

SolidWorks Tutorial and Example Task

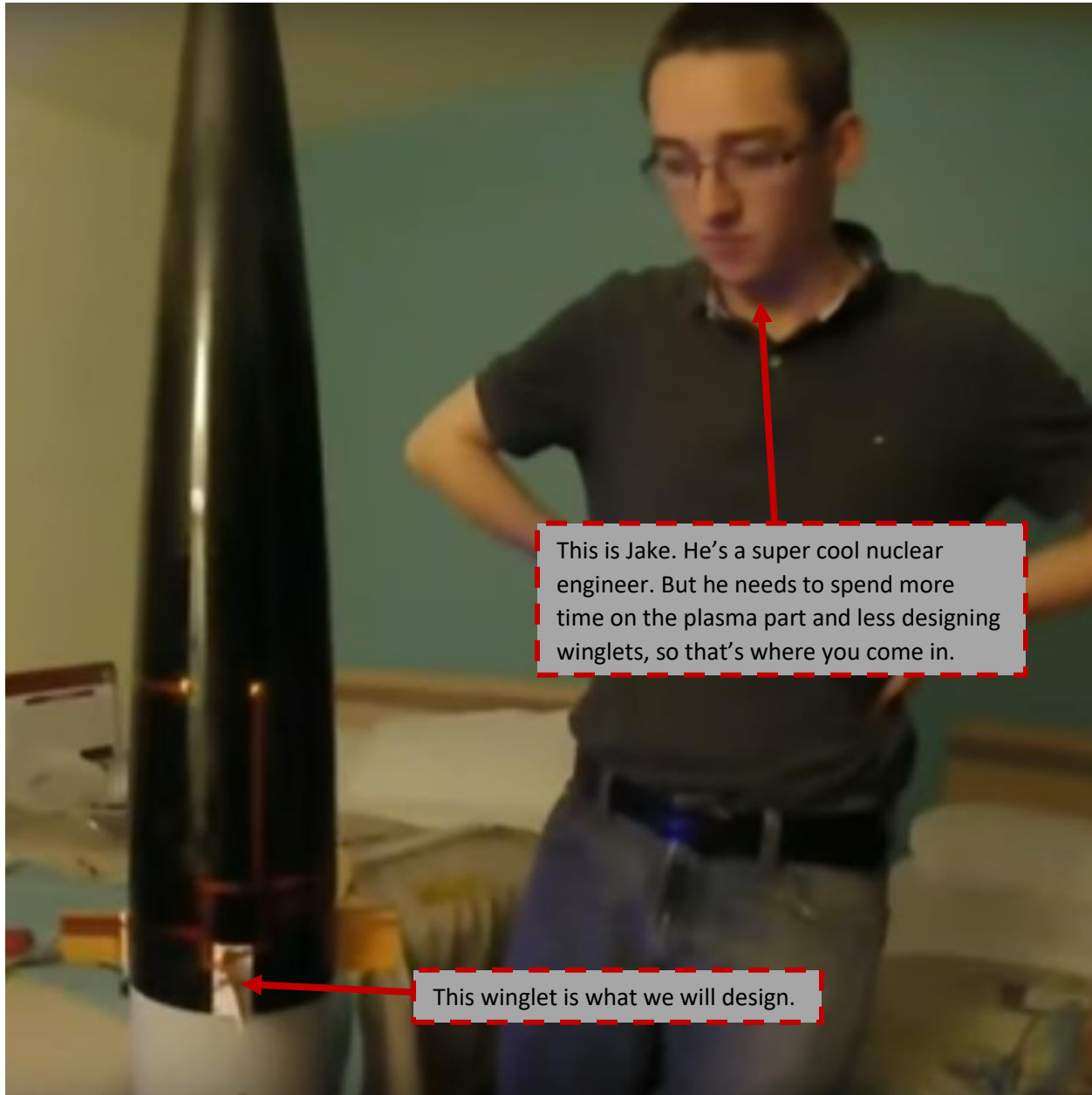
What is Rocket Team?

Rocket Team is an MIT Student Club that designs, builds and launches rockets to supplement the MIT curriculum. In recent years, the team has typically built one rocket each year for the Intercollegiate Rocket Engineering Competition (IREC), which focuses on accurate altitude, build quality and payload.

To build a single rocket, the team consists of several *subteams* (propulsion, recovery, payload, avionics, structures, etc). Each subteam has a lead, who will assign other members of the subteam certain tasks or projects to complete. Obviously, we have all sorts of tasks, but this tutorial will show you how to complete a typical task you might be assigned.

The Example Task

The 2015 rocket investigated the effect of a *Dielectric Barrier Discharge Actuator* on a stalling aerofoil; basically, we made a winglet that was stalling (at too much of an angle to produce much lift) and wanted to see if a thin layer of high voltage plasma moving with the surface could unstall the winglet.



So one task you might be assigned is to design and make these little winglets. Not a trivial project by any means, but it's definitely achievable for a newbie and easy enough to test. Here are the requirements:

- Must attach to the nosecone
- Must stall airflow during flight
- Must not break
- Must not be conductive (because of plasma)

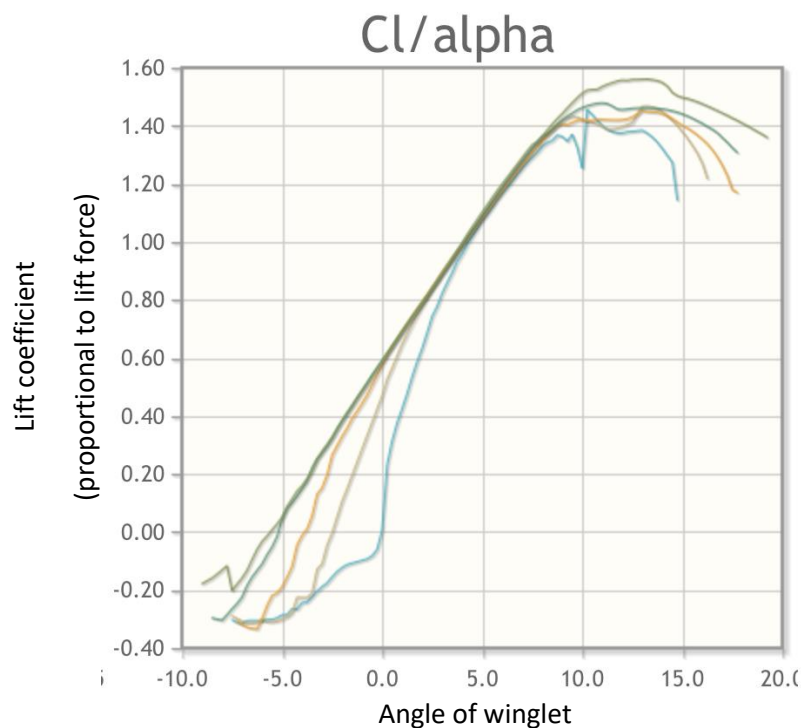
Initial Design

Before you start on SolidWorks, we need an idea of what we're going to make in the first place! If you already know what you want, take a minute to draw it on paper.

INSERT DRAWING HERE

If you aren't quite sure what to make, try talking with a fellow engineer about the task; there's something about bouncing ideas back and forth that seems to result in good ideas coming up quickly (particularly if you both use a whiteboard). If you still aren't sure, ask your subteam lead; they're always happy to think about a rough solution to the problem with you.

For a winglet in particular, we need to know the angle relative to the airflow. Let's take a look at the below plot:



This plot shows the angle of the winglet against the lift force, and we can see lift increases up to about 10-15 degrees. Therefore, a 20 degree angle should ensure the winglet is stalling (which meets our requirement).

Also, your subteam lead will tell you the winglet should be about 5cm tall and 5cm long.

There are a few other issues you should sort out before you open SolidWorks:

How will your part attach to the rocket, or whatever its parent assembly is?



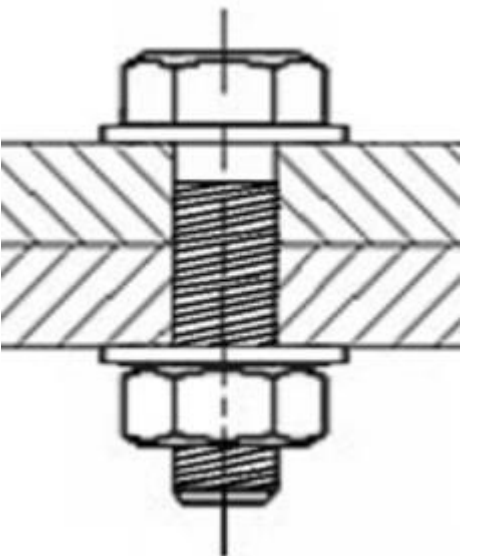
The first challenge is attaching your part to the rocket or another assembly. Here's what you need to think about:

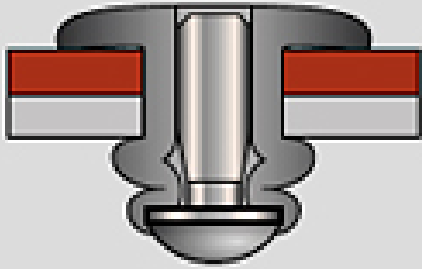
- Does your part need to move? Rotate or slide?
- How strong does the attachment need to be?

- Does your part need to be removable/replaceable?
- Can your part be installed easily?
 - If you can't reach the part with two hands or can't see it easily, it will be hard to install

In our case, the part doesn't need to move, but it does need to be replaceable (if the winglet breaks, we want to replace it). Also, the part should be easy to install, because it's on the outside of the rocket. The attachment needs to be quite strong though, since the winglet will be pushed back by the air. We're not sure how strong at this point, but probably stronger than just duct tape...

So, what options are available?

Method	Picture	Pros	Cons
Tape (duct, electrical, masking)		Super fast Super easy	Prone to peeling Looks terrible
Glue (epoxy, hot glue, superglue)		Fairly fast Can look nice Can be strong	Can't be removed May not adhere super well
Screws		Very strong Adjustable tightness Can be removed Can look nice	Slow to install (holes need to line up) Might need access to other side Bumpy

Rivets		Fairly strong Can look nice	Slow to install Can't be removed Only thin materials
--------	---	--------------------------------	--

In this case, we want the winglet to be removable, so we must choose screws. Specifically, we will put a screw in from the outside, and attach to a nut inside the nosecone.

You may want to think about using threadlock or a lockwasher to make sure the screws don't accidentally loosen. But this is just an intro, and your subteam lead would totally point that out if you need it.




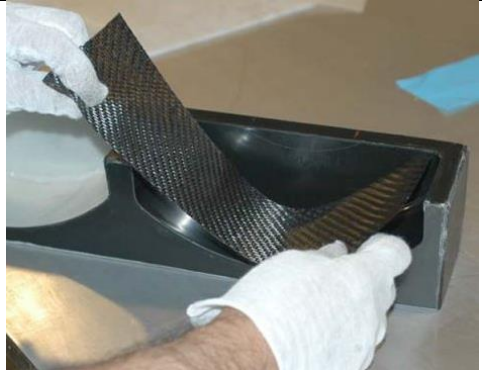
How will you make the part?

Of course, this part needs to eventually be real, not just a SolidWorks model. What should we think about?

- How strong does the material need to be?
 - Most of the time, you can make it stronger just by making it thicker. But sometimes, you can't because there isn't space, or because weight is super important; then you need super-strong materials.
- How complicated is the part?
 - Some processes can make super complicated parts, others might be faster or cheaper because they can only make certain kinds of parts.
- What tools are available?
 - Unfortunately, we don't have any titanium 3D printers. So you'll have to find another way.

Again, here are some options we have:

Process	Picture	Pros	Cons
---------	---------	------	------

<p>Machining (mill, lathe)</p>		<p>Can cut almost any material Very accurate Very smooth surfaces</p>	<p>Very manual Slow Not as flexible as 3D printing</p>
<p>3D printing</p>		<p>Very automatic Can print any shape parts, with super complicated features</p>	<p>Slow Few materials Not so accurate Ridged surfaces</p>
<p>Waterjet</p>		<p>Very fast Accurate Can make smooth surfaces Almost any material</p>	<p>Can only make 2D parts</p>
<p>Composite layup</p>		<p>Very strong materials Very lightweight Very accurate Very smooth surfaces</p>	<p>Super slow and hard to setup Can only make thin parts Very manual</p>

Our part is clearly going to be 3D, and isn't worth the complicated setup to do a composite layup. Machining might be a good choice (even though metal isn't allowed due to conductivity), because of the

surface finish. 3D printing might also be a good choice, because it's super easy, and we can sand it to make it smooth. Both are good choices, but we chose 3D printing in 2016 just because it was easier and faster.

How strong does the part need to be?

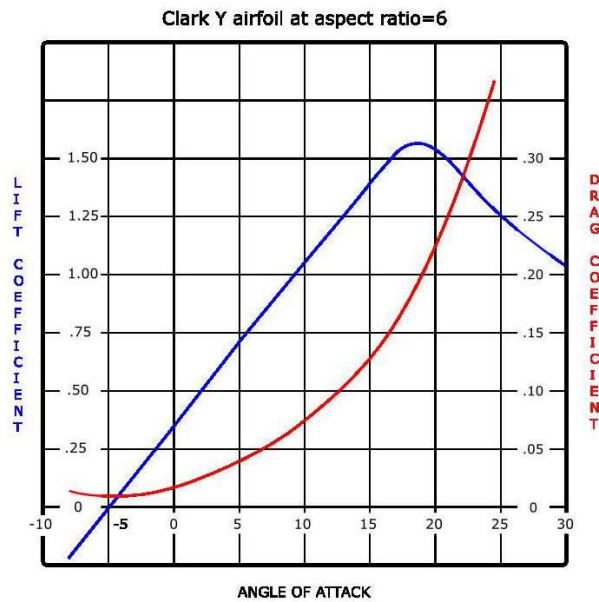
This can be the mathiest part of the challenge; if you don't know the right equations, ask your subteam lead where you should start. If you do know all the math, double check with your subteam lead to make sure you did it right!

The first part is: how much force will there be?

We want the maximum possible force on the wing, which intuitively will happen at maximum speed (about 400 m/s). We'll use a couple of equations from aerodynamics:

$$L = \frac{1}{2} C_L \rho v^2 A \text{ and } D = \frac{1}{2} C_D \rho v^2 A$$

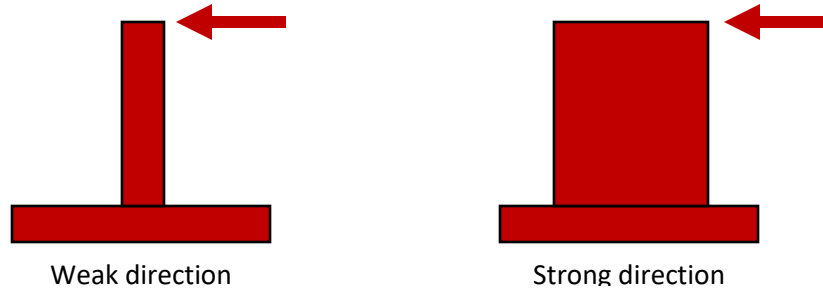
We're not sure exactly what C_L and C_D are, so let's look online for an example:



Now, to apply the equations, we just need $C_L = 1.55$ and $C_D = 0.23$ from the above plot, $A = (5 \text{ cm})(5 \text{ cm}) = 25 \text{ cm}^2$ from the length and height, and $\rho = 1.2 \text{ kg/m}^3$ from online. That gives us a lift force of $L = 370 \text{ N}$ and $D = 55 \text{ N}$.

Then the second part is: how thick does the wing need to be to not break?

The wing has a strong direction and a weak direction. If we want to see when it breaks, we should look at the weak direction, because that will break first.



The force in the weak direction is partially lift, and partially drag:

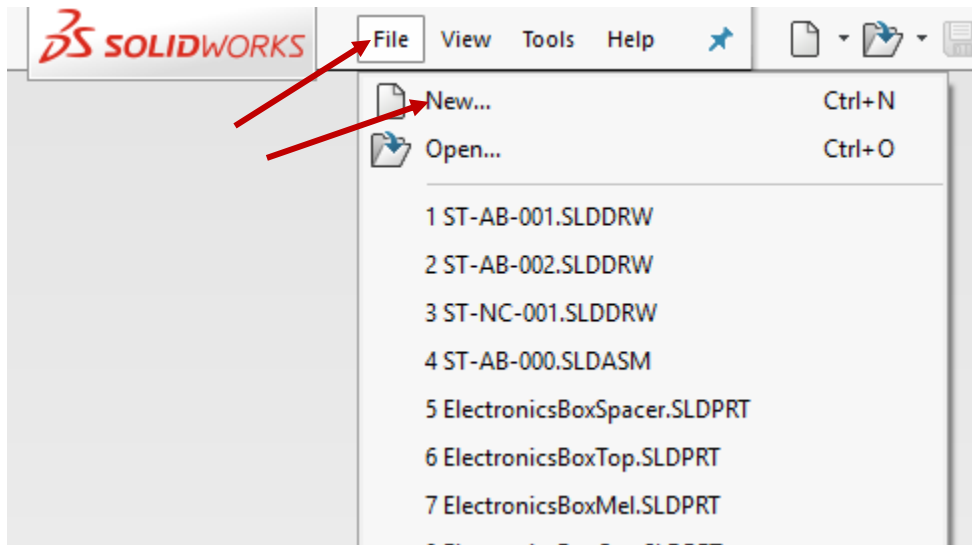
$$F = L \cos(20^\circ) + D \sin(20^\circ) = 366 \text{ N}$$

You definitely could calculate exactly what the right thickness would be, but we're not really sure about the material properties of our shoddy 3D printer. So I find the best way to do this is to go play around with a similar material, and then choose a thickness that feels comfortable to you and your subteam lead. Make sure you test it before you fly though!

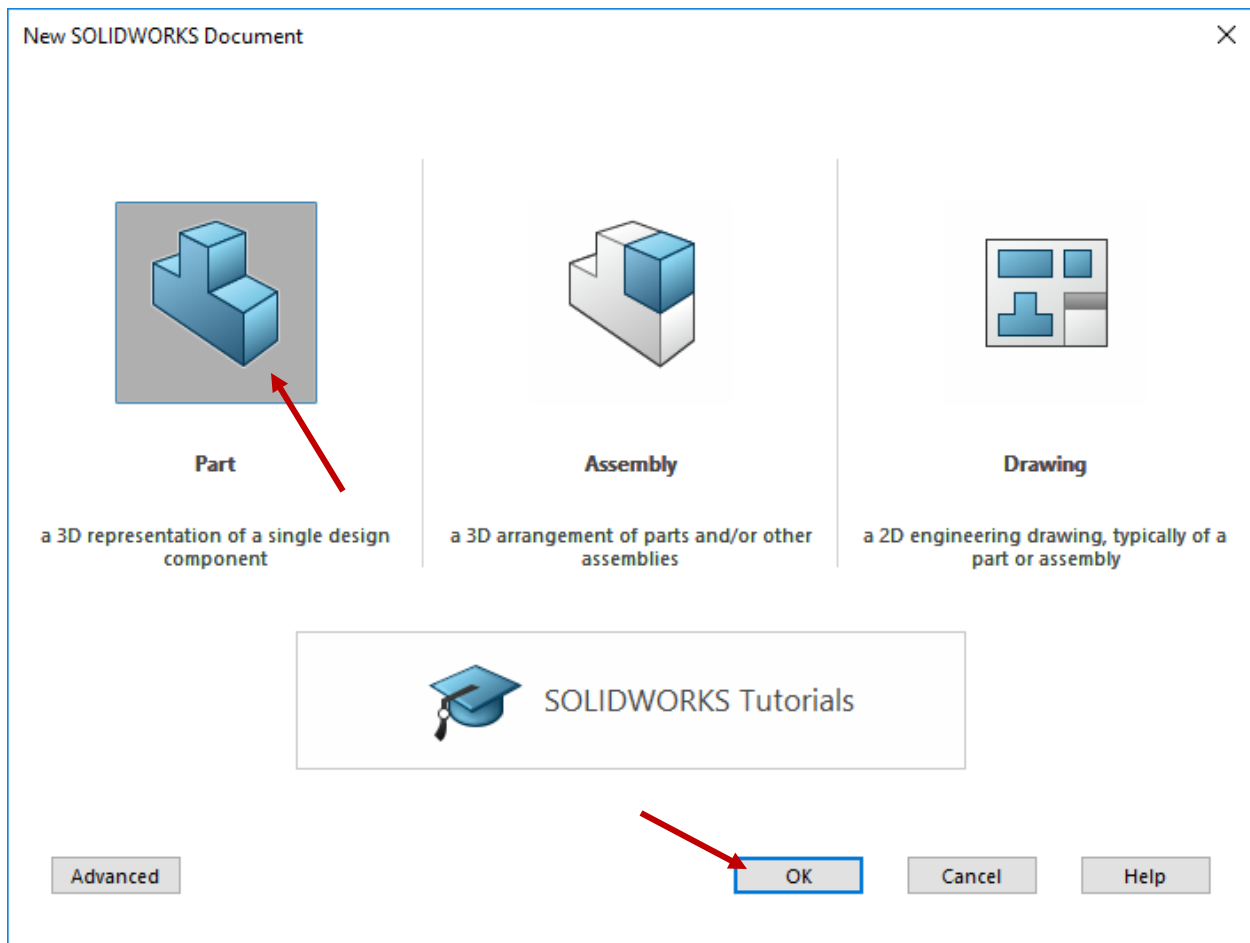
In this case, we'll make the middle of the wing about 0.8cm thick.

Finally, making the part in SolidWorks!

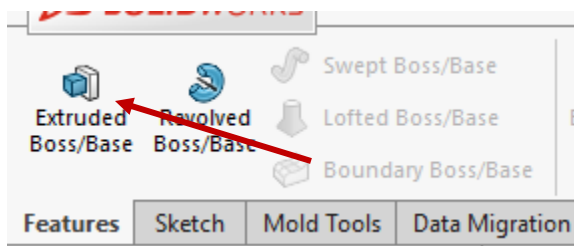
Fortunately, this part isn't too hard to make! Let's open up SolidWorks, and make a new part. Click File, then New...



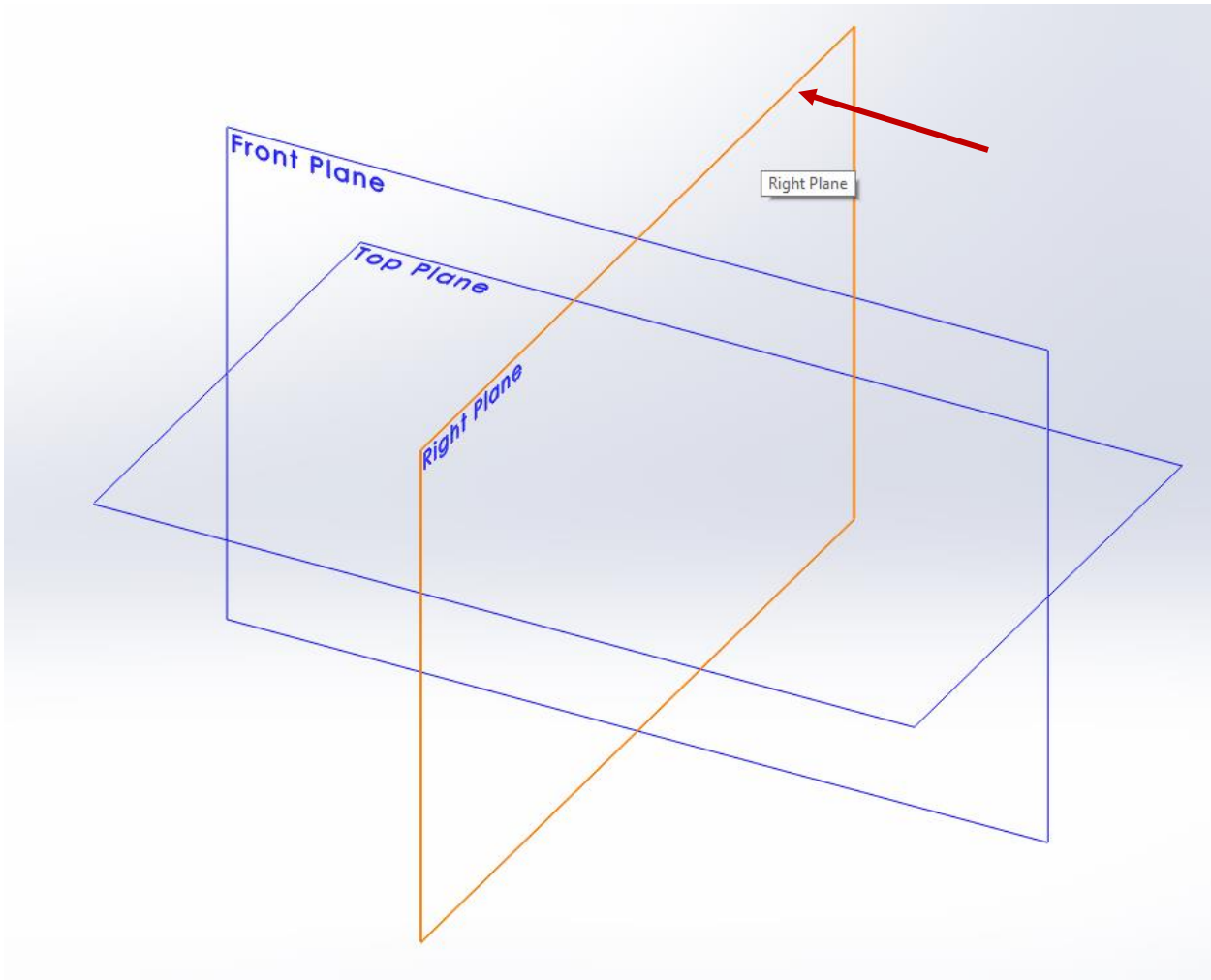
Select Part, then click OK



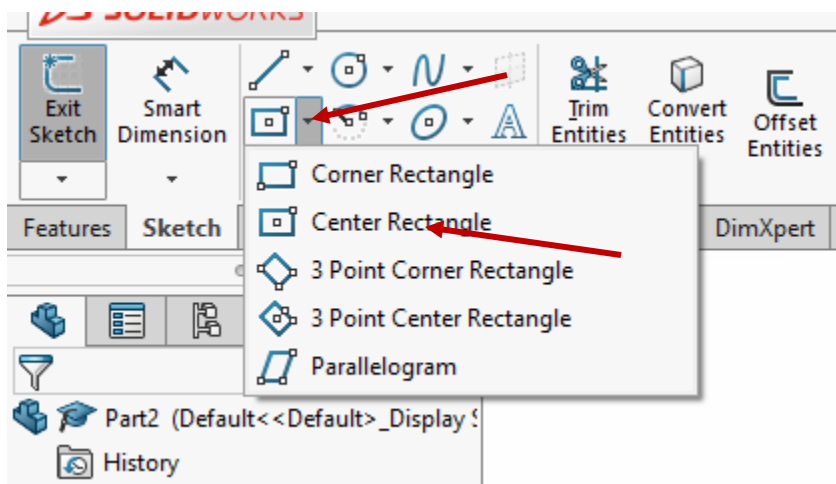
Click Extruded Boss/Base:



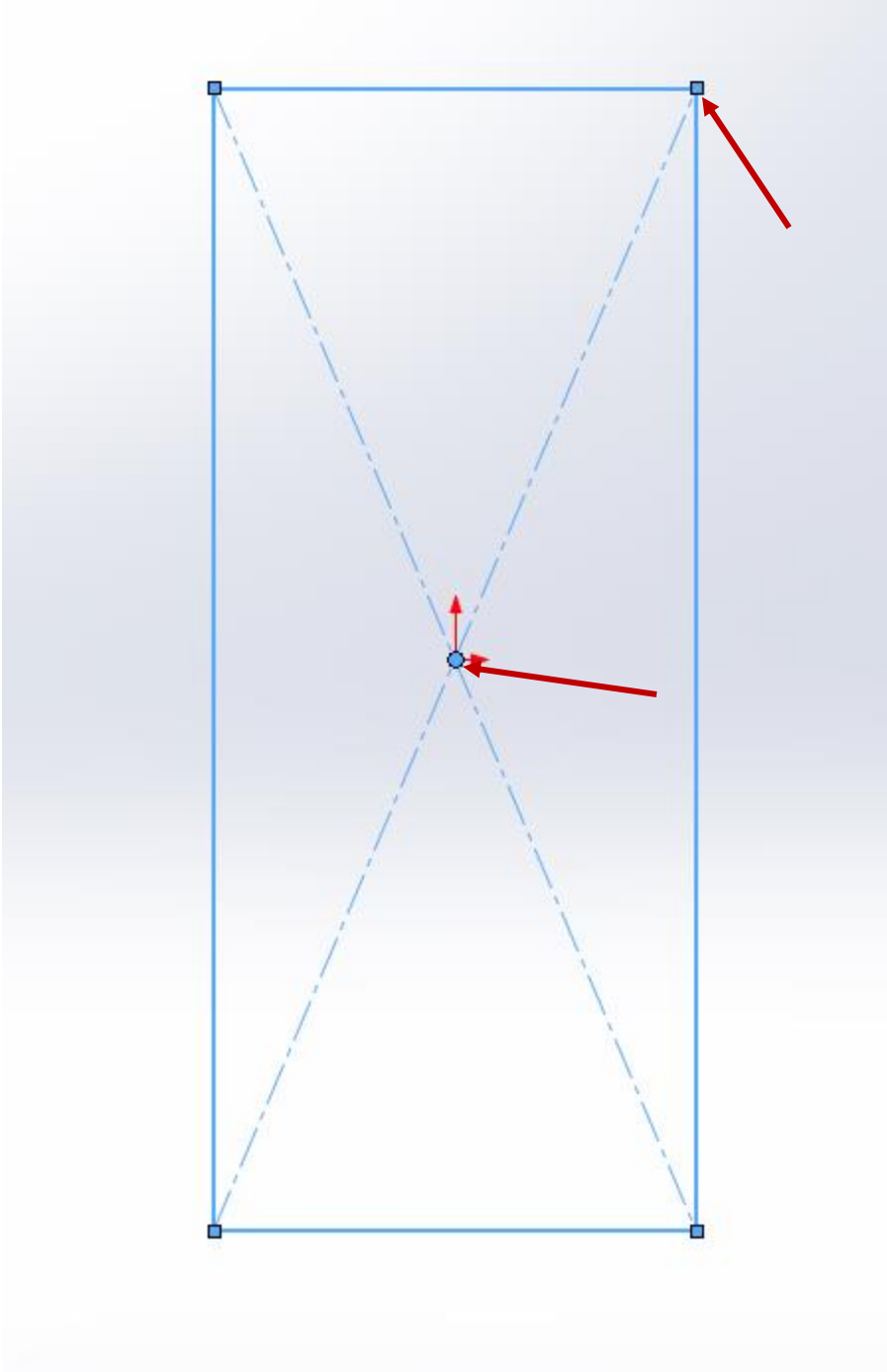
Select the *Right Plane*:



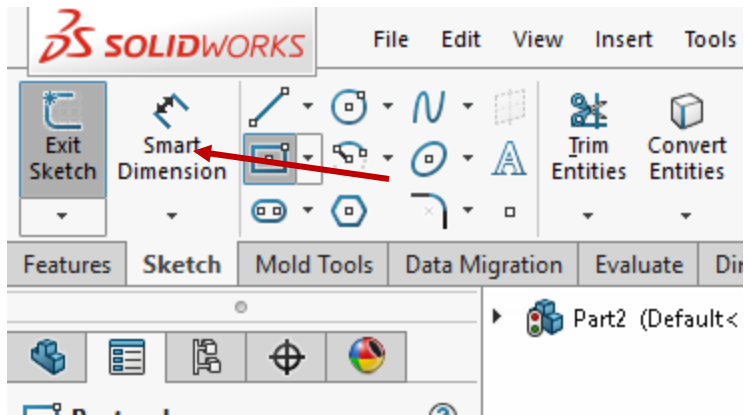
Click the triangle next to the *Rectangle tool*, and select Center Rectangle:



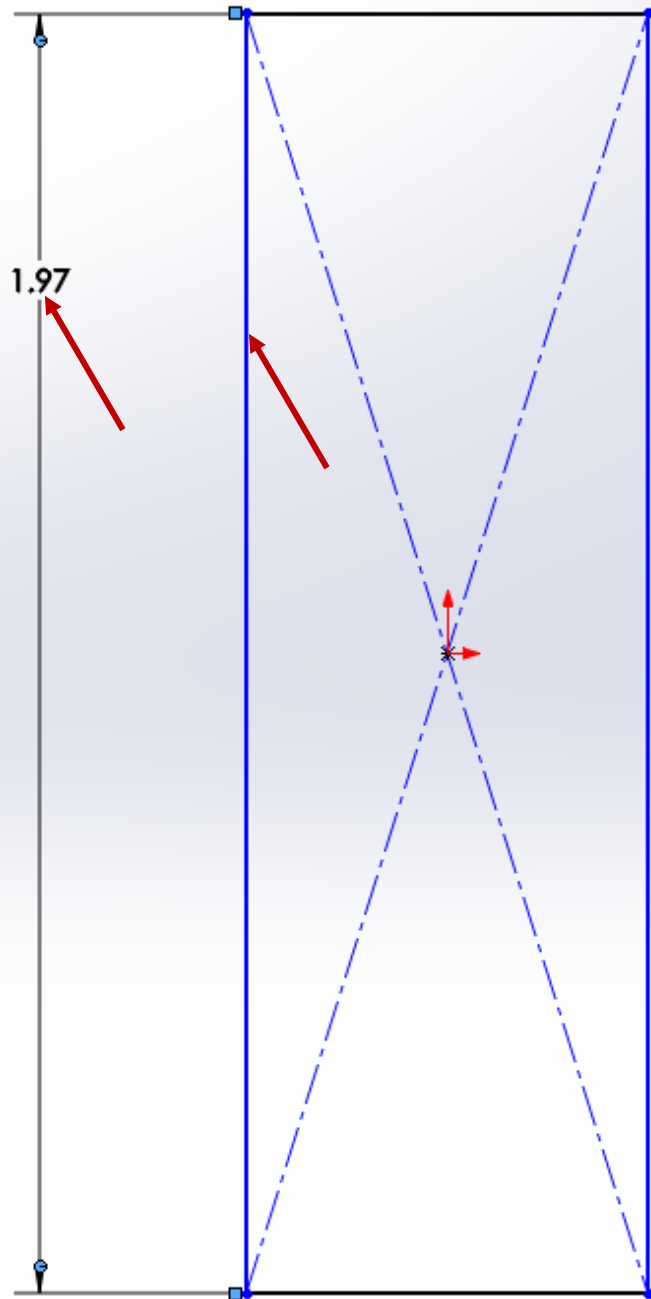
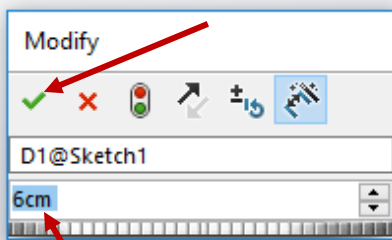
Click at the center mark to start drawing a rectangle, then click again at a corner to finish the rectangle.



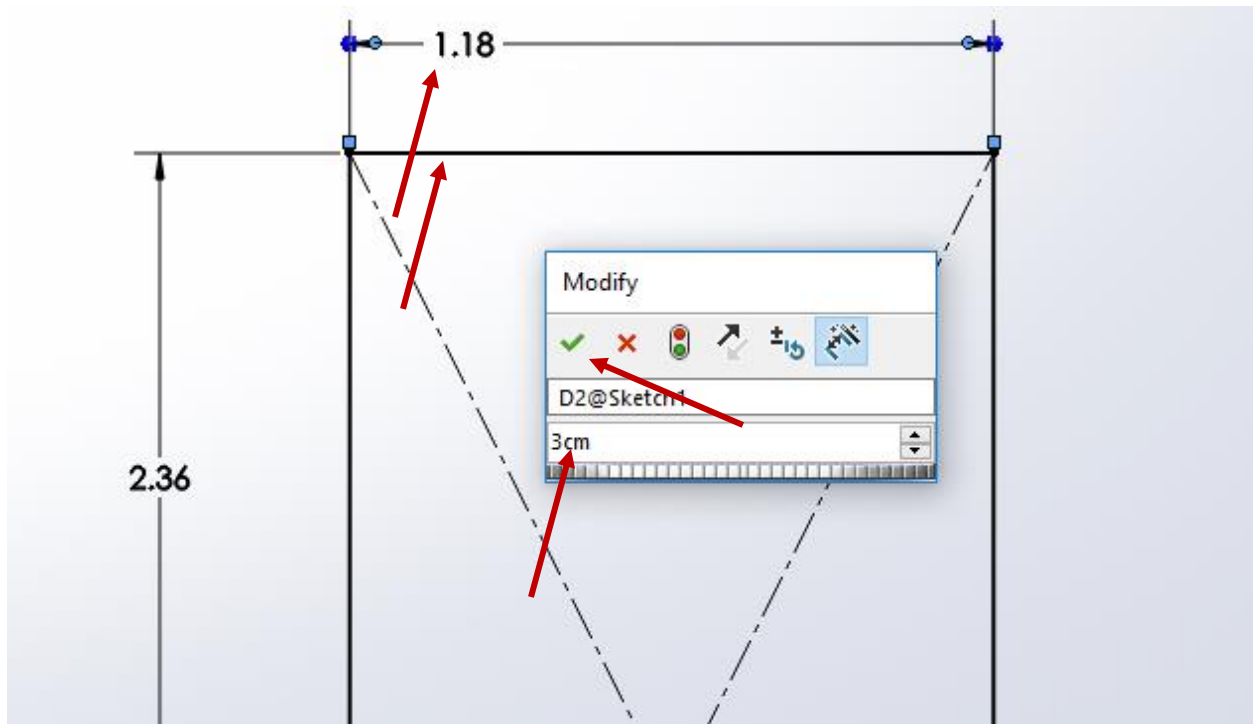
Click Smart Dimension:



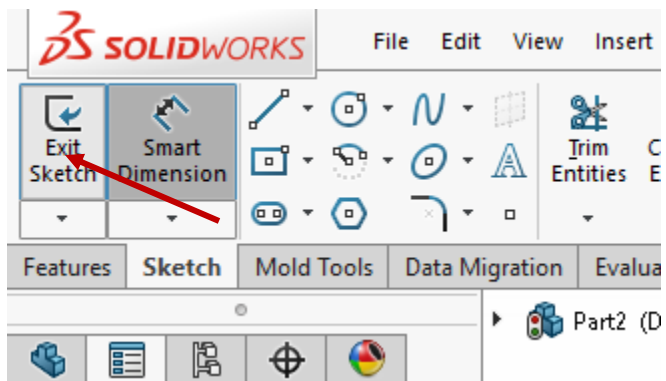
Click the left line, click next to it it, then type in 6cm, and click the check mark:



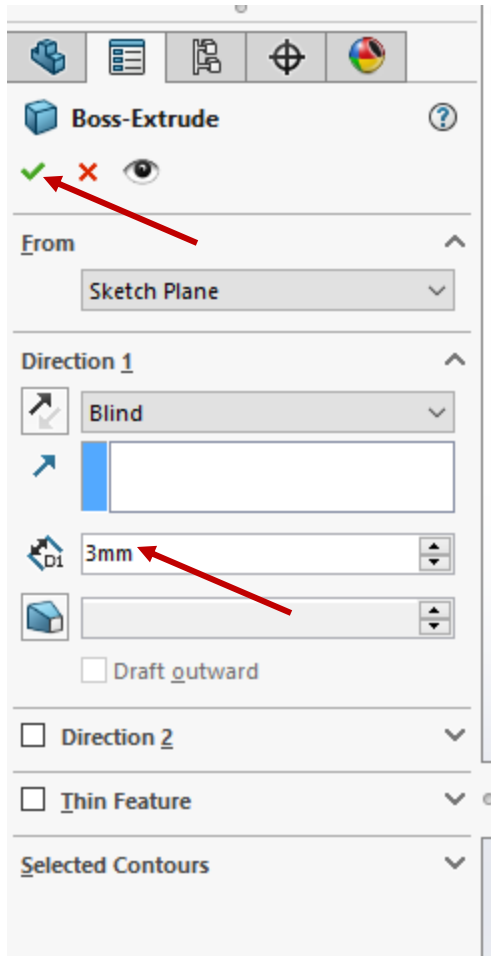
Click the top line, click above it, type 3cm, and click the green check mark. If there isn't space above the rectangle, try using the scroll wheel to zoom out.



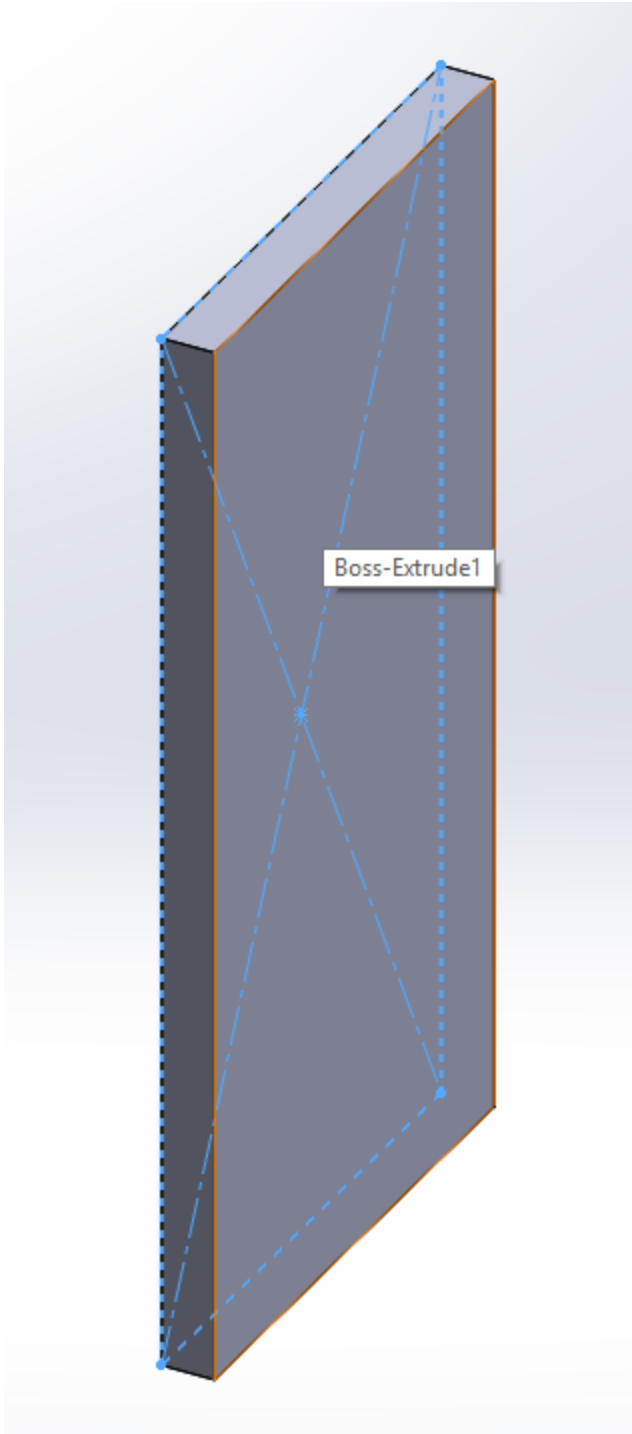
Click Exit Sketch:



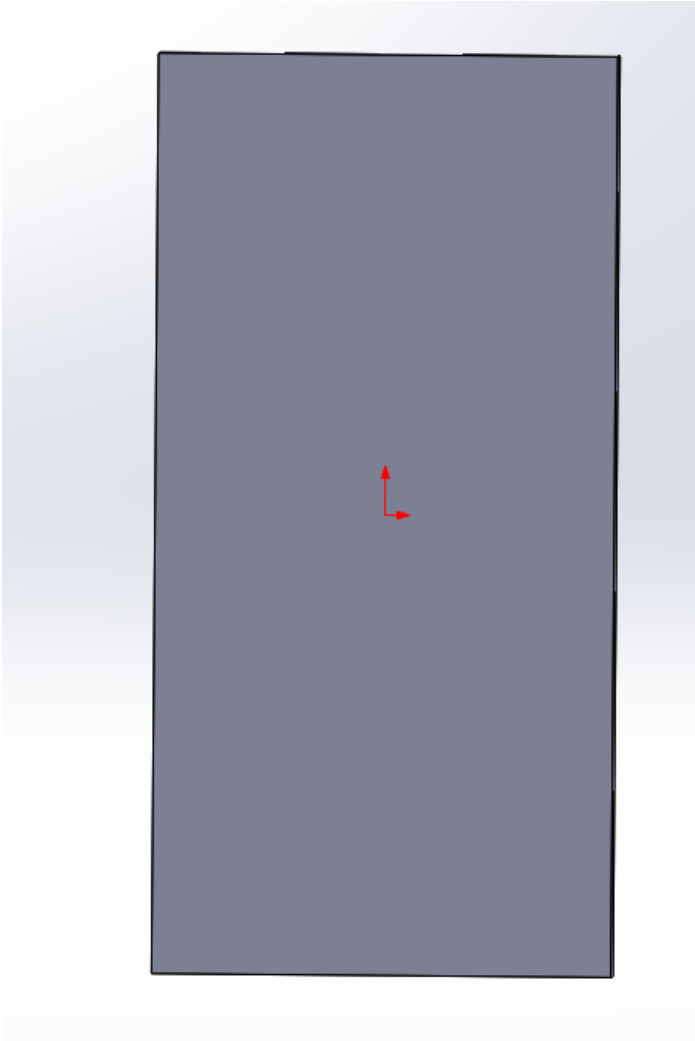
Type 3mm, and click the green check mark:



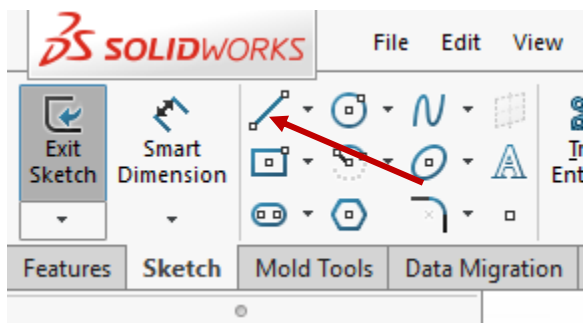
Click Extruded Boss/Base again, then click the rightmost face:



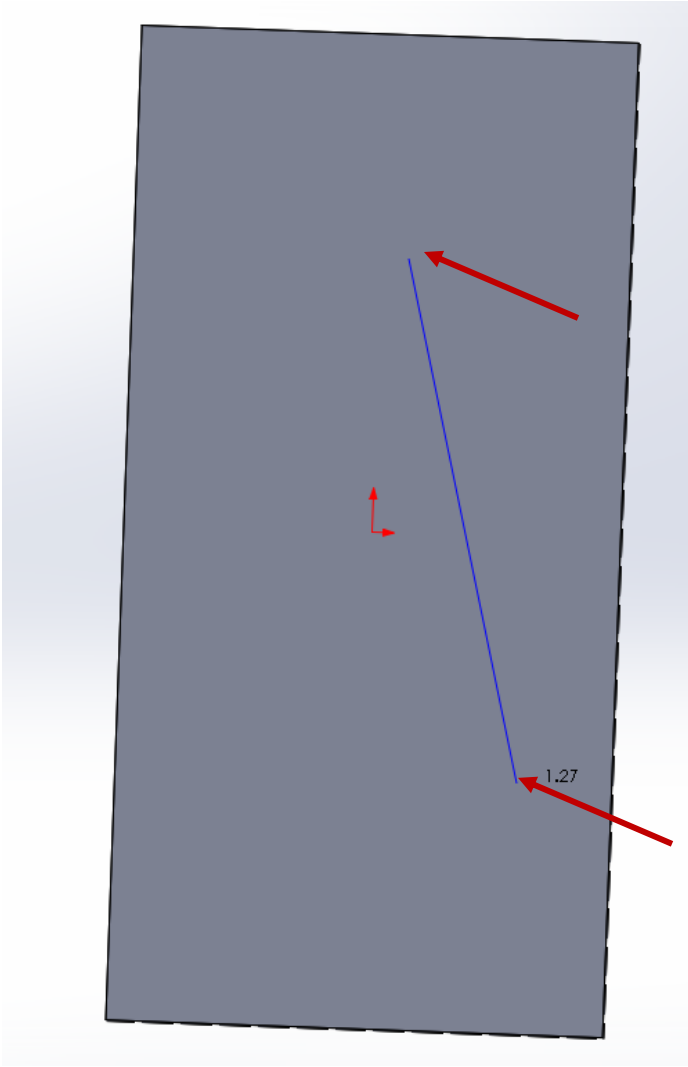
Hold down your middle mouse button and move the mouse to rotate until you're mostly normal to the face. (if you don't have a mouse, try the arrow keys)



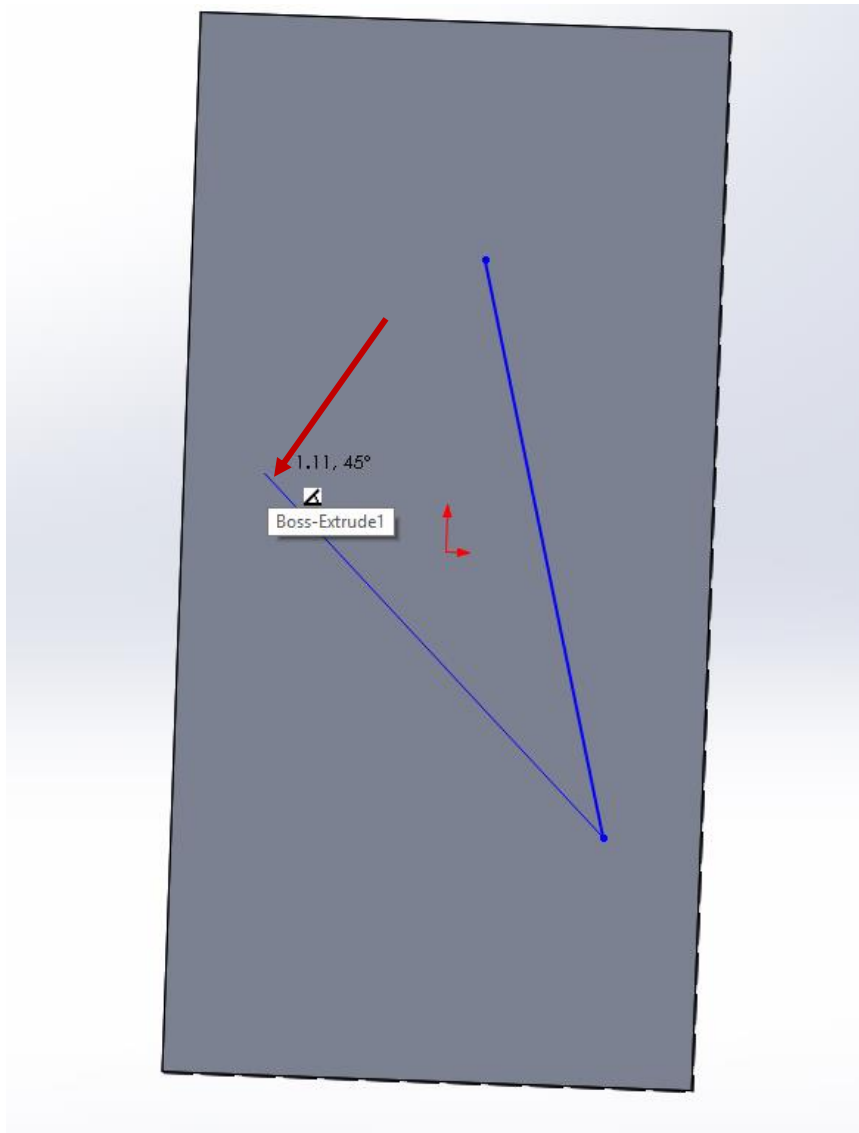
Click the *line tool*:



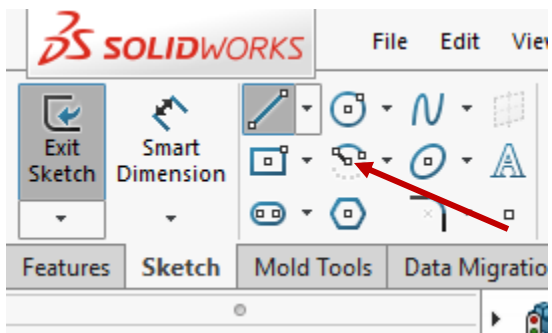
Draw a line from just above the center to bottom right by clicking twice:



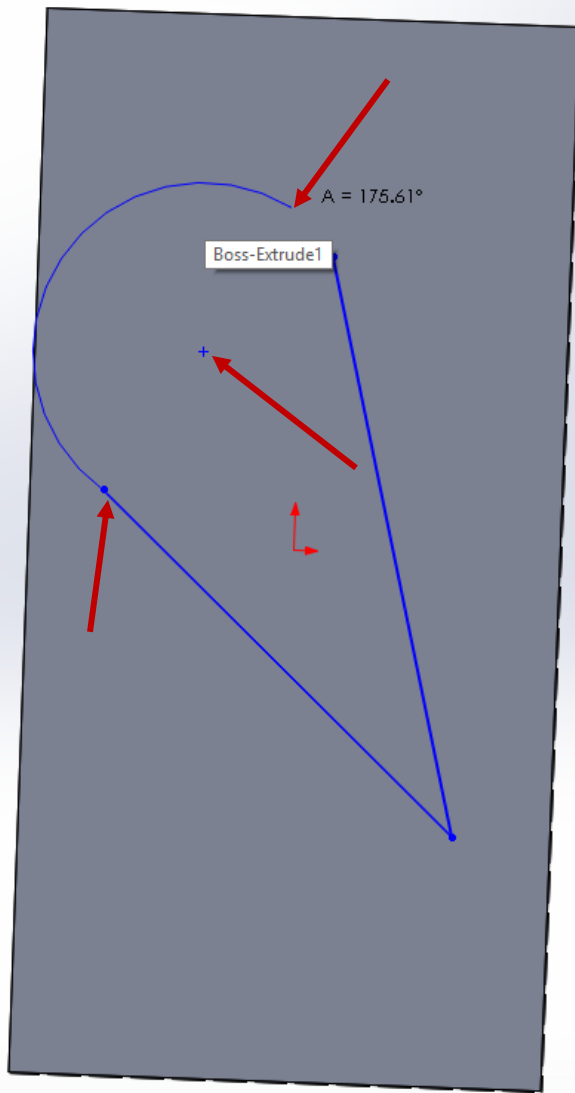
Click again to the left of the center:



Click the *Centerpoint Arc* tool:

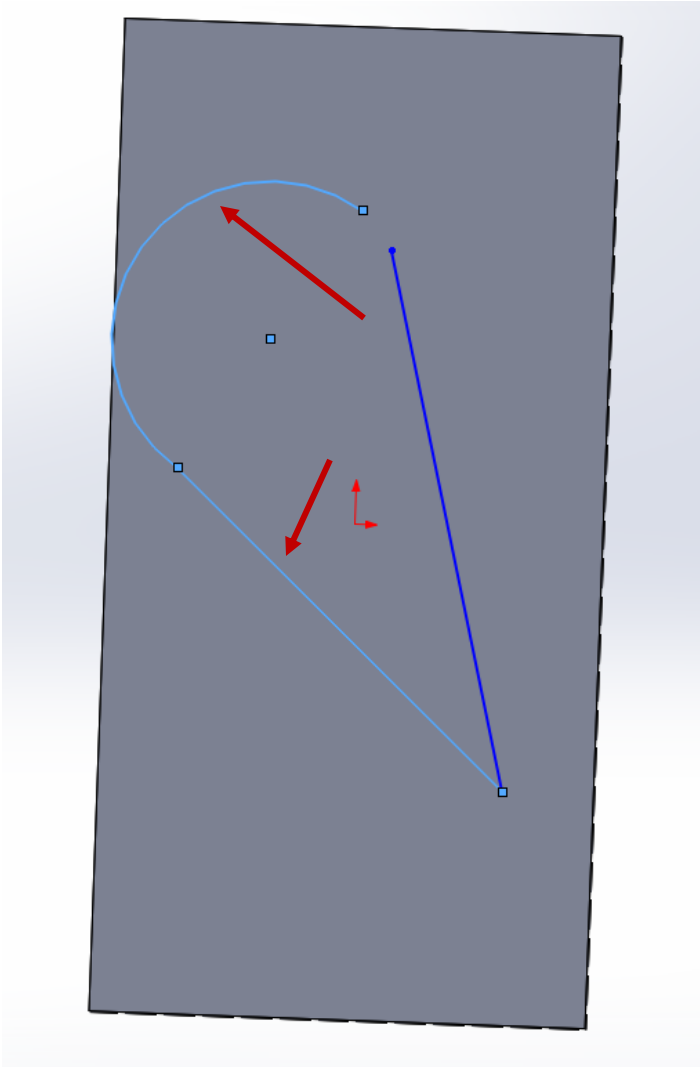


Click between the two endpoints, then click the left endpoint, move the mouse around in a circle to the other endpoint, then click somewhere near the right endpoint.

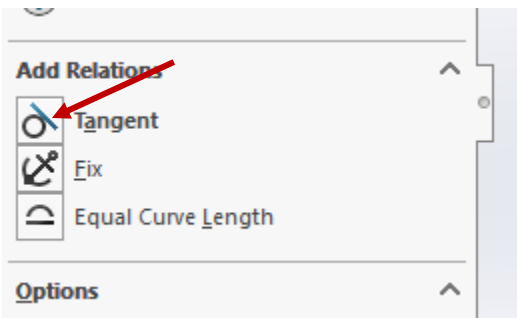


Press ESCAPE to quit the *Centerpoint Arc tool*.

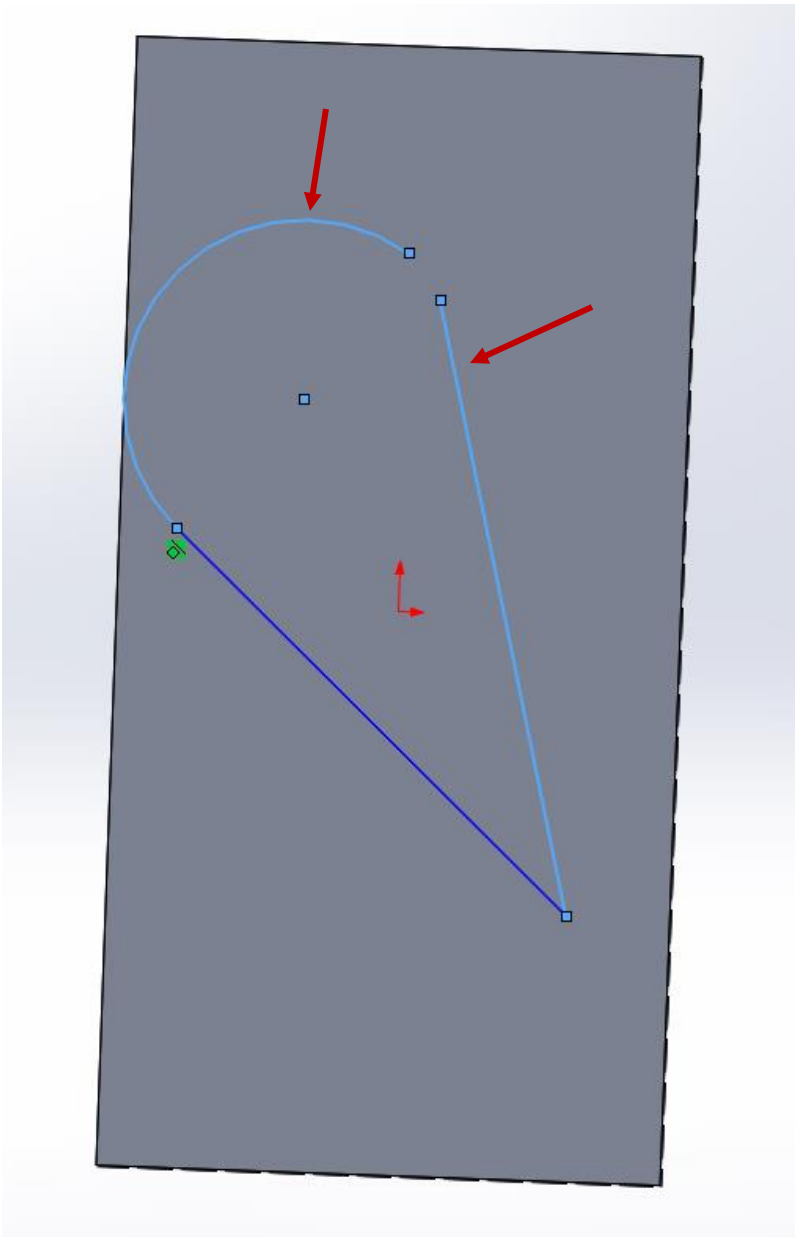
Click the arc, then hold CTRL and click the lower line.



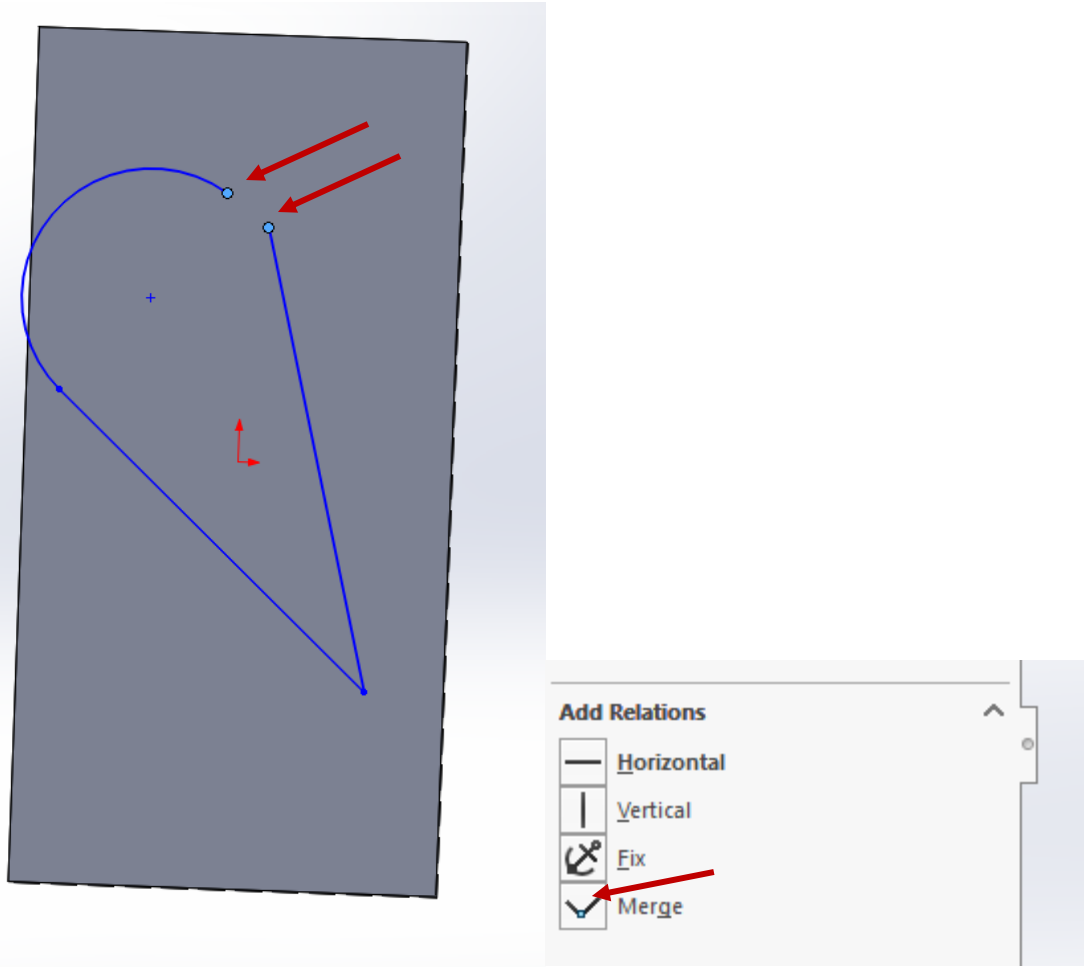
Under *Add Relations*, click Tangent:



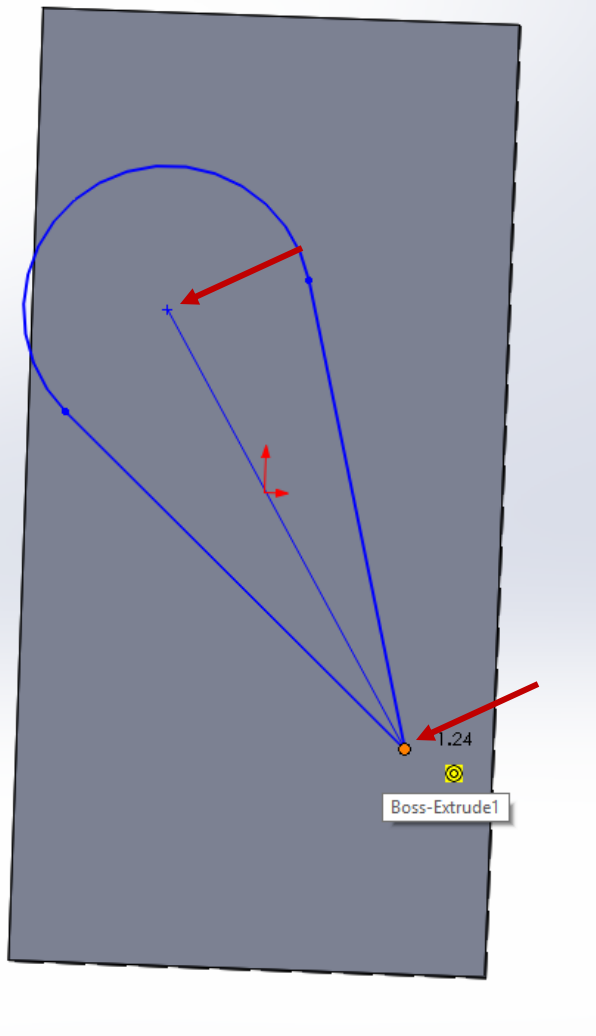
Click the arc, then hold CTRL and click the upper line. Add a Tangent relation to these as well.



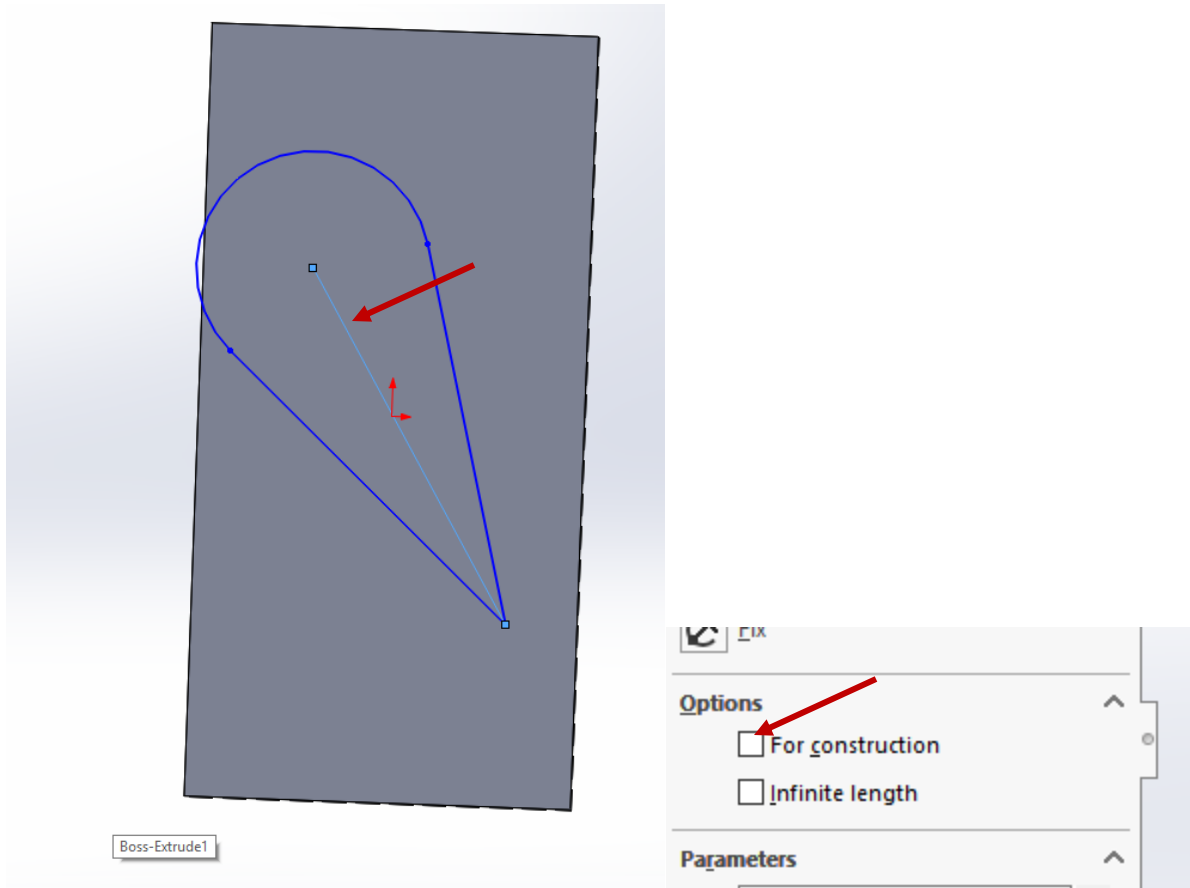
Click the unconnected arc endpoint, and the unconnected line endpoint, and add a Merge relation.



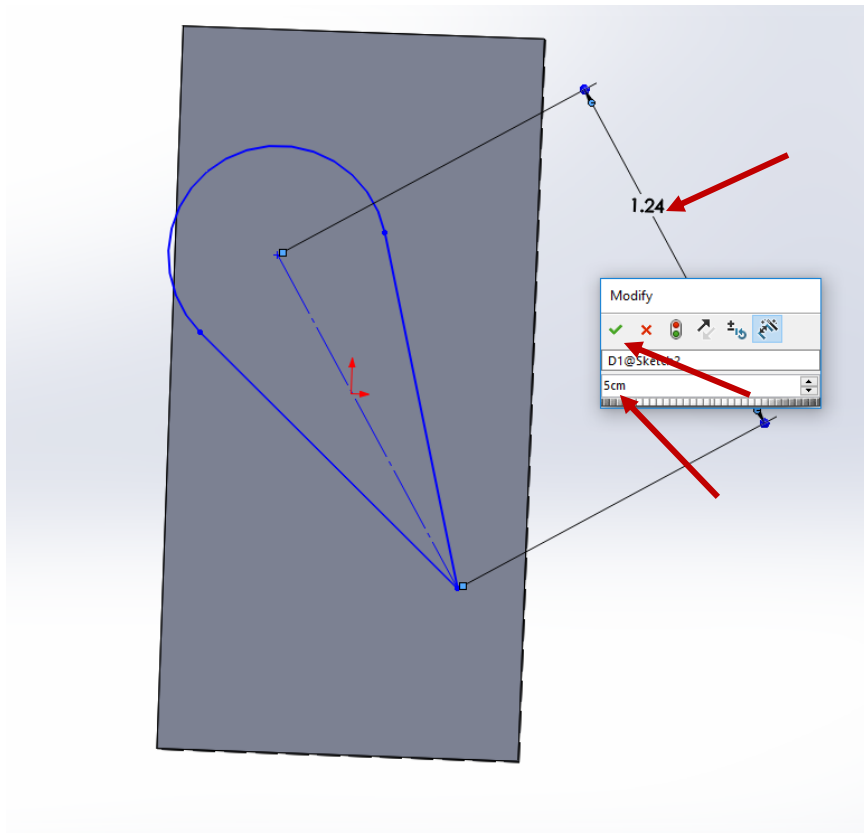
Click the *Line tool*, and draw a line from the arc center to the pointy end:



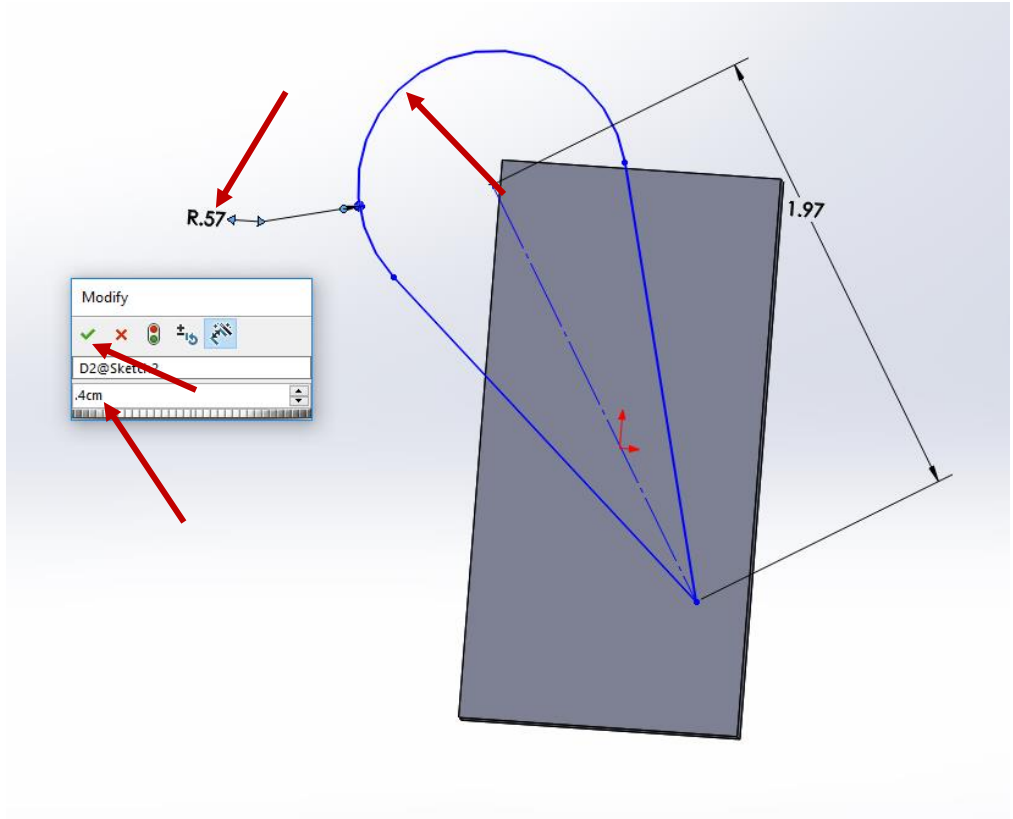
Press ESCAPE to exit the *Line tool*. Click the line to select it, and check the *For Construction* box.



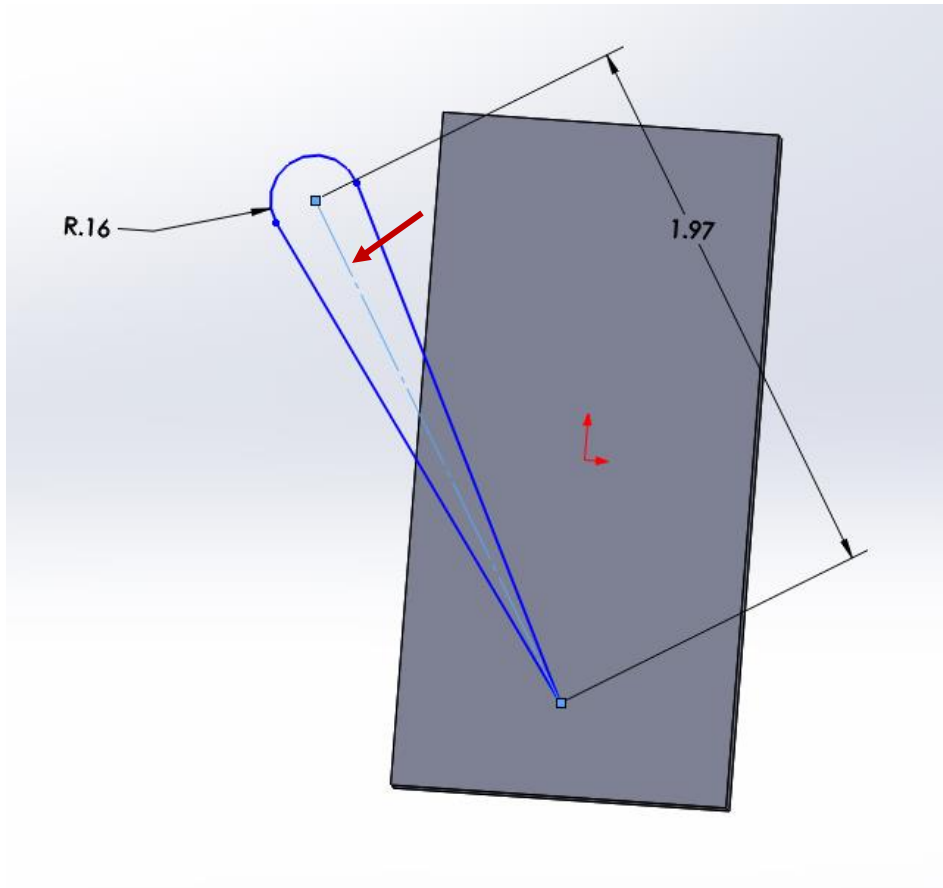
Click Smart Dimension again, and place the dimension **parallel to the line**. Type in 5cm, and click the green check mark.



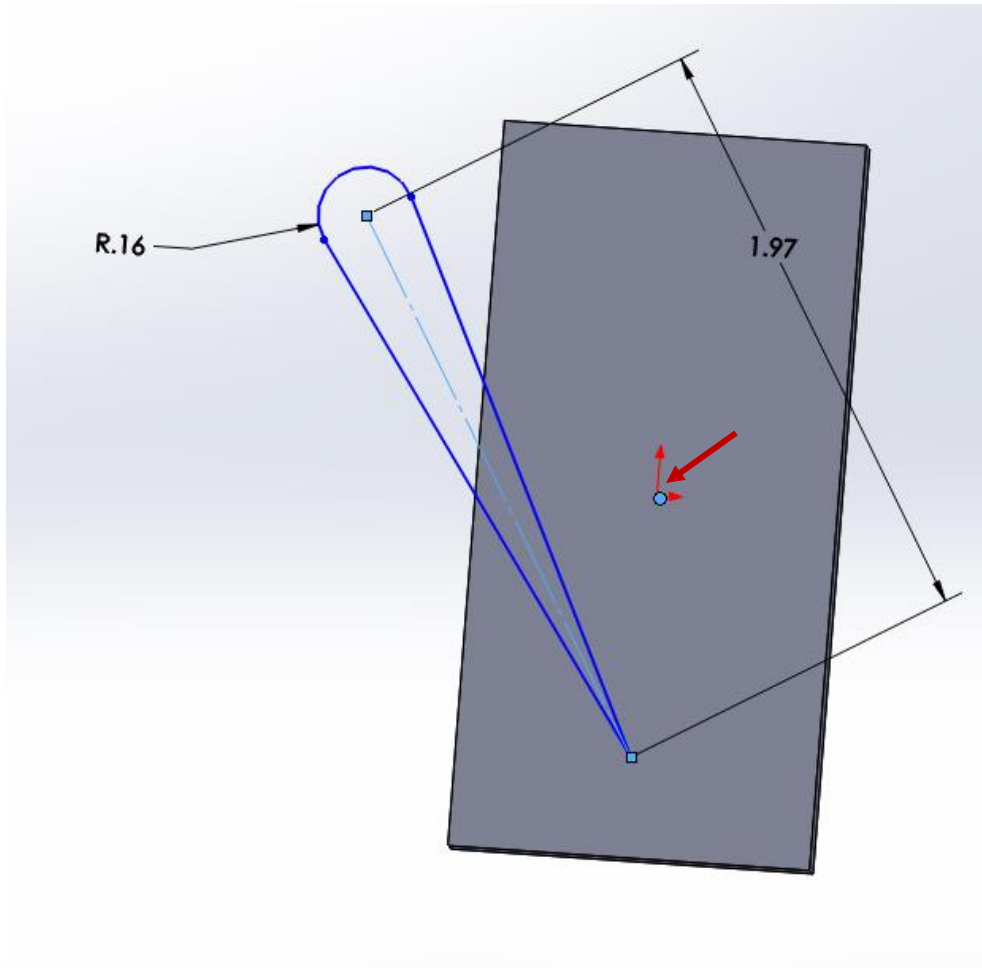
Click the arc, put a dimension next to it, type in .4cm, and click the green check mark:



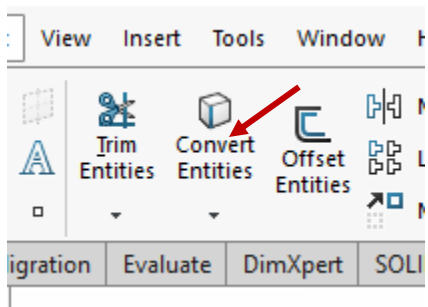
Drag the centerline to the side:



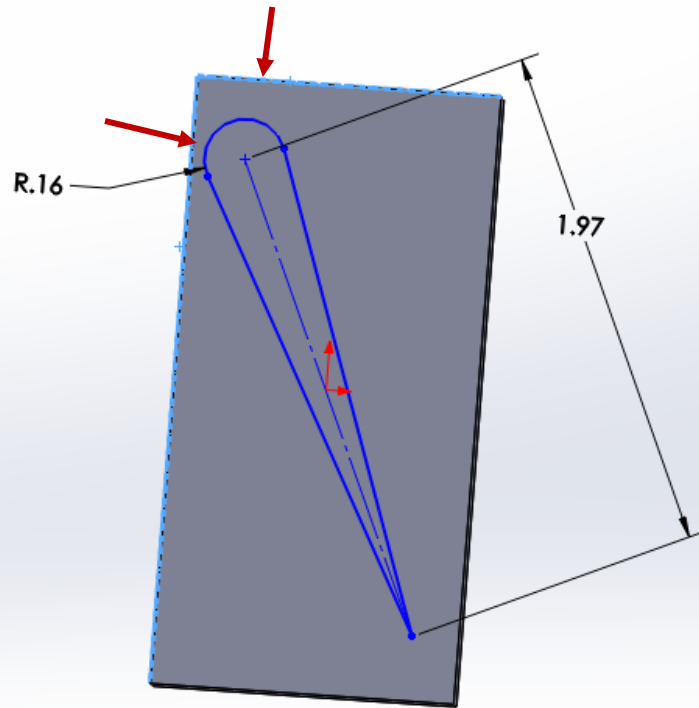
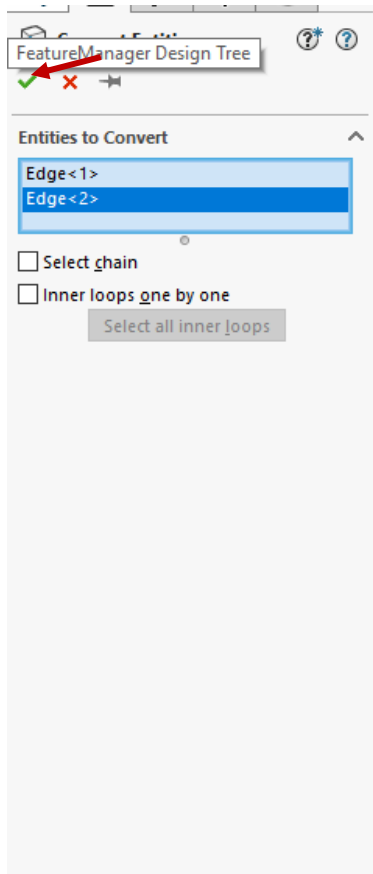
Hold CTRL, and click the center point. Then add a Coincident relation:



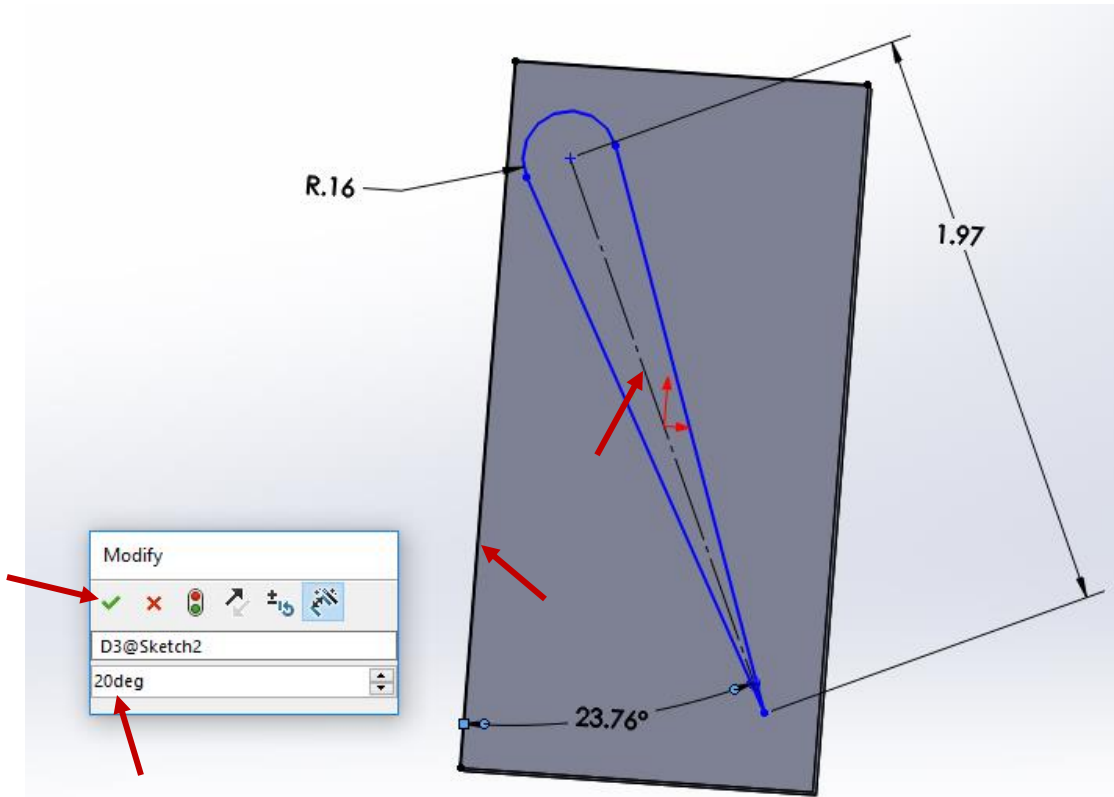
Click the background to deselect everything. Then click Convert Entities:



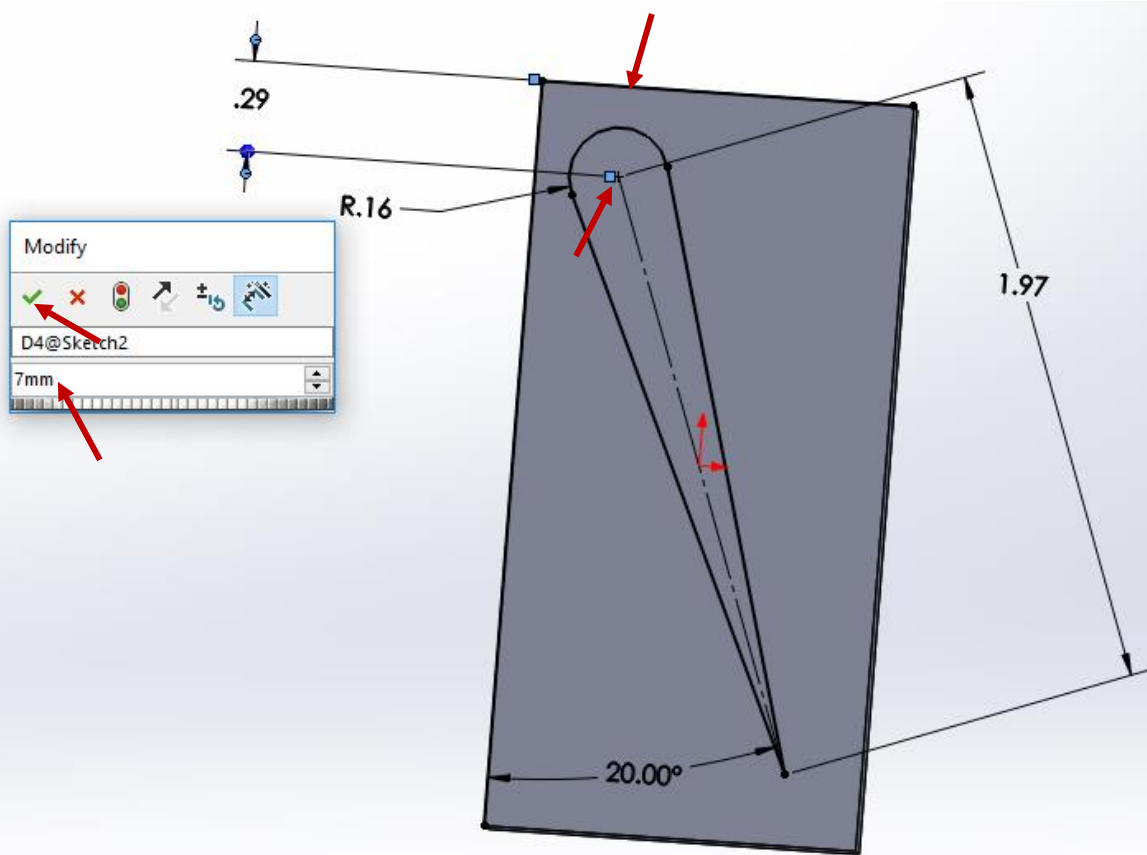
Click the left edge of the rectangle, and then the top edge. Click the green check mark.



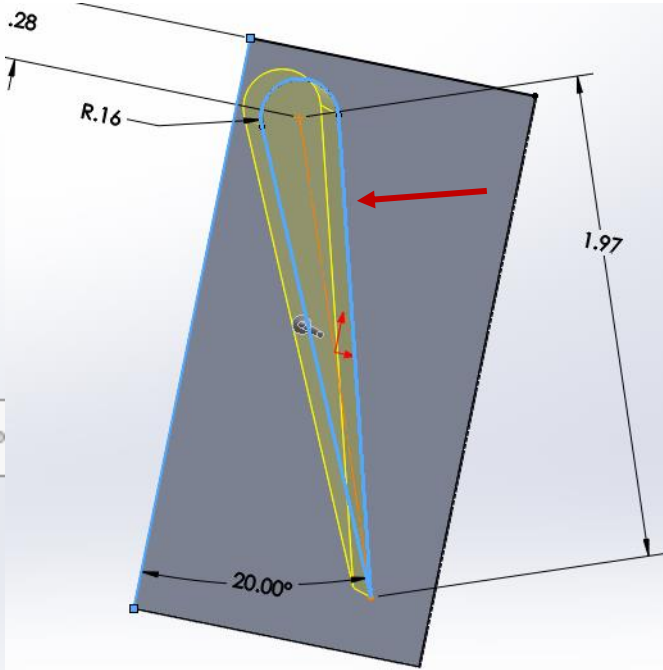
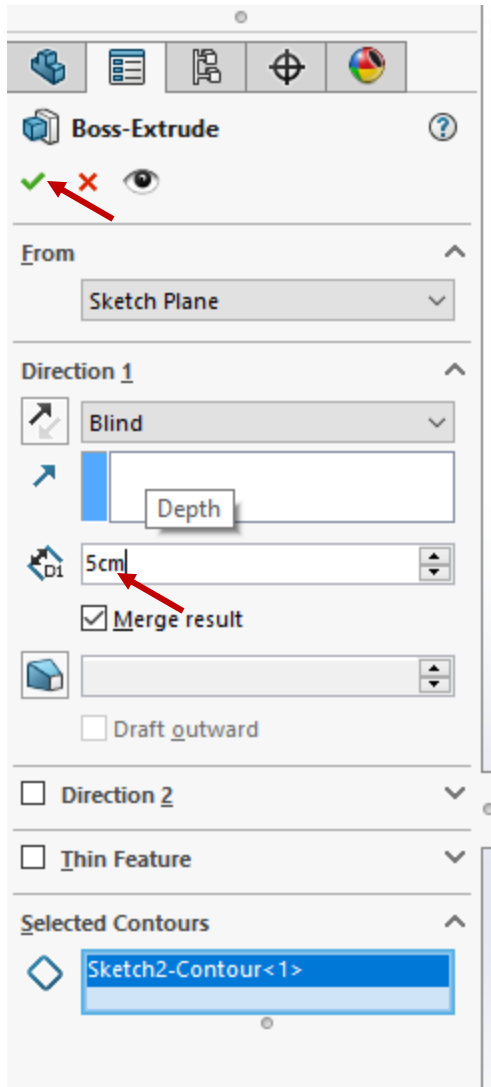
Click Smart Dimension, then click the left edge, then the center line. Type 20deg, and click the green check mark.



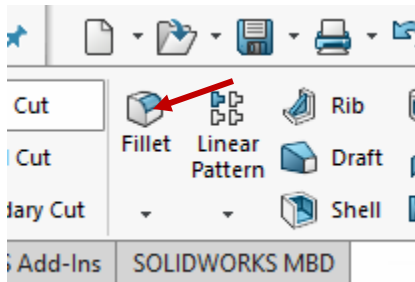
Click the centerpoint, then the top line, type 7mm, and click the green check mark:



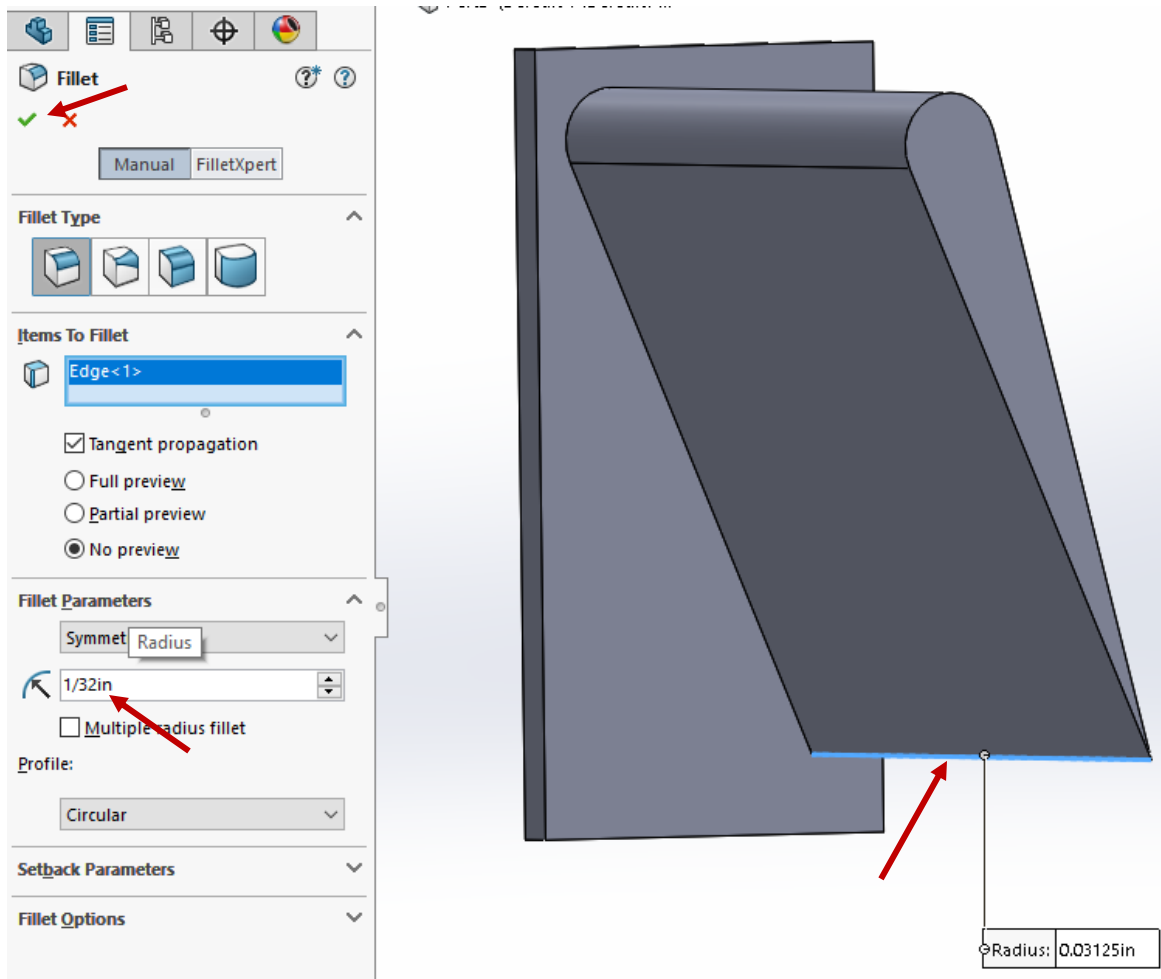
Click Exit Sketch, then type 5cm, click the outline of the winglet you just drew, and click the green check mark.



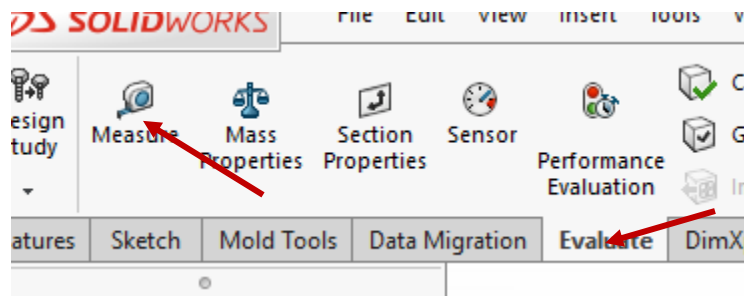
Click Fillet:



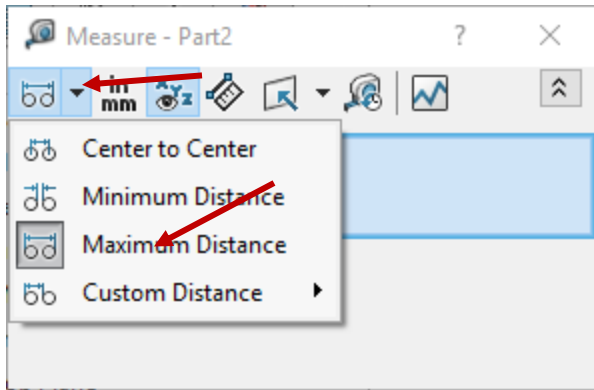
Click the trailing edge of the winglet, then type in 1/16in, and press the green check mark.



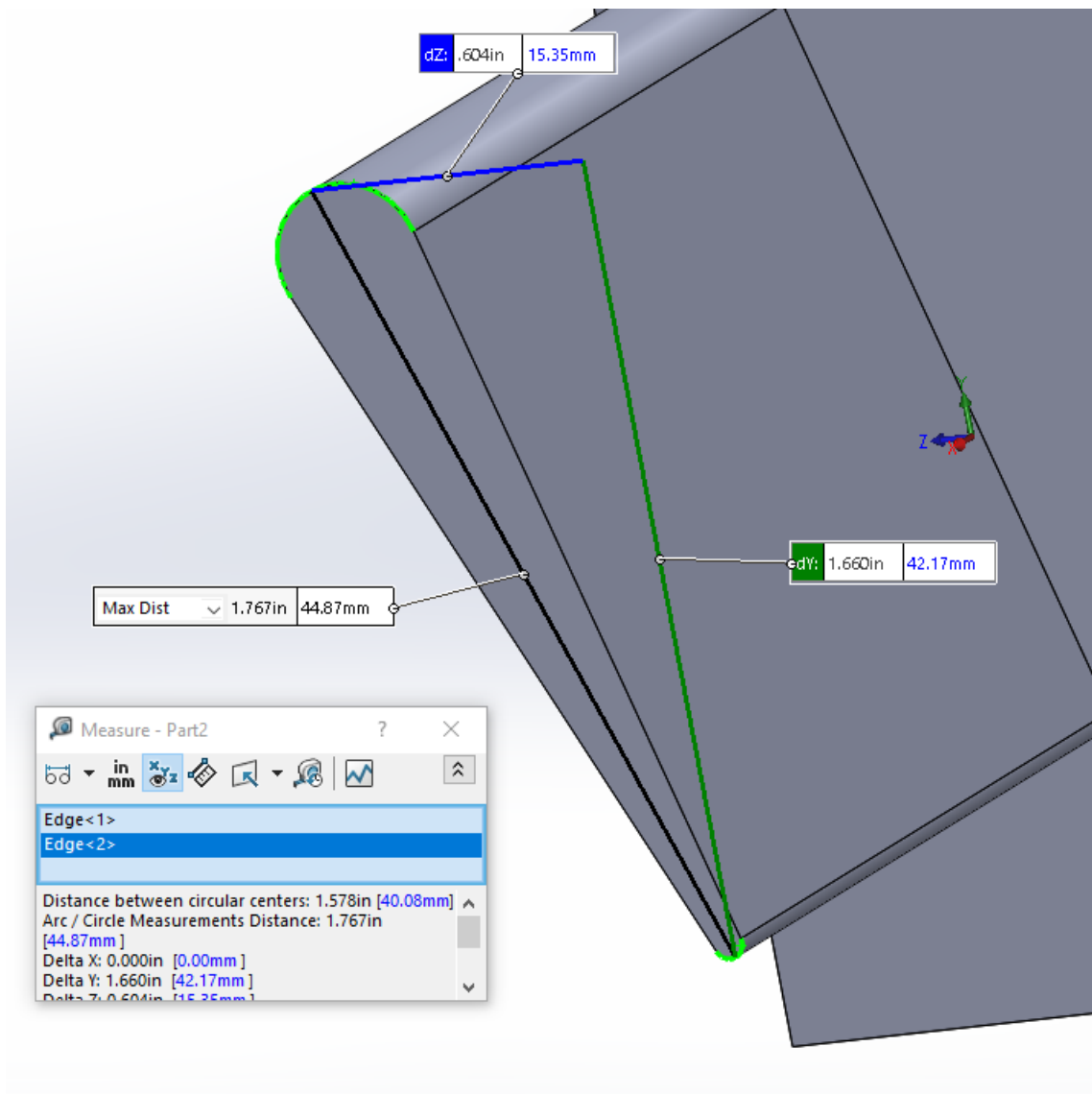
Oh no! The fillet made our wing a lot shorter! Let's see how much. Click Evaluate, then Measure.



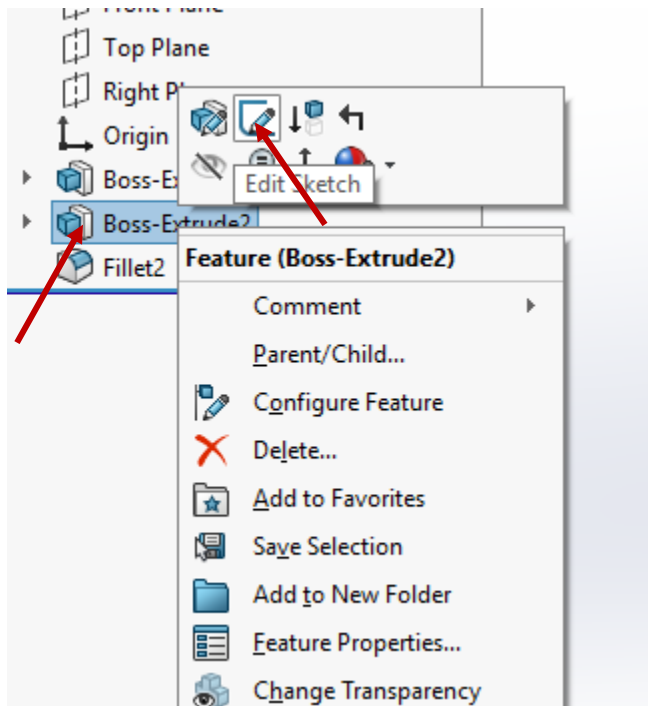
Click the arrow next to the two circles, and click Maximum Distance:



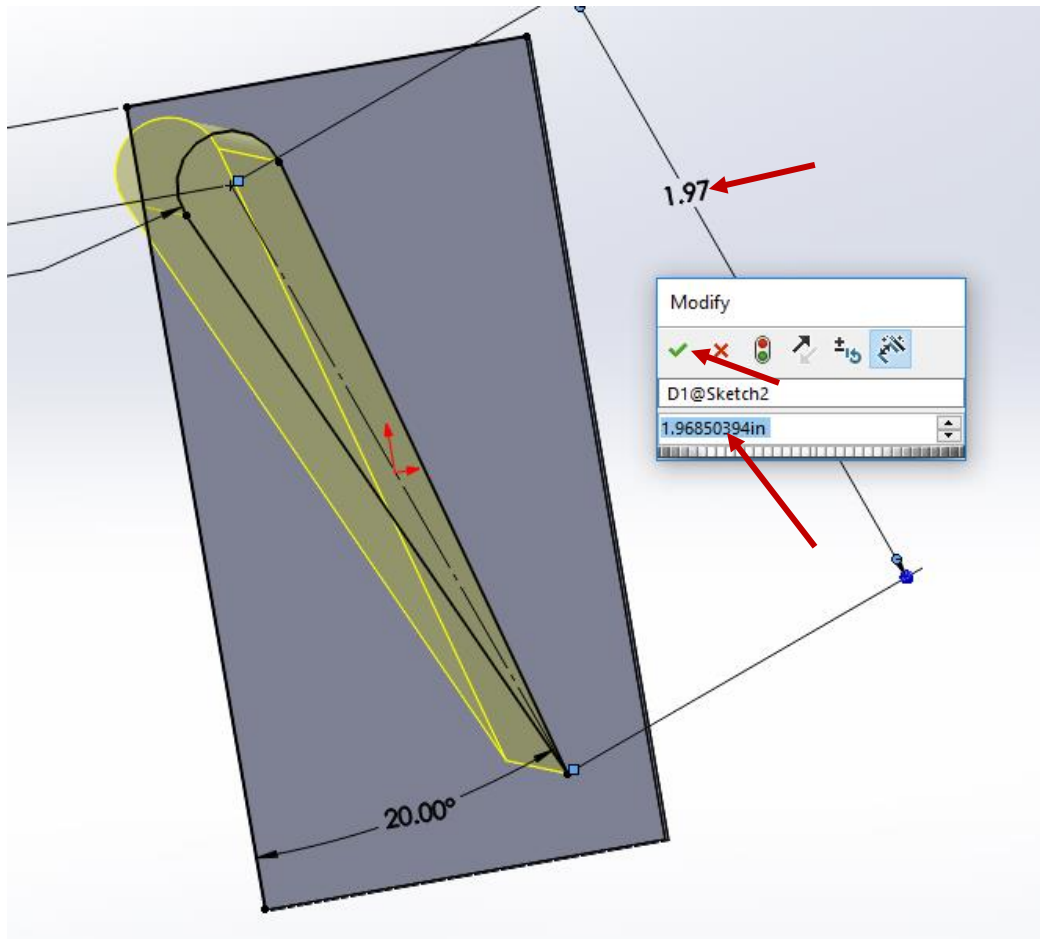
Click the top arc, then the bottom arc. You may need to zoom way in to click them right; don't select the points by accident. If you mess up, click the background to try again.



So the length is only about 4.5cm. That's OK, we'll go increase the original wing length by 0.5cm. Right click Boss-Extrude2, and click the Edit Sketch button:

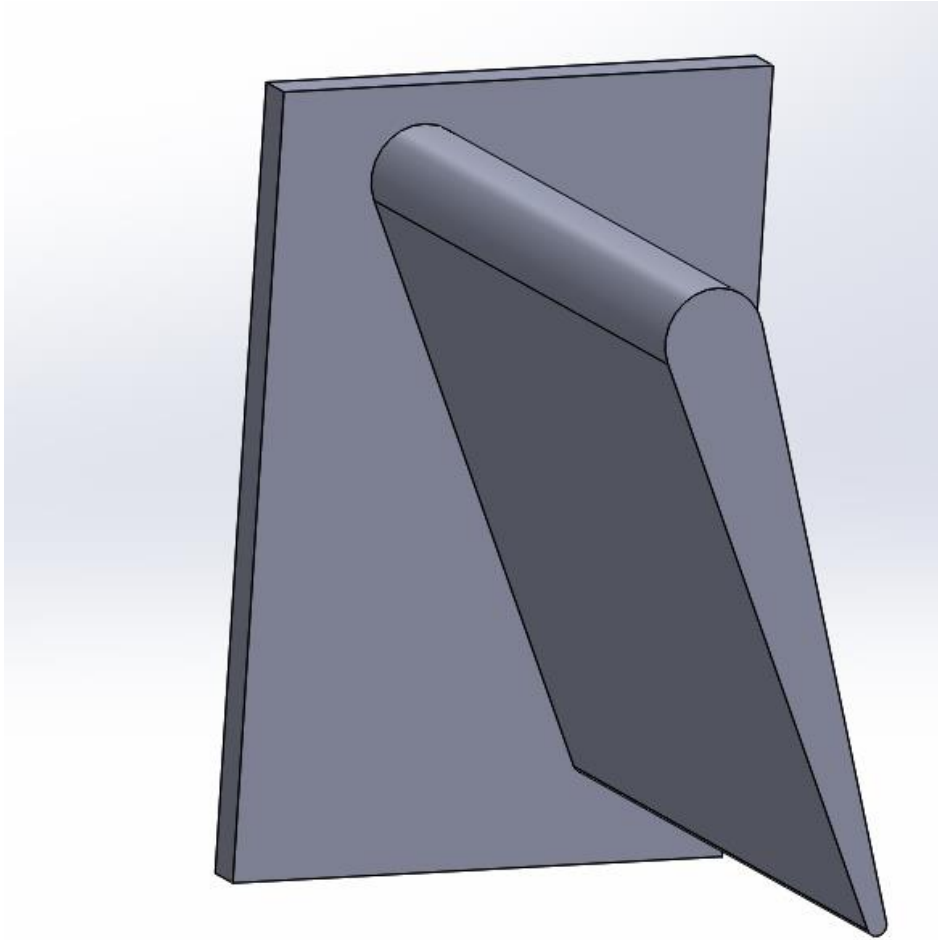


Double click the length dimension, and type in 5.5cm; press the green check mark and then Exit Sketch:

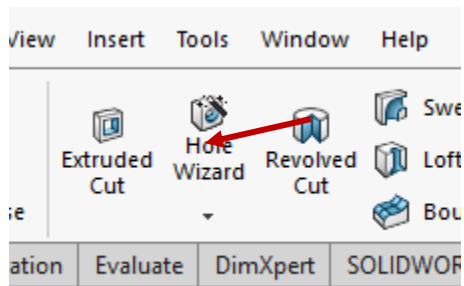


Now that we know how to edit, we should get some more space to add those screws. Right click Boss-Extrude1, and click Edit Sketch again. Double click the width dimension, change it to 4cm, and green check mark, and Exit Sketch. No hints here ;-)

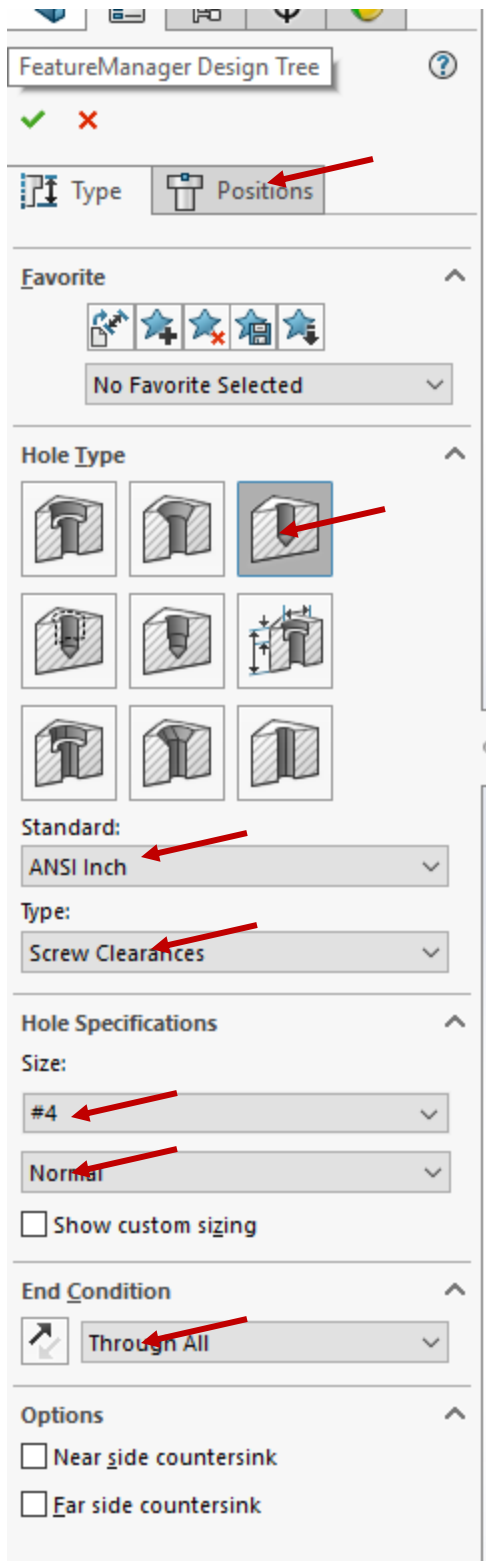
But it should look like this afterwards:



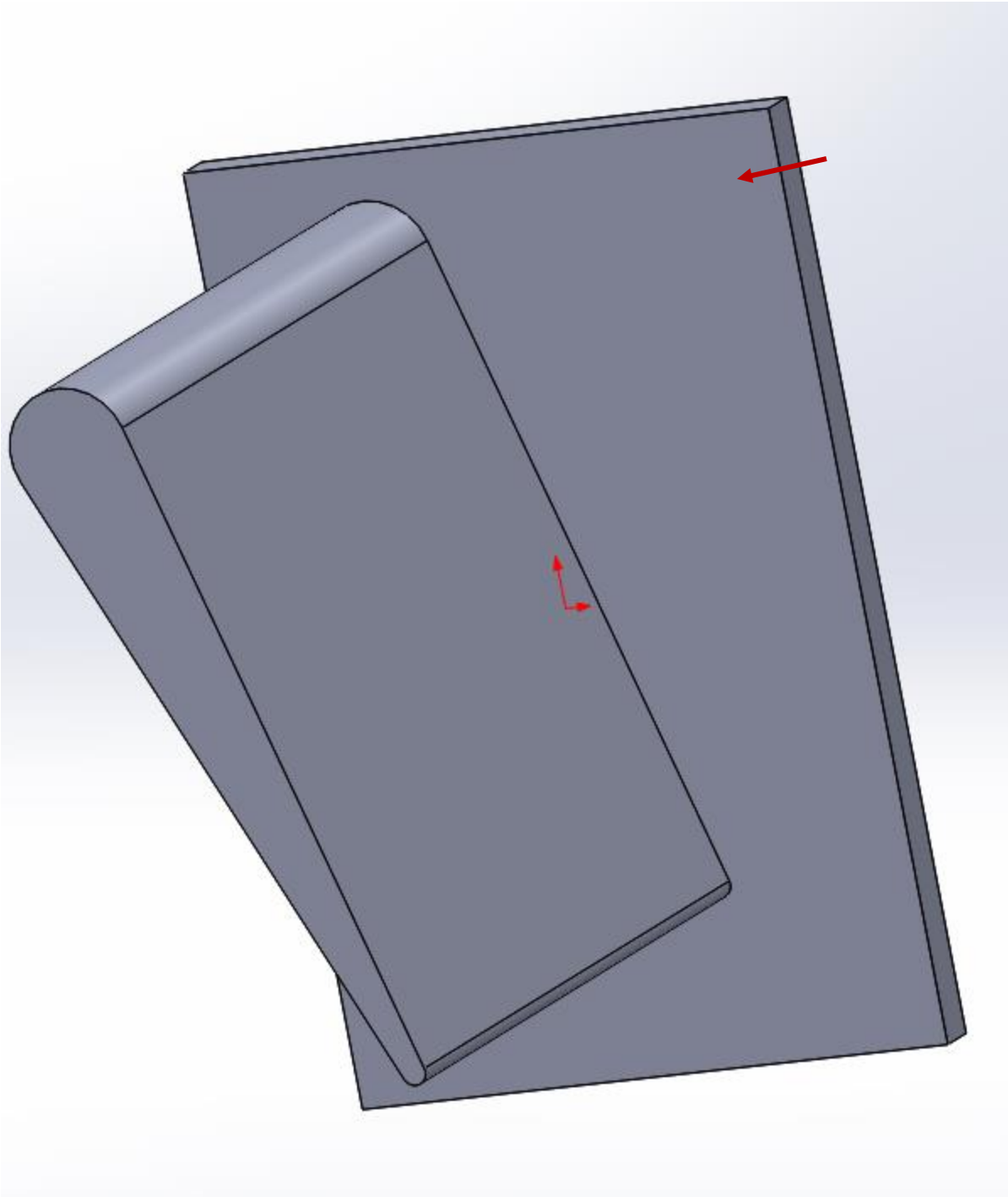
Click Features, then Hole Wizard:



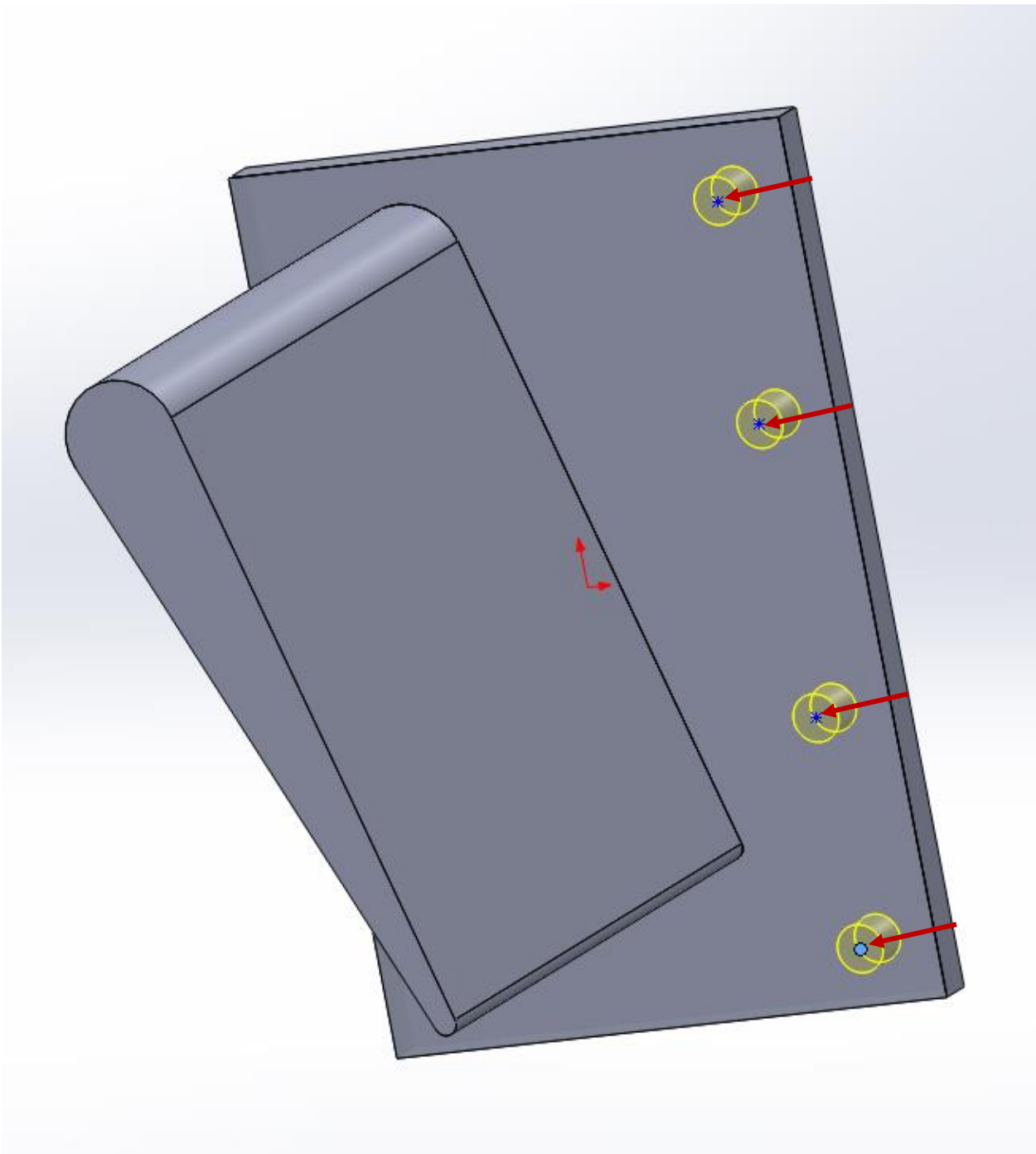
Select a plain *Hole*, using *ANSI Inch* with the *Screw Clearance Type*, size *#4 (Normal)*. Make sure the end condition is *Through All*, and countersinks are deselected. Finally, click Positions.



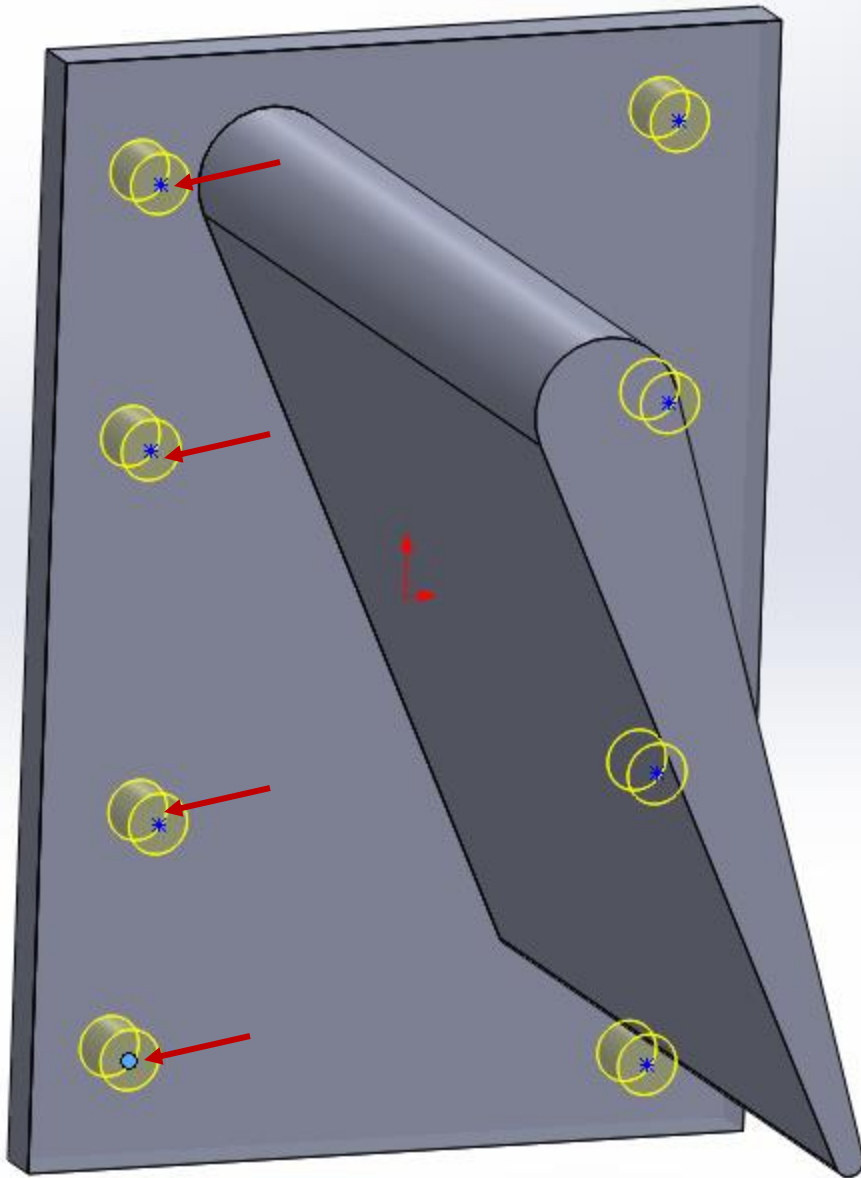
Click the front face:



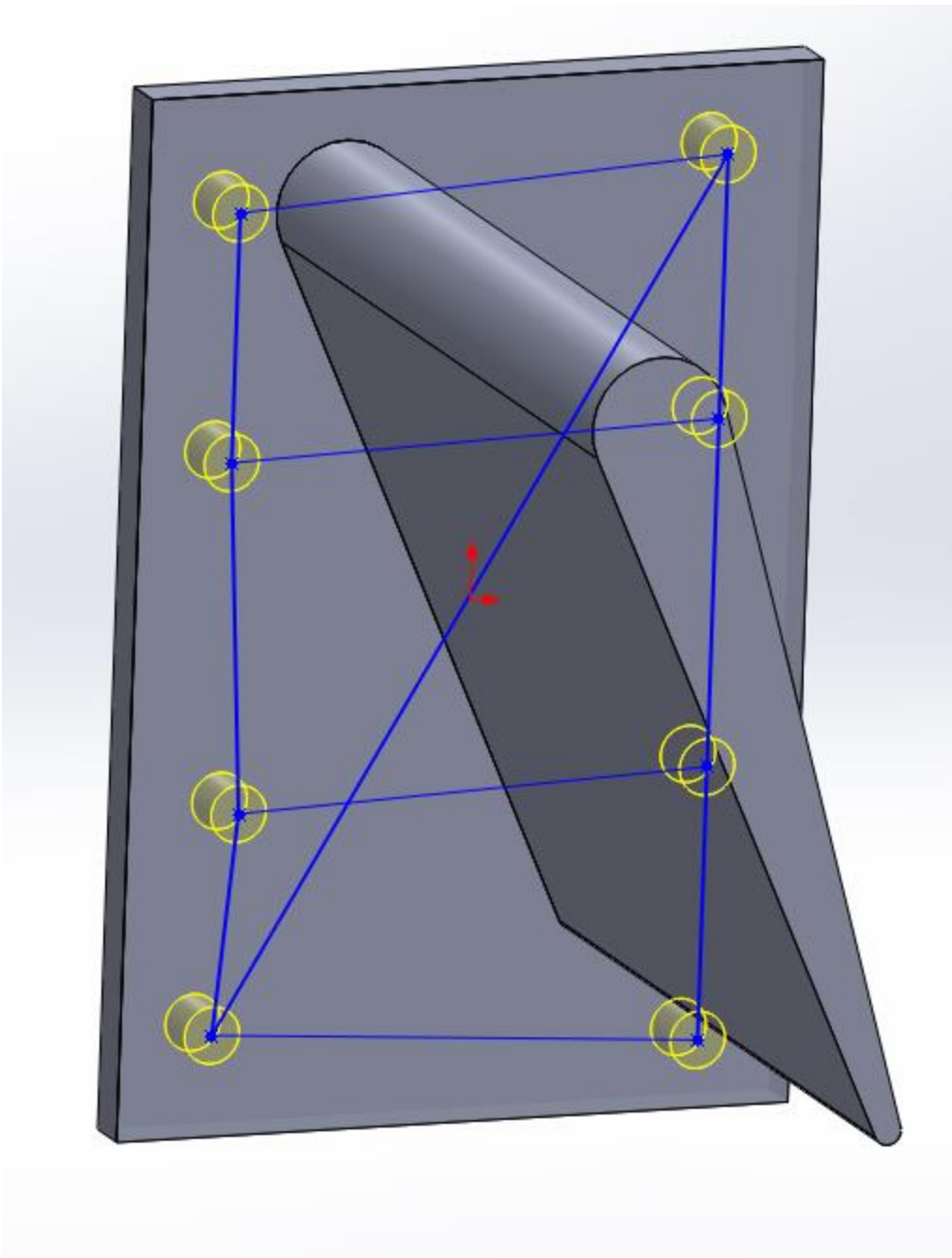
Place 4 holes by clicking:



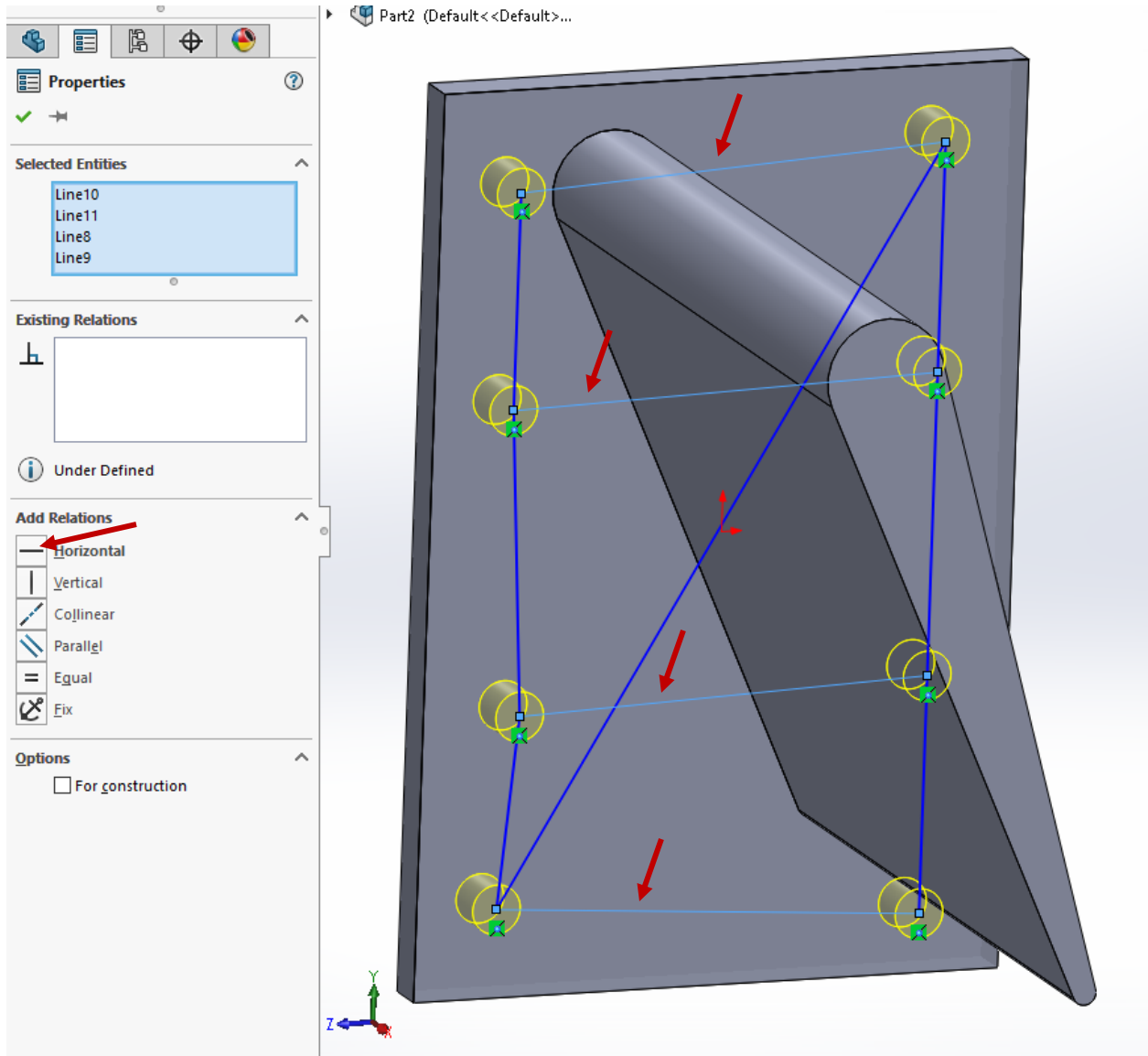
Place another 4 on the other side, after rotating the view:



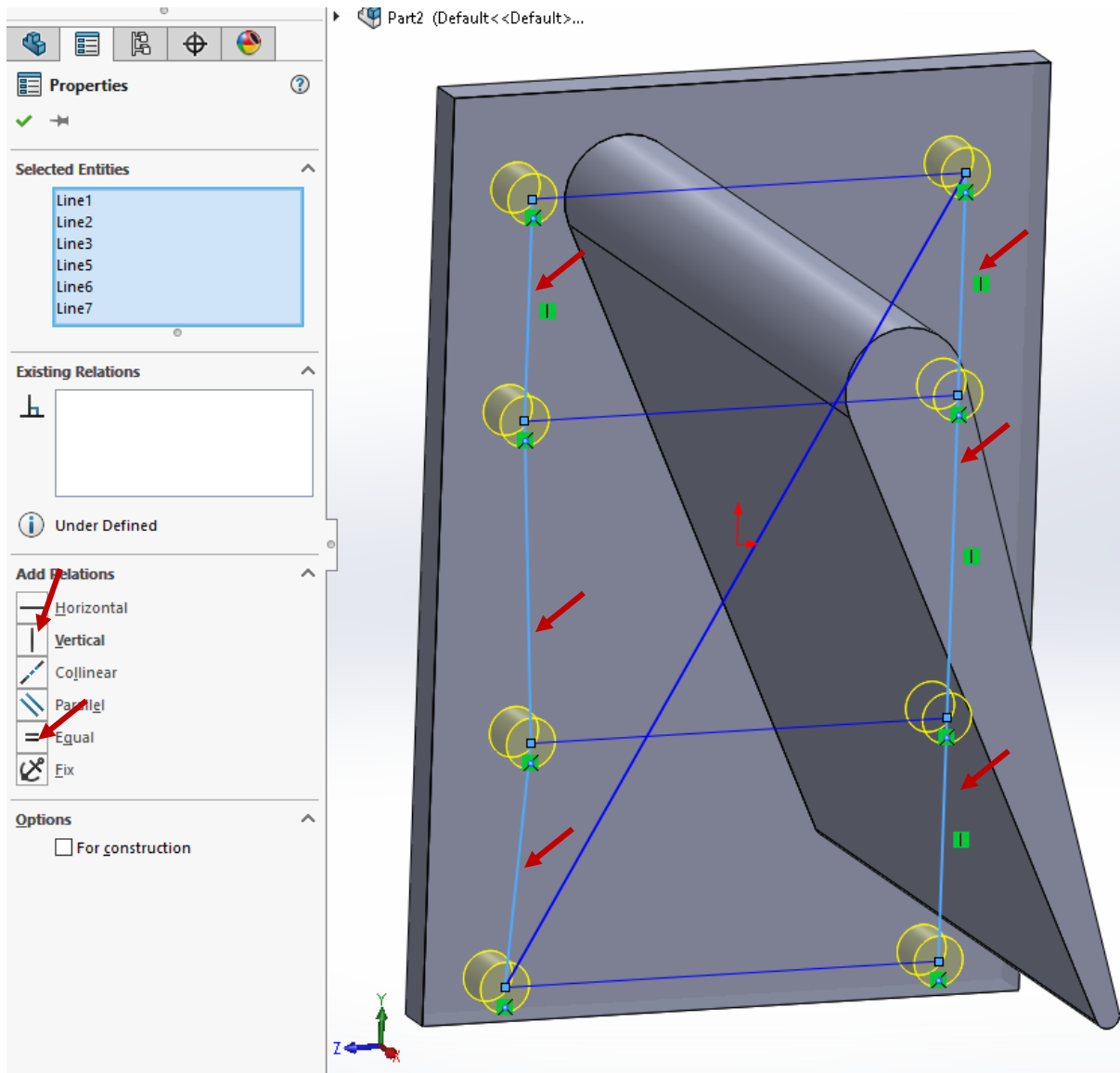
Click the *Line tool*, and connect the dots together like so:



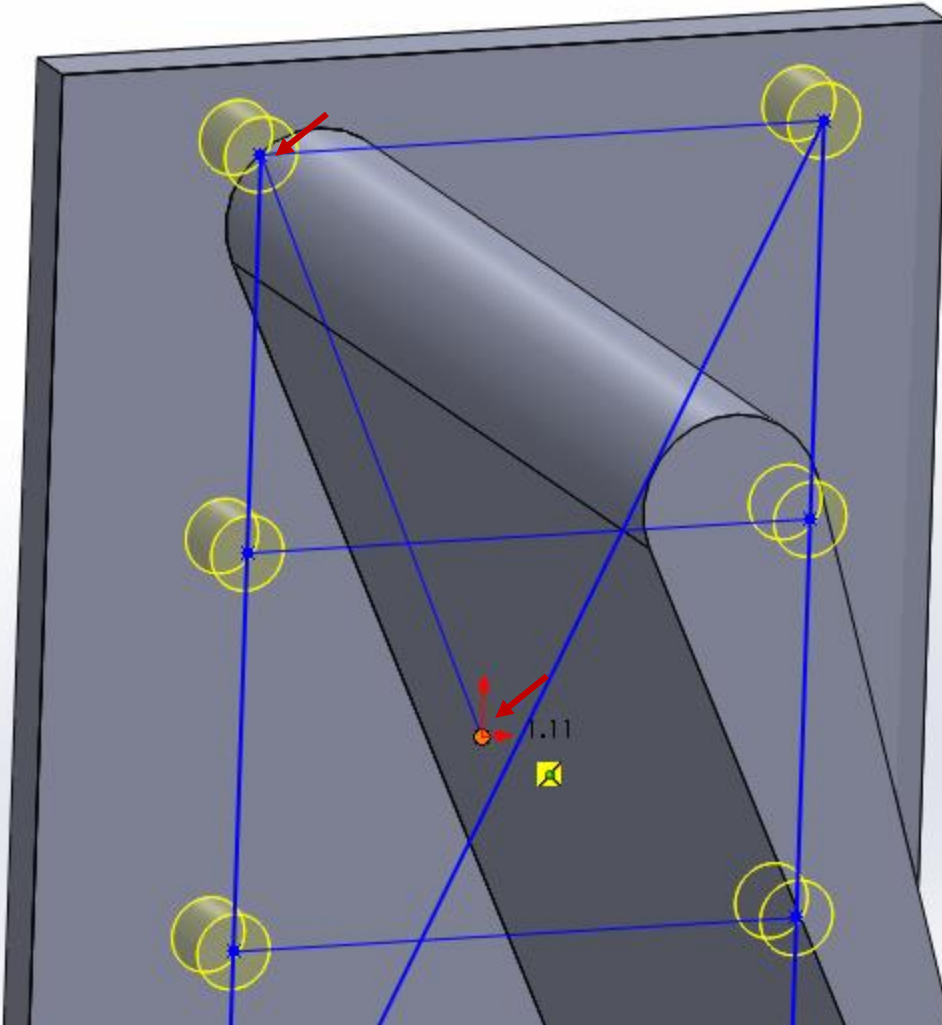
CTRL click to select all the horizontal-ish lines, and then add a Horizontal relation:



CTRL click to select all the vertical-ish lines, and then add a Vertical relation and an Equal relation:

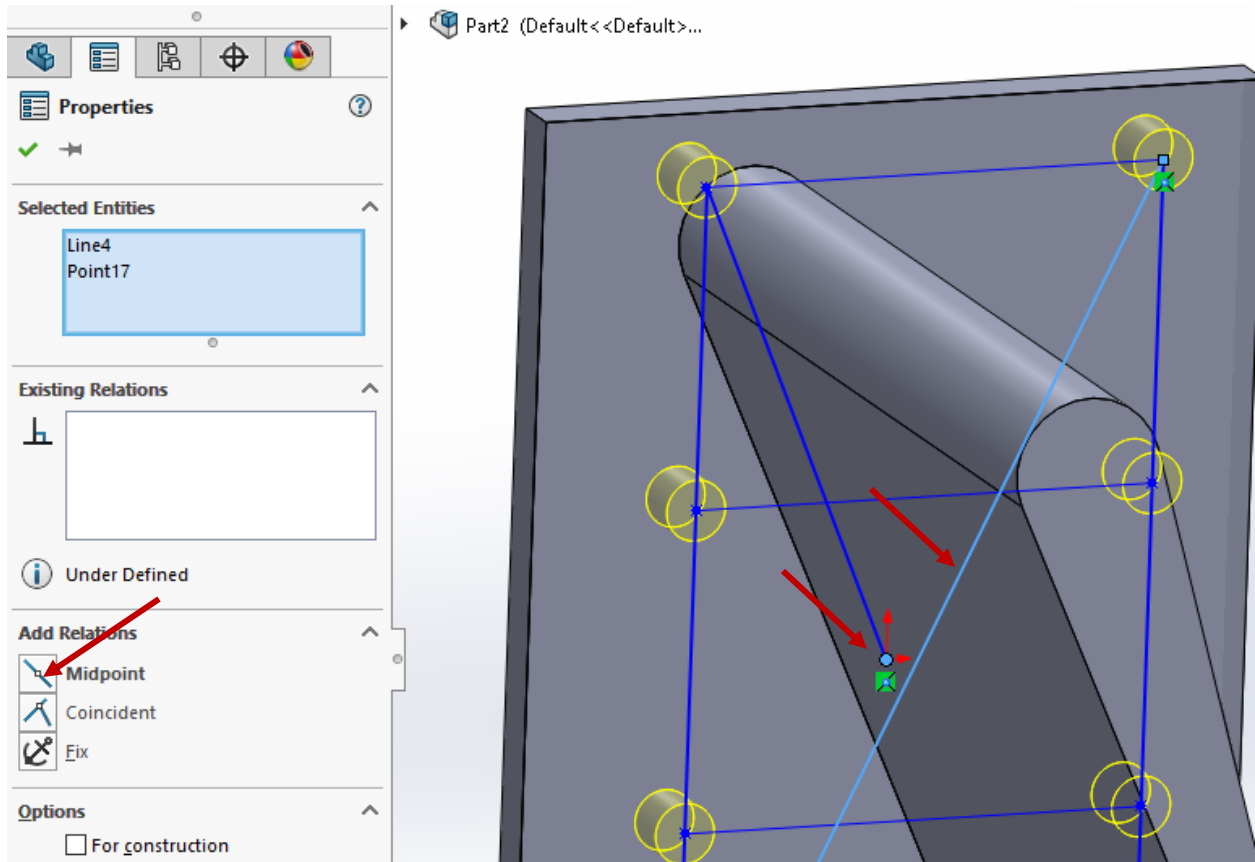


Click the *Line* tool. Draw a line from the top left corner to the center point:

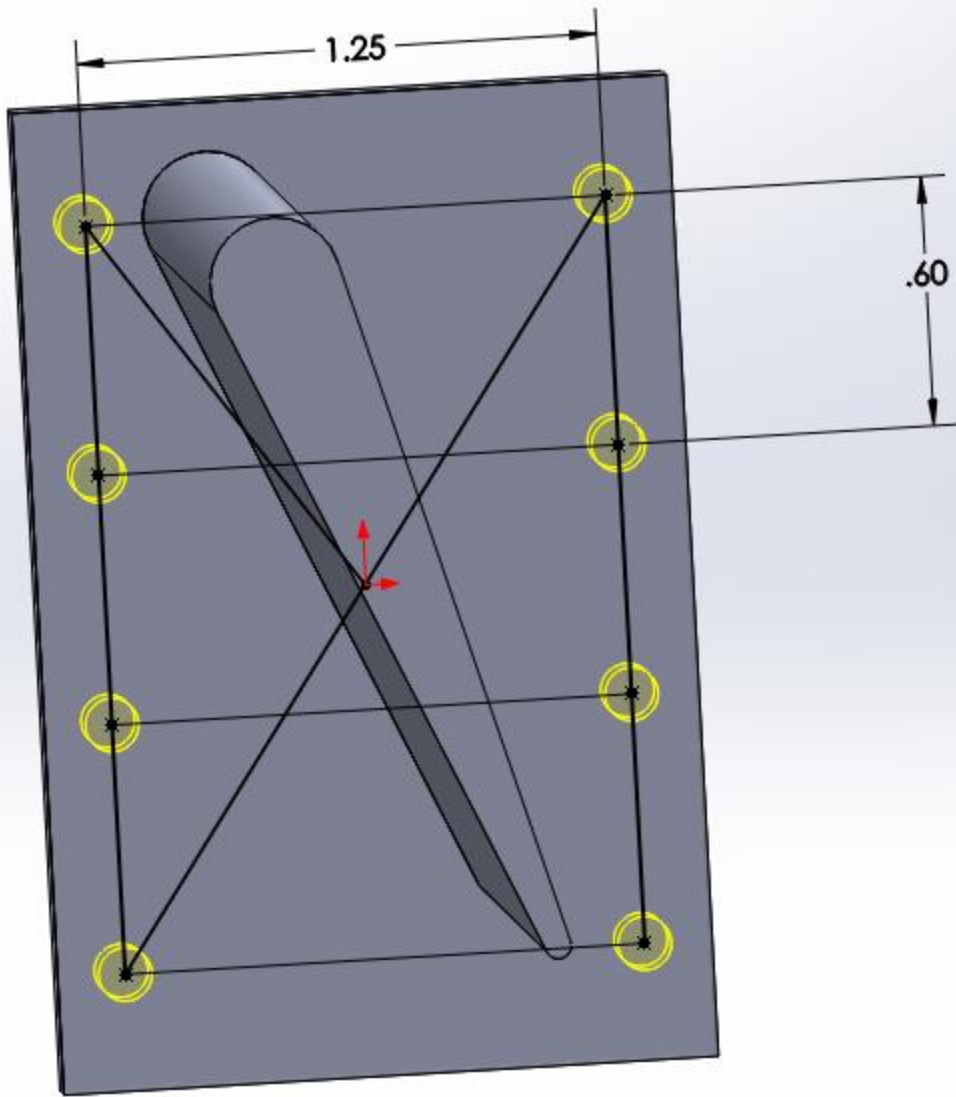


CTRL click the new center point and the diagonal line. Add a Midpoint relation:

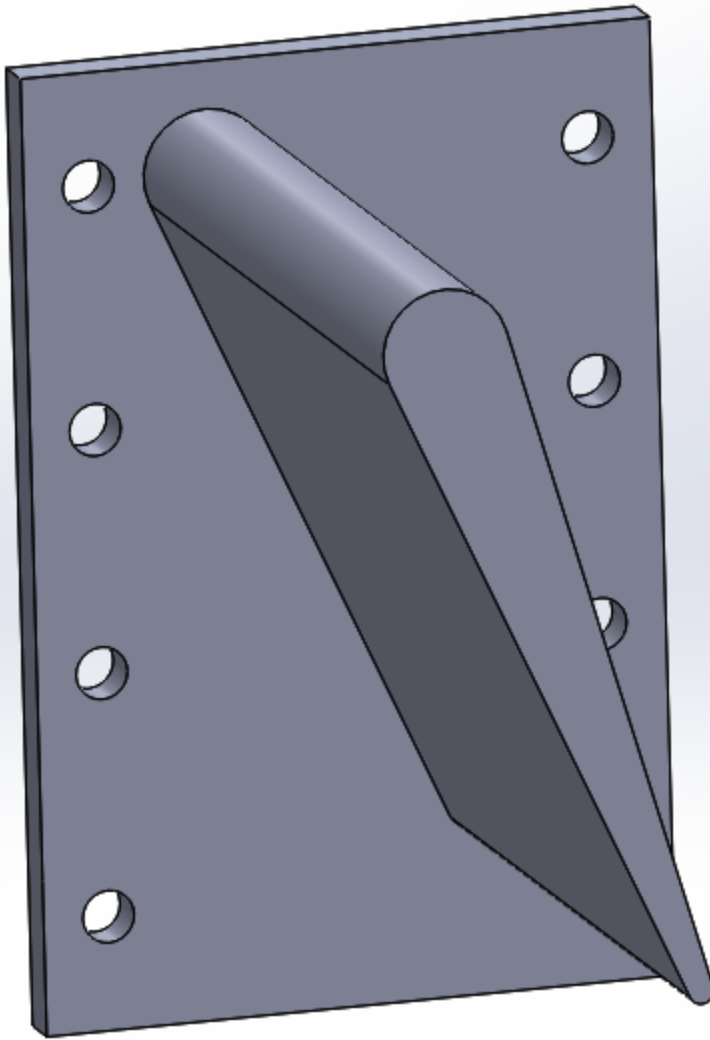
(Yes, it would have made more sense to just put a point on the center point. But in the Hole Wizard mode, SolidWorks thinks every *point* is a hole to be drilled, which is not what we want. So using the endpoint of a line is an easy workaround)



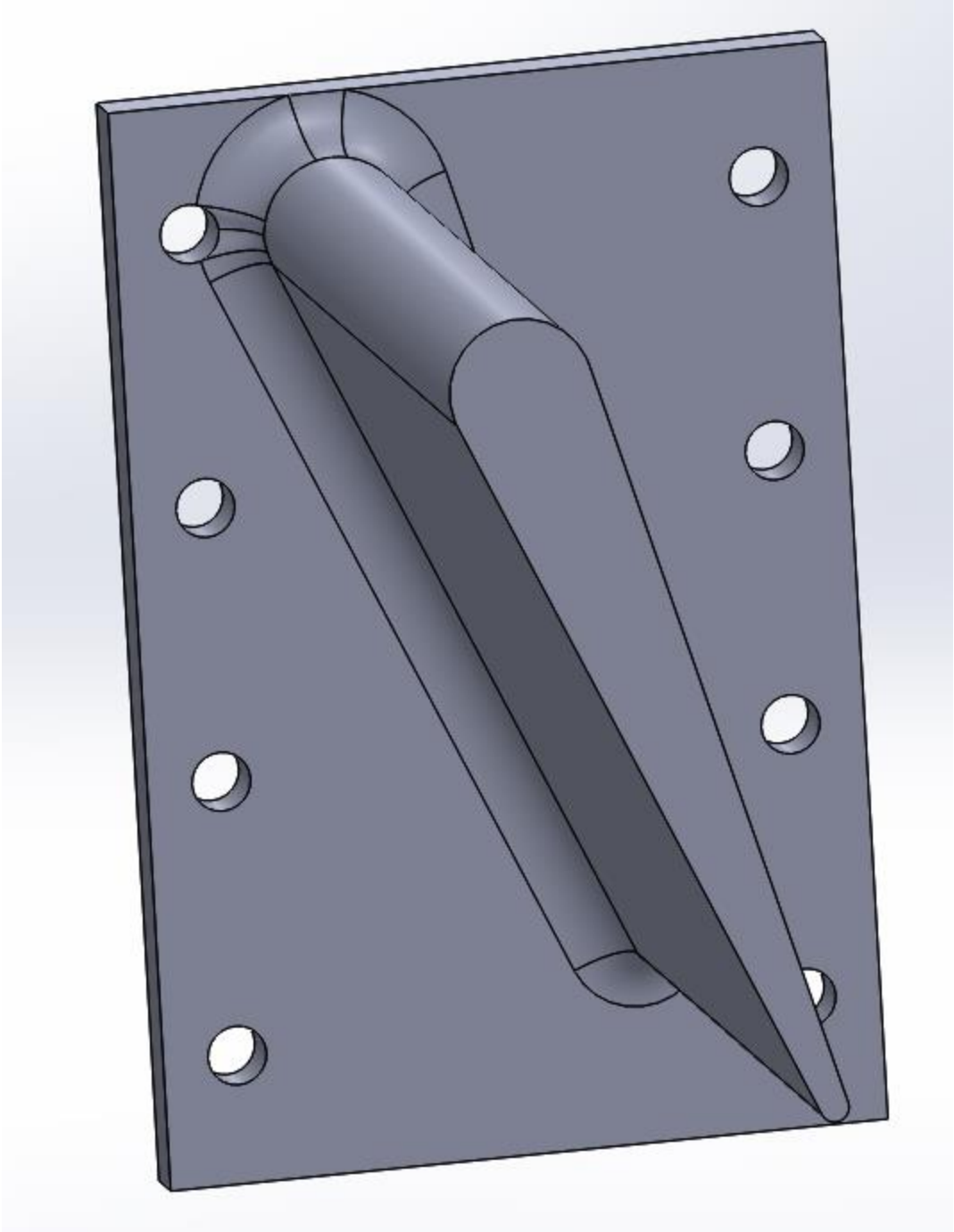
Click Smart Dimension, and add the following dimensions (.6in and 1.25in):



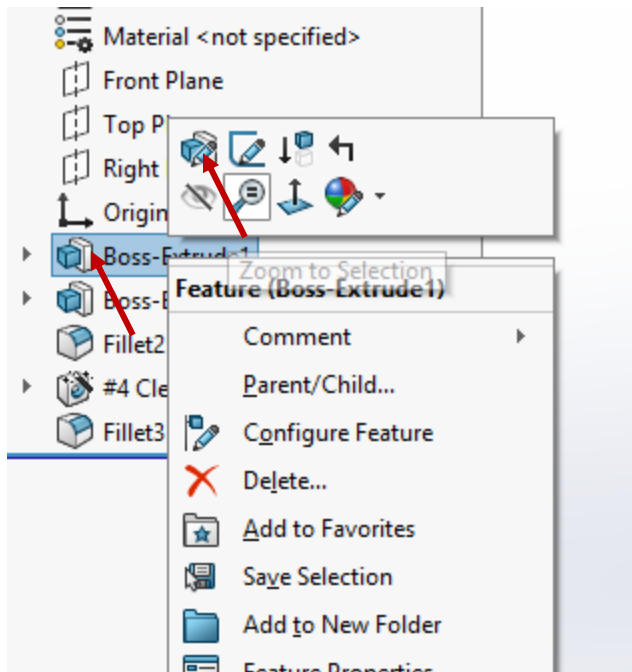
Click the green check mark (press ESCAPE if you don't see it).



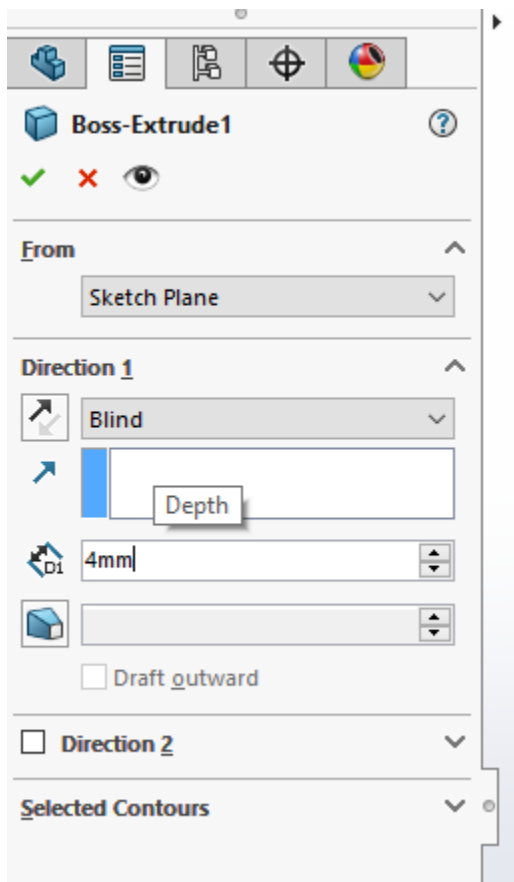
Click Fillet again. Click one of the base edges of the wing, then enter 1/8in as the radius. This fillet prevents stress concentrations at the base of the winglet.



Shoot, that base looks a little thin. Let's make it thinner. Right click Extrude-Boss1, and click Edit Feature.



Change the thickness to 4mm, and click the green check mark:



Congratulations! That's your first part in SolidWorks! Go ahead and play around with the model, or ask me how to do other advanced things:

- Curved base to match nosecone
- Countersunk screw holes
- Wing camber
- Chamfered base for aerodynamics