CAD Tutorial: Fuselage

Below is a tutorial on how to create a fuselage using Solidworks. Without any further ado, let's jump right into it.

Prerequisites

There is only one prerequisite for this tutorial:

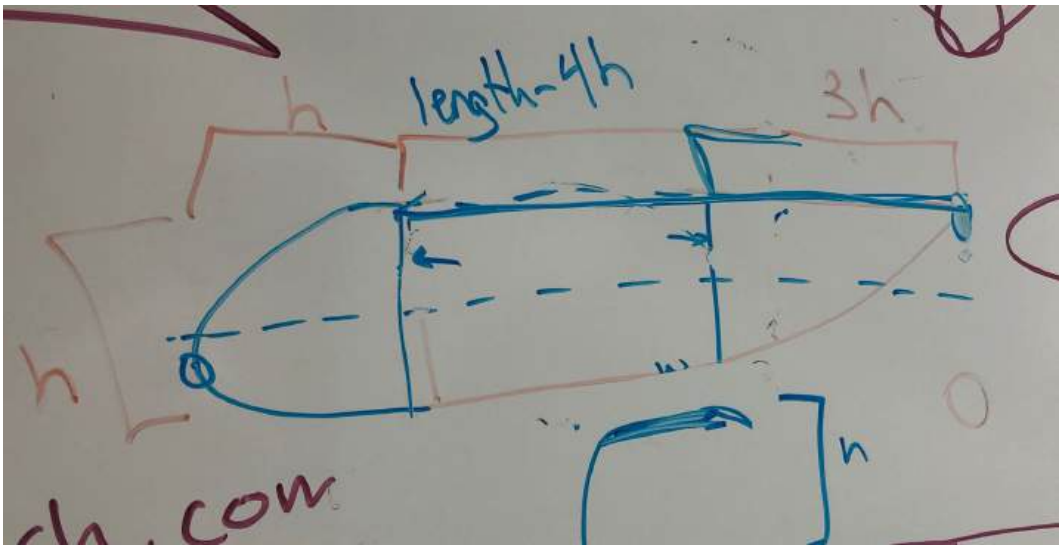
- Solidworks (or VMWare, see VMWare tutorial for specific instructions)

There are no starting files or anything of that sort.

IMPORTANT NOTE: This tutorial in particular will rely heavily on a proficiency of basic Solidworks functionalities such as creating sketches on different planes and faces, constraining different points and lines, as well as dimensioning lengths, radii, and diameters using equations. If any of these are foreign to you, give the last series of tutorials a good read. Anything new will be explained thoroughly here. In fact, we are going to learn about lofting and 3D sketches today. Fun stuff!

Fuselage: preliminary ideas and planning

We are going to create something that looks roughly like this:



Okay, well, maybe not like *that*, but it gives the rough idea.

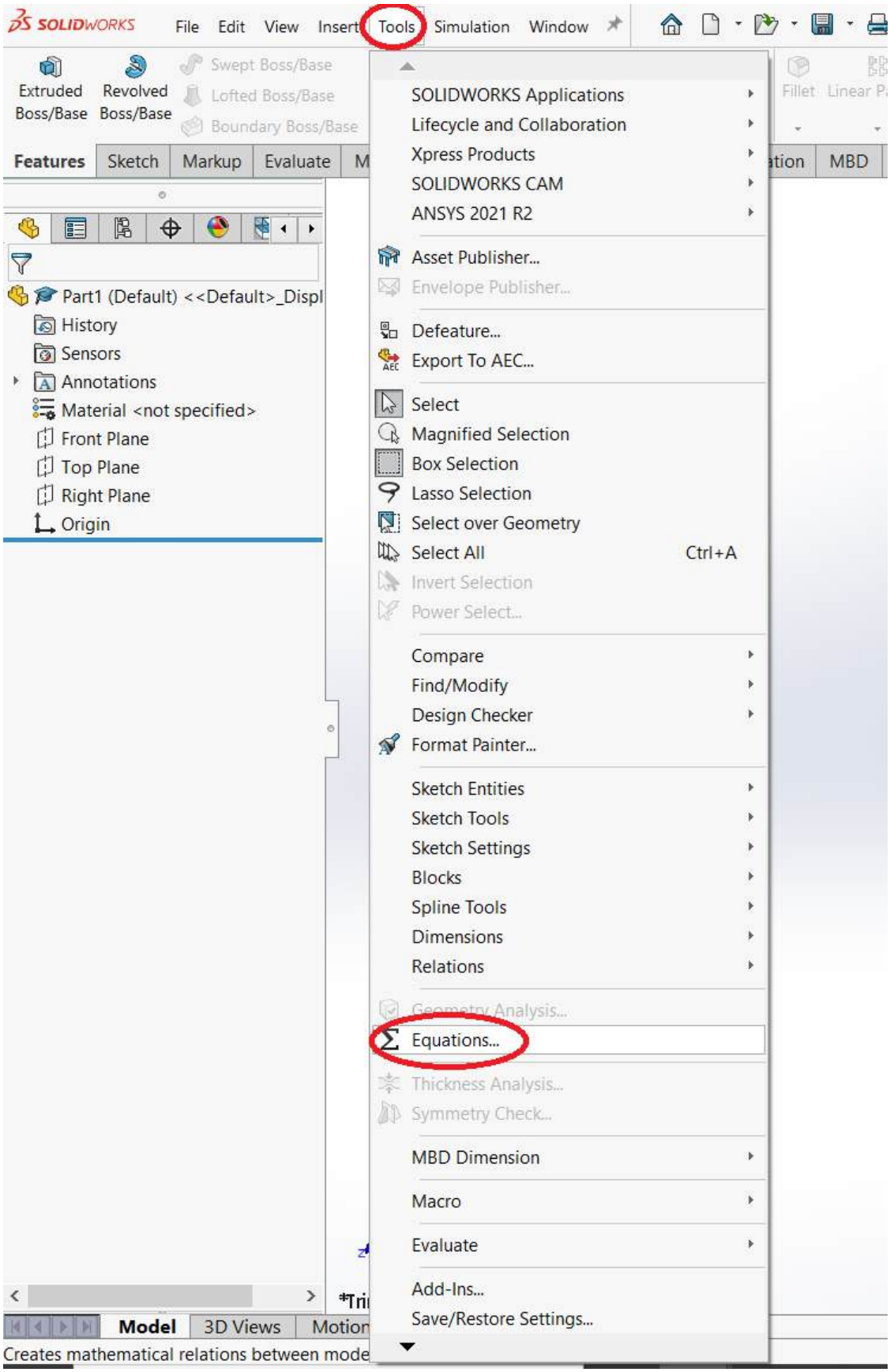
With enough creativity and CAD wit, you might observe that this general fuselage model has three different cross-sections: a circle right at the nose, a square-ish in the middle, and another circle in the tail. Note two things: the circular nose can be conveniently aligned with the origin if we want, which will makes things easier; also, the square section in the middle isn't really square, it has its corners filleted a bit. We will see that a bit later.

Finally, note the different lengths/ We define the height, length, and width of the fuselage, and then use the distances marked in the drawing above to dimension the sketches we are about to produce. Let's get started then!

Defining the variables and making the nose

Before we start CADing anything, let's first create the variables we need to create.

You should remember this from the equations tutorial, but let's recap it really quickly. On top, go to **Tools** → **Equations** as shown:


SOLIDWORKS
File
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Tools
Simulation
Window
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Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
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History
Sensors
Annotations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
SOLIDWORKS Applications
Lifecycle and Collaboration
Xpress Products
SOLIDWORKS CAM
ANSYS 2021 R2
Asset Publisher...
Envelope Publisher...
Defeature...
Export To AEC...
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Symmetry Check...
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Motion
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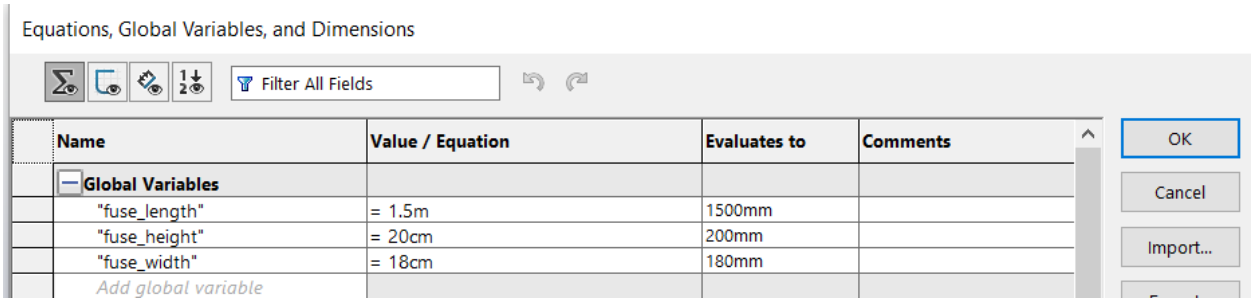
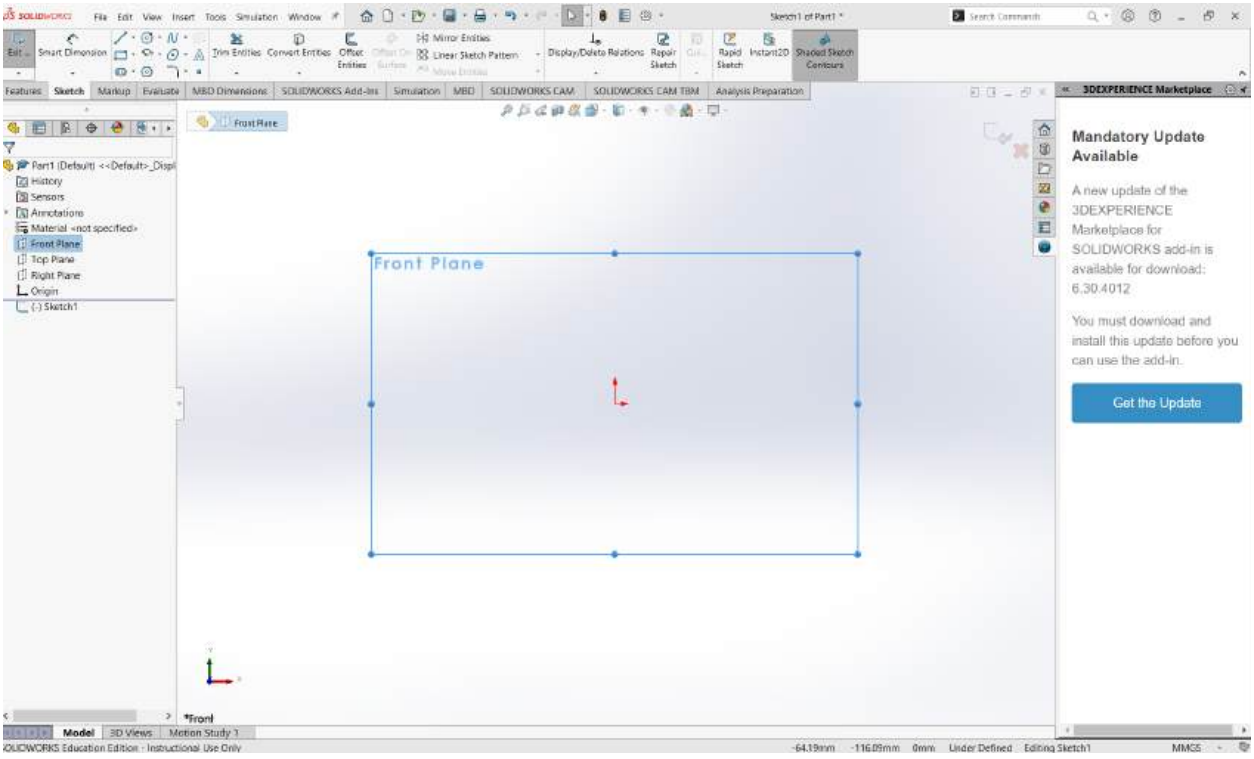
The equations window should open. Create the following global variables:

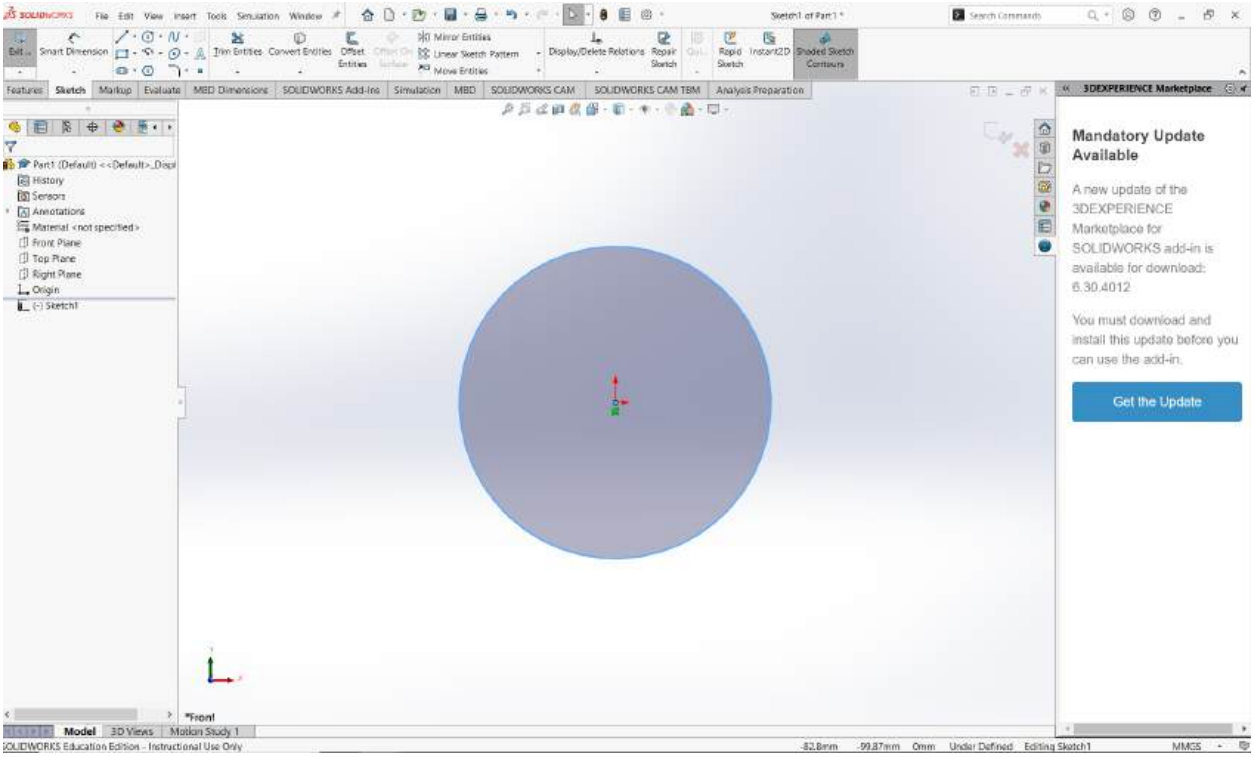


These three variables will suffice to define all the lengths we will need. As expected, ‘fuse_length’ is the length of the fuselage, ‘fuse_height’ is the height of the fuselage, and ‘fuse_width’ is the width of the fuselage.

Now, let’s get started with the nose. Create a sketch on the front plane by clicking the front plane and under Sketch clicking “Sketch.” You should see this:

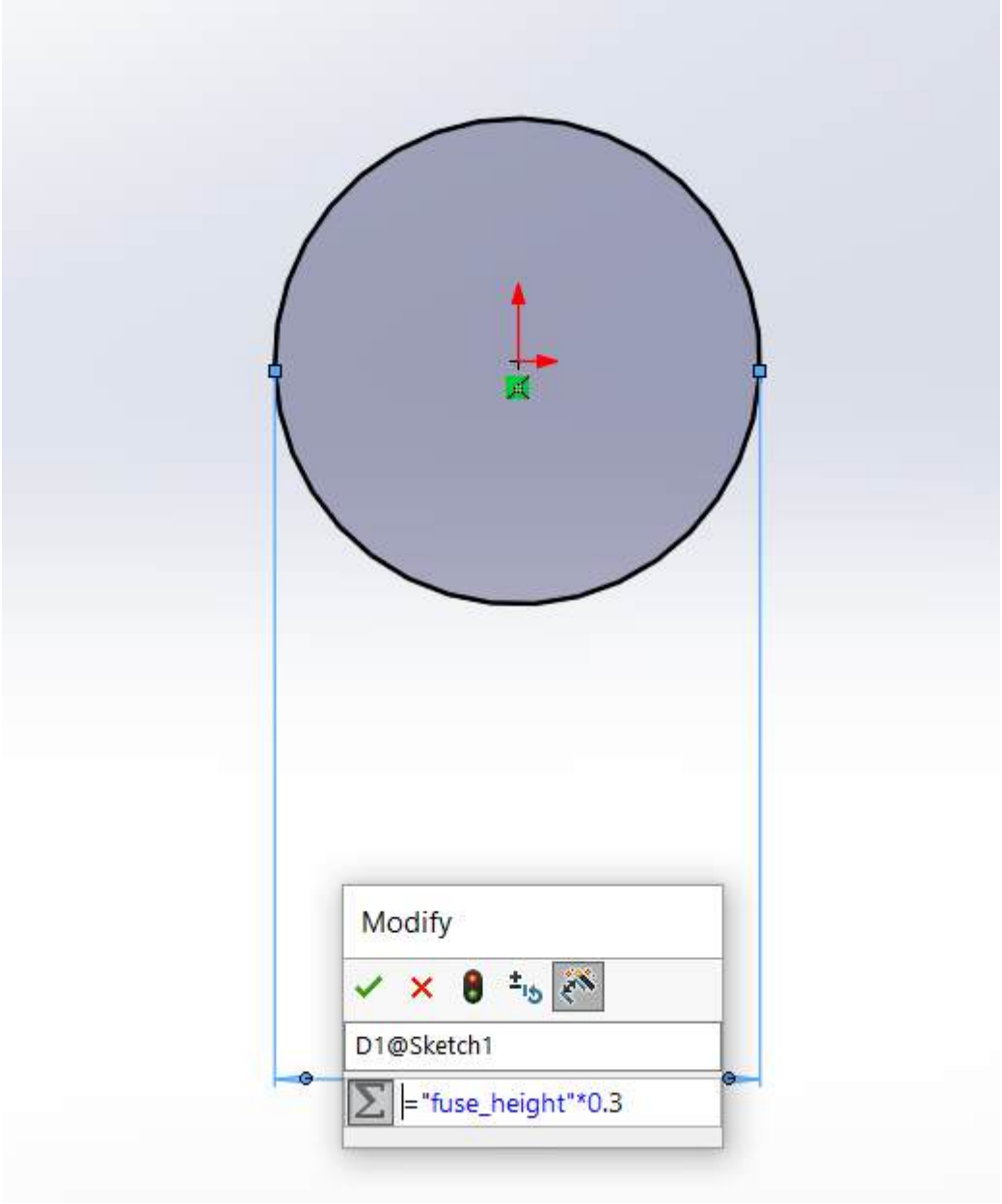


Recall from our initial considerations that the nose should be a circle centered about the center line. Let’s therefore make a circle centered at the origin with an arbitrary diameter that we will set soon:

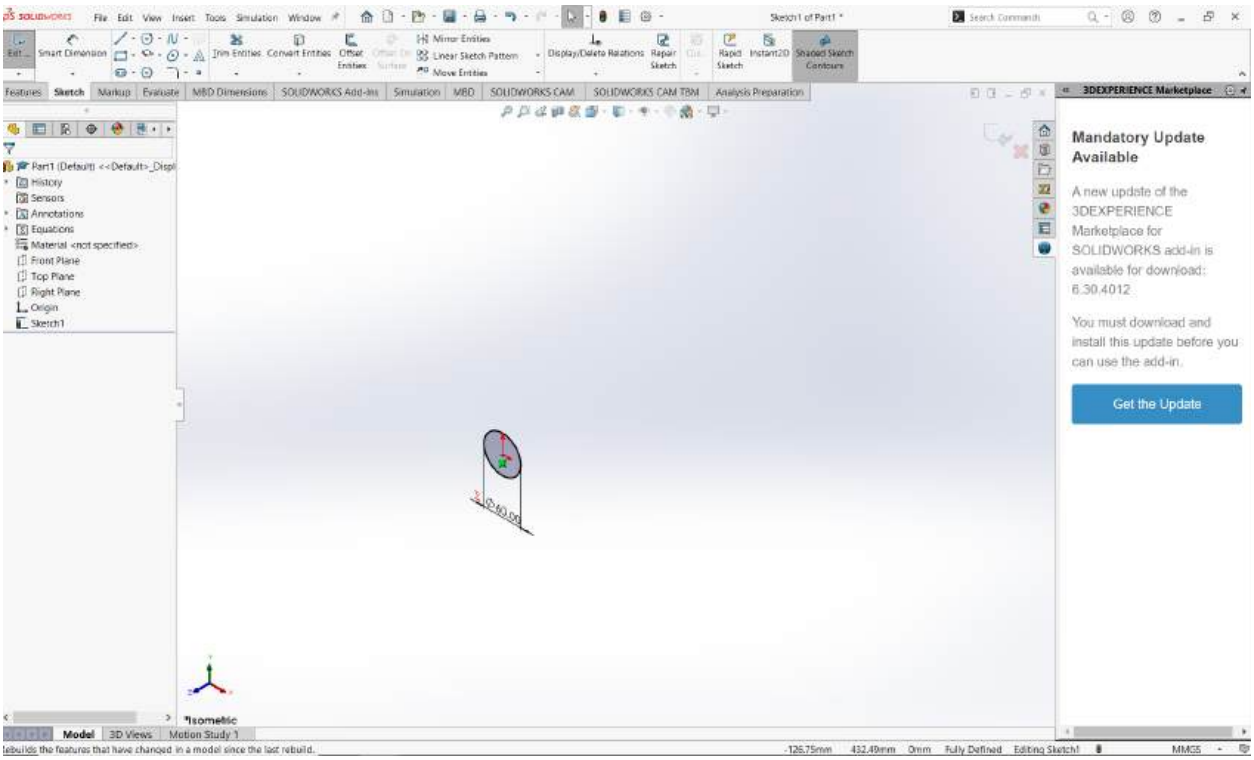


Perfect. To define the dimension of the circle, use the Smart Dimension tool and click the circle. Again, I will assume from now on that you are proficient with constraints and dimensions, so I will not repeat the exact instructions again.

With regards to what length to use, it would be ideal to define this diameter as a function of one of the variables declared earlier. This way, if we were to change the height, length, or width for some reason, the nose would scale accordingly. I will set the diameter to be 30% of the height of the fuselage, like so:



Perfect. Exiting out of the sketch and looking at the isometric view, we can see our blueprint of a nose:



The rest of the fuselage, however, is missing.

Creating the guideline

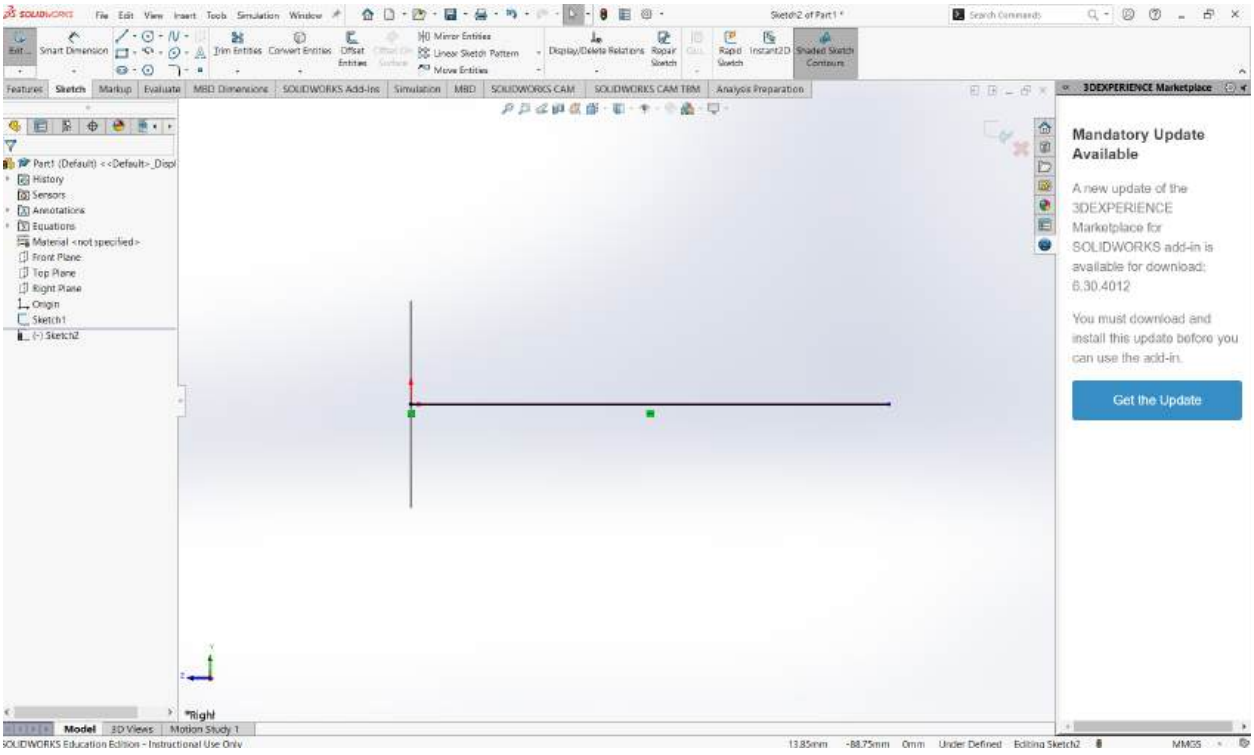
Remember that I mentioned we would have two other cross sections. One for the square-ish center piece and another circle for the tail. We need, therefore, to somehow sketch these other two cross sections. However, they won't be in the front, right, or top plane. They will be in planes parallel to the plane where the nose lies but offset by certain distances. The distance from the plane where the nose is and the plane where the tail is is the length of the fuselage.

We will, therefore, do just that: make our own planes. Well, we've been doing this, but now we'll make another type of plane. A geometrical plane. So far, we have only used the standard front, right, or top, so this is a big step.

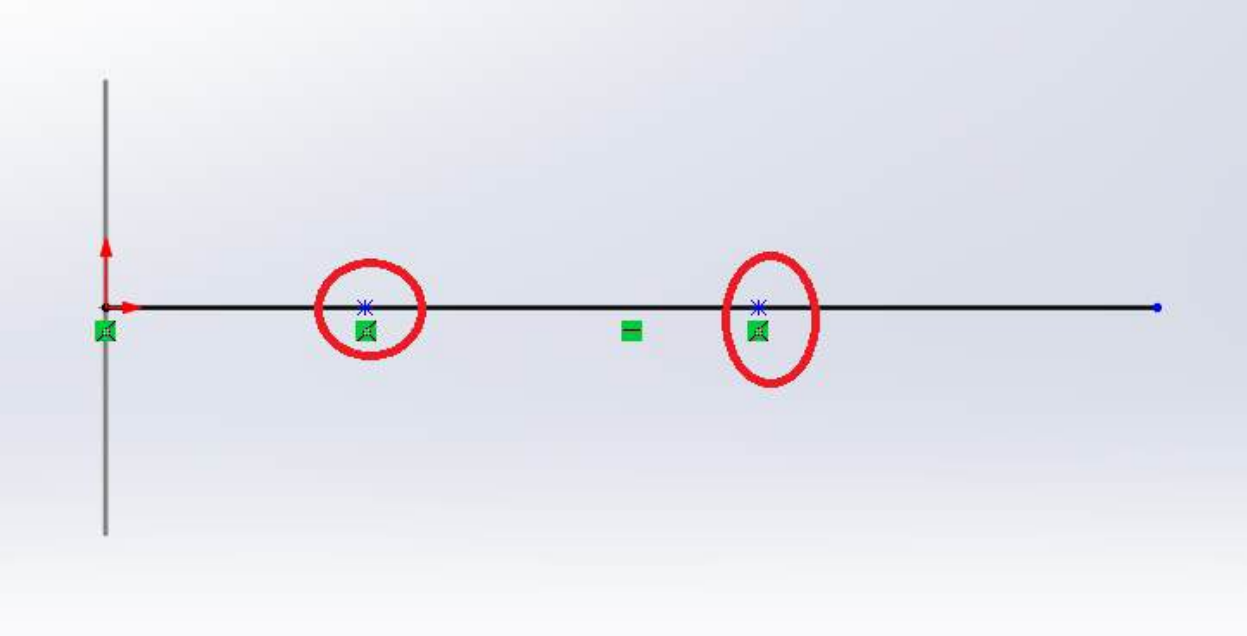
To define where these planes will be, it would be good if we could mark a point in space to later indicate where this plane will be. After all, we already know that it is parallel to the front plane, so Solidworks will be able to tell its normal vector from this information. Recall from Calc 3 (or you will learn in Calc 3, if you have not taken it yet) that a plane is defined by a normal vector and a point. We're missing a point!

Remember that we also want that centerline for dimensioning purposes. Let's combine these needs! Let's make a line in the right plane that contains the origin and points scattered along it that mark where the planes for the cross-sections will be.

To do so, exit the current sketch for the nose circle. Create a sketch in the **right plane** and draw a horizontal line from the origin to the right with an arbitrary distance. We will dimension it later. You should now see something like this:

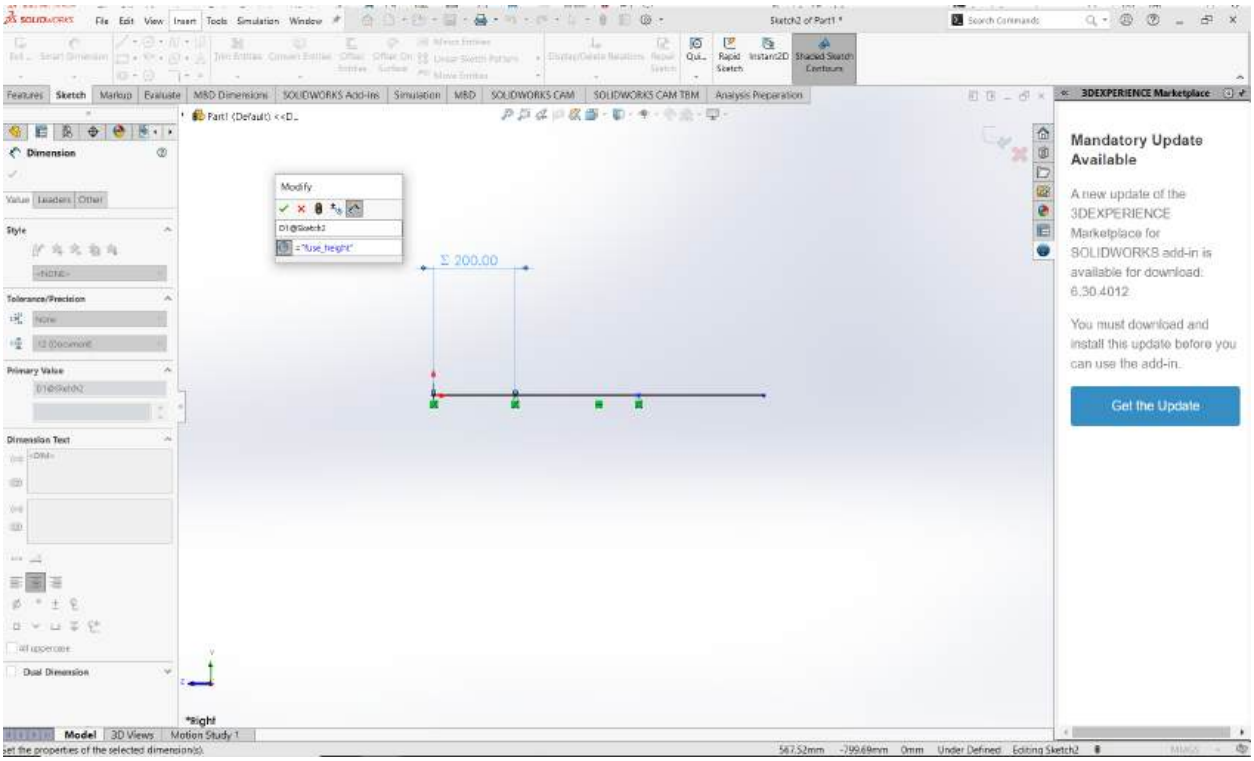


Great! Because we only have two more cross sections, one for the center part (squareish) and one for the tail, we only need two points on this line. With the point tool, mark two points arbitrarily in this line, like so:



Awesome! Now time to dimension them.

From the drawing on the whiteboard, note that the distance from the left most point to the origin should be the height, the distance from the right most point to the end of the line should be 3 times the height, and that the distance between them should be the fuselage length minus 4 times the height. We dimension them like so: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Mandatory Update Available
A new update of the 3DEXPERIENCE Marketplace for SOLIDWORKS add-in is available for download: 6.30.4012
You must download and install this update before you can use the add-in.
Get the Update
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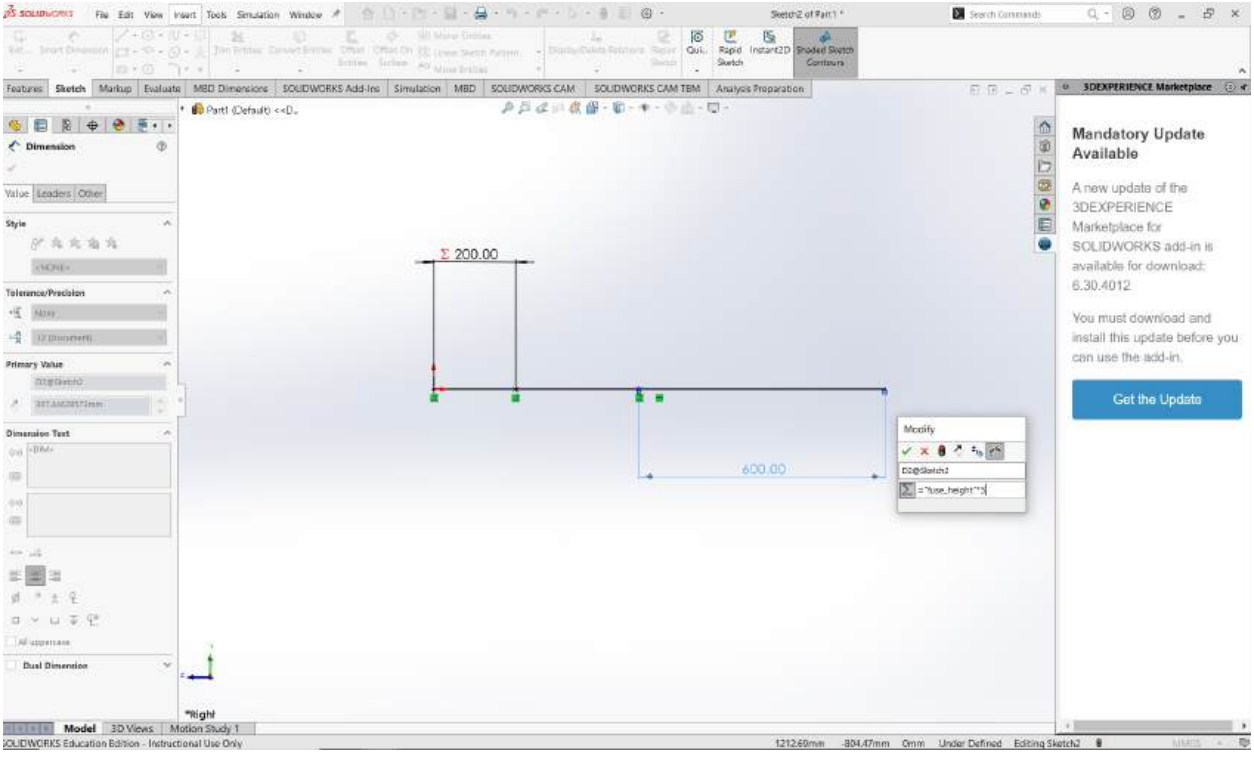
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You must download and install this update before you can use the add-in.
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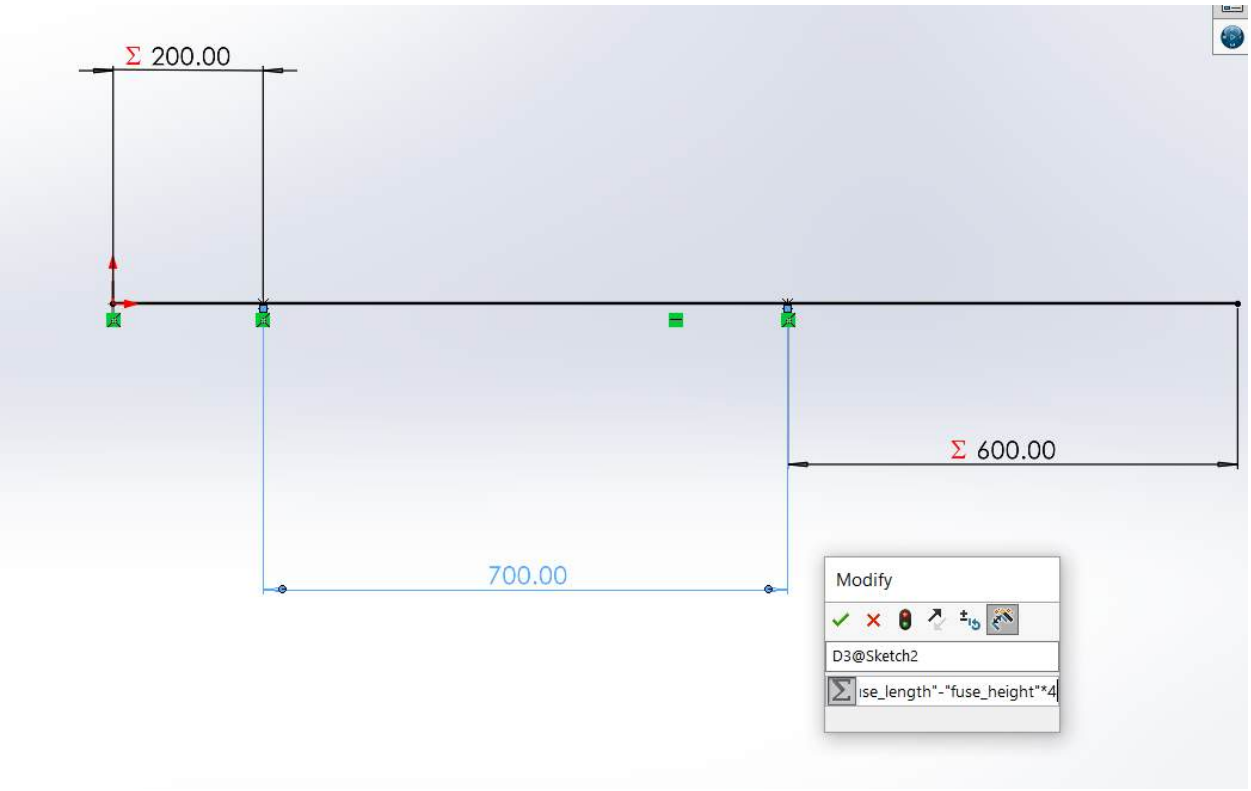
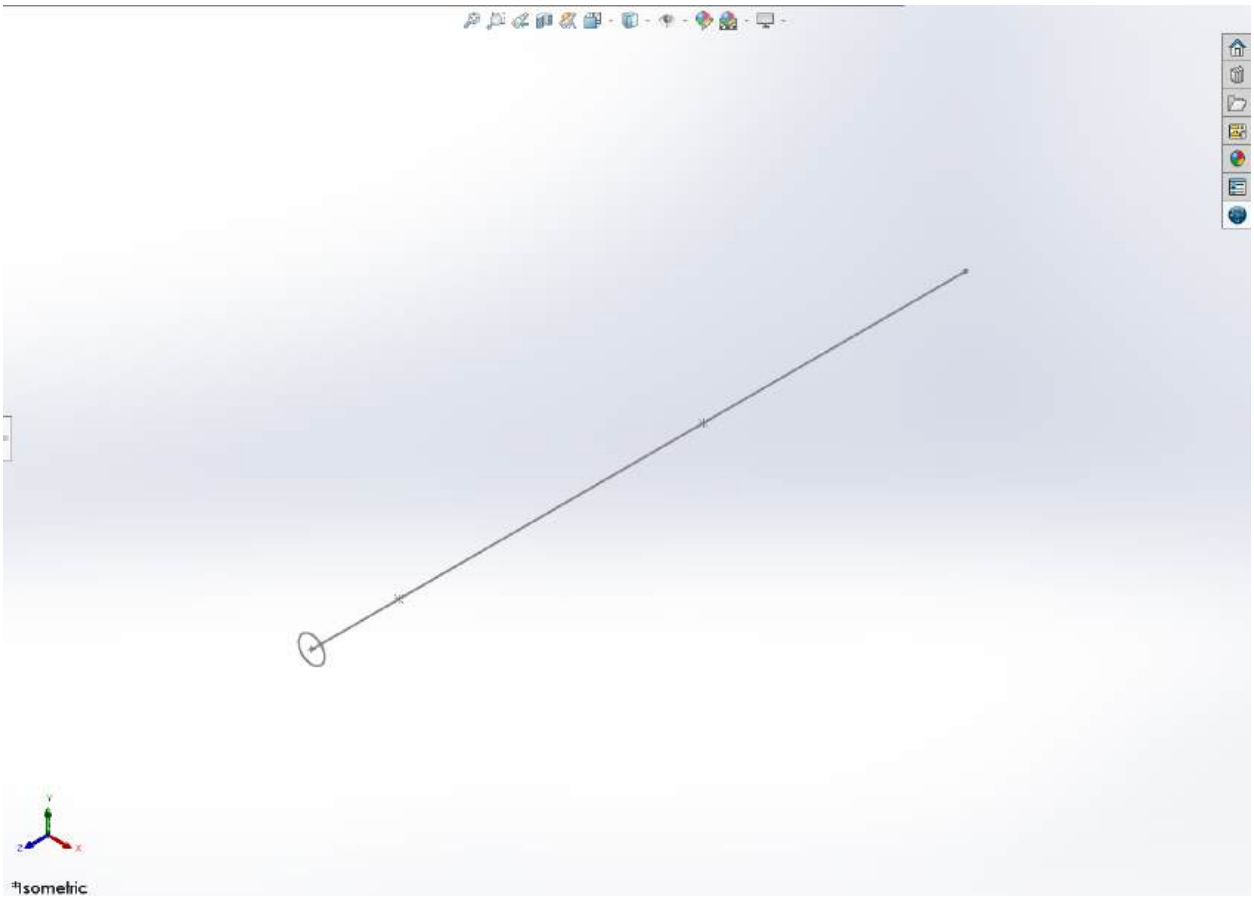


Perfect!

Exiting the sketch and looking at the isometric view, you should see this:

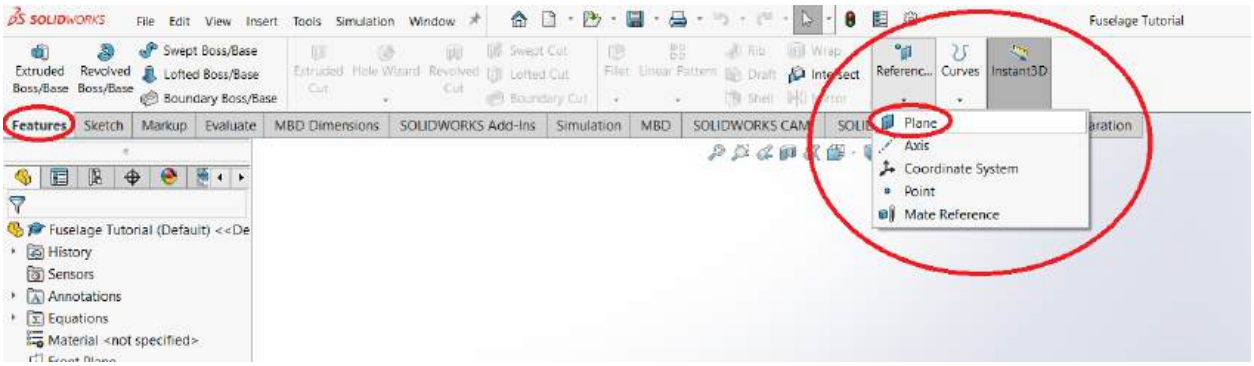


Looking good! We have our nose profile and three points, two of them indicating the places where we'll create planes to sketch the cross-sections. Let's get on with it then!

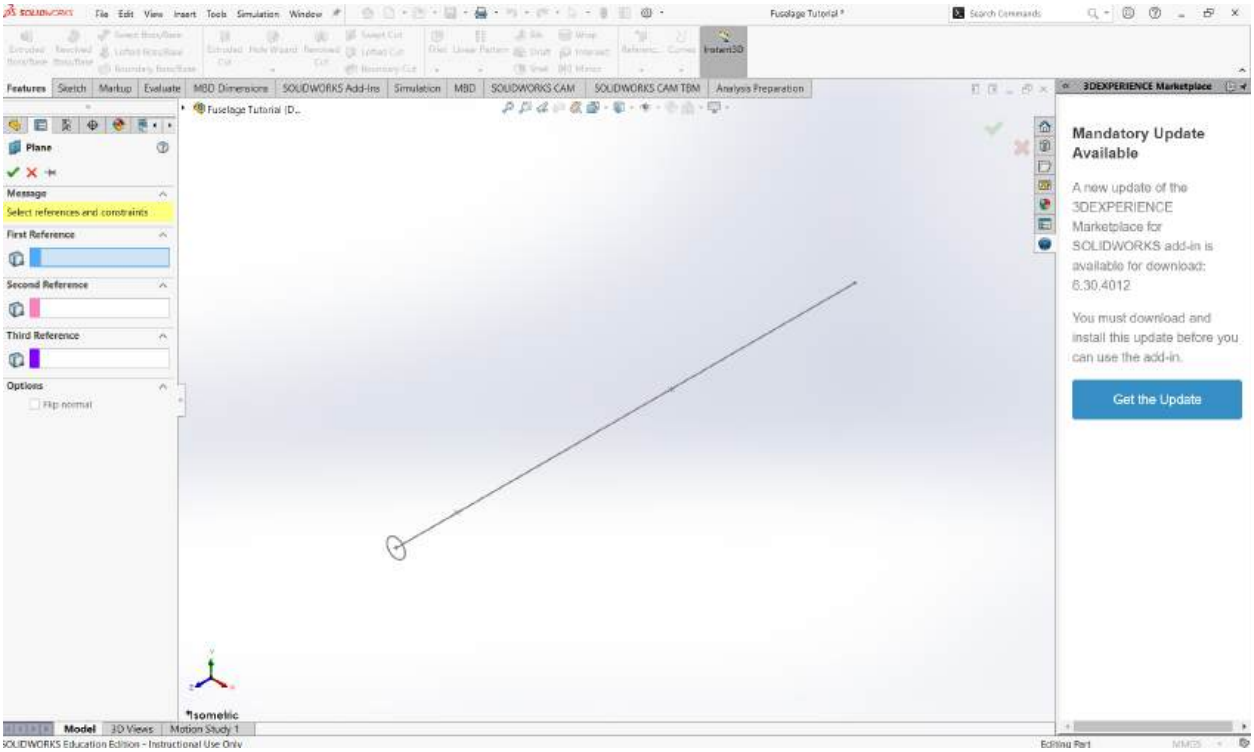
Center part cross-section

To create the center part cross-section, we must first create a plane to sketch the cross-section in. Looking at the screenshot above of the guiding line, the first point after the nose is where we want that plane. It should also be parallel to the front plane. To create this plane, we will use a new tool: reference geometry.

Under Features, select the Reference Geometries dropdown arrow, and select plane, like so:

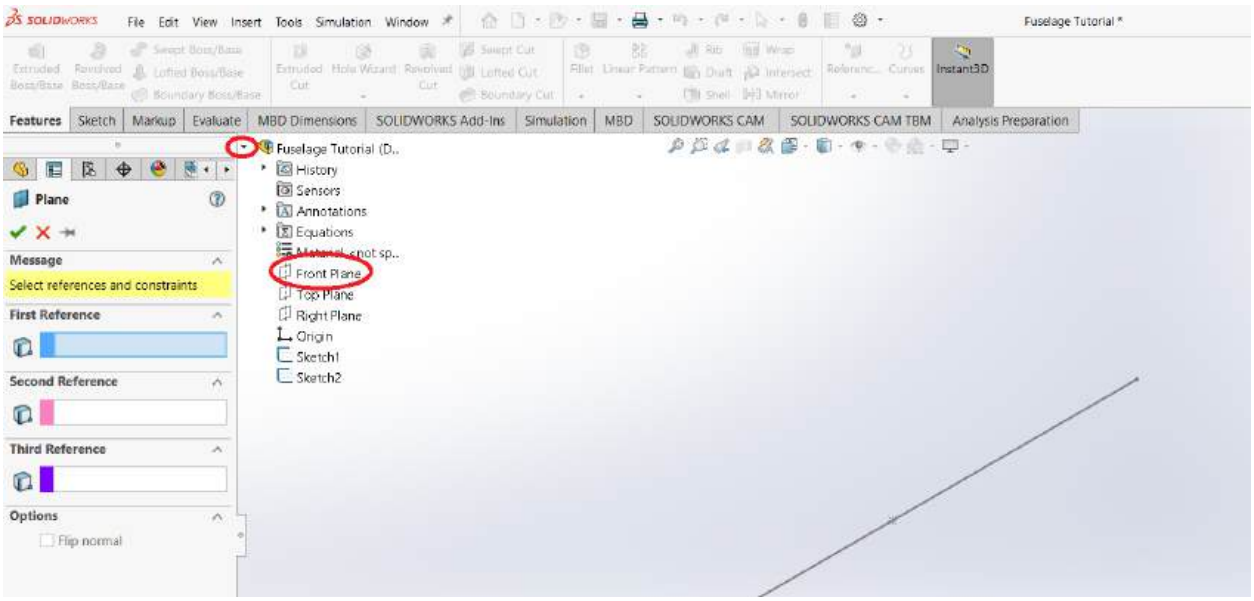


You should now see something like this:

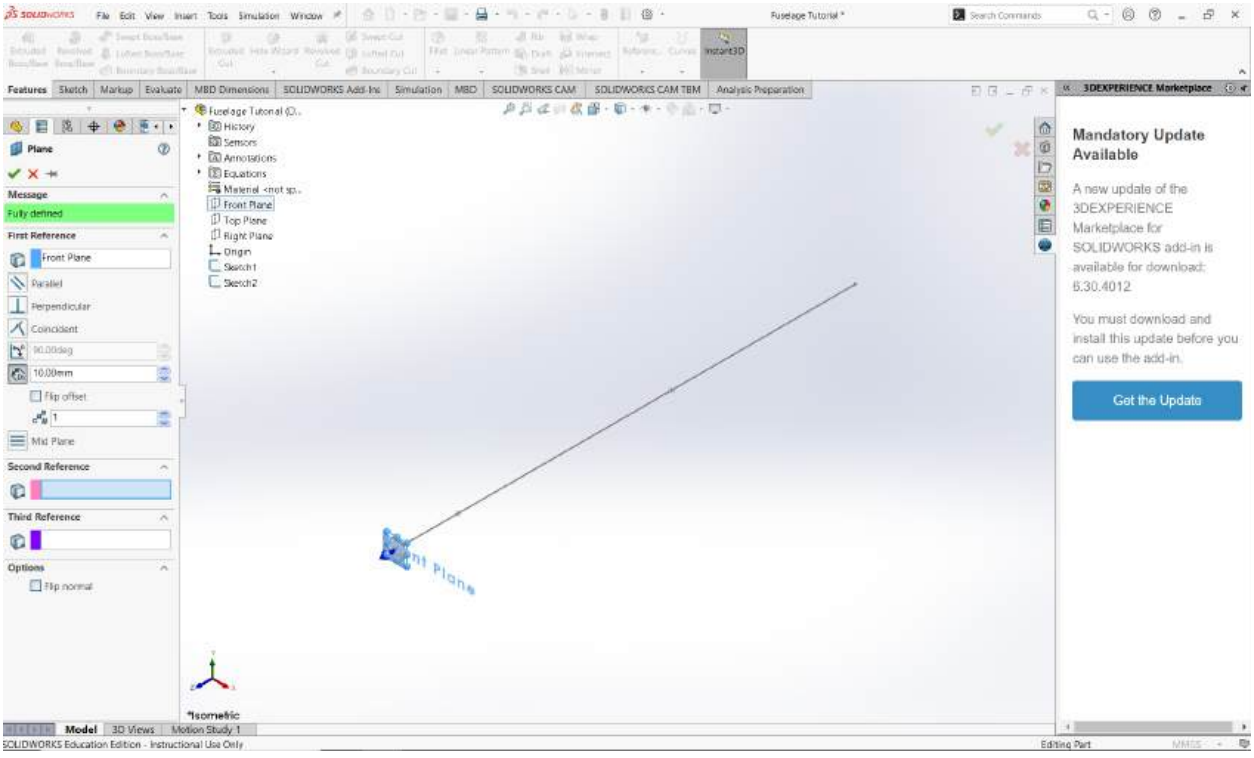


Note how we are asked to select references. Recall that we only need a plane and a point to sufficiently define our plane. Thus, our first reference will be the plane - the right plane, in this case - and the second will be that point in the guide line where our plane needs to be.

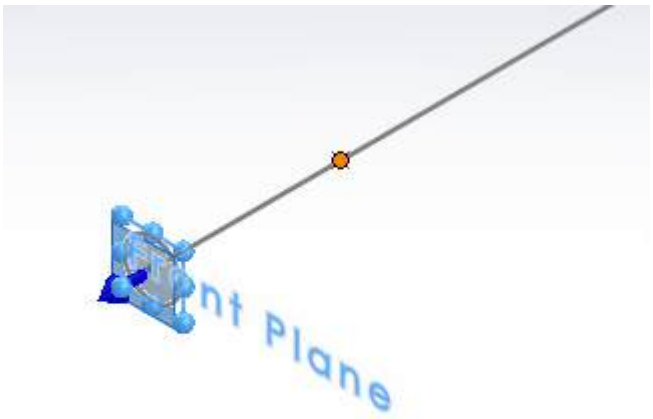
To choose the right plane, click on the dropdown arrow to expand the tree, like so:



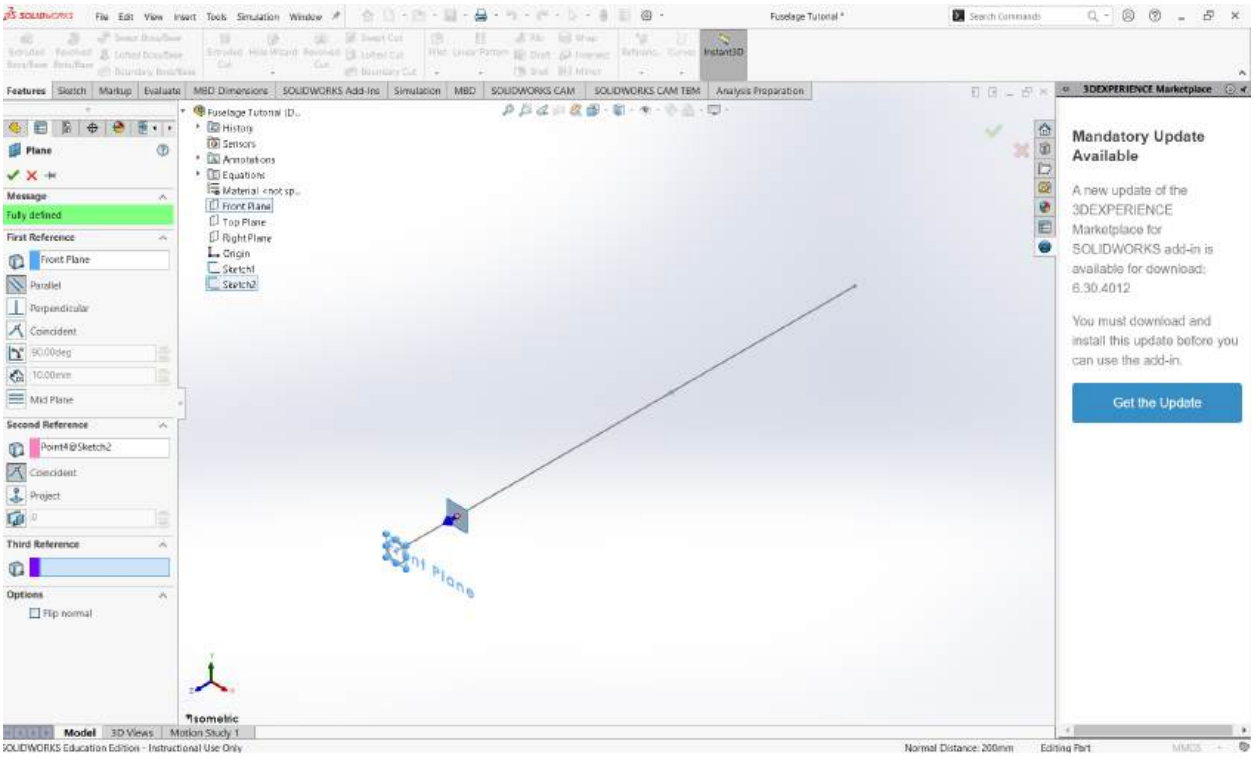
Then, click the front plane, highlighted in the picture above. You should now see this:



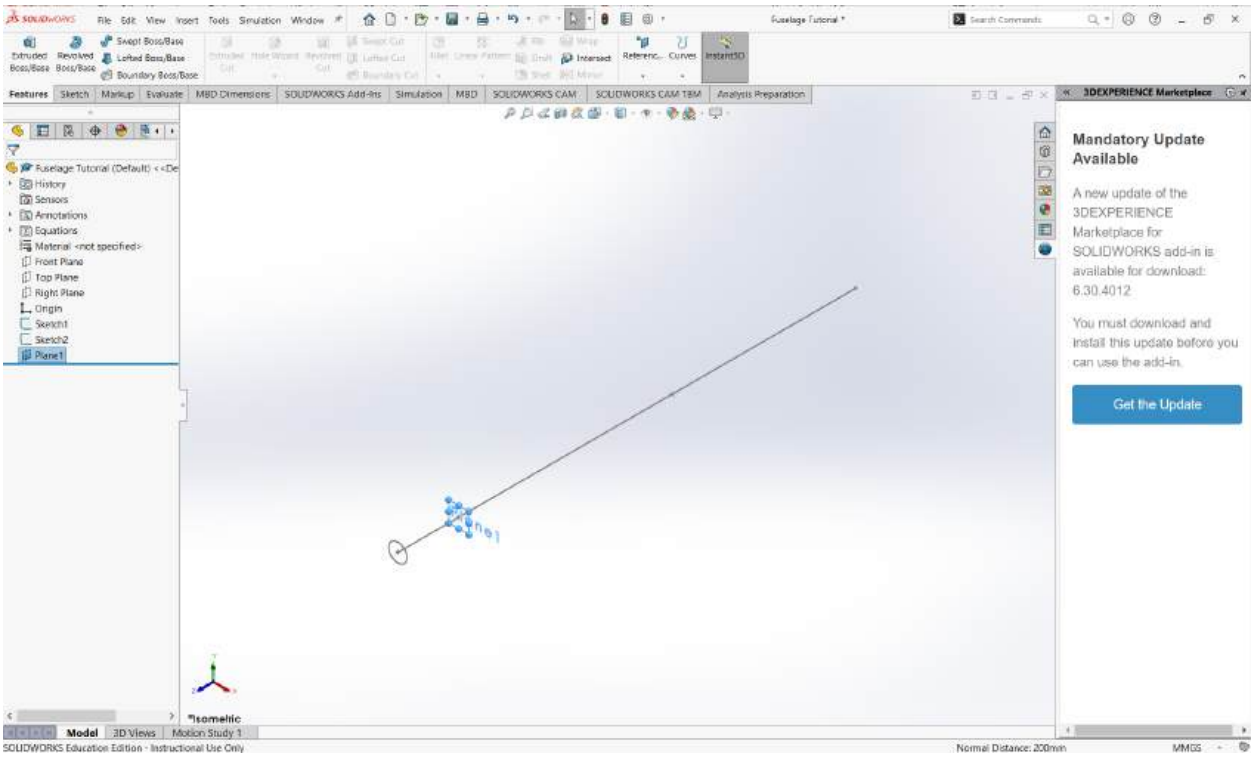
Perfect! Solidworks should automatically take you to the second reference option. If not, simply click it on the left. Now, to select the point, merely click on it! It might be a bit hard to get the mouse to highlight it, but once it is highlighted, it should look like this:



Notice how the point is highlighted. When this happens, simply right-click. You should now see this:

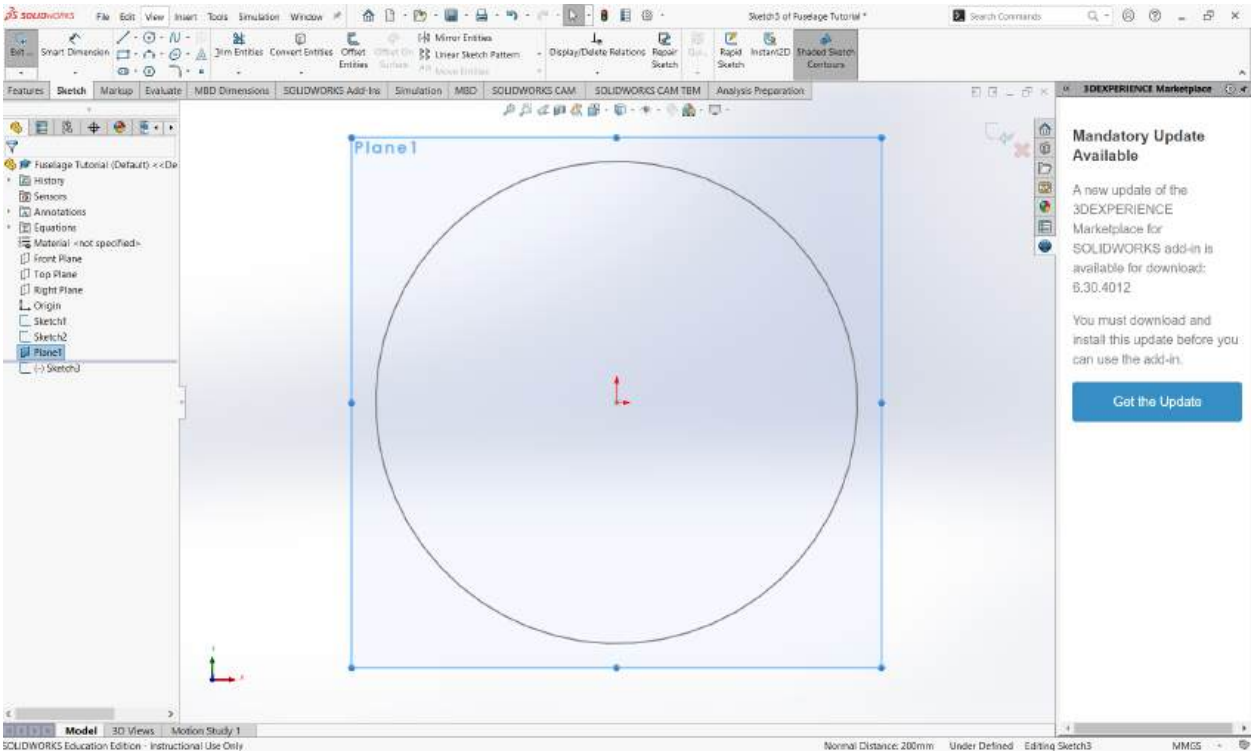


Awesome! It’s now in the right spot! Confirm by clicking on the green tick on the left, and you should now see the plane created:



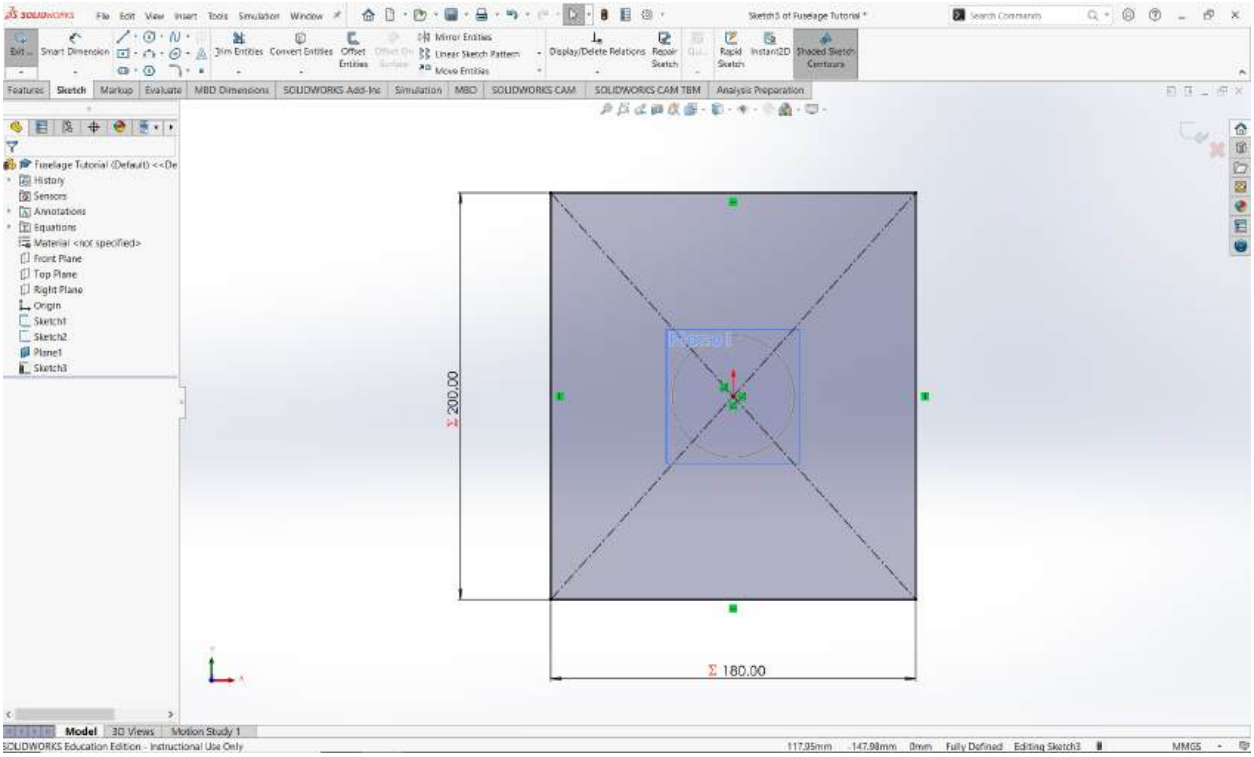
Great! We now need to create a sketch on that plane to make the cross-section.

With the plane selected, as it should automatically be (if not, simply click on it in the tree on the left), create a sketch by going under Sketch and clicking “Sketch:”



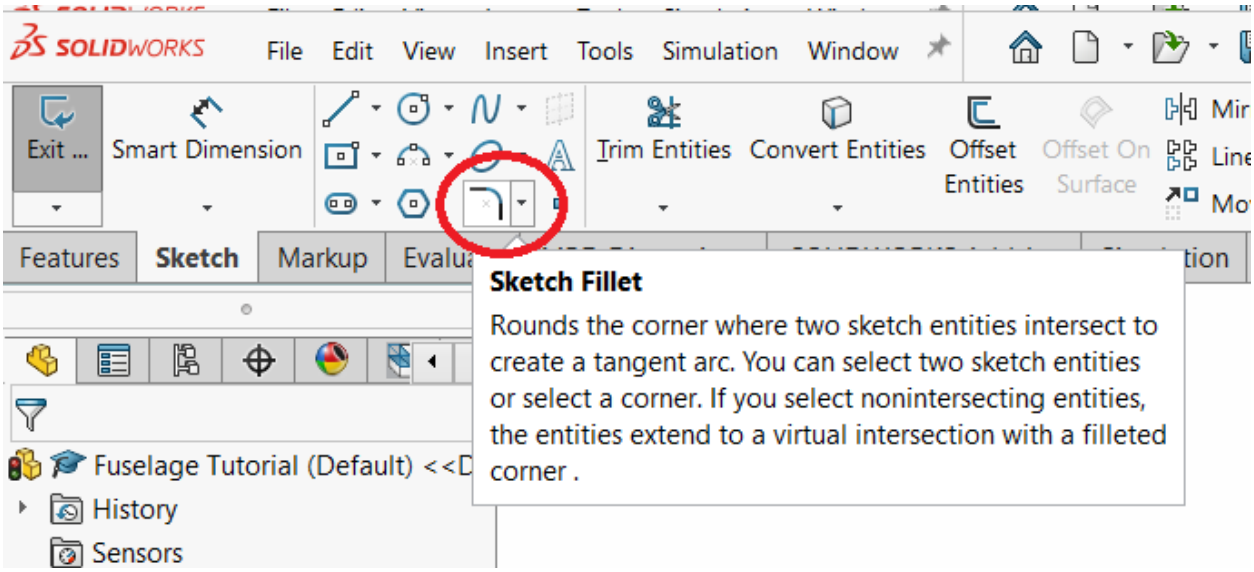
Great! Although you see this circumference, which you should recall it to be the nose, we are not in the same plane as the nose, as indicated by the “Plane1” in blue. It just happens that the sketch for the nose is not hidden. We don’t want to hide it, it’s actually a good thing it is there, since we can line things up with the origin (and, subsequently, the nose itself.)

Let’s create the cross-section for the center part. First, we create a rectangle centered at the origin with width equal to the fuse_width variable and height equal to the fuse_height variable. After doing so, you should see this:

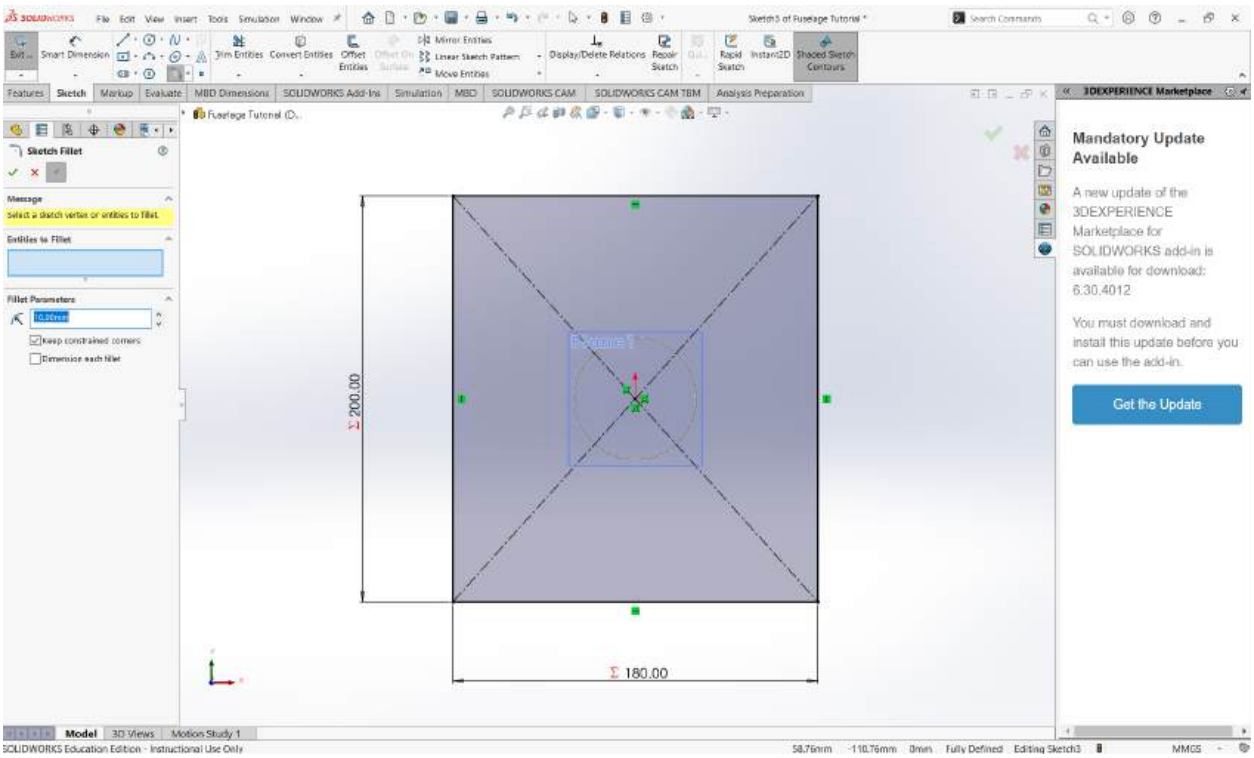


Awesome. However, fuselages aren't square like that. We want to add a fillet. So far, however, we have only used these corner-altering tools as features of a 3D body, not a sketch. Remember we used fillets for the wheel of the landing gear. We didn't use it on the sketch, though, but rather on the solid body.

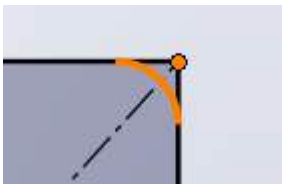
Although fillets are generally used in solid bodies, there is also a tool for sketches. Under sketches, there is something called a "Sketch Fillet," appropriately denoted with the icon showing a corner being rounded:



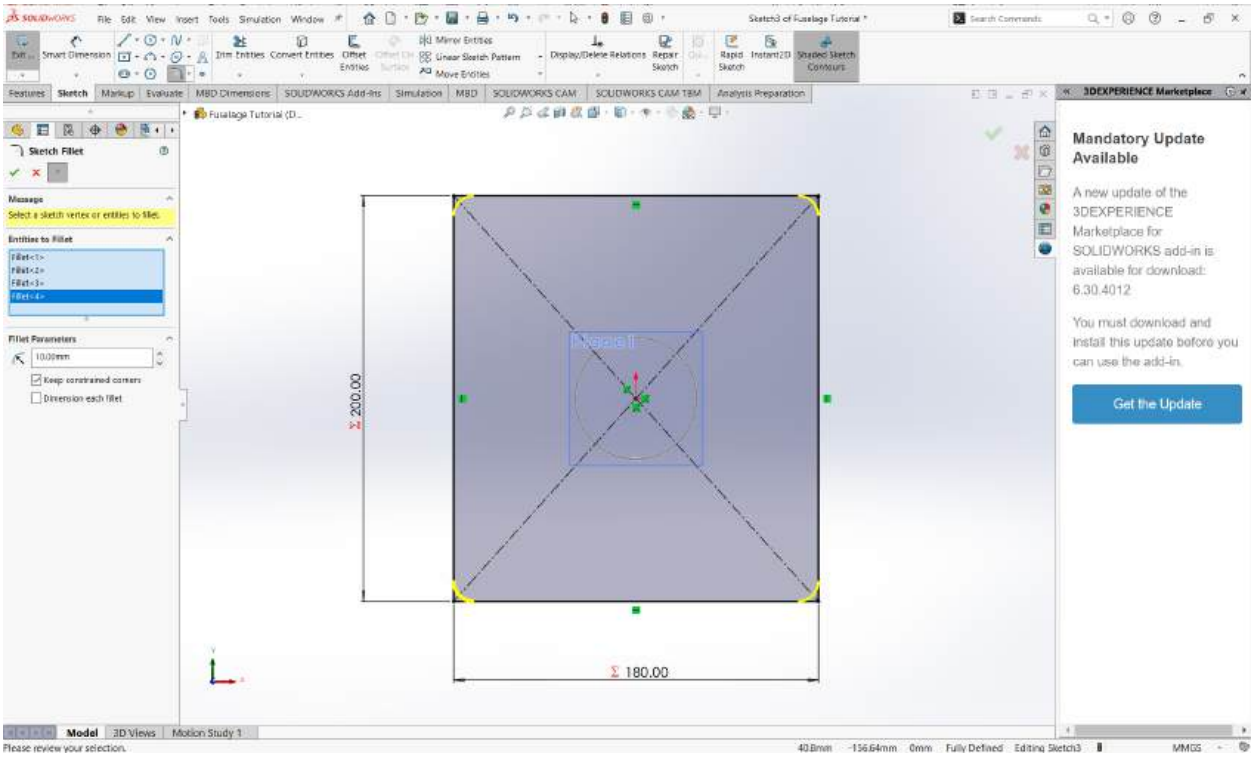
Select this tool. You should now see this:



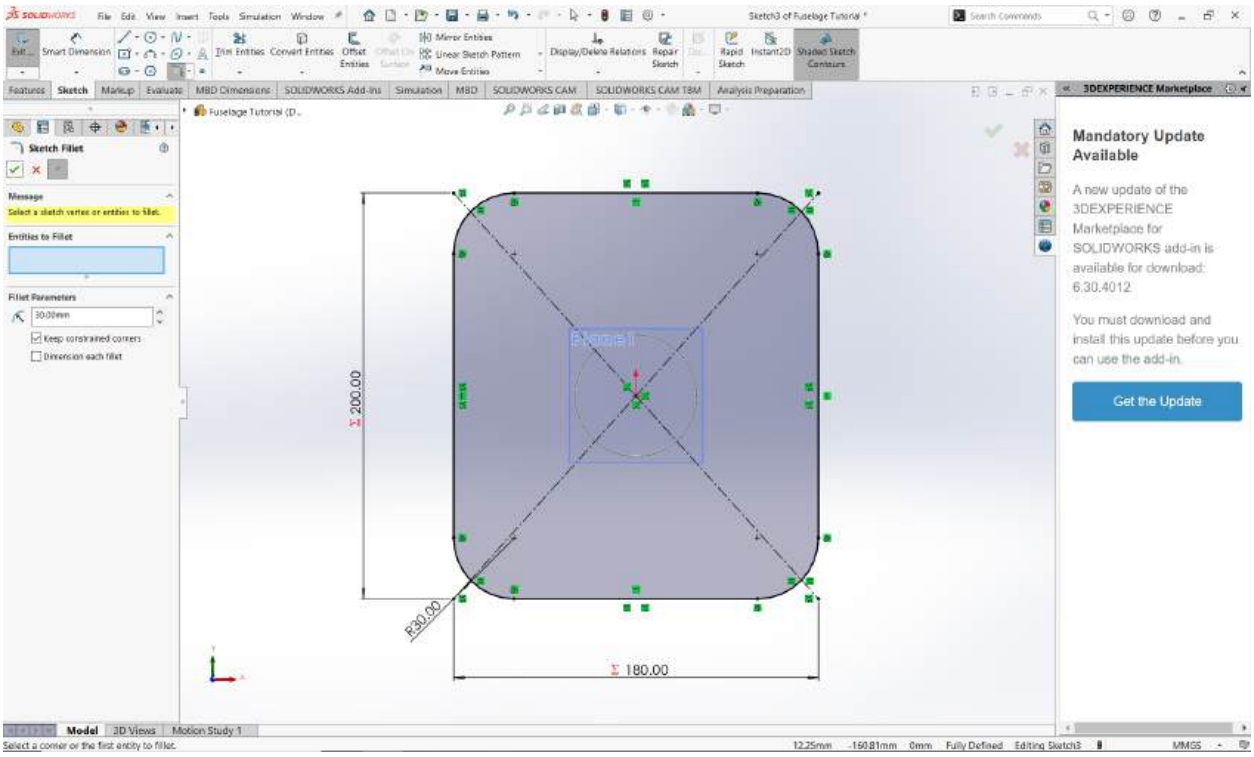
It asks us to select the corners that we want to fillet. Select all four corners of the sketch. To select the corner, simply hover over the point where two sides intersect and select that point. When you are hovering, you should see something like this:



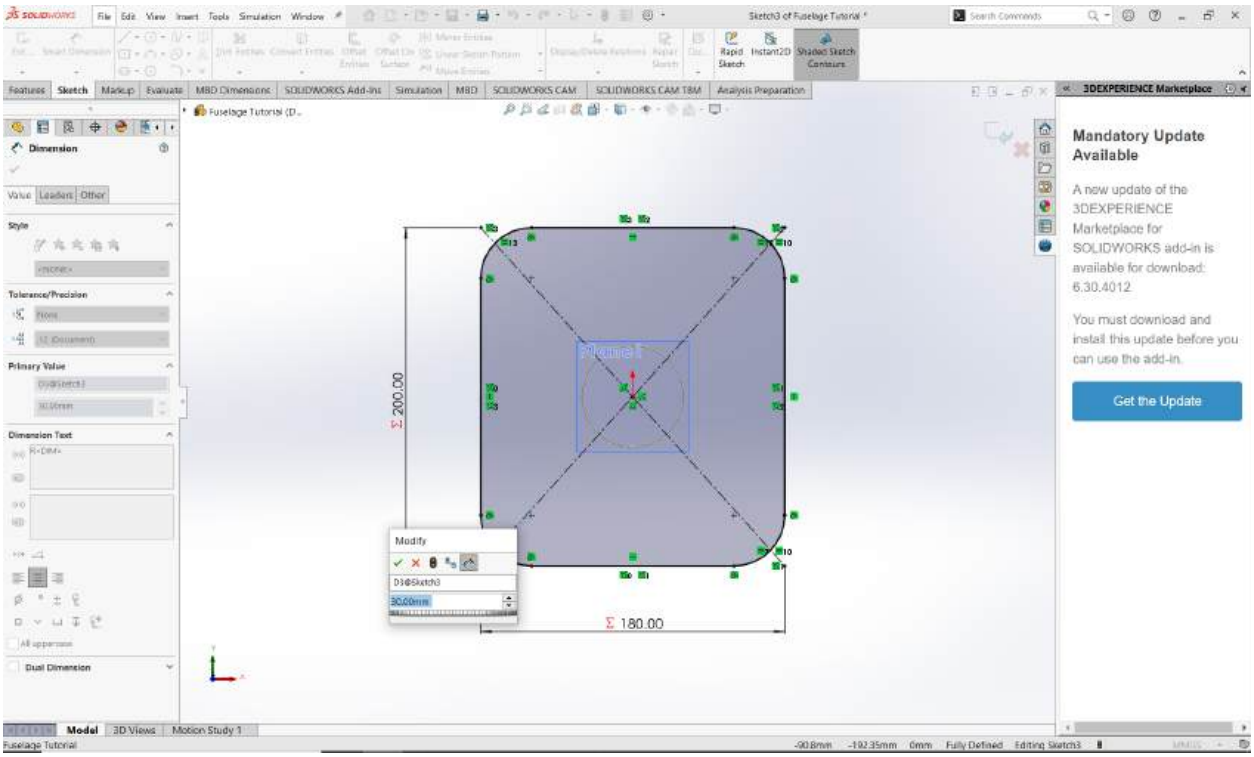
After selecting all four corners, you should see this:



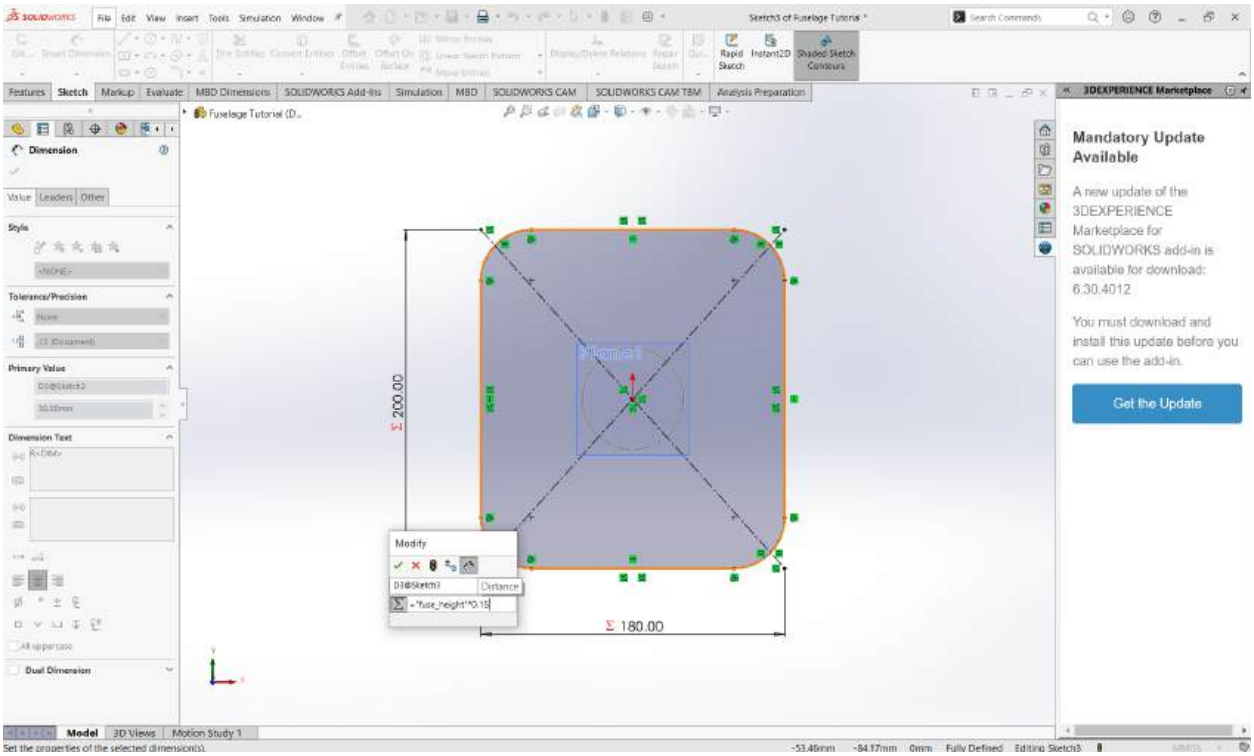
Now, we need to select a radius for these corners. This is more of a matter of artistic freedom, what looks right. For this, I have found that around 30mm works fine. Under “Fillet Parameters,” type 30mm and confirm the fillet by clicking the green tick on the left menu. You should see a beautifully filleted cross-section:



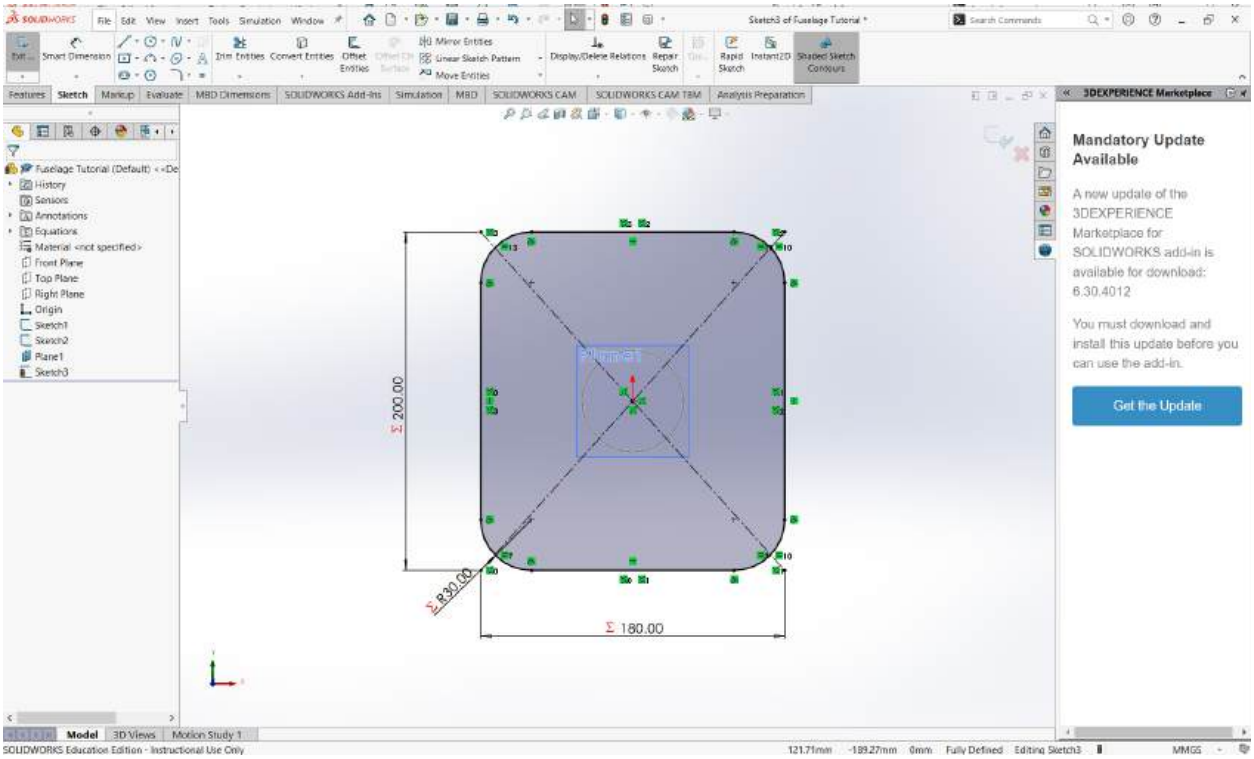
Awesome! We’re almost done. You might be wondering why we did not make this radius as a function of some other variable to make our design more dynamic. Good observation! For some reason, the sketch fillet won’t allow you to use a variable as a radius (weird). However, after the fillet is applied, we can change the radius using smart dimensioning. See that “R30.00” on the sketch? With the Smart Dimension tool selected, double click that. You should now see this:



Now, simply type “=” to bring up the equations menu, navigate to the fuselage height, and make this dimension 15% of the height by multiplying “fuse_height” by 0.15, like so:

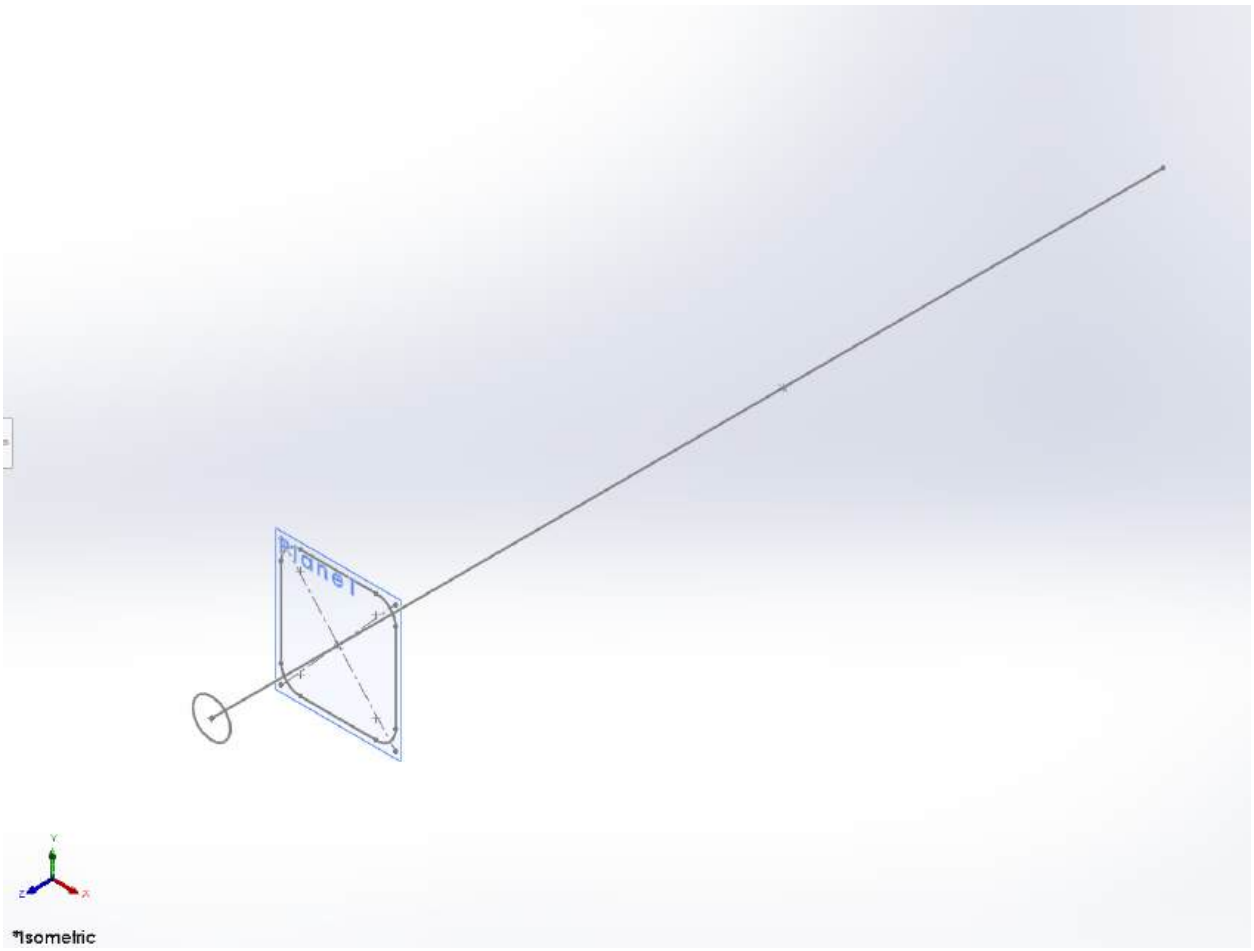


Confirm the dimensioning. The sketch should look almost the same, like so:



However, note the sigma next to the radius. It’s dynamic now!

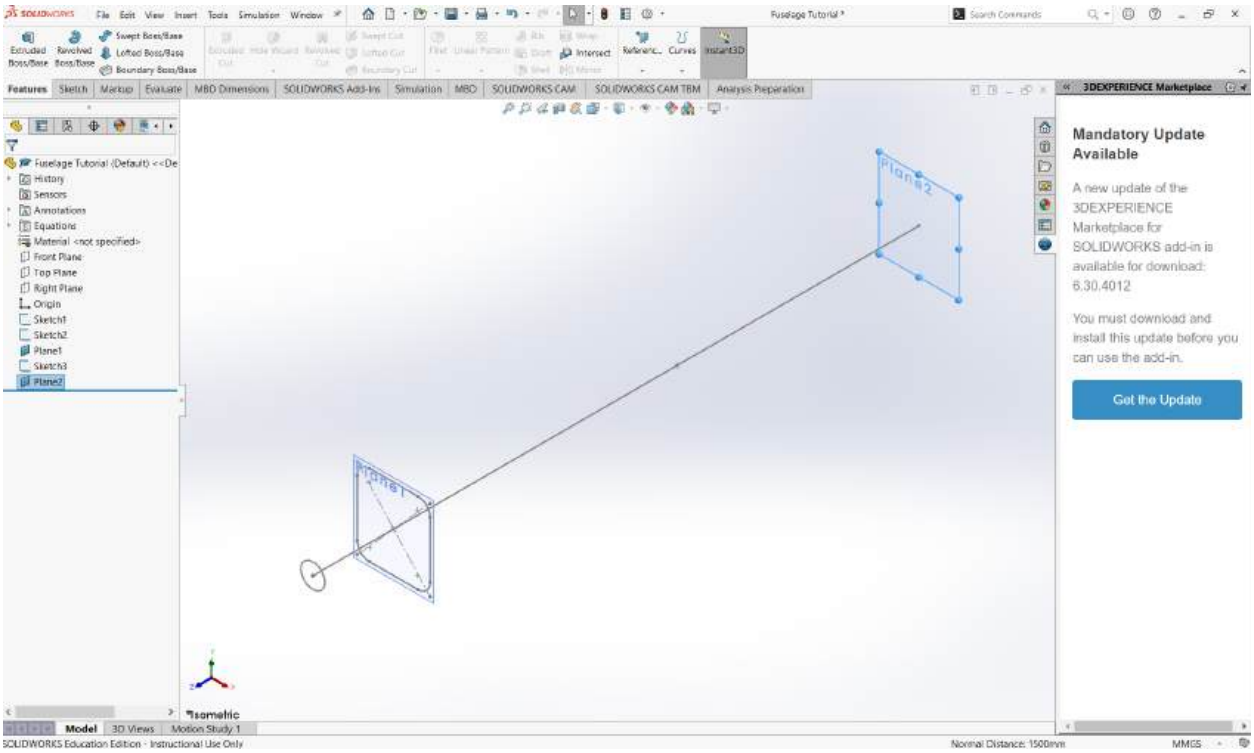
This is it for the cross section. Awesome! Looking at the isometric view of the fuselage so far, it should look like this:



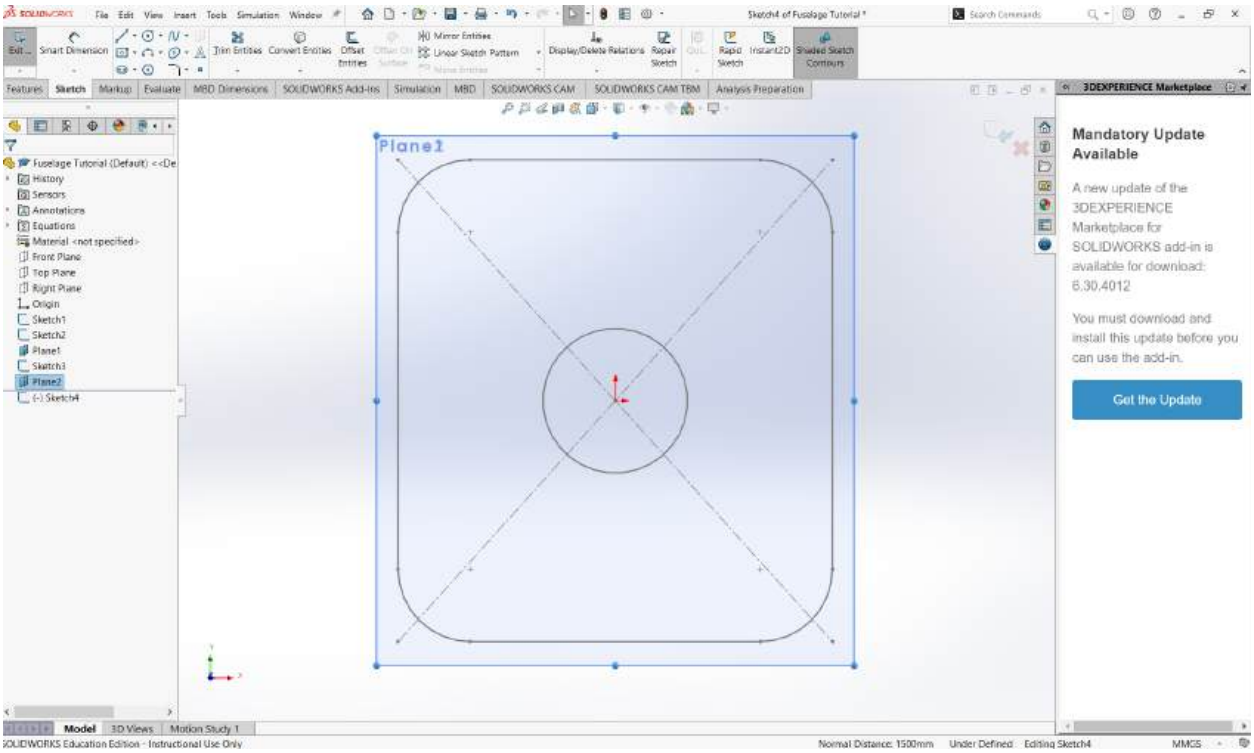
Beautiful! Let’s now work on the last cross-section: the tail!

Tail cross-section

To start with the tail, we need to once again create a reference plane, but this time on the furthest point of the guide line (the point in the tail). Repeat the process for creating the center part plane. As the first reference, select the front plane again. This time, however, select the last point as the second reference. You should have a plane like so:



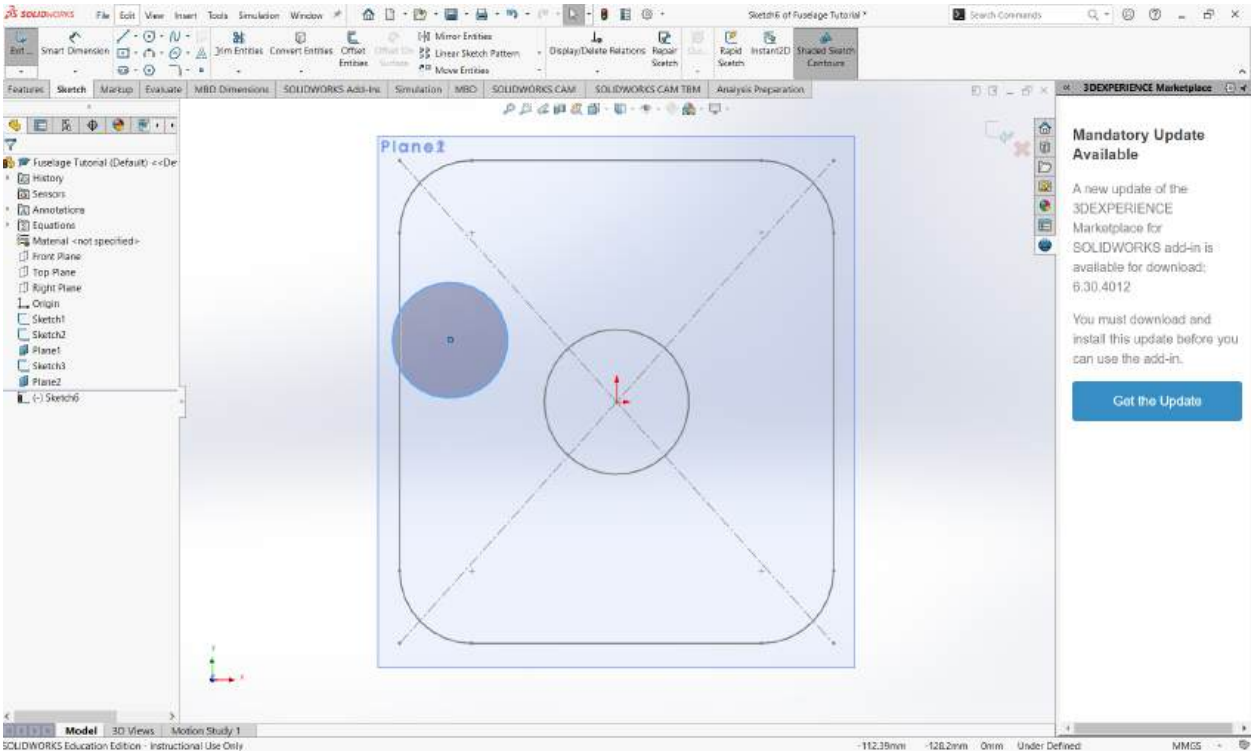
Create a sketch on that plane by going to Sketch, then Sketch. You should see this:



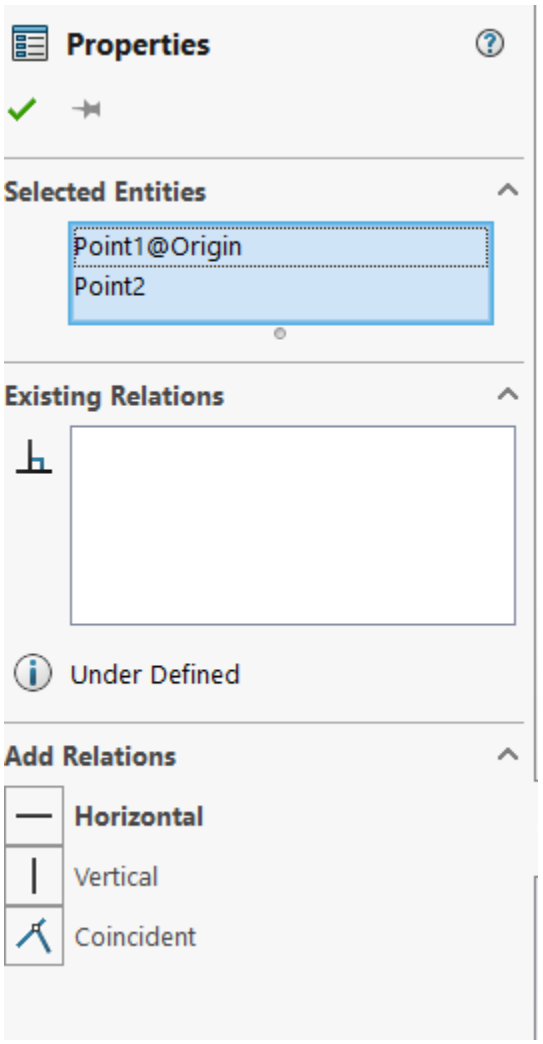
Perfect. Once again, no need to hide the other sketches. We will use them to our advantage.

The cross-section for the tail should be once again a circle. However, its diameter should be 15% of the fuselage width (”fuse_width”*0.15) and it should not be centered about the origin. Rather, its center should be aligned vertically with the origin, but it should be tangent to the top of the center part cross-section.

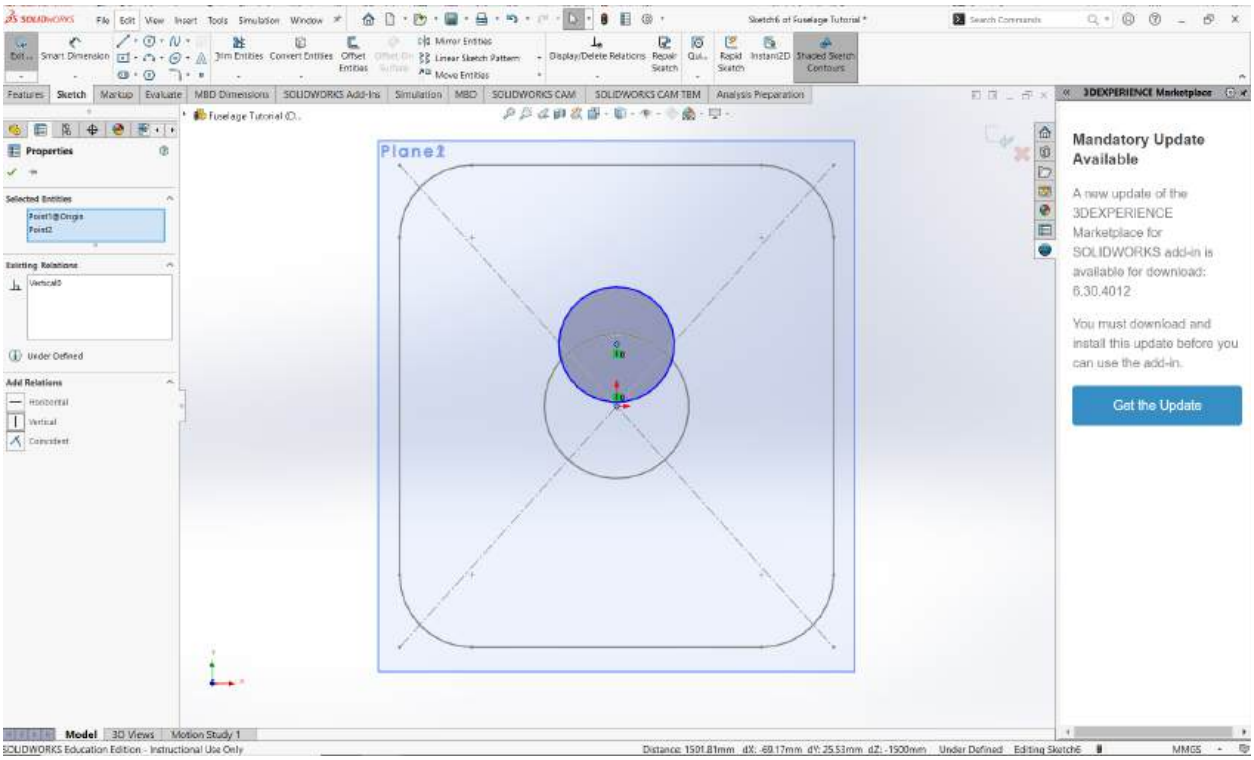
To make this, create an arbitrary circle not centered about the origin:



Then, select the center of the circle and the origin. On the left menu, the following will be shown:

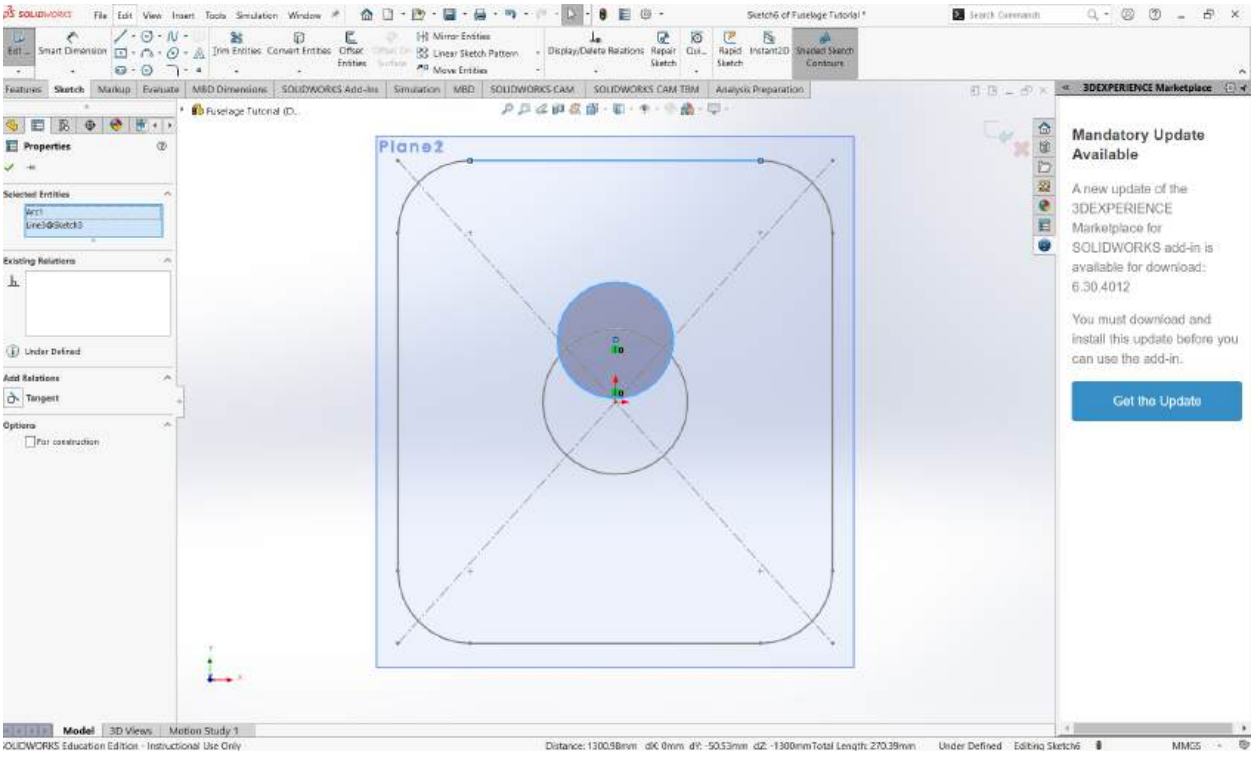


Choose the Vertical relation, and confirm. You should see something like this:

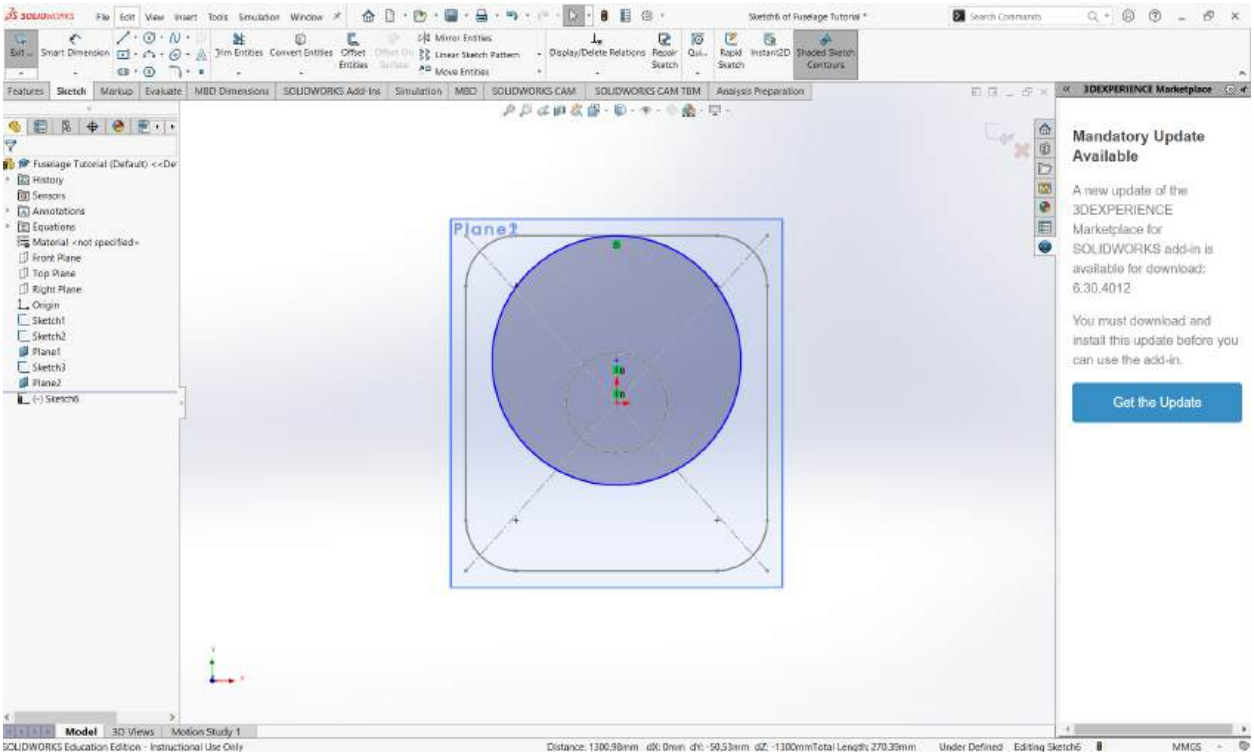


Awesome. confirm out of the constraint menu. Let’s now make this tangent to the top of the center part cross-section.

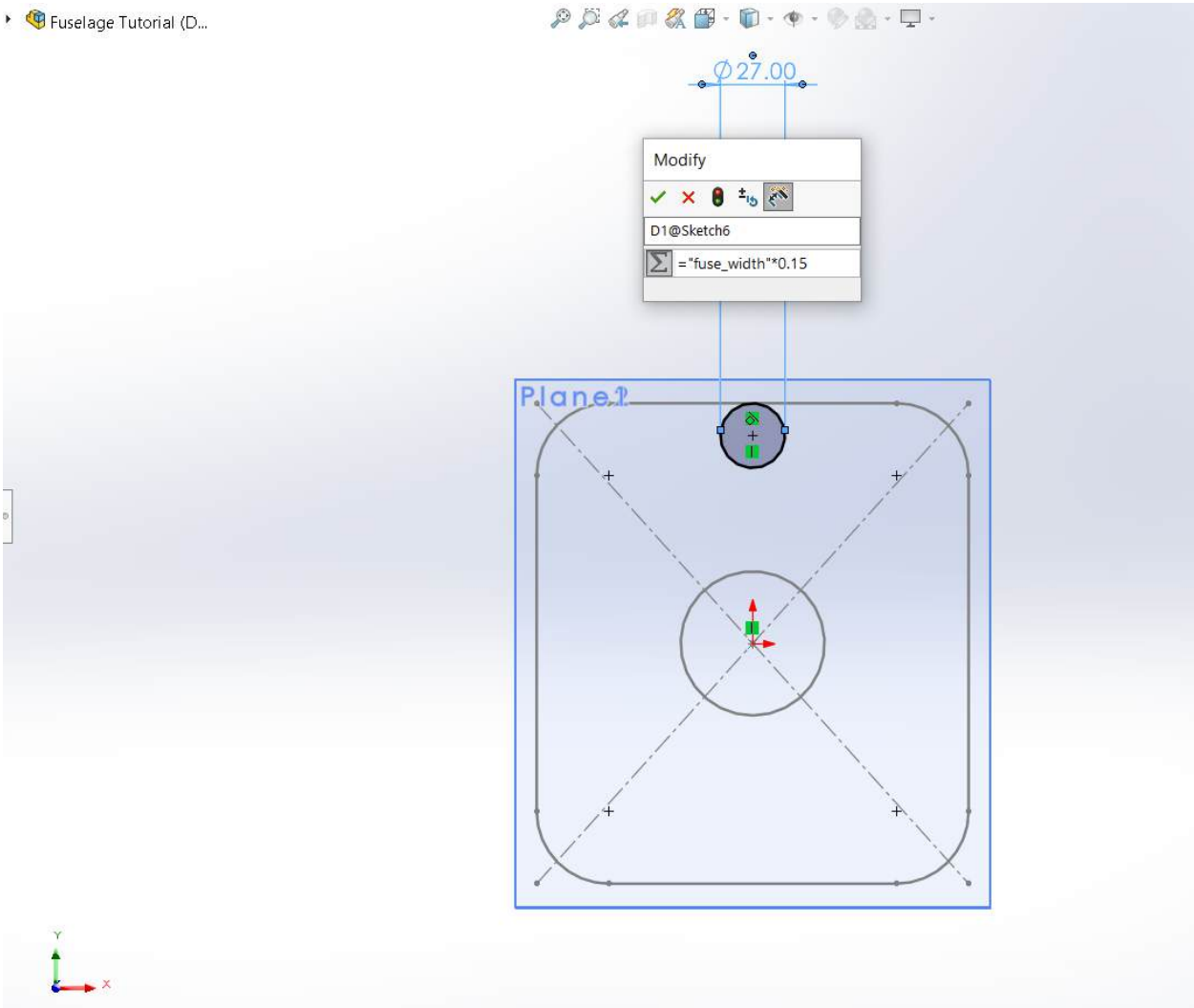
Simply select both the circle (its circumference) and the top of the center cross-section:



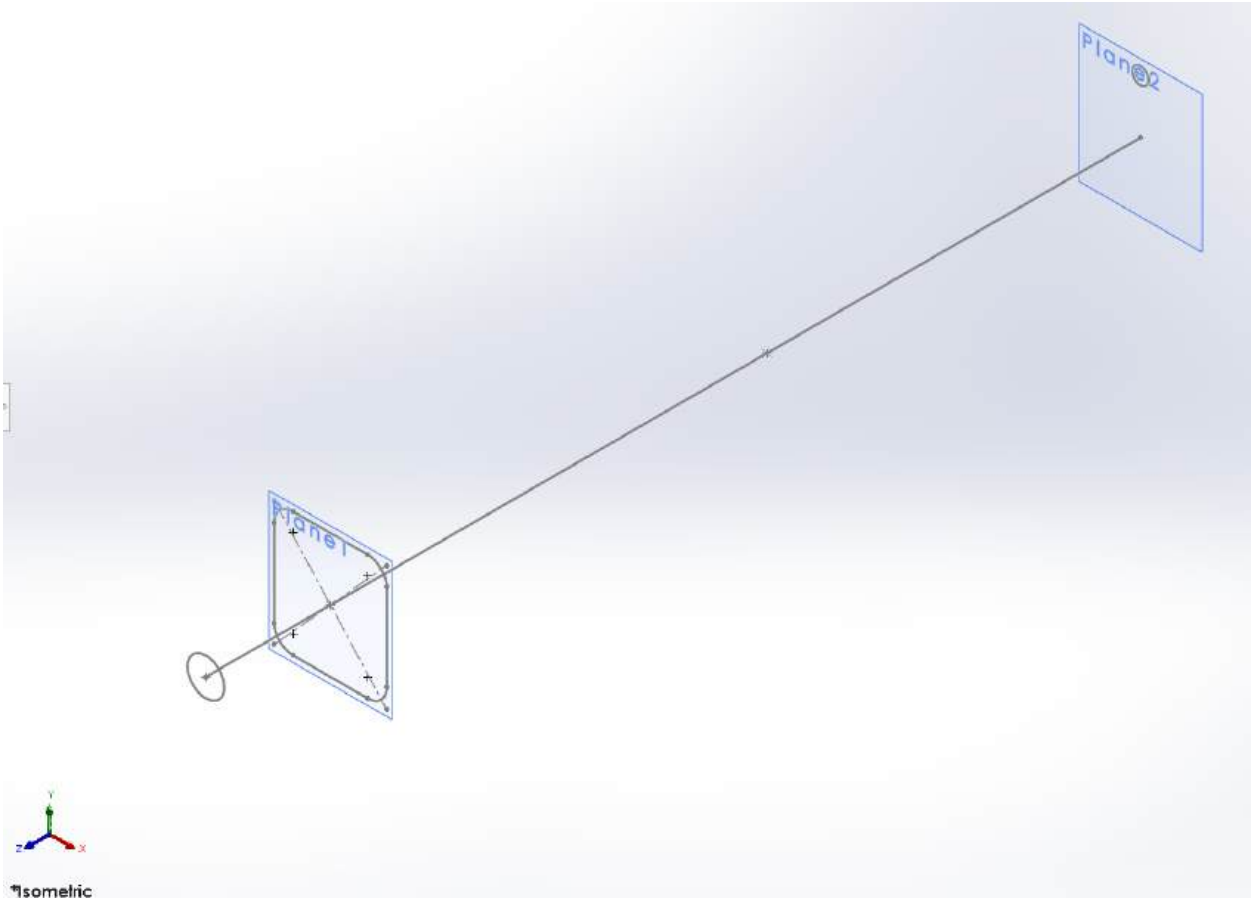
Select the tangent constraint and confirm. You should now see this:



Lastly, let’s dimension the diameter. Select the Smart Dimension tool, the circle’s circumference, and make the diameter “fuse_width”*0.15:



Great! Confirm out of everything. We’re done for cross-sections! The isometric view should look like this:



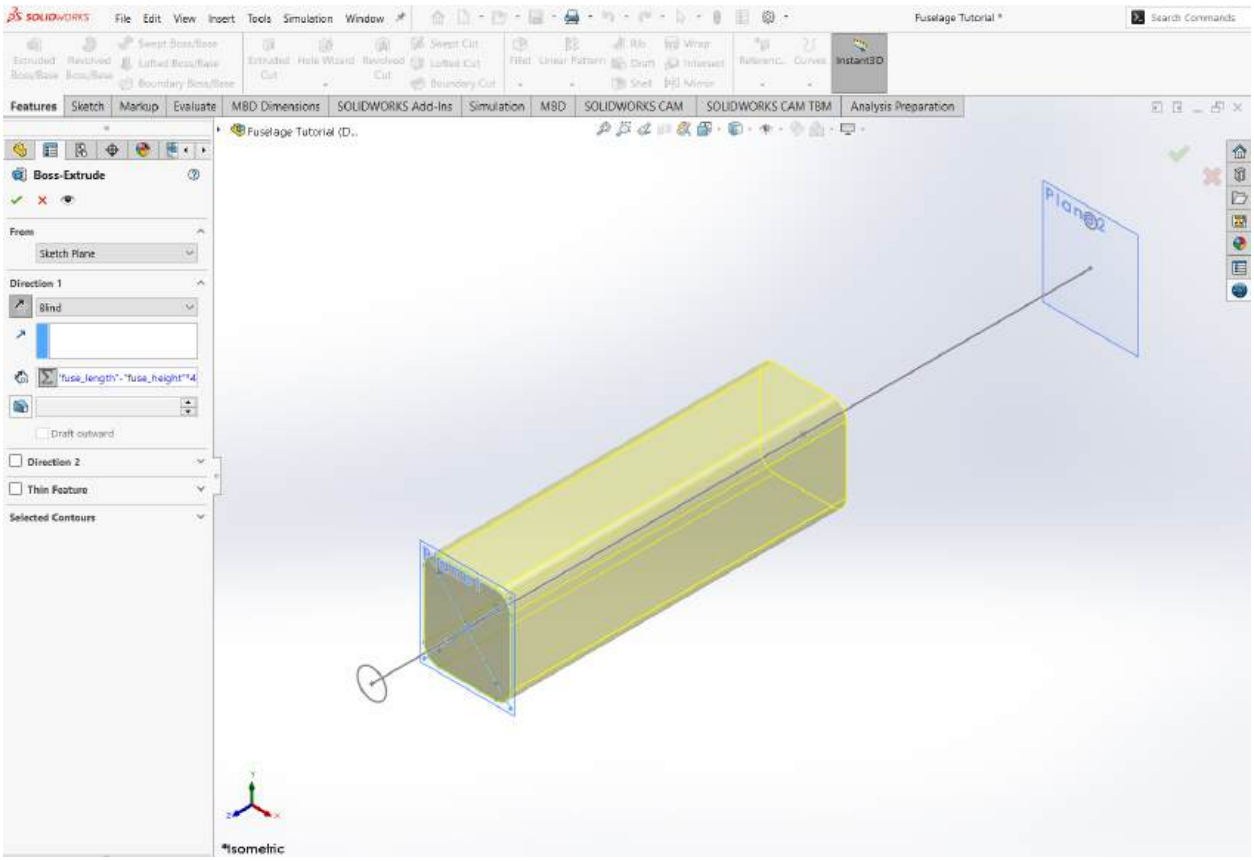
Looking great! Almost like a skeleton of the fuselage. Time to start extruding stuff!

Let’s start with that center part, since it really is a simple extrude.

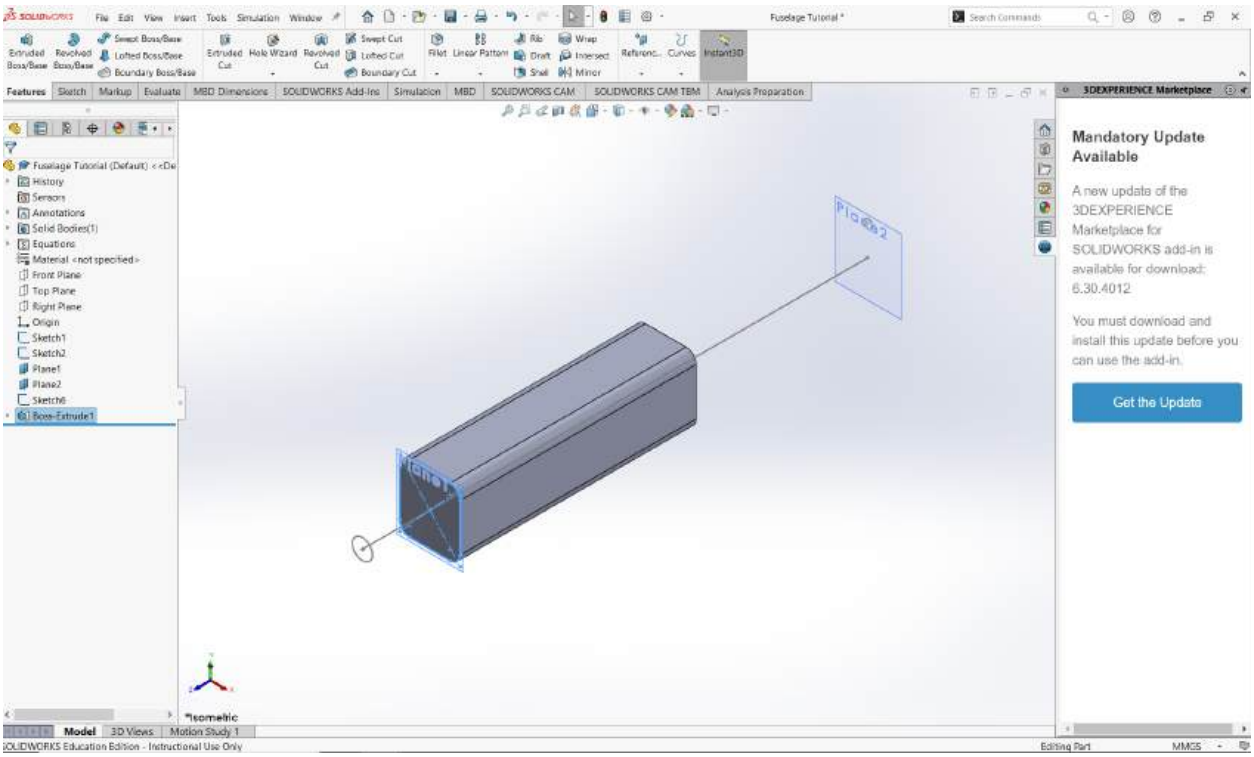
Extrusion: center part

This should be pretty easy by this point. Let’s simply extrude the center part cross-section.

Make sure to select the sketch on the tree and, under Features, click on Extrude Boss/Base. Also make sure that the direction of extrusion is correct by clicking the two arrows next to “Blind” to toggle directions. Under the extrusion amount, following the diagram at the beginning of this tutorial, the length should be “fuse_length”-”fuse_height”*4:



Looks good. Confirm the extrude and you should now see this:



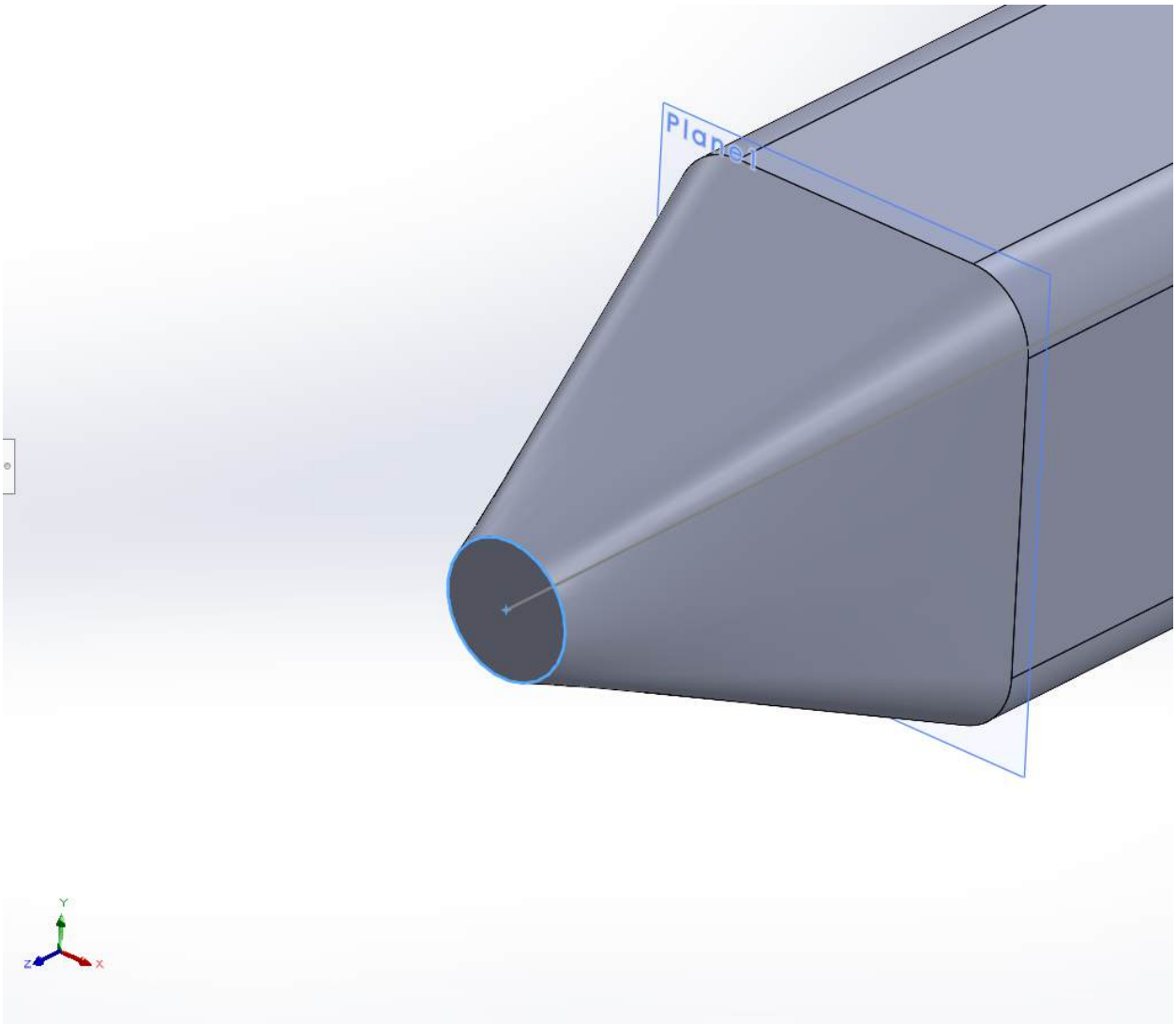
Starting to look more like a fuselage! Now, the hard part: the lofts.

Lofts: what we'll do

So far, to turn a 2D sketch into a 3D body, we have only used extrusions. We get that cross-section and expand it onto a section of a new axis. This "addition" of a new axis makes it 3D.

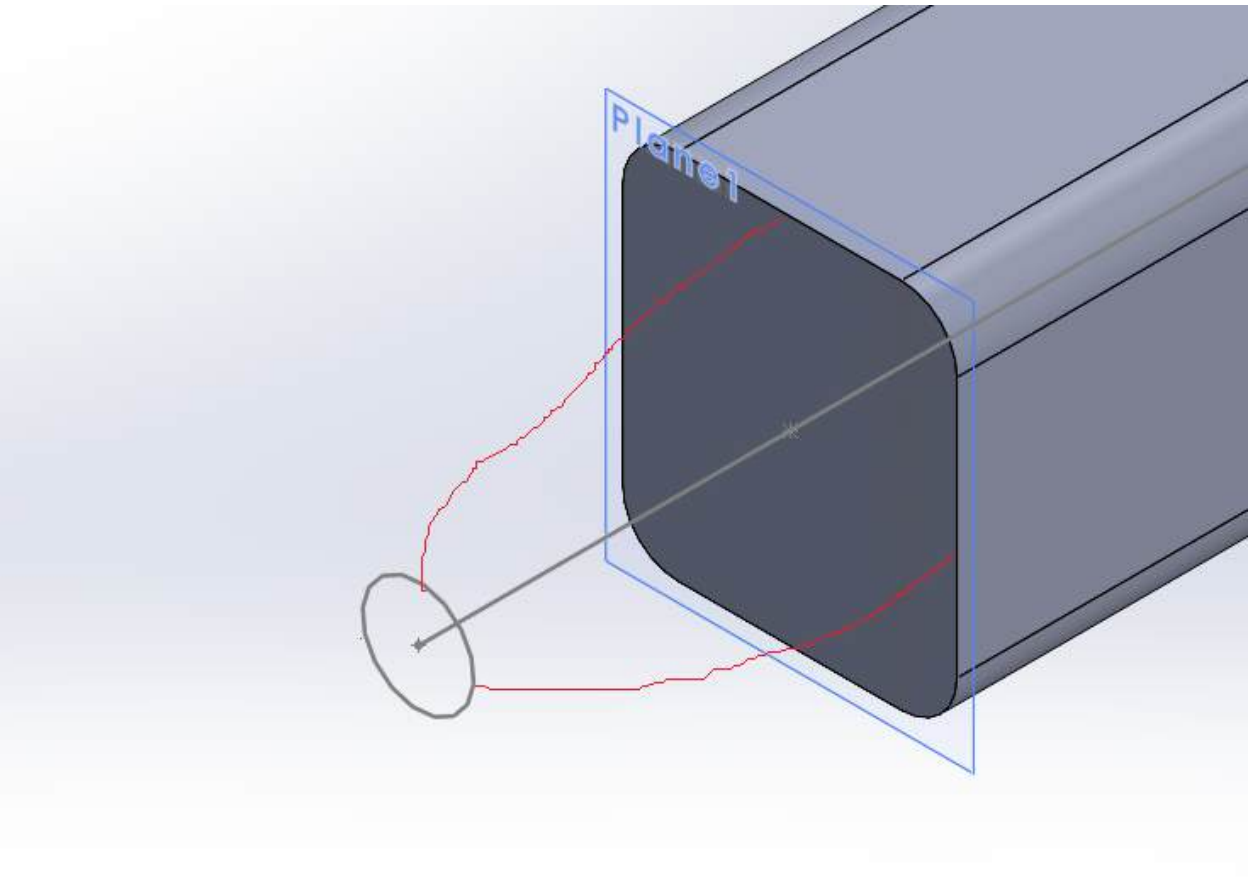
What if the cross-section changes along the axis? This must be the case for the tail-to-center and center-to-nose parts, right? We start with a circle and go to a filleted rectangle in the nose and go from a filleted rectangle to a circle in the tail.

What do we use now? We use a loft. Lofts are just that: they connect two cross sections into a 3D shape. Problem is, if you tell a loft to loft between two shapes, it will do so in the most simple way. This is what happens if we simply loft the nose and the center-piece:



Doesn’t look bad, but doesn’t really look like a fuselage nose, does it? Note how it simply draws some straight lines from points at the circumference to the rectangle. This looks too straight. Not smooth enough.

To make it a more smooth loft, we can give it guiding lines. The guiding lines we are idealizing should look something like this (and ignore my terrible artistic skills):



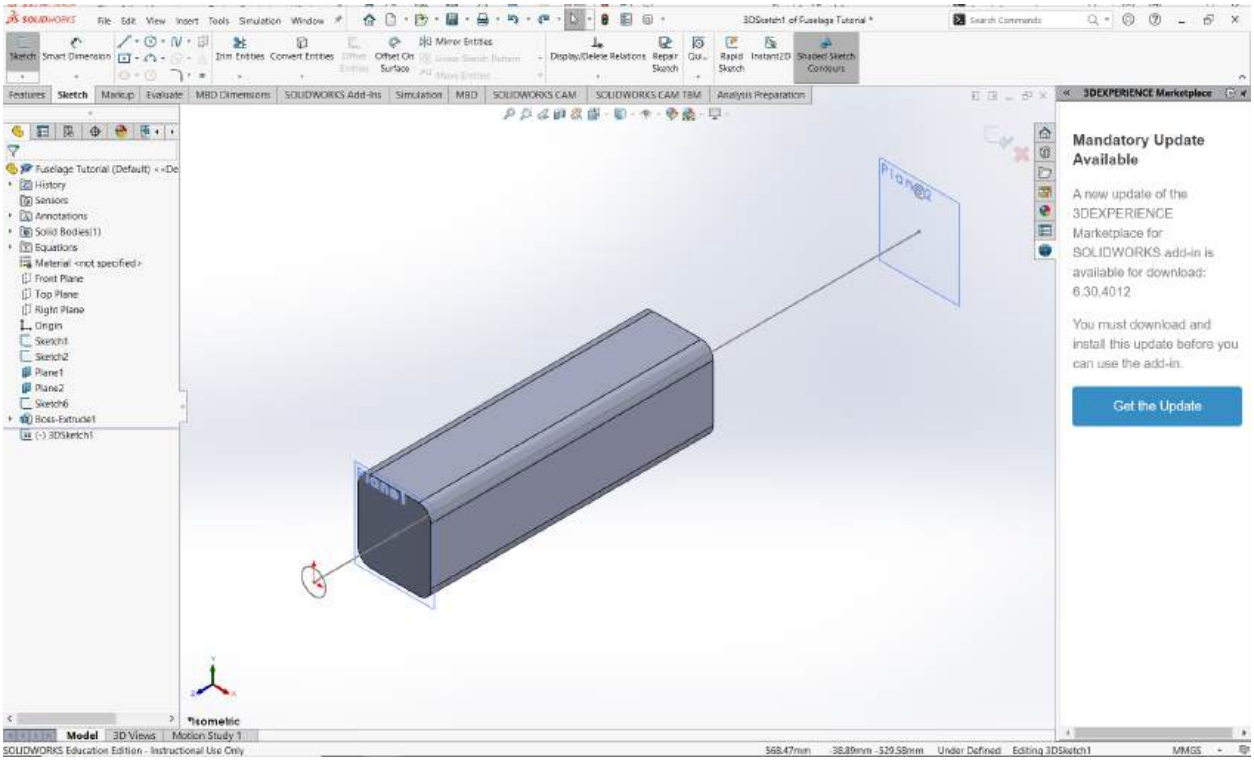
This attempt at a diagram shows the top and right guiding lines. The bottom and left should be simple reflections of these about the top and right sketch planes, respectively. How in the world do we make these though? We use something called a 3D sketch.

Nose guiding lines

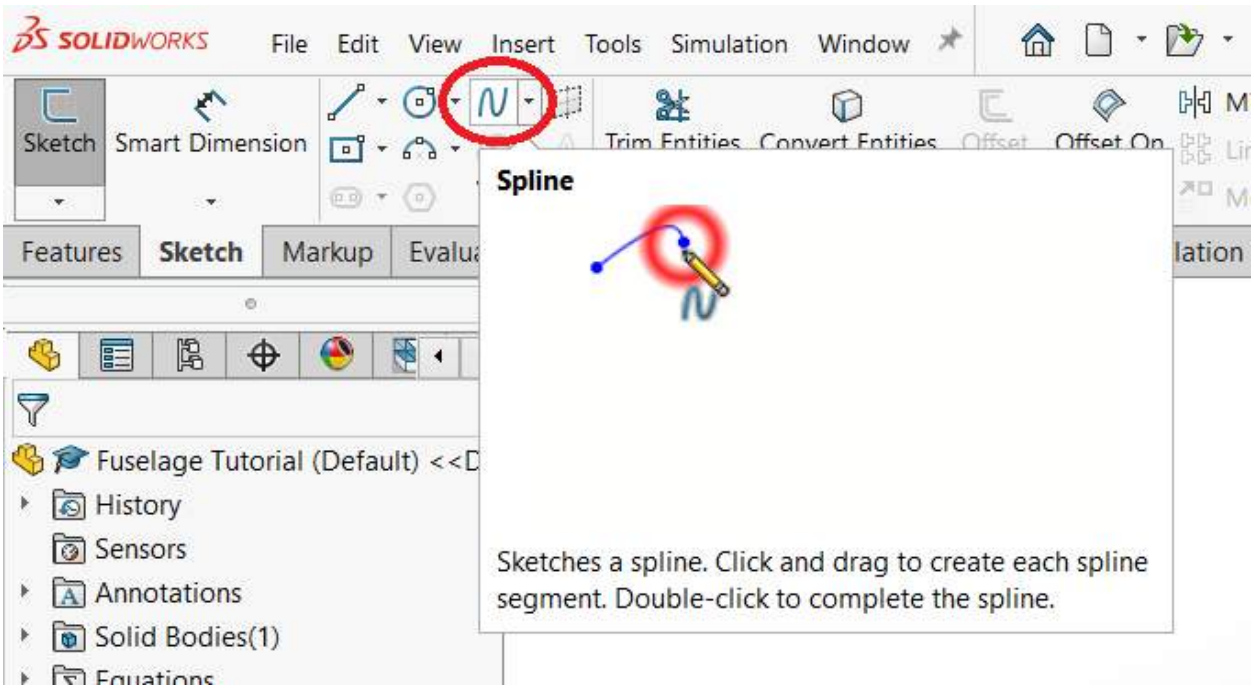
3D sketches are simply sketches that are not constrained to a single plane. Remember how we need to choose a plane to create a sketch in? That is not the case for 3D sketches.

To create a 3D sketch, go under Sketch and click the dropdown menu under Sketch:

Click 3D Sketch. You should now see this:

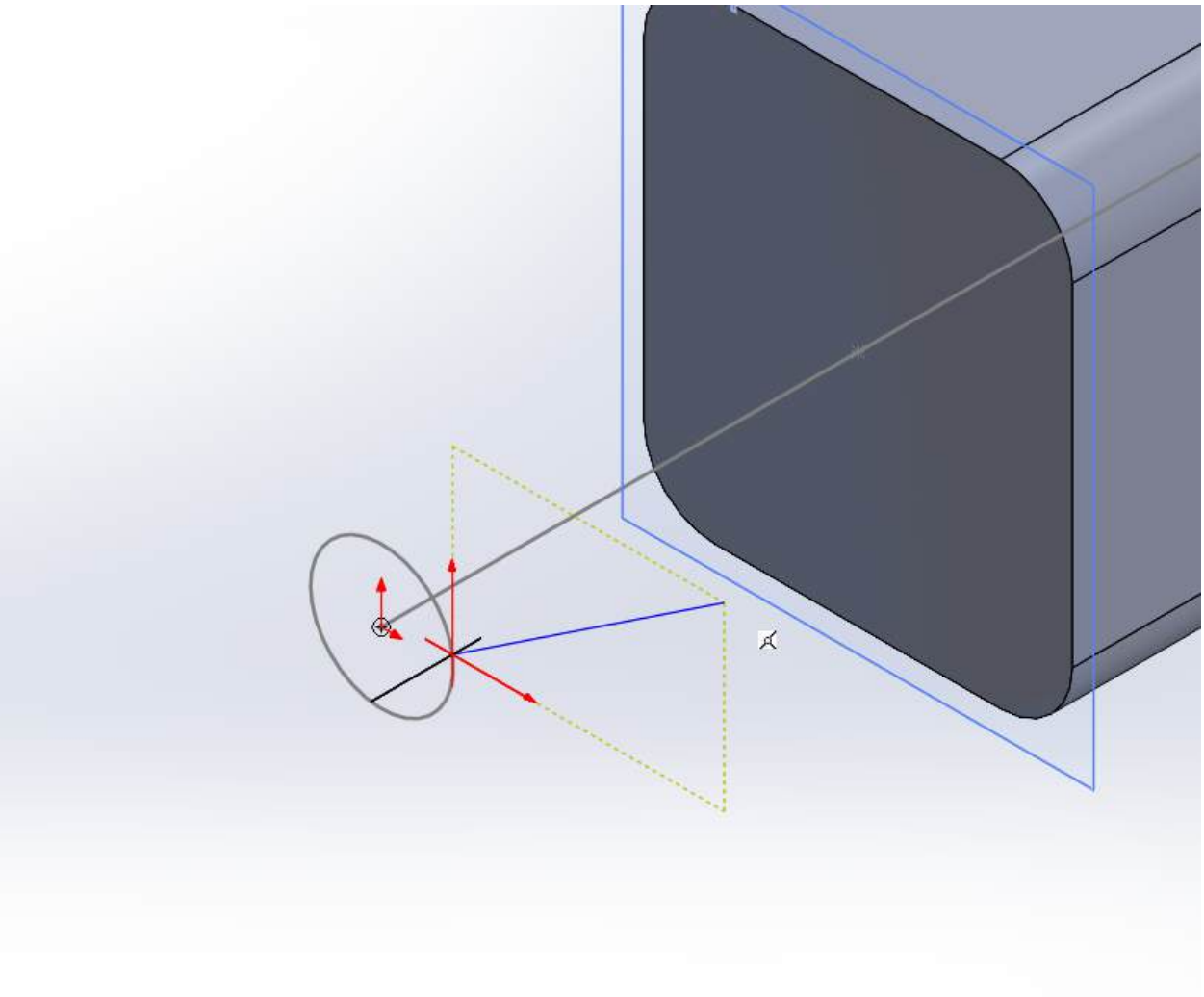


Not much changes. Notice we are now in a sketch judging by the darkened Sketch icon. We can now use all of the sketch tools that we used before. Circles, rectangles, lines, etc. For the nose guiding lines we will primarily use the Spline tool, highlighted below:

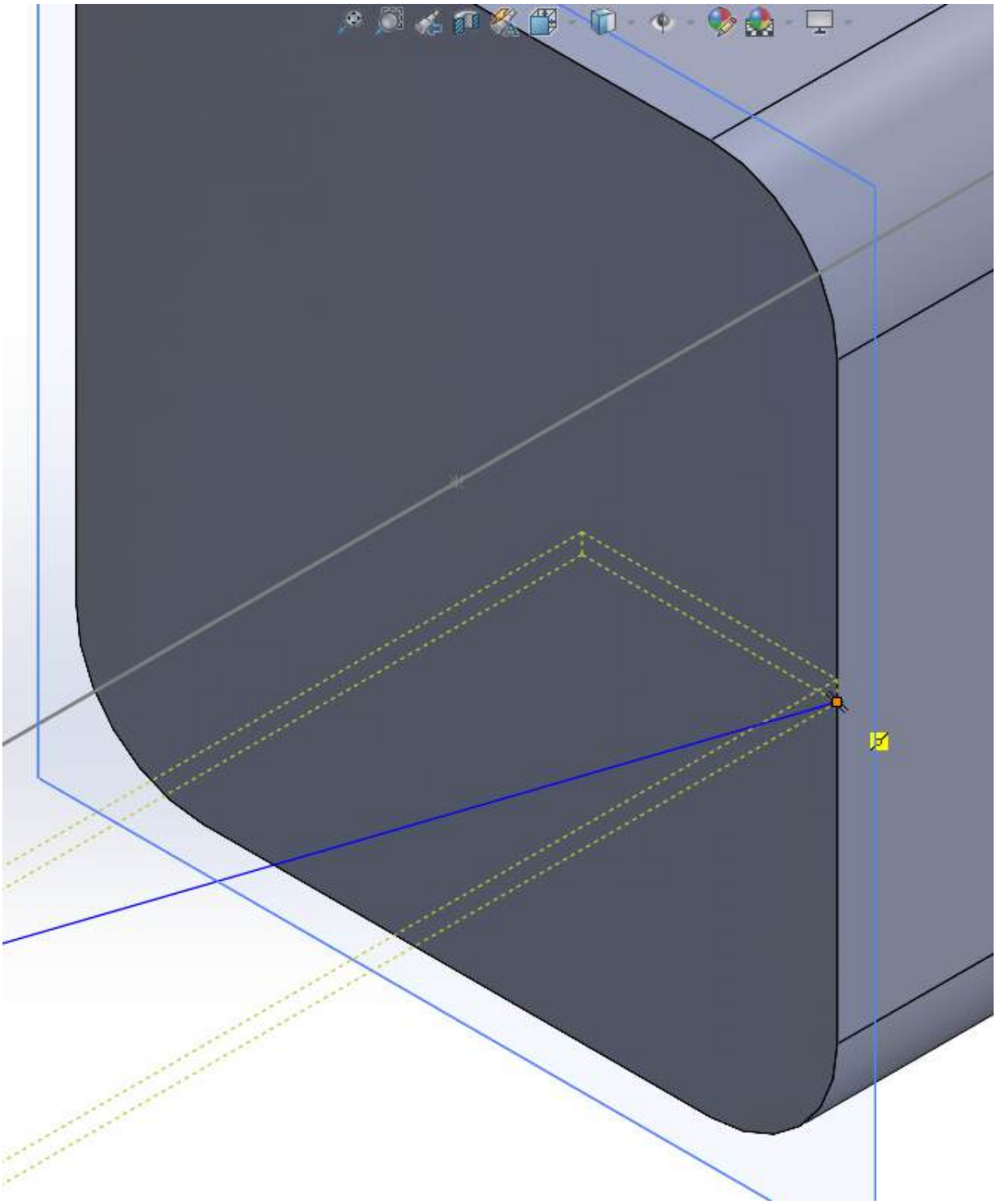


A Spline creates a curve between two points. If we click on the origin point and the target point, it will create a curve that connects the two points. We can then drag a couple of arrows to make this curve more or less curved. Perfect for the guiding lines we want. Remember that the whole point of this is to make the loft more round, less straight!

Go ahead and select the spline tool. Zoom in on the nose and center piece. First, click on the right part of the nose circle. If you move your mouse around, you should see a curve being created:

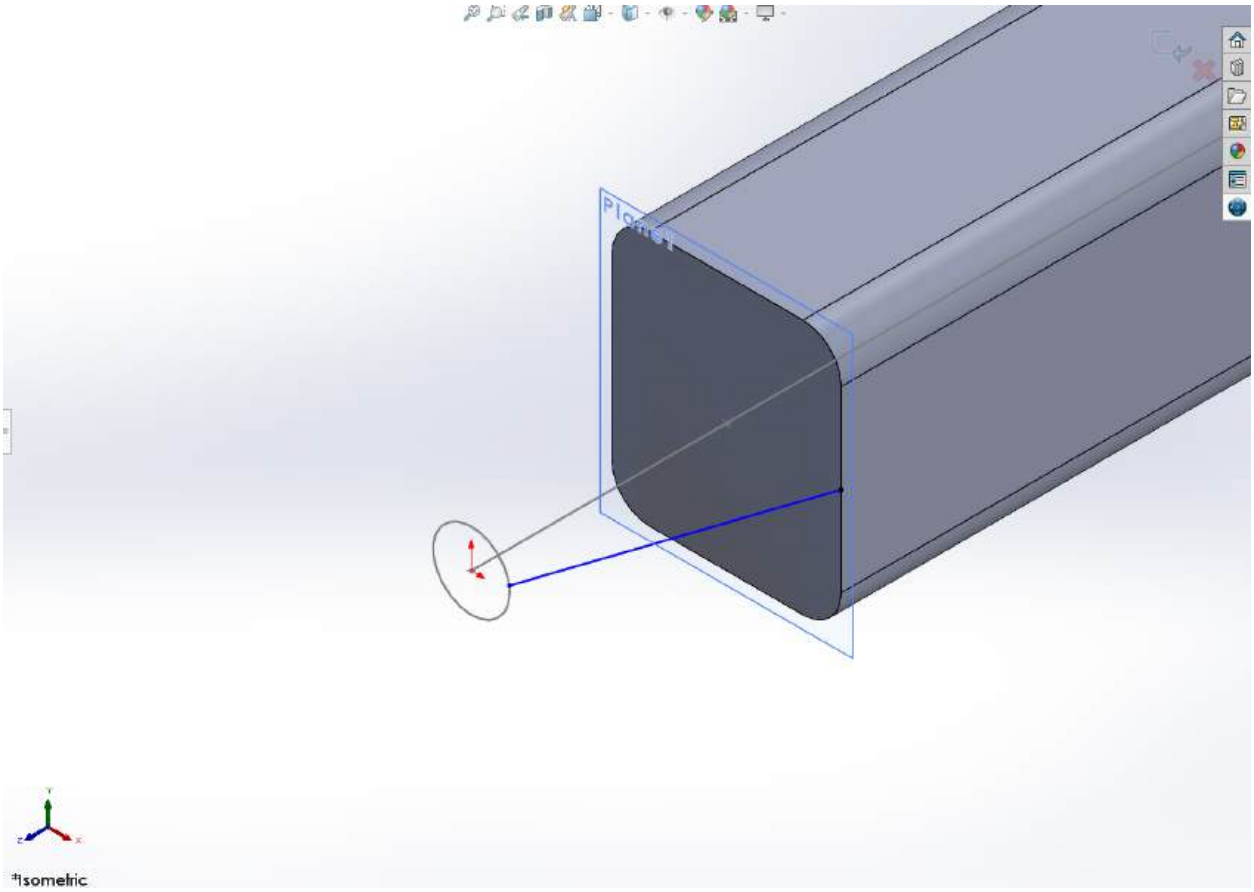


Neat. Now, connect this point to the midpoint of the right side of the center part of the fuselage. You should hover and click over this point:

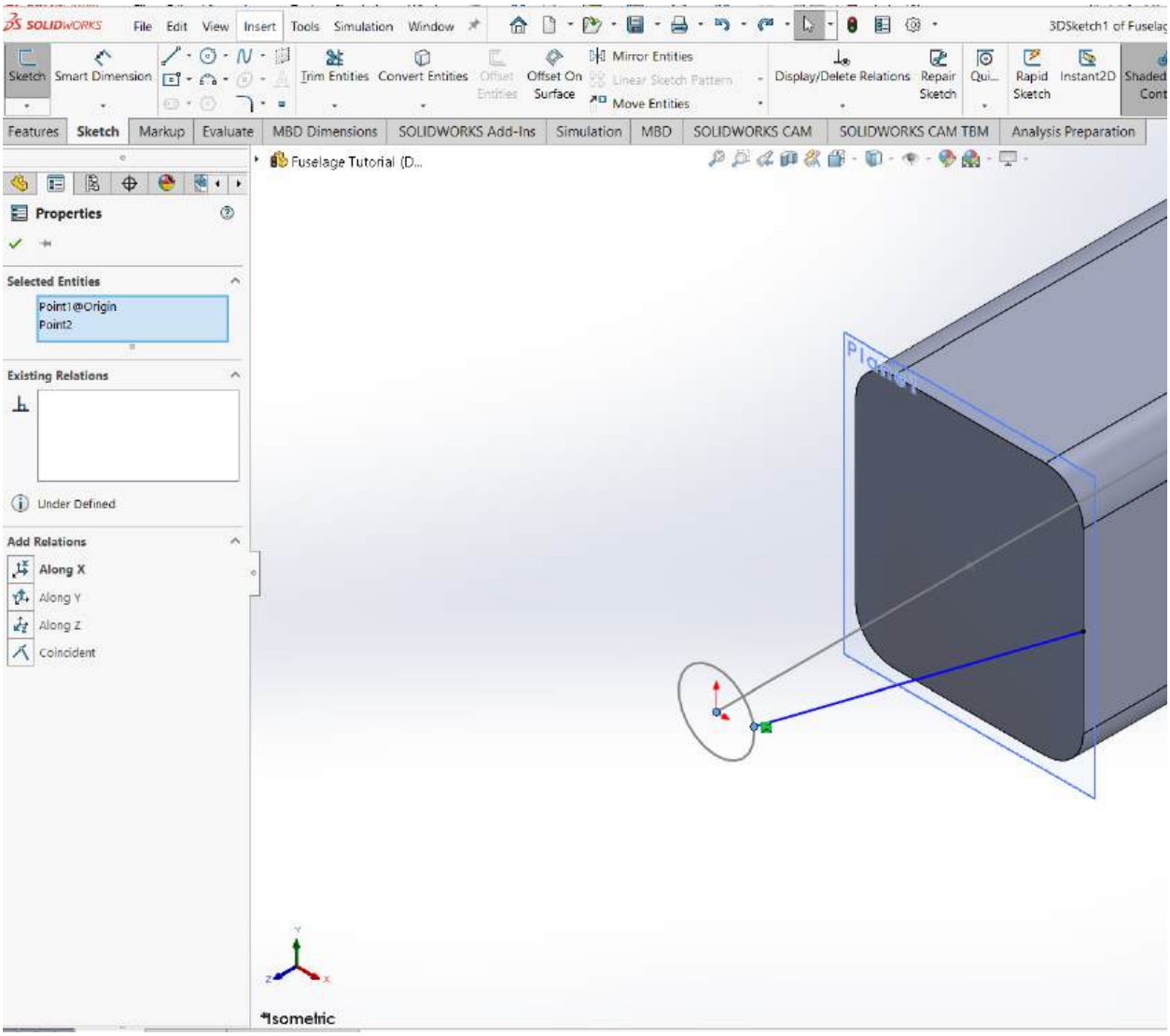


Click over that point. Because we are not making a line, it will not automatically leave the tool. The spline tool can take many points. If you give it more than two, it will use the previous point as a point in the path that connects the first point you click and the last

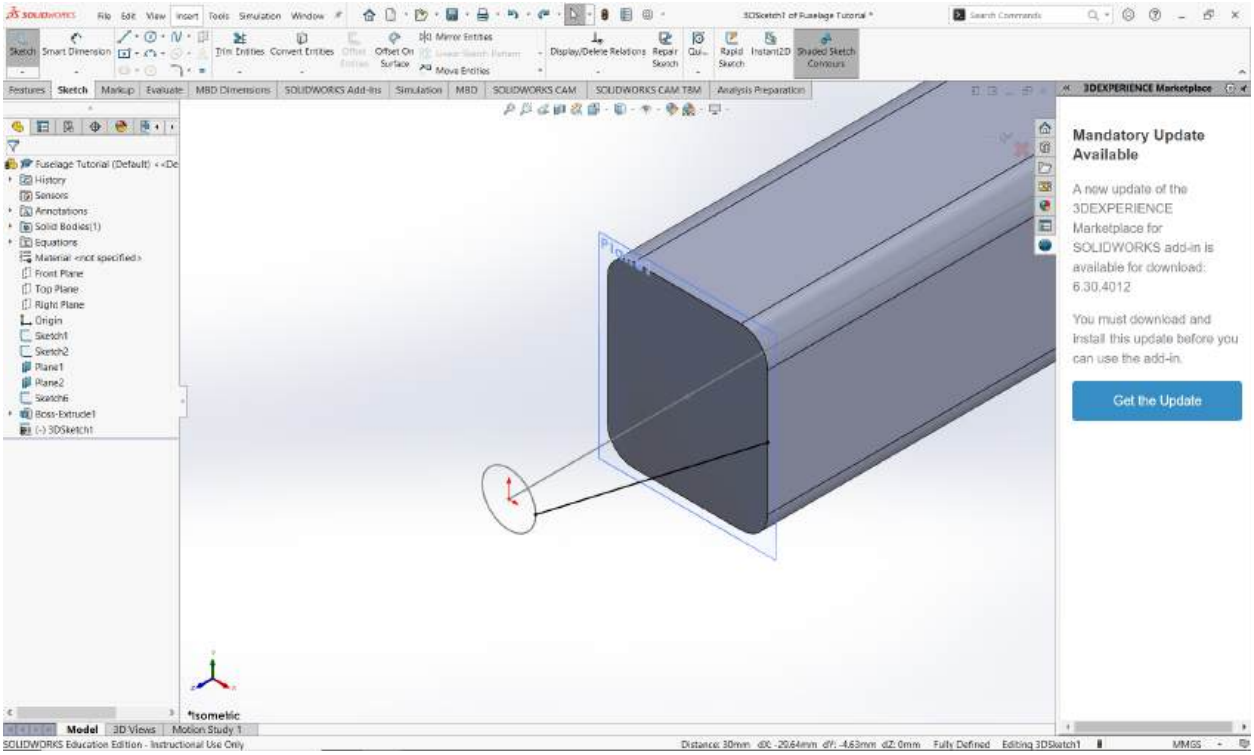
point you click. In this case, two points is enough. After you click the point in the center part of the fuselage, click escape to leave the tool. You should now see this:



Awesome! But still looks pretty straight, right? We will soon change this. First, notice how the yellow dotted lines when creating the spline indicate that the two points are not in the same xz-plane (also, small note, I will denote any plane parallel to the xz-plane as the xz-plane. I'm an engineering student, so I can do this). To change this, we need to make the point that lies on the nose circle line up with the origin. To do this, click the two points. The left menu should show this:

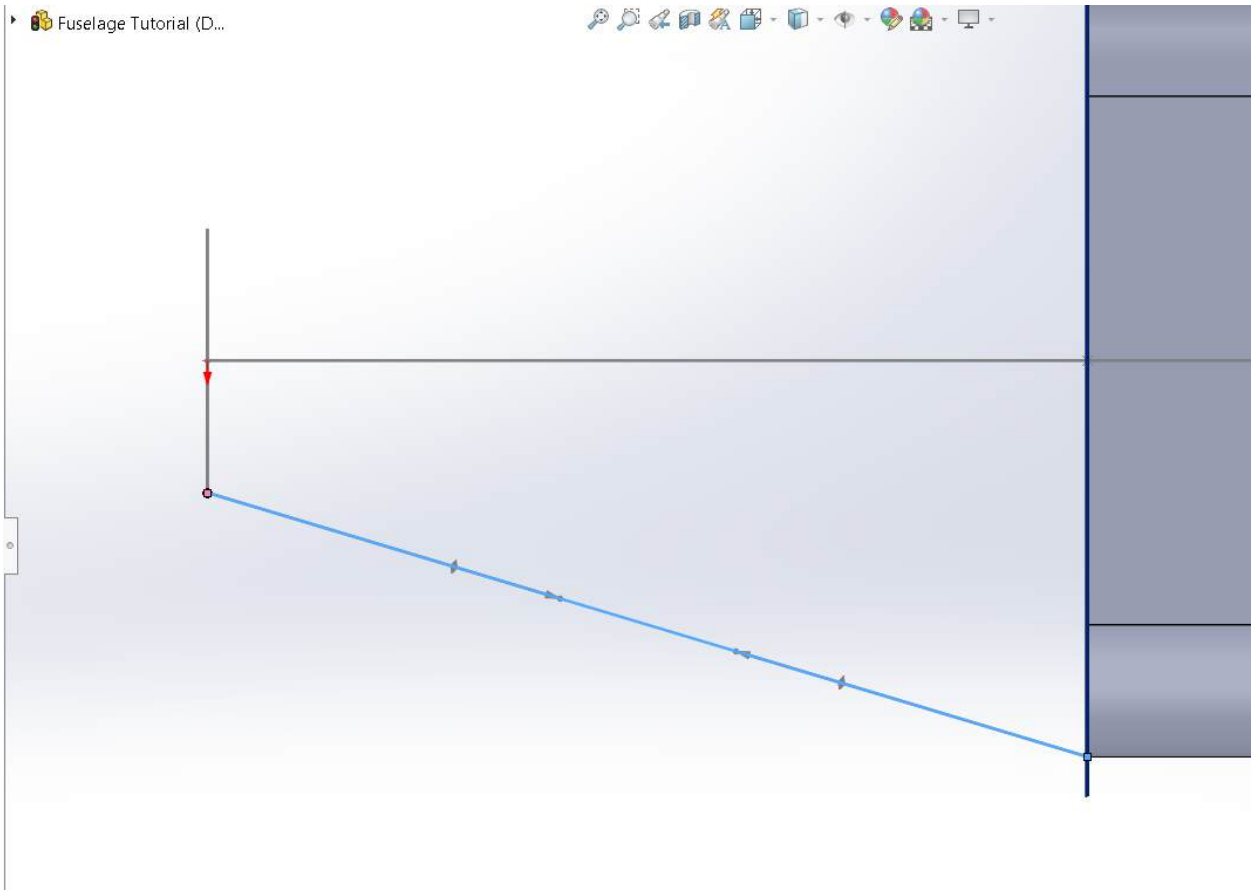


Because we are now in 3D, we no longer have those convenient “parallel,” “vertical,” “horizontal,” etc. This is because we have no fixed plane as reference now. We must constrain things along different axes. I will tell you the axes, but it should become pretty intuitive pretty fast. Take a guess at the axis that we will constrain these two points to. That’s right! The x-axis! Go ahead and click “along X” and confirm. You should see the points line up:

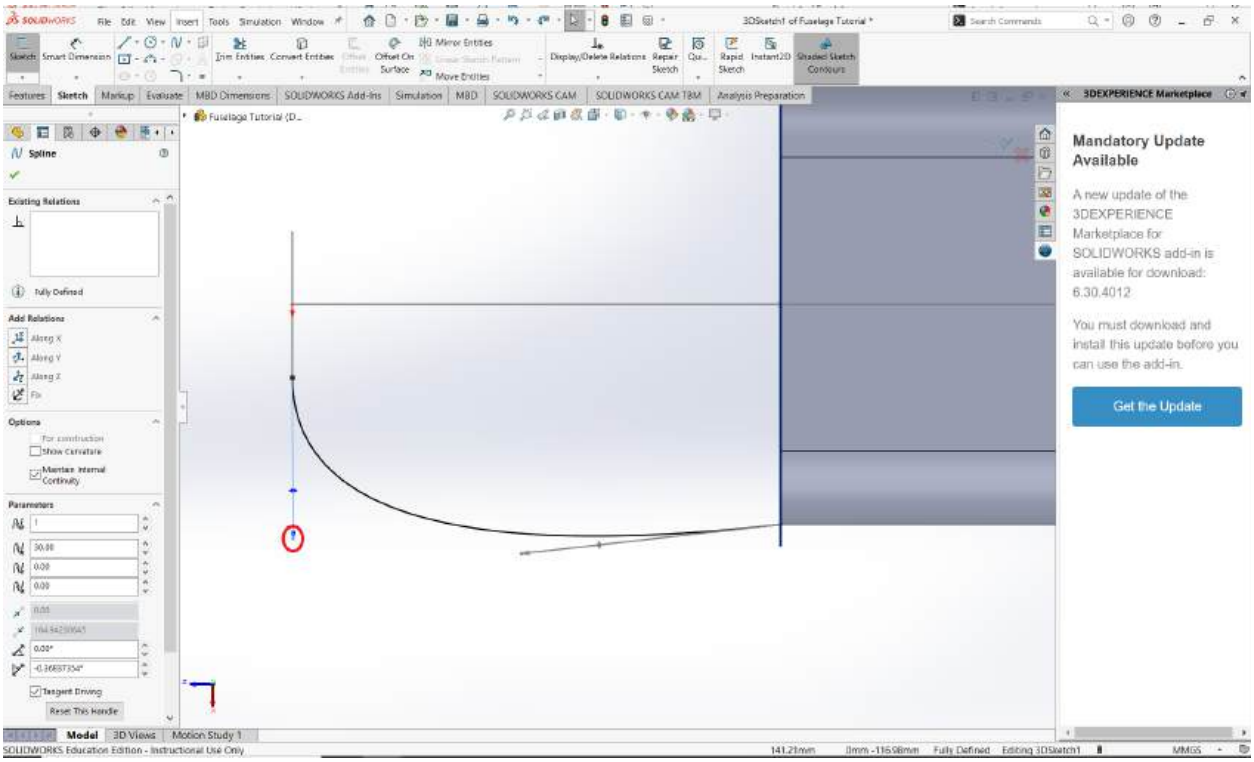


Looks good! We will be doing a lot of the “Along [axis]” constraints in the next few pages, so get used to it!

Now, time to make this thing more round. If you click on top of this curve we just drew, and switch to the top view for convenience, you should see this:



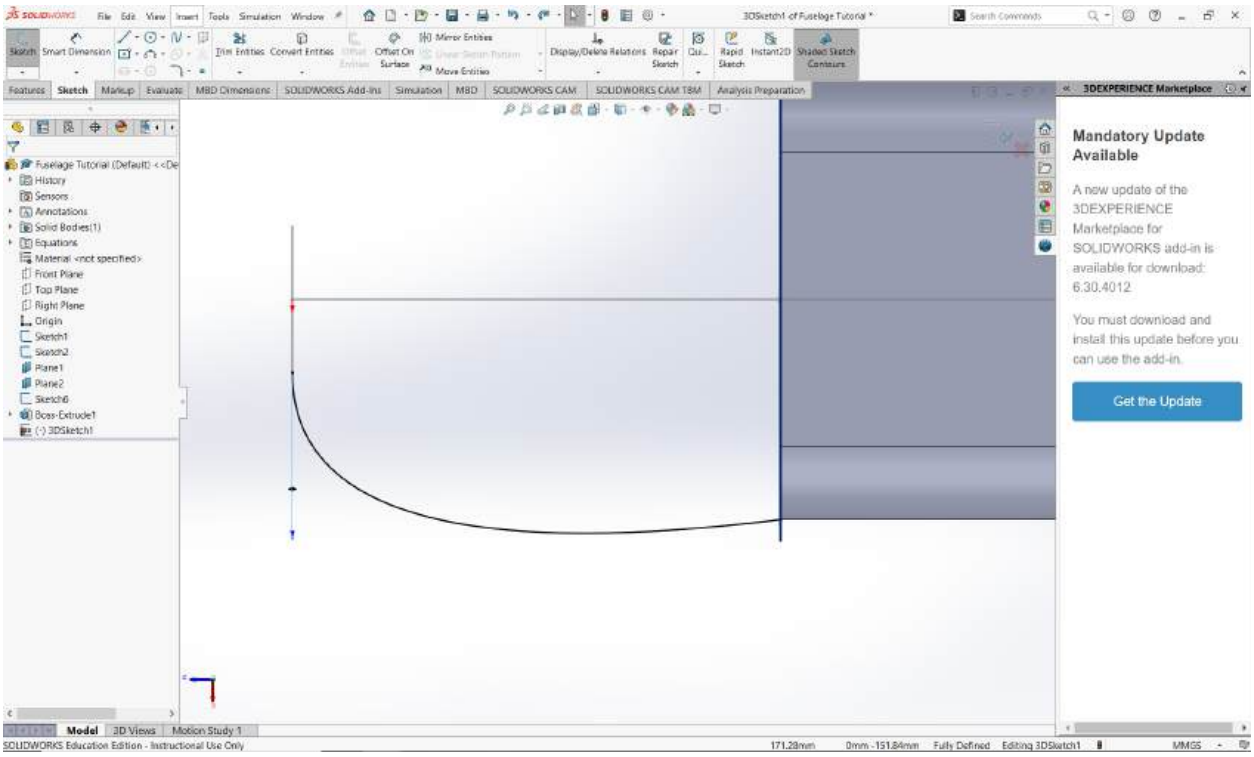
Notice the two arrows, one that goes from the nose to the center part and one that goes from the center part to the nose. For now, I am more interested in the arrow from the nose to the center. If you click the point highlighted below and drag it around, you will see how useful the spline is:



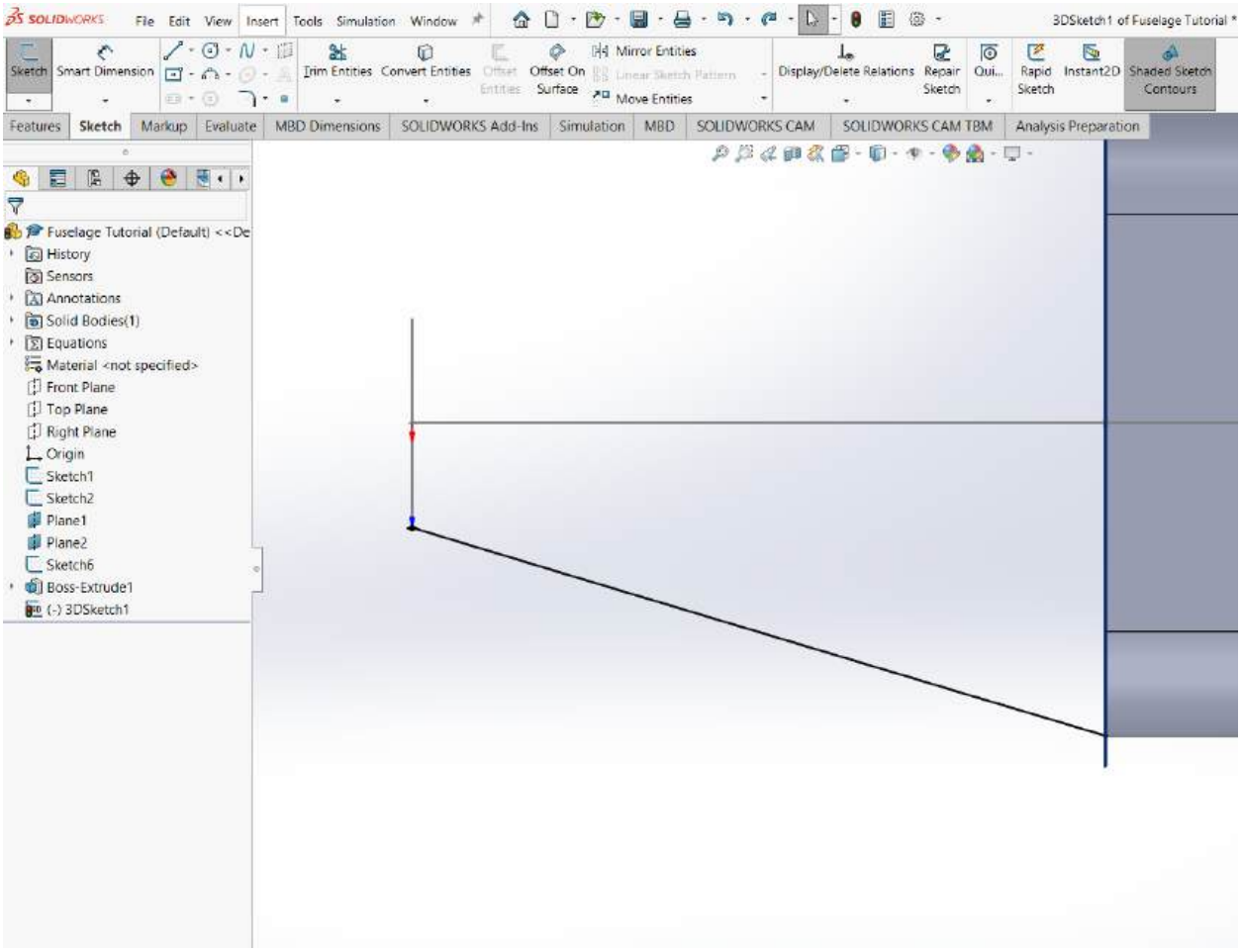
This round nature makes it look much more like what it should!

One small caveat: for reasons that I will not elaborate too much on this tutorial, this arrow that we are moving must be constrained to the X axis and it should be dragged to be as small as possible. This has to do with the way we want the loft to transition close to the nose circle. It should be as round as possible as close to the nose as possible, so the transition close to the nose must be sharp. To make it sharp, we make the arrows coming out of the nose very very small.

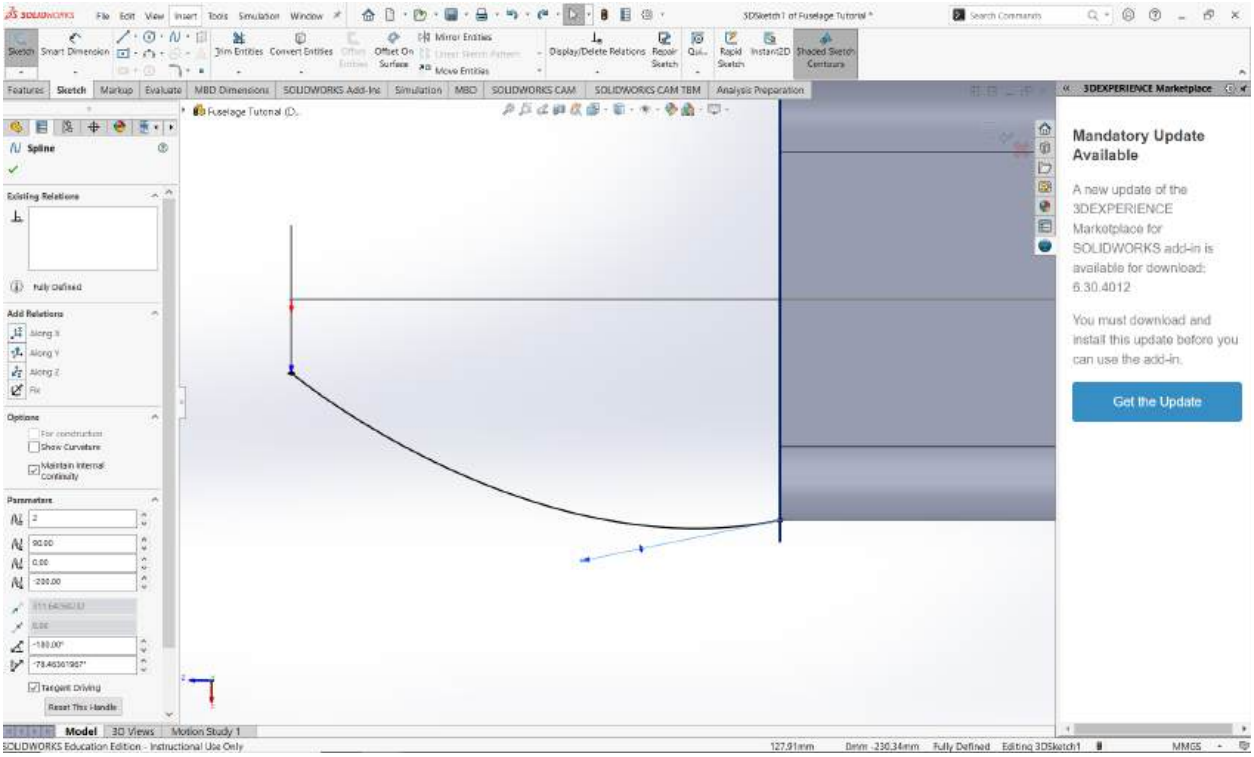
Therefore, on the menu on the left, select “Along X” once again and confirm. You should see this:



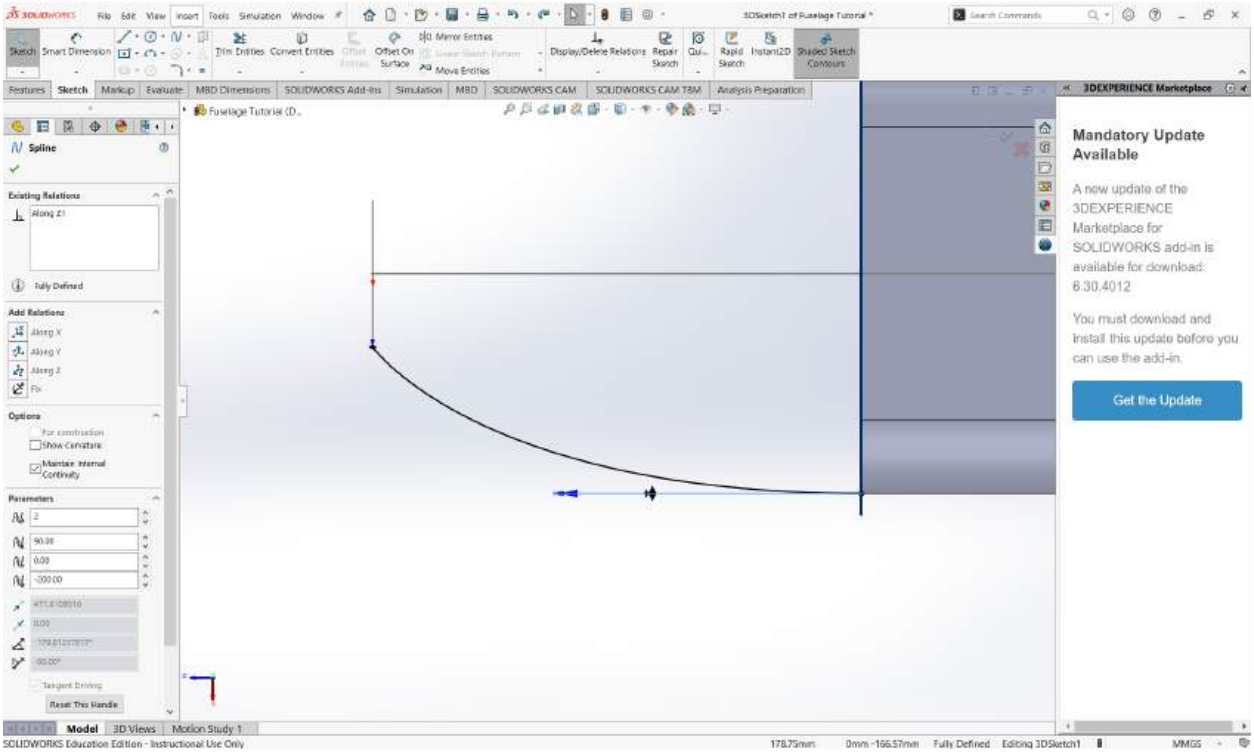
Now, drag that arrow to be as small as possible, something like this:



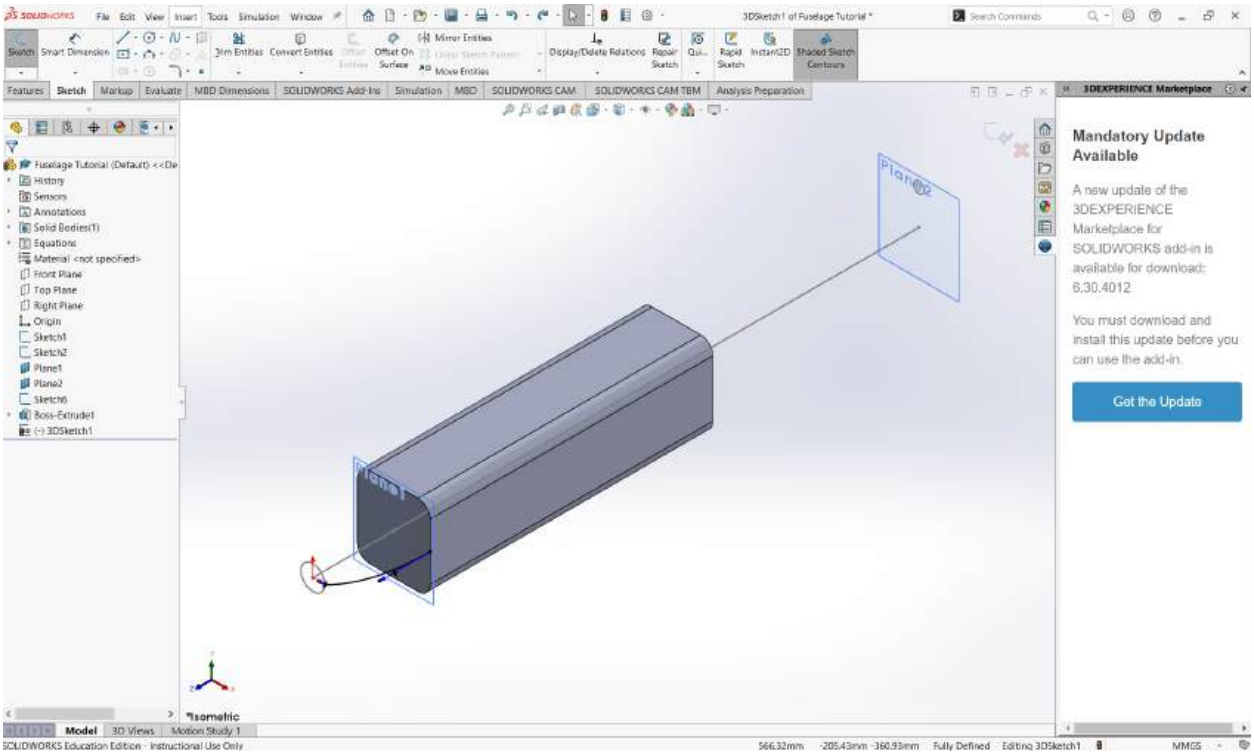
It is back to being a line, but we'll change this now. Click on the other arrow, the one that goes from the center part to the nose, and drag it around a bit:



This arrow needs to be constrained to the Z axis, so click “Along Z” on the menu on the left:



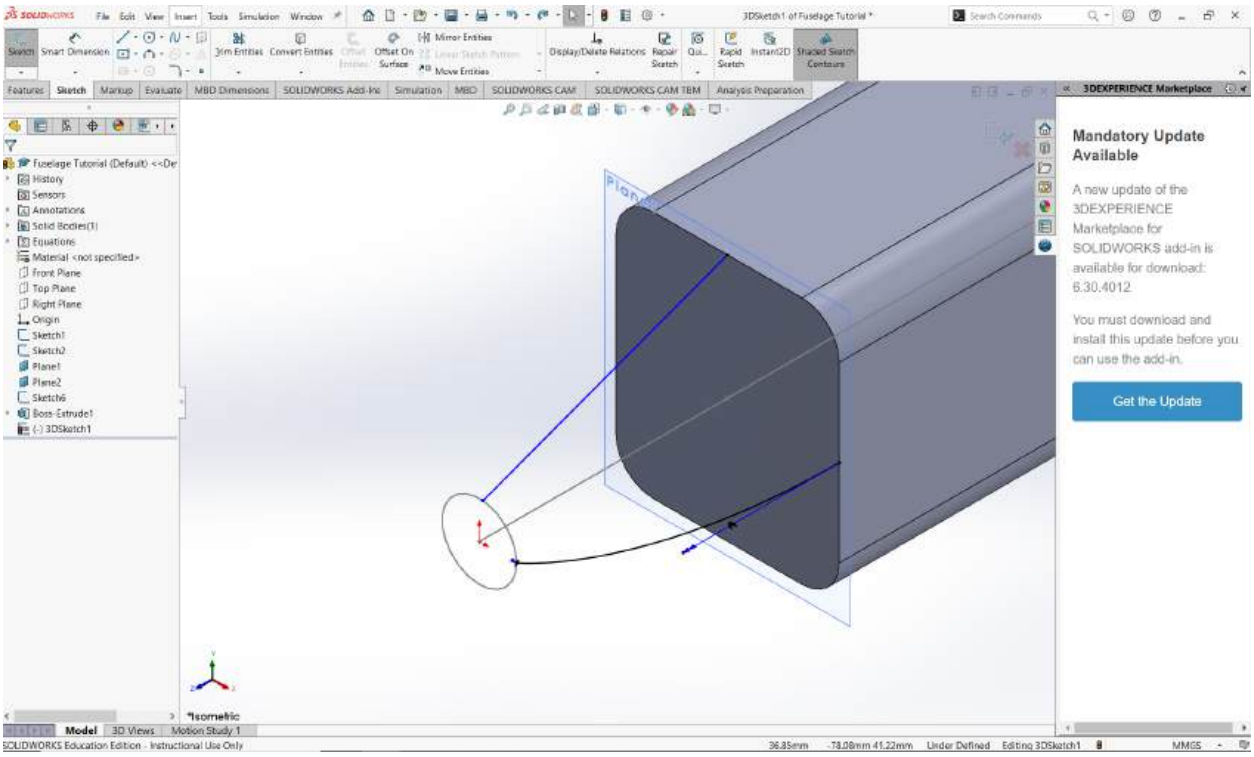
Now, if you drag the arrow around, you will notice that it only moves on the Z-axis, as it should. Move it to a spot where it just looks right. There's a bit of artistic freedom here once again. Once it looks pretty good, confirm with the green tick. Going back to isometric, this is roughly what it should look like:



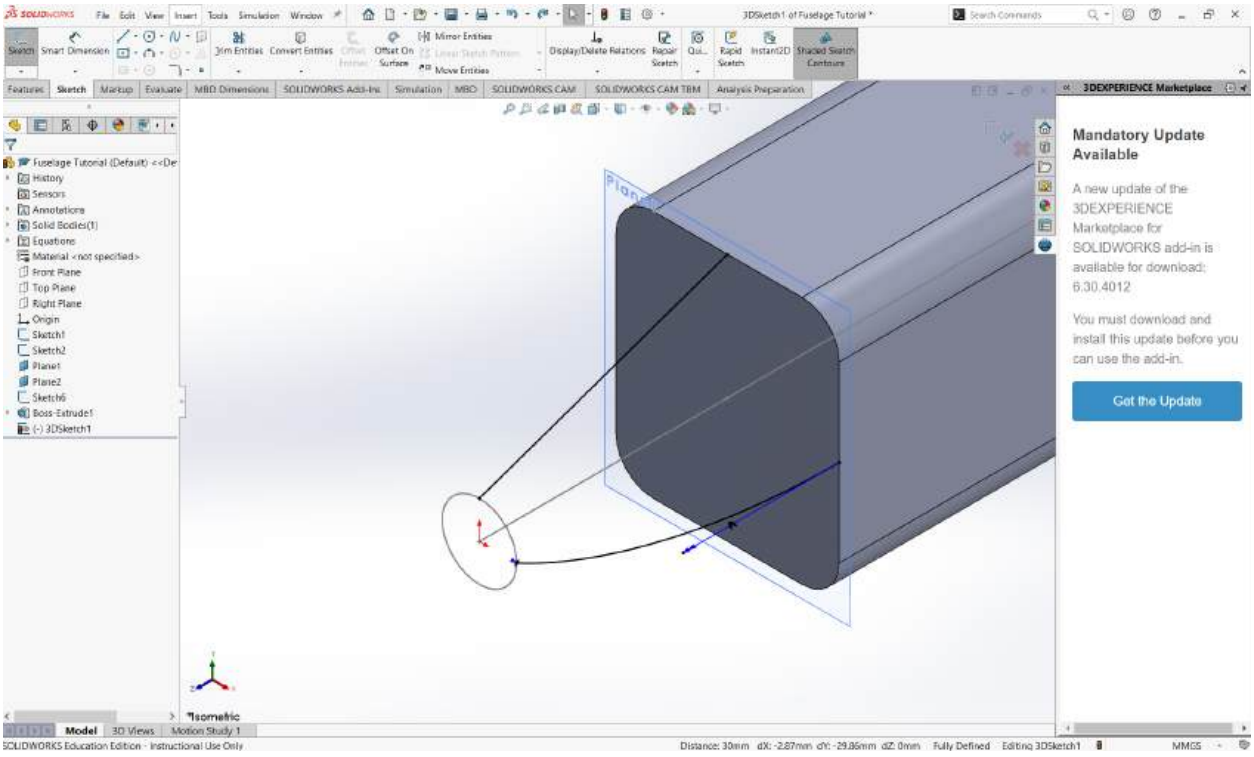
Looking good!

Now, we must do the same but for the guiding line that connects the top of the circle to the top side of the center part. I will move through this a bit faster now. The steps are basically the same, but the constraints are along different axes for some of these.

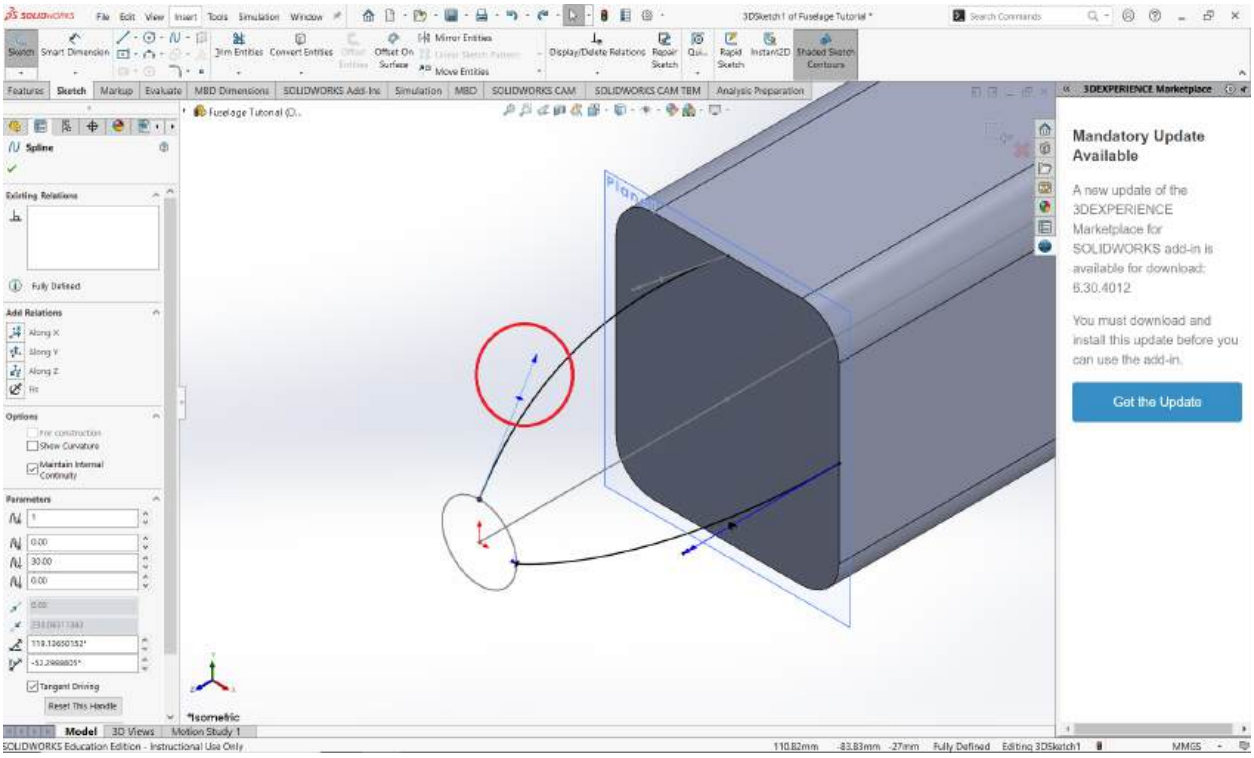
First, create the spline between the circle and the midpoint of the top side of the center part. Be very careful to select the midpoint. That's very important. You should see something like this:



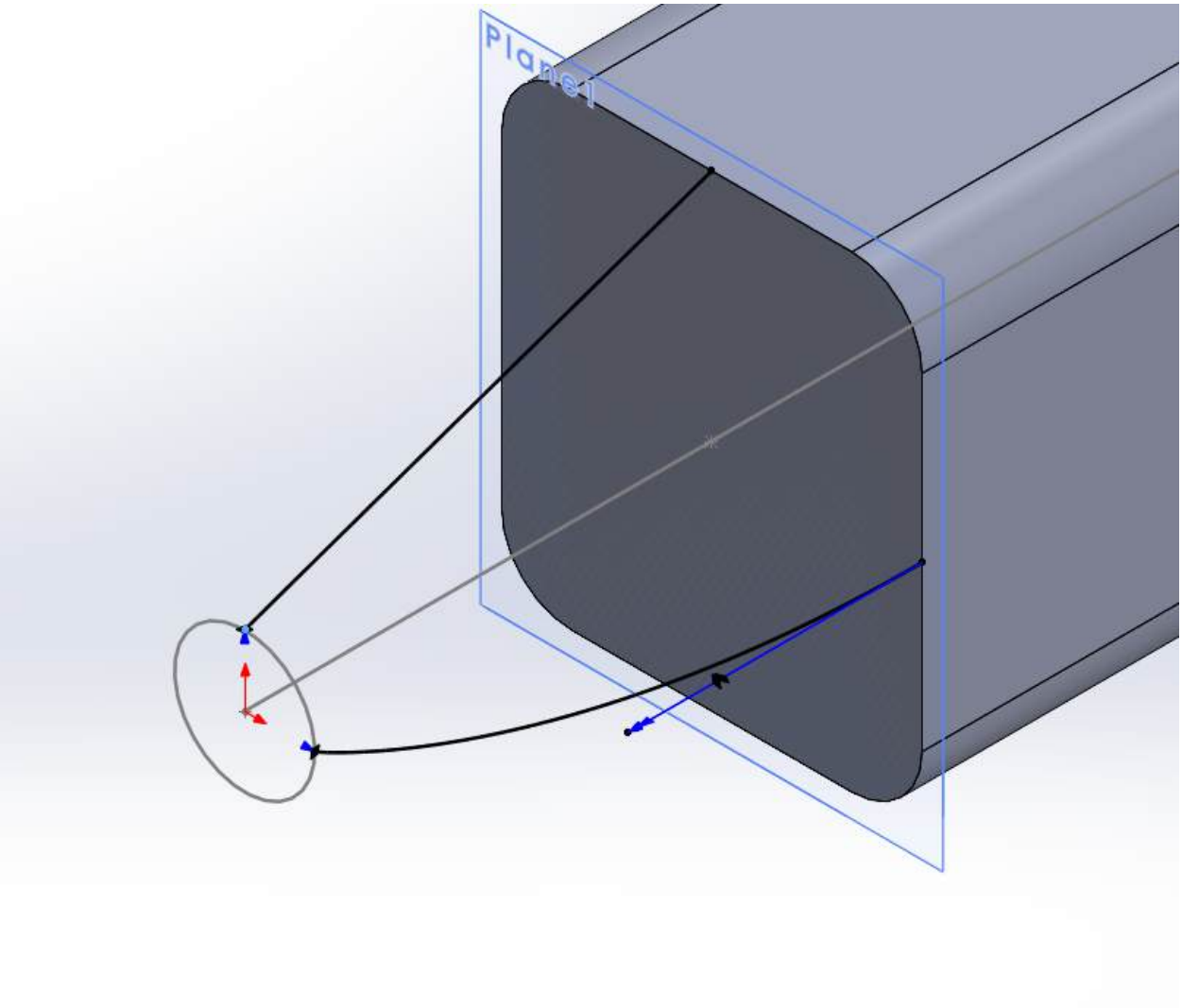
Let’s line up the point in the circle with the origin. Now, we must constrain it along the Y-axis. This is somewhat intuitive. Notice how the line that should connect the origin and that point should lie on the y-axis. After constraining them, you should have this:



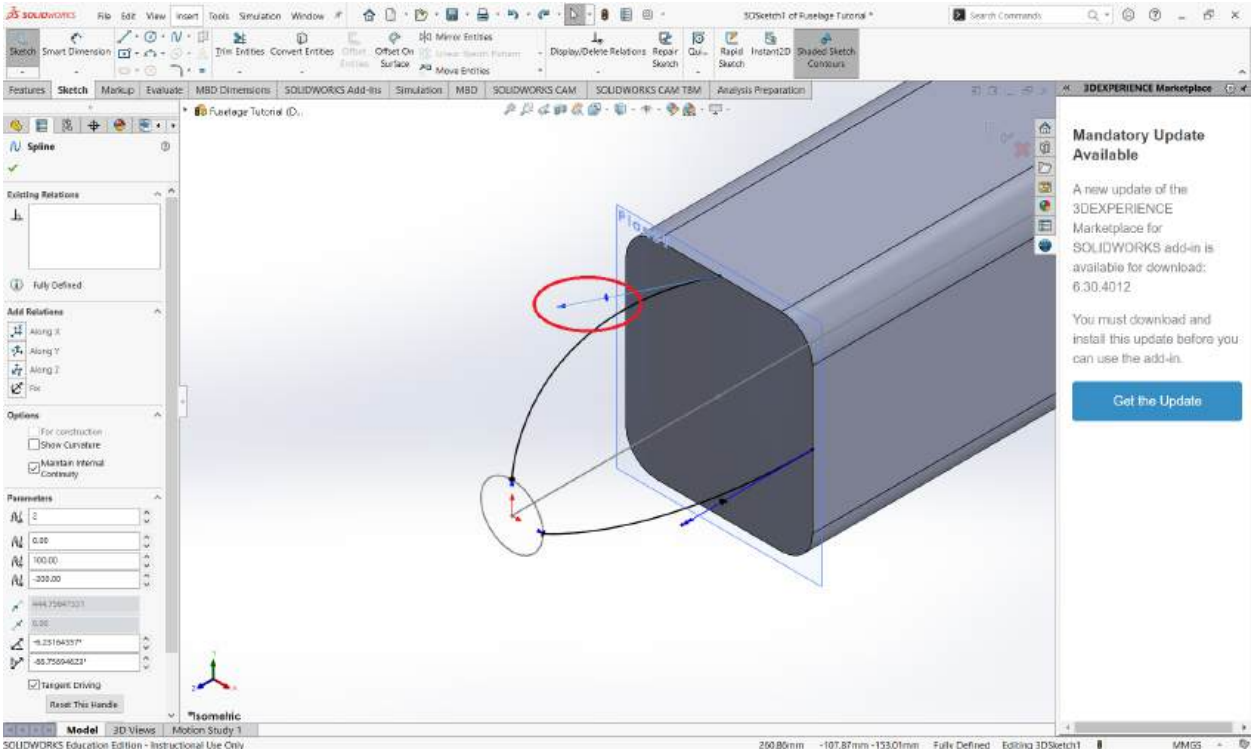
Click on the curve we drew. Select the arrow that goes from the nose to the fuselage and drag it around:



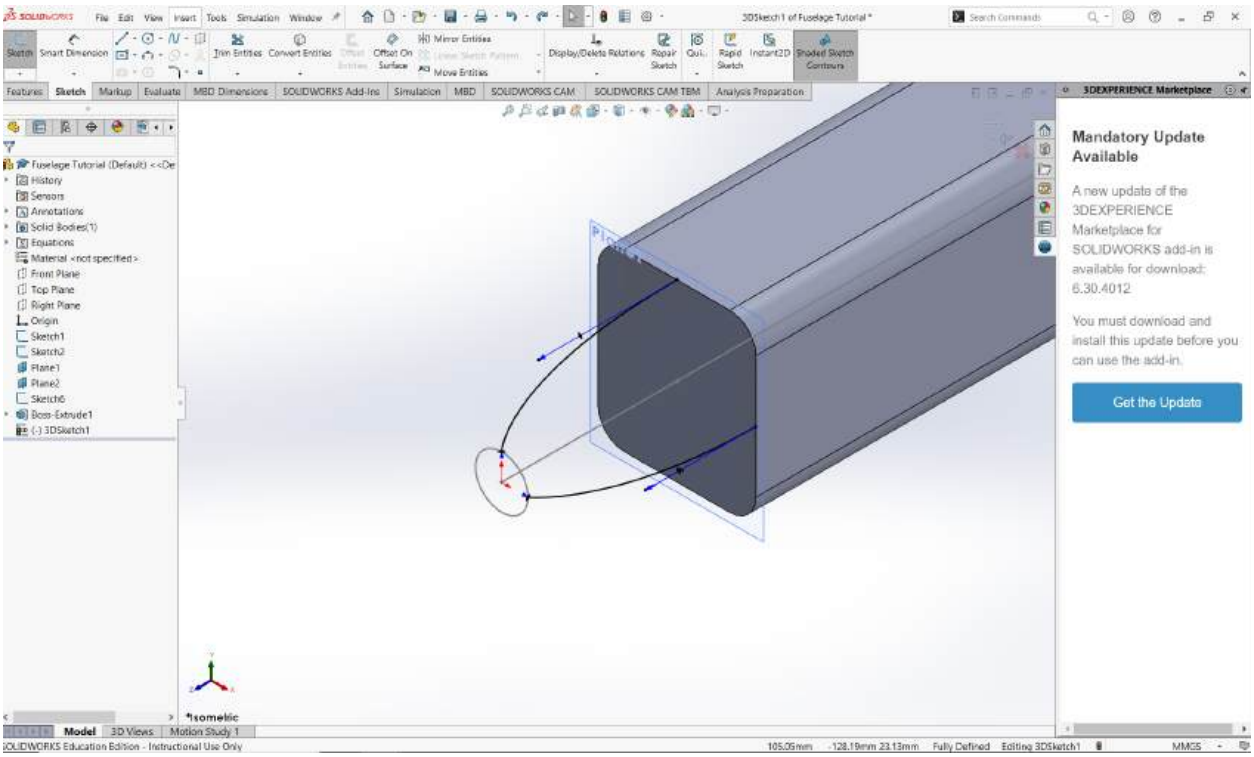
This one should be constrained along y, so select “Along Y” on the left menu. Then, drag it as small as possible, just like we did before. You should see something like this:



Let’s work on the other arrow. Click on it, and drag it around:

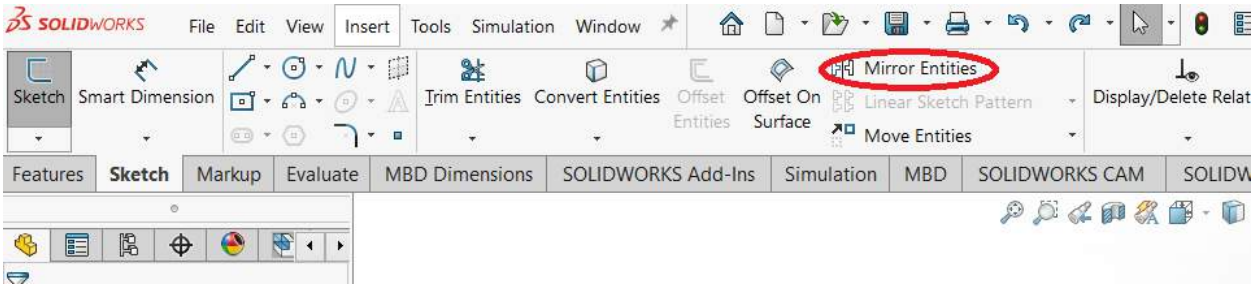


This one should be constrained along the z-axis, so select “Along Z” on the menu. Drag it until it kind of looks right (artistic freedom again!) and confirm. You should now see this:

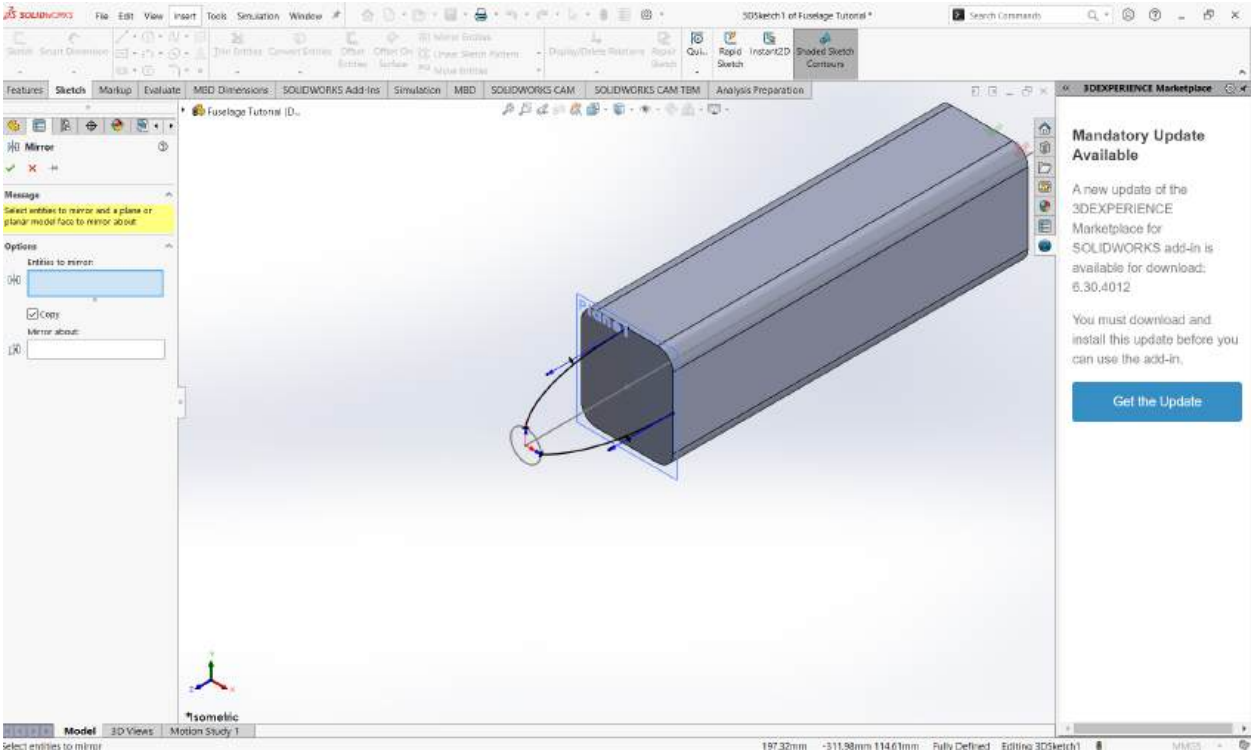


Great! Last step on the nose before lofting: let's mirror these about the right plane and top plane.

Still under Sketch, select Mirror Entities:

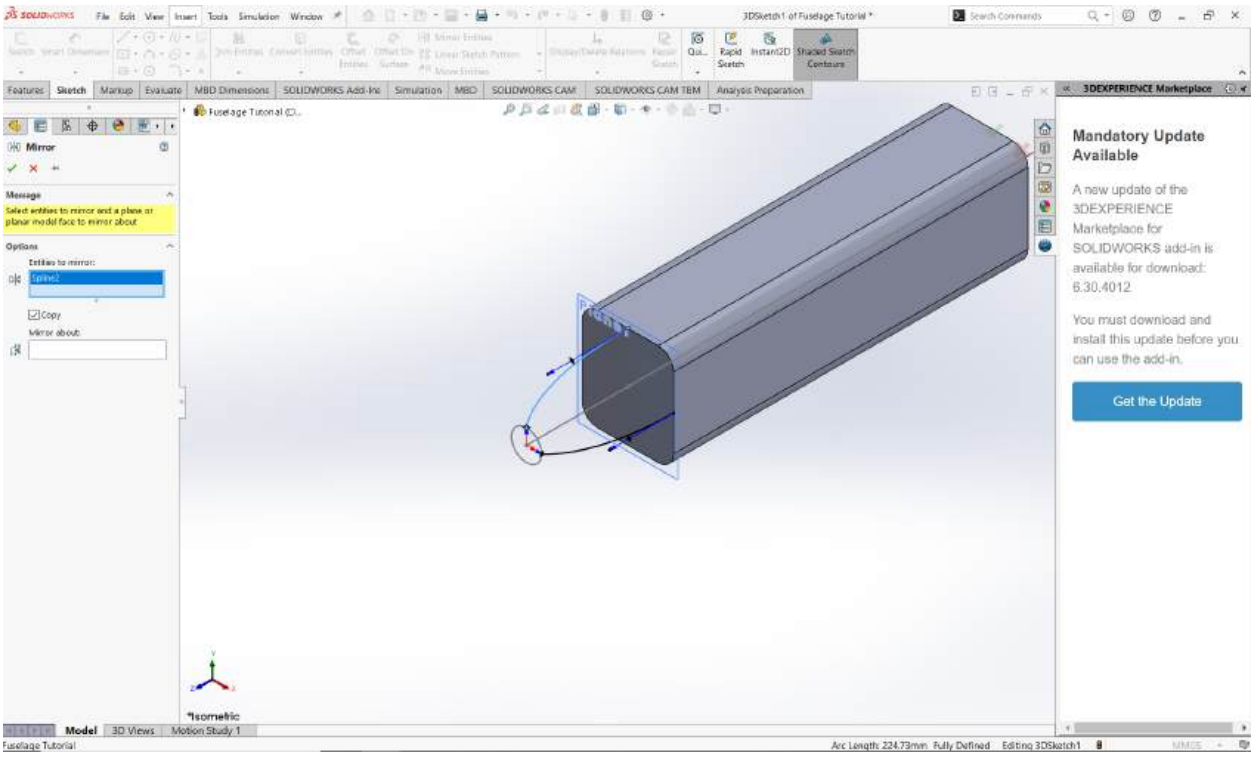


You should now see this:

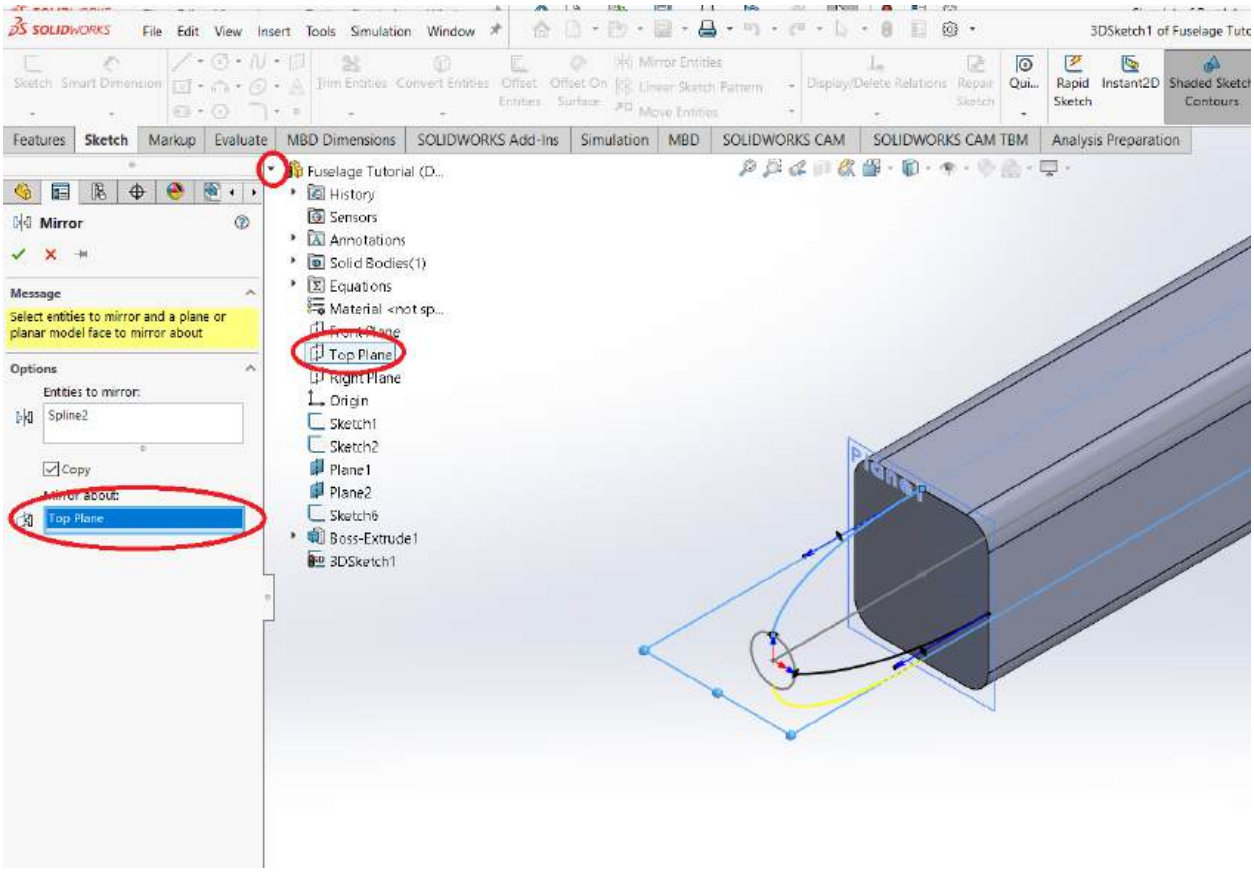


It is asking us to select the entities to mirror and then the plane to mirror about.

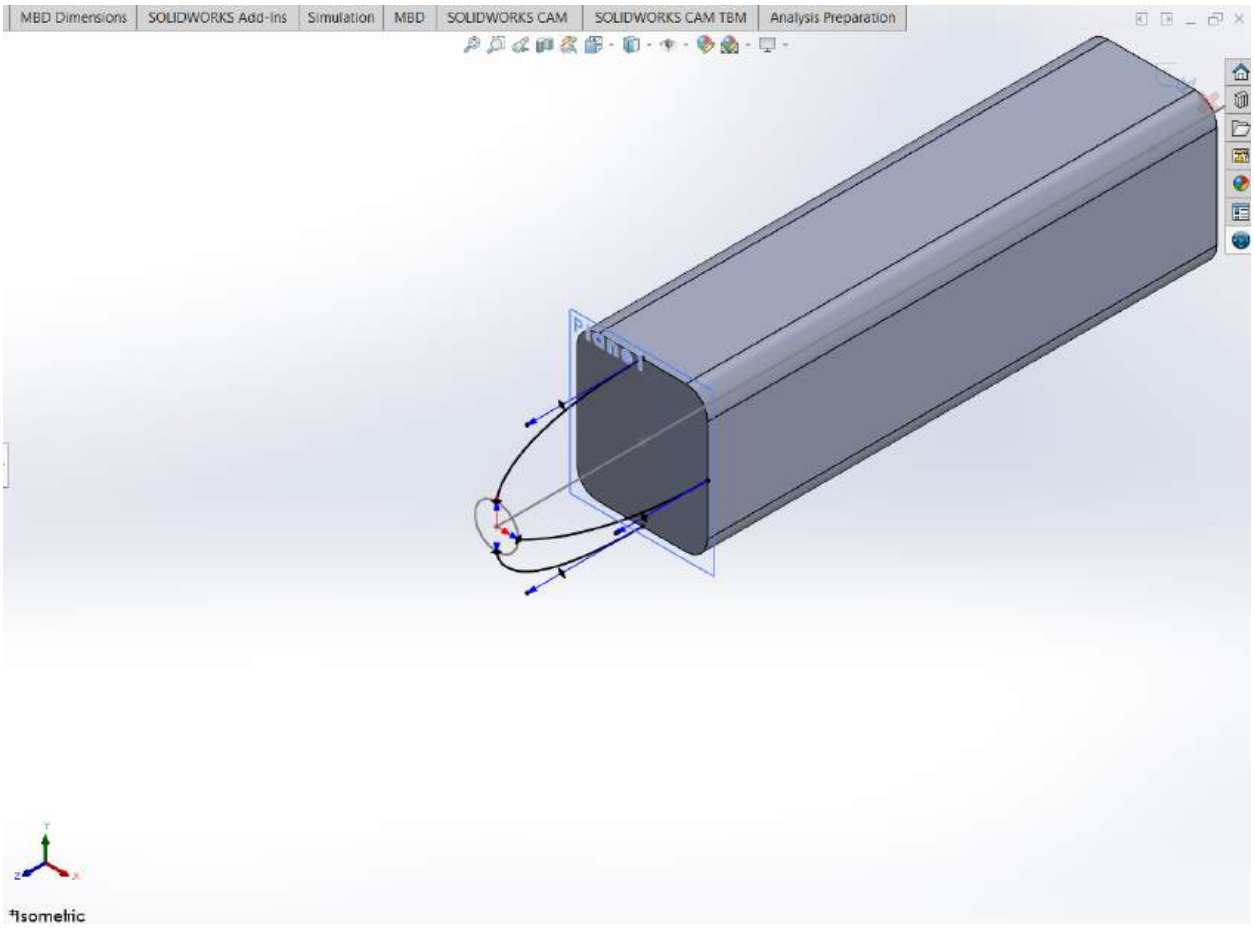
To select the entities to mirror, just click on one of the curves we just drew. I'll do the top one first:



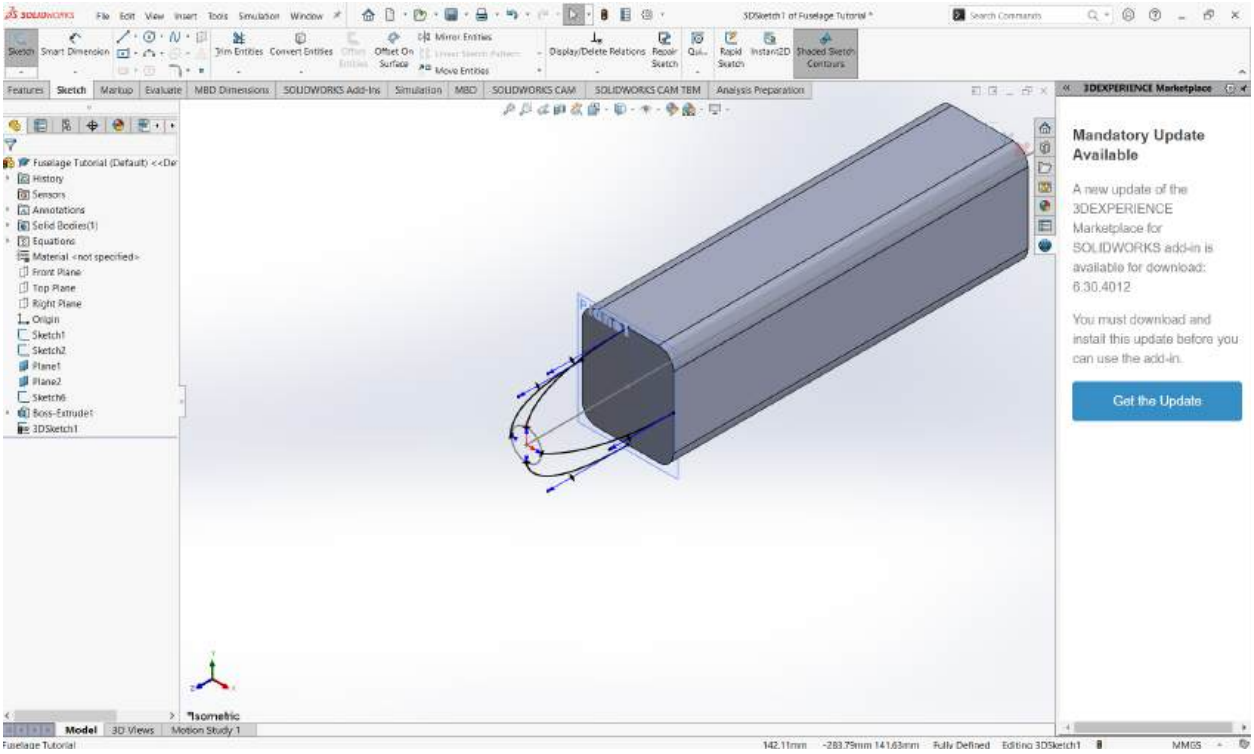
Great. Now, let’s choose the plane. Solidworks doesn’t automatically move us to the plane selection, so you’ll have to click under “Mirror about” on the left menu. Then, to choose the top plane, expand the tree and choose the plane, as highlighted below:



Voilla! Confirm. You should see the mirrored curve:



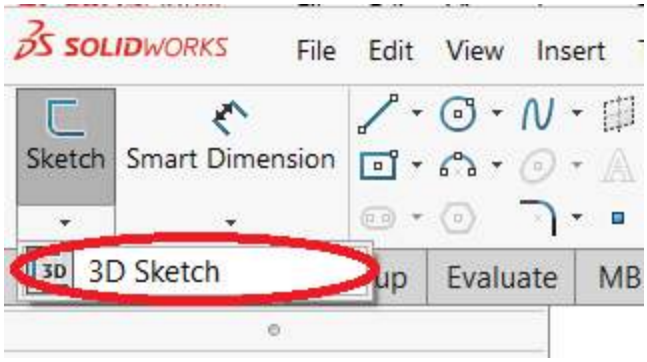
Perfect! I'll leave it as a small challenge for you to mirror the other curve. The only difference is that you'll select a different curve (duh) and you'll mirror it about the right plane. If you do it right, you should see this:



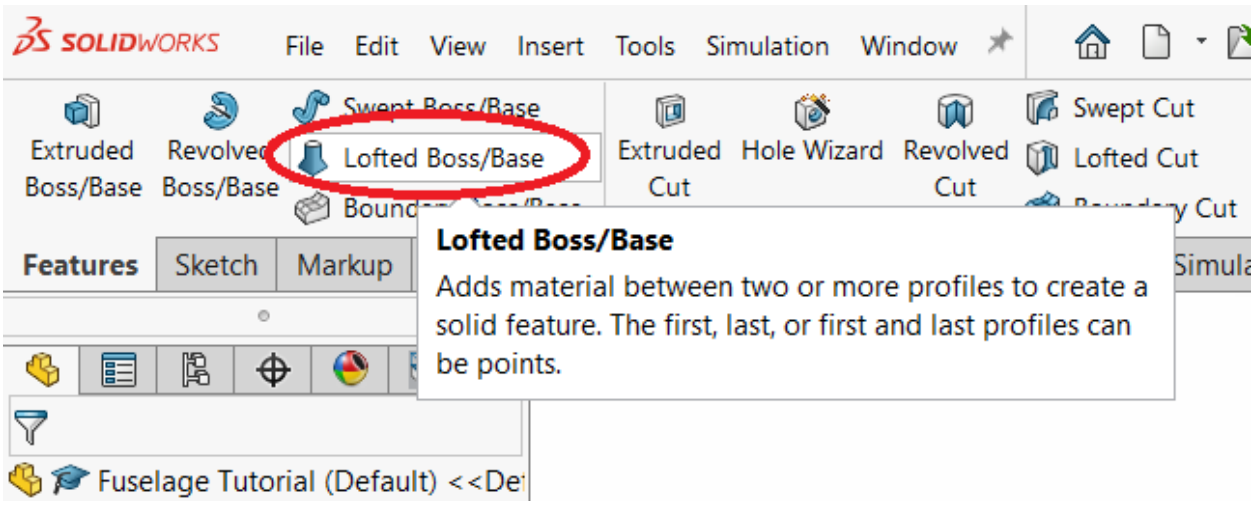
Perfect! We're ready to loft now!

Nose-center loft

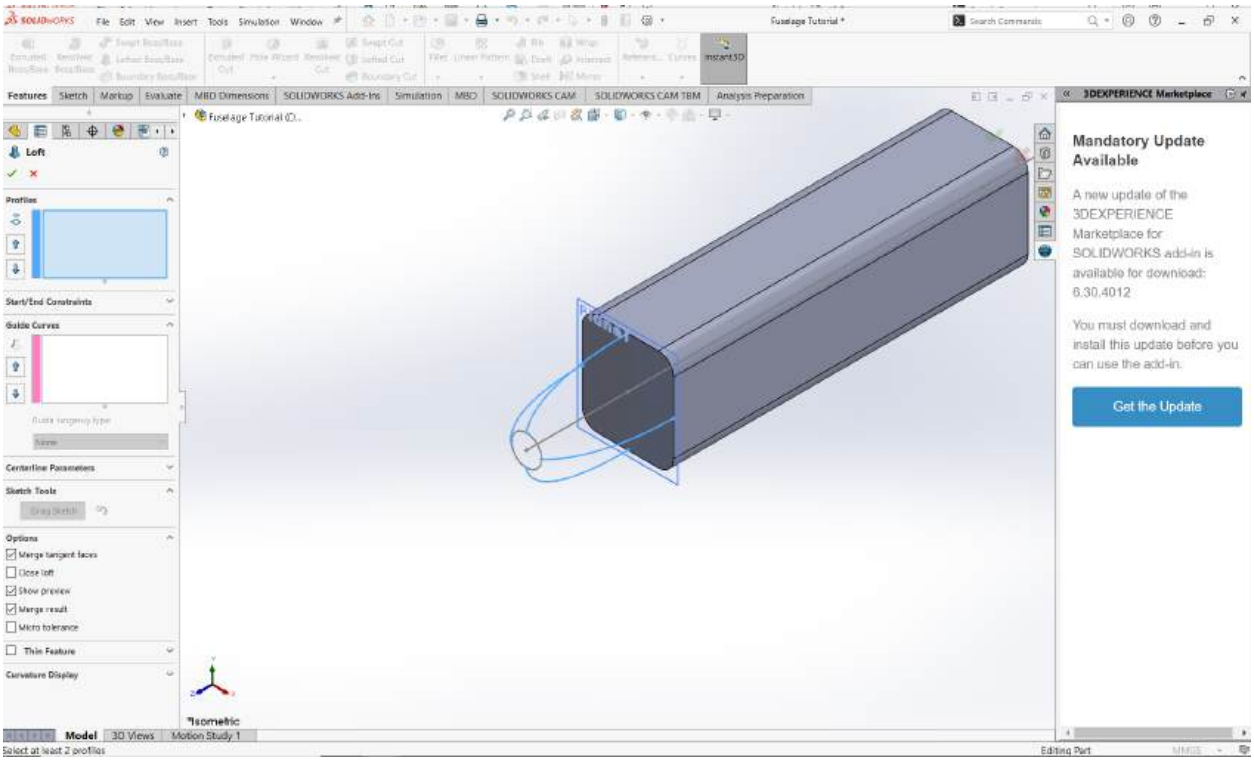
First, you'll have to exit the 3D sketch. To do so, go under Sketch and click the dropdown menu under Sketch. Then, click 3D sketch, which should be dark, indicating you are currently in it:



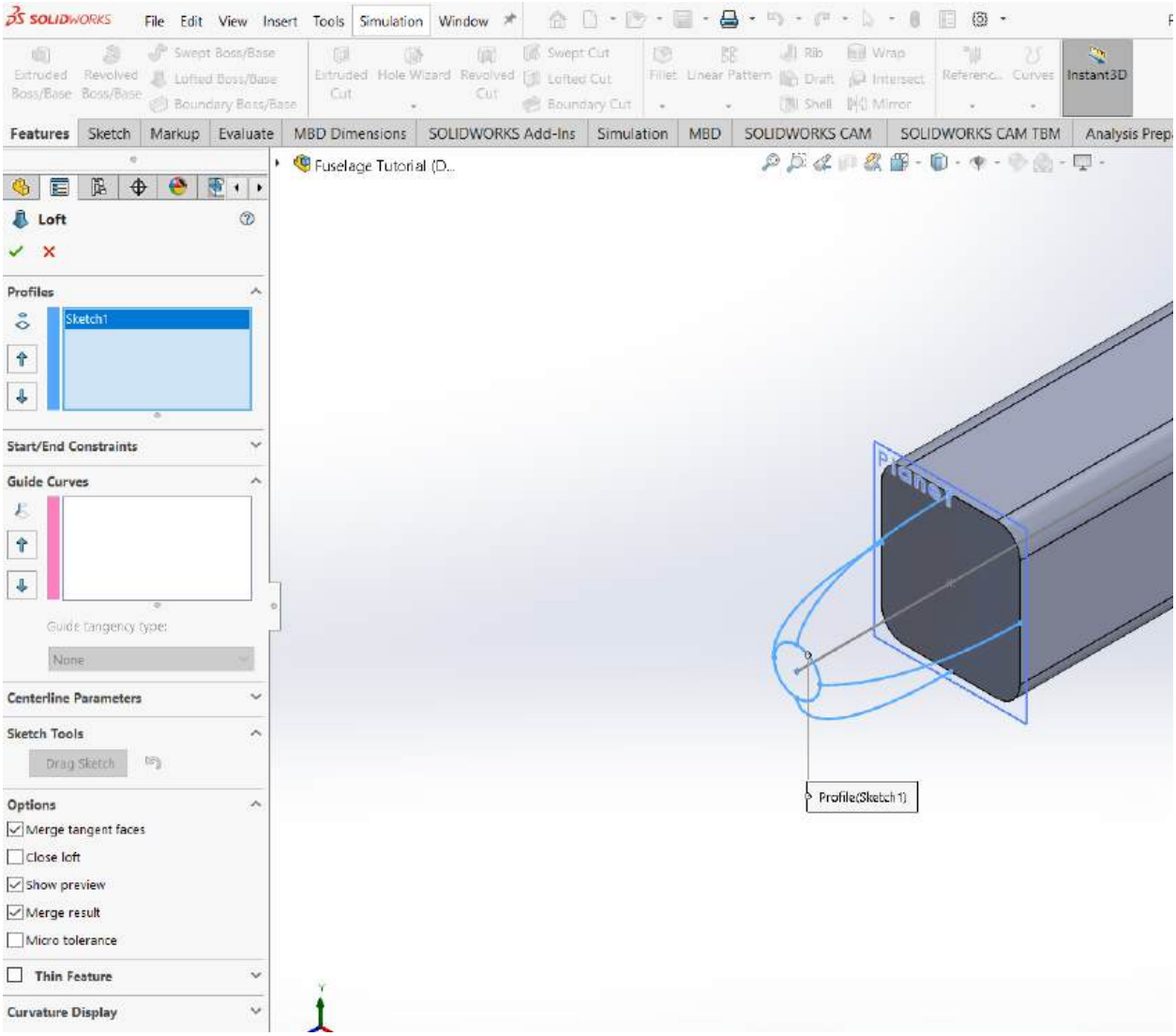
Click that to exit the 3D sketch. Then, under Features, select Lofted Boss/Base:



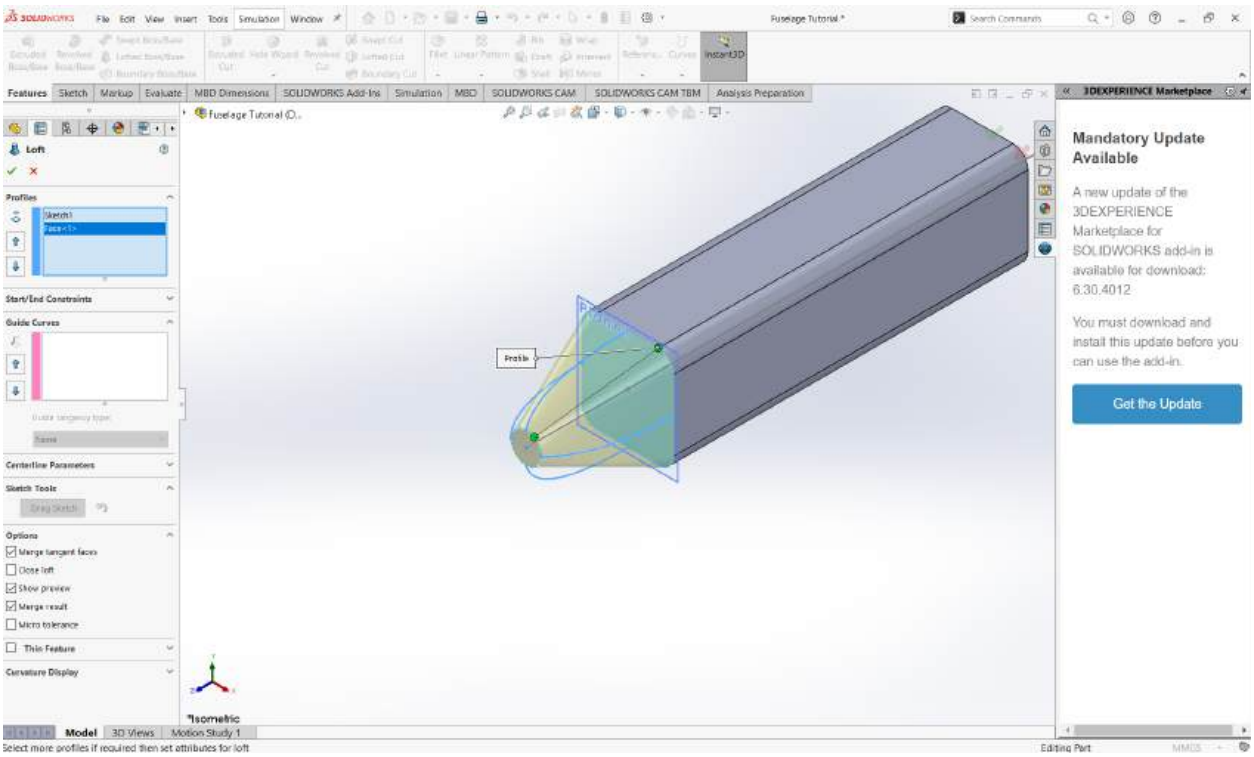
It should show you this menu:



First, we will choose the profiles. First click on top of the circle, like so:

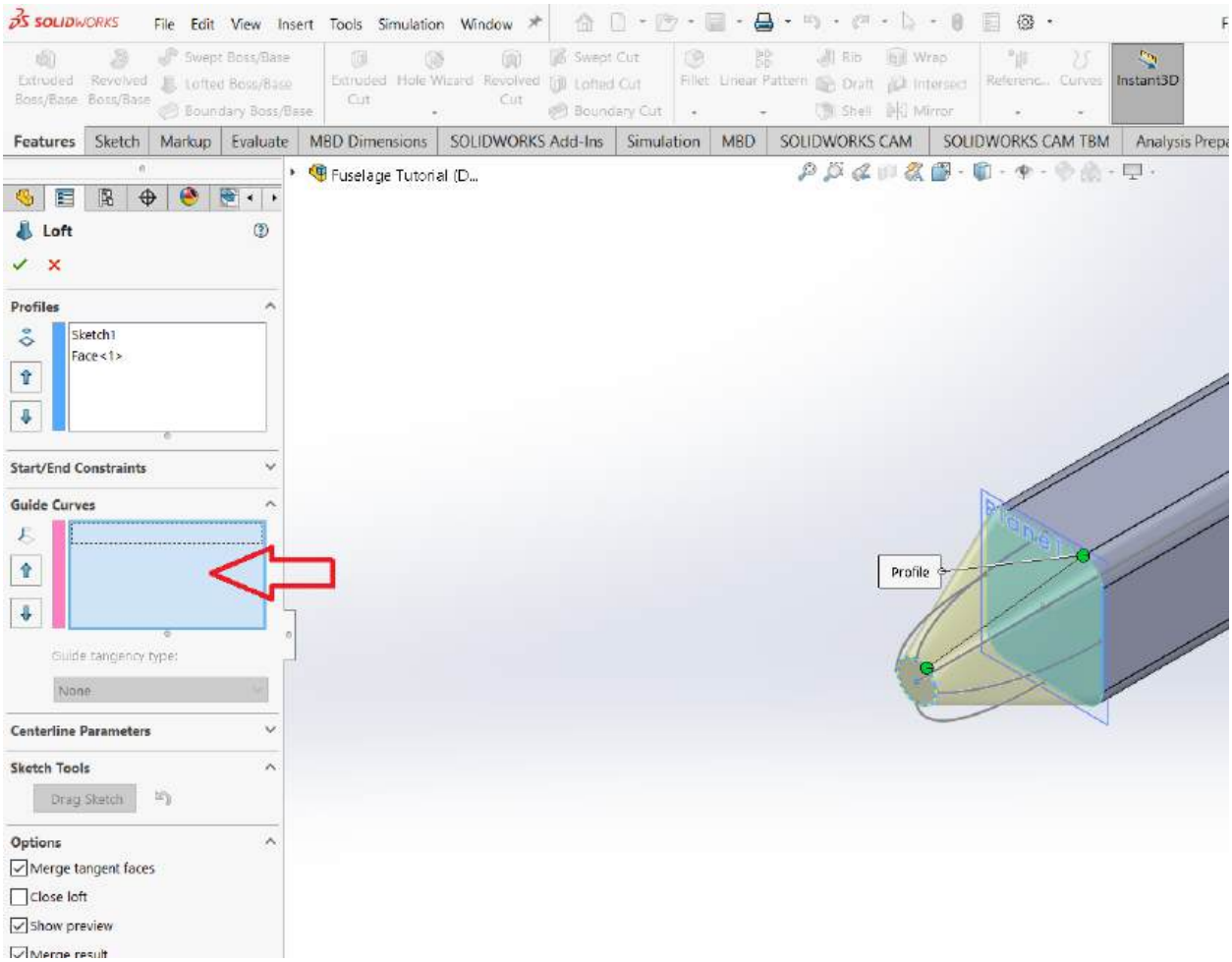


Then, click on the face of the fuselage. That's right, the face. If you try to click on the profile like you did with the circle, it will only highlight one side at a time, which is not what we want. So just go ahead and click on the face of the fuselage facing the nose. Solidworks will understand what we're trying to loft. You should now see this:



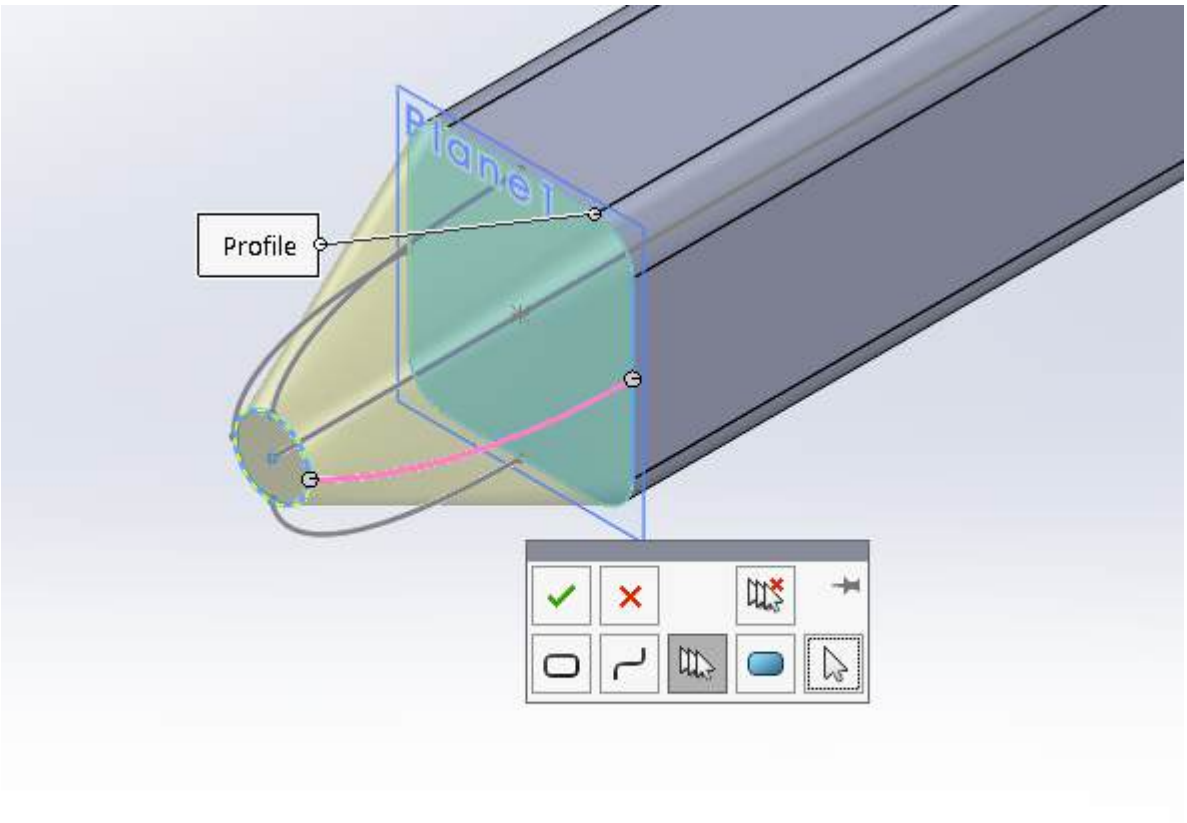
Not quite there. This is a direct loft of the circle to the fuselage center. This is what I showed you before we started this ordeal. We need to give it the guiding lines!

To do so, click under the Guide Curves menu on the left:

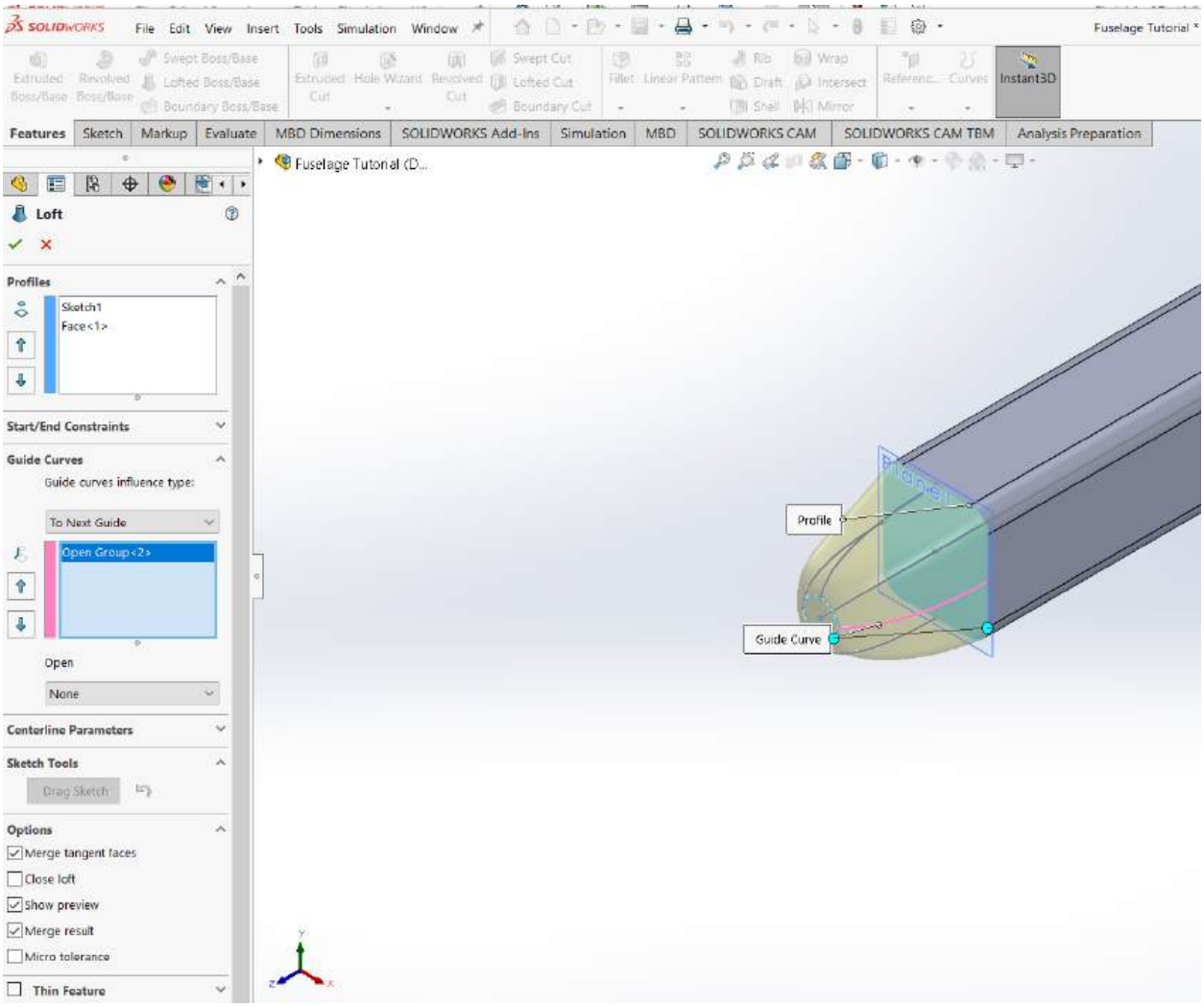


Now, a very annoying thing: you cannot simply select all of the guiding curves and then confirm. You will need to do the following for each guiding curve:

First, click on a single guiding curve. The following small menu should appear next to it:

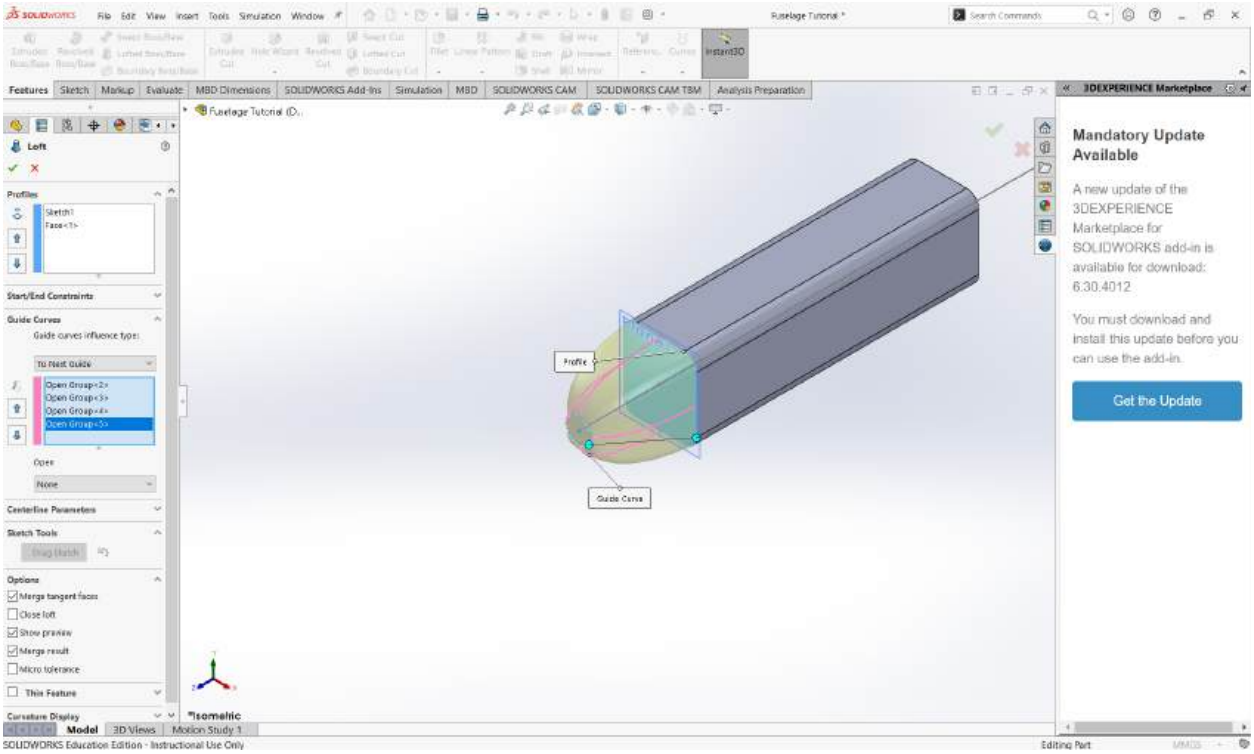


Confirm by clicking the green tick. The Guiding Curves menu will now look like this:

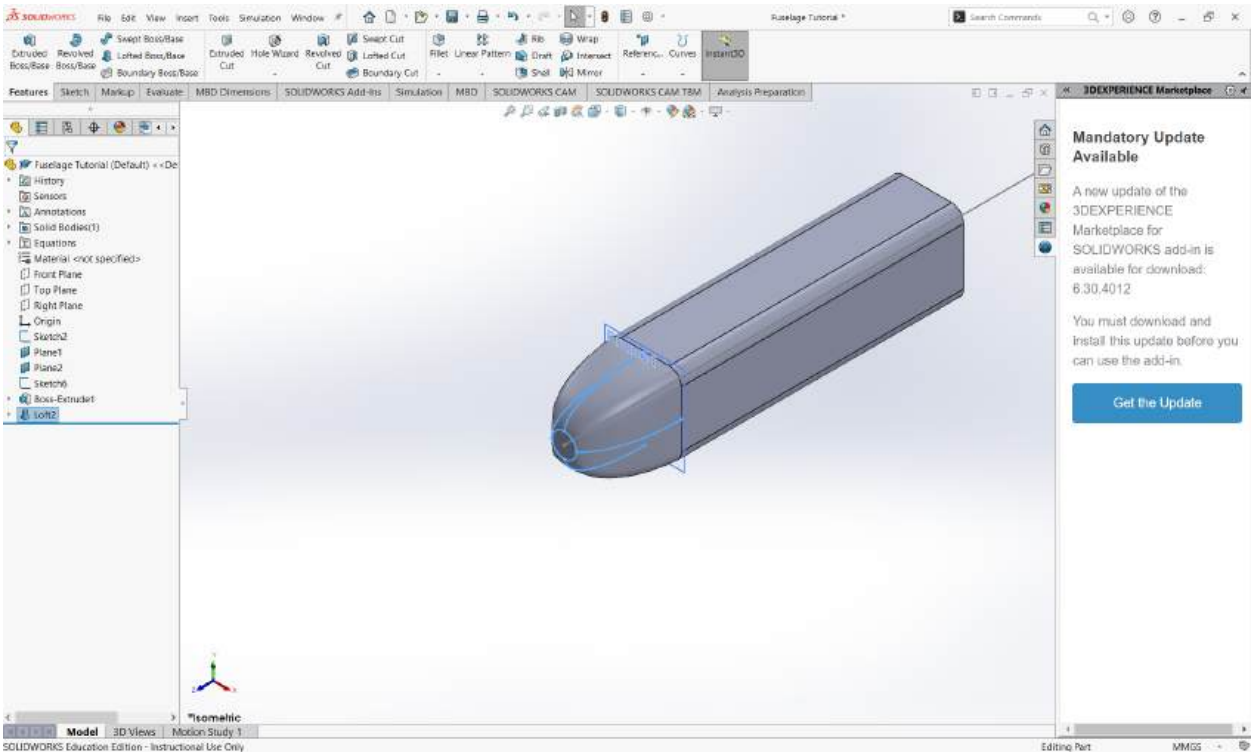


Notice how the nose also changed. It looks a bit more round. This is because the loft is now adhering to that guiding curve.

Repeat this process for each guiding curve. You should see this at the end:



Confirm the loft by clicking the green tick on the left menu:



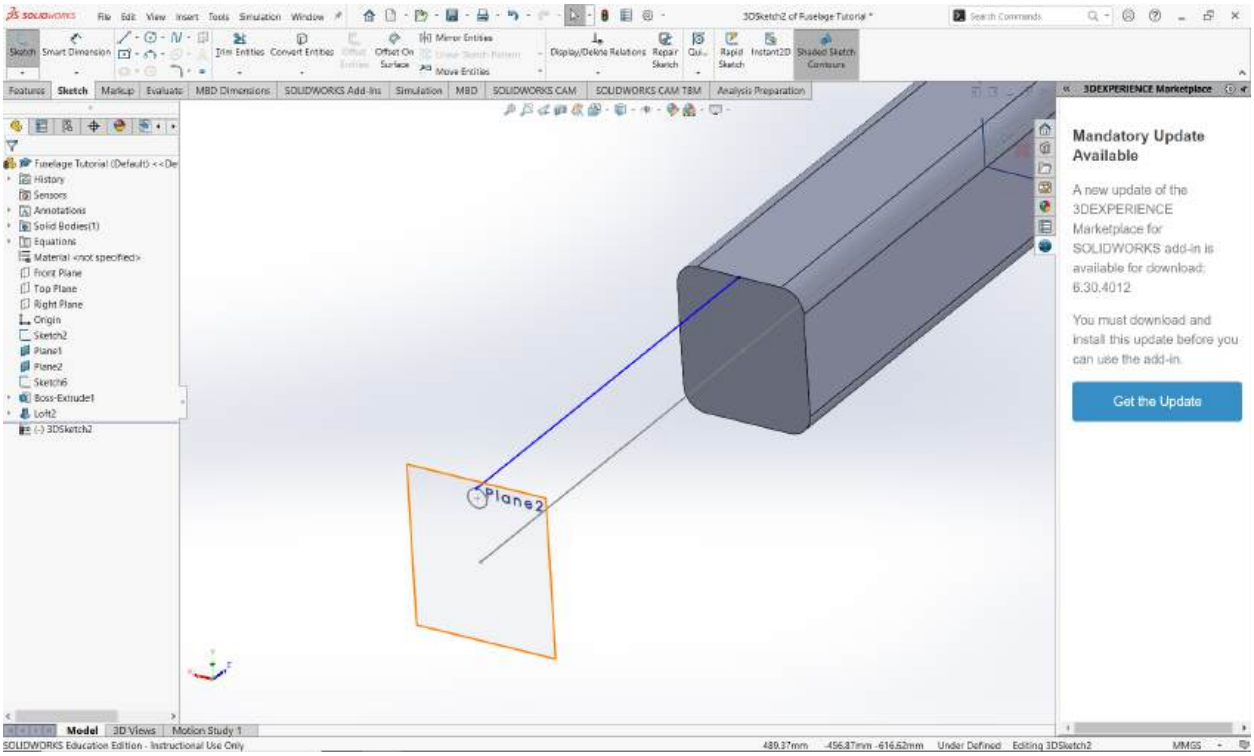
Boom! We have a nose! And it is nice and round, not a simple connection of the two faces.

That's great. We will now do the same for the tail!

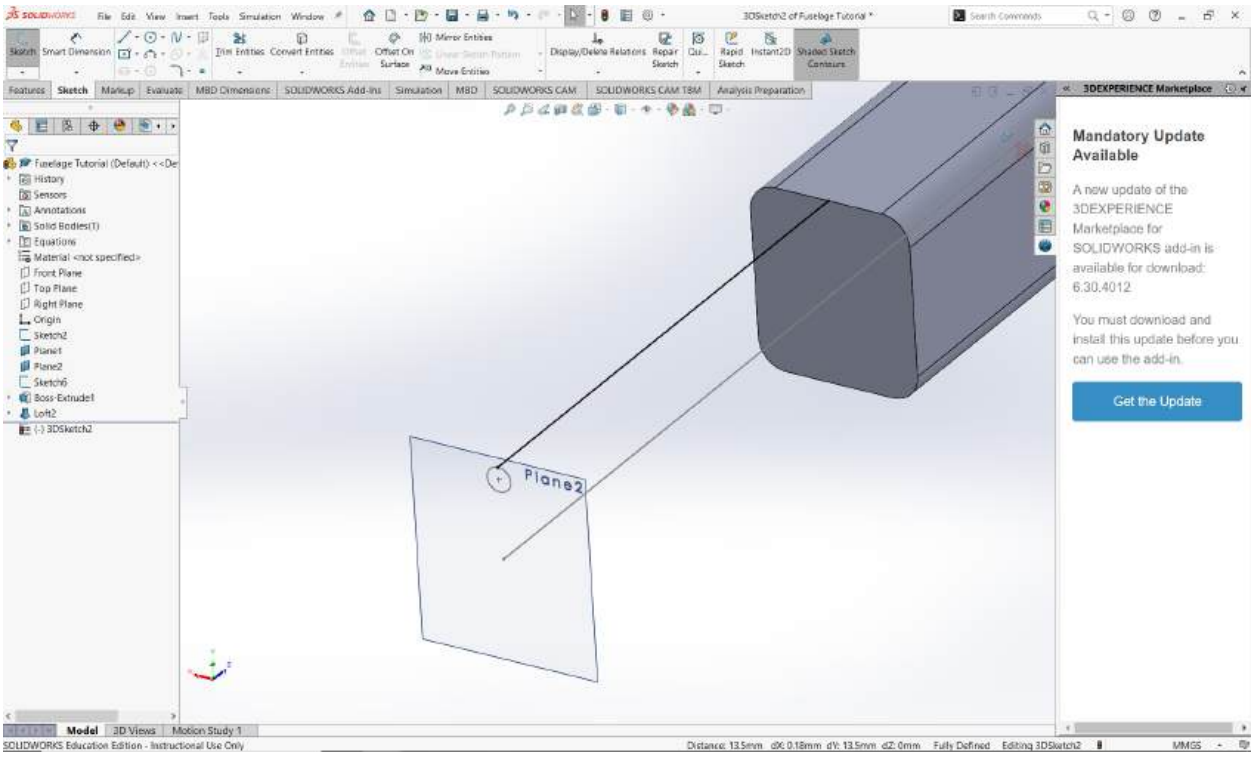
Tail guiding lines

The tail will be mostly the same, except for the top guide line. Because the cross-section for the tail is a circle tangent to the top of the center part, the guide line on top is just a straight line.

First, go back into the 3D sketch just the way we went in it last time. Then, select the line tool and draw a line from the midpoint of the center part to the top of the tail circle, like so:

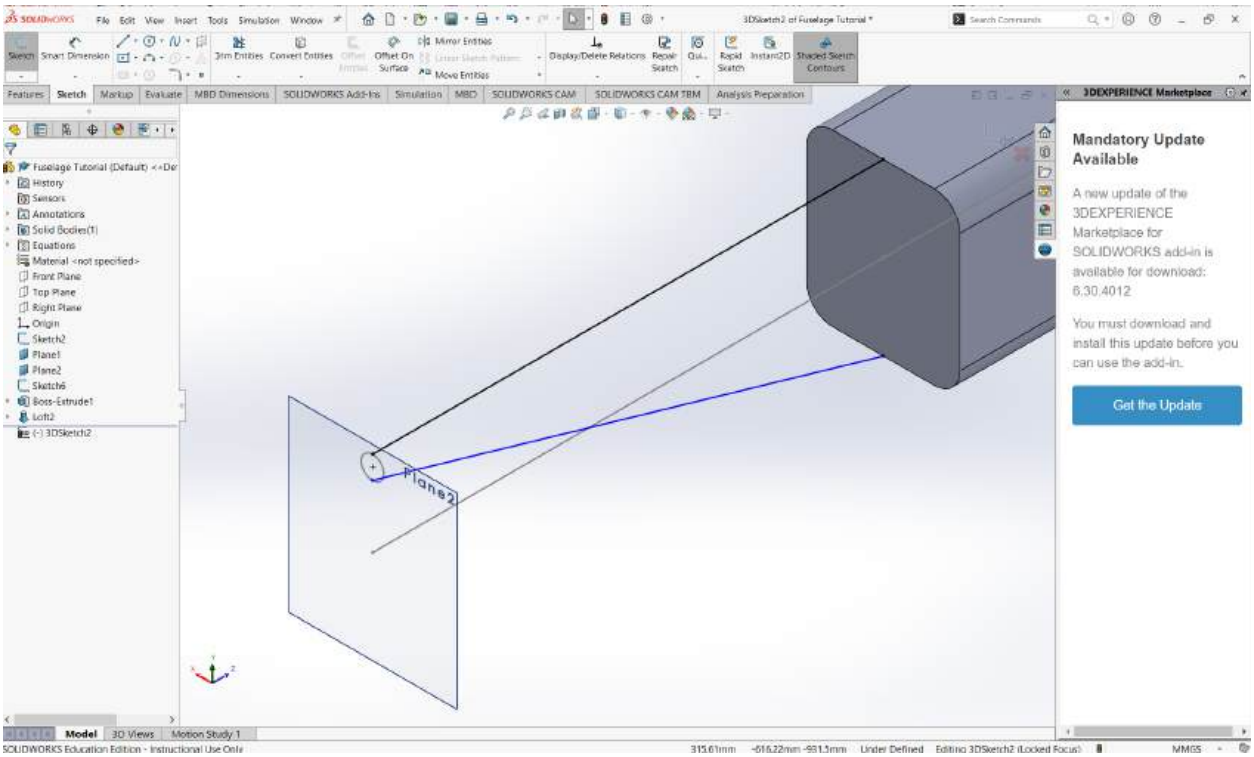


We still have to align the point in the circle with the origin. We do the same thing we did for the top plane. Select both the point and the center of the circle and hit "Along Y" in the menu. It should now look like this:

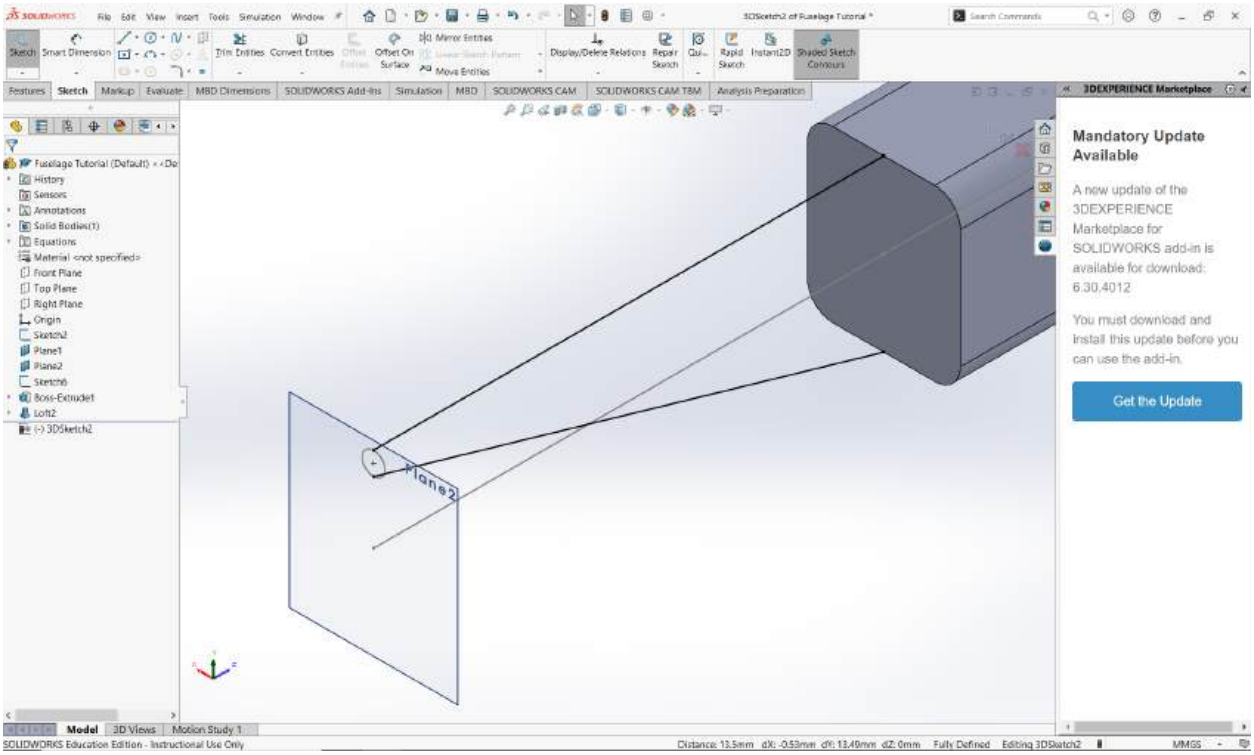


Great! That’s done! Let’s do the bottom now.

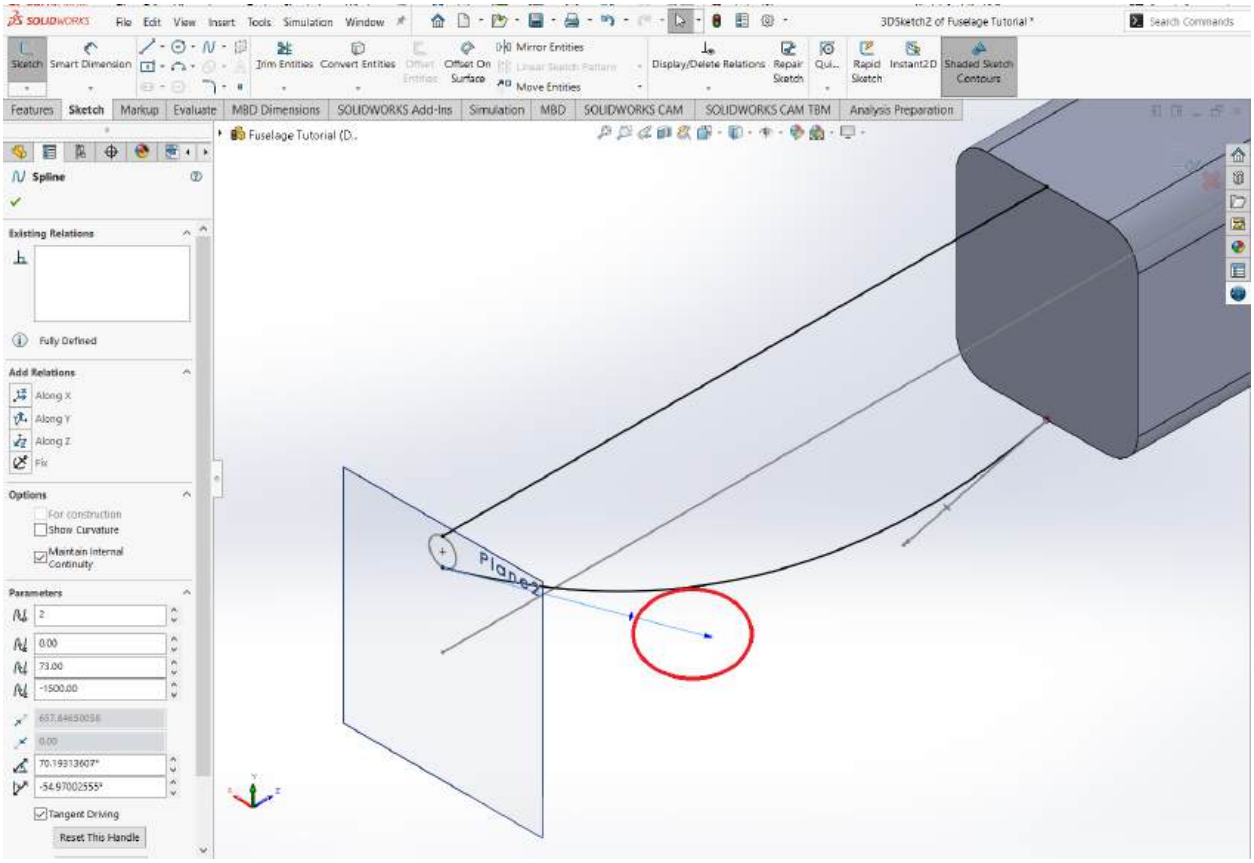
For the bottom, we use the spline tool just like before. First, using the Spline tool, draw a curve from the midpoint of the bottom side of the center to the circle, like so:



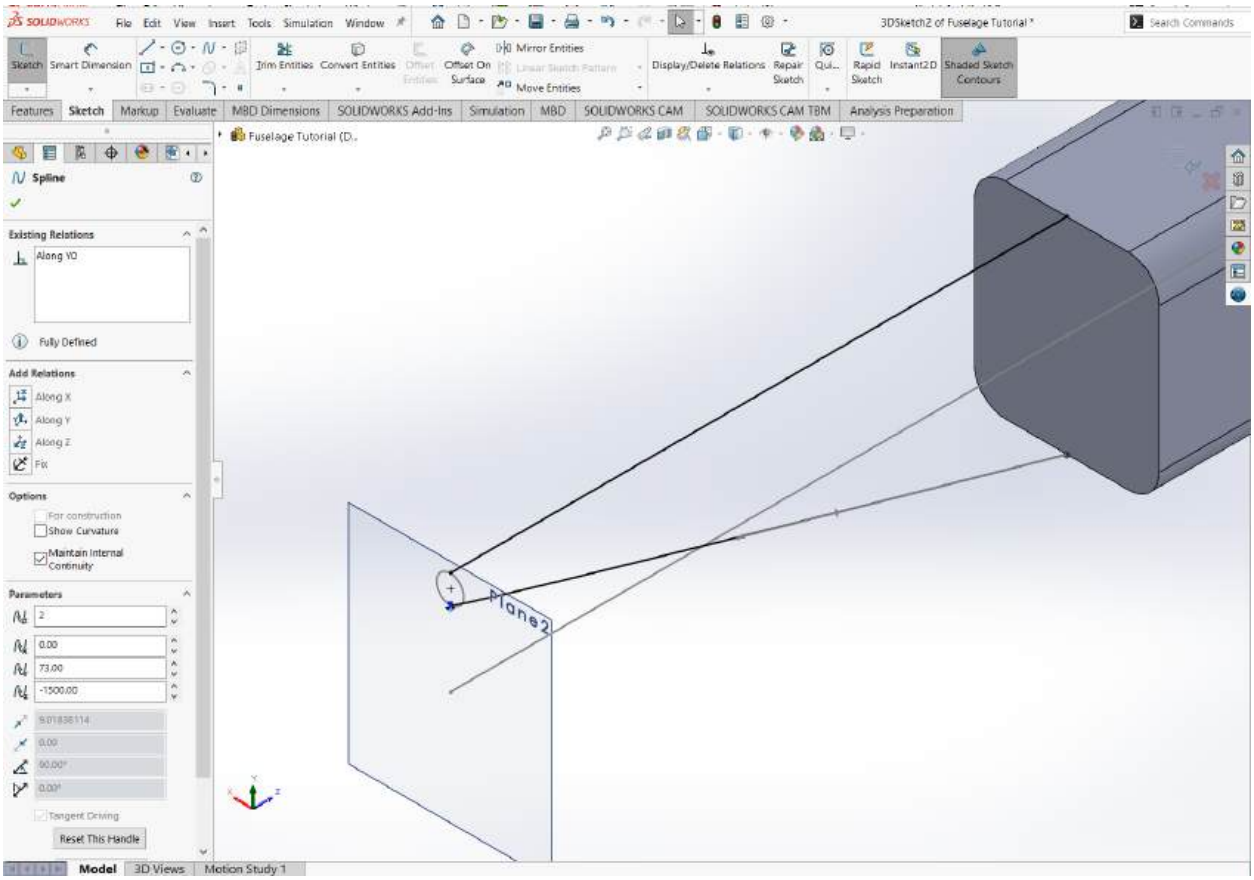
Once again, we need to align the point in the circle to the center of the circle. Select both the point and the center of the circle, and click “Along Y.” You should now see this:



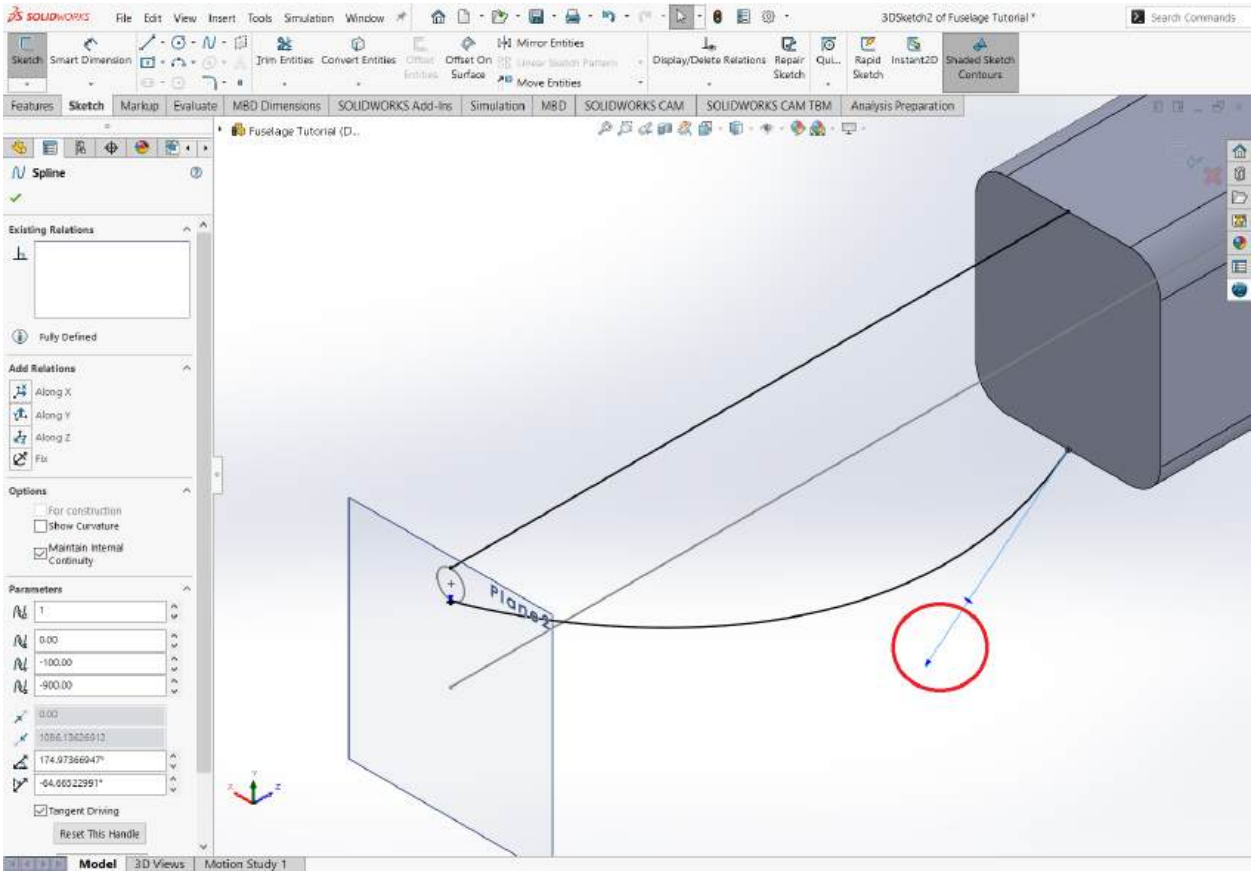
Perfect! Now let’s align the arrows again. Once more, the arrows going from the circle to the center need to be very very small. Click on the bottom curve, and select the arrow from the circle to the center. Drag it around, and you should see this:



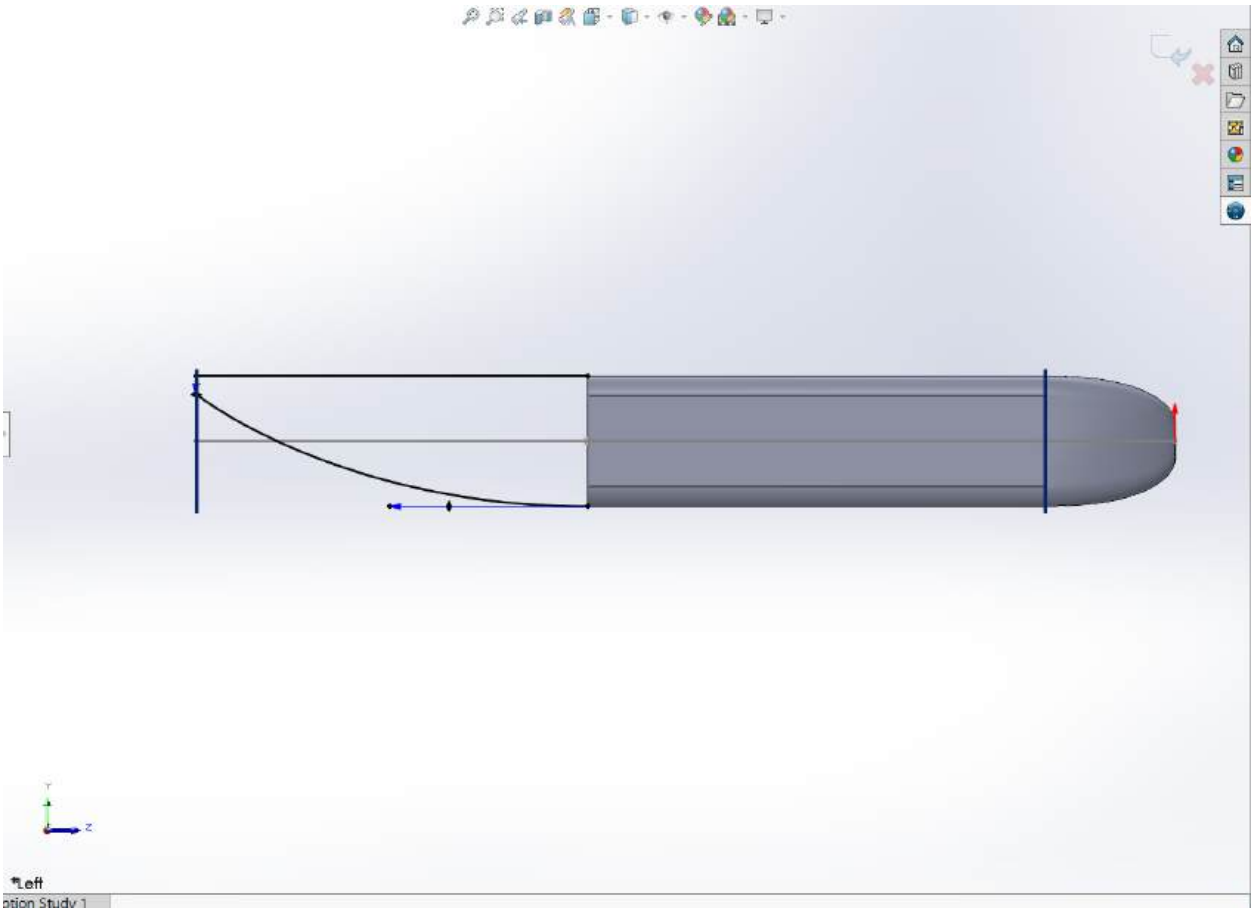
This arrow should be constrained along y, so hit “Along Y” on the left and, once again, make it very, very small. You should now see this:



Great! Now, select the other arrow and drag it around:

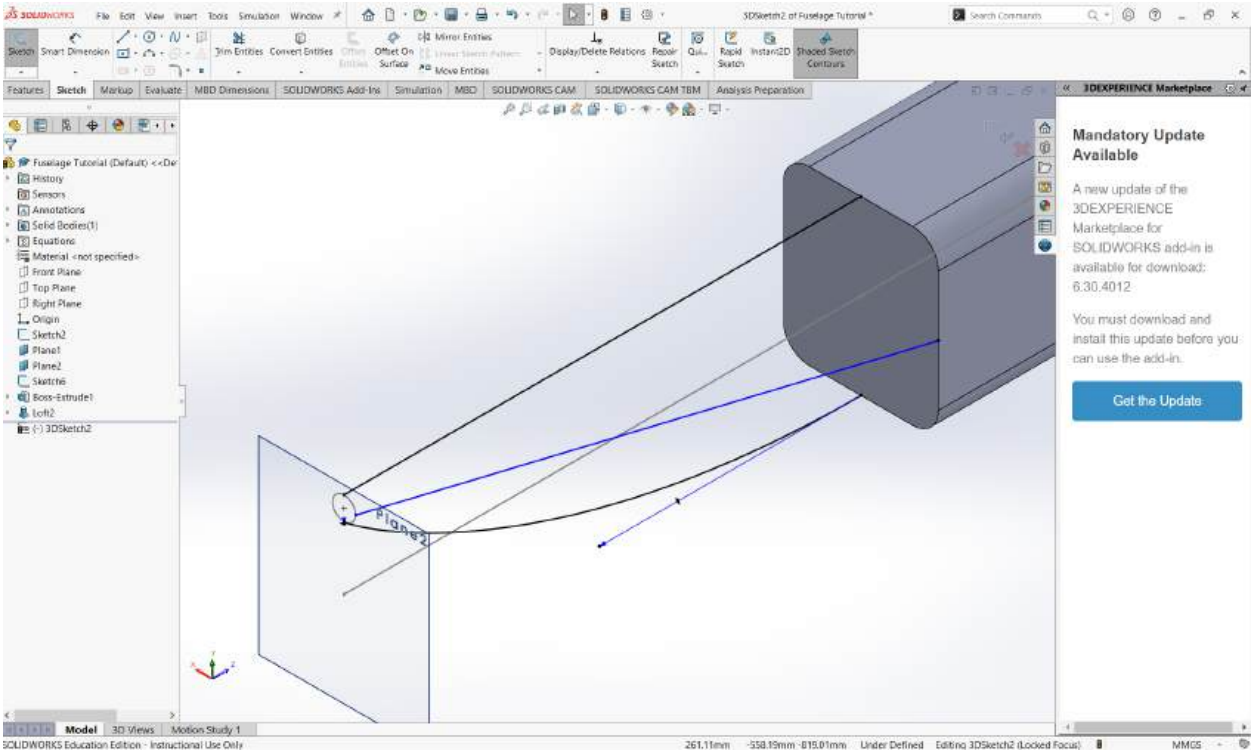


This one should be “Along Z” and, with some creative freedom, drag it around until it looks good. I arrived at something like this (from the side):

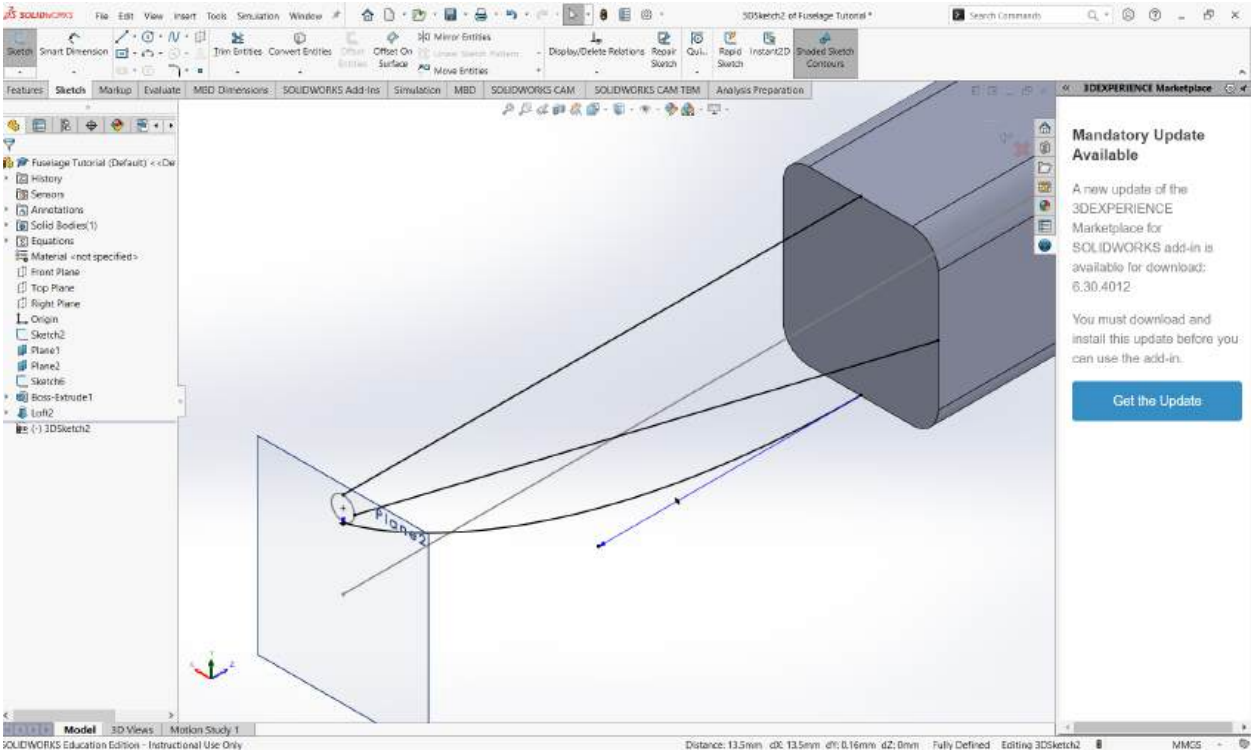


Great! Let’s do the side ones now. We only need to do one because we can just mirror it about the right plane.

We do the exact same thing. First off, a spline from the midpoint (always remember the midpoint!) to the circle:

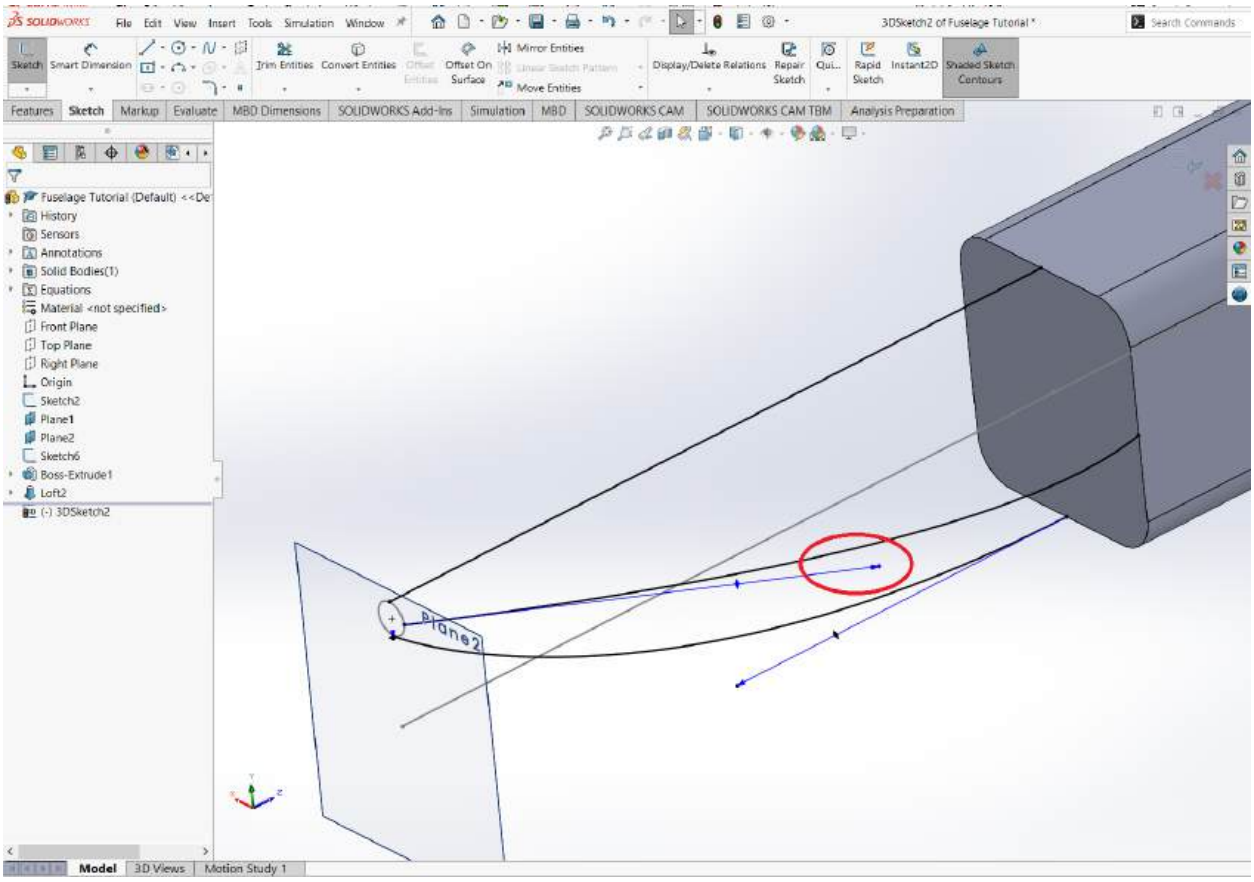


Now, we must align the point to the center of the circle. Select both the point and the center of the circle. This time, constrain “Along X” and confirm. You should now have something like this:

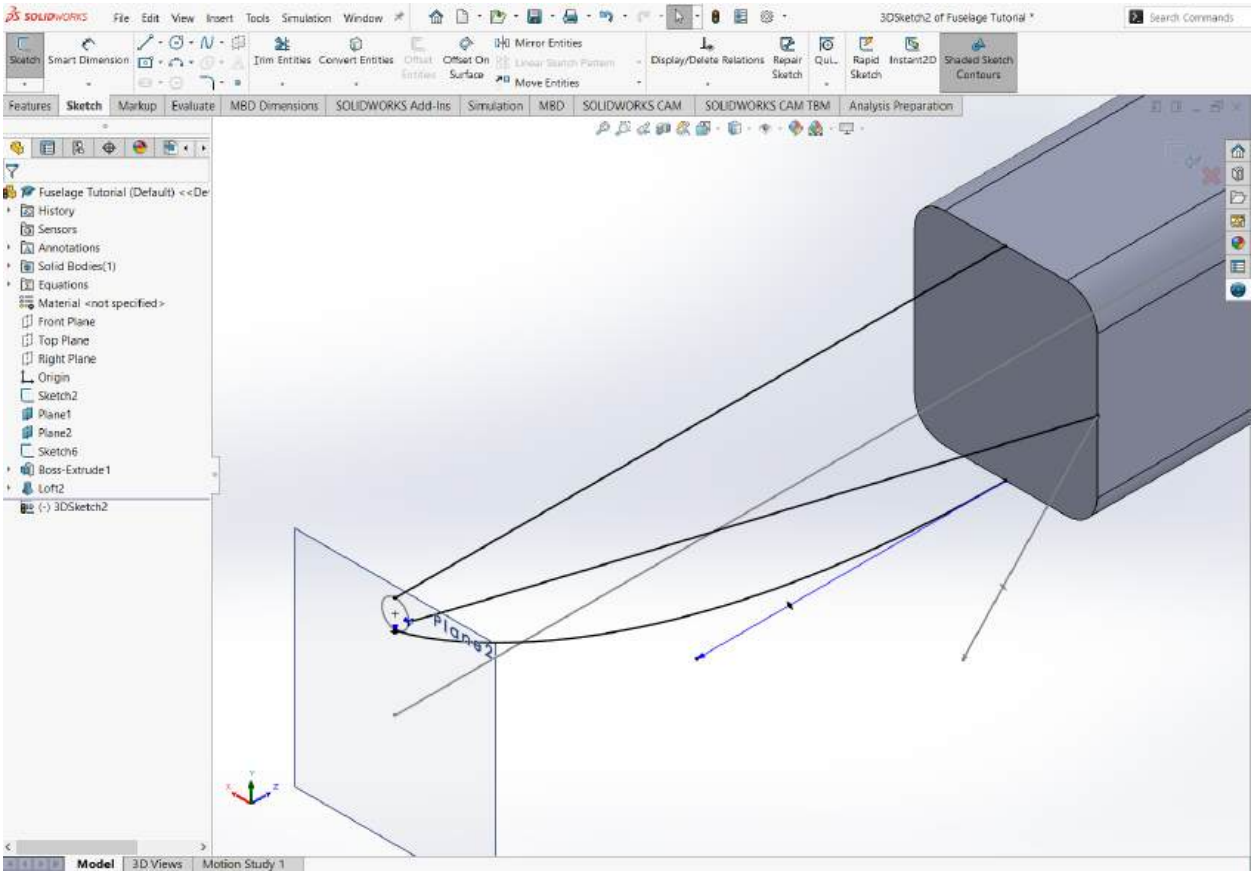


Great! Onto the arrows.

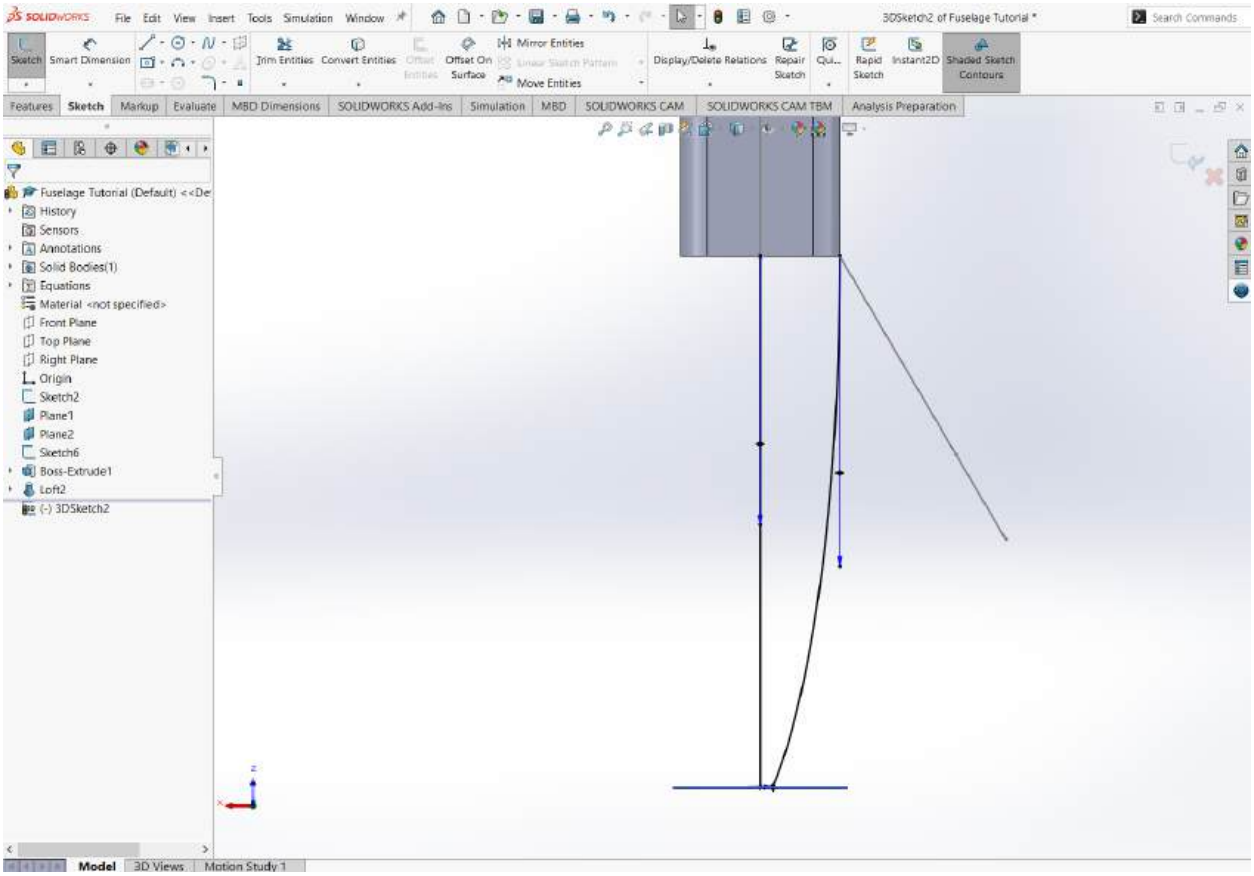
Click the spline and drag the arrow going from the circle to the fuselage around:



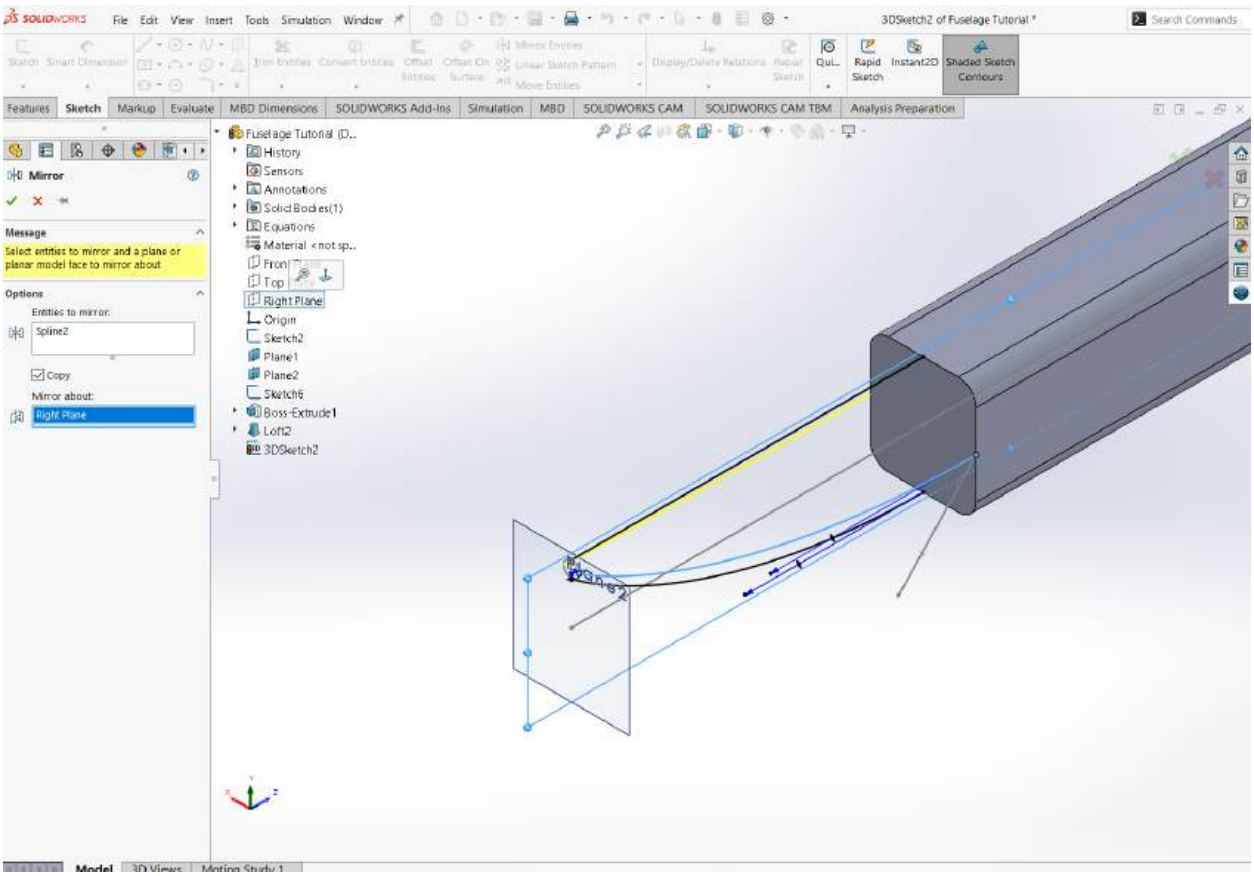
This arrow should be along x, so choose “Along X” and make it very, very small:



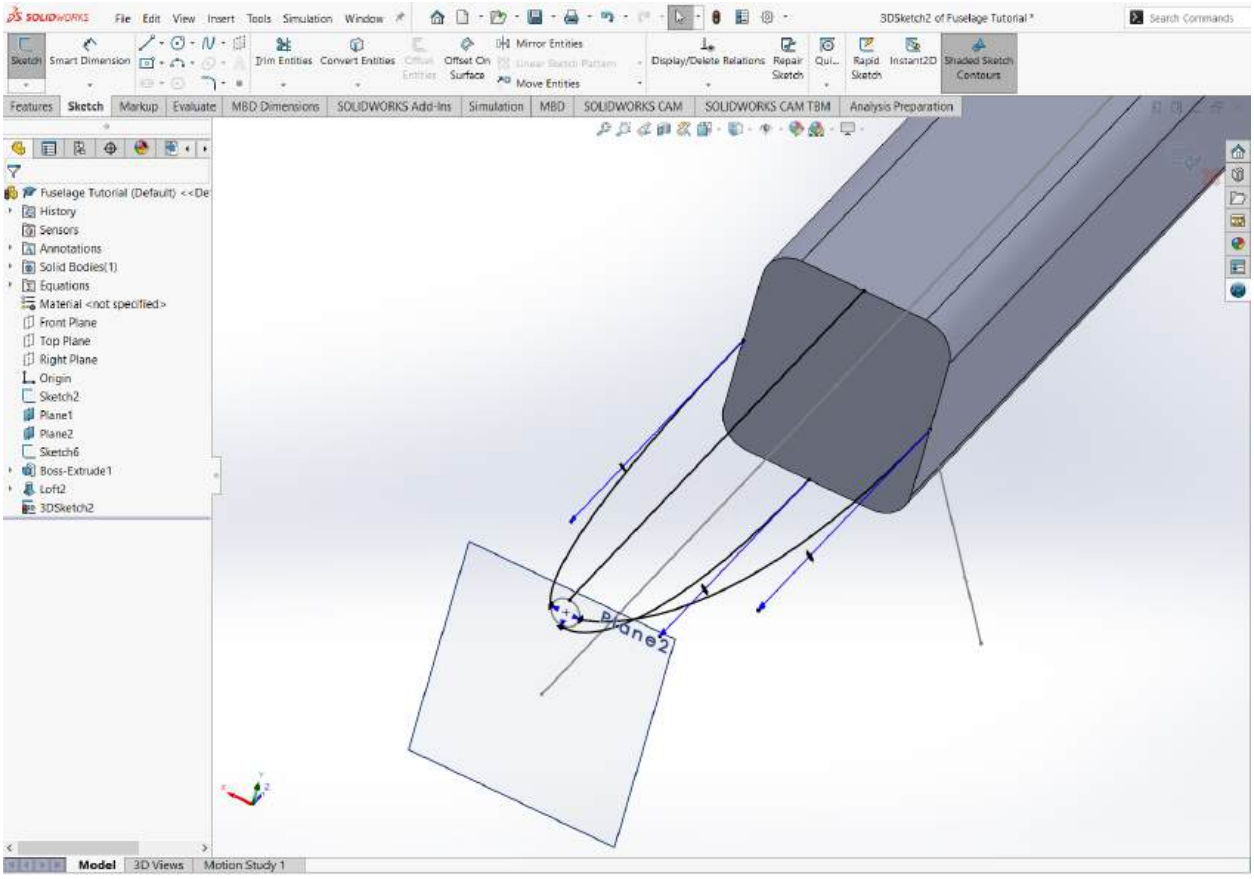
The other arrow should be Along z, so choose “Along Z” and, once again, move it around until it looks pretty good (creative freedom). I got to something like this from the top view:



Perfect. Now, let’s mirror this curve. Same thing we did with the nose. Select the Mirror Entities tool. Choose the spline and the right mirror:



Confirm and voila! Our guide curves!

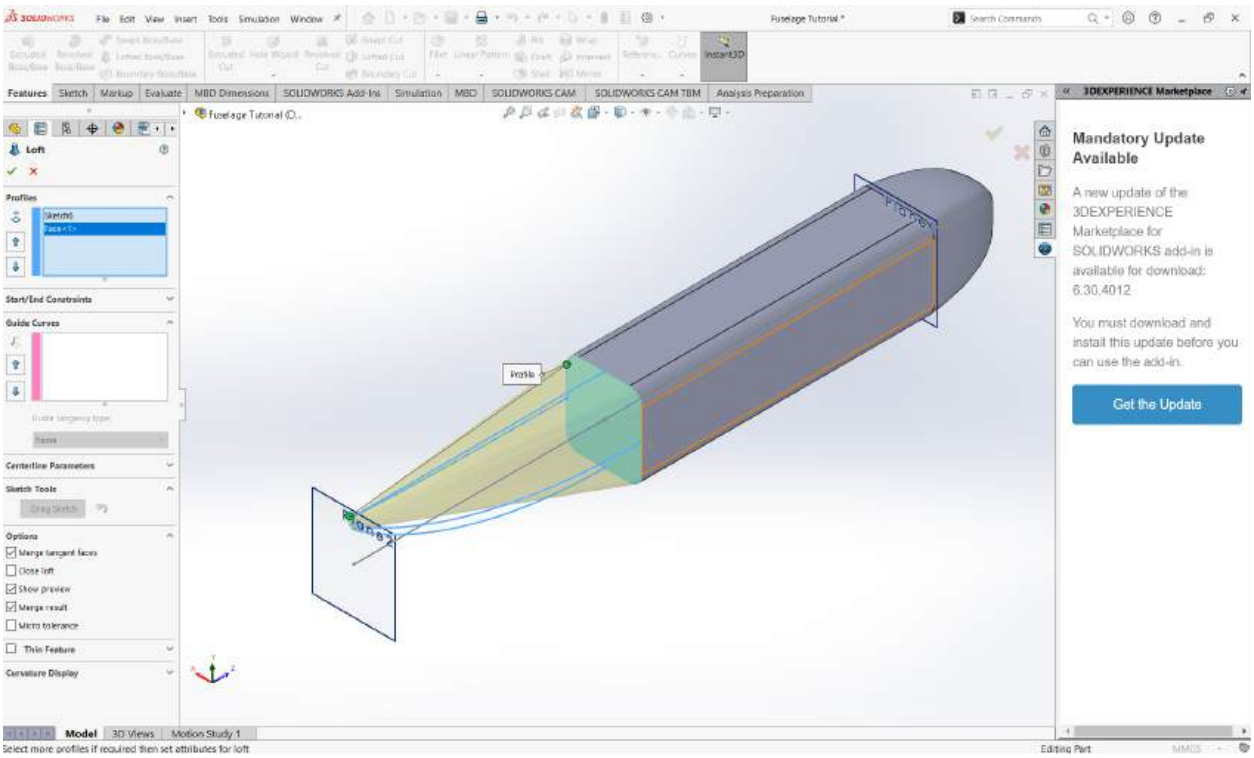


We are now ready to loft the tail.

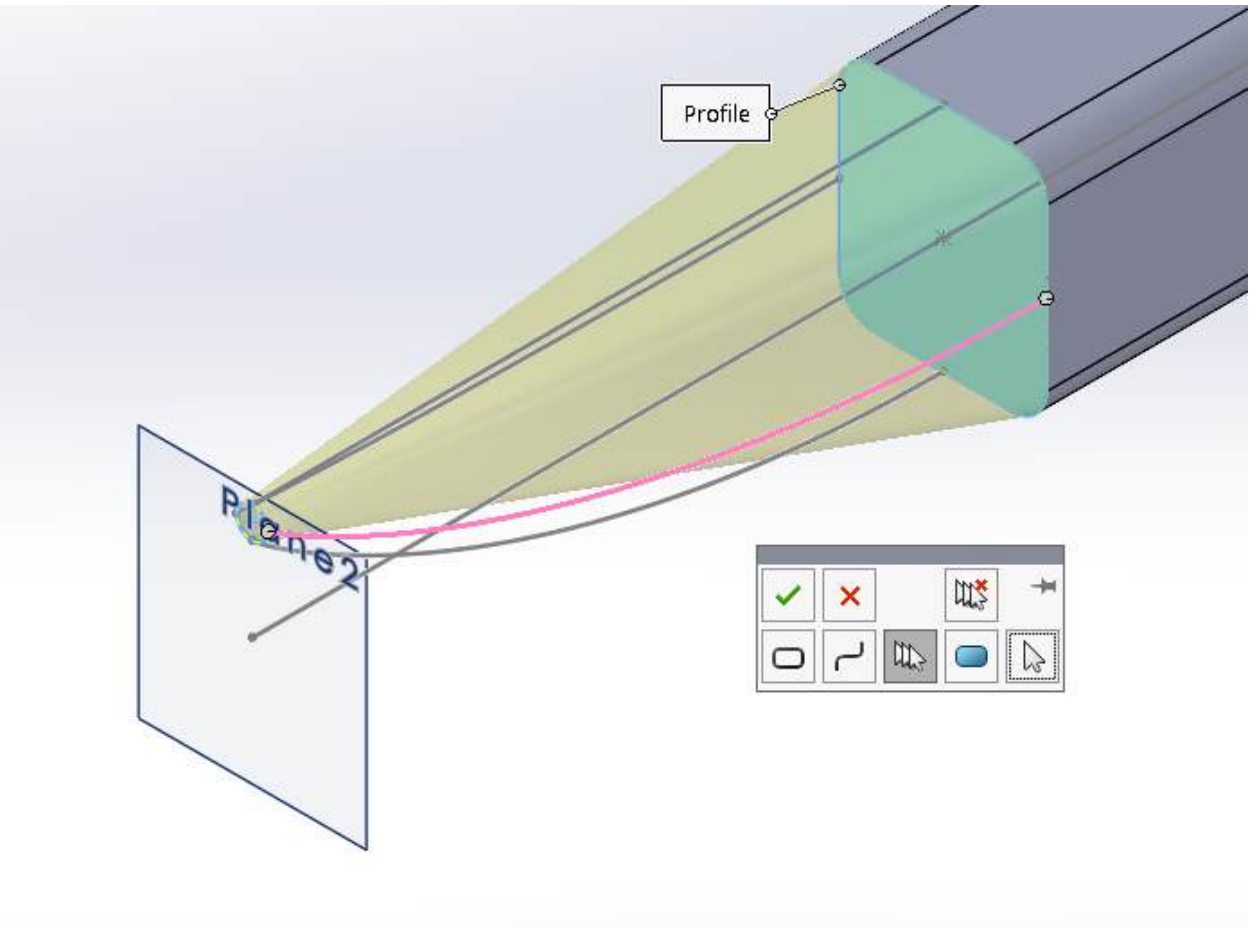
Center-tail lofting

Exit the 3D Sketch the same way as you did before. Go to Sketch, click the dropdown menu on Sketch, and click on 3D Sketch.

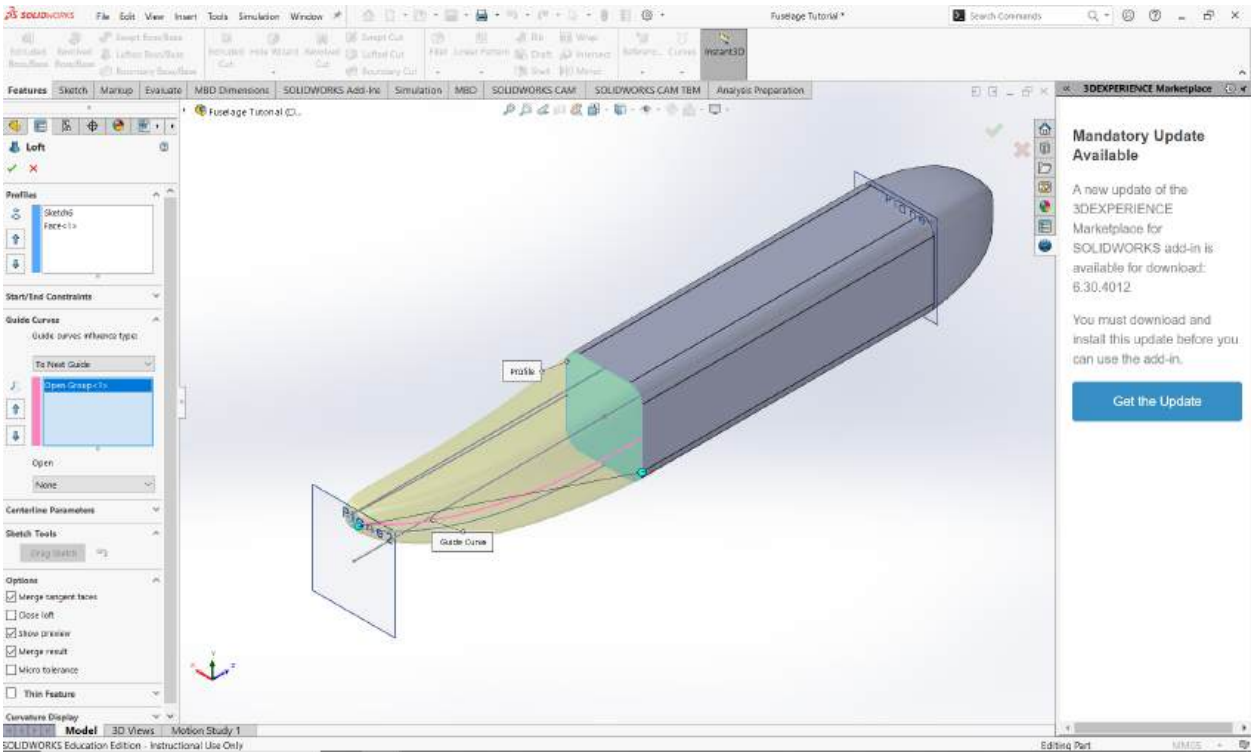
Then, under Features, select Loft Boss/Base. Just as you did before, select both the circle and the face of the center part facing the tail. Remember you need to click **on the face**, not on the lines of the rectangle. Solidworks will only let you select a side, but we want the whole rectangle, so we click on the face. You should see this:



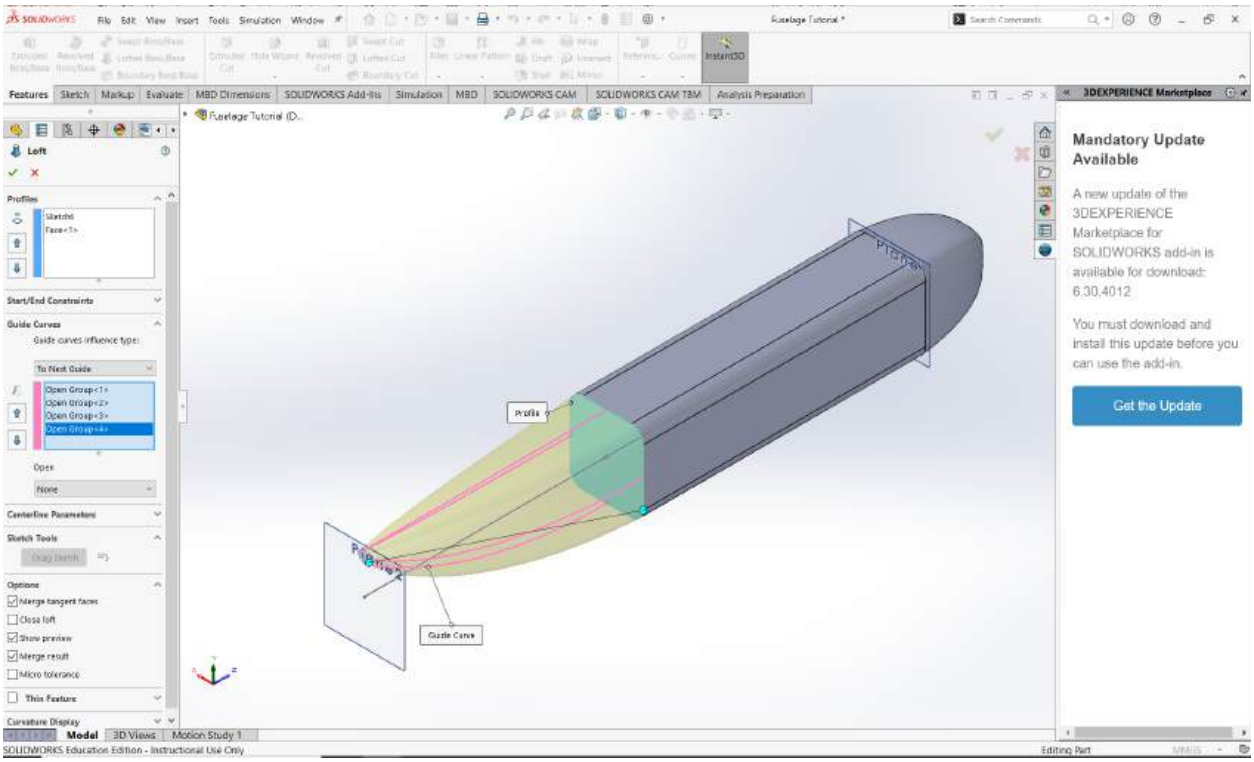
One thing left: the guide curves. We do the exact same thing once again. Remember that we must select one guide curve at a time. First, click under Guide Curves on the left menu to enter the Guide Curves selection. Then, select a single guide curve:



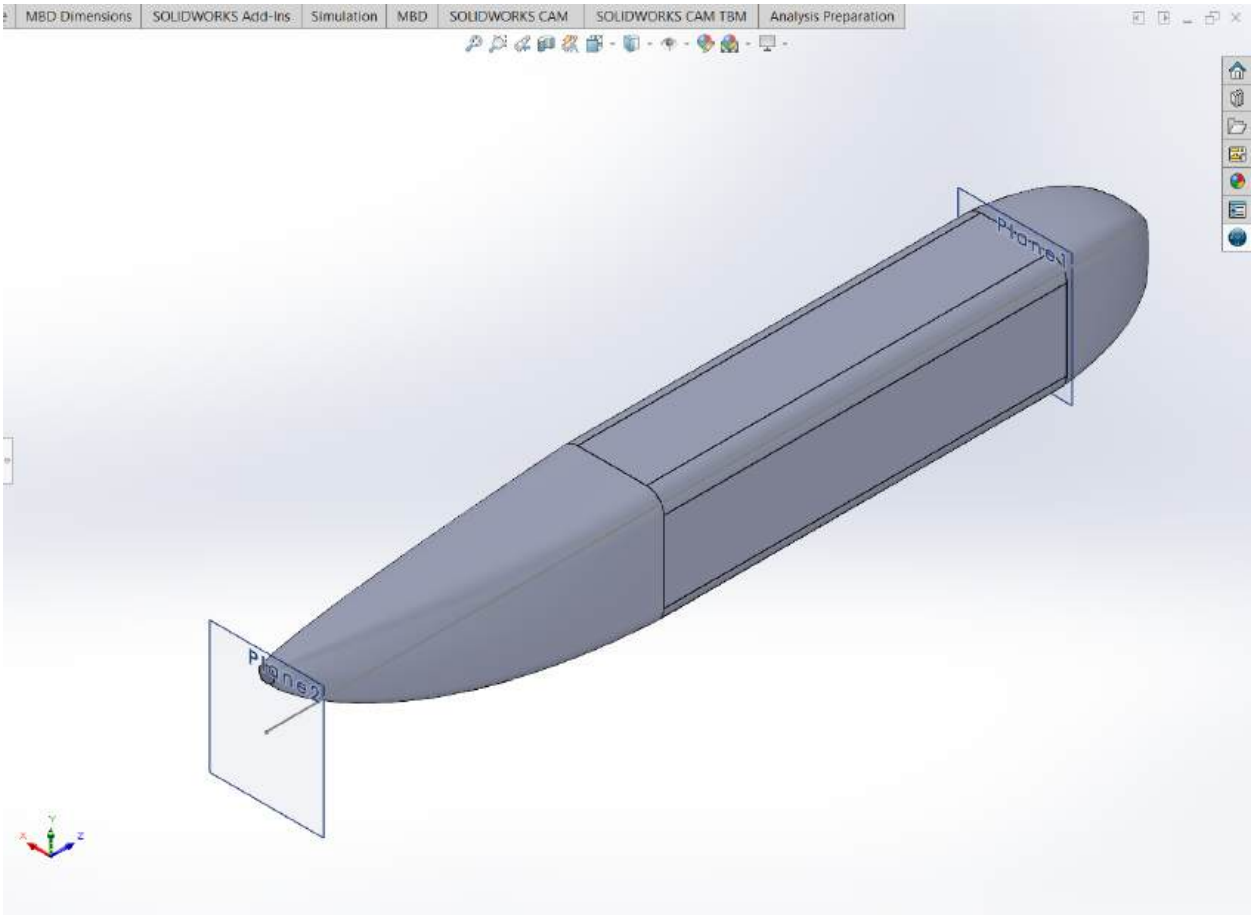
Confirm by clicking the green tick on the menu that appears:



Repeat the process for every single guide curve, all four of them. By the end, you should see this:

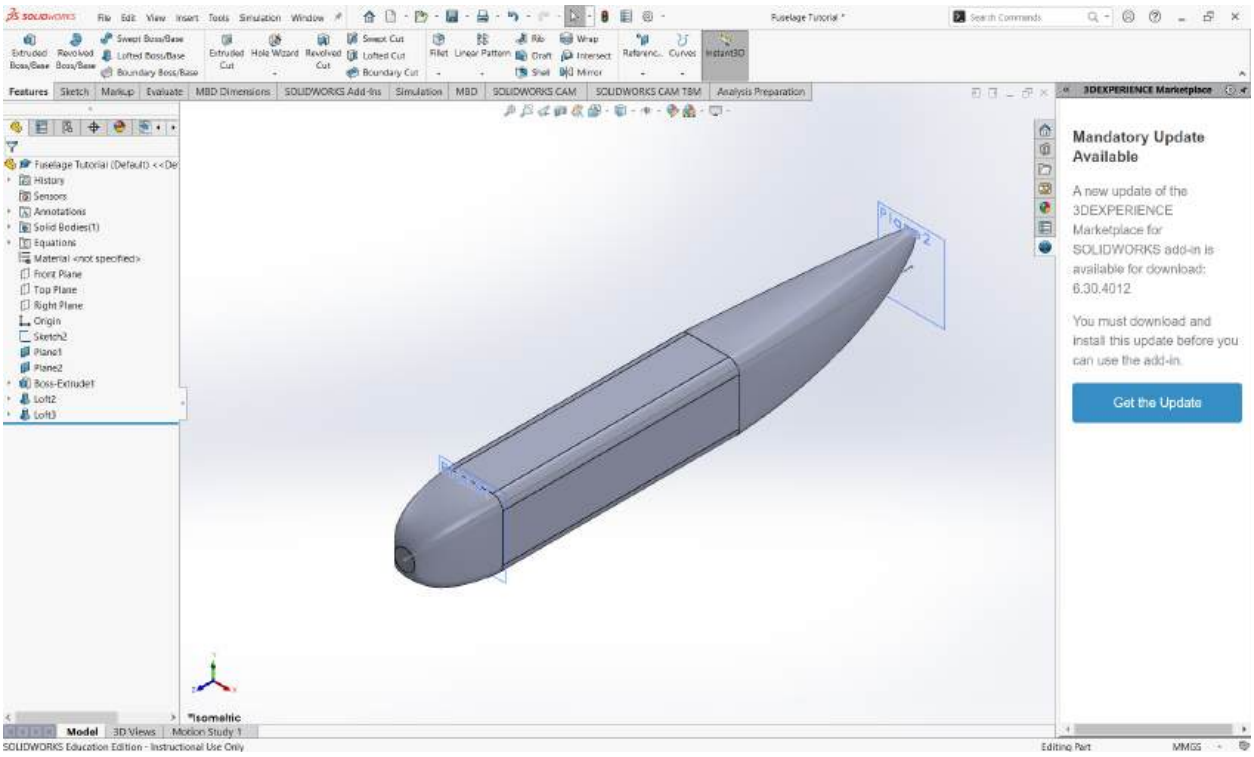


Perfect! Now, confirm the loft. You should be greeted with this view:

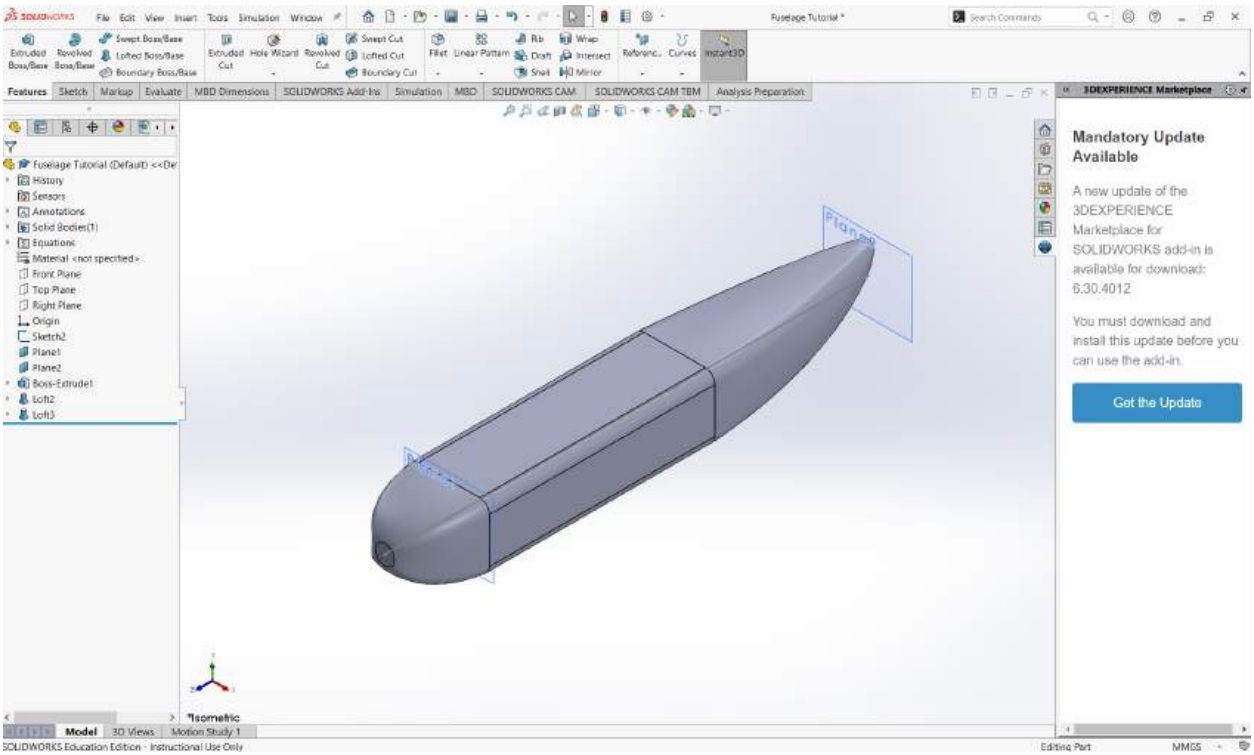


Fantastic!

We did it! This is the complete fuselage. Its isometric view looks very much so like what a standard fuselage should look like:



And what's even better, we made it all with variables! We can change the width of the fuselage by simply going over to Tools → Equations and changing the value of fuse_width. Changing it to 25cm yields us this:



Perfect, a wider body. Much easier than changing it everywhere we input the width, huh?

Comments and Conclusion

There it is: how to CAD a conventional fuselage. The design might not always be like this, of course. Some nuances might arise from different configurations, special payloads, etc. But this is the gist of it, and these are the tools you will find yourself using when making fuselage-like designs.

As always, I hope this was fun, and please don’t hesitate on reaching out for help in any steps. The 3D Sketch part is very complex here and has many little details that might be hard to follow, so if you need, message away.

Thanks for reading through and happy CADing!