



Surface Modeling Tutorial

Complex Surfacing in
SolidWorks

By

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Who is this tutorial for?

This tutorial assumes that you have prior surfacing knowledge as well as a general understanding of modeling concepts and the SolidWorks environment. You will not be walked through the user interface, the location of features on the ribbon or specifics which you should know such as applying dimensions or relations. If you try this tutorial and you lack some basic knowledge your first resource should be the Help menu. If the help menu can not clarify the problem then you should seek help from one of the many forums or online resources.

Good luck!

Step1. Set Up Images



In this case our “side view” isn't a true side view. Its best to find a good front, side and top view BUT in this case we will have to use good judgment and some imagination. This speaker is a JBL Duet and I suggest you do a web search and find the best/largest images you can find to use for reference. We wont be exactly replicating the speaker, but this is a good exercise.

Step1. Set Up Images

Start a new Metric Part.

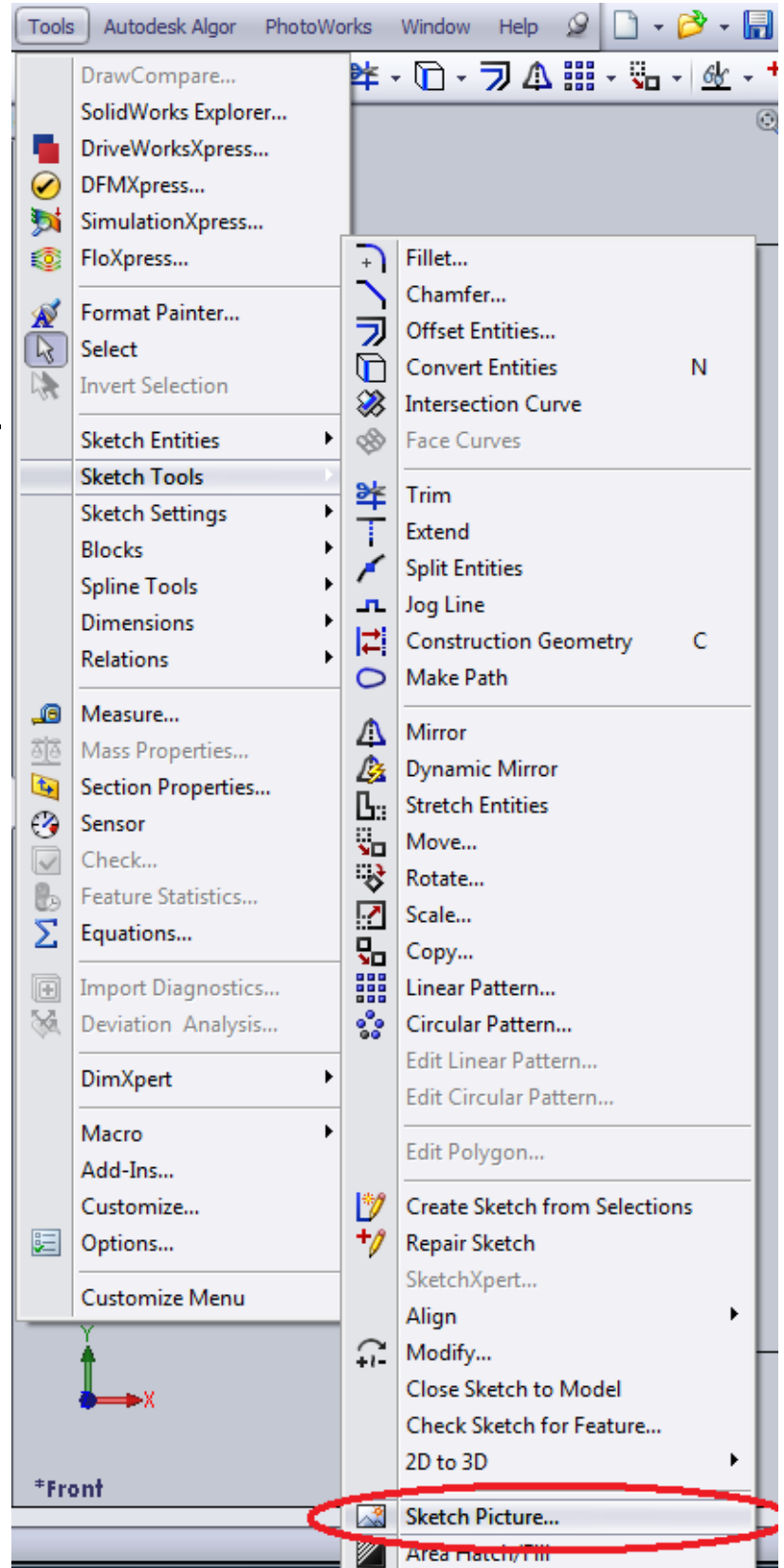
Start a sketch on the Front Plane.

To determine a scale start by drawing a vertical construction line from the origin. Dimension this line to 200mm.

Insert a Sketch Picture:

Tools>Sketch

Tools>Sketch Picture



Step1. Set Up Images

The sketch picture properties box will give you options for scale, location and angle of the picture as well as a few other options such as transparency. You can also grab the control corners of the picture to scale it. Scale and move the picture until its roughly centered on your vertical reference line like the image below.



Step1. Set Up Images

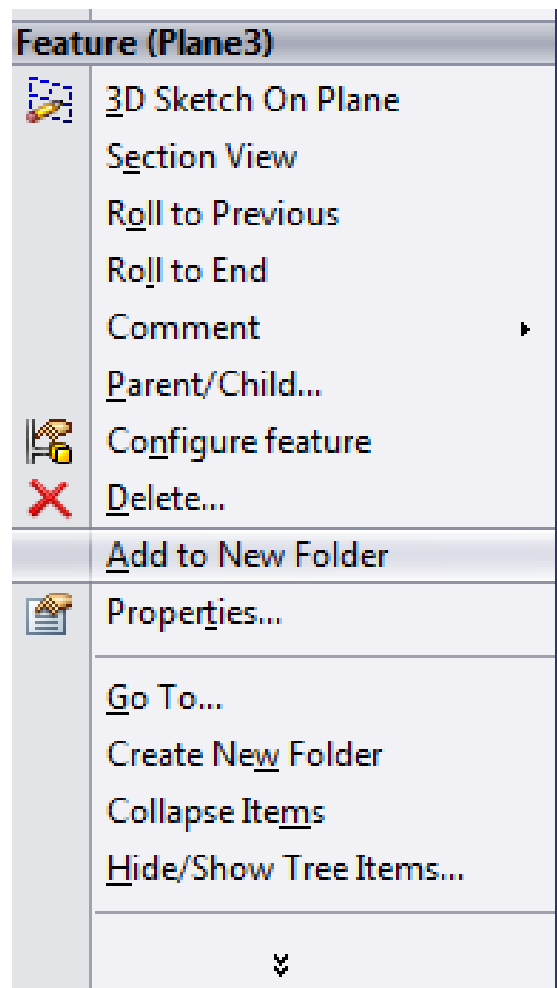
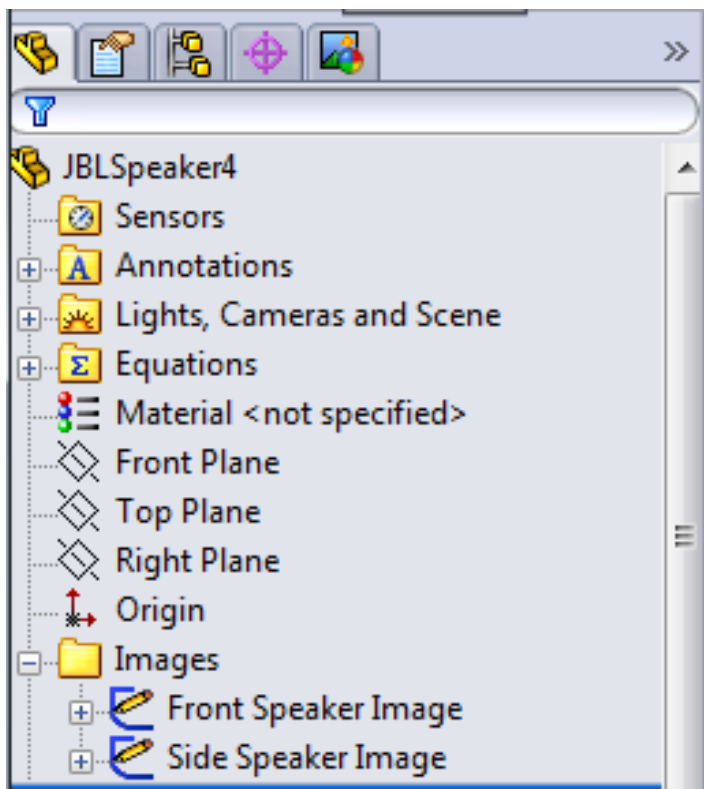
Follow the same procedure and insert a Sketch Picture on the Right Plane. Scale it to be roughly the same size. Remember this isn't a true side picture so we are only using it as a loose reference, but in general you want to get these images/blueprints as close as possible.



Step1. Set Up Images

Now that we have our images in place take this opportunity to organize your feature tree. It's a good idea to name your sketches as well as place them in a folder.

To place sketches or features into a folder simply select them(everything you want in one folder), right click and choose “Add to New Folder”.

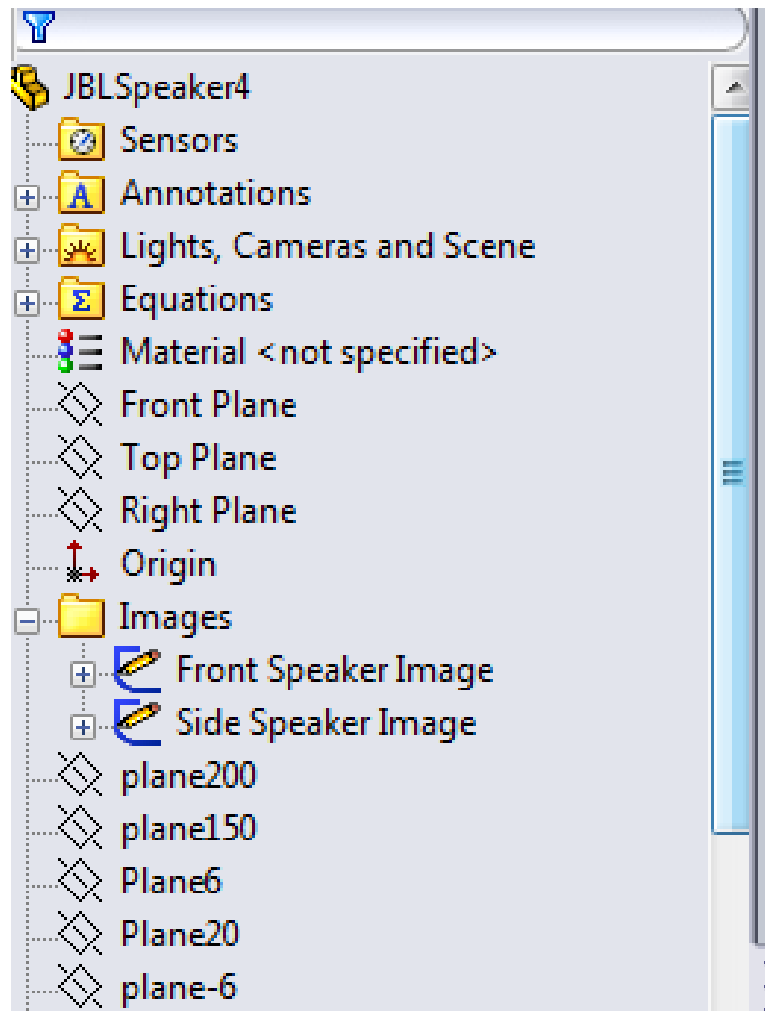


Step2. Setting up work planes

I like to keep any work planes that are based off the Top, Right or Front planes at the top of my feature tree. Since I know all of the planes already we can lay them out now, but during the modeling process you can drag the created planes back up the feature tree to better organize. Its also good to name them something meaningful. In this case I named them their offset amount.

Create 5 planes that are offset from the Top Plane in these amounts: 200,150,20,6,-6.

Naming them in this manner not only helps when selecting a plane for sketching, but when looking at Parent/Child relationships.



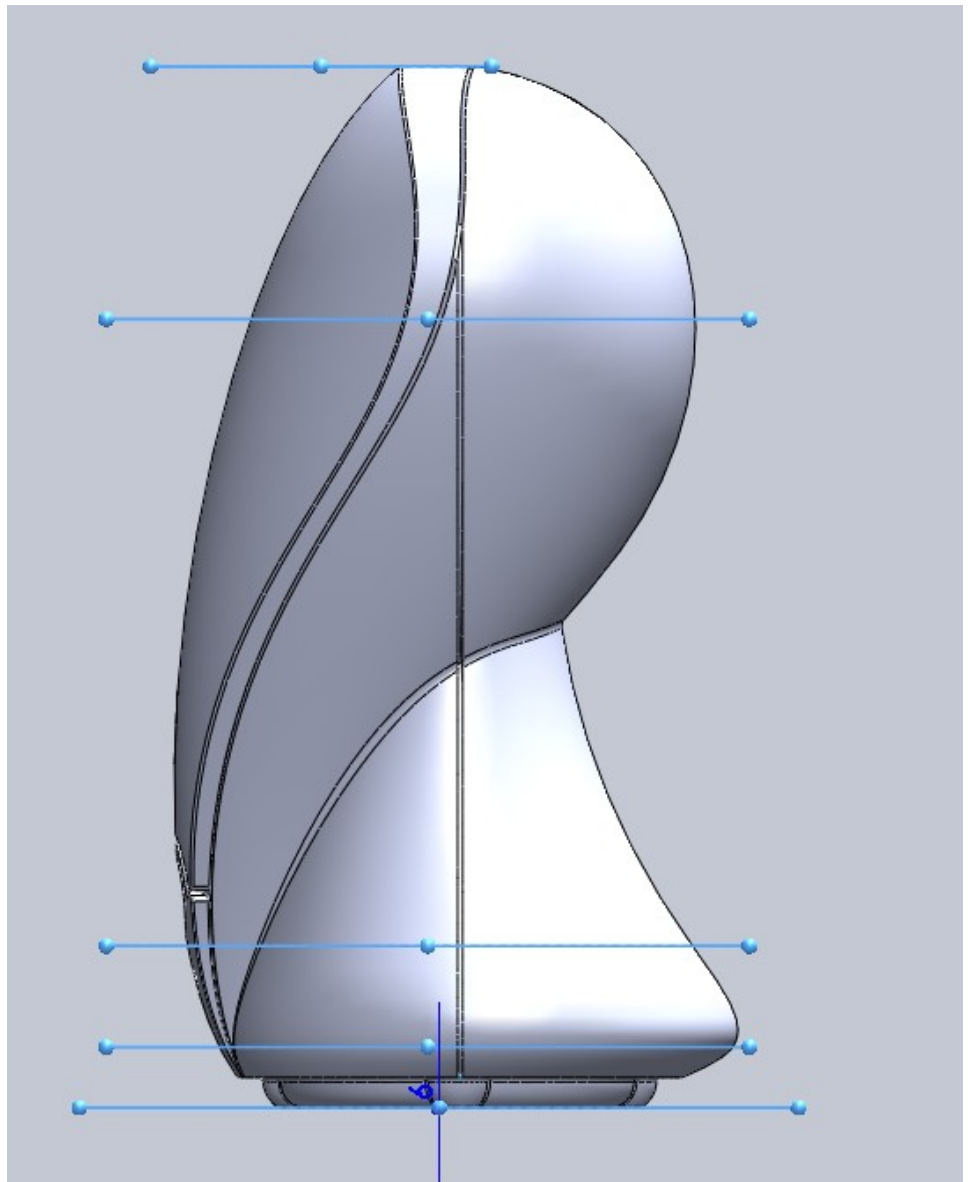
Step2. Setting up work planes

You might be wondering how I came about the Plane offsets. The 200mm plane came from the 200mm line we added to our front sketch. The 150 plane really could have been a wide range. We needed a plane to add a contour for our boundary surface. For whatever surface you are drawing you have to plan it out by looking at the shape of your part and planning a good location for

guides. Same thing for the 20mm offset.

The -6mm offset plane was a decision for the location of the “feet” for the speaker.

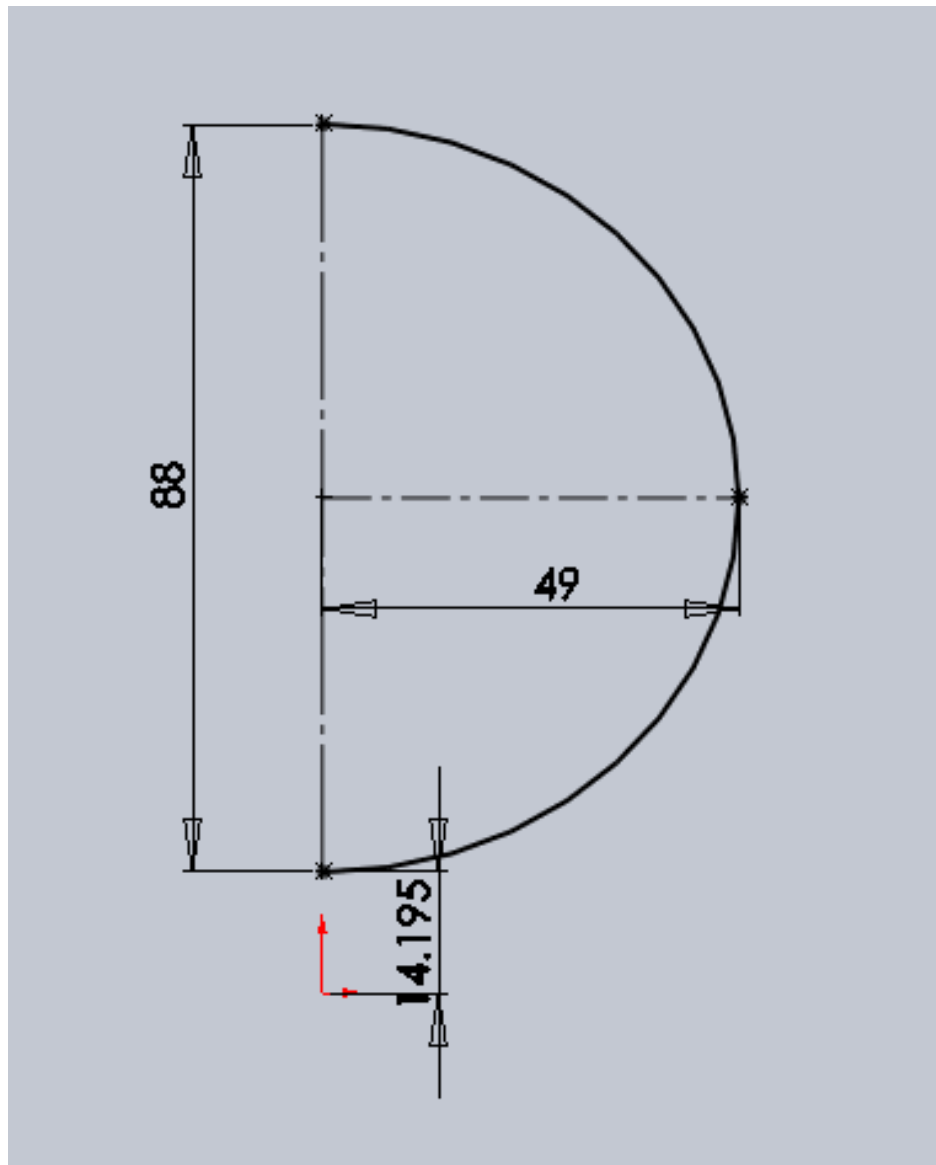
When you plan this out you may move planes around as needed.



Step3. Creating a Lofted Surface

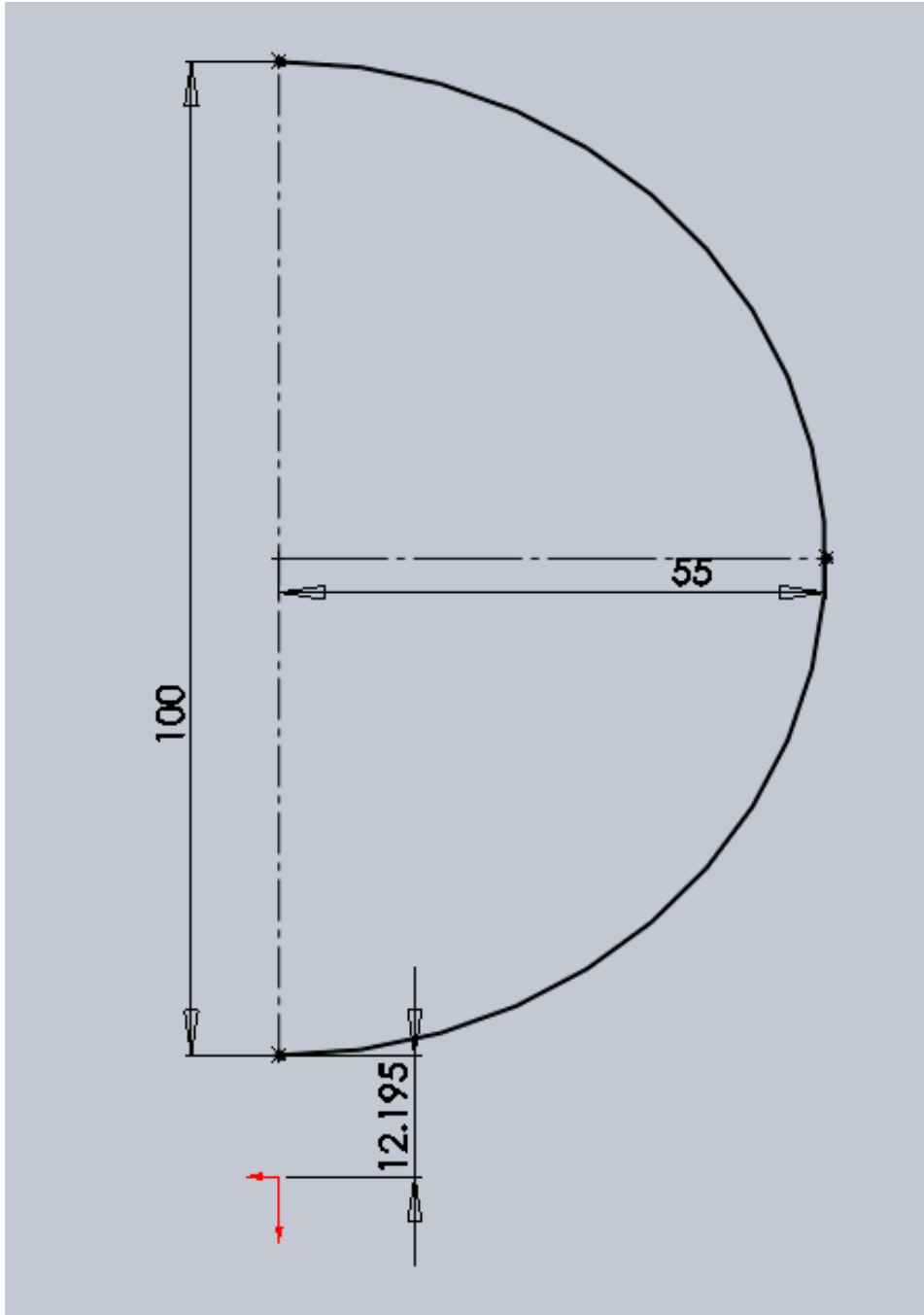
On your Top Plane create a sketch as shown below.

First Create the construction lines followed by an ellipse while snapping to the line intersection and end points. Then trim the ellipse so you are left with the right half(in the positive X direction)



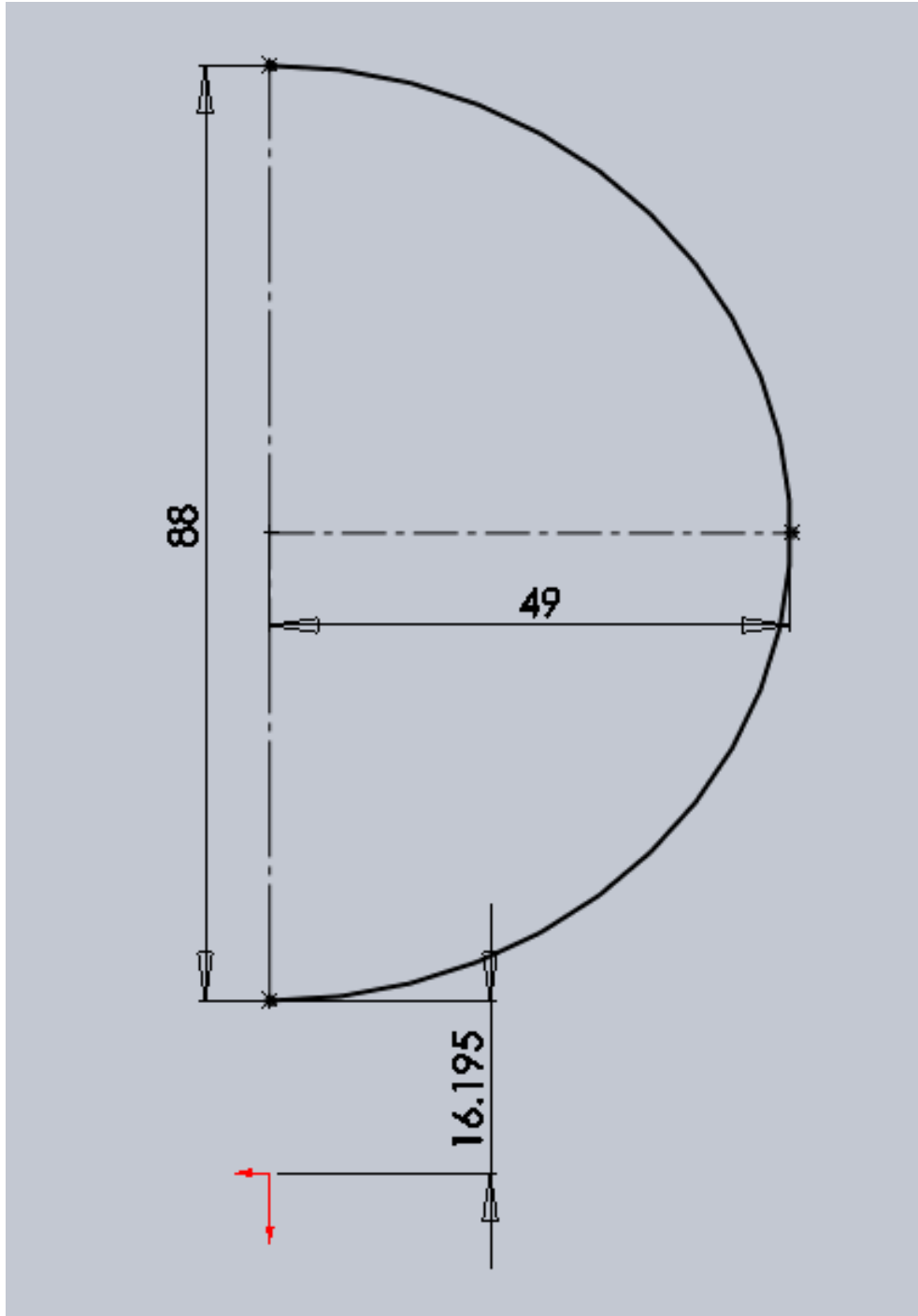
Step3. Creating a Lofted Surface

On Plane6(6mm offset in the positive direction) draw a similar sketch with the dimensions shown.



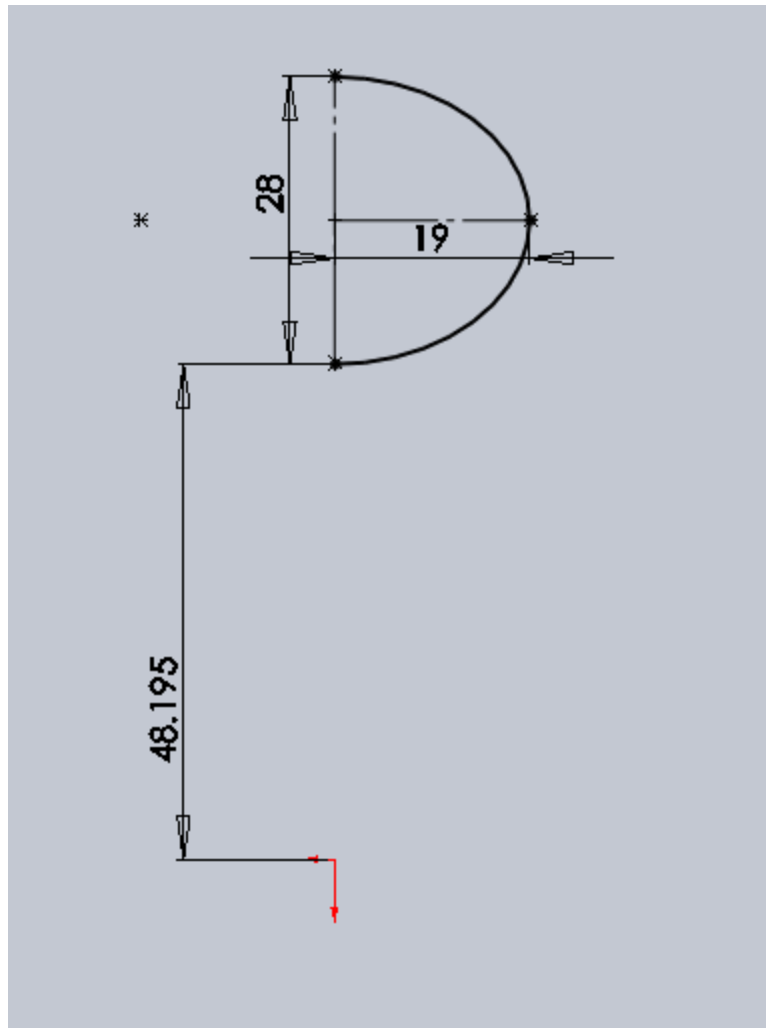
Step3. Creating a Lofted Surface

On Plane20(20mm offset in the positive direction) draw a similar sketch with the dimensions shown.



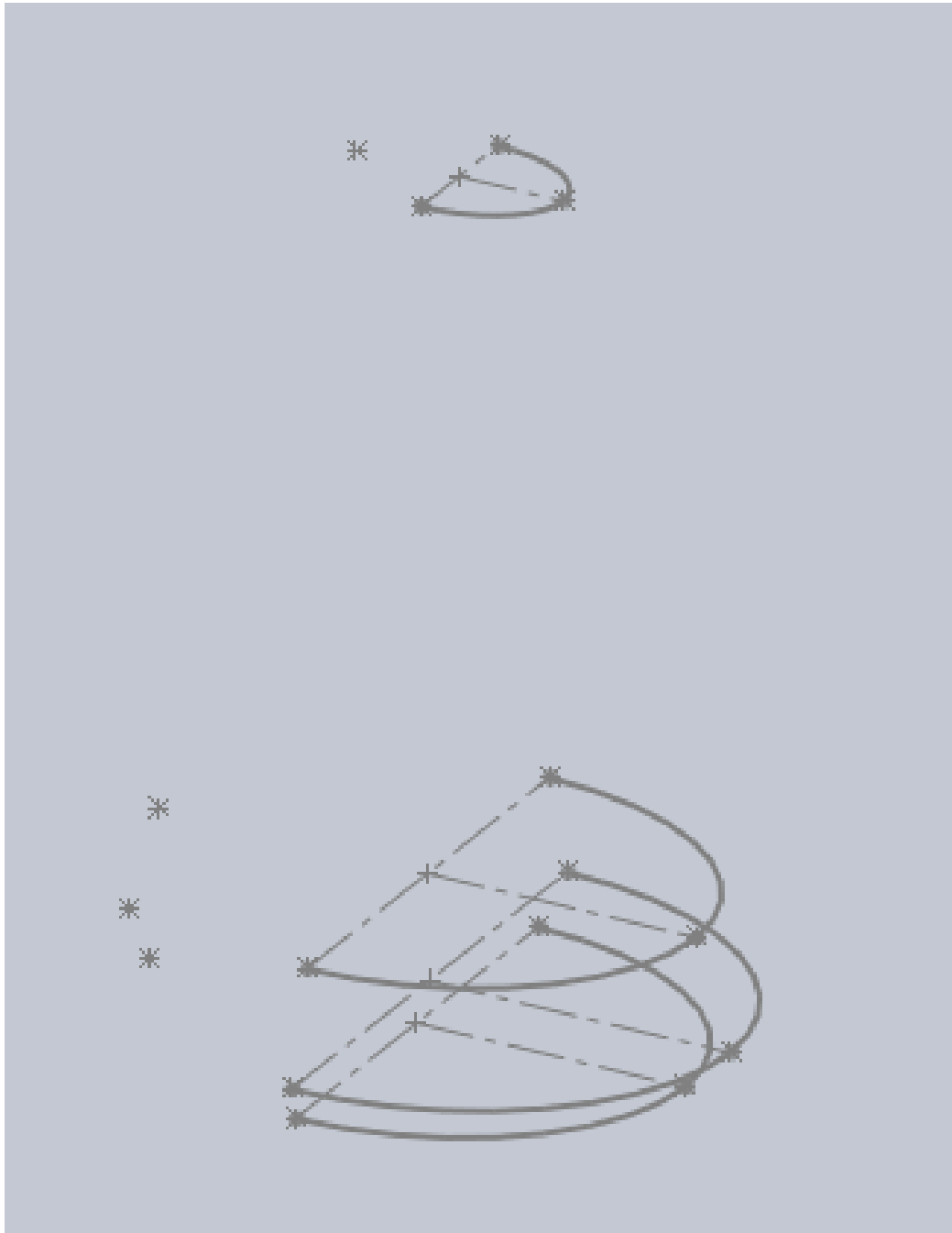
Step3. Creating a Lofted Surface

On Plane150(150mm offset in the positive direction)
draw a similar sketch with the dimensions shown.



Step3. Creating a Lofted Surface

You should now have a skeleton of sketches that will setup our first loft.



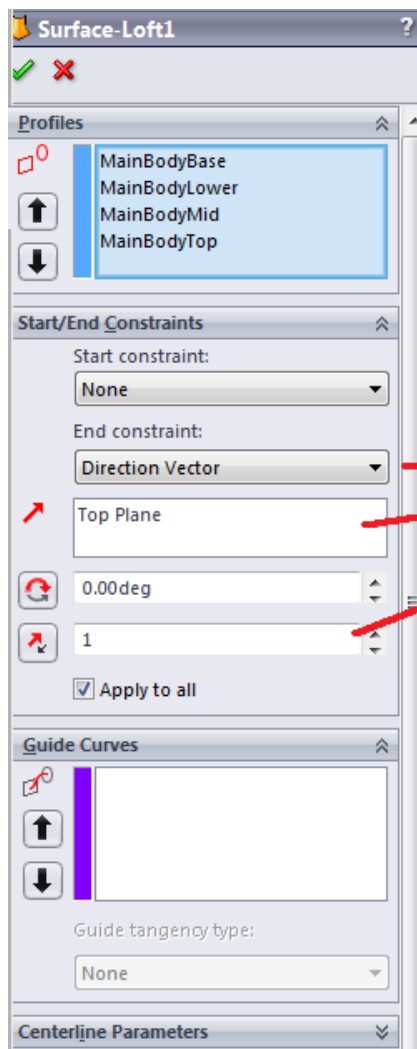
Note: notice all sketches were fully defined.

Step3. Creating a Lofted Surface

Again it's a good idea to name your sketches. Mine are named MainBodyBase, Lower, Mid and Top. This will help you out later if/when you need to modify things.

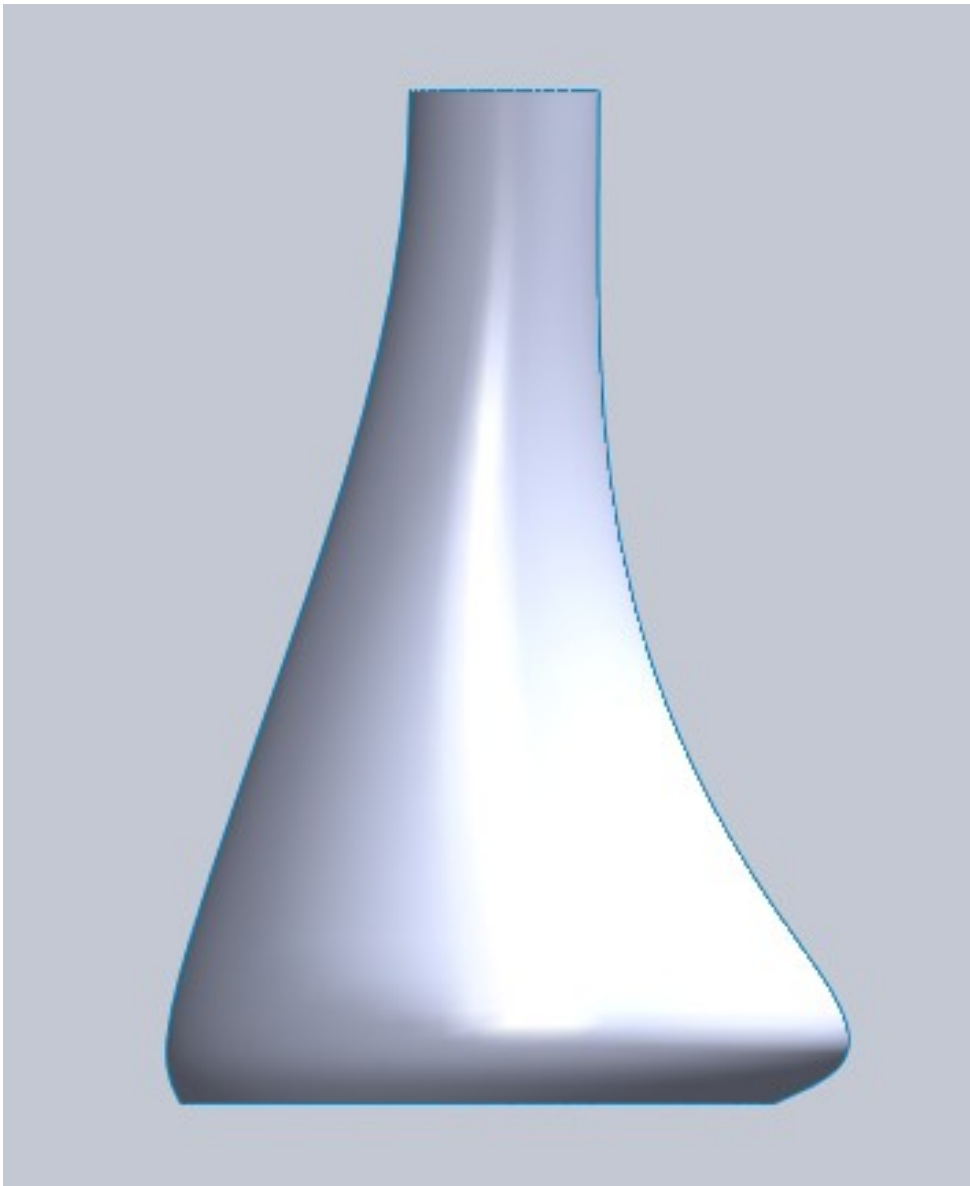
Start a Lofted Surface and select each sketch as your Profiles starting from Bottom to Top. In the Start/End constraints section change End Constraint to “Direction Vector” and select your Top Plane for the reference.

Note: Tangent Length Is 1 but this can be helpful to control lofted shapes without the use of guide curves.



Step3. Creating a Lofted Surface

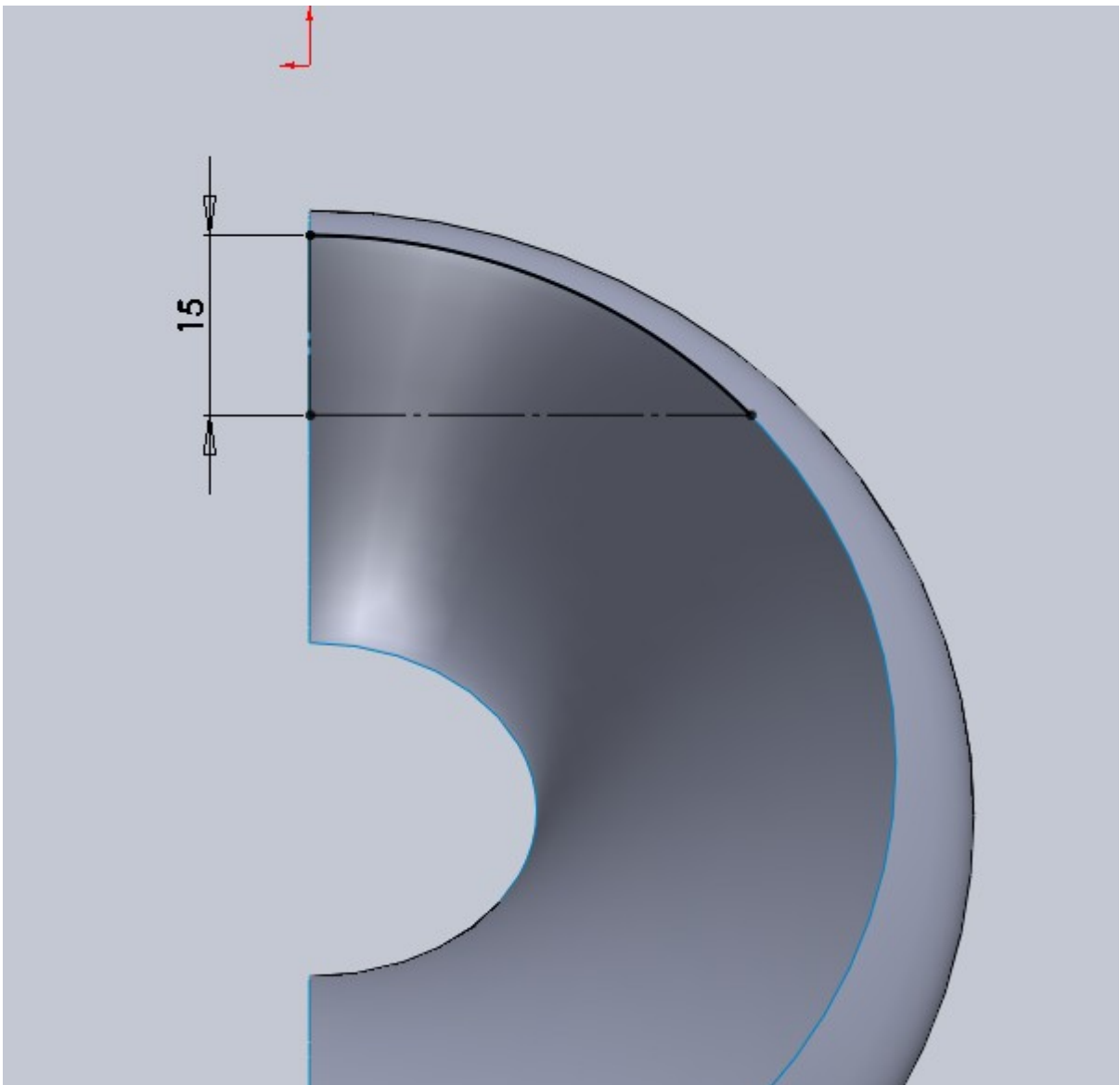
This lofted surface makes up half of the base shape. There are other ways to achieve this shape so don't limit yourself in the future. Play around with guide curves, tangent lengths and directions and so on.



Step4. Creating the Speaker Face

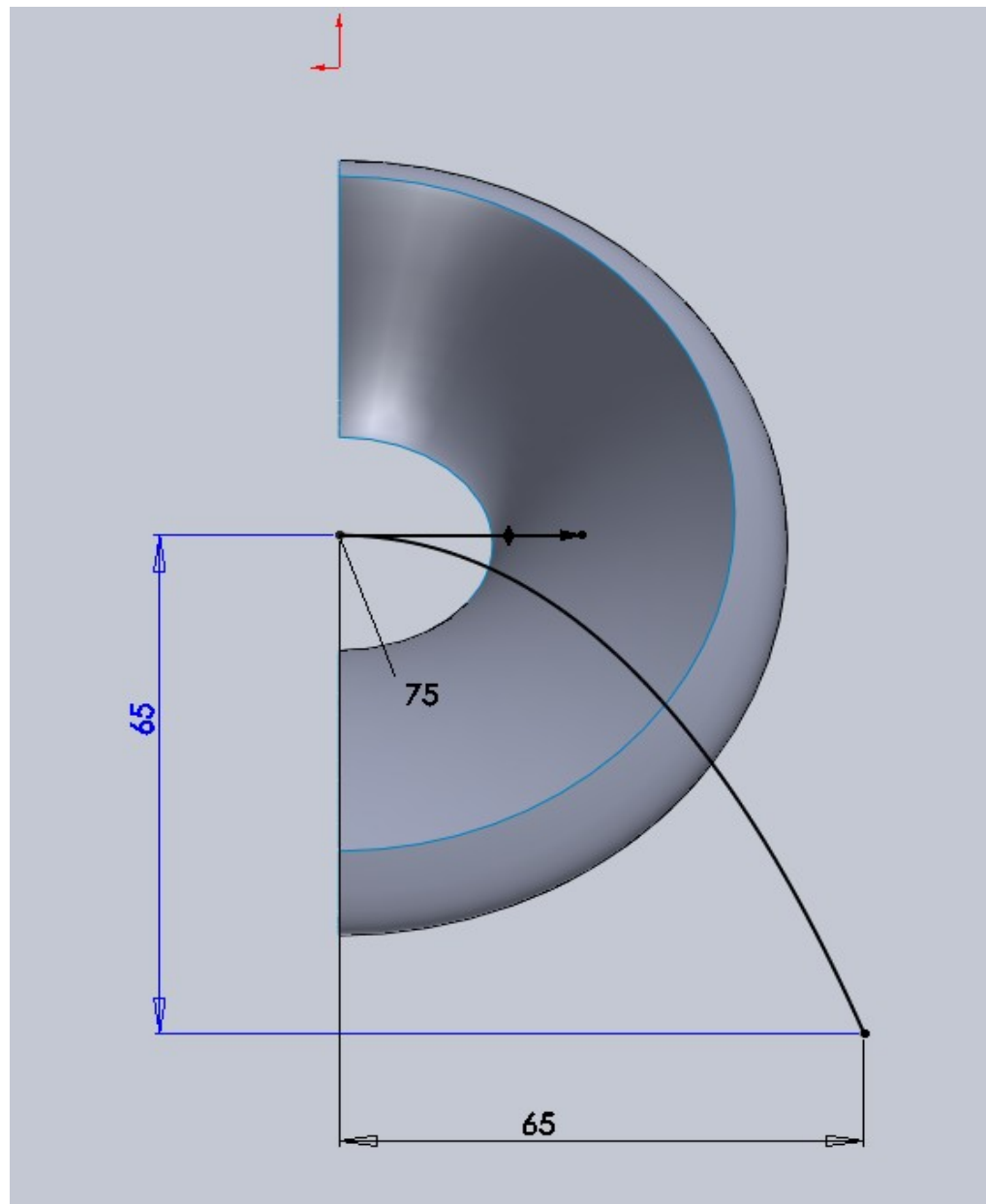
Start a sketch on the Top Plane.

Draw two construction lines with the dimensions shown that intersect the loft base. Use Convert Entities on the loft base edge and trim it so you are left with only the part inside the two construction lines as shown.



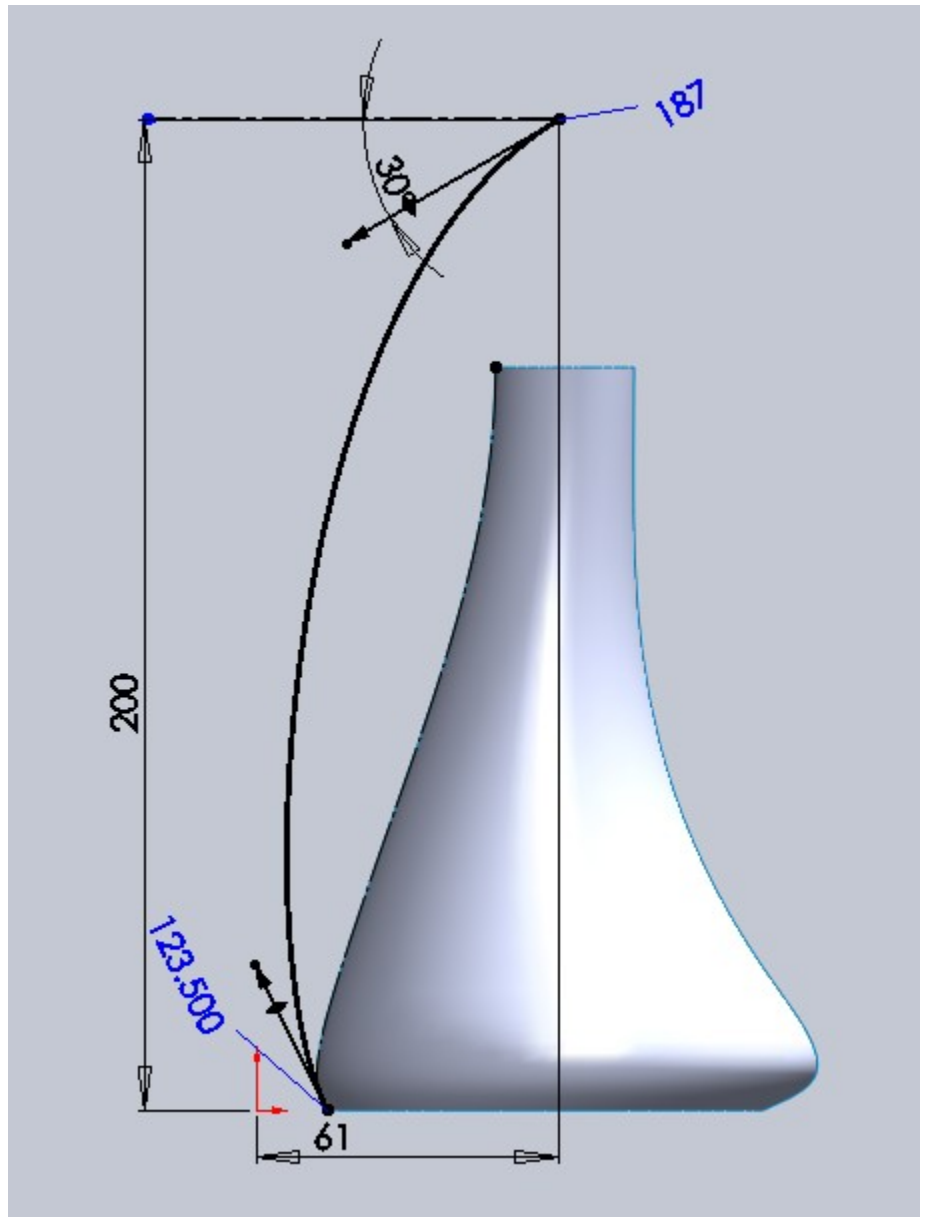
Step4. Creating the Speaker Face

Start a sketch on Plane200 and draw a spline as shown using only a start and end point. The spline is coincident with the center of the top oval. Its handle has a horizontal relation as well as a 75mm dimension for its tangent length. The bottom right end of the spline is dimensioned as shown and its handle was never adjusted.



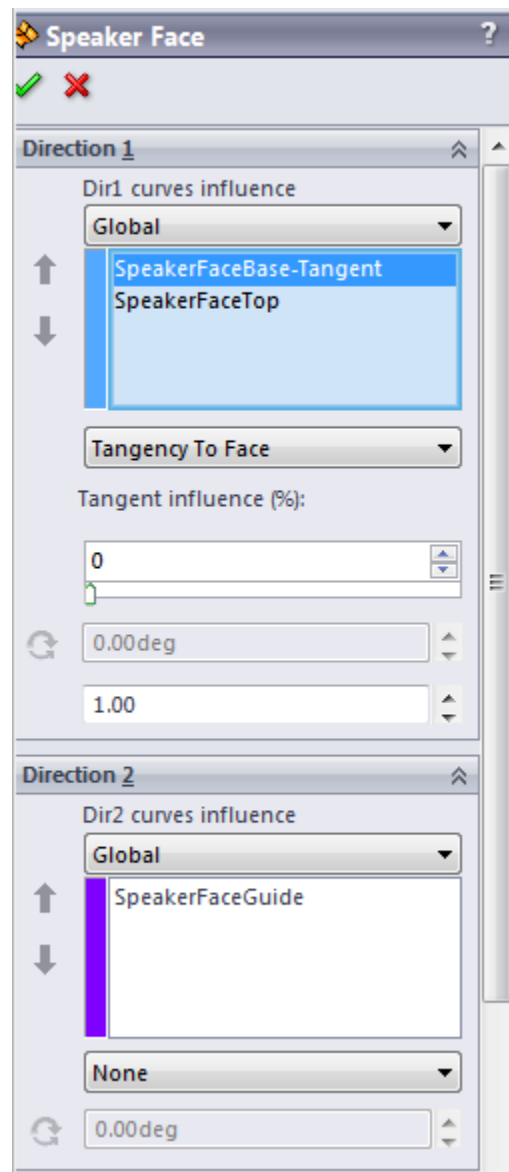
Step4. Creating the Speaker Face

Start a sketch on the Right Plane and draw a spline as follows. Spline is coincident with both sketches just created. The top point also has a horizontal construction line so it can be dimensioned 30degrees below as well as having 187mm tangent length. Use Convert Entities on our loft front edge and give the spline a Tangent relation as well as 123.5 tangent length dimension. Make sure your sketches are fully defined! The 200 in this image should not be necessary if you are coincident to our previous sketches. Its merely there for reference on the image.



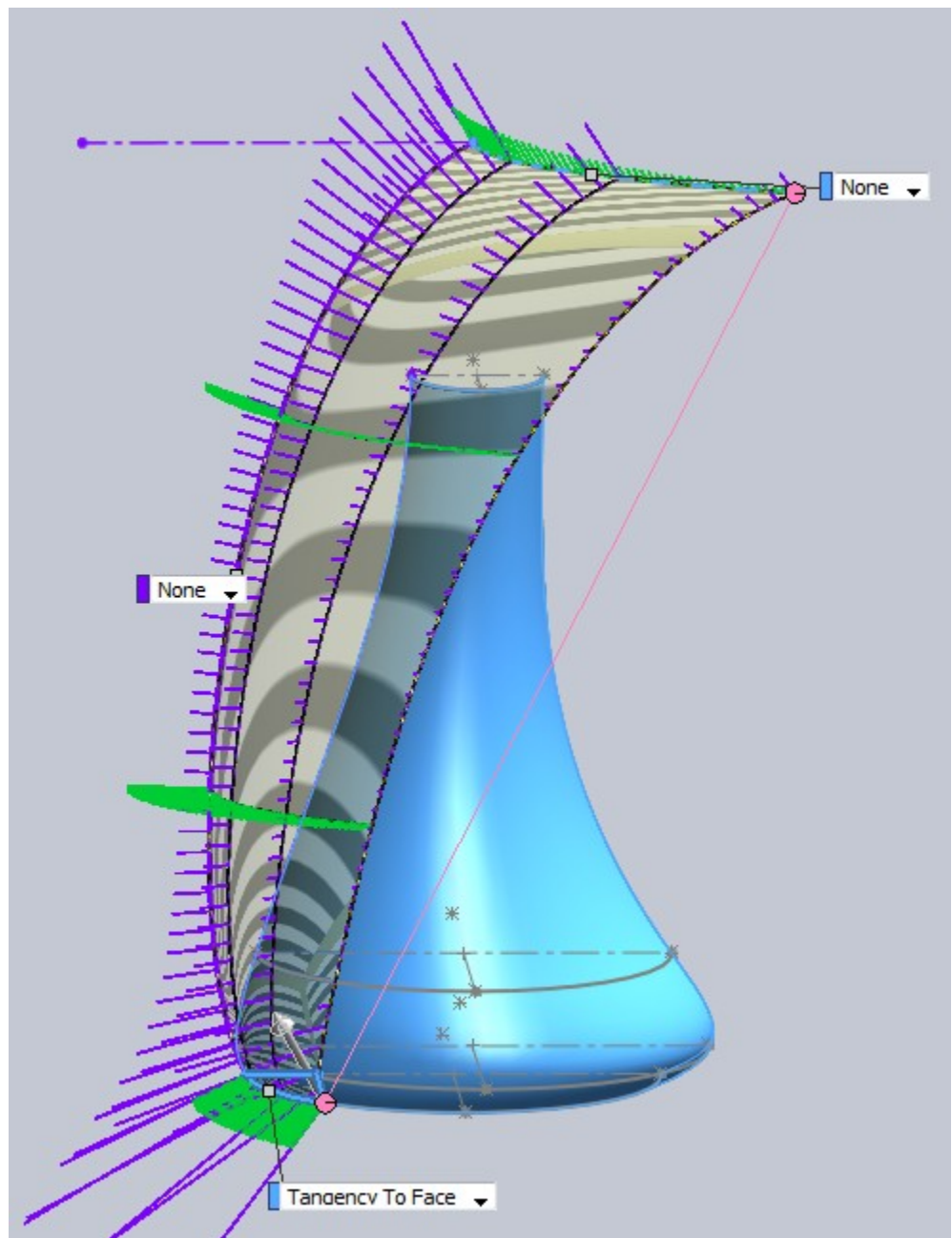
Step4. Creating the Speaker Face

Create a Boundary Surface using our sketch on the top plane and Plane200 as Direction1 and our sketch on the Right plane as Direction2. For Tangency Type, on the bottom sketch place a Tangency to Face. This will ensure our boundary surface is tangent to our lofted surface at the base. This is needed for our speaker to look right.



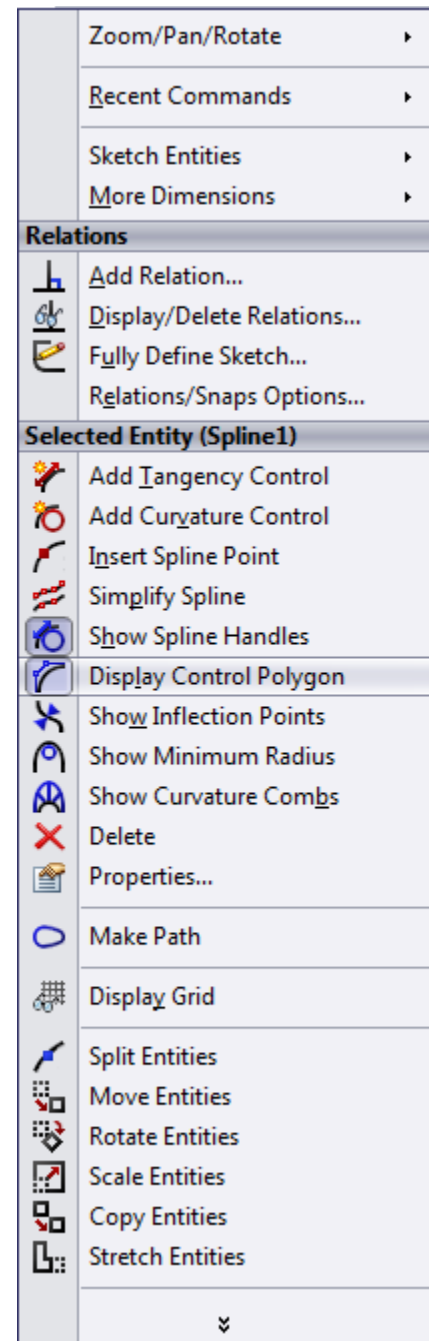
Step4. Creating the Speaker Face

One thing you will need to make use of in the display section of the Boundary properties box is Zebra Stripes and Curvature Combs. This are useful to show you (in the preview) any major problems with the geometry. You want your curvature combs to have a decent transition and not jump from long to short very quickly.



Step4. Creating the Speaker Face

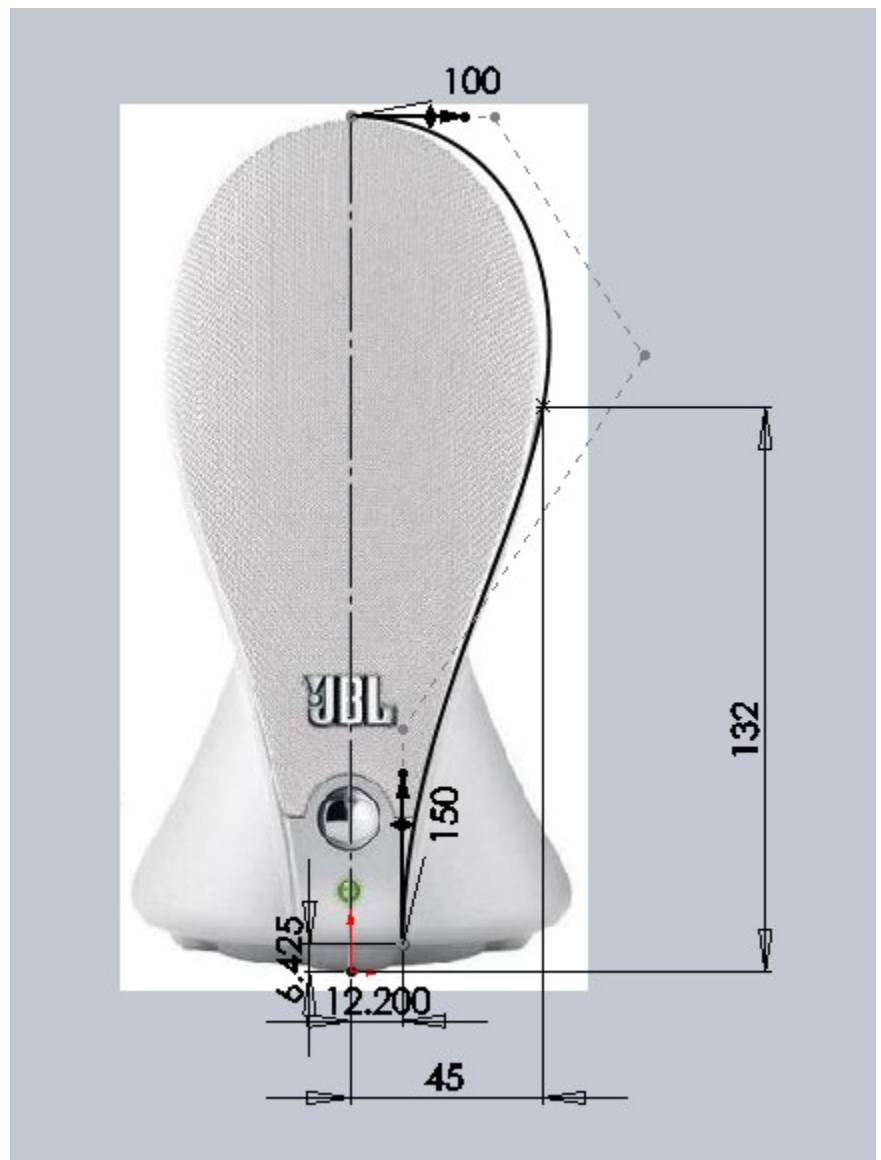
Start a new sketch on the Front Plane and turn on the visibility of our Front Speaker Image. You will need to create a spline with a control point and its end points. As a note I like to Right Click on the spline and Display Control Polygon.



Step4. Creating the Speaker Face

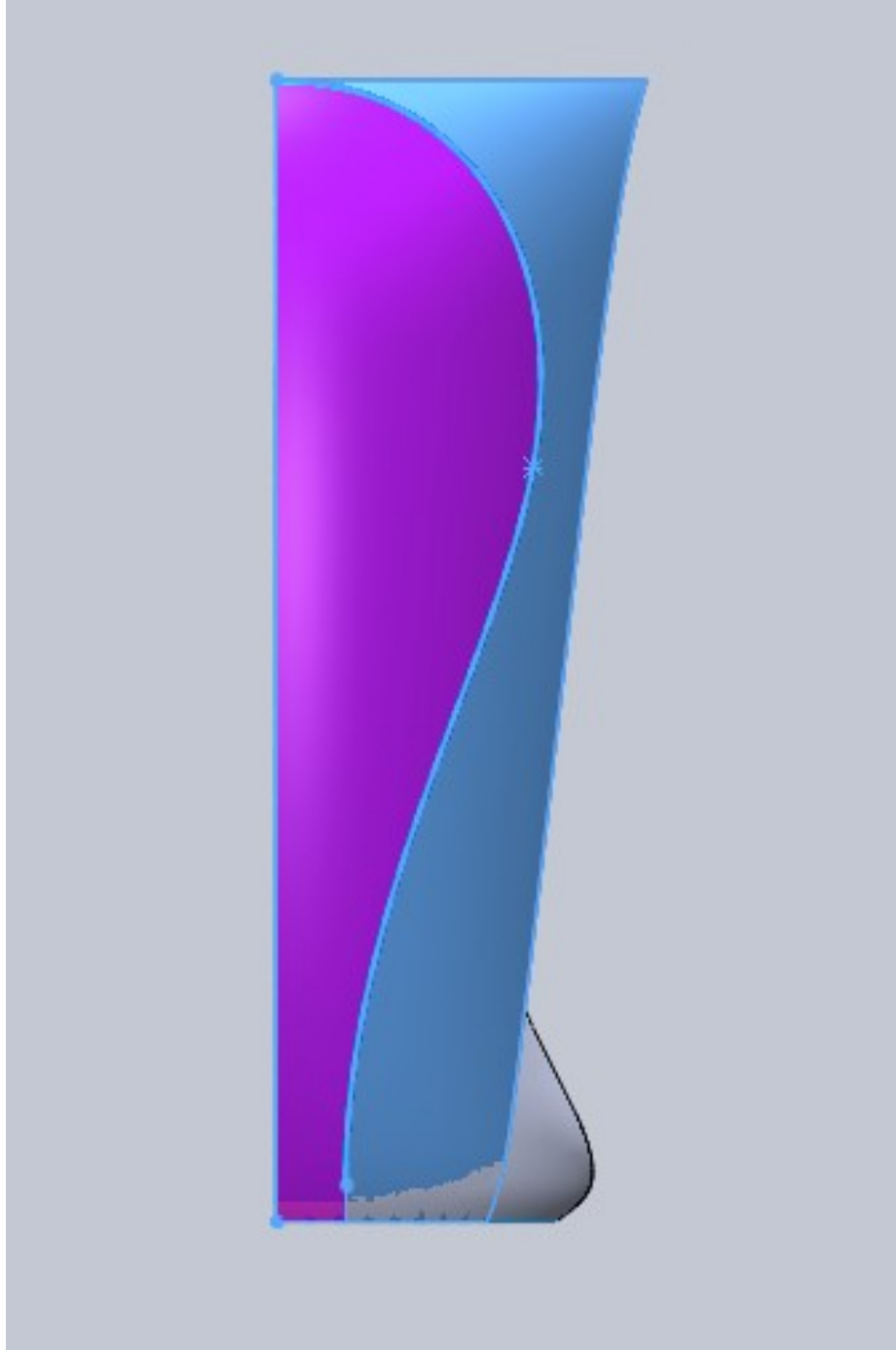
For the spline remember these images are only for reference and they have perspective so don't be afraid to deviate from the shape slightly to achieve a better end result. You will see the spline doesn't follow the speaker exactly.

Make sure the top spline handle has a horizontal relation and is coincident with the vertical line end point on our image sketch and that the bottom spline handle has a vertical relation.



Step4. Creating the Speaker Face

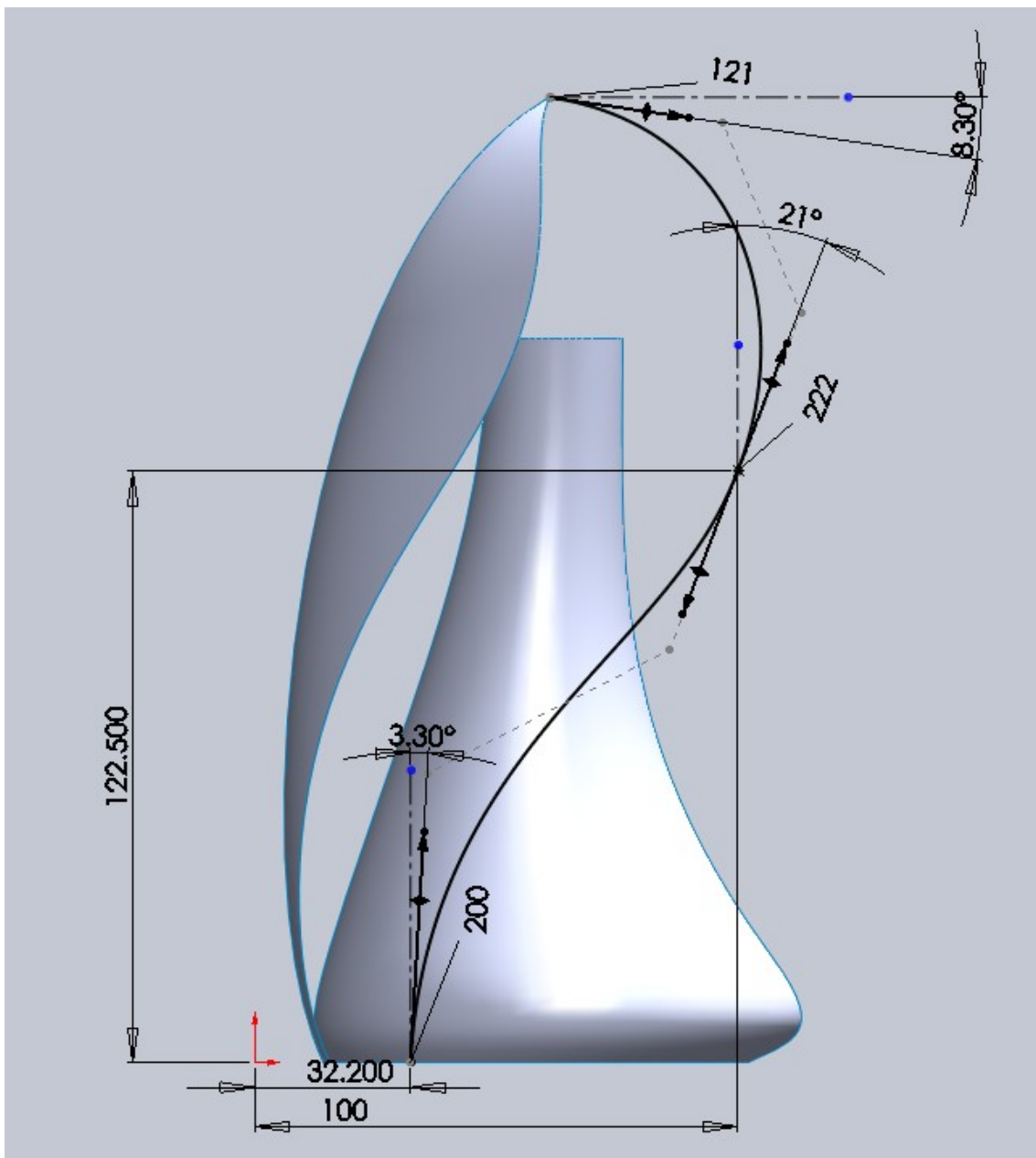
Use this sketch to Trim Surface and remove the blue section from the image below.



Step5. Creating the Speaker Body

Start a sketch on the Right Plane.

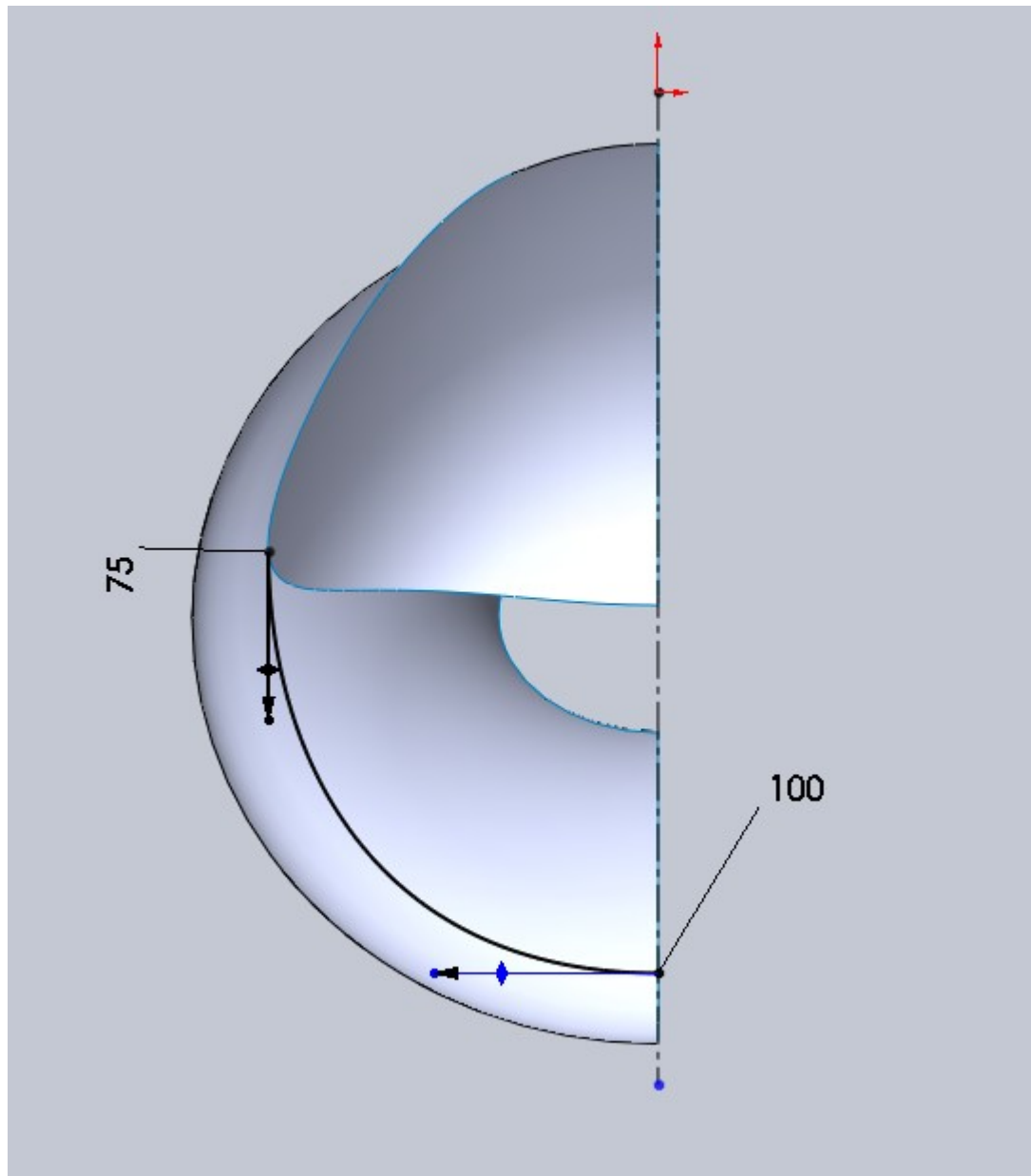
Recreate this spline making note that the lower point has a horizontal relation with the origin and the upper point is coincident with the end point of the speaker face surface we just created.



Step5. Creating the Speaker Body

Start a new sketch on Plane150.

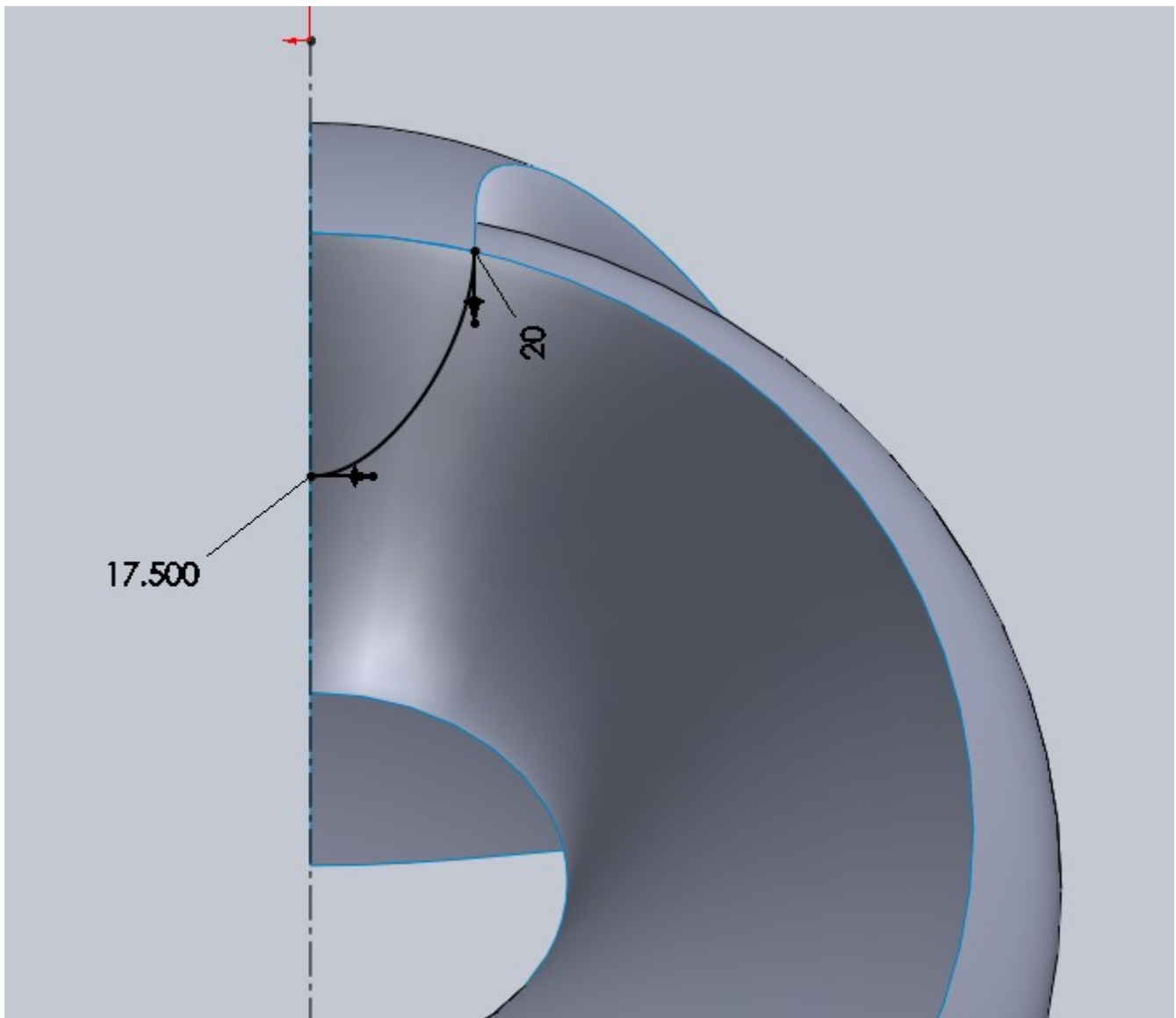
Create a spline as shown. You can create a vertical construction line or simply add a vertical relation between your spline endpoint and the origin. The spline has vertical and horizontal relations on the tangency handles.



Step5. Creating the Speaker Body

Start a sketch on the Top Plane.

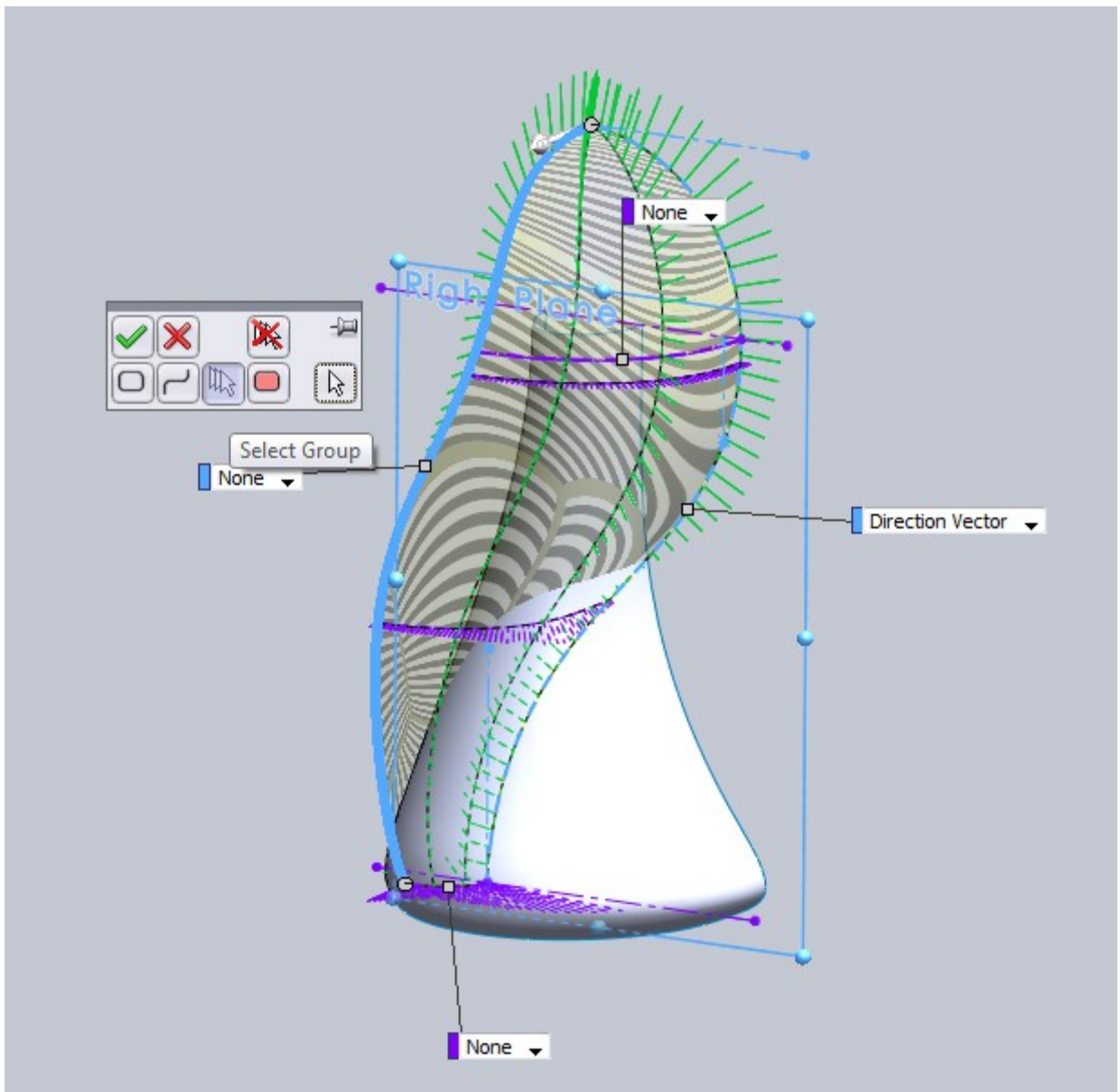
Create a spline as shown. Spline handles have horizontal and vertical relations as well as dimensions. Again you can create a vertical construction line from the origin or use a vertical relations for the spline end



Step5. Creating the Speaker Body

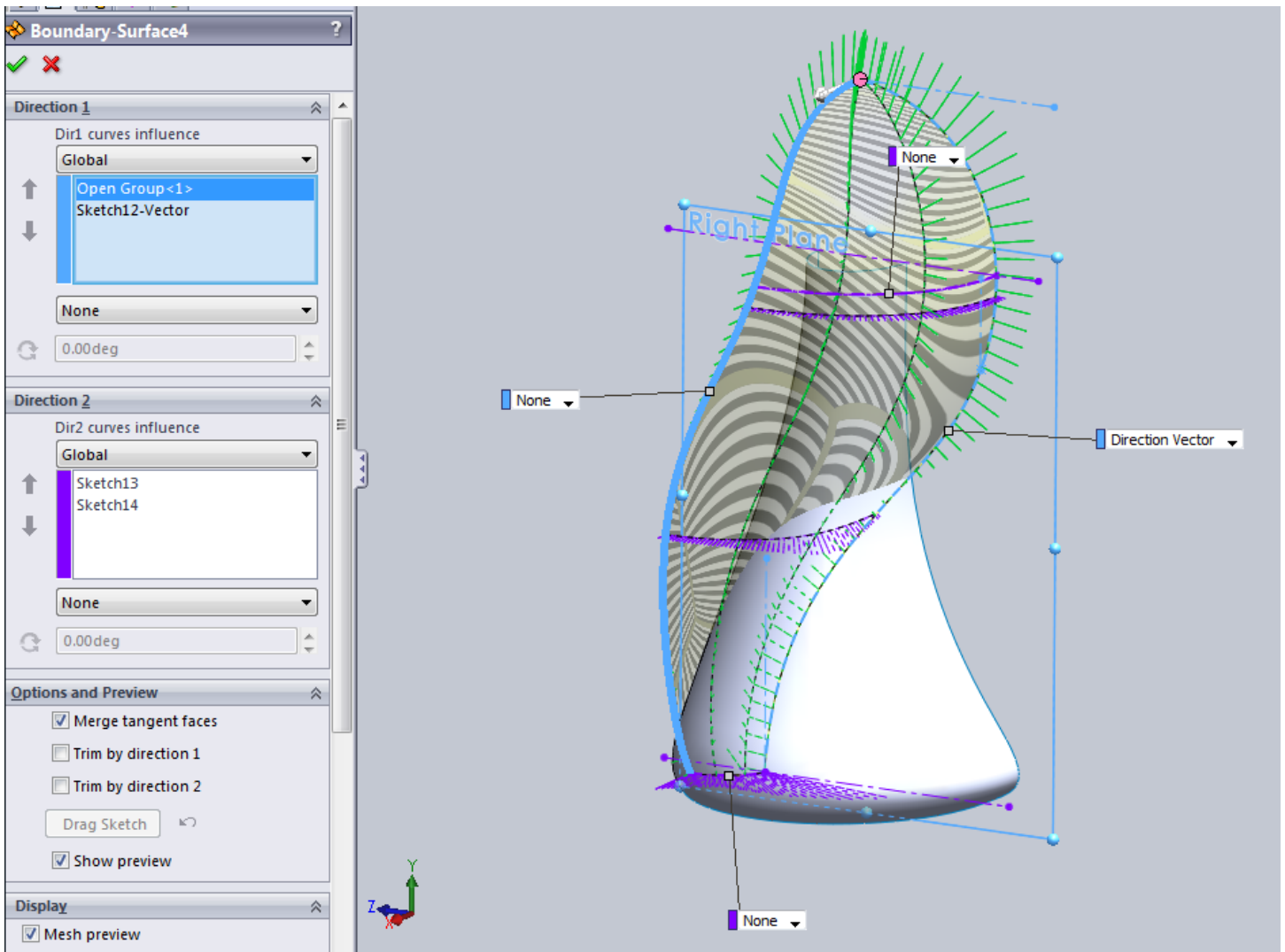
Create a boundary surface.

Select the boundary surface edge for your first curve. Depending on your trim and geometry this may have more than one section so be aware of this. If it does make sure you Right Click and use the selection man-



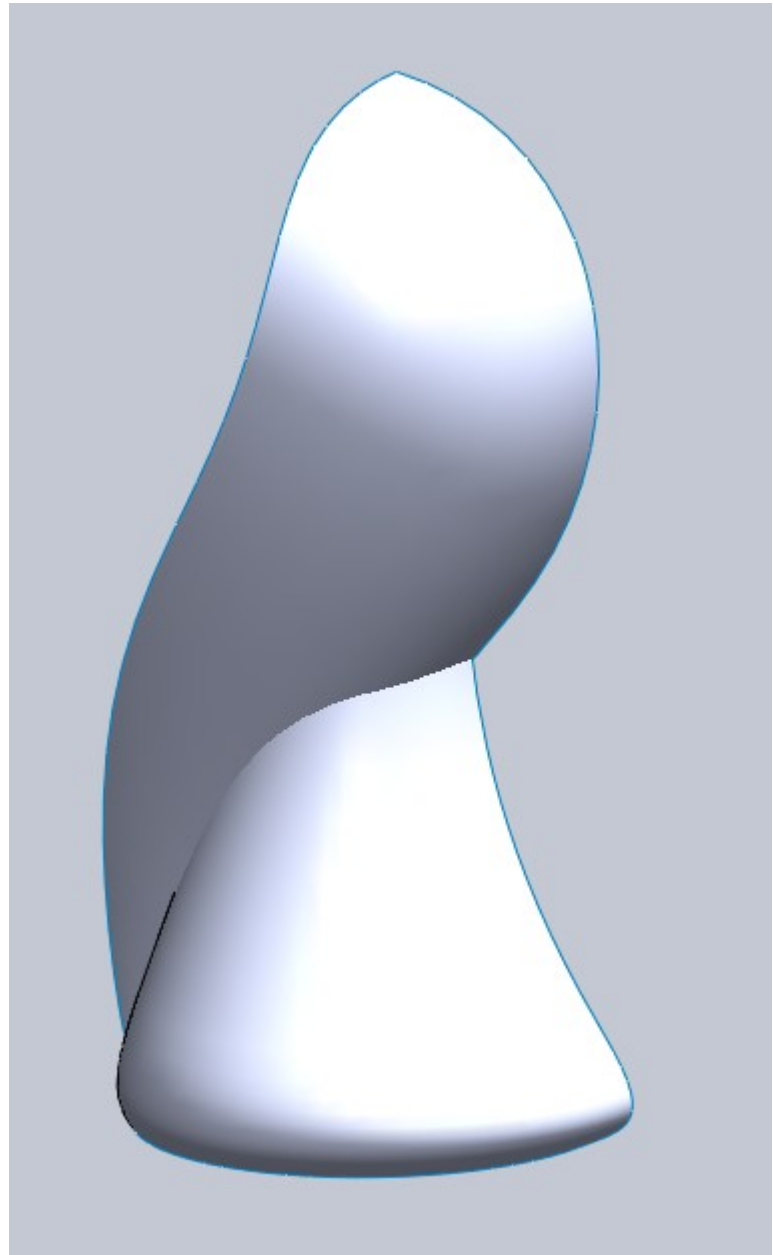
Step5. Creating the Speaker Body

Select our Right Plane sketch for our second curve for Direction1. In the Tangent Type drop down select Direction Vector. Select the Right Plane as your reference and give it a 45% influence. For Direction 2 curves select our top plane sketch and our plane150 sketch.



Step5. Creating the Speaker Body

Some clarification. The reason we chose to add a direction vector influence is because even though we have guide curves that we ensured were horizontal, they will not control the entire boundary surface. Adding this direction influence to the surface will ensure that when we mirror our body we can avoid a visible Ridge at the back.

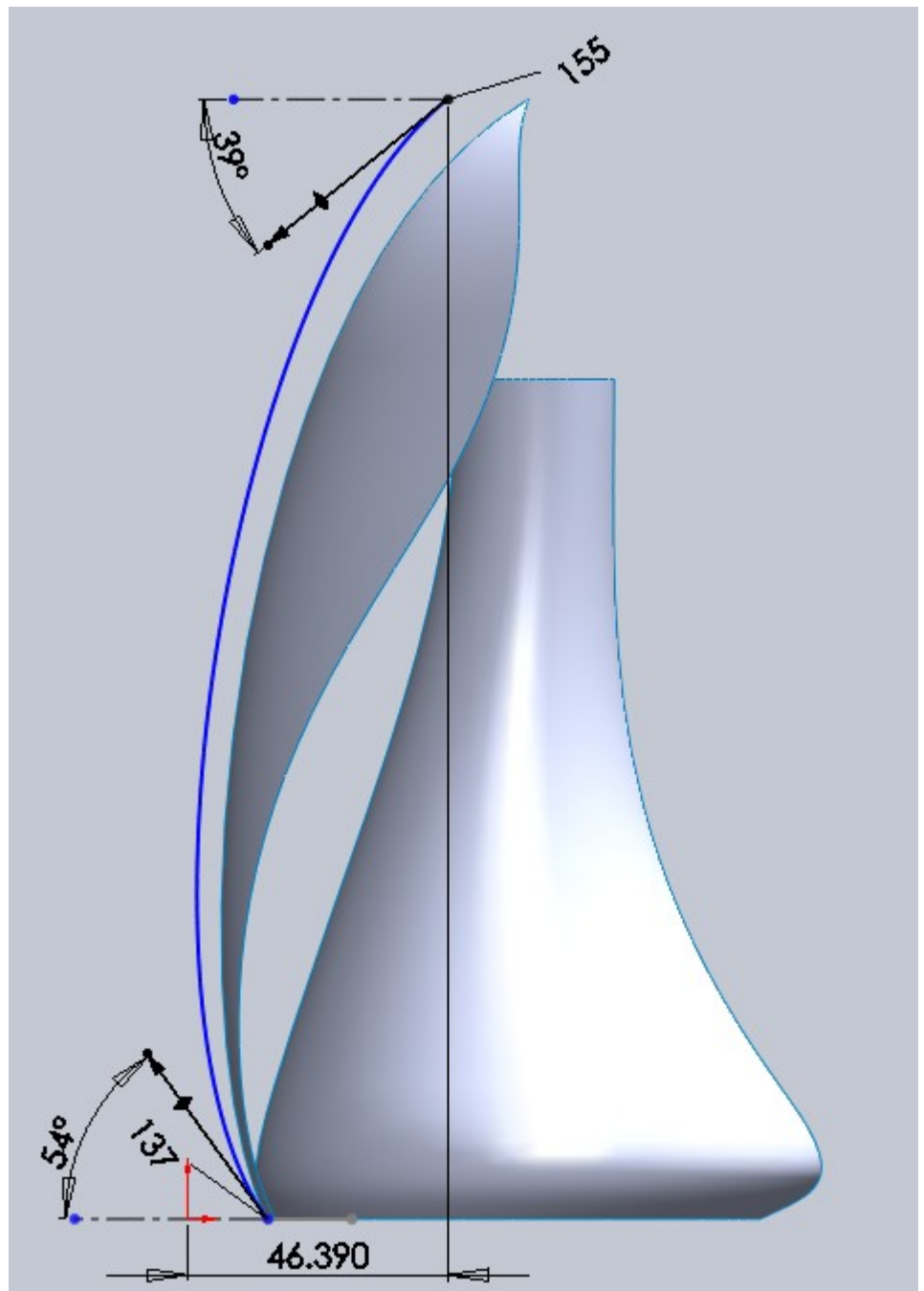


Step6. Creating a second speaker face.

Create a sketch on Right Plane.

Draw a spline that is coincident to the bottom of the speaker face and coincident at the top with Plane200. Add the dimensions shown.

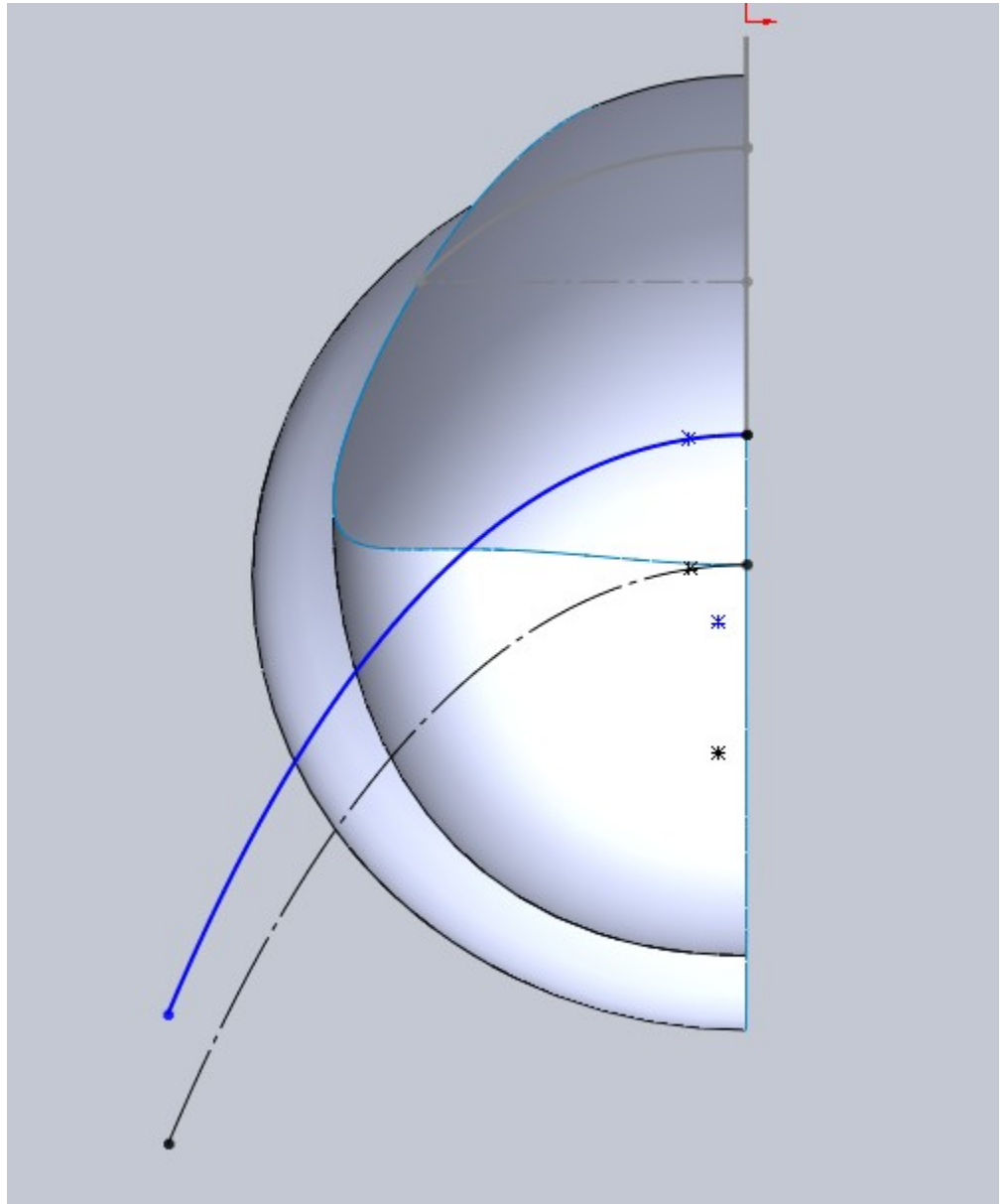
You may be wondering where some of these numbers are coming from like 46.390. In cases where you are dealing with complex shapes its often easier to draw a curve so it looks right and add the dimension after to locate it. That is the case with some of these.



Step6. Creating a second speaker face.

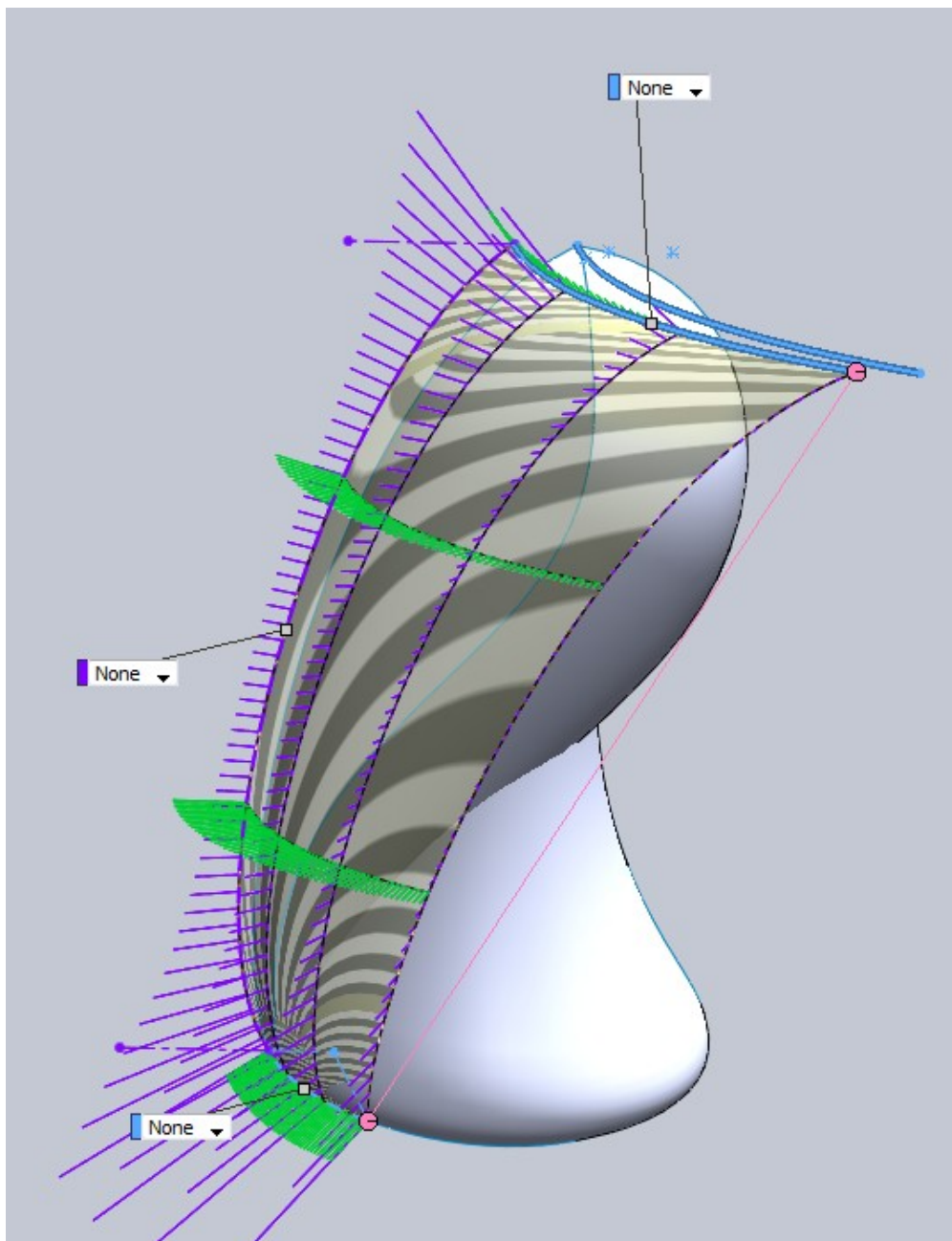
Create a sketch on Plane200.

Make the sketch used for the top edge of the speaker face boundary surface visible. What we are doing is creating a surface that will be offset to create the outer face of the speaker. Draw a spline similar to the one shown making it coincident to the previous sketch.



Step6. Creating a second speaker face.

Create a Boundary Surface using the bottom sketch from our first speaker face and our new top profile as the Direction1 curves. Use our Right Plane sketch spline as Direction2. Again pay attention to the curve combs in the preview.

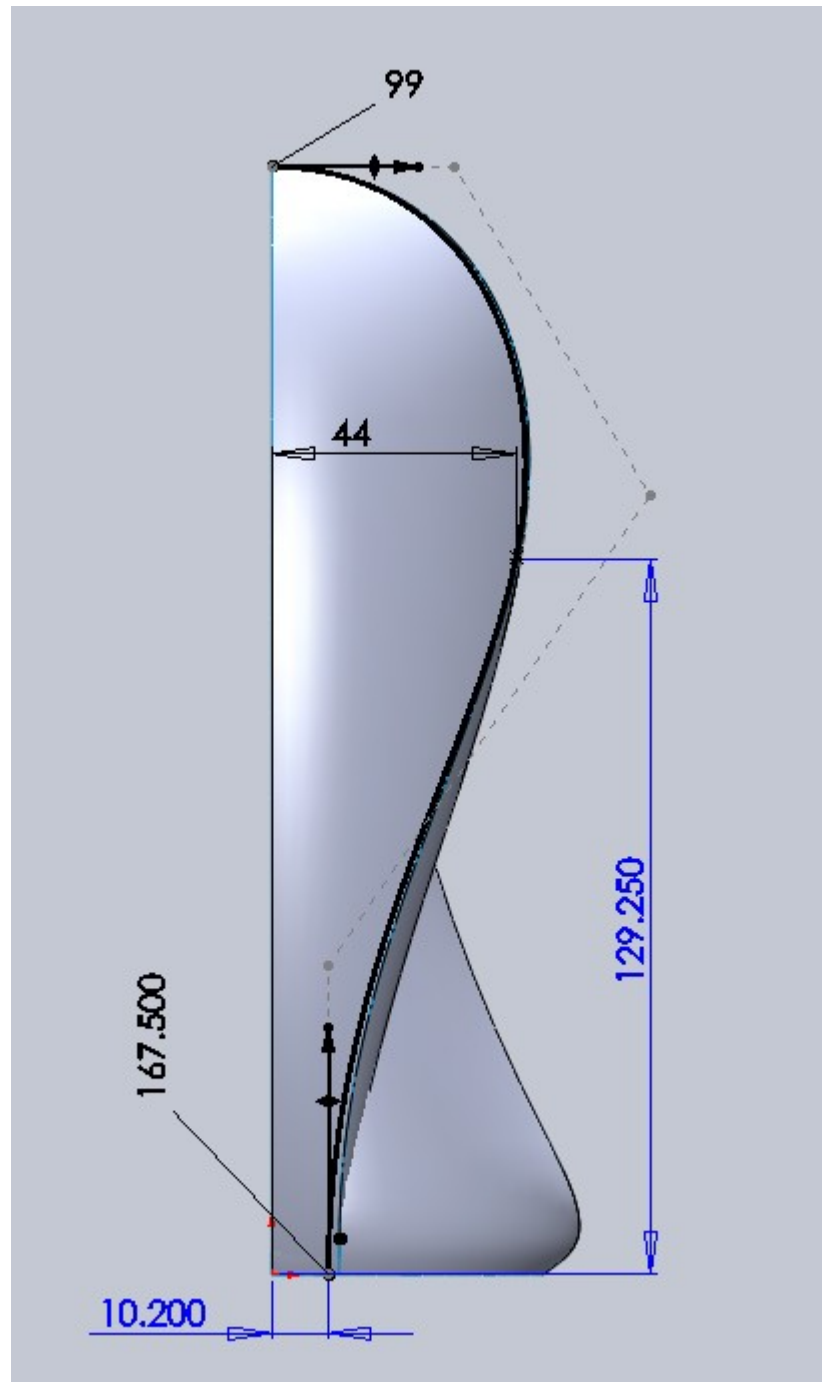


Step6. Creating a second speaker face.

Create a sketch on the Front Plane.

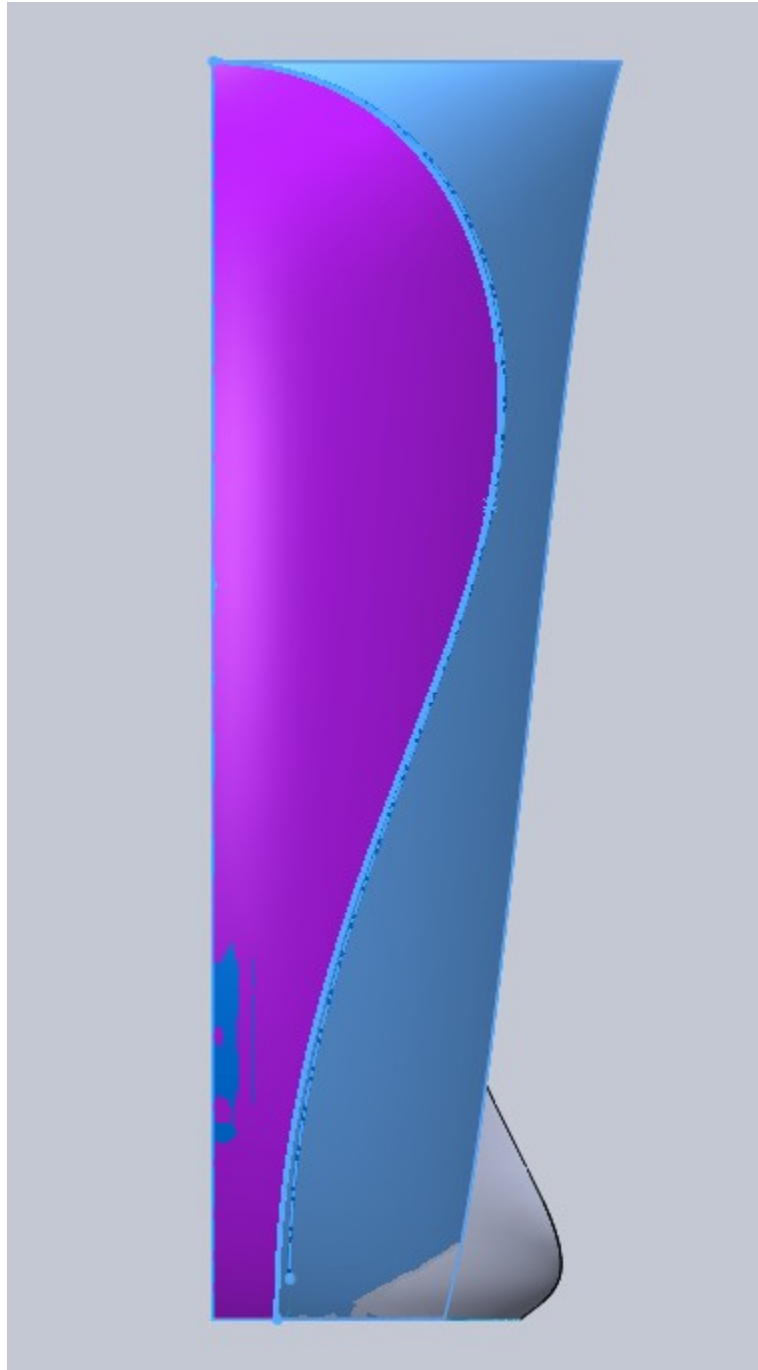
We need to draw a spline similar to the one used to trim our other speaker face, but this one will need to be inset at the bottom.

The reason for this is based on the image of the speaker. The area around the button and led is inset a bit so we need to be aware of this when creating the geometry. This spline has a control point as well as end points. Note the vertical and horizontal relations for the end handles and that we have not touched the mid-point handle.



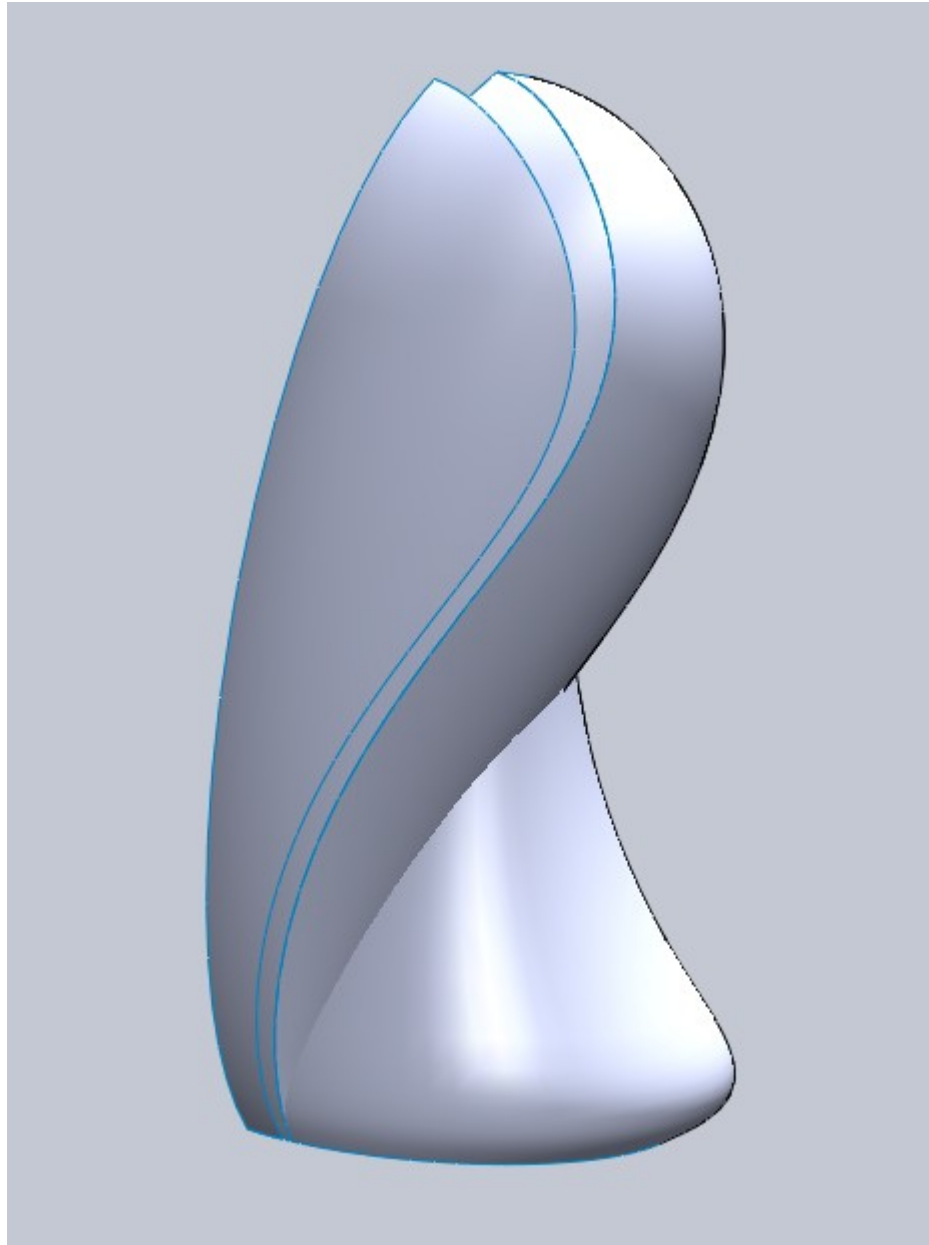
Step6. Creating a second speaker face.

Use this sketch to trim our boundary surface by removing the blue area as shown below.



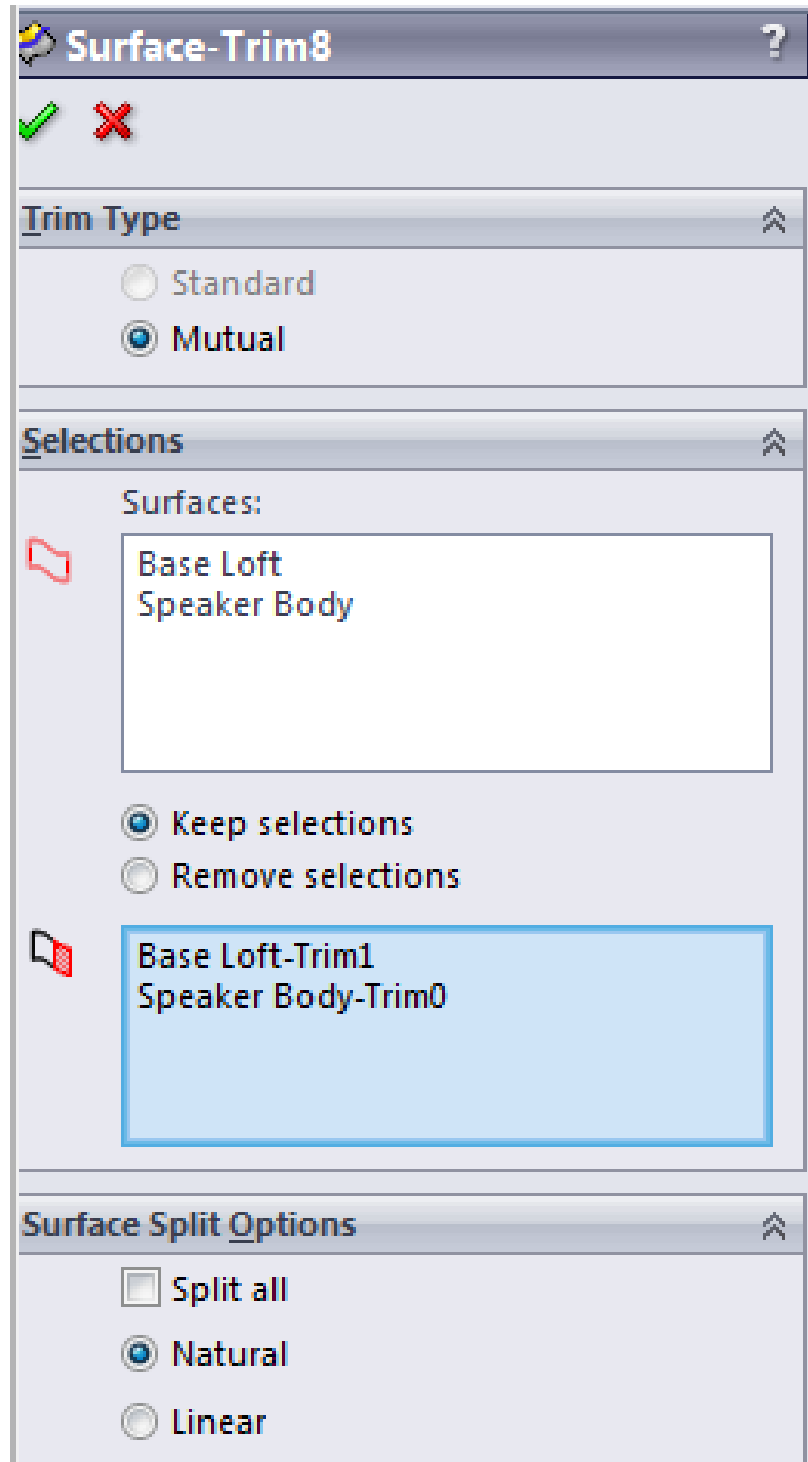
Step6. Creating a second speaker face.

We should now have 4 different surfaces that are starting to resemble the speaker.



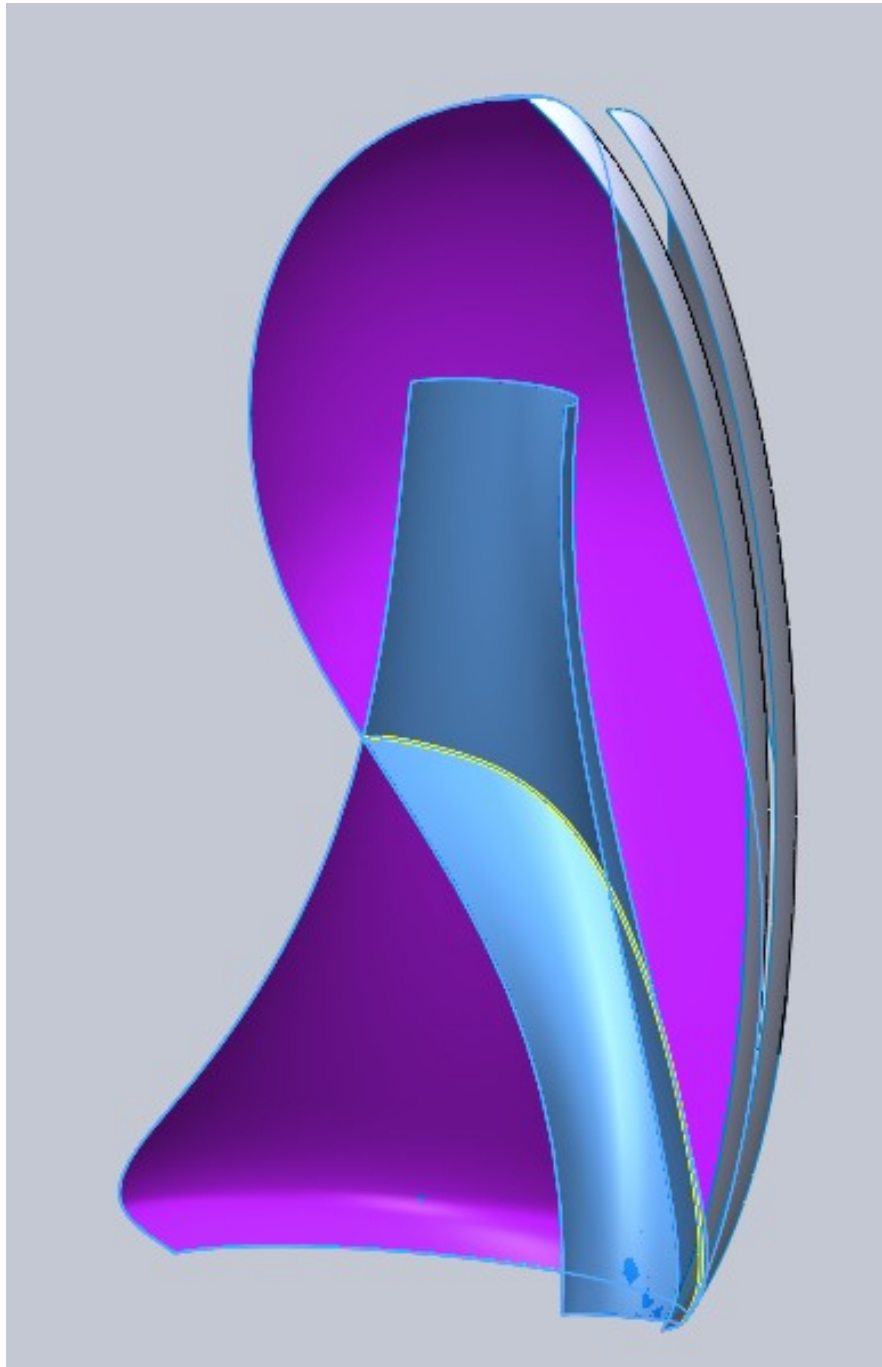
Step7. Trim Main Bodies

Using Trim, we are going to make a Mutual Trim of the main loft and boundary surfaces.



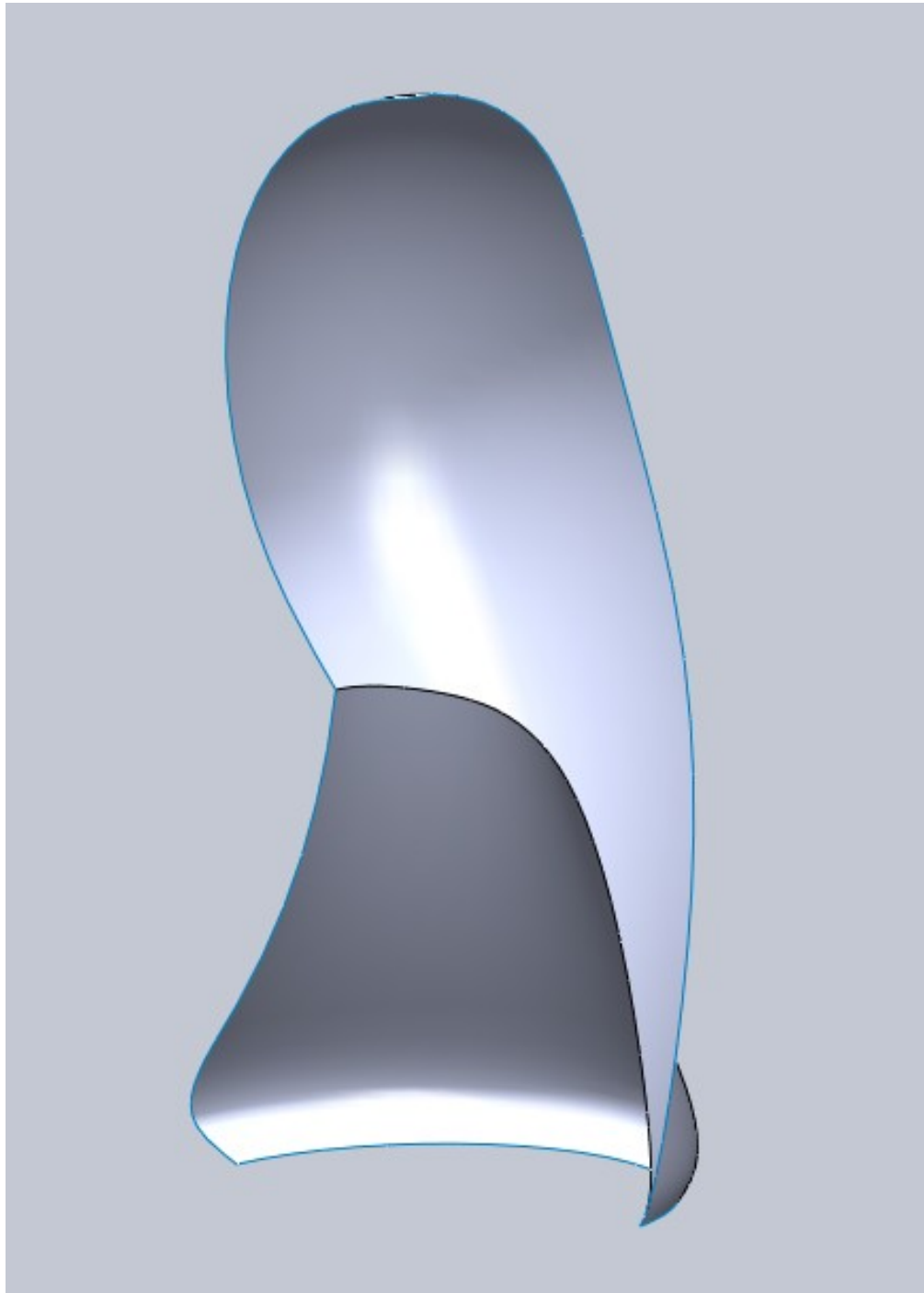
Step7. Trim Main Bodies

We want to keep the purple section below and remove the blue section so make your choices appropriate based on your settings. Also note that using mutual trim with knit these two surfaces together.



Step7. Trim Main Bodies

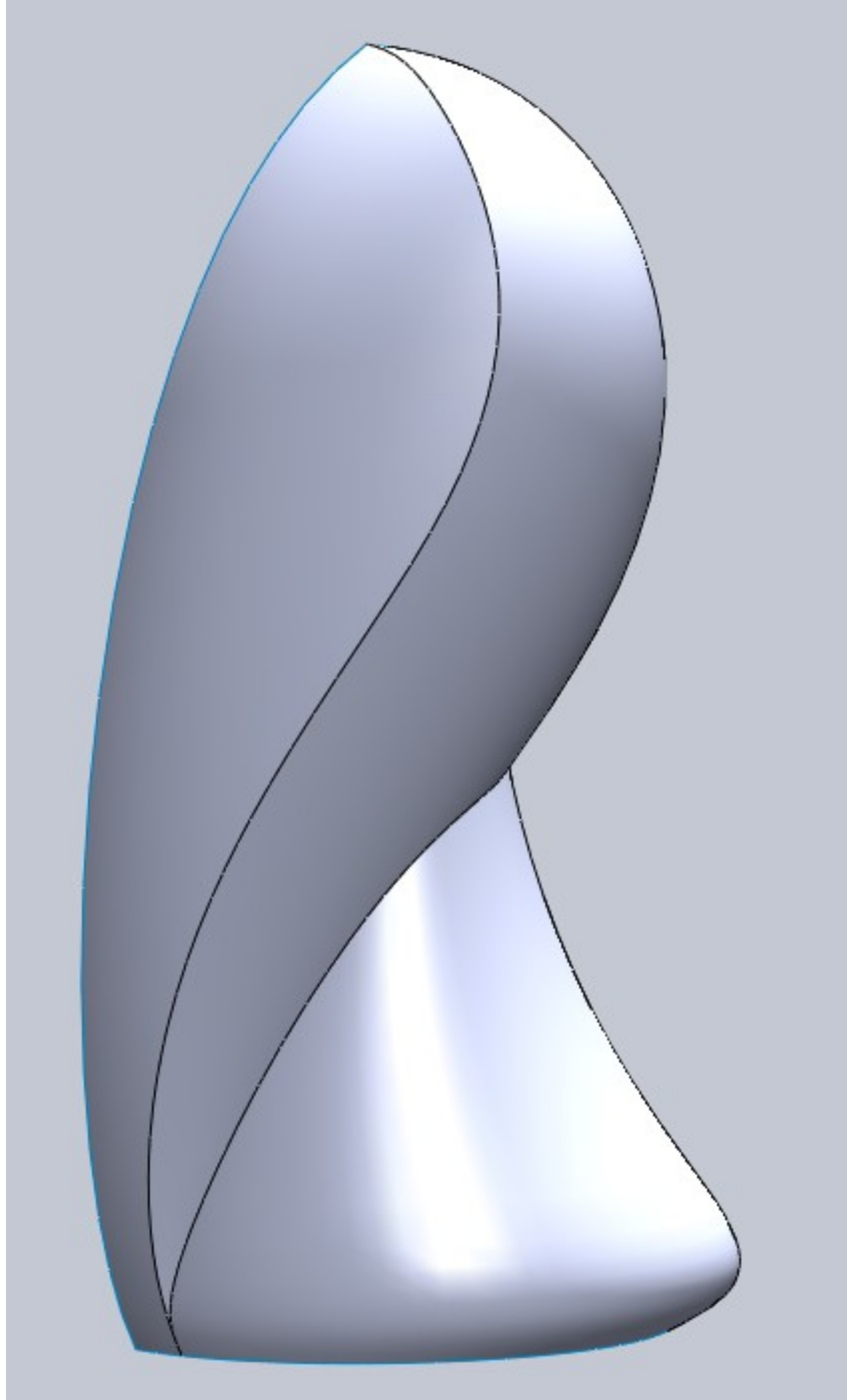
The result will be a nice trimmed surface. Note that I have hidden the two speaker face surfaces, they were not part of the trim operation.



Step8. Knit Bodies

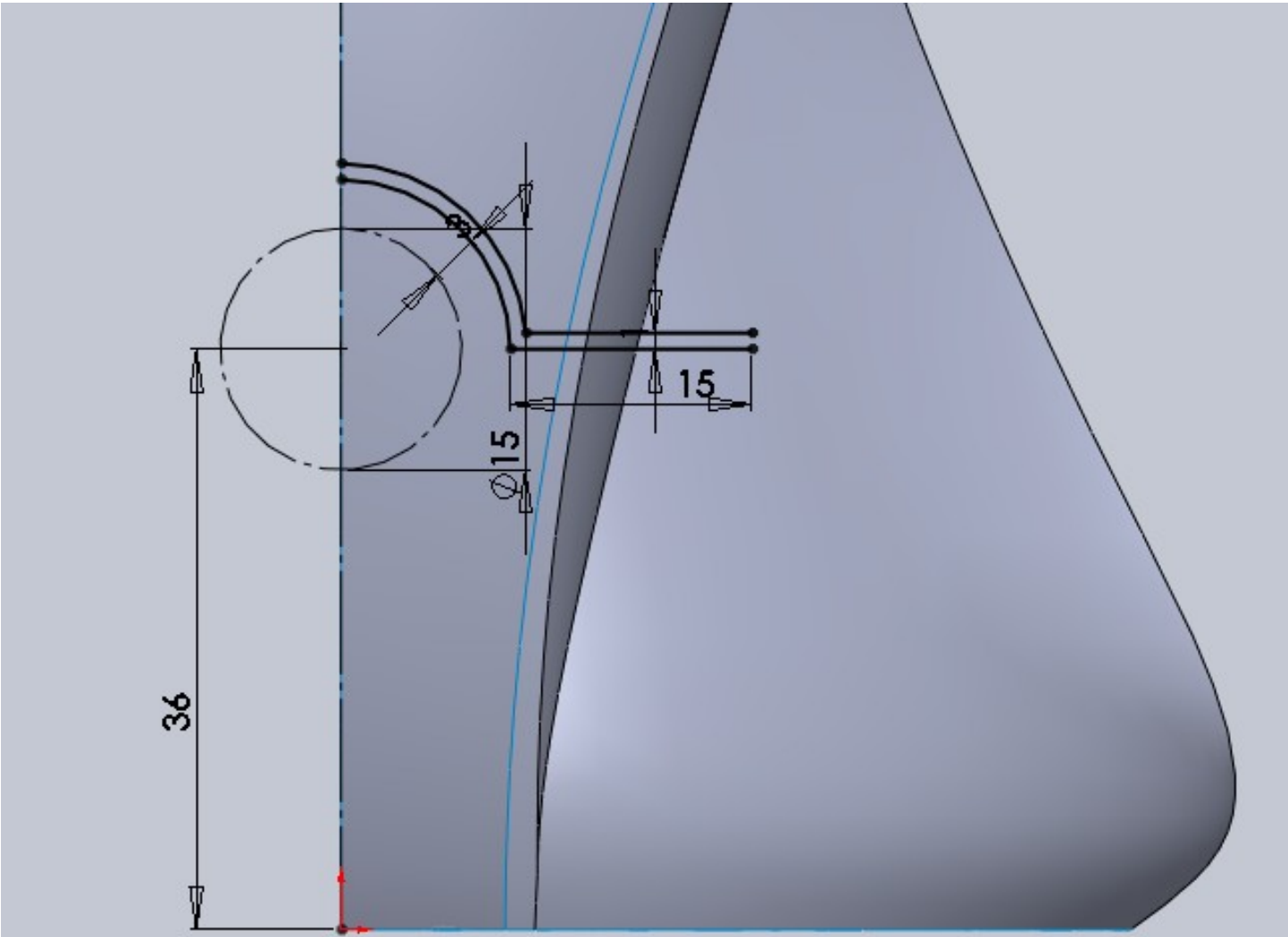
Knit the trimmed surface with the first Speaker Face boundary surface. Knit surface (unless you specify) will make appropriate changes to surfaces in order to knit them together.

This change will help us and is why we knitted the face to the trimmed body.



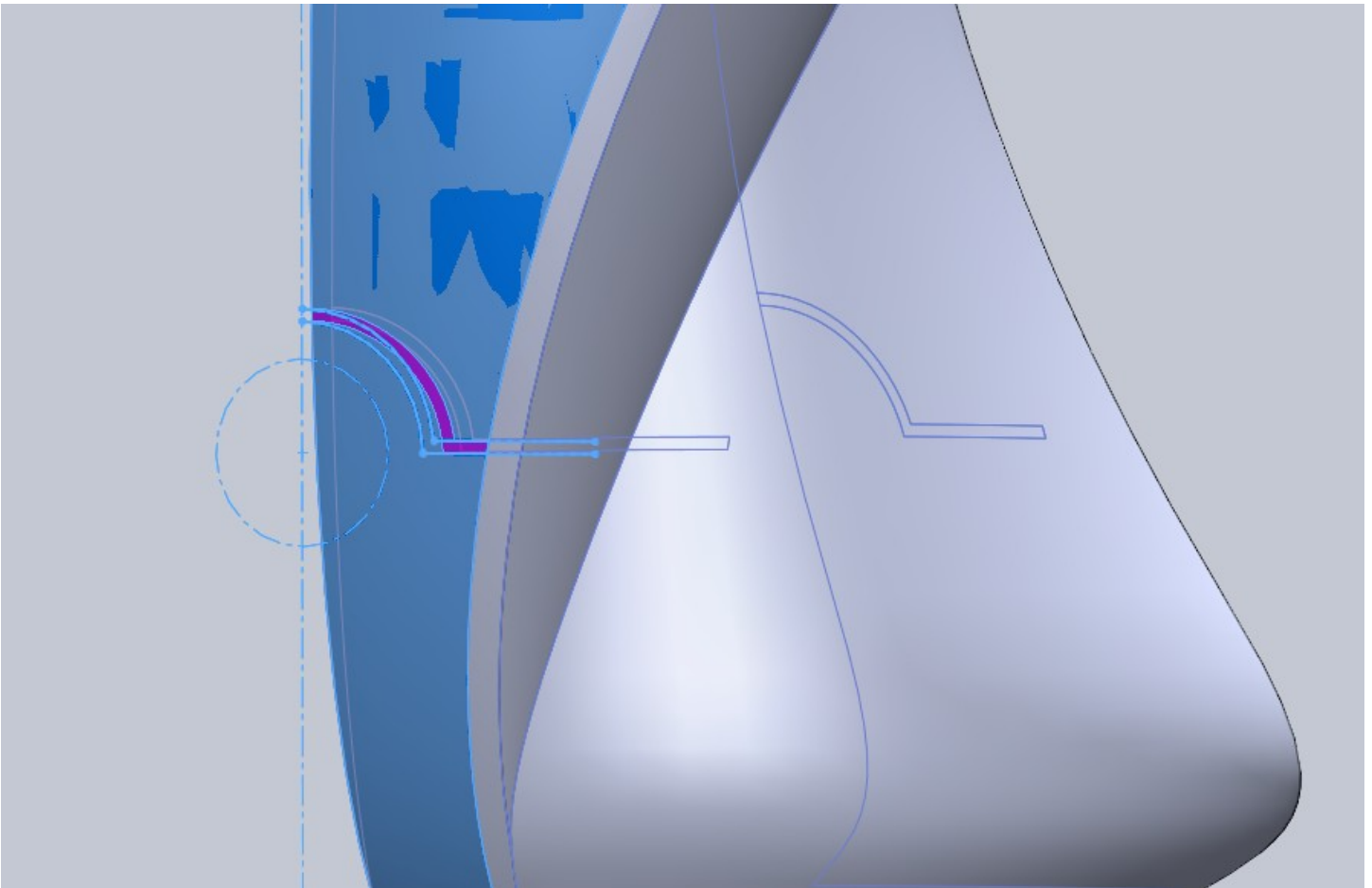
Step9. Trim Surface for Speaker Face Split

Start a sketch on the Front Plane.



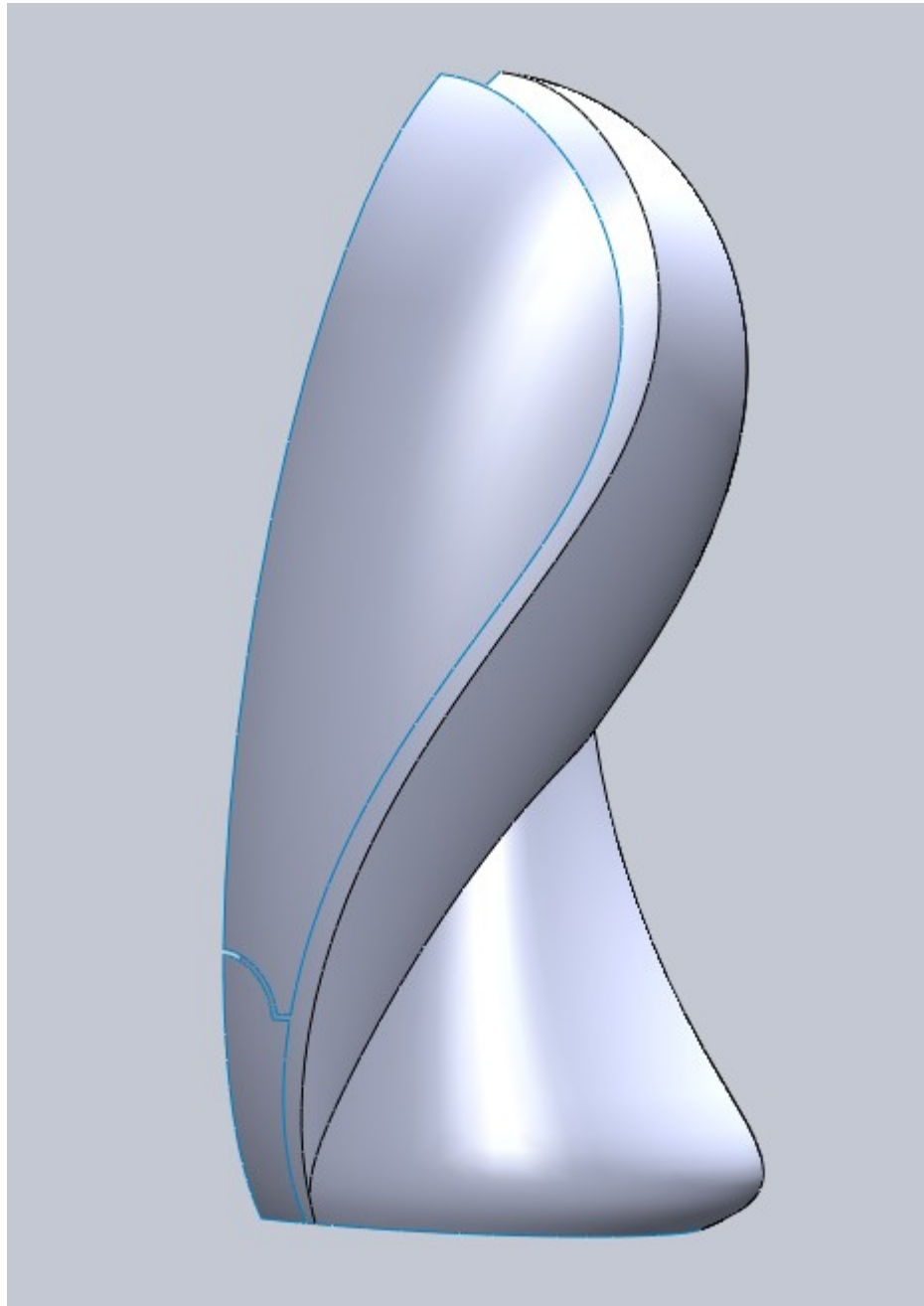
Step9. Trim Surface for Speaker Face Split

Trim the 3mm section away from the second speaker face boundary surface. Remove the purple section shown.



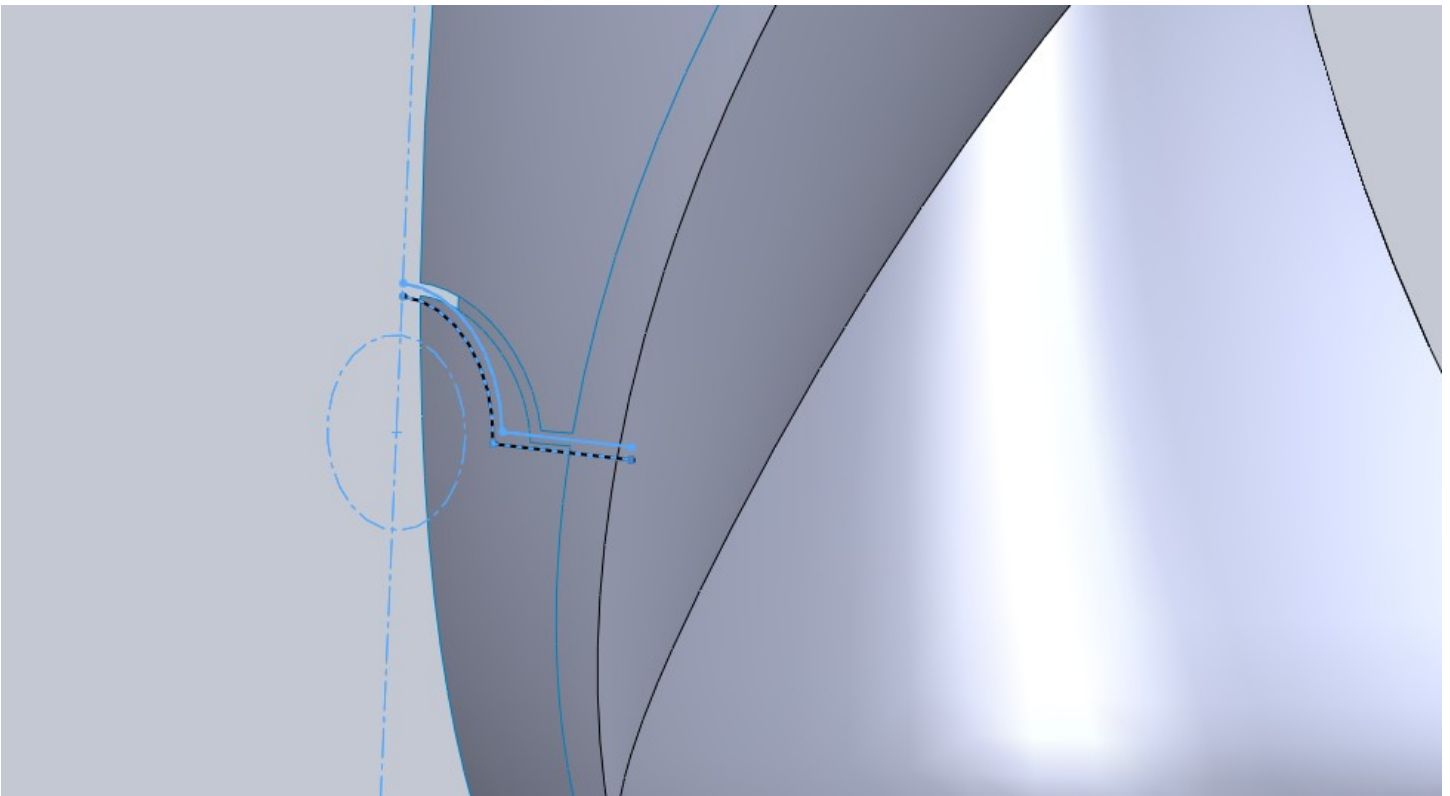
Step9. Trim Surface for Speaker Face Split

You should now have 3 surfaces. The main body and two surfaces that make up the speaker face.



Step10. Split Faces and Create Connecting Surfaces

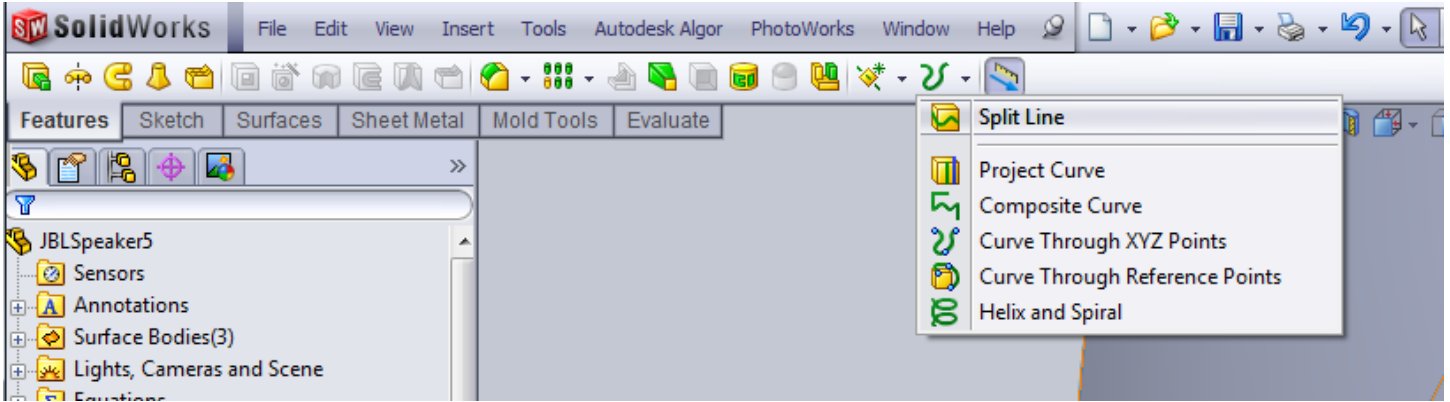
When using Split you have a limitation. Split will not allow you to use a sketch with more than one open or closed profile. This means our last sketch used in the Trim operation could not be used to split a face. Because of this we will need to start a new sketch on the Front Plane and use Convert Entities on just the bottom set of lines.



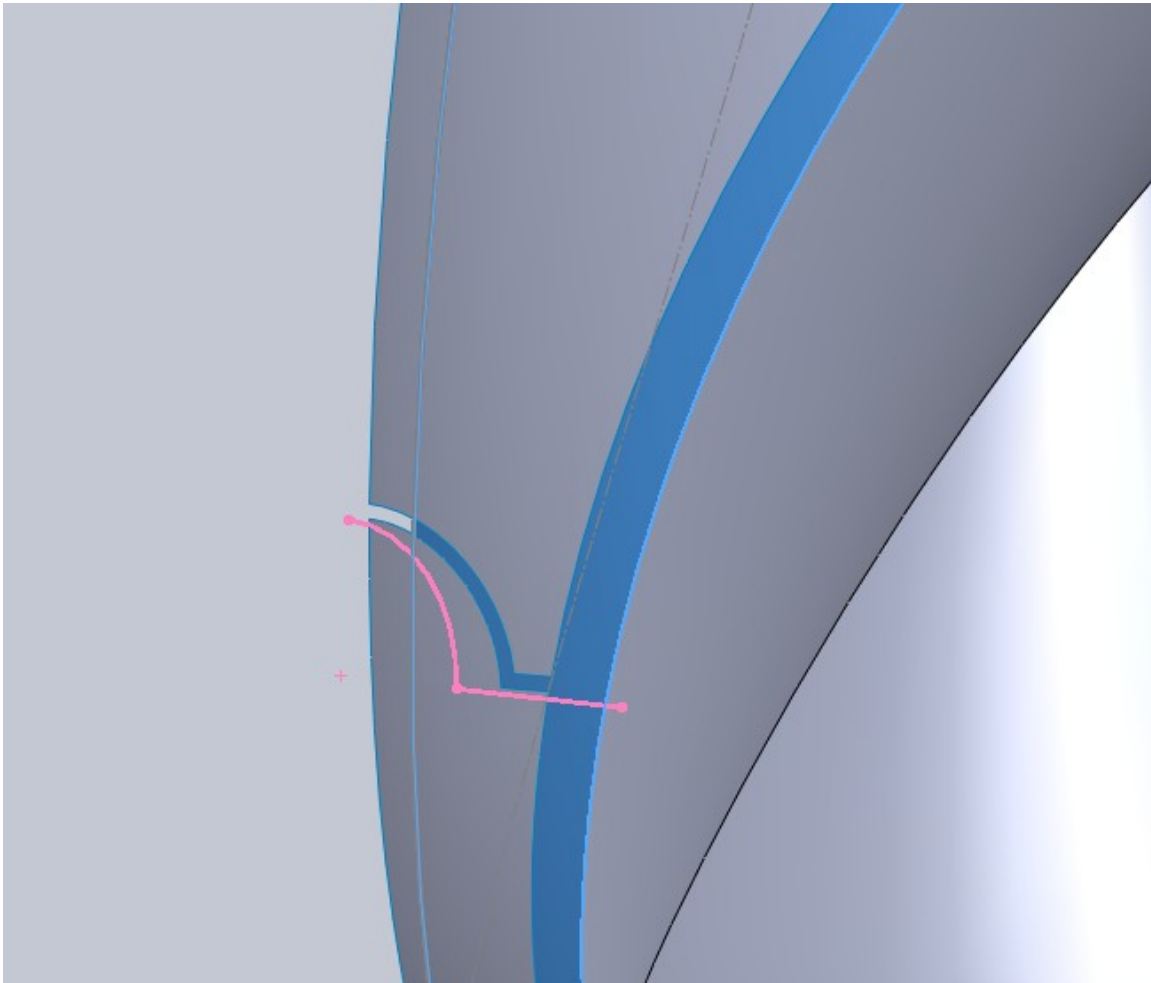
Splitting a face allows us to use these new edges for things like lofts or boundary surfaces.

Step10. Split Faces and Create Connecting Surfaces

Using the new sketch and Split Line split the faces as shown.

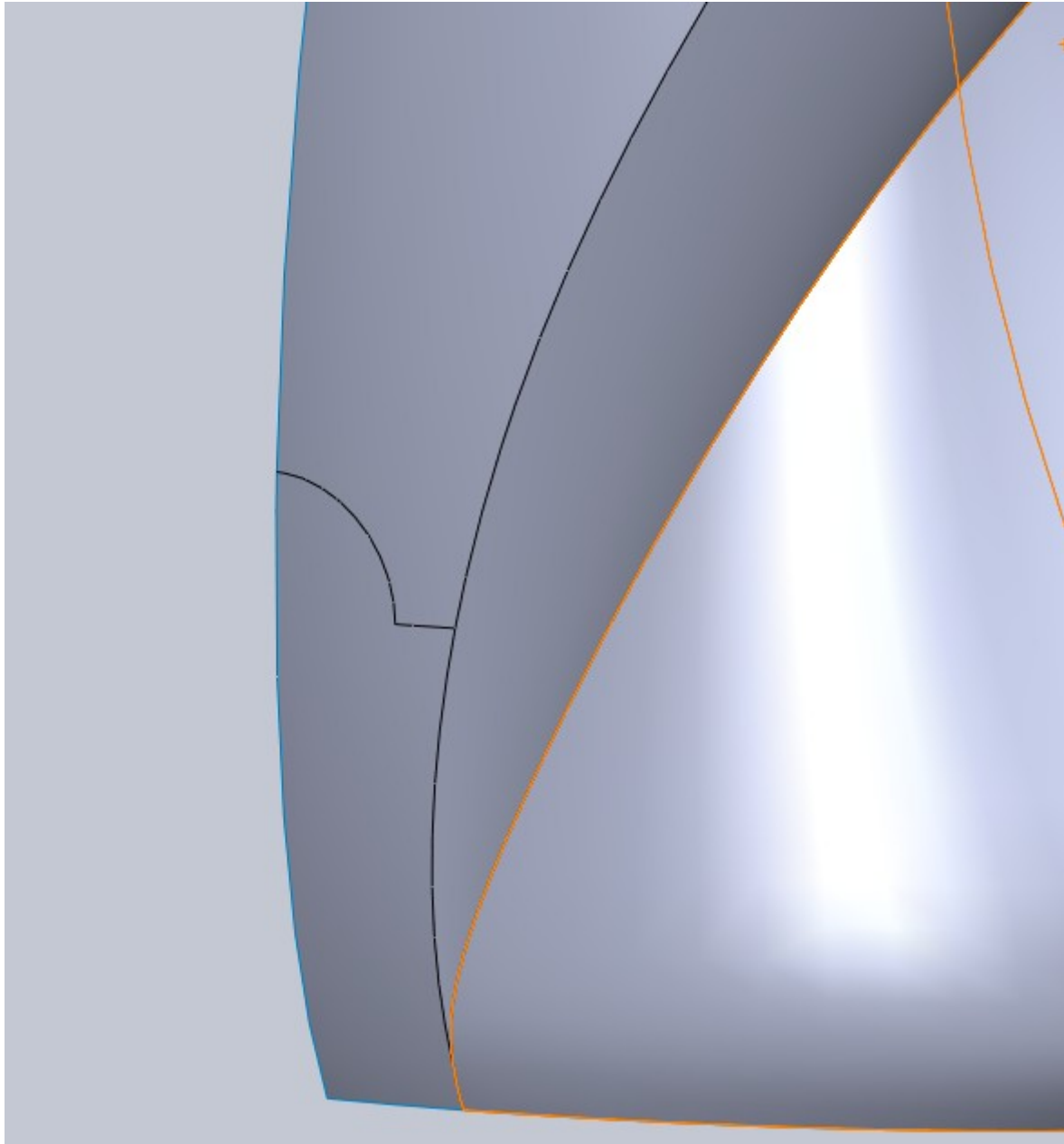


The pink line is our split tool and the blue face is our selected face to split.



Step10. Split Faces and Create Connecting Surfaces

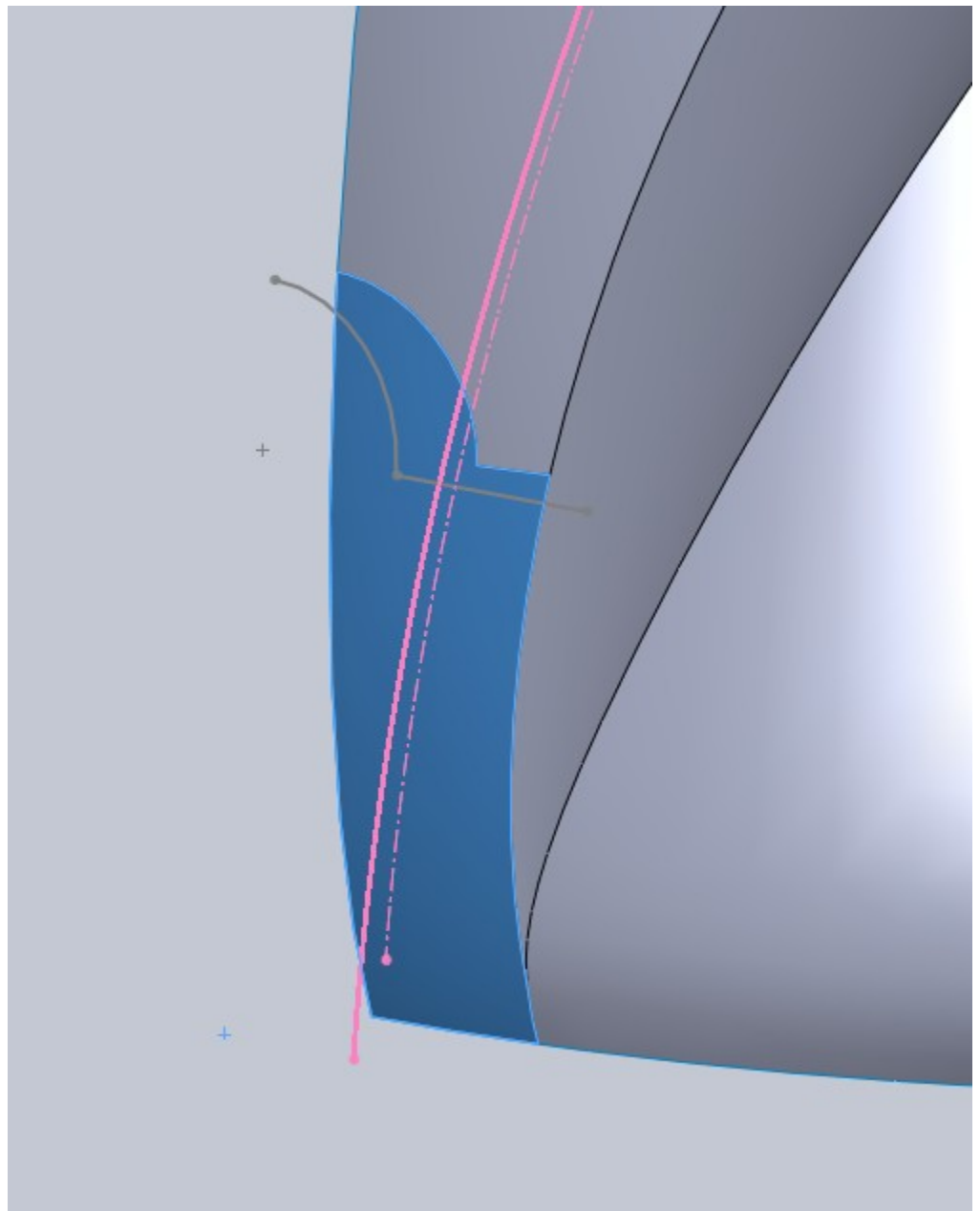
You can now see our surface has a split on its face but is still one knitted surface. I have hidden the other two surfaces from view.



Step10. Split Faces and Create Connecting Surfaces

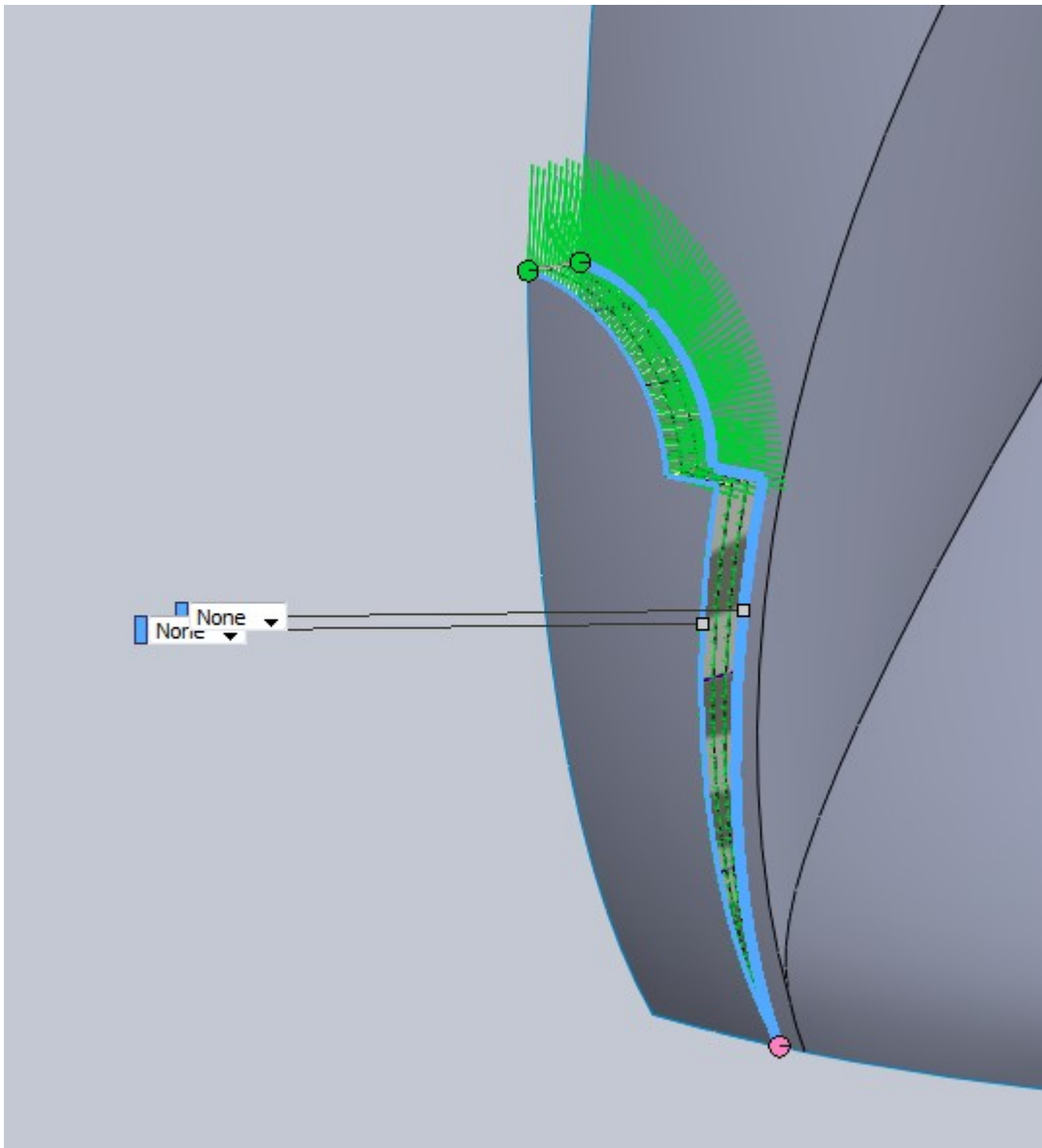
We are going to make another split but this time we are going to re-use a sketch we have already created. Make the sketch we used to trim our secondary speaker face visible. This sketch was slightly inset from our first speaker face sketch at the bottom.

Split only the blue face shown here.



Step10. Split Faces and Create Connecting Surfaces

Now that our face is split go ahead and create a boundary surface between this split edge and the matching surface in front of it. You will need to use the selection manager and select the 3 edges needed for each.



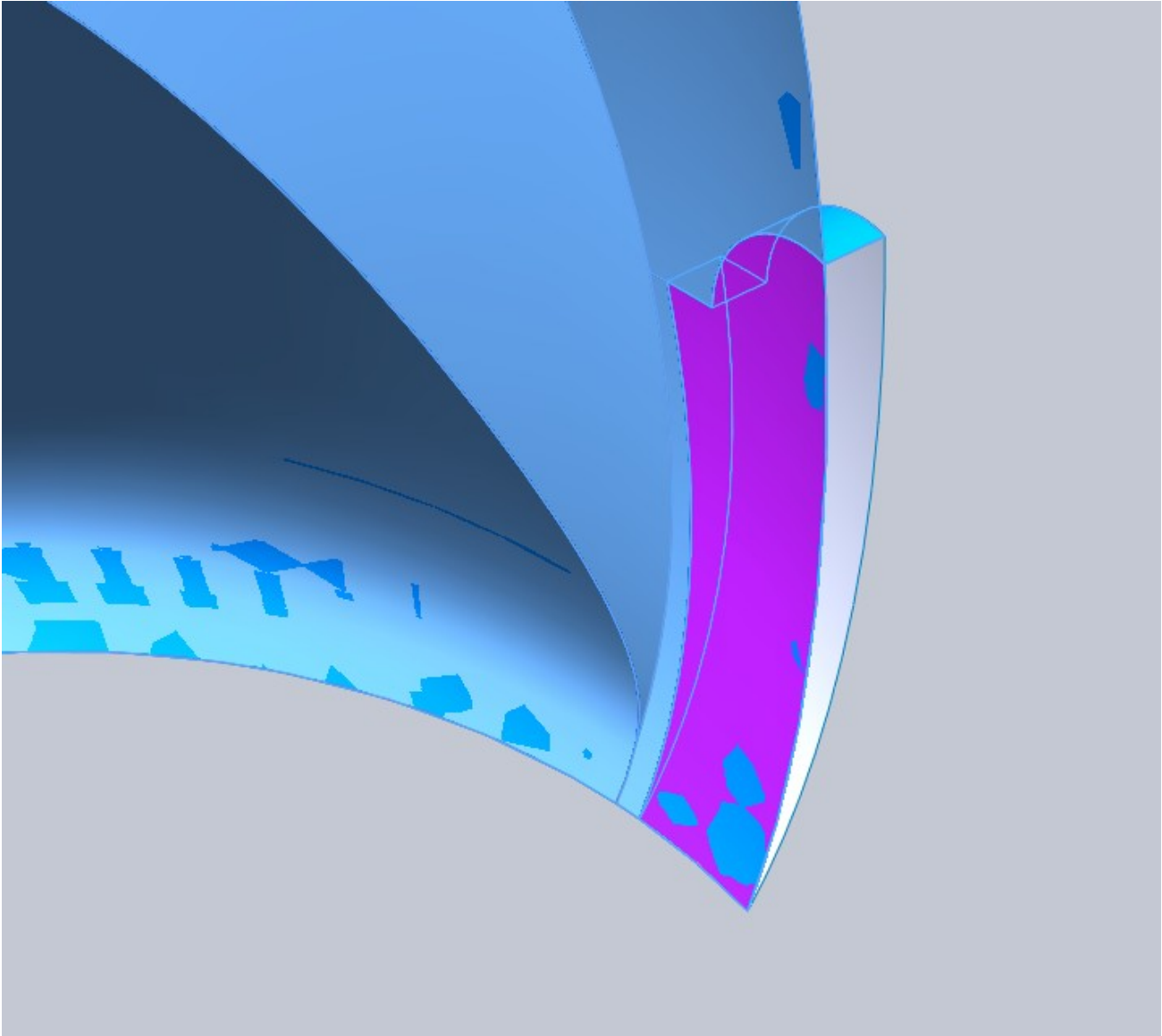
Step10. Split Faces and Create Connecting Surfaces

You might be wondering why we needed to make the second split for this surface. Looking at the picture it appears as if there is a small “step” in this area. It really comes down to a judgment call on the person making the drawing. I drew it the other way first and it didn't appear right to me so I made the call to do it this way.

One note that I feel is important. When dealing with complex surfaces its often a good idea to have a quick try at the general shape you are dealing with. This will let you know if a certain approach will or will not work for you. In the speakers case my first approach was to use a boundary surface for the back first, then model the base using a boundary and split lines on the first face. This worked well enough but produced problems down the road with fillets and surface intersections. So I learned by taking a shot at it first that my original method was flawed. Producing the base first with a loft then using a boundary and trimming both together gave me a much more stable set of surfaces to work with.

Step11. More Trimming

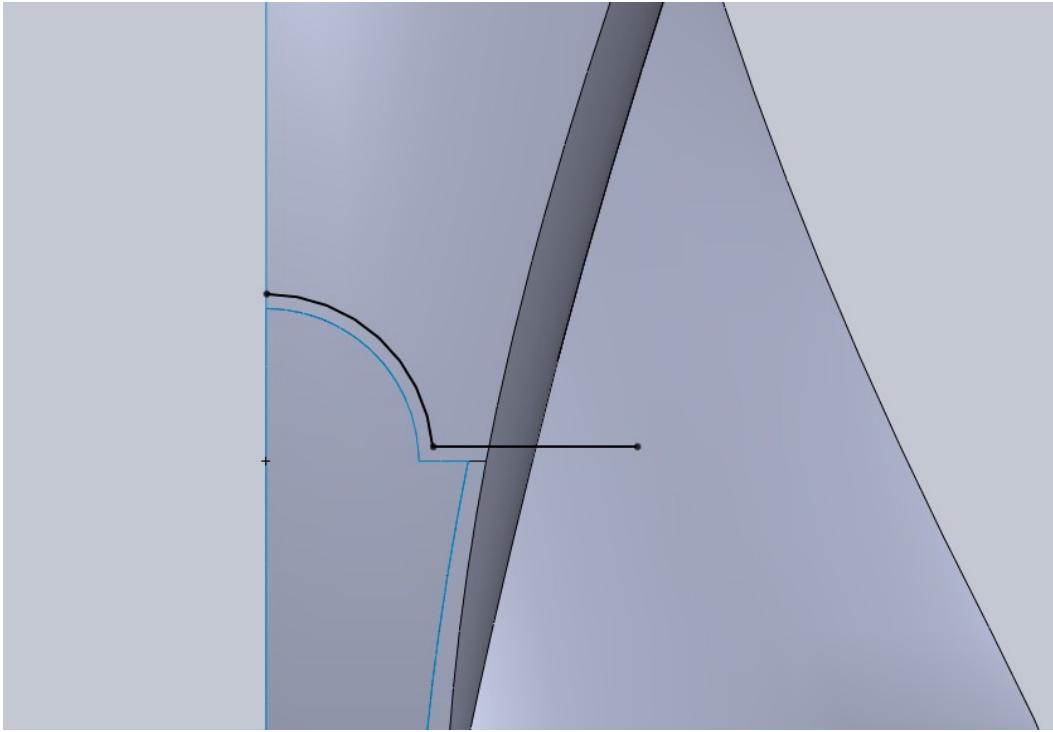
Using the new boundary surface as a trim tool, trim away part of the face behind it shown in purple.



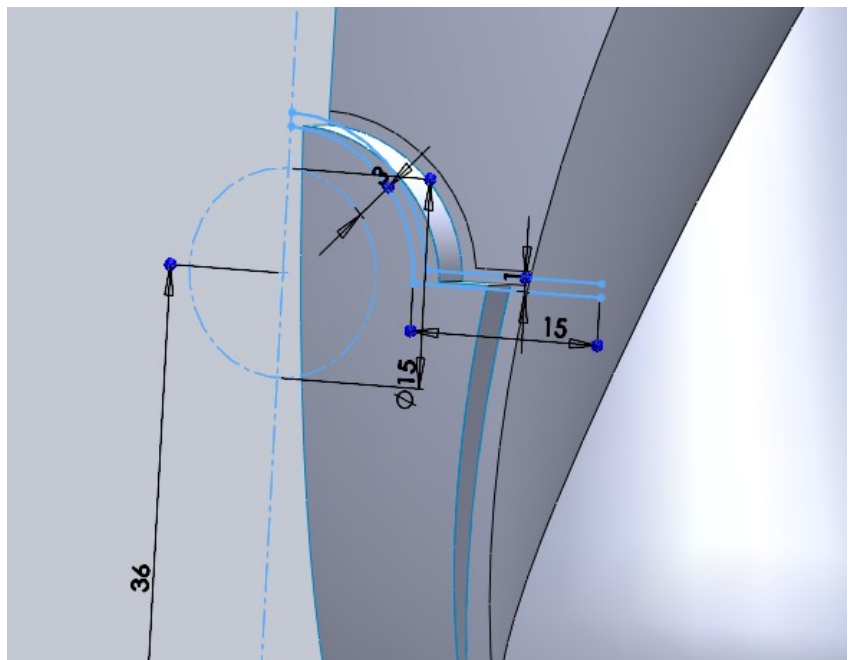
Just as a note, since we already split this face to use for our surface, we could have also used the Delete Face command and arrived at the same result.

Step12. Another Split Face

Start a new sketch on the Front Plane and Convert Entities on the edge shown below.

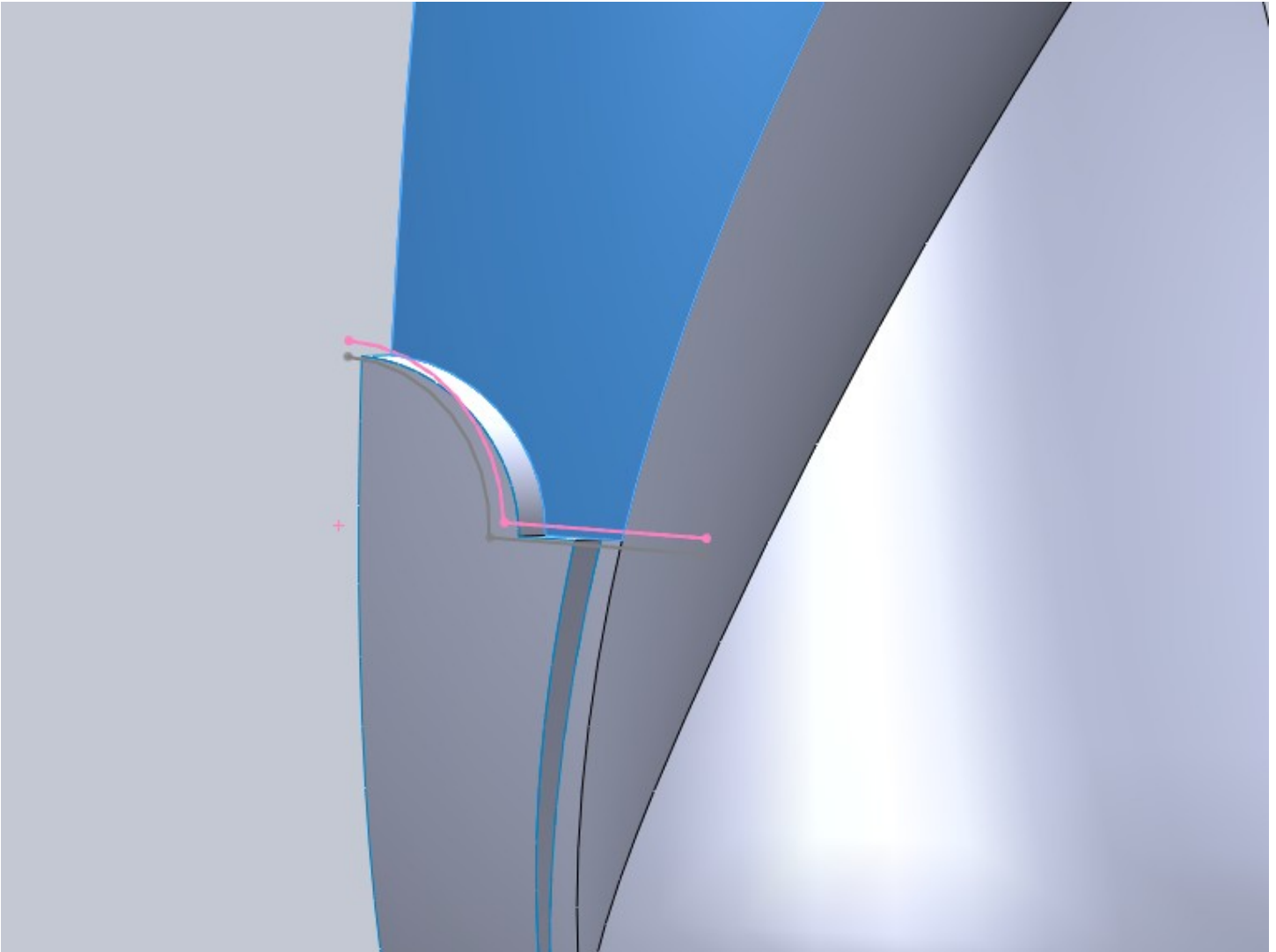


This edge comes from our sketch used to trim the second speaker face.



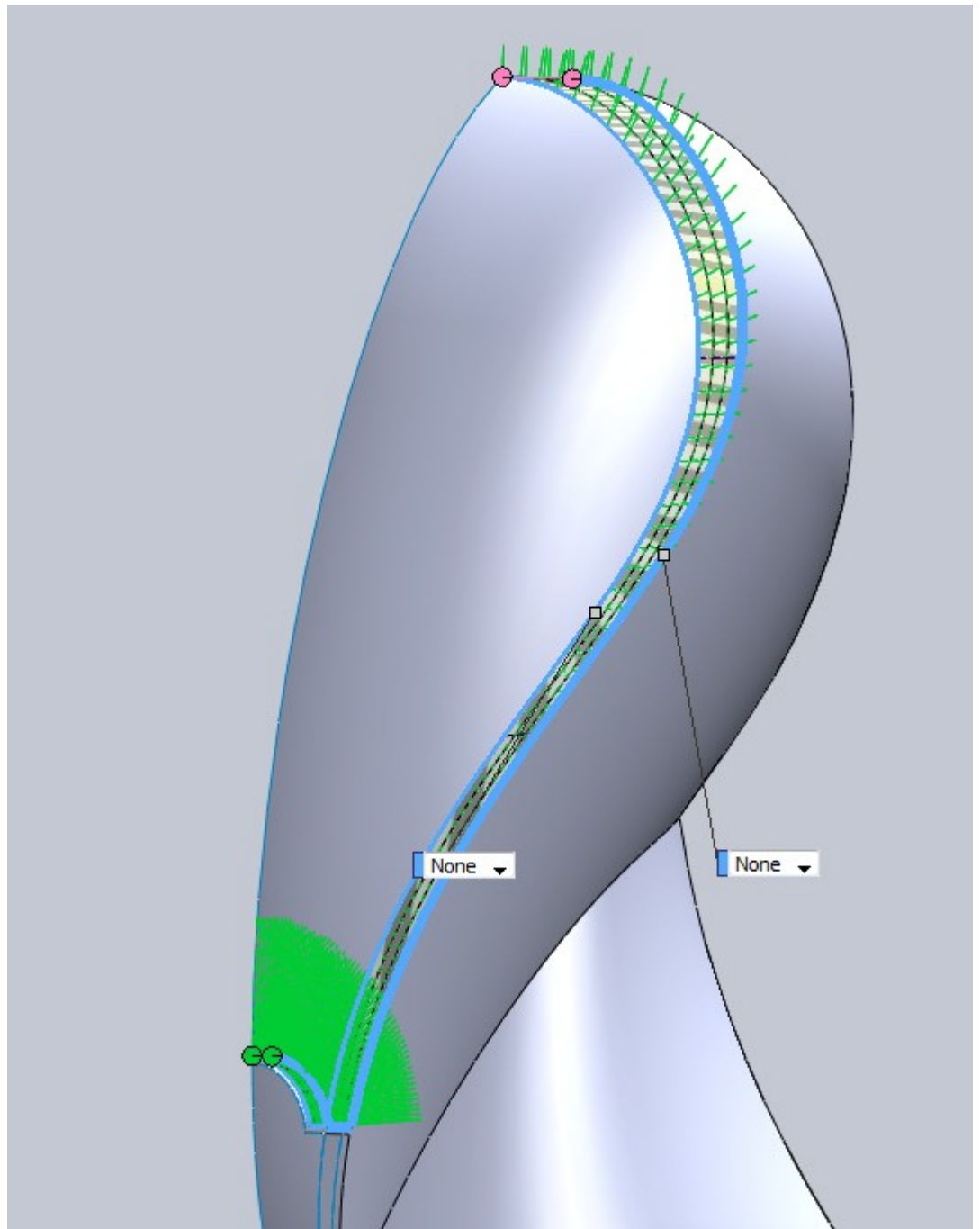
Step12. Another Split Face

Use this new sketch as the split tool and split the blue face shown below.



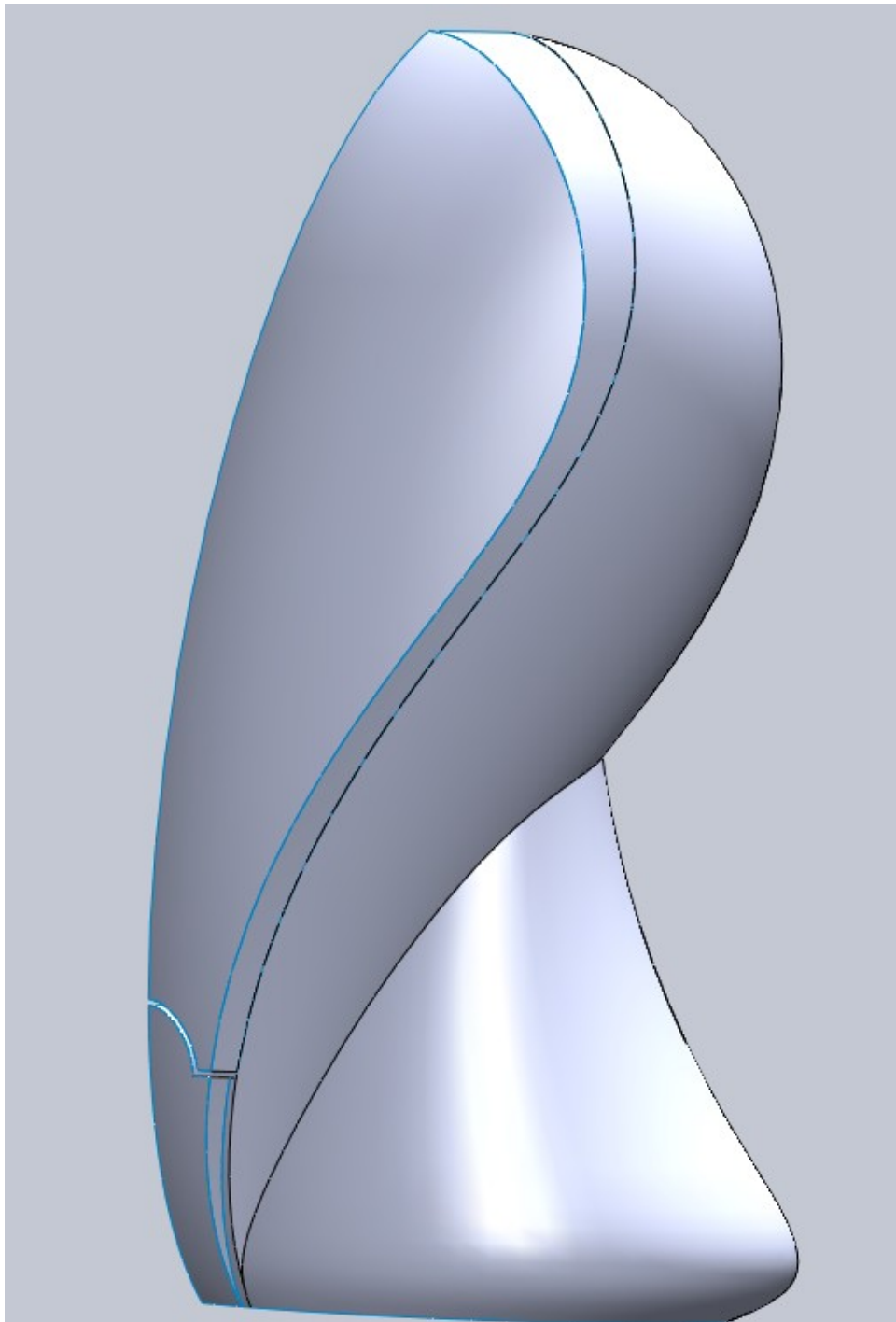
Step13. Speaker Face Boundary

Using the new bottom split line we can now make the boundary surface for the speaker face. In the boundary surface selection box you will need to use the selection manager and select each edge needed. This will be 3 edges for each selection.



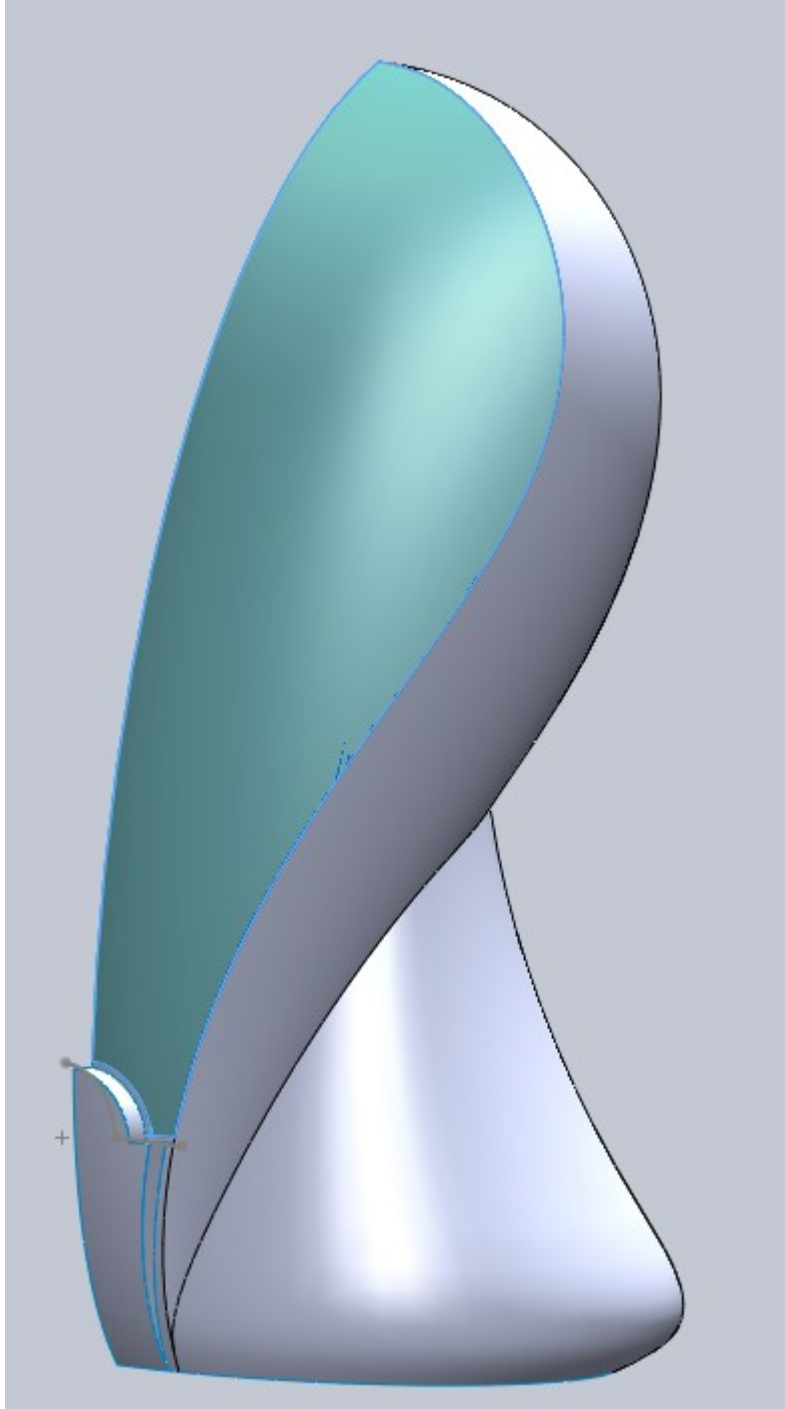
Step13. Speaker Face Boundary

Now we have our speaker face edge filled in we have complete almost all of the major surfaces. Now the devil is in the detail work!



Step14. Offset Surface

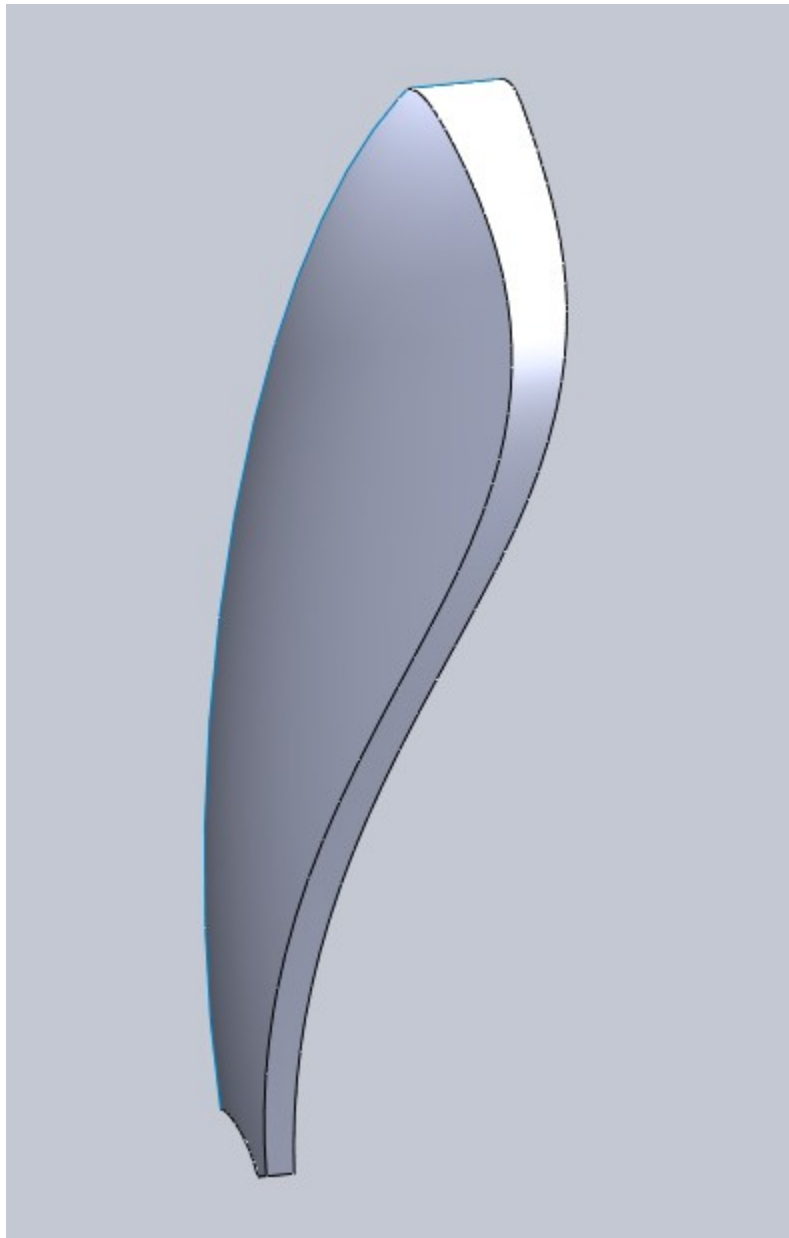
Now we want to take our inside speaker face surface (which is part of the main body) and make a 0mm offset of that. So make a 0mm offset of the surface shown below.



Step14. Offset Surface

Now hide the main body and the two lower surfaces that go with it but leave the new offset surface visible and make our speaker face and its edge visible.

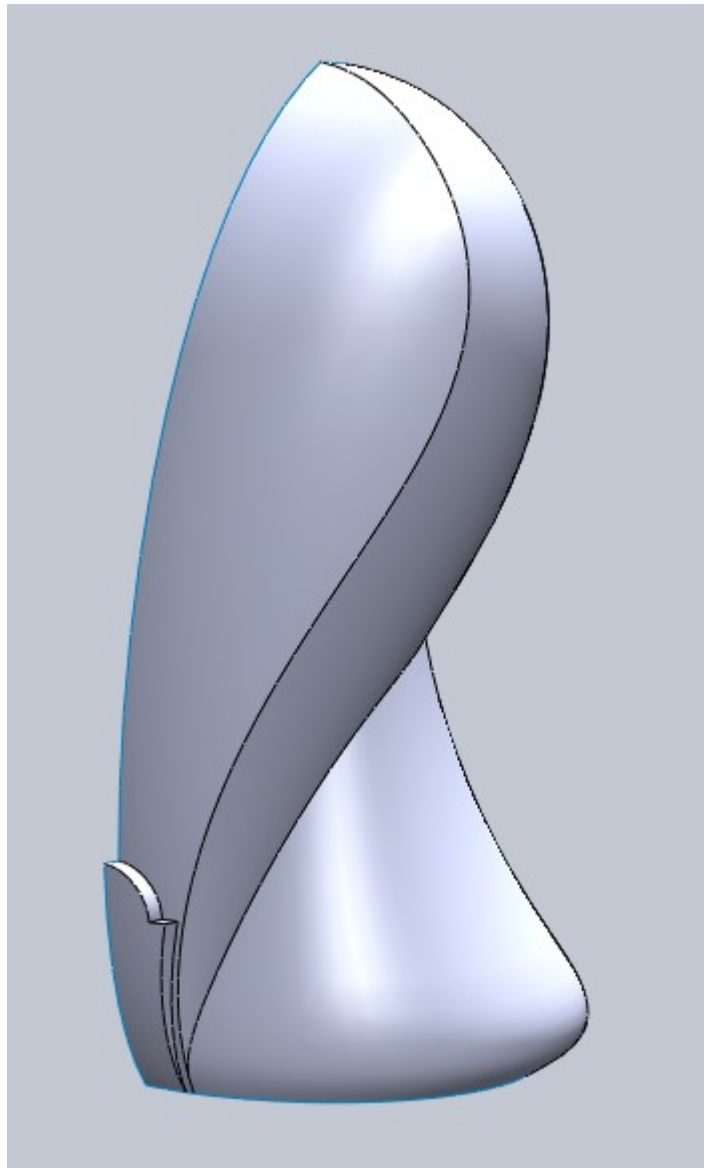
Knit the offset surface and the front and edge surfaces of the speaker face together.



Step15. Knitting Surfaces

Hide the newly knitted surface and then make the other 3 surfaces visible. Now knit these surfaces together. Take note of what happens to the face with our split line on it. Once you have knitted that face with another, the split line is no longer there.

This can be both valuable knowledge and annoying. Since you know this will happen make sure you use your split line edges on surfaces before knit operations.

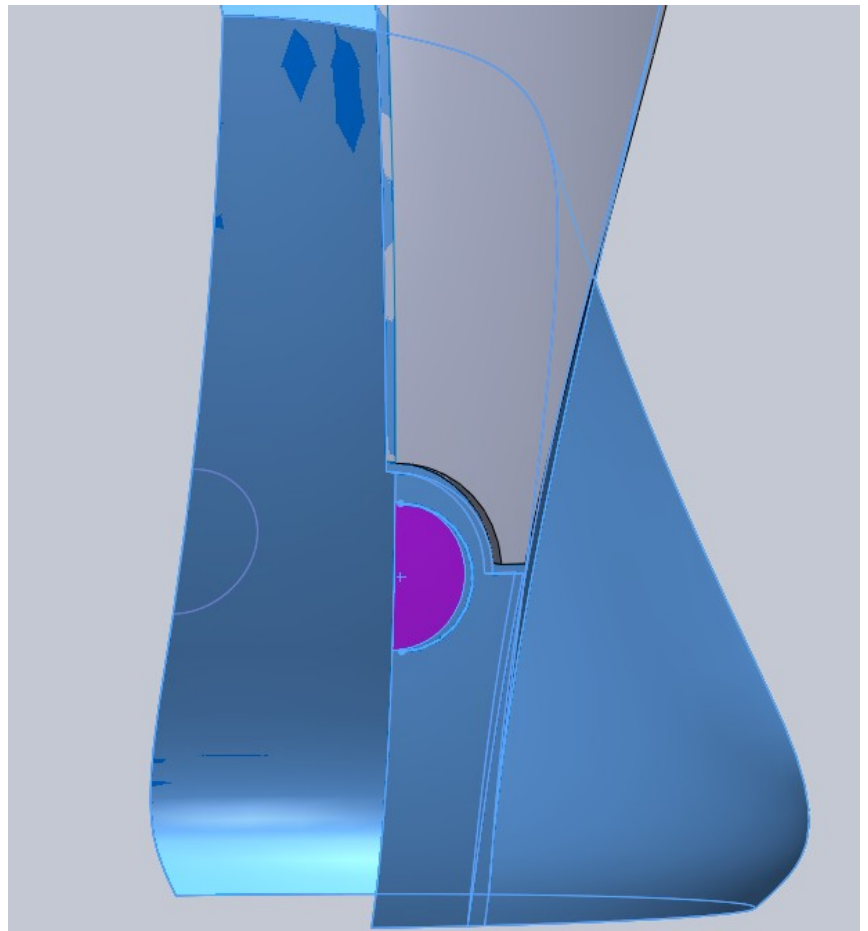


Step16. Trimming Surfaces

Now we have two main bodies to work with. The speaker face and the main body. Now we are going to make some holes for the button and led on the bottom portion of the speaker body.

Start a new sketch on the Front Plane. If you remember when we made our sketch to trim the speaker face away from this portion we started with a construction circle. Use Convert Entities to bring that circle into our new sketch. If you like you can create a vertical line to trim the circle or you can leave it complete.

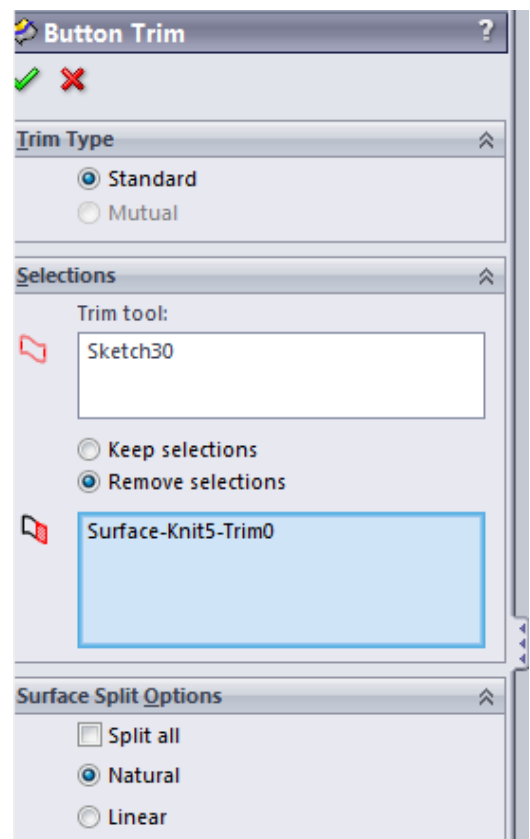
Use this sketch to trim away the area for the power button.



Step16. Trimming Surfaces

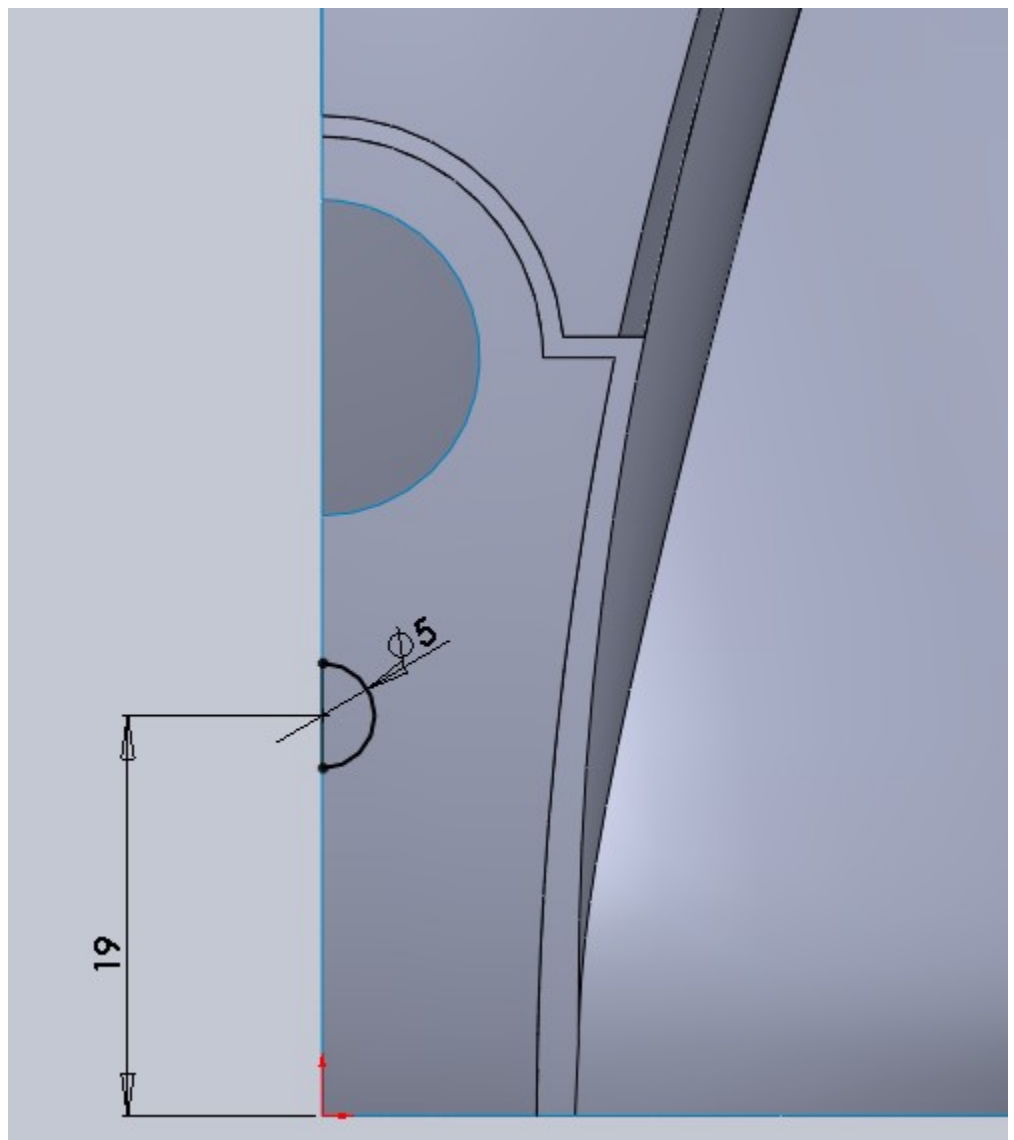
Note on trimming. In this case we are dealing with a sketch that would intersect our surface body in two places. With cases like this you have to be very careful of your selection. I like to “Remove Selections” as opposed to “Keep Selections” because of this. With Remove Selections only the surfaces you select will be removed. With Keep selections, depending on your view, you might not notice that this sketch would trim the surface in two places. This would leave a hole all the way through the surface. Of course this is an easy fix, just edit the trim operation, but I just want you to keep this in mind when you make your selections.

Notice I have Remove Selections option toggled on.



Step16. Trimming Surfaces

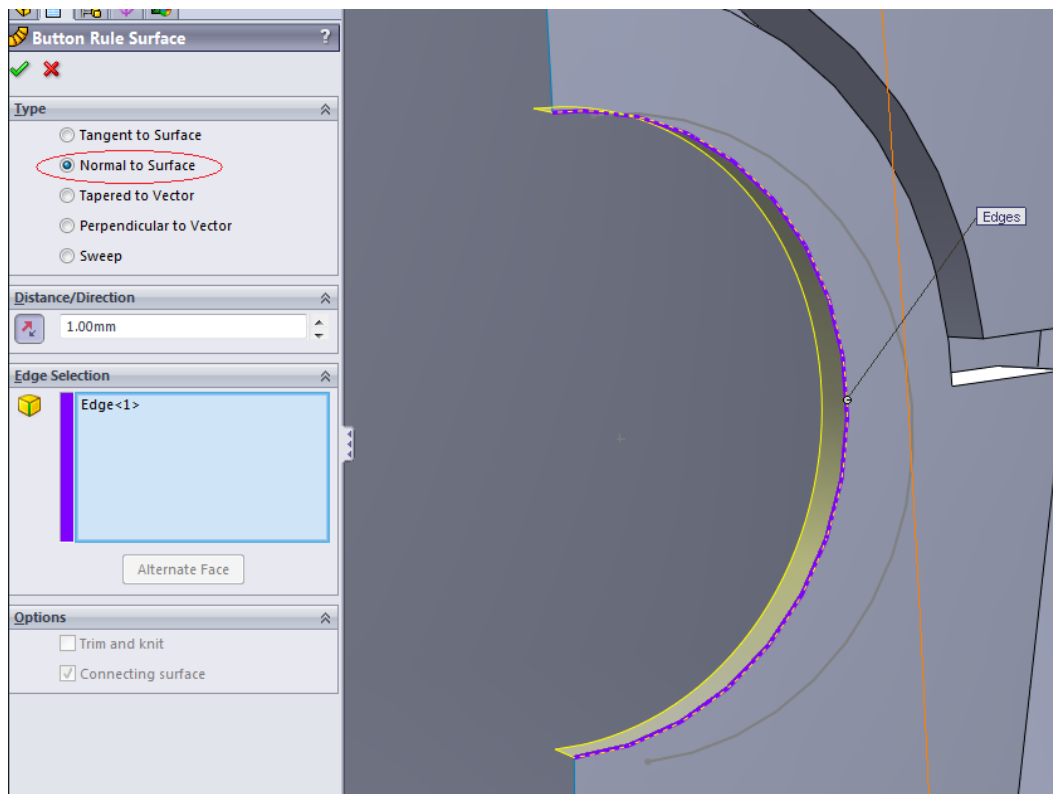
Start a new sketch on the Front Plane for the led hole trim. For this position you should make your front speaker image visible and choose its location. I chose 19mm up from the origin and drew a vertical reference line that was above the origin. I chose 5mm as my LED hole size. Again you can draw a full circle or a half circle like I did. Use this sketch to trim away for the led hole.



Step17. Rule Surfaces

Rule Surfaces are very versatile. Whenever you are dealing with surfaces that are complex and you need to add edges or lips to them, rule surfaces are the way to go. The context in which they are used here is only a small fraction of what can be done so be sure to play around with them.

Choose Rule Surface from your surfaces tab on the ribbon. You will notice 5 different options for “type”. We are going to use “Normal to Surface”. Select the edge of our power button trim and enter a distance of 1mm. Ensure the direction is going into the body of the speaker.



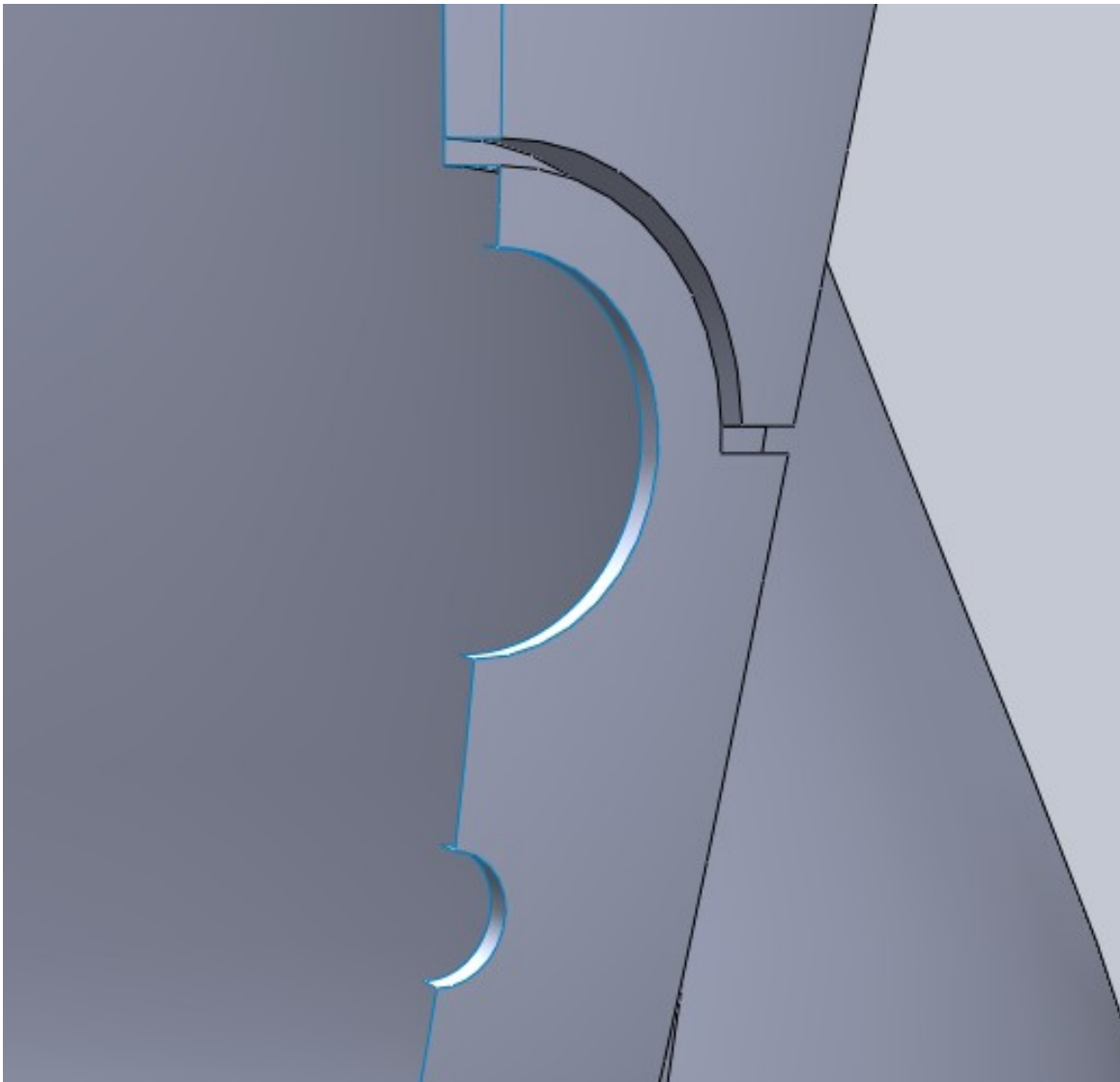
Step17. Rule Surfaces

Follow the same procedure for the LED hole with the same 1mm Normal Rule Surface.

Looking at a side view you will notice a little bit of the power of Rule Surfaces. These 1mm surfaces stayed normal to the face where they intersect. Now I will make one note here about “tapered to vector”. In reality this speaker body would be injection molded, meaning everything would need an appropriate draft angle. While the rule surface normal to our face works great for this visual representation, the real thing would probably use a tapered to vector surface where you would pick a reference plane that suits your direction of pull. This is a bit much to digest at the moment but just understand difference here between a visual model and production part.

Step18. Knit

You should now have a 1mm surface that is normal to the Power Button hole and LED hole. Go ahead and Knit both of those to the main body.

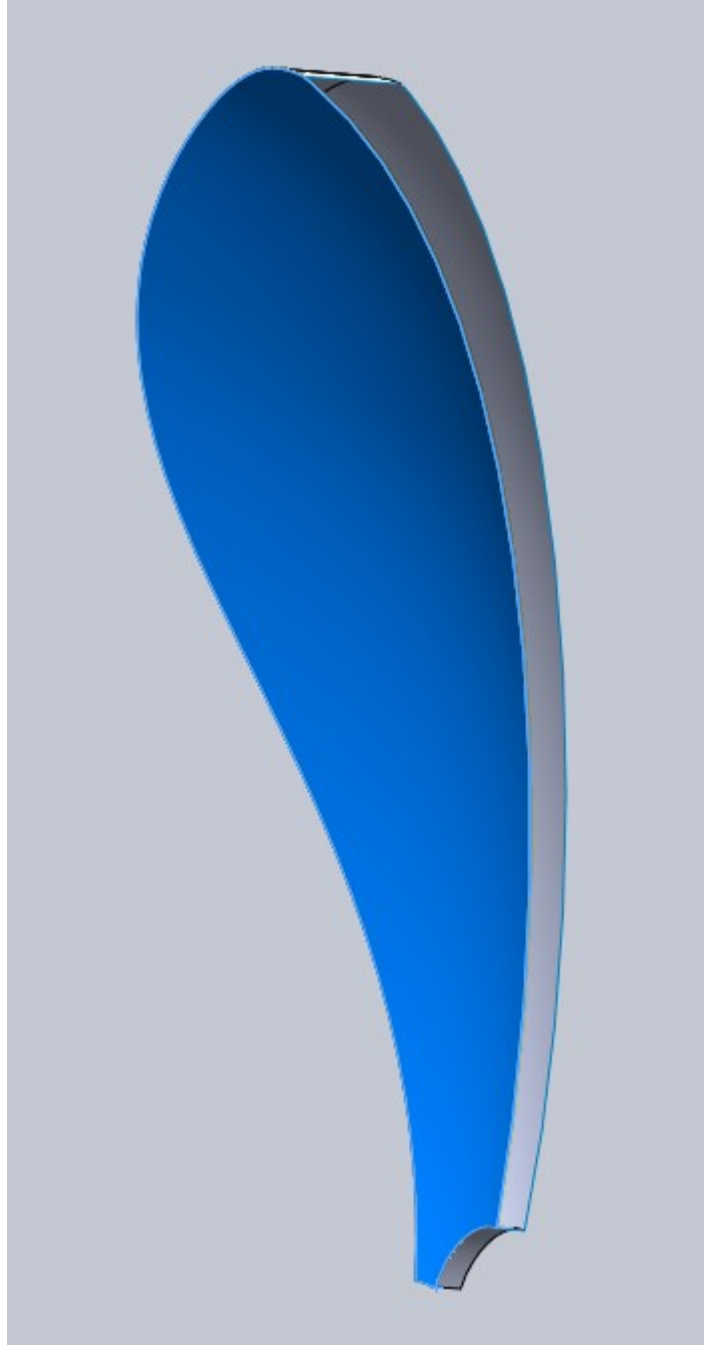


Step19. Reveals

When you look at production plastic parts they have what's called a reveal. This is a gap left between parts for several reasons. One of which allows a margin for manufacture tolerances so you don't have pieces that can't fit together. Some times these reveals are made very apparent by the designer and some times they are hidden. In the case of the speaker, the main body is split right up the middle. Also there is a gap between the speaker face and the main body. The part we are drawing is less than accurate because it wont be wrapped in speaker cloth. The real part would be undersized an appropriate amount to allow room for the cloth as well as an additional gap. We wont be going that far into it because we don't have enough information so our focus is "looks". We are going to leave a 1/2mm gap between the speaker face and main body to start with and see how things look. Because the material between this part and the main body will be different, the space between the two parts will be very hard to distinguish anyways so that is why I chose such a small number.

Step19. Reveals

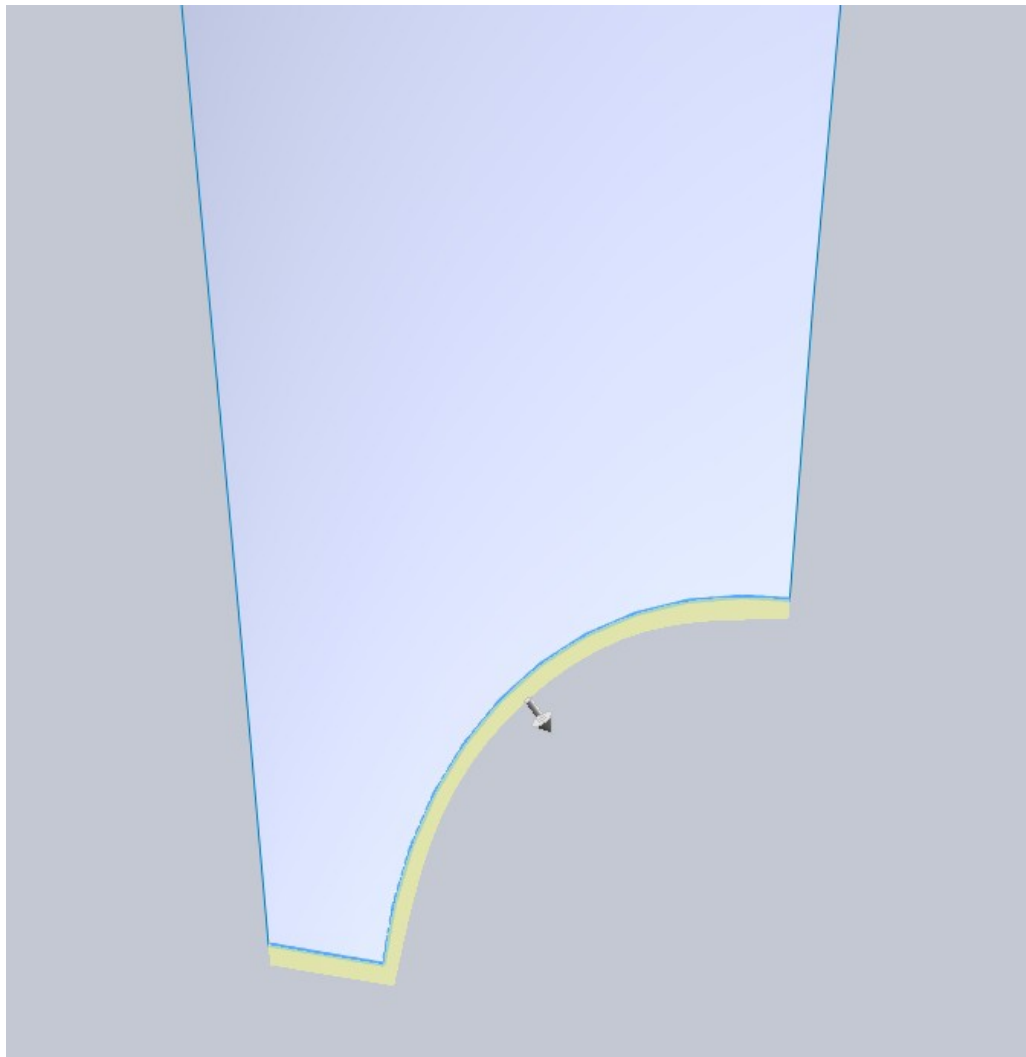
Start by creating a 0.5mm offset surface of the inside face of our speaker “cover”. This needs to be offset into this part because we are going to use it to trim away part of it.



Step19. Reveals

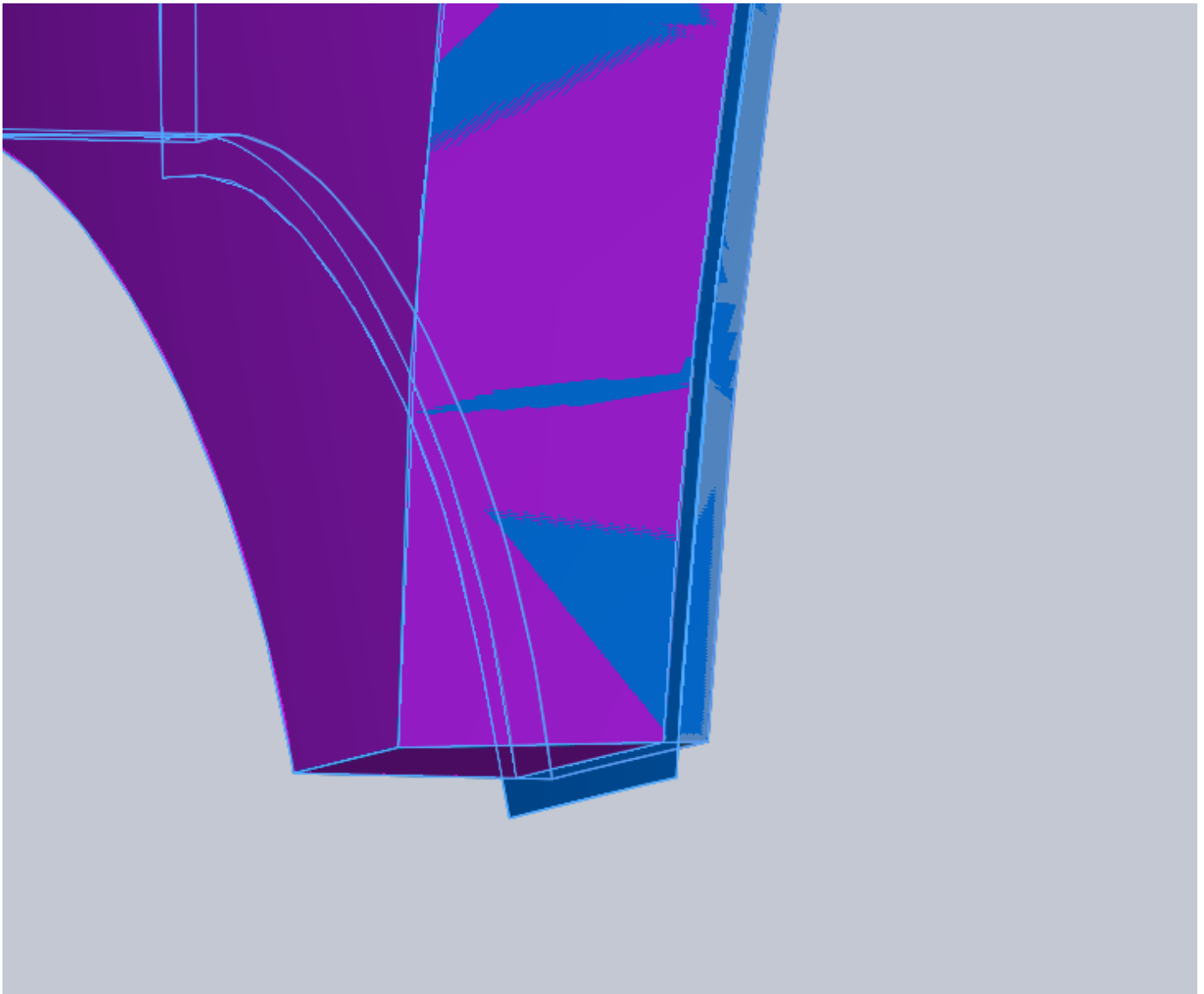
Hide everything except this offset surface. We are going to extend part of it near the bottom. The reason we have to do this is because it does not completely intersect these areas. If you try to use it as a trim surface you will get an error.

Use Extended Surface and select the two edges shown to extend them 0.5mm. This is enough to allow us to trim what we need.



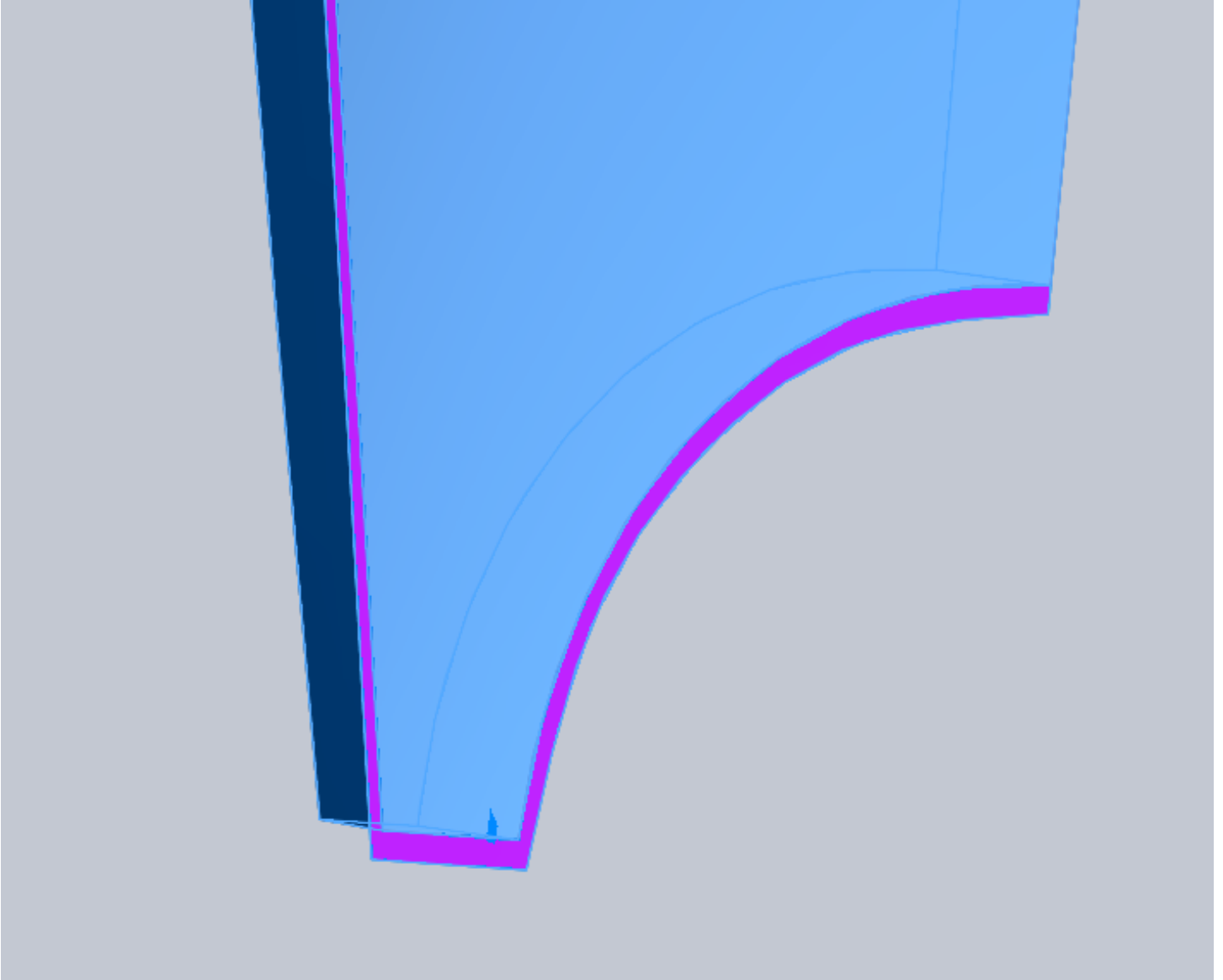
Step19. Reveals

Make the speaker face visible again and use this offset surface as a trim tool to remove the back portion (shown in blue).



Step19. Reveals

Now use the speaker face body to trim our offset surface. Remove the purple section shown below.

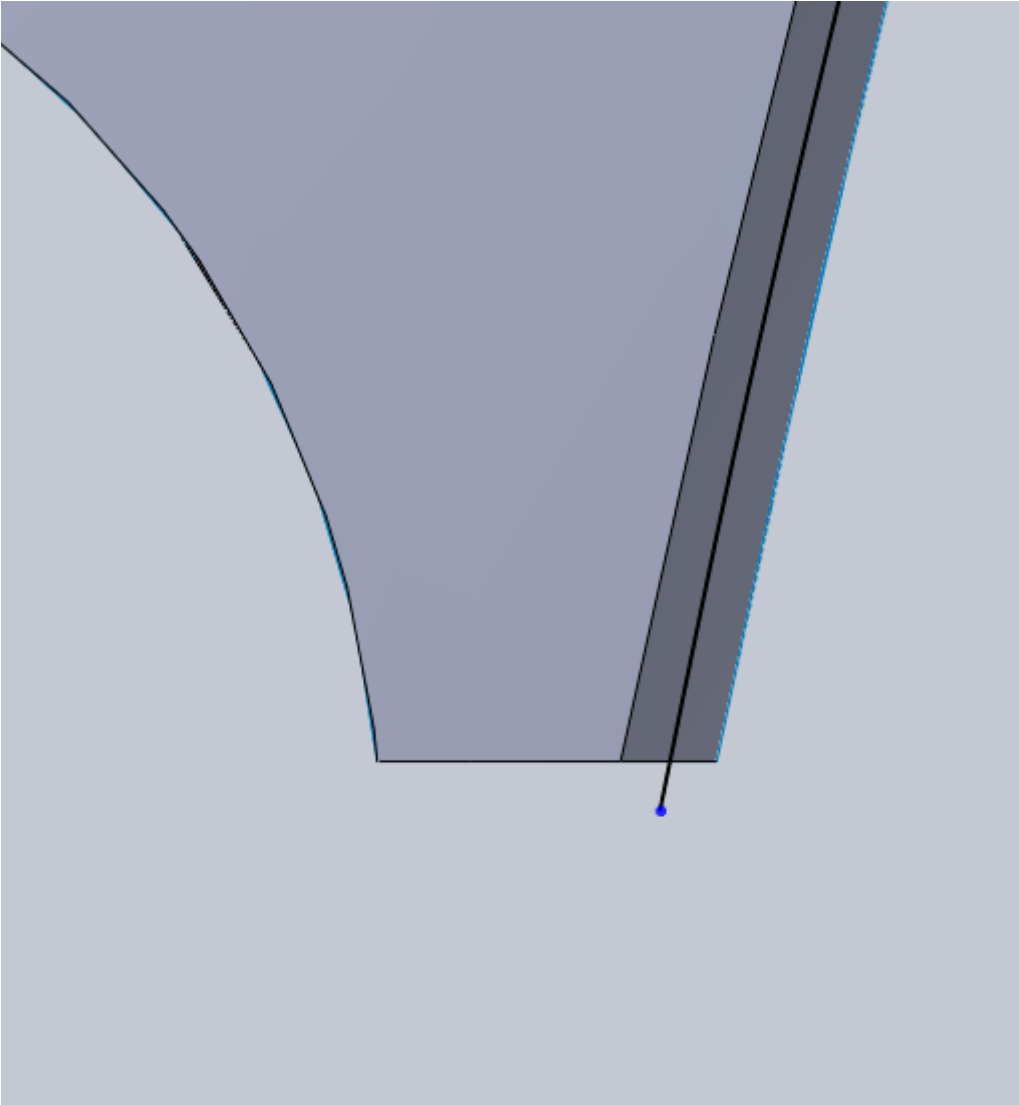


Step19. Reveals

Some of you who have played with the trim tool might be wondering why we didn't use the “mutual” option in the trim tool and take care of both trim operations at once. The reason we didn't do that is because using mutual trim will knit both surfaces together. Its very powerful, but unless you know you want these two surfaces knitted together, having the extra operation in there gives you more freedom. I have a plan for this surface which makes having it separate a benefit.

Start a new sketch on the Front Plane. Offset the outer most edge of our newest face 0.5mm inward. You do not need to use Convert Entities in order to offset this, you can simply select the edge and then the offset button in your sketch. When you offset an edge like this (or any) it is fully defined, but you can still grab its endpoints and drag them. They will stay tangent to the line, or follow the curvature of a spline. Go ahead and drag the lower endpoint of your offset edge to ensure it goes past the surfaces we are working with.

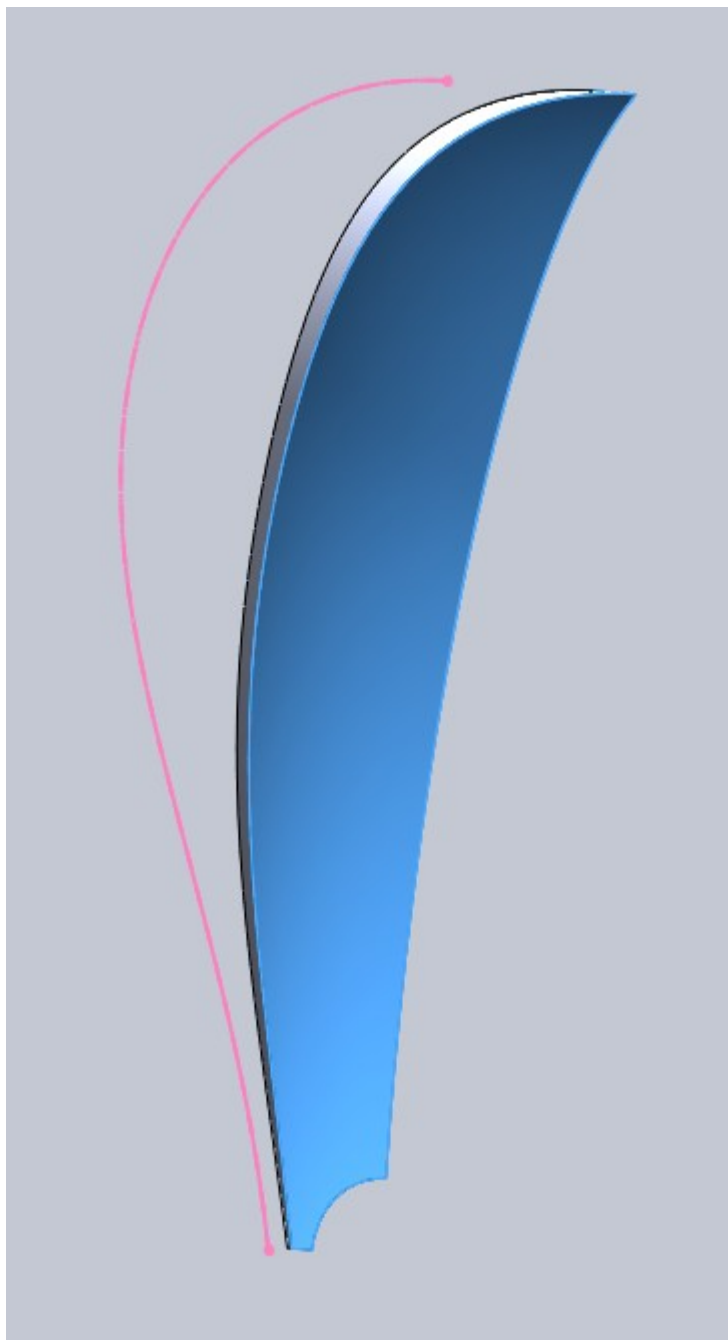
Step19. Reveals



The reason we do this is because we are dealing with offset entities and surfaces that taper in space. This edge is going to split the back face and we don't want it to end short of the edge of the face.

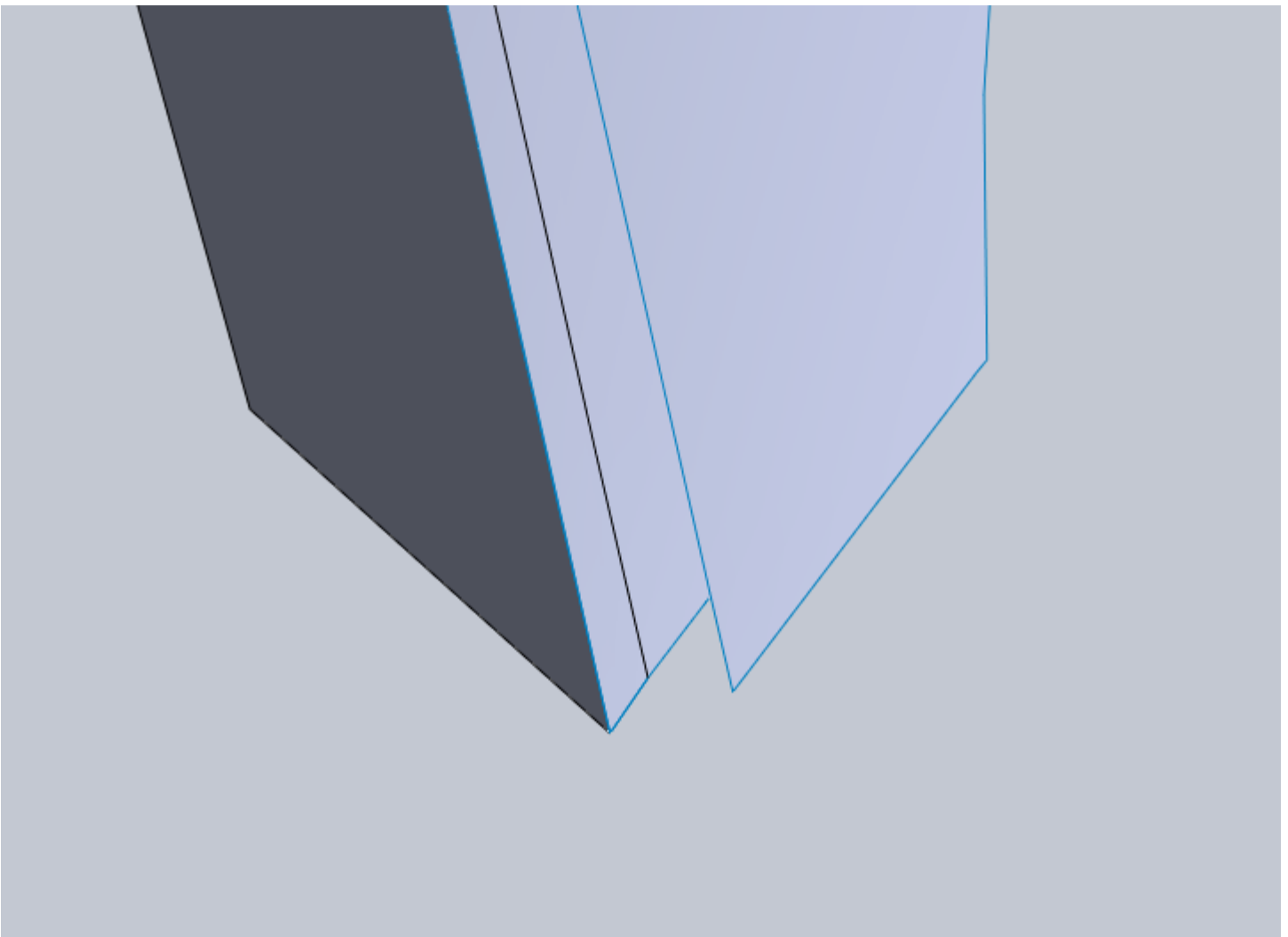
Step19. Reveals

Use this sketch as a Split Line to split the back face shown in blue.



Step19. Reveals

Now offset the larger portion of that face 0.5mm back towards the main body of the speaker. This is hard to show in a screen shot but here is the bottom corner. Notice the split line on the face and the offset surface.

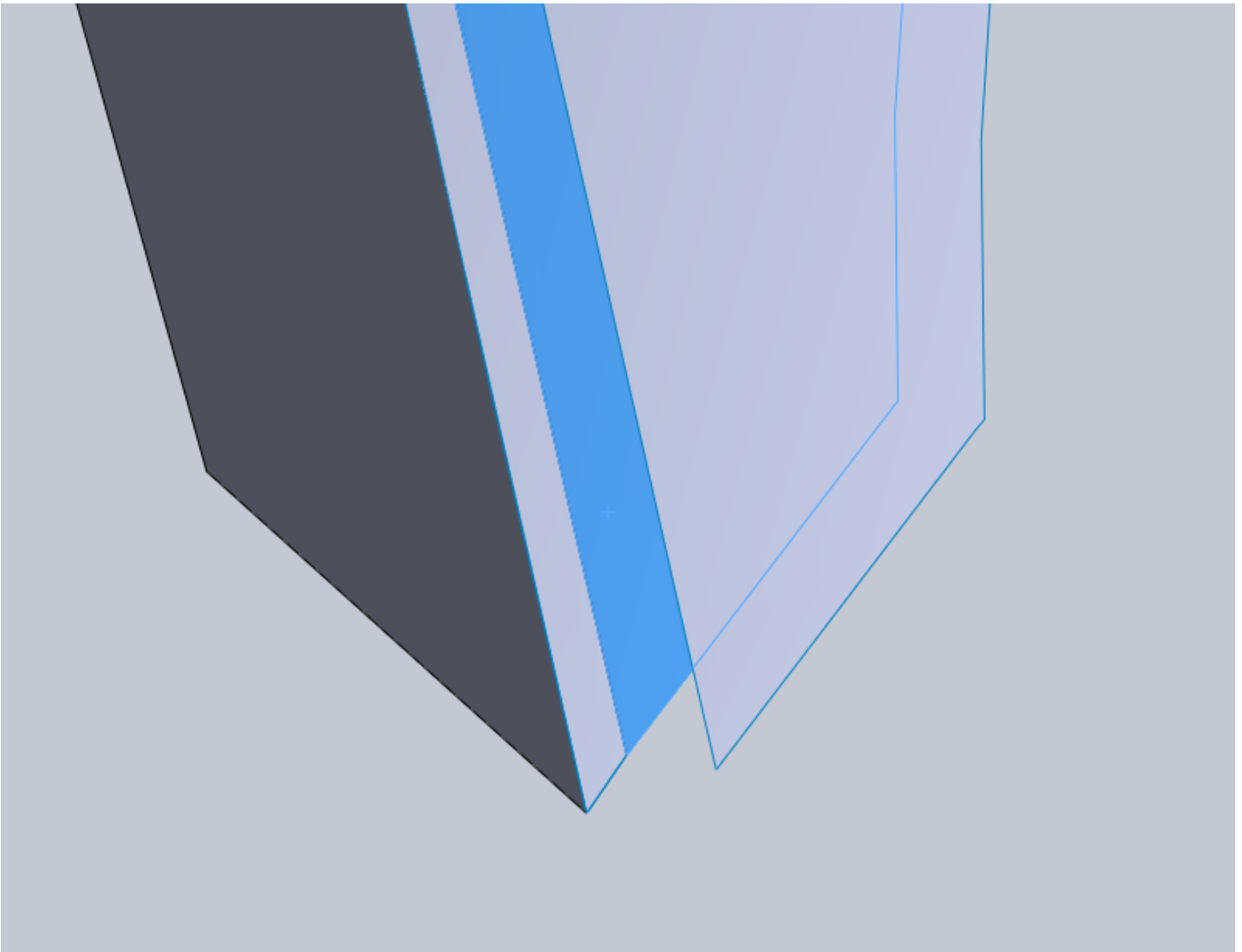


Step19. Reveals

The reason we are doing this is because we want to end up with a step at the edge of our part. Sure with the curvature of this part we could have just left a gap between it and the main body, but this way is better. Some of you might wonder why we offset one surface, trimmed it, then offset our new surface back after a split rather than just splitting the first. The main reason is because looking from the Front Plane we drew the shape of the outer surface to be slightly smaller than the inner one. This means when we offset that first surface 1/2mm we came up with a whole new edge. We could have offset the original edge and really it would have looked nearly the same. Because our inner and outer edges for these faces were two different splines, I wanted to use the actual trimmed edge for mine. Again these types of things are judgment calls. Some times there isn't a good reason to do one over the other. Some times you will try one, it wont work and you will have to go back and try another. Only practice will let you decide which is best but you should always be thinking of alternative options when modeling anything.

Step19. Reveals

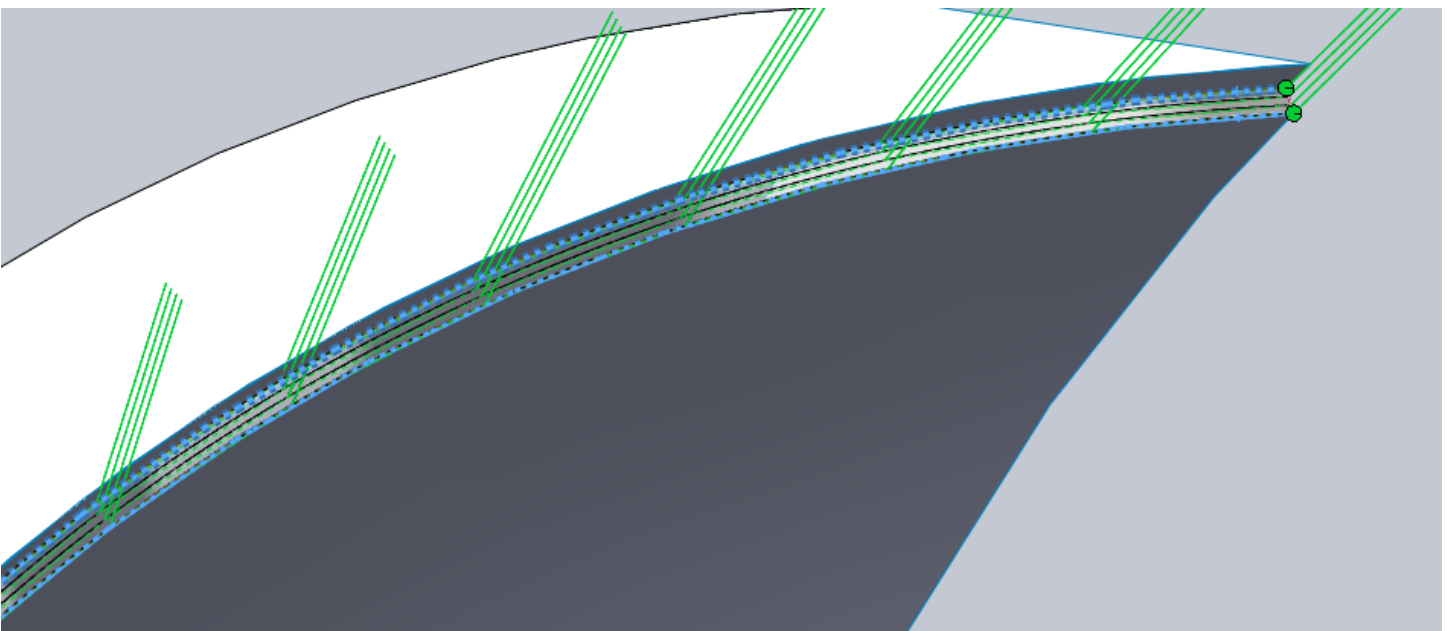
Now use the Delete Face command and remove the face shown in blue. Since our recently offset surface will be the “back” of our speaker face, there is no need for this inner surface.



Step19. Reveals

Now we want to make a boundary surface to finish off this reveal. There is one small issue that may arise with this. We are dealing with several different surfaces now so edges may have some discontinuity. Knitting surfaces together fixes these so we are going to approach this boundary a little different. If you try to create a single boundary surface you may get a curvature error. I encourage you to first try that using the selection manager and selecting the proper edges. If it works that's great, if not this is a valid second option.

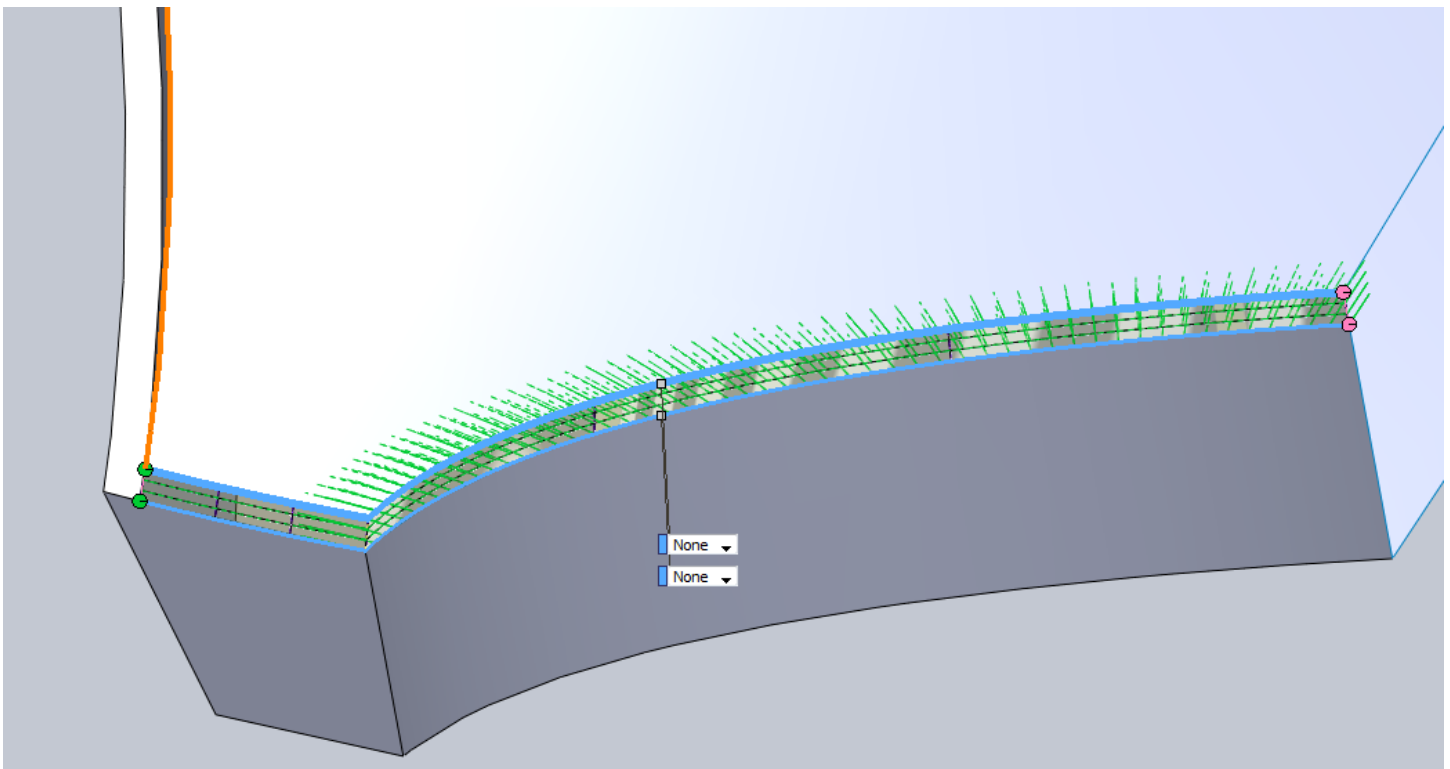
Start with a single selection boundary surface going from our split line to the offset surface.



Step19. Reveals

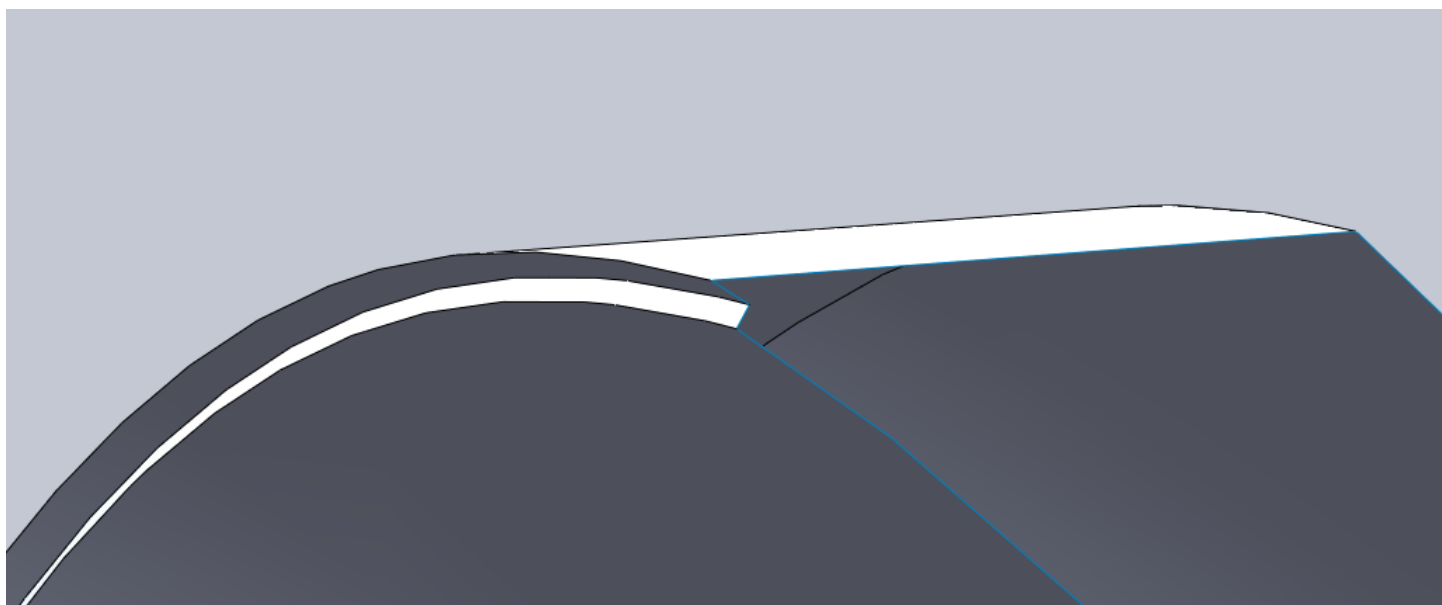
Now knit that boundary surface, the offset surface and remaining bodies(4 surfaces total) together. This will heal some edges where curvature may cause a problem. I suspected that the large spline edges were the ones causing the issue which is why I chose those as my first boundary surface before the knit operation.

Now create a boundary surface using the selection manager and selecting the bottom edges.



Step19. Reveals

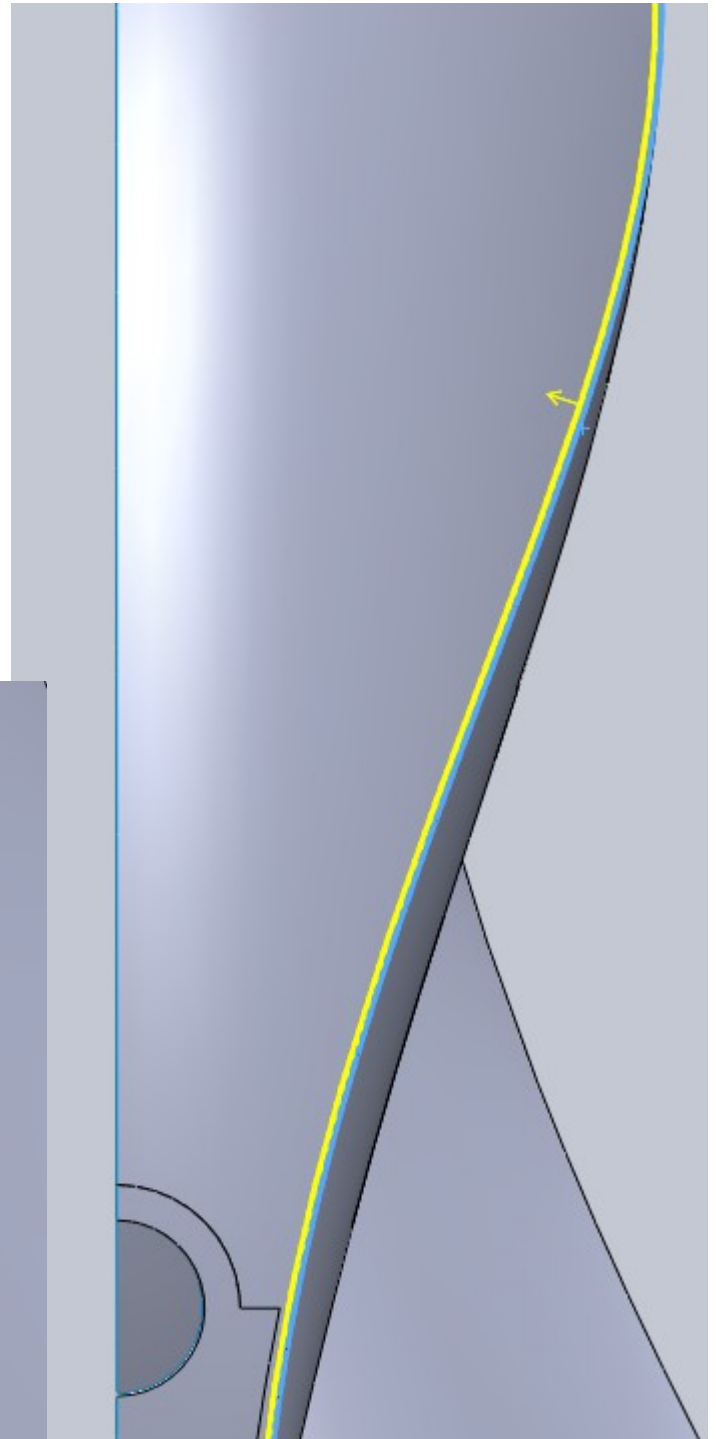
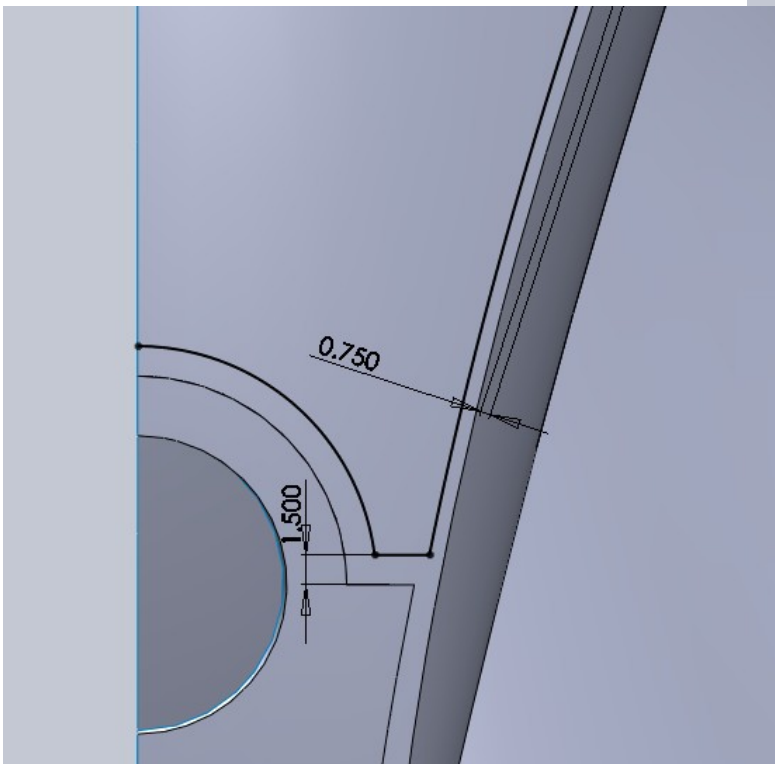
Knit that boundary surface to the rest of the knitted surface. You should now have a single surface for the speaker face with a reveal lip all the way around it (except at the bottom). We didn't need one at the bottom because we already created a space between this part and the area around the button/led.



Step19. Reveals

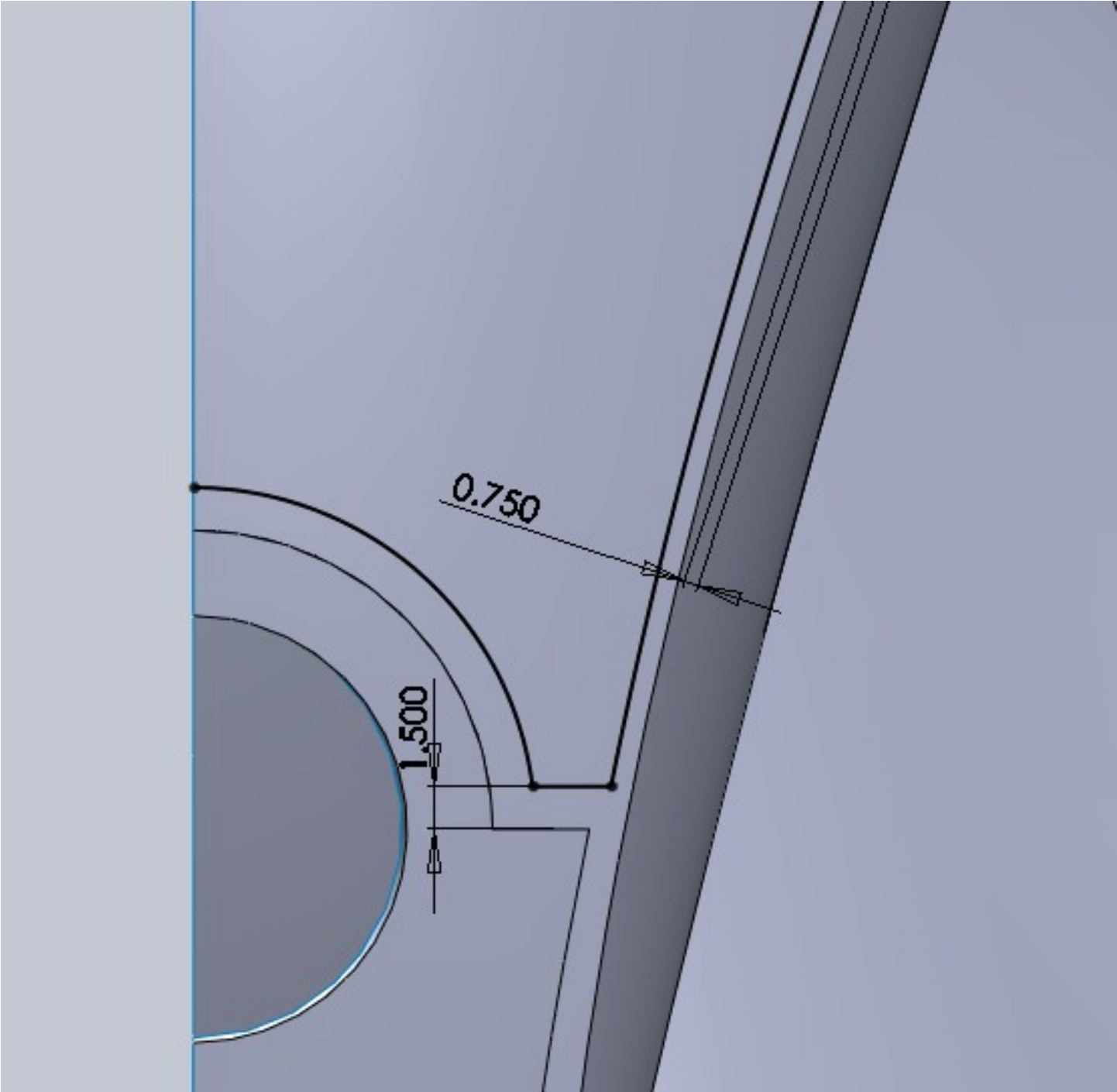
Moving onto the main body. Hide the speaker face and make the main body visible. Start a new sketch on the Front Plane. We need to offset a few edges. First offset the main shape of the speaker 0.75mm.

Next offset the arc and line around the power button 1.5mm.



Step19. Reveals

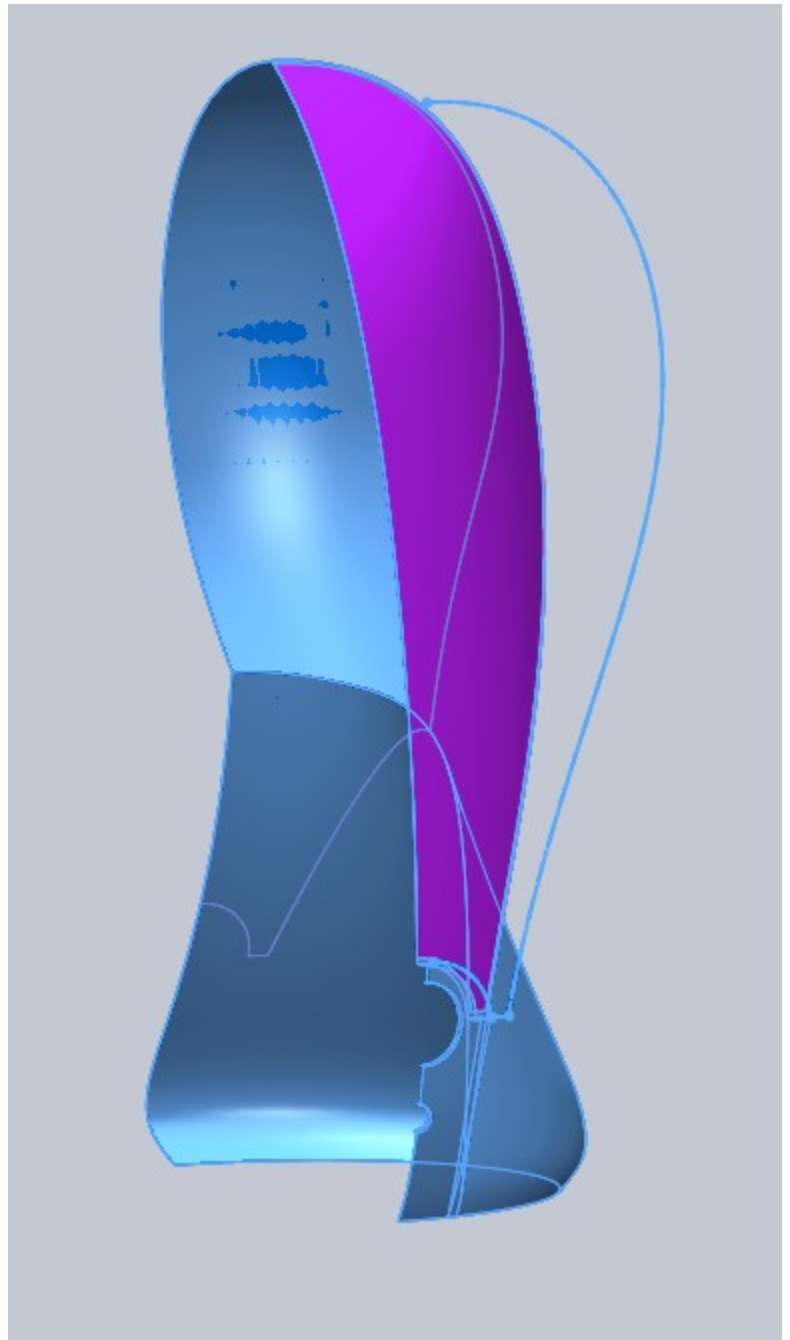
We need to extend the straight line and trim everything so you are left with this.



Step19. Reveals

Use this sketch as a trim tool and remove the purple section pictured below.

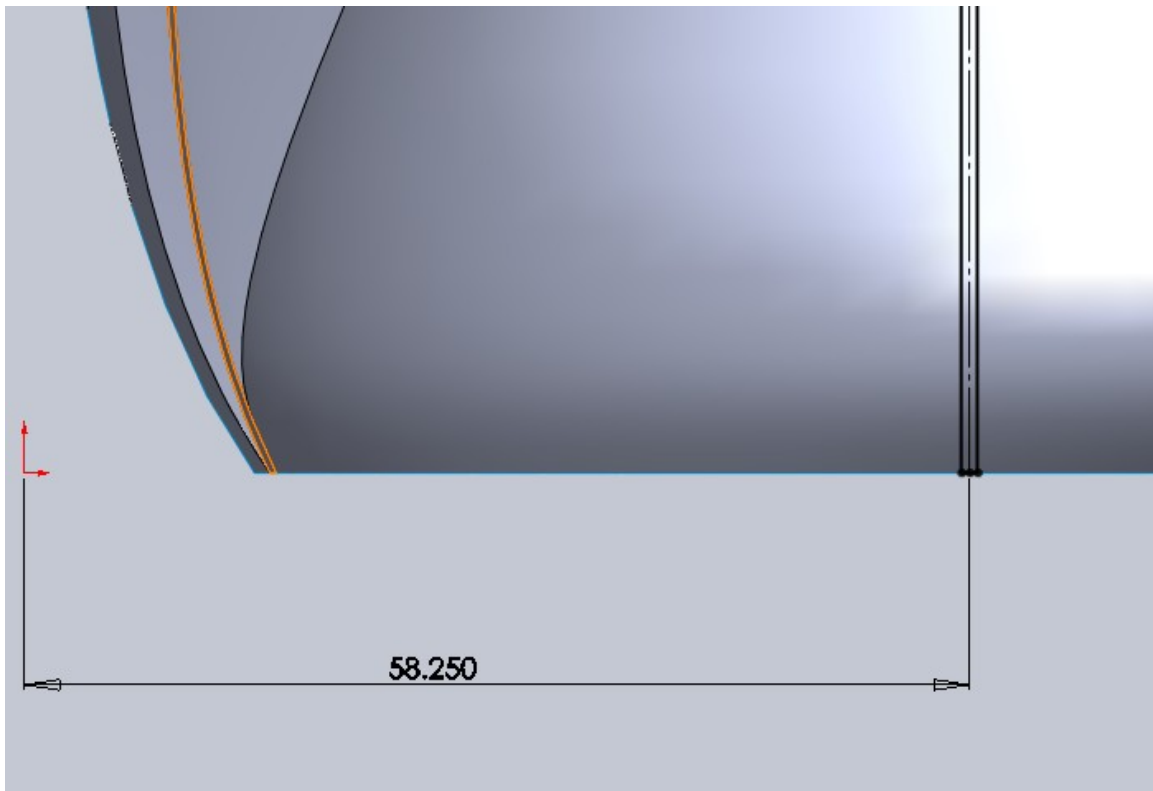
The reason we need to trim this away is because it will make creating a reveal very difficult. Or at least more difficult that it needs to be for this tutorial.



Step19. Reveals

Start a new sketch on the Right Plane.

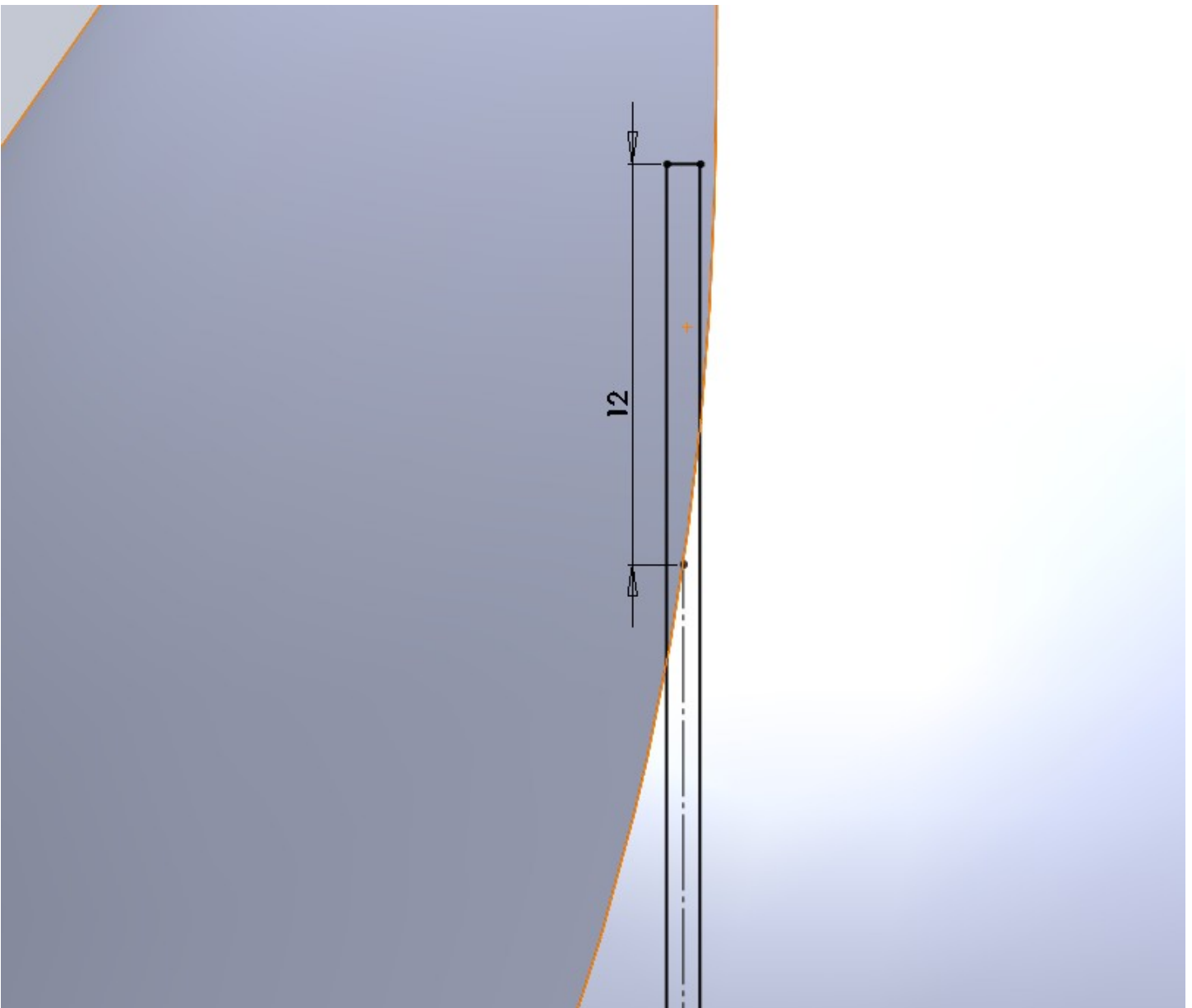
We are going to draw a rectangle that we will use to split our main body for a reveal. Start by drawing a vertical construction line. Make its end points coincident with the base edge and upper edge shown.



Use a bi-direction offset with a dimension of 0.5mm. Draw a line to join the two offset lines together at the bottom. At the top we will need to add a few lines.

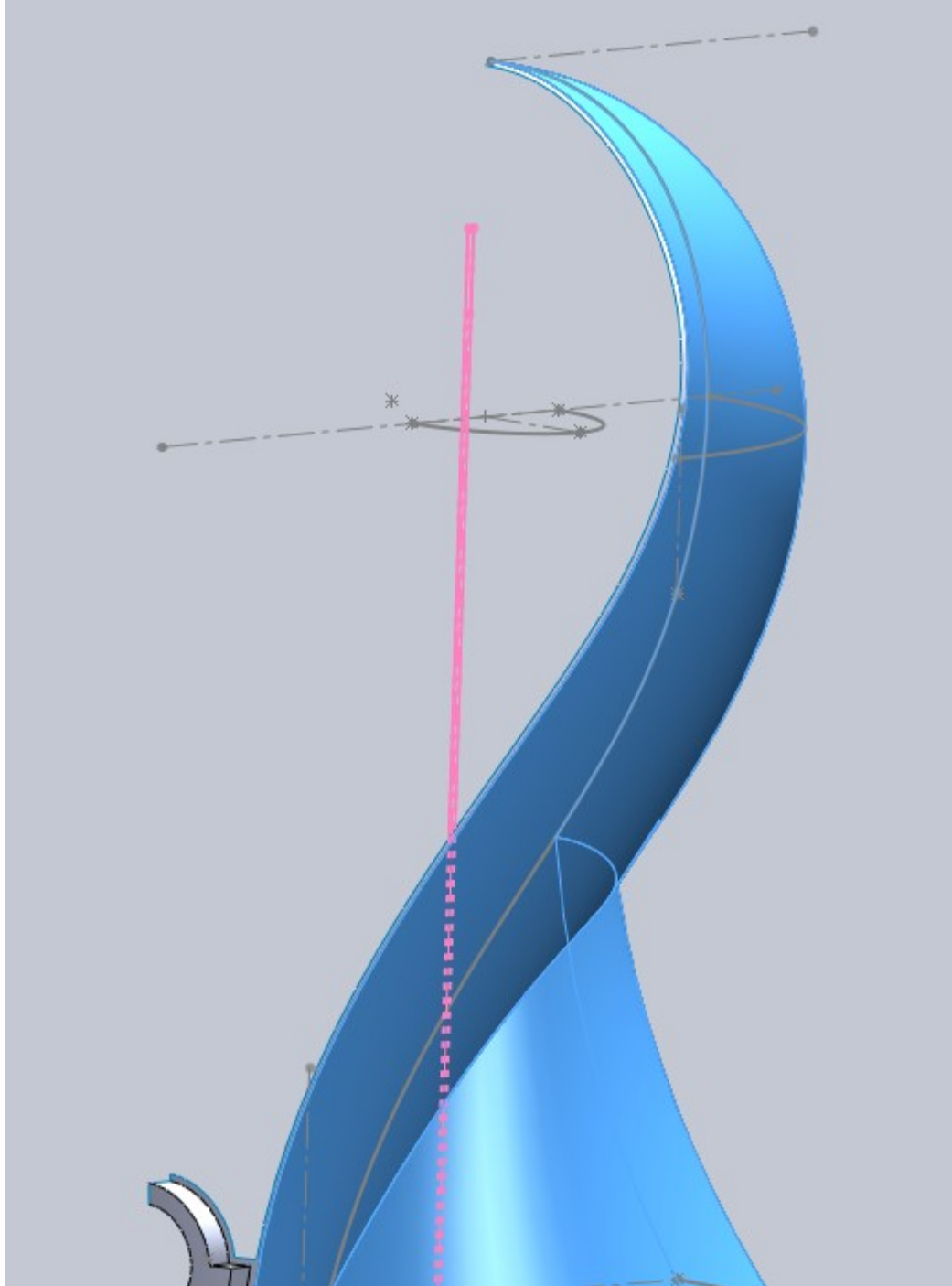
Step19. Reveals

You can either add lines or “pull” the endpoints of the offset lines up. I added a 12mm dimension but the important thing here is that the lines go past the edge shown in orange. The reason is because we need our split to split the entire face.



Step19. Reveals

Use the sketch as a split tool and split only the 2 faces shown in blue. We are not splitting the small lip left by the previous trim operation with this sketch.



Step19. Reveals

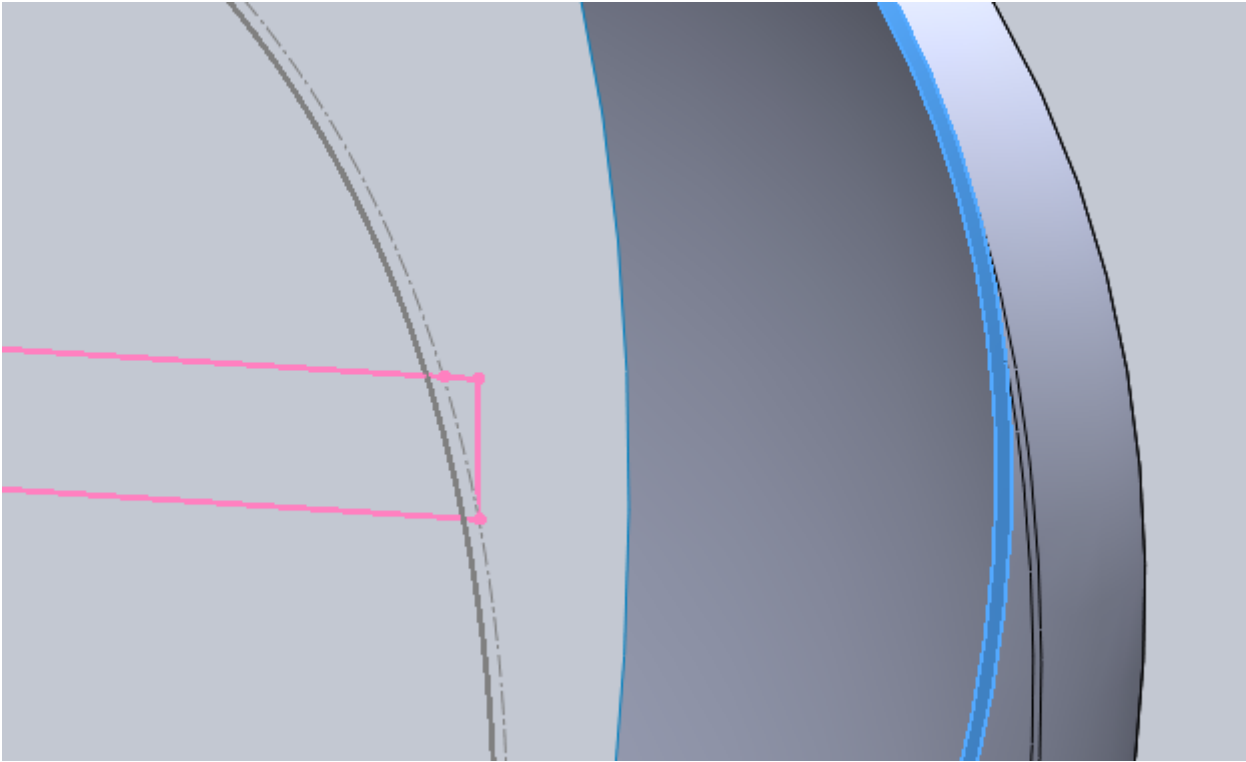
Start a new sketch on the Front Plane.

We are going to draw another rectangle this time using the edges of our split line for coincident relations with our rectangle. For simplicity sake I make my left vertical line above the origin even though we do not need this large of a rectangle. It allows me to use no dimensions. You can also see the top edge is made up of two lines. This could have been done with one that went through our reference point but this works as well. Since this sketch is fully defined with no dimensions, this extra line will not cause any undesirable results during updates.

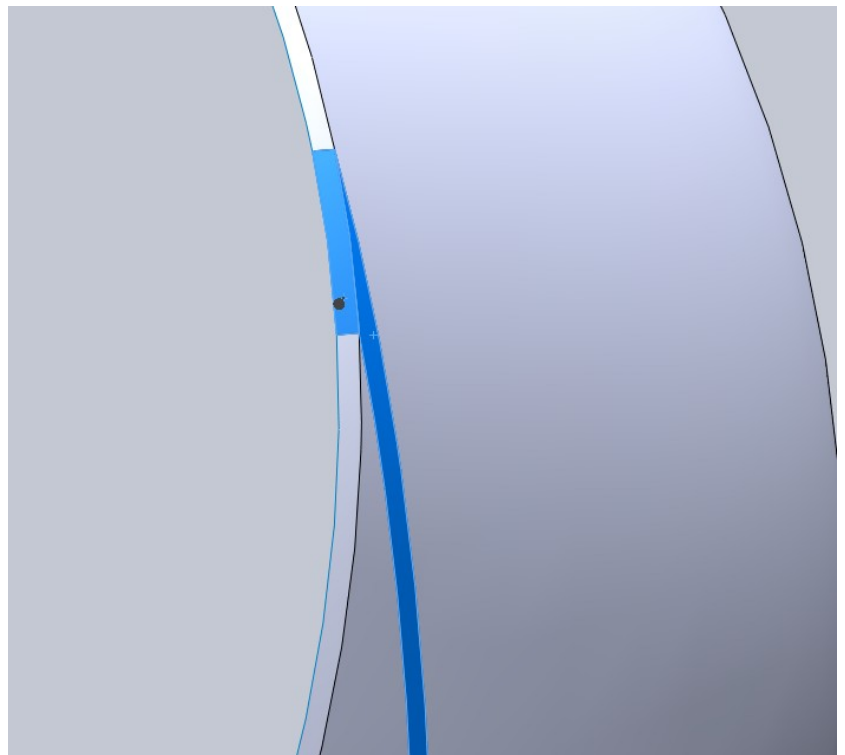


Step19. Reveals

Use this sketch to split the face as shown.



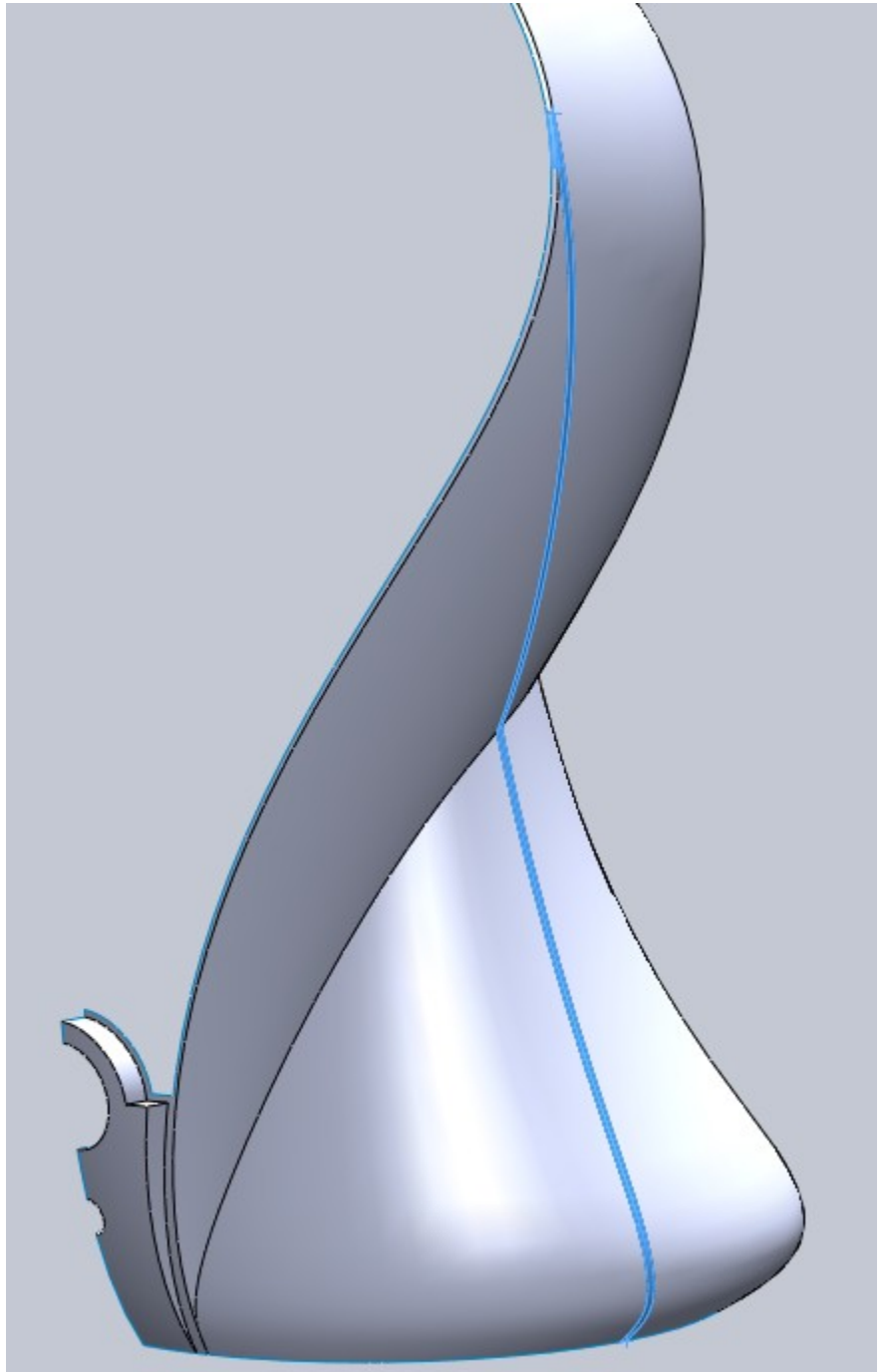
The end result is a set of split faces that intersect properly. If we used our first sketch to split all 3 faces, the results would have been less than satisfactory because of the curvature.



Step19. Reveals

Use Delete Face and delete all 3 faces that we have just created with our split.

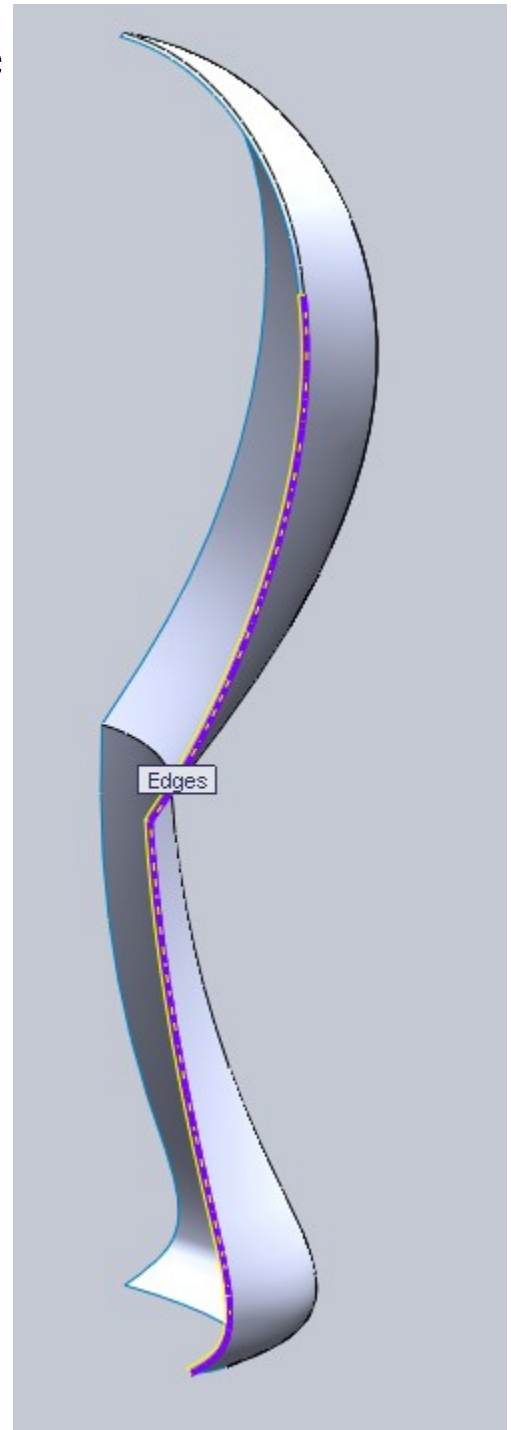
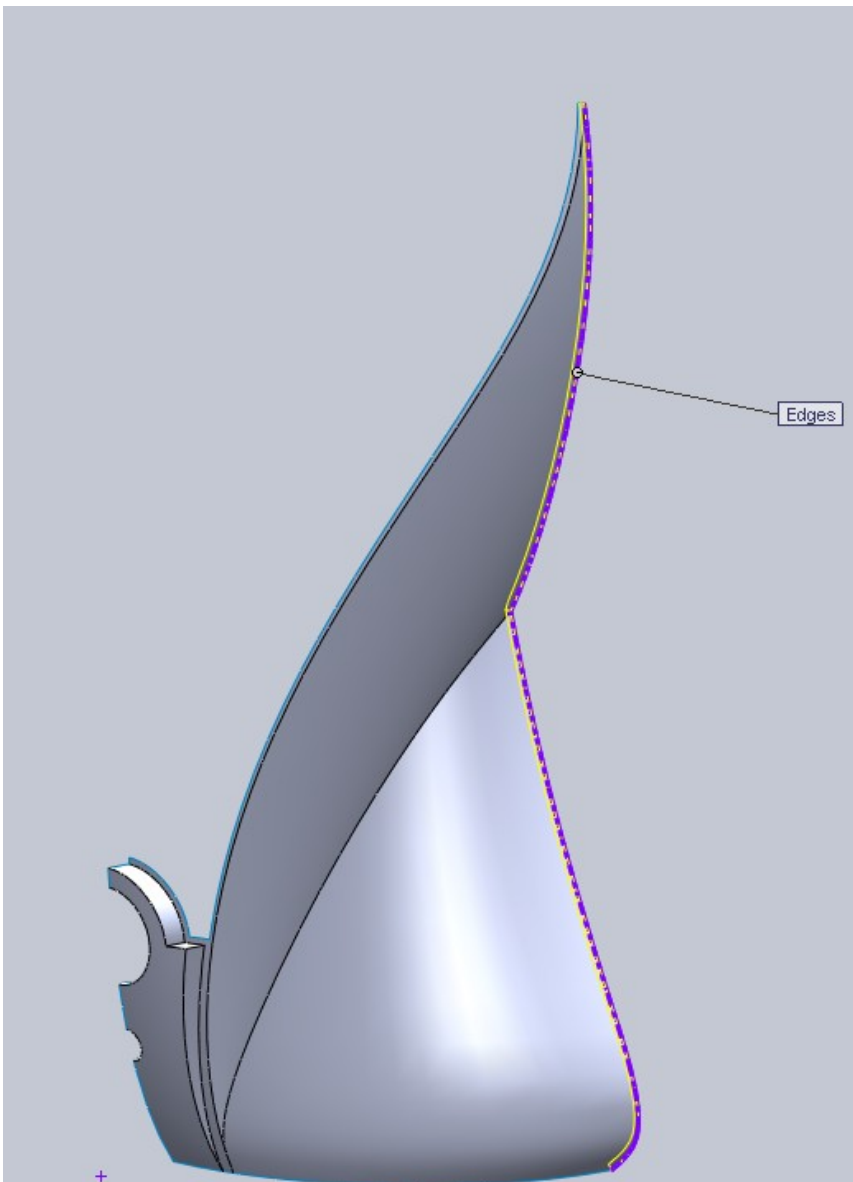
This will turn the single speaker body into two separate bodies since these faces are the only thing keeping them together. We want to do this because we want to have a front and back of our speaker body.



Step19. Reveals

Create a 1mm Rule Surface using “Normal to Surface” type. Select the two edges shown on the back half of our speaker body.

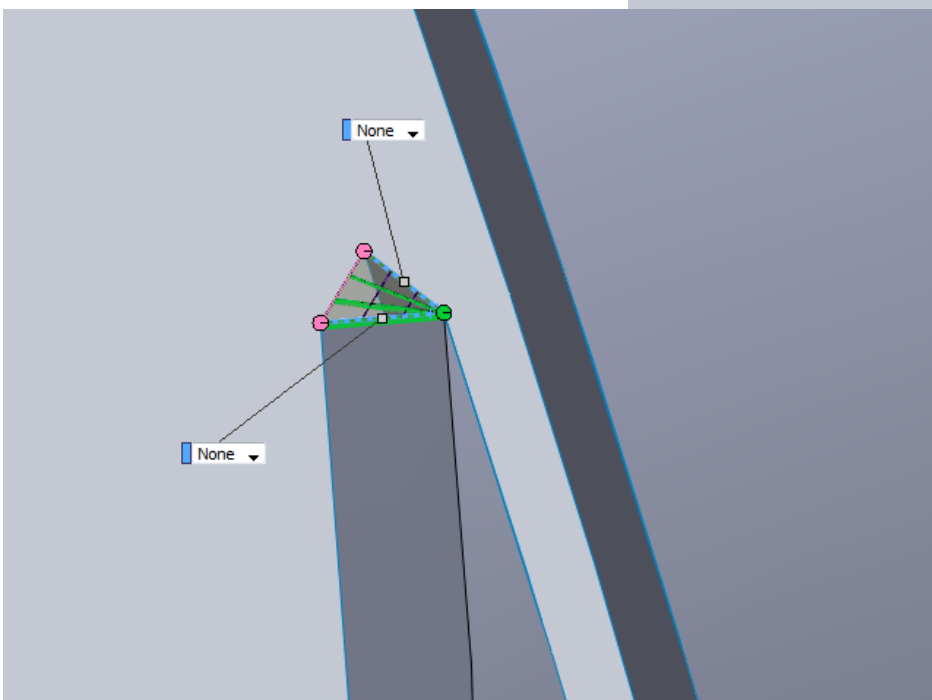
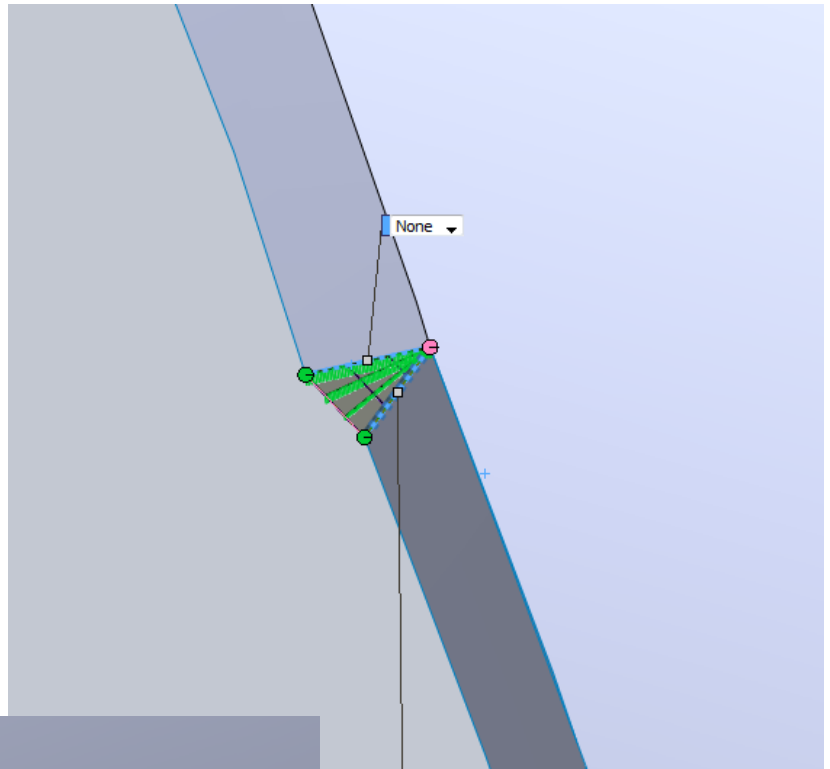
Create another 1mm Rule Surface for the same edges on the front half of the speaker body.



Step19. Reveals

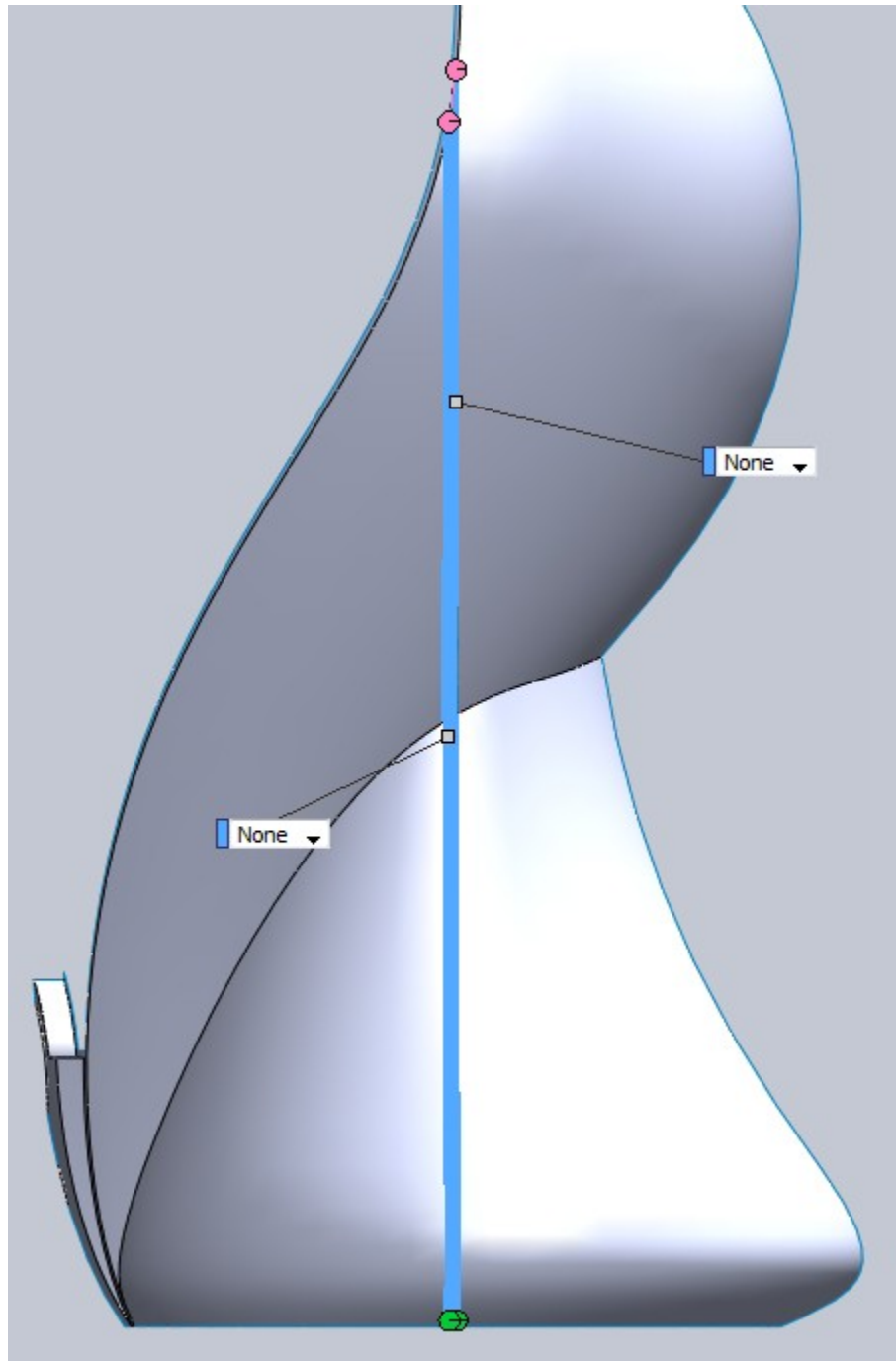
Now we need to fill in the small gaps between the rule surfaces and the rest of the trimmed surfaces. At the top on the back half of the speaker body there is a small triangular section. Create a Boundary Surface to fill this in.

Do the same for the top edges on the front half of the speaker body.



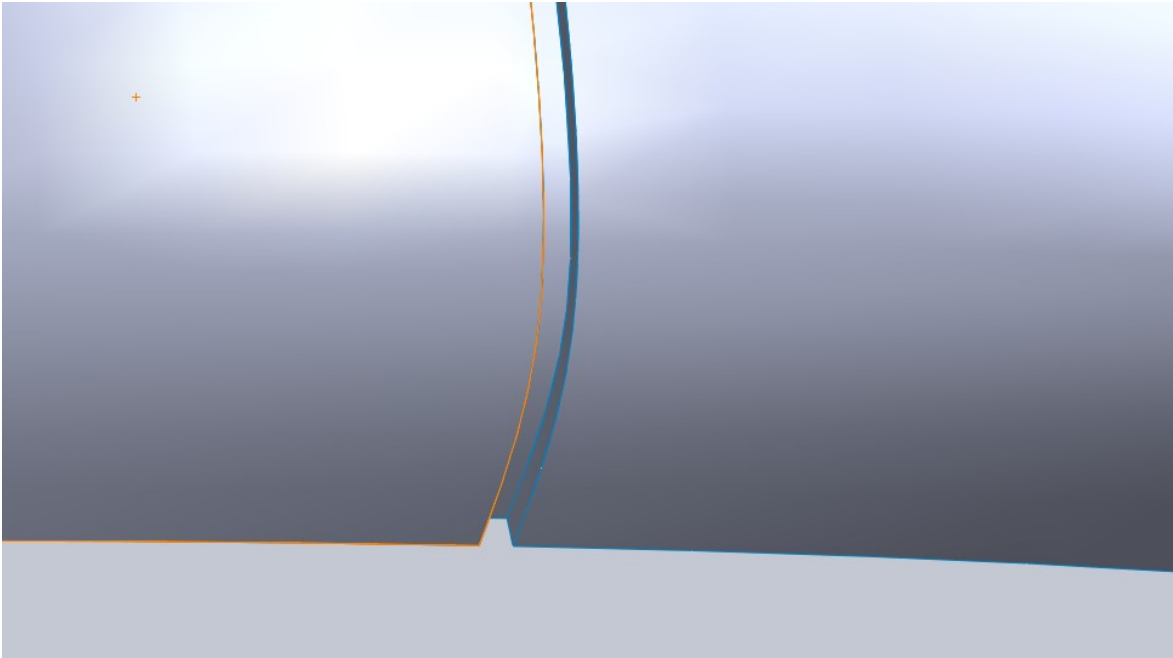
Step19. Reveals

Now create a boundary surface between the two ruled surfaces to fill in this reveal area. You will need to use the selection manager and select two edges for each selection.



Step19. Reveals

Here is a close up at the bottom of the bodies to show you what we have done.

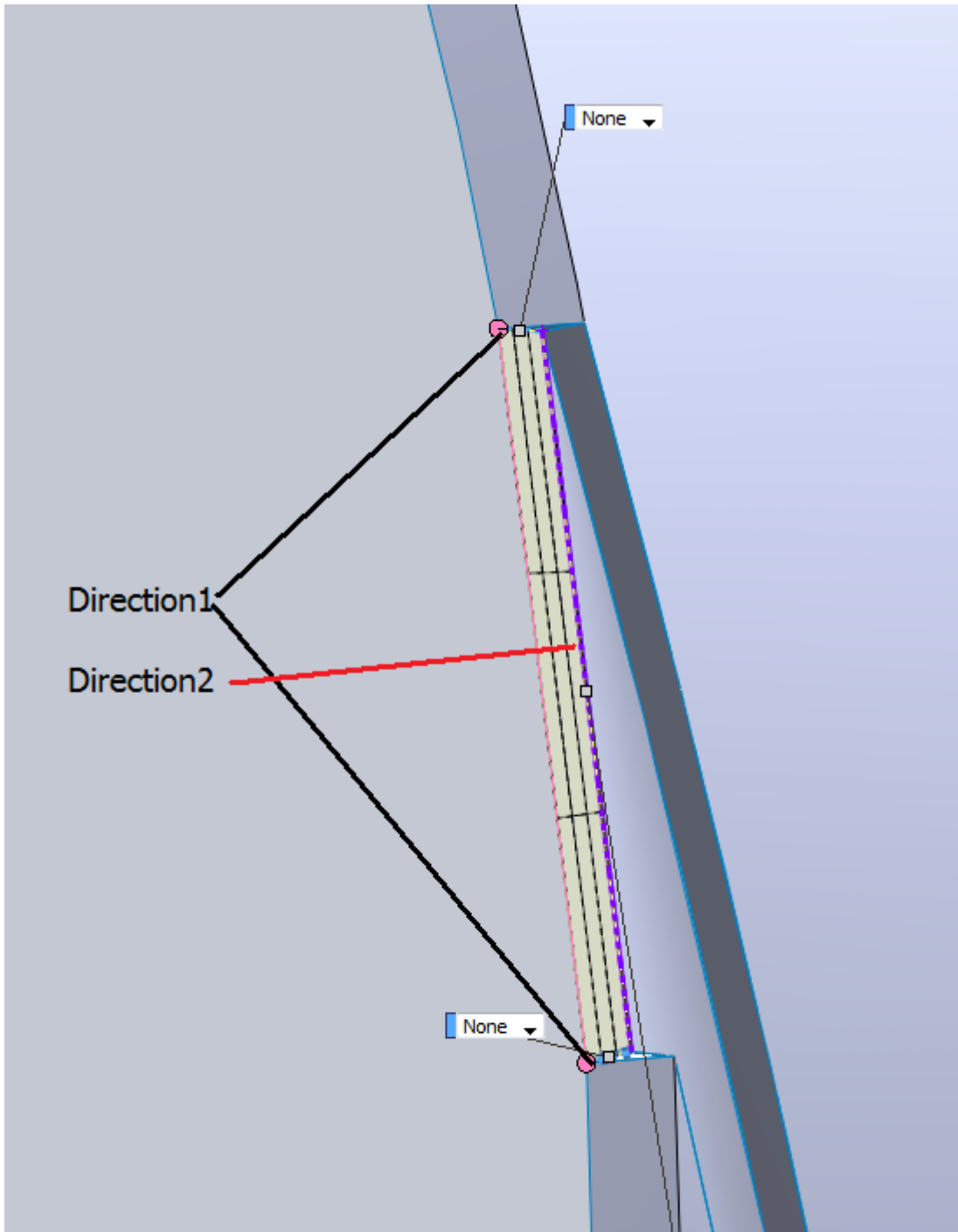


But we need one more piece to finish off this reveal. At the top we will need one more boundary surface. This surface will have selections in both the Direction1 and Direction2 box.

Think of the Direction1 selections as the From and To curves. We will make a surface between those. Direction2 selection will give the surface a path to follow between the two Direction1 selections.

Step19. Reveals

This surface fills in the last gap we have at the top.



Step19. Reveals

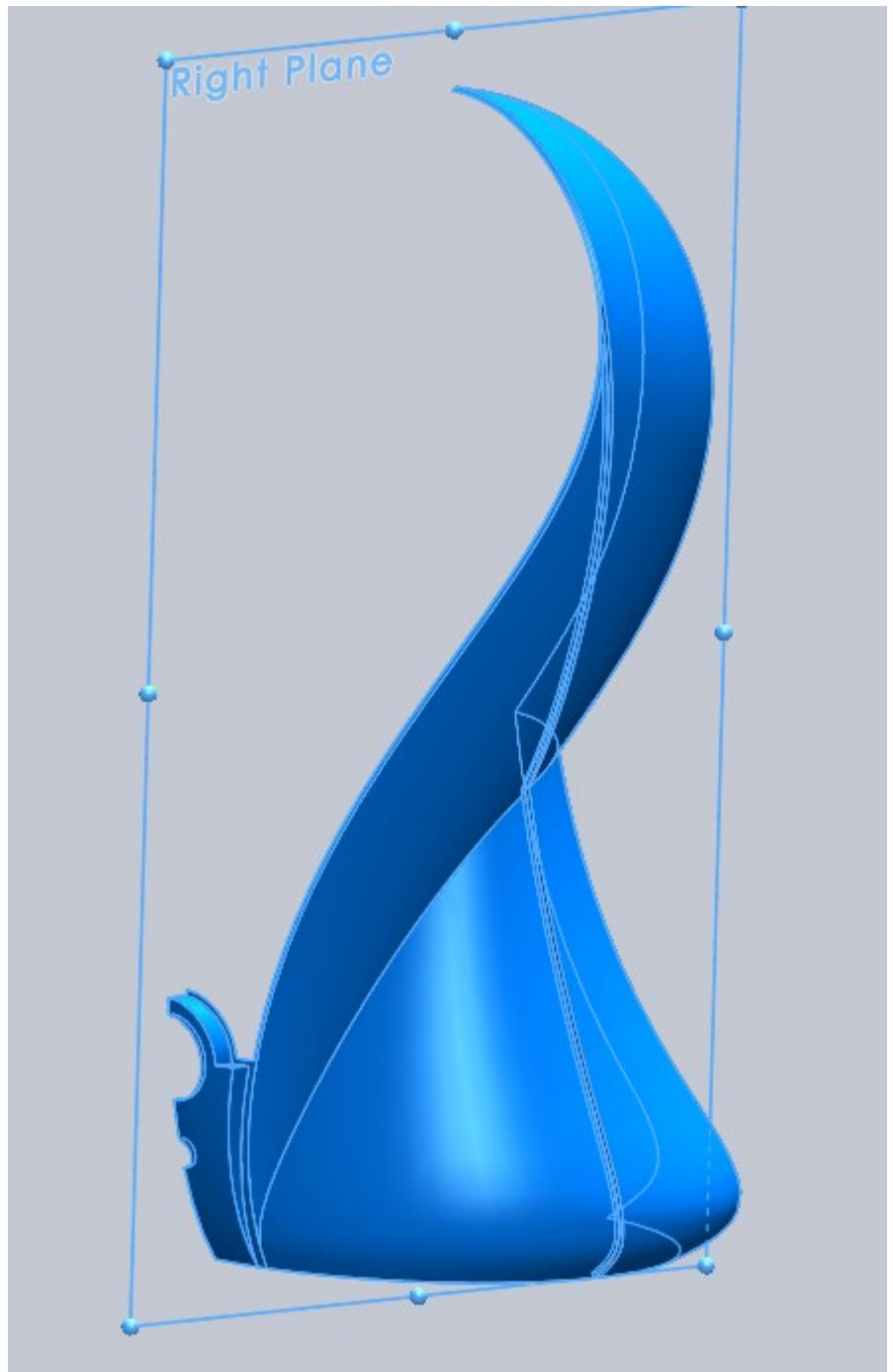
Earlier I mentioned we wanted to split the speaker body up into two pieces. Since we are only creating a visual representation we are going to knit both pieces and all these rule and boundary surfaces back together to make one body again. Again I want to note the difference between modeling for “looks” and modeling for production. The purpose of this is only to show you how to work with the surfaces to create these parts and not actually give you all the info to produce them in reality. Knowing how to work with and manipulate the surfaces is the first step.

At this point we have two surface bodies. The main body and the speaker face. We have a little bit more of the speaker to produce but at this point we can go ahead and mirror what we have in order to make the remaining surfaces a little easier.

Step20. Mirror

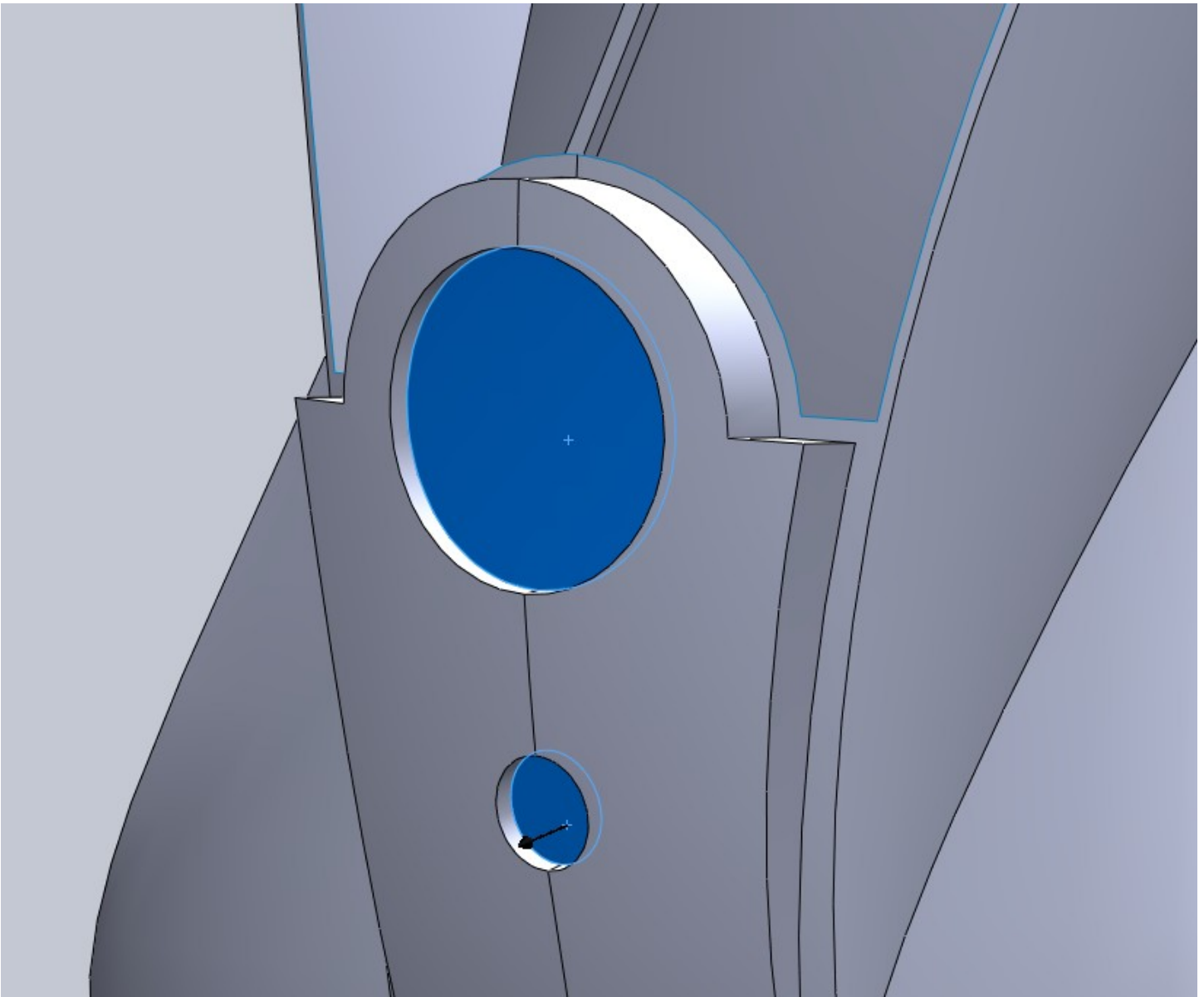
Mirror the main speaker body across the Right Plane and knit the two surface bodies together.

Note: you can use the Knit Surfaces option during the mirror or perform a knit after the mirror operation. I like to separate these during the tutorial to remind us what we are doing, but you can reduce the size of your feature tree by combining operations like that.



Step21. Filled Surfaces

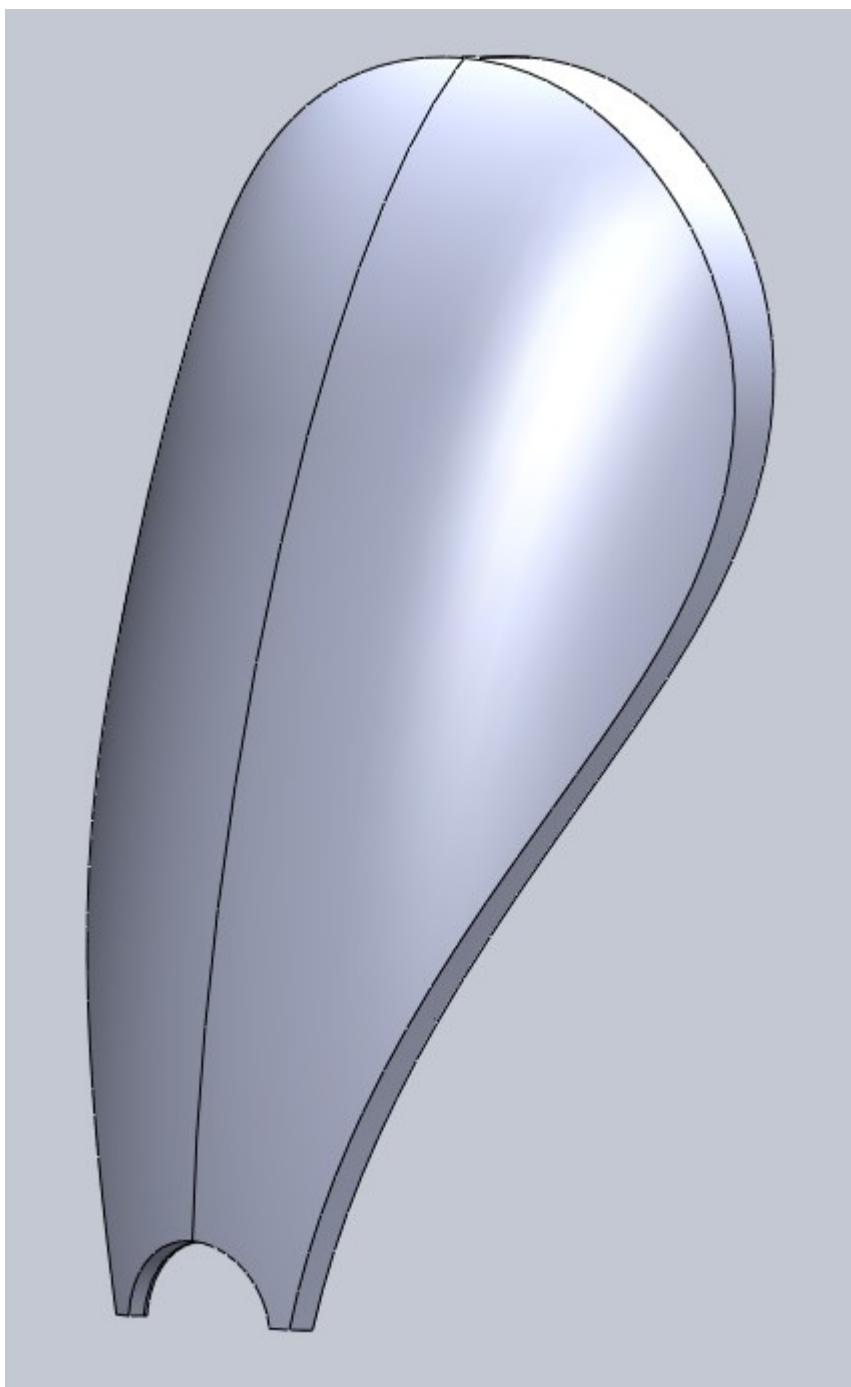
One of the reasons for the mirror was to more easily fill the power button and led hole. Using Filled Surface make your selections to fill both holes. These will be two separate Filled Surface operations.



Knit these two filled surfaces to the rest of the body.

Step22. Mirror and Knit

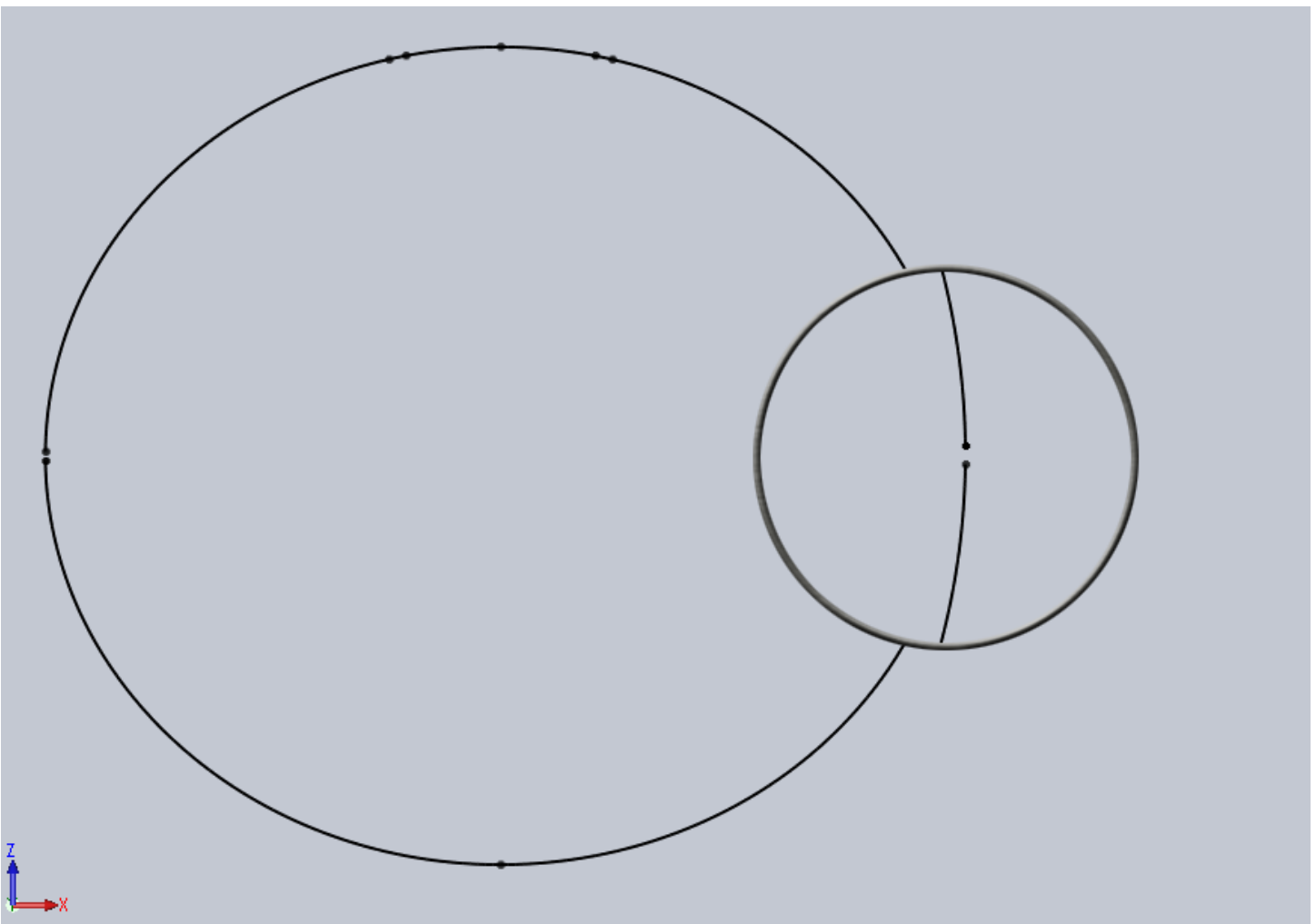
Now mirror the speaker face surface and knit.



Step23. Finishing the Bottom of the Speaker

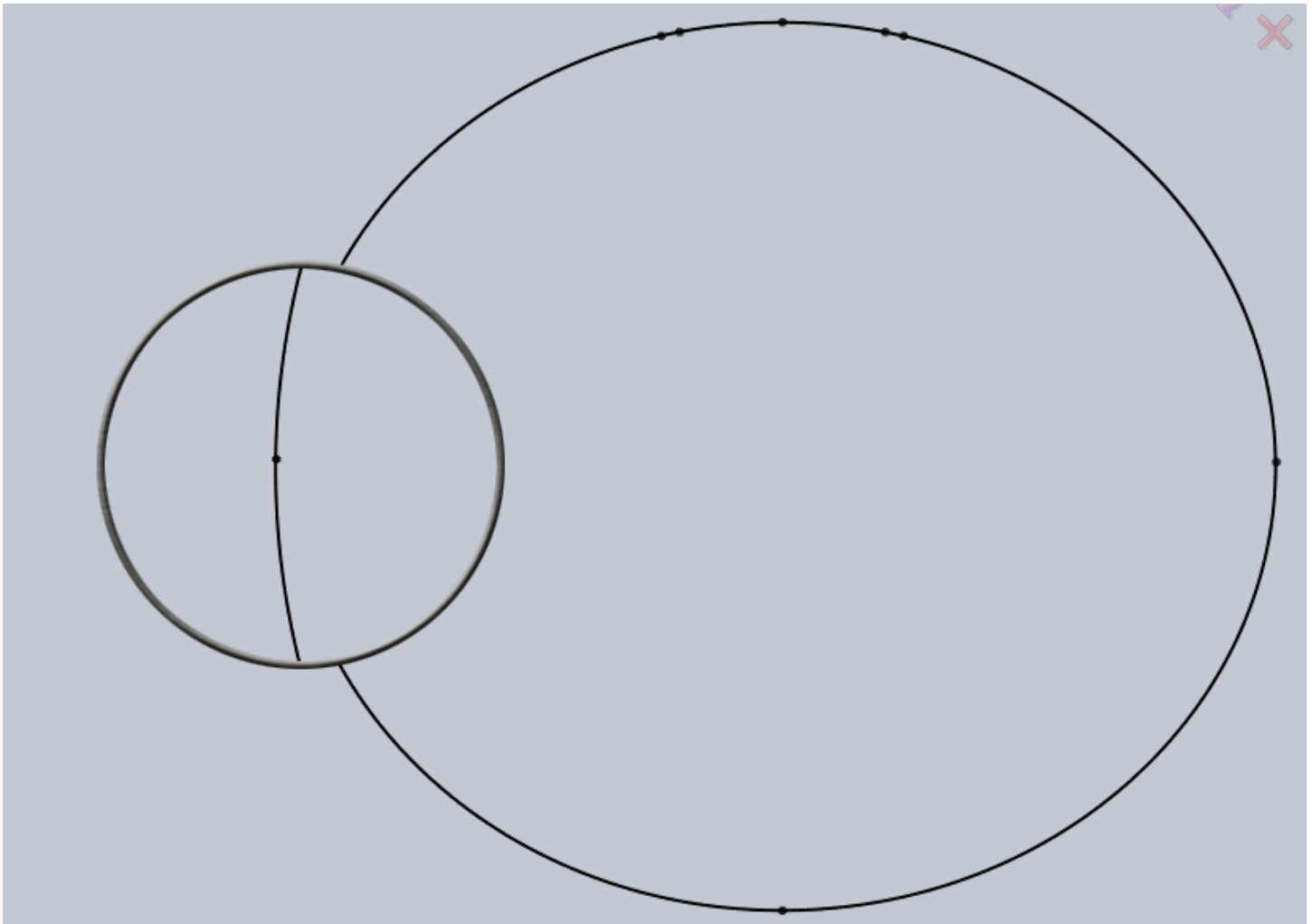
Start a new sketch on the Top Plane.

Use Convert Entities to convert all of the edges around the base of the speaker. There will be two small gaps where we created our reveal.



Step23. Finishing the Bottom of the Speaker

To fix the gap you want to grab the lower curves end points and drag them up to merge the two.

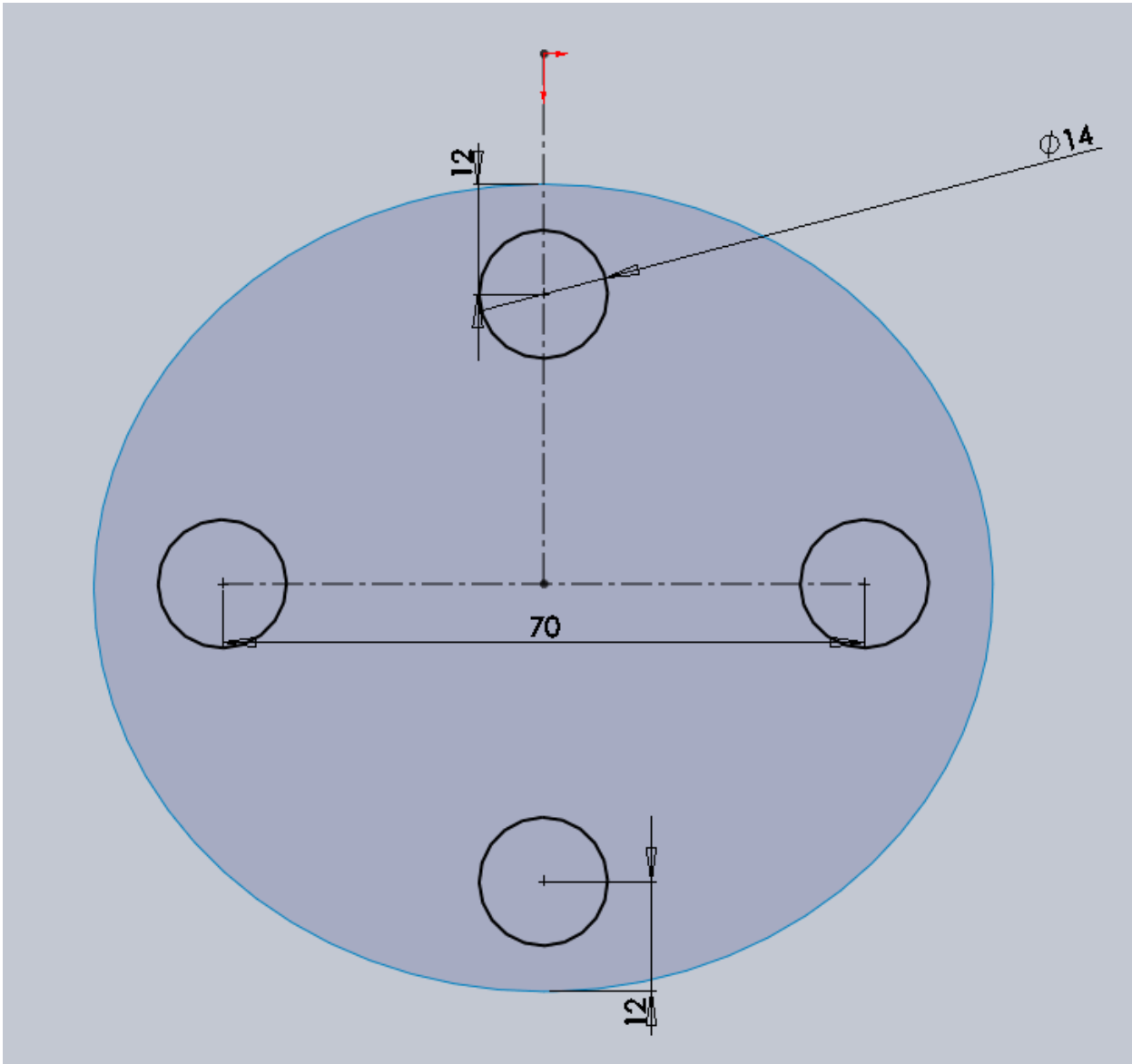


Once the sketch is closed, use it with the Planar Surface operation.

Step23. Finishing the Bottom of the Speaker

Start a new sketch on the Top Plane.

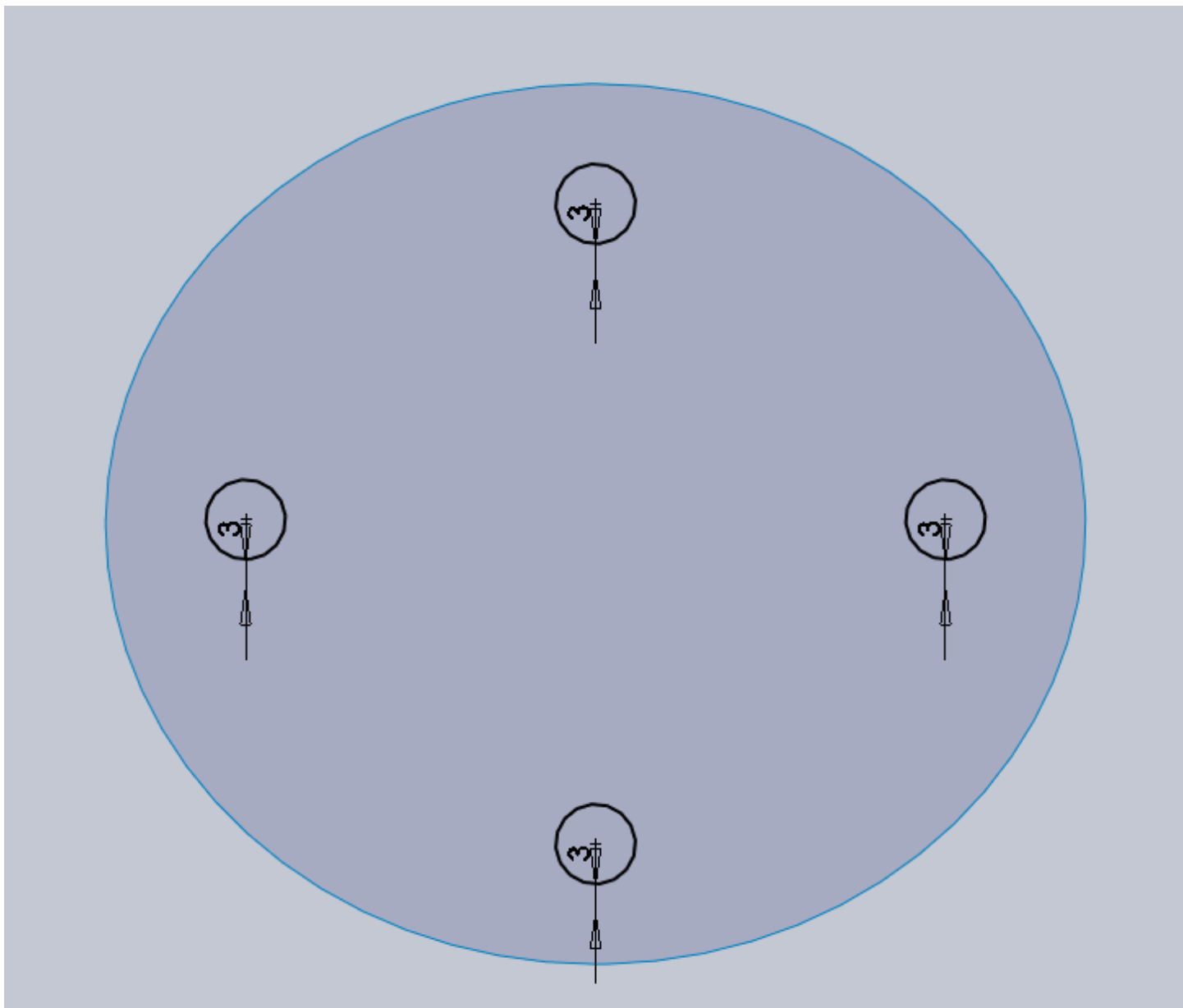
Create the sketch shown below.



The 70mm wide holes are horizontal to the midpoint of the ellipse. The vertical holes have a vertical relation with the origin.

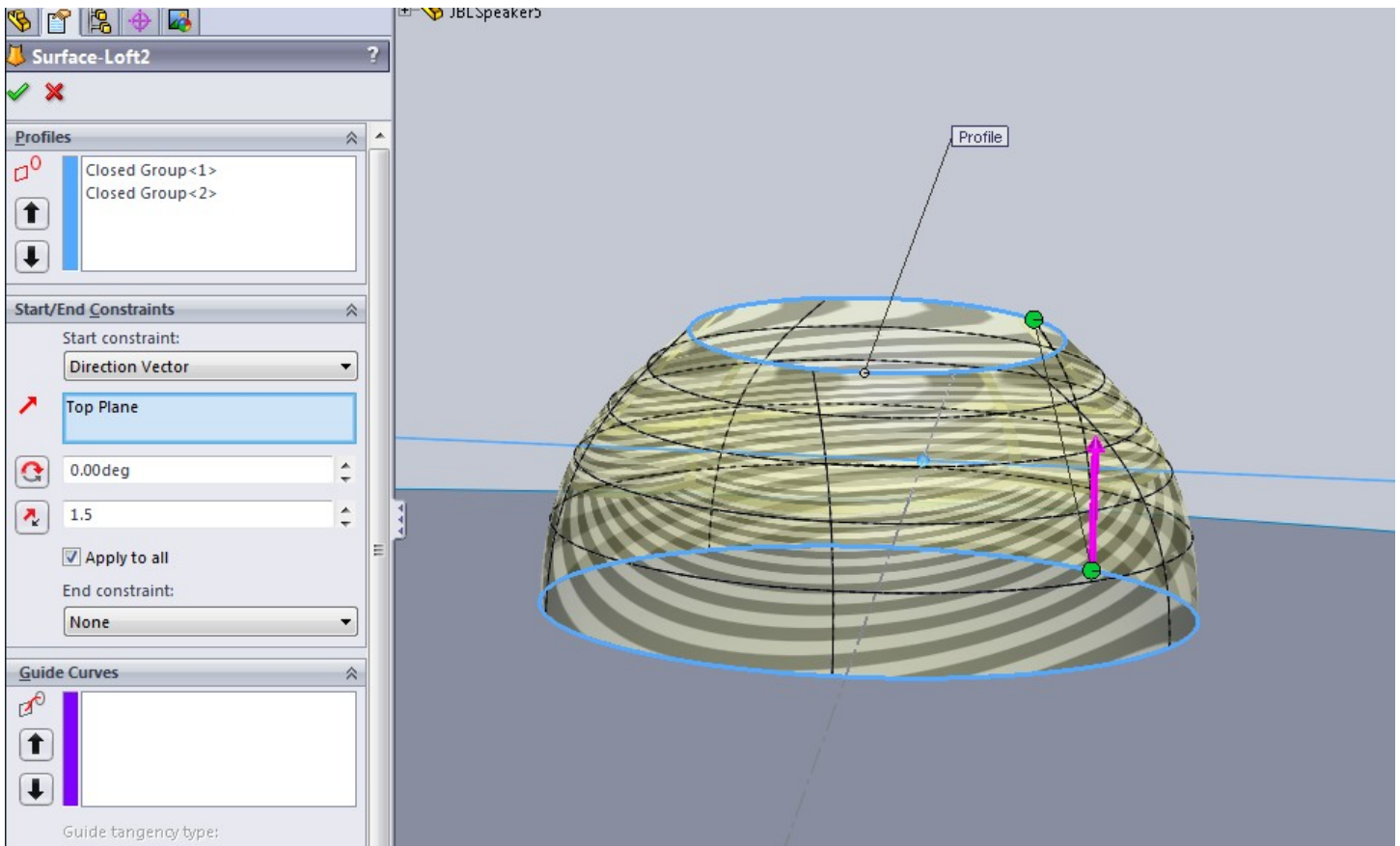
Step23. Finishing the Bottom of the Speaker

Start a new sketch on Plane-6. Offset the four 14mm circles 3mm inside.



Step23. Finishing the Bottom of the Speaker

Create four different Lofted Surfaces between the 14mm circles and the 3mm offset circles. Add a direction Vector constraint to the 14mm circle with the Top Plane. Change the Tangency Length to 1.5. This will change the shape of our loft from a cone to a rounded surface because that tangency make the loft follow a certain direction for a certain length. The 1.5 doesn't mean 1.5mm. It's a unit-less value.

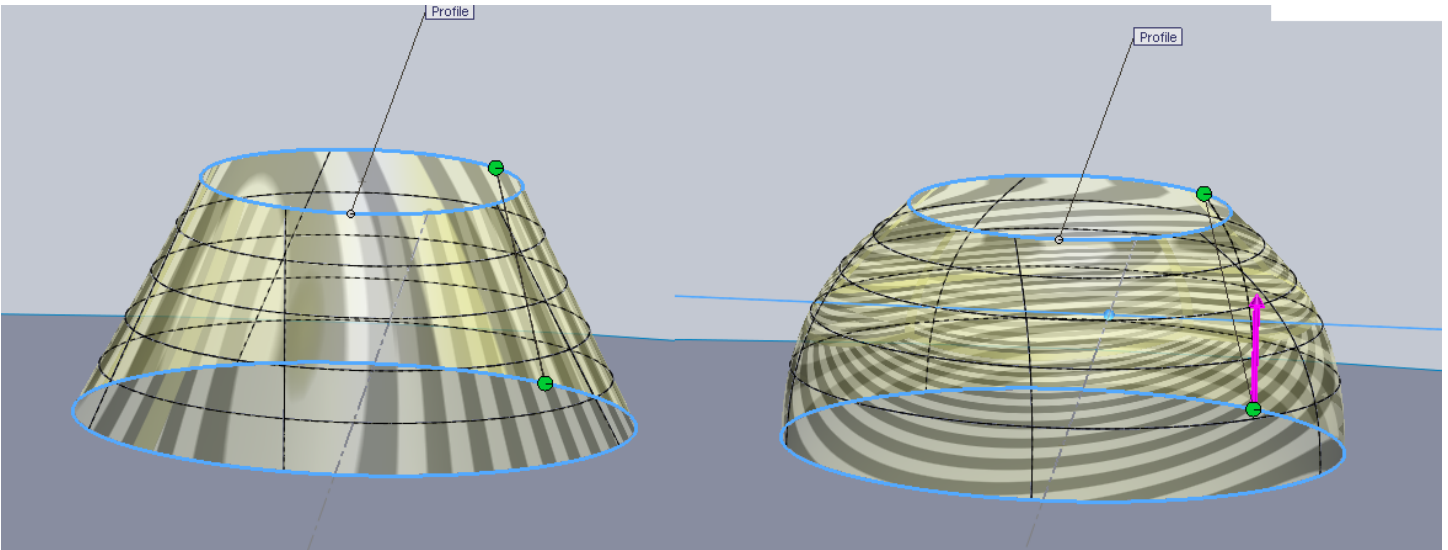


Step23. Finishing the Bottom of the Speaker

Here is a side by side shot of the loft with and without the tangency added.

No Tangency

Tangency

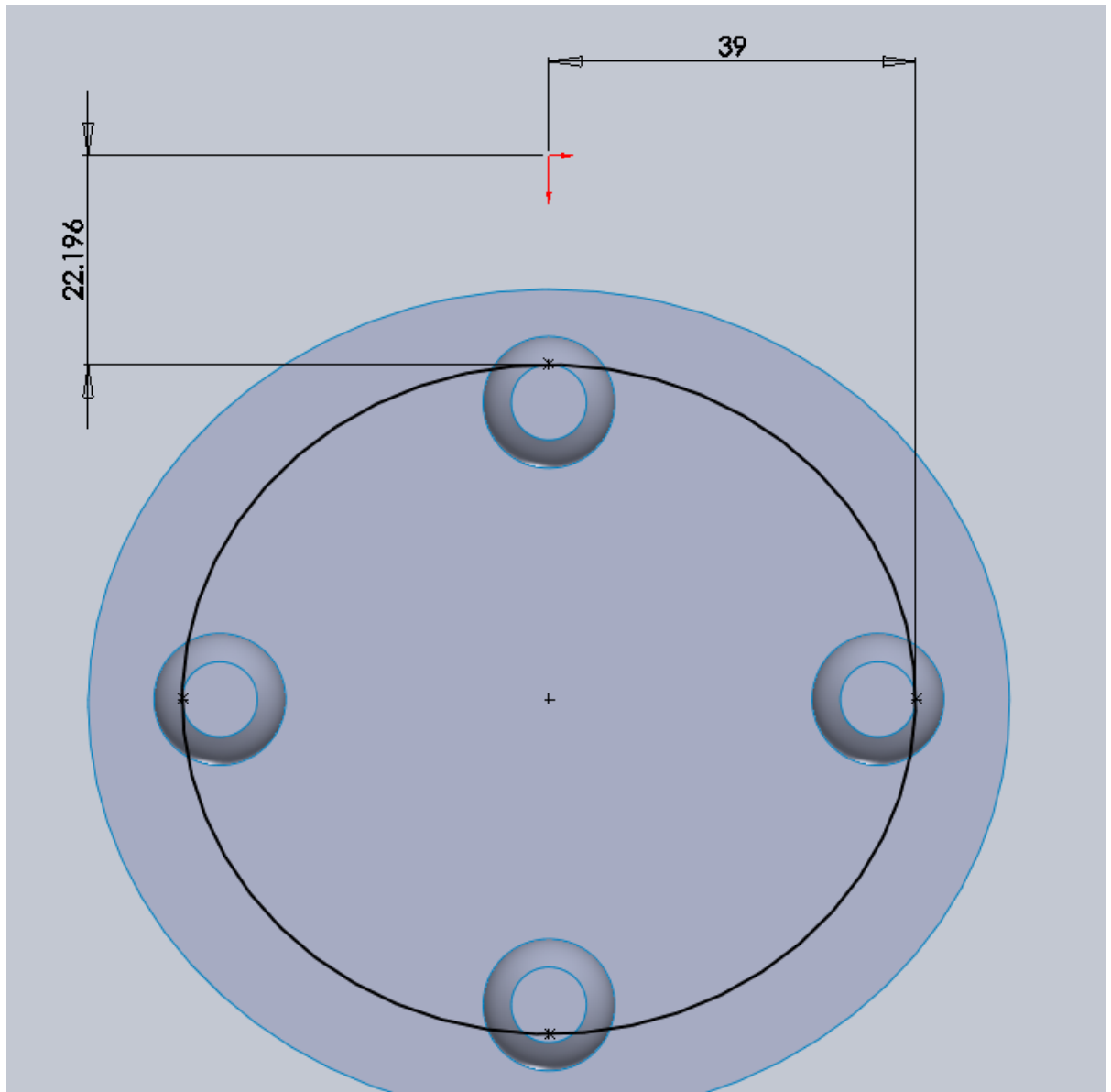


Using the tangency to control the shape allows us to create some nice shapes without the use of guide rails.

Step23. Finishing the Bottom of the Speaker

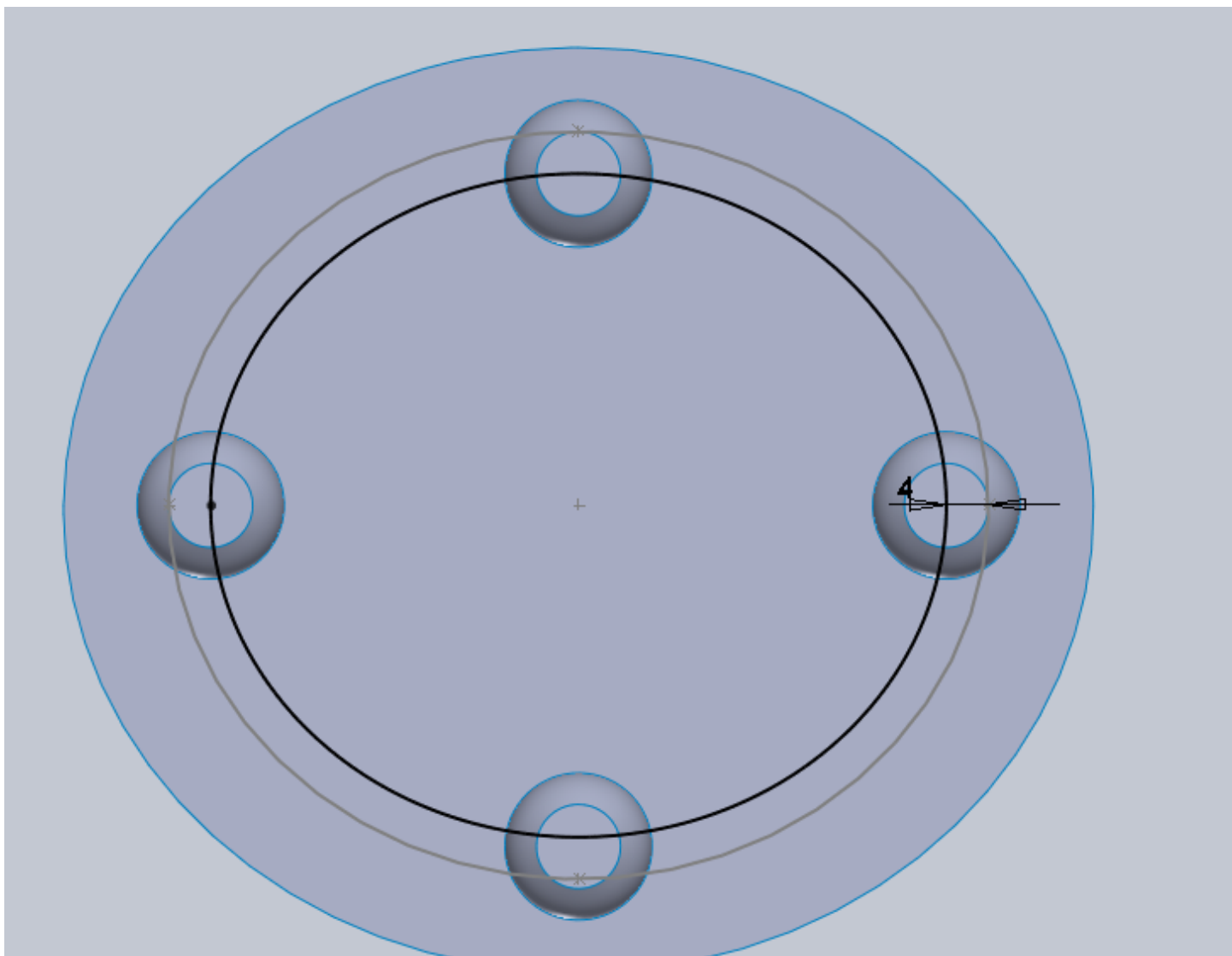
Once you create the first loft, the sketches will automatically hide. You will have to make them visible in order to have the circles available for selection for your other three lofts.

Start a new sketch on the Top Plane. Draw an ellipse as shown.



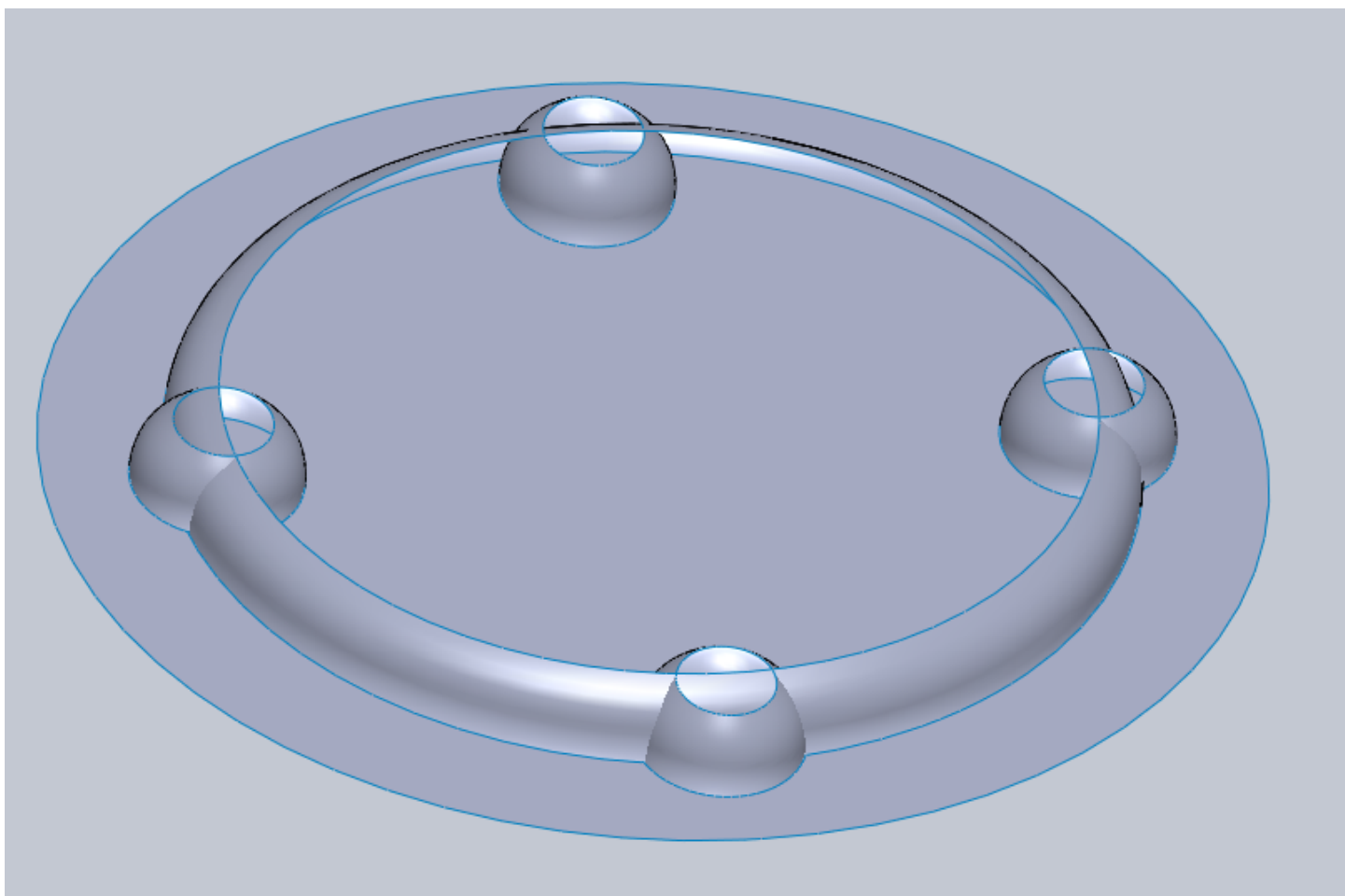
Step23. Finishing the Bottom of the Speaker

Start a new sketch on Plane-6. Offset the ellipse 4mm to the inside.



Step23. Finishing the Bottom of the Speaker

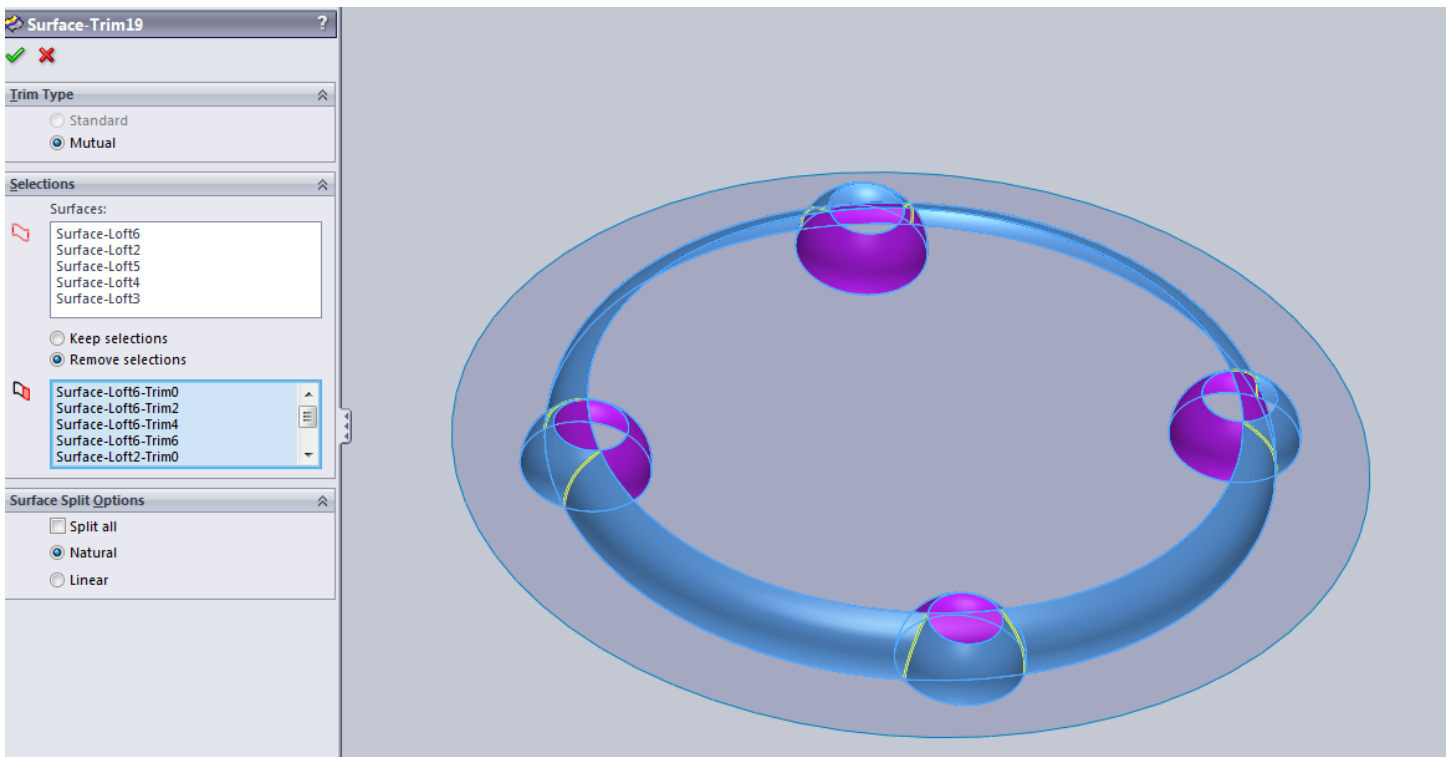
Create a Lofted Surface between the ellipses. Add a Direction Tangency for the larger ellipse. Use the Top Plane as the references and make the Tangency Length 1.5.



Step23. Finishing the Bottom of the Speaker

Start a trim operation and select “mutual” for the trim type. Select all 5 lofted surfaces for the Trimming Surfaces selections.

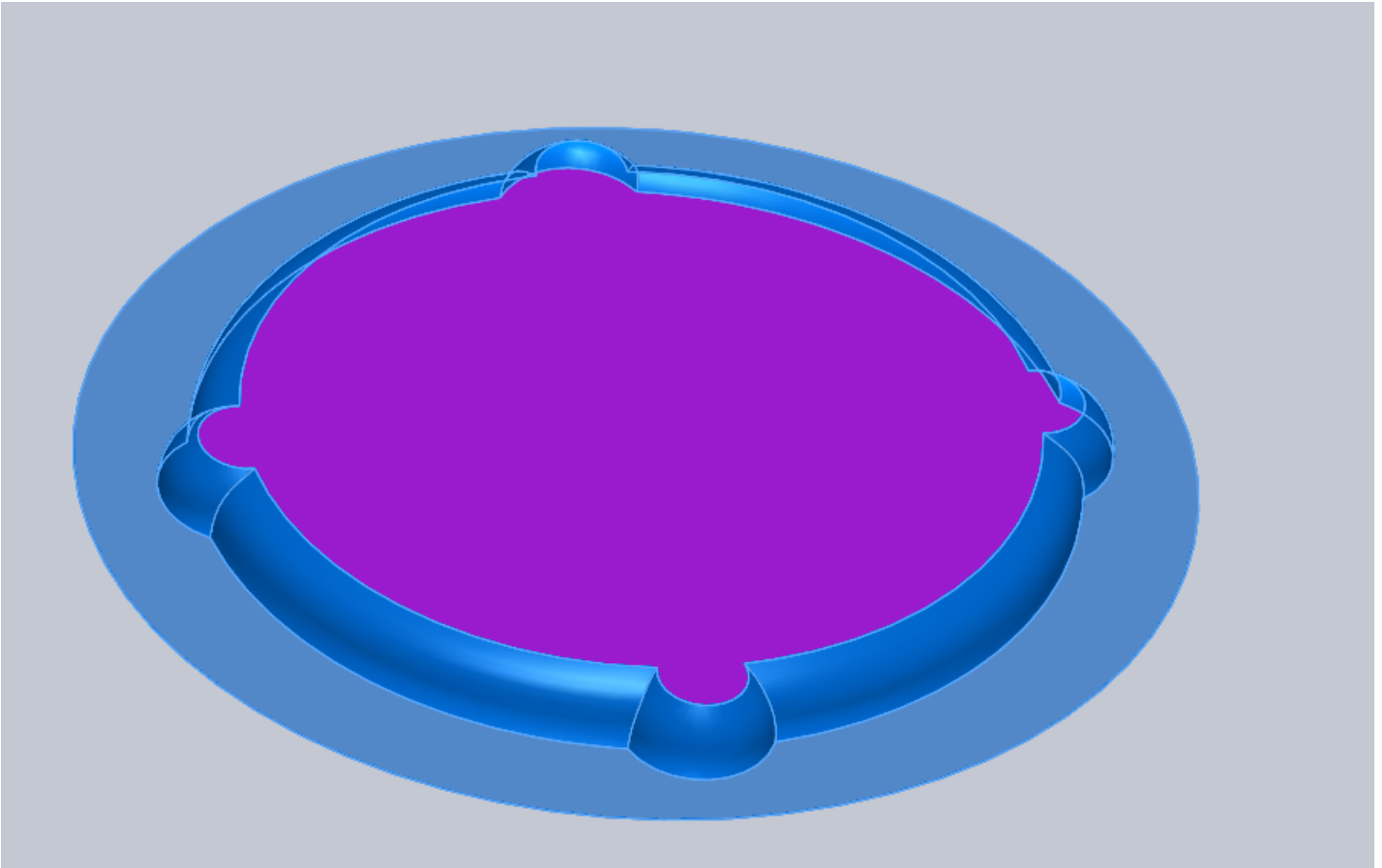
Remove the purple sections shown below.



This is the first time we have used Mutual Trim and I mentioned earlier that it knits the sections together. This will now be apparent when we use this new surface as a trim surface.

Step23. Finishing the Bottom of the Speaker

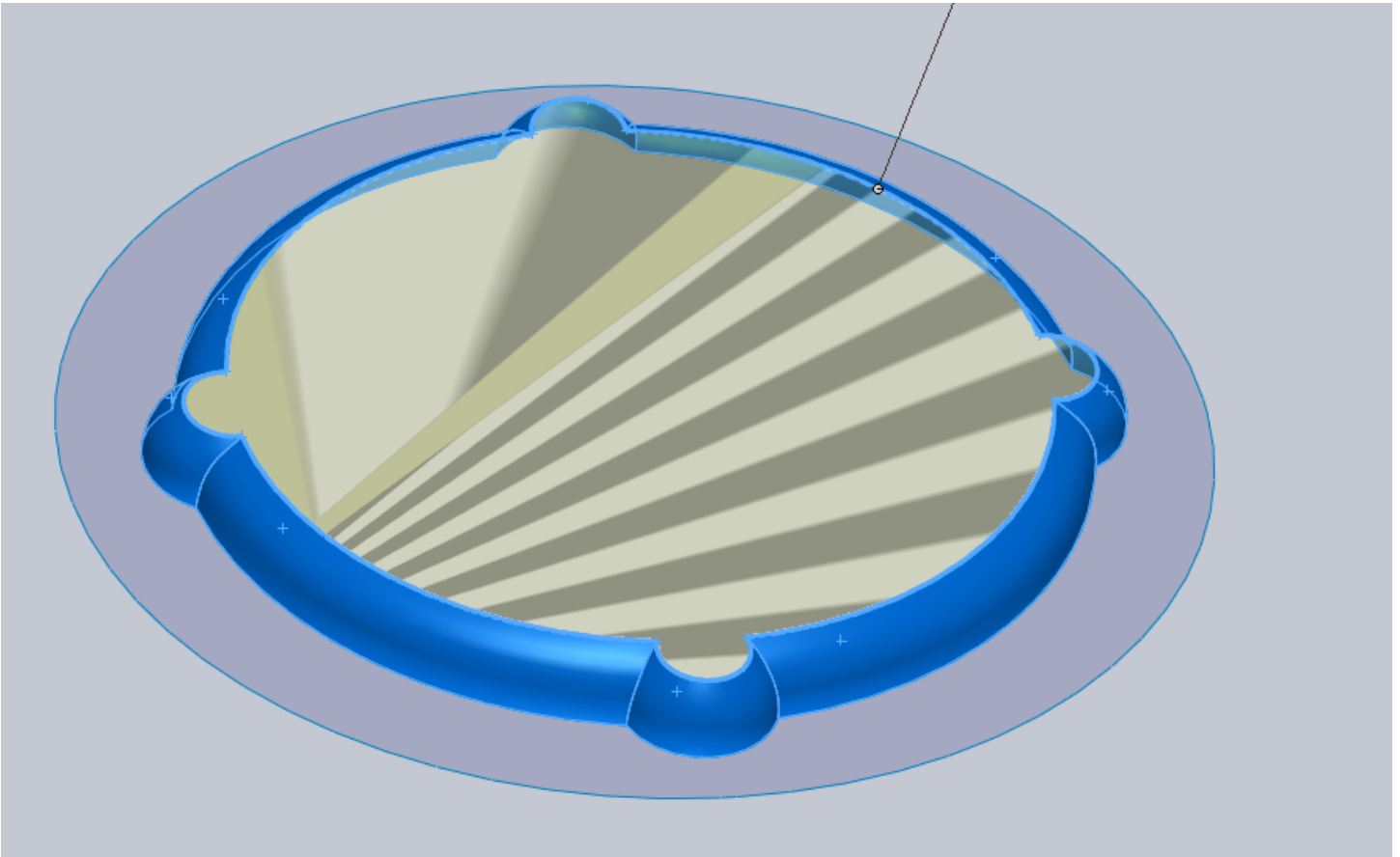
Use this new surface as a Standard Trim Tool (not mutual) and remove the purple section shown below.



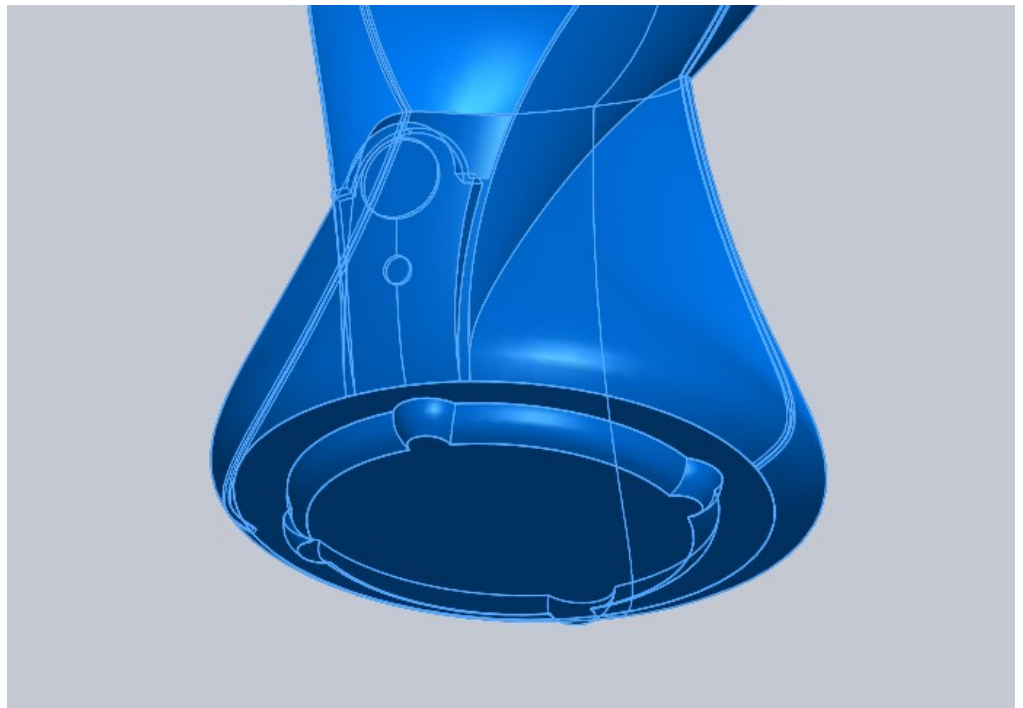
There is no good reason to not use Mutual Trim here as well, but I may want to work with this surface a little more so I made the call to use Standard Trim here.

Step23. Finishing the Bottom of the Speaker

Create a Filled Surface to fill in the bottom as shown.

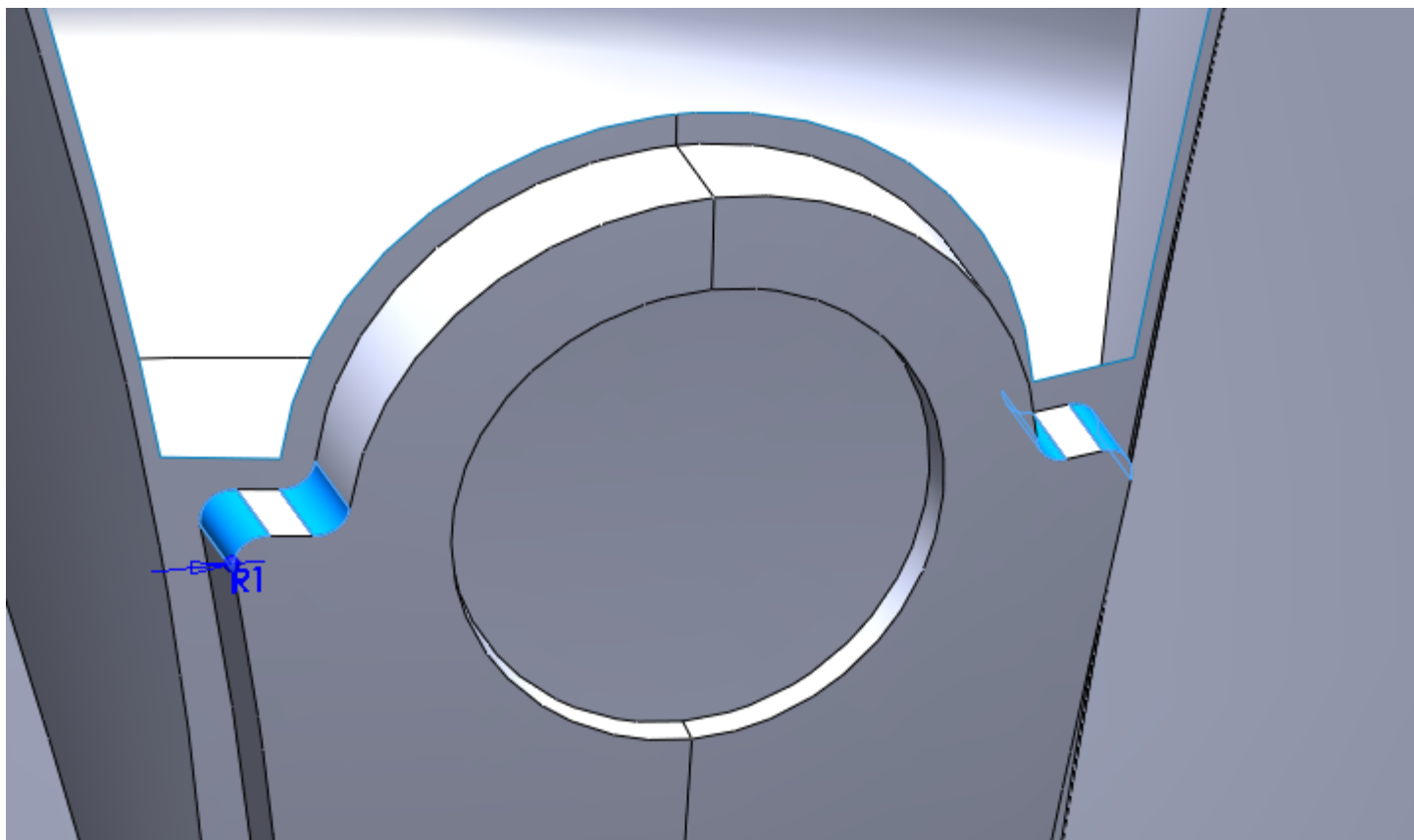


Knit these 3 surfaces to the main body of the speaker.



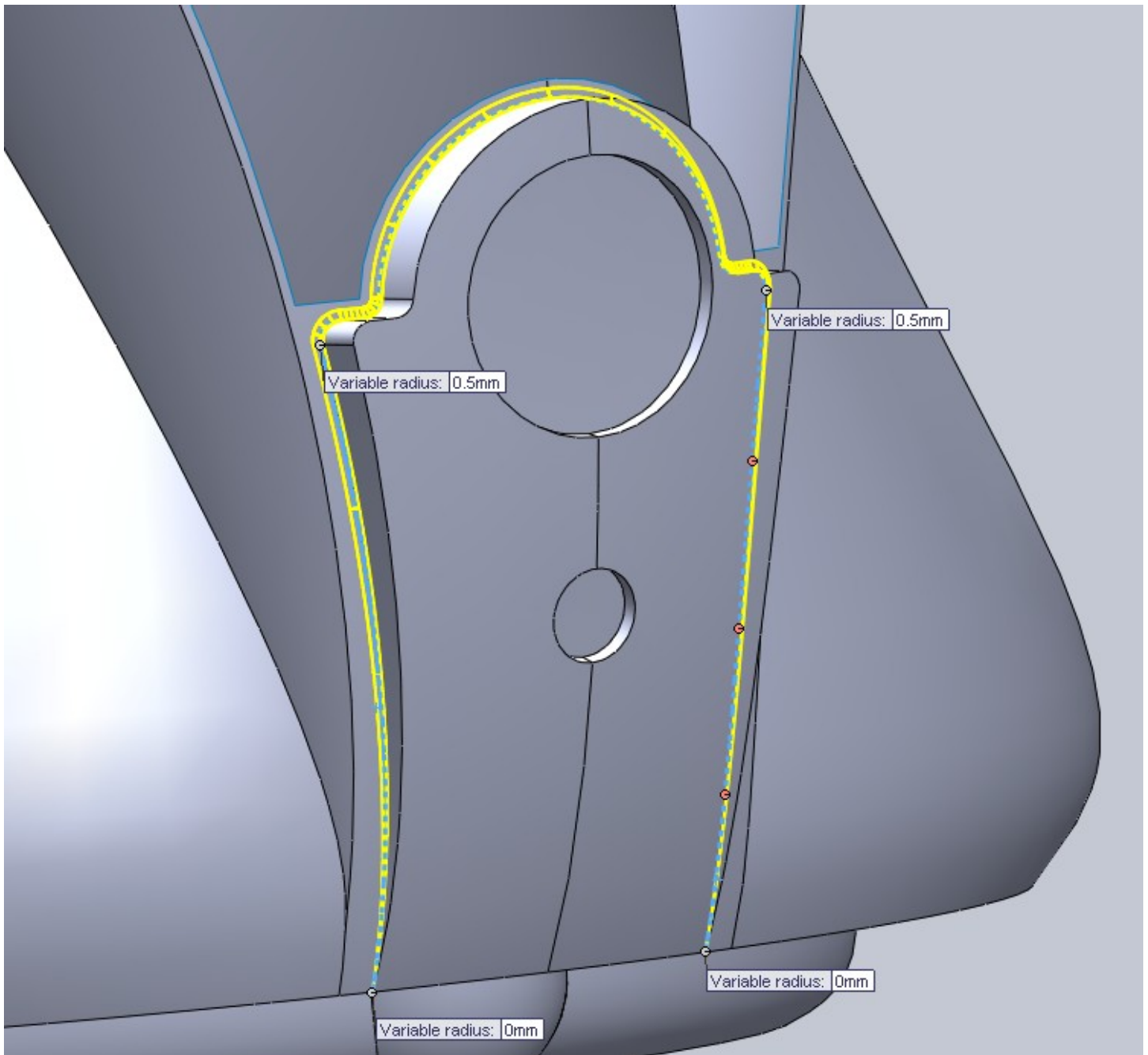
Step24. Some Finishing Touches

Start with a 1mm fillet to the 4 edges shown.



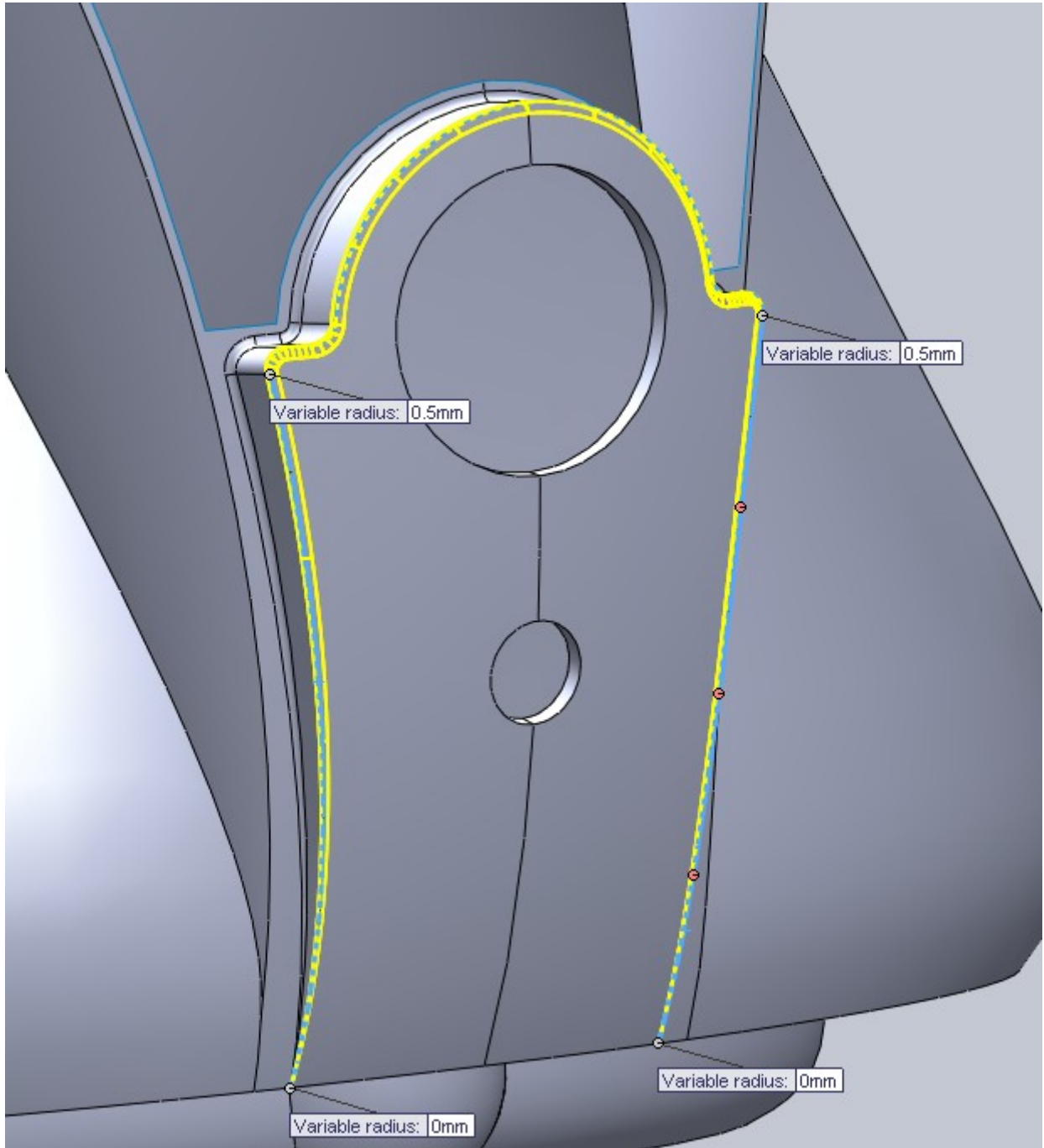
Step24. Some Finishing Touches

Add a variable fillet to the edges shown. It goes from 0mm to 0.5mm and back to zero. The reason we do this is because the curve starts and ends in a place where you can not fillet. A conventional fillet would fail.



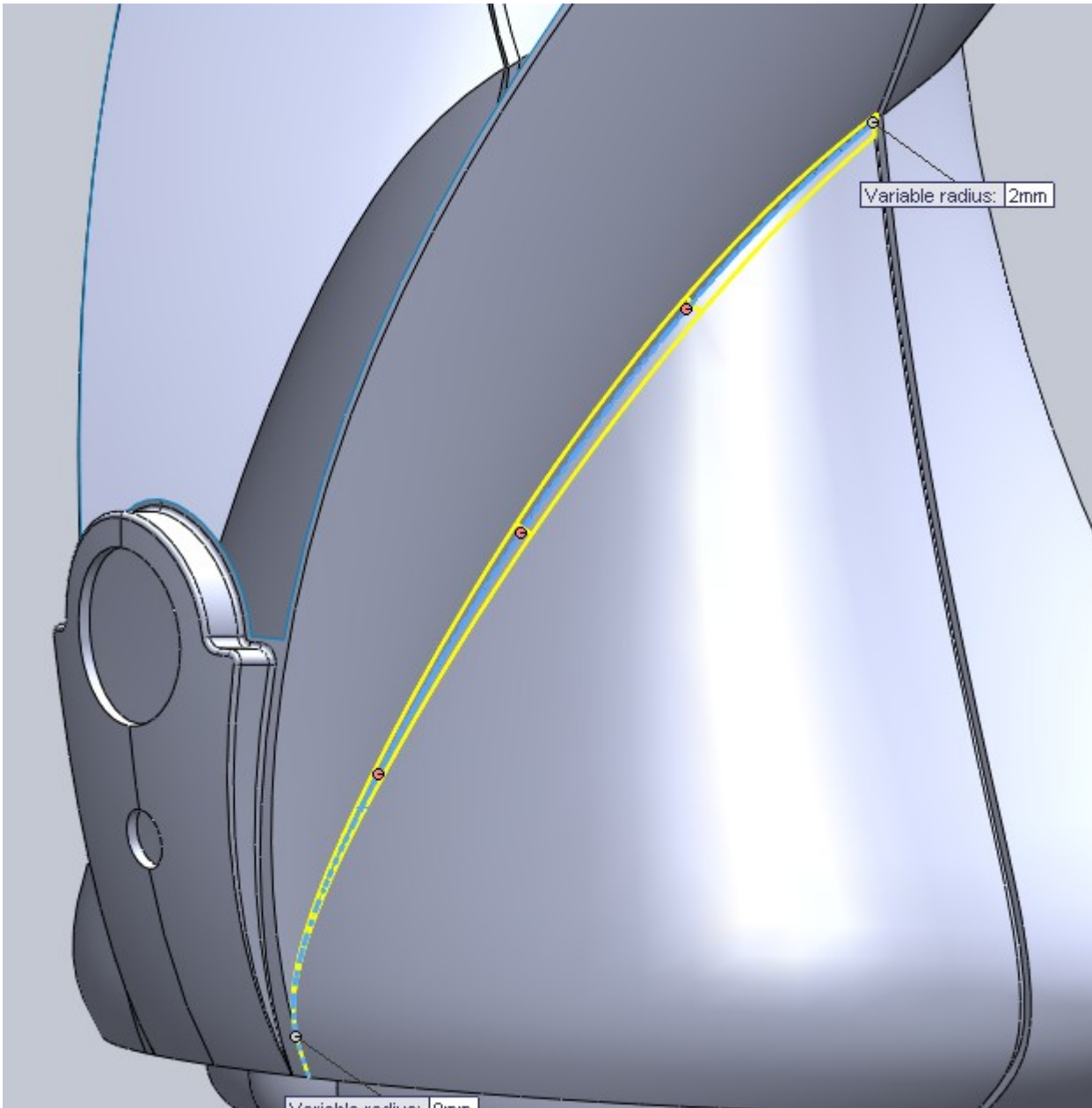
Step24. Some Finishing Touches

Add a variable fillet to the front edge using the same numbers.



Step24. Some Finishing Touches

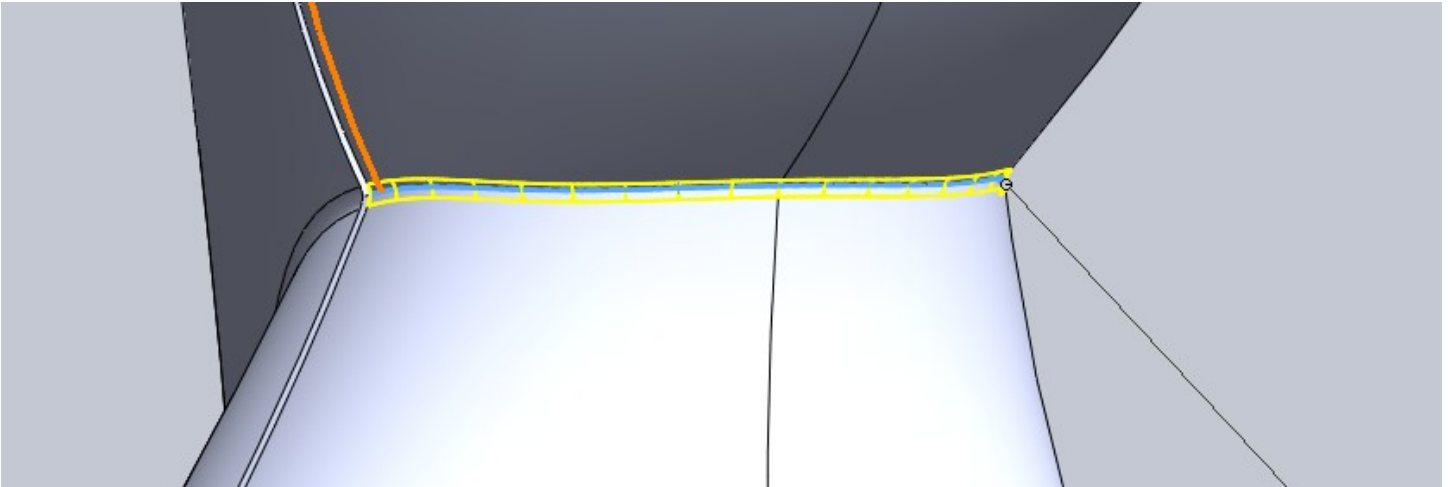
Add a variable fillet to the edge shown going from 0mm to 2mm. Note that there is a small section of this edge near the bottom that was not selected.



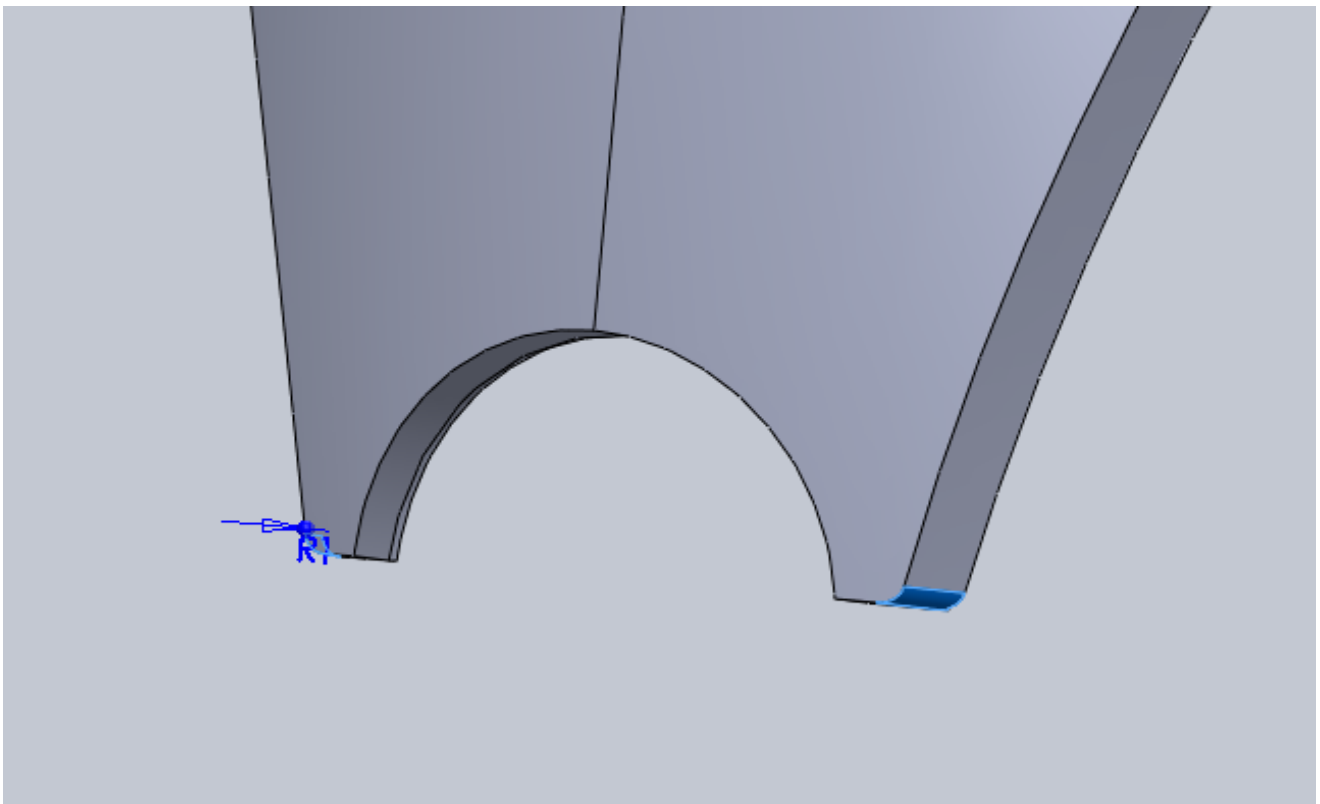
Repeat for the edge on the other side (its mirror).

Step24. Some Finishing Touches

Add a 2mm constant radius fillet to the edge shown around the back of the speaker body.

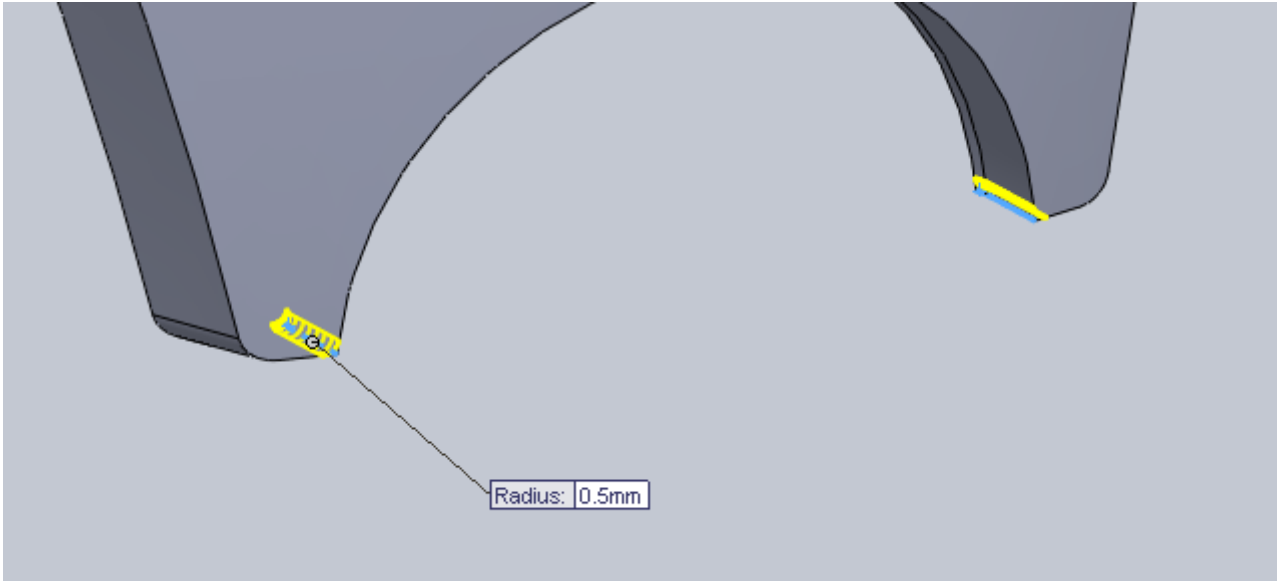


Add a 1mm constant radius fillet to the two edges shown below.

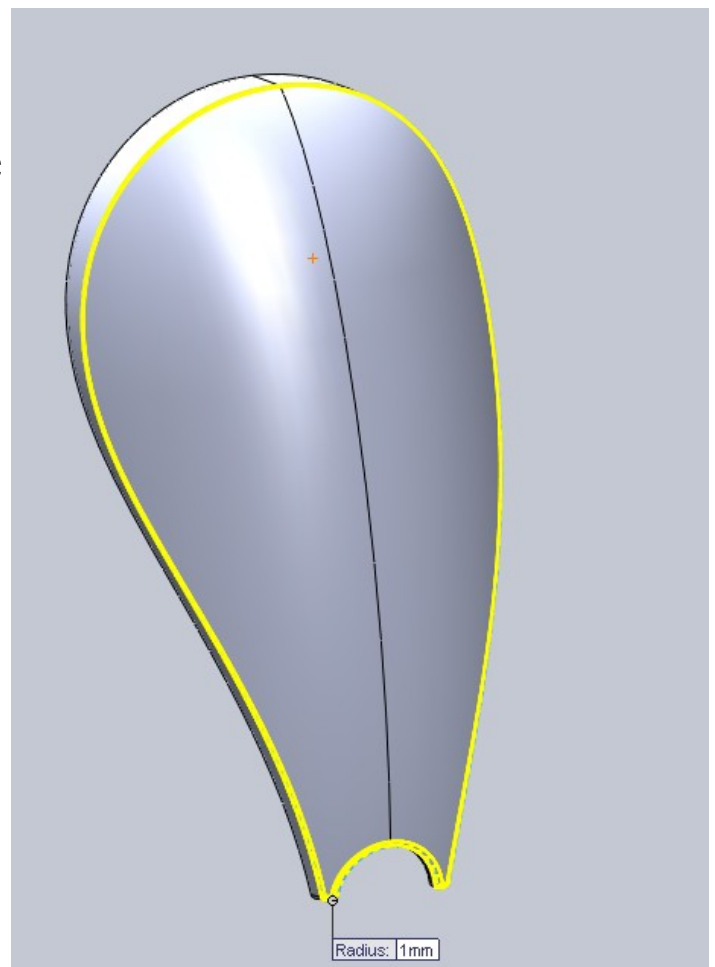


Step24. Some Finishing Touches

Add a 0.5mm constant radius fillet to the edges shown below.

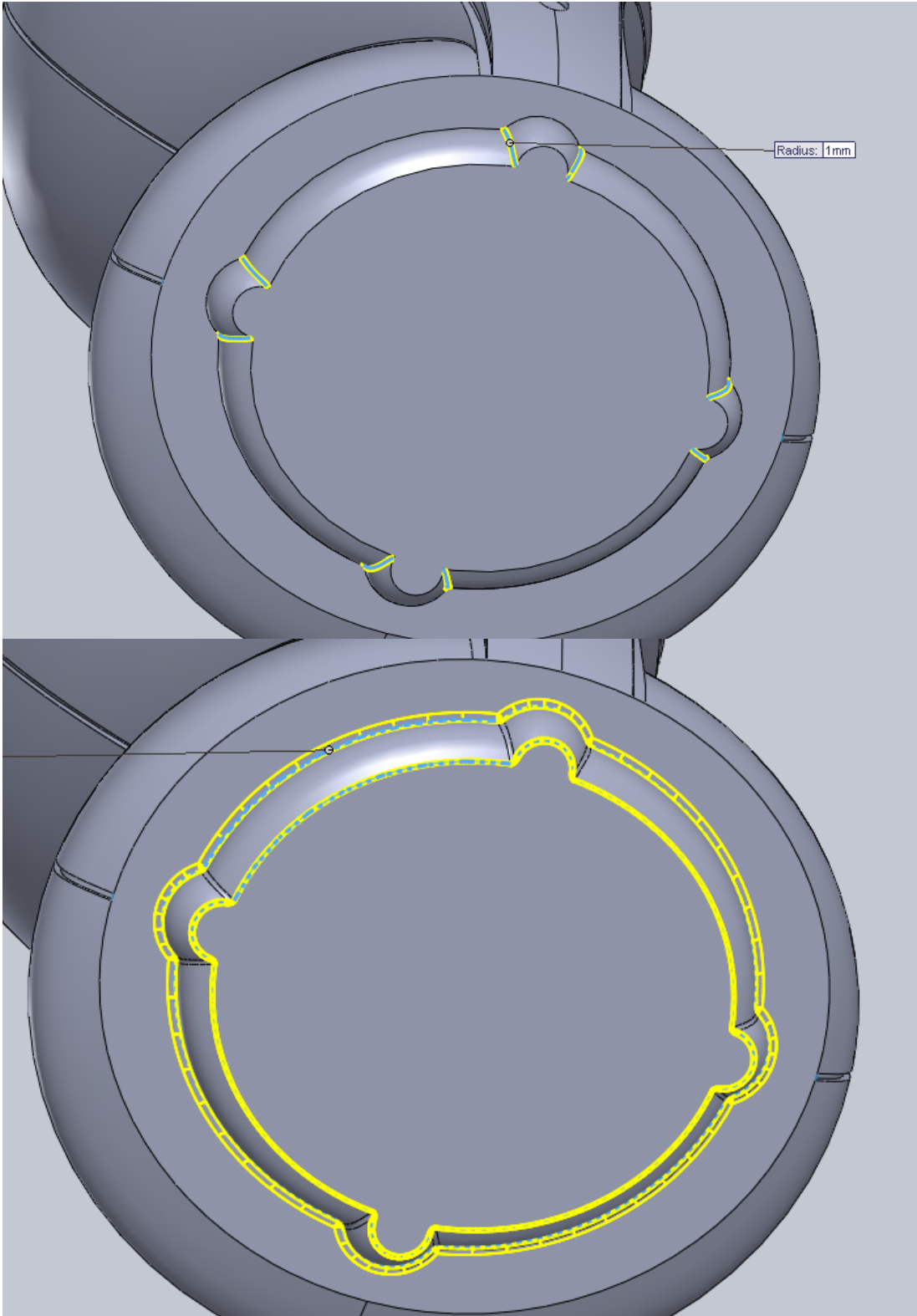


Add a 1mm constant radius fillet to the entire outer edge of the speaker face.



Step24. Some Finishing Touches

Add a 1mm constant radius fillet to the edges shown below in two separate operations.



Step25. Rendering

If you want to add other things like a JBL logo, LED or power button you should be able to tackle those things. If you just want to render what you have you can apply a Cotton Cloth material to the speaker face, and a high gloss plastic to the rest of the body and your done!

