of my stand body. While you are drawing the rectangle, tab between the two dimensions. Set the horizontal dimension to standWidth and the vertical

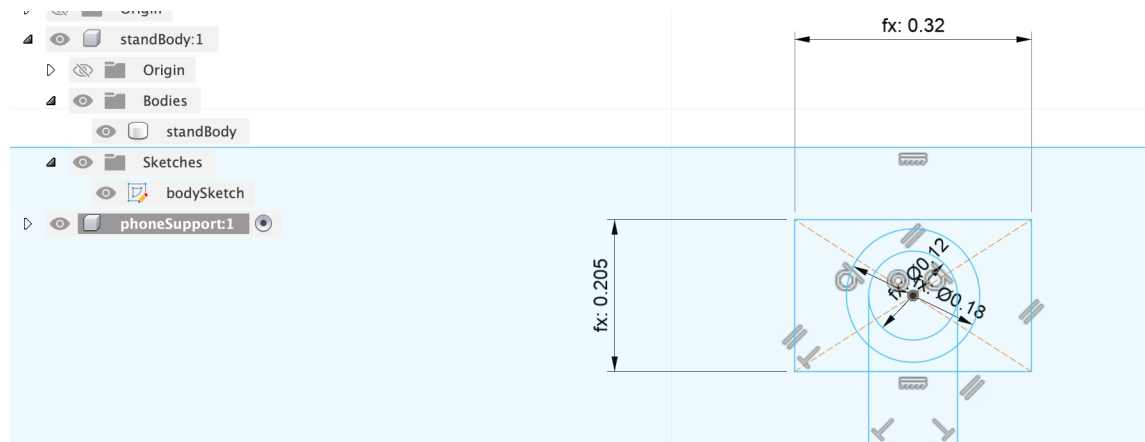
dimension to ledgeWidth.



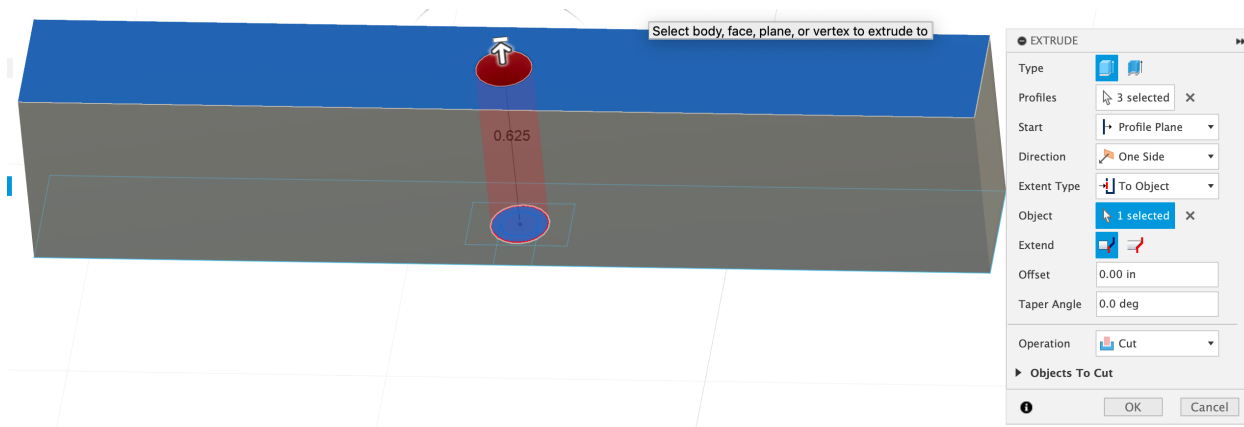
- Zoom in on your new sketch. To make the next step easier, we are going to define a point that is at the center of our rectangle. Select Point from the Create menu. Hover over the middle of the left side of the rectangle until you see a pale green triangle icon, which represents a center point. Hover over the middle of the top of the rectangle until you see the triangle icon again, then move your mouse to the center of the rectangle. When you see pale lines extending to each midpoint (see below), click your mouse to define the center point.



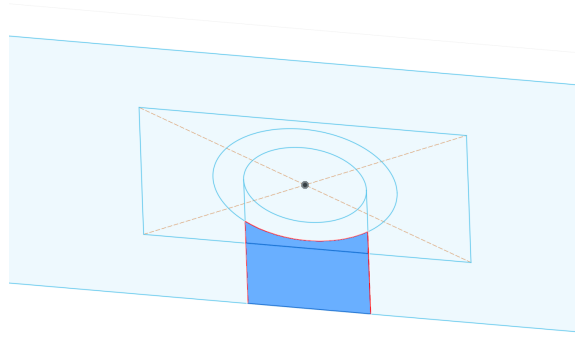
- Draw two concentric circles around the center point using the Create Center Diameter Circle command under Create > Circle. The first circle should have a diameter of cableDiameter, and the second circle should have a diameter of cableGrommet.
- Draw a rectangle centered at the center point using the Create > Rectangle > Center Rectangle command. The rectangle should have a horizontal dimension of plugWidth and a vertical dimension of plugDepth.
- Draw a line from the left hand side of the smaller circle straight down to the bottom of the support. Draw a second line from the right hand side of the smaller circle straight down to the bottom of the support. Note the little \perp icons which signify that the vertical lines are perpendicular to the base. At this stage your sketch should resemble



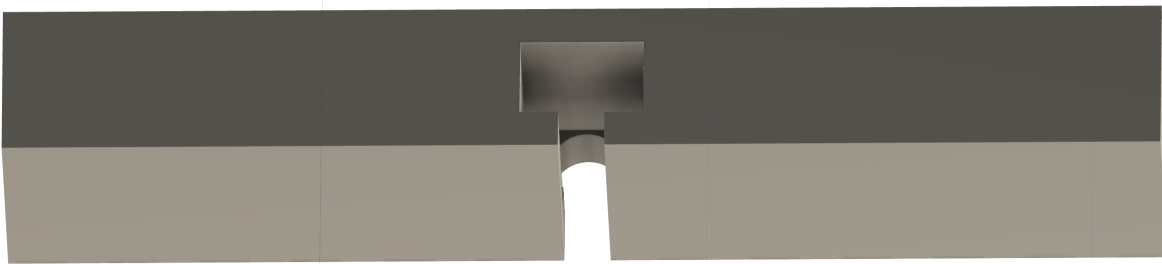
- The next step is to extrude the phone support. Click on Finish Sketch in the tool bar and give your new sketch a descriptive name. Select everything in your drawing of the phone support. The entire rectangle and everything inside it should turn dark blue. Under the Create menu, select Extrude. In the Extrude panel, set the Distance to `plugHeight + 0.15`.
- You should now have a new Body in your component. Give the new body a descriptive name.
- The next step is to cut out a hole all the way through our phone support for the cable grommet. To do this, make sure that your phone support sketch is visible (dark eyeball next to sketch). On your sketch, select the entire area inside the larger circle, then select Create > Extrude. (It may help to temporarily hide the phone support body.) In the Extrude panel, change the Extent Type to “To Object,” then click on the opposite face of the support. Your Operation should automatically be set to “Cut.” See the screen shot below.



- Click OK to make the cut. Your support should now have a circular hole all the way through it.
- Select everything inside the small rectangle on your phone support. Select Create > Extrude and set the Distance to `plugHeight`. Your phone support should now have a recess for the plug.
- The last step is to create a slot for the phone cable. Select the region shown in the sketch below, then create an extrusion that cuts this region away from the entire phone support.



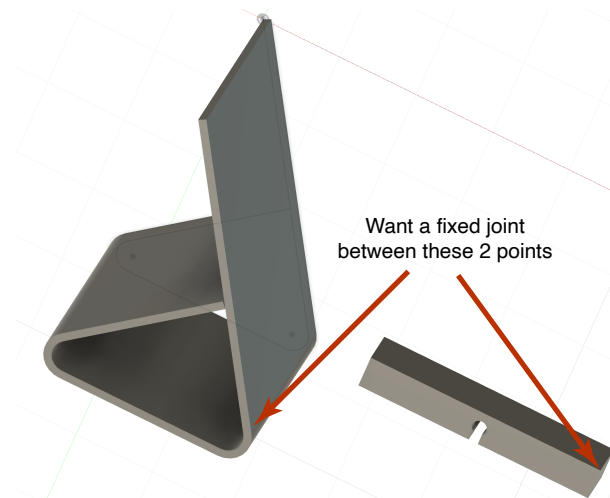
- Your phone support is now finished. It should resemble the sketch below.



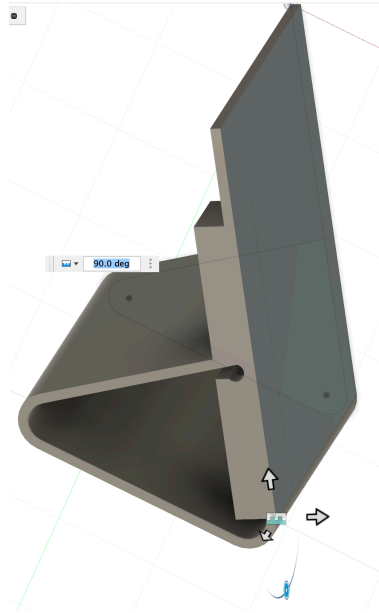
Assembling the Stand

Now we need to attach the phone support to the stand body. We will do this by making a *joint* between the two components. This joint will be between a point on the lower left backside of the phone support and the face of the phone stand.

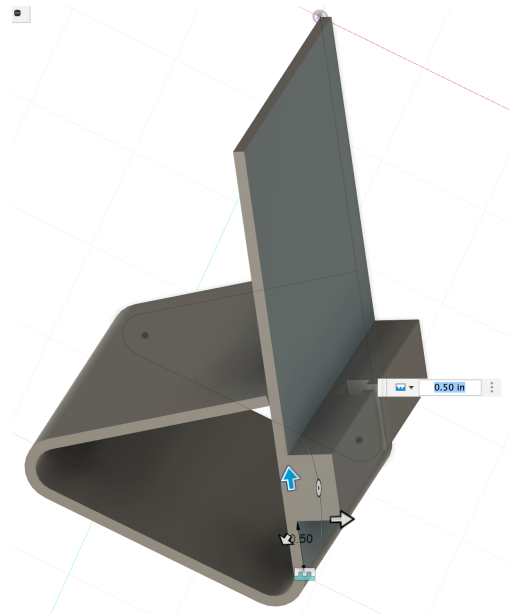
- At this point Fusion 360 knows about the points that defined the phone stand (*i.e.*, the curvy line), but it doesn't "know" about points on the face of the support. To rectify this, click on the circle to the right of the standBody component to make it active. Use the Create menu to make a new sketch on the standBody. Click on the front face of the stand body to define the plane of the new sketch.
- Click on the circle to the right of the name of your drawing at the top of your Browser list. This will make both components active.
- Rotate your drawing so that you can see the back of the phone support and the front of the stand as shown below.



- Under the Assemble menu in the tool bar, select Joint. Hover over the desired face on the support until it is highlighted, press command on Mac or control on Windows, then click on the desired point on the support. Hover over the face of the stand body until it is highlighted, press command/control, then click on the desired point on the stand body. The support should now fly over to the stand body.
- There is a 50:50 chance that your joint will have the wrong orientation. Here is what my initial joint looked like.



As you can see, there is a joint between the correct two points, but the orientation is funky. To fix this, select “flip” in the Joint window. Use the circular slider to set the correct orientation, then slide the support 0.5" up the face of the support. My assembled joint is shown below.



Possible Modifications

The basic stand is complete, but you are free to do additional modifications. For example, I added a hole underneath the phone support so I could thread the cable from the back of the stand to the front of the stand.

My Apple charging cable has an oval cross section, not a rectangular cross section. To fix this issue, I added fillets to the recess for the charging cable.

Printing the Stand

The first step in printing the stand is exporting a .stl file. Then the .stl file needs to be “sliced” to create a file of print commands. I opened the .stl file in Prusa Slicer, which is the slicer recommended by my 3D printer company, Prusa. A screenshot of the sliced design is shown below. The phone stand is in orange. The green “arms” are supports added by the slicer to enable printing. At the bottom right, you’ll see this design requires ~15.4 meters of filament at a cost of ~1.67 € and will take ~ 4 hours to print.

