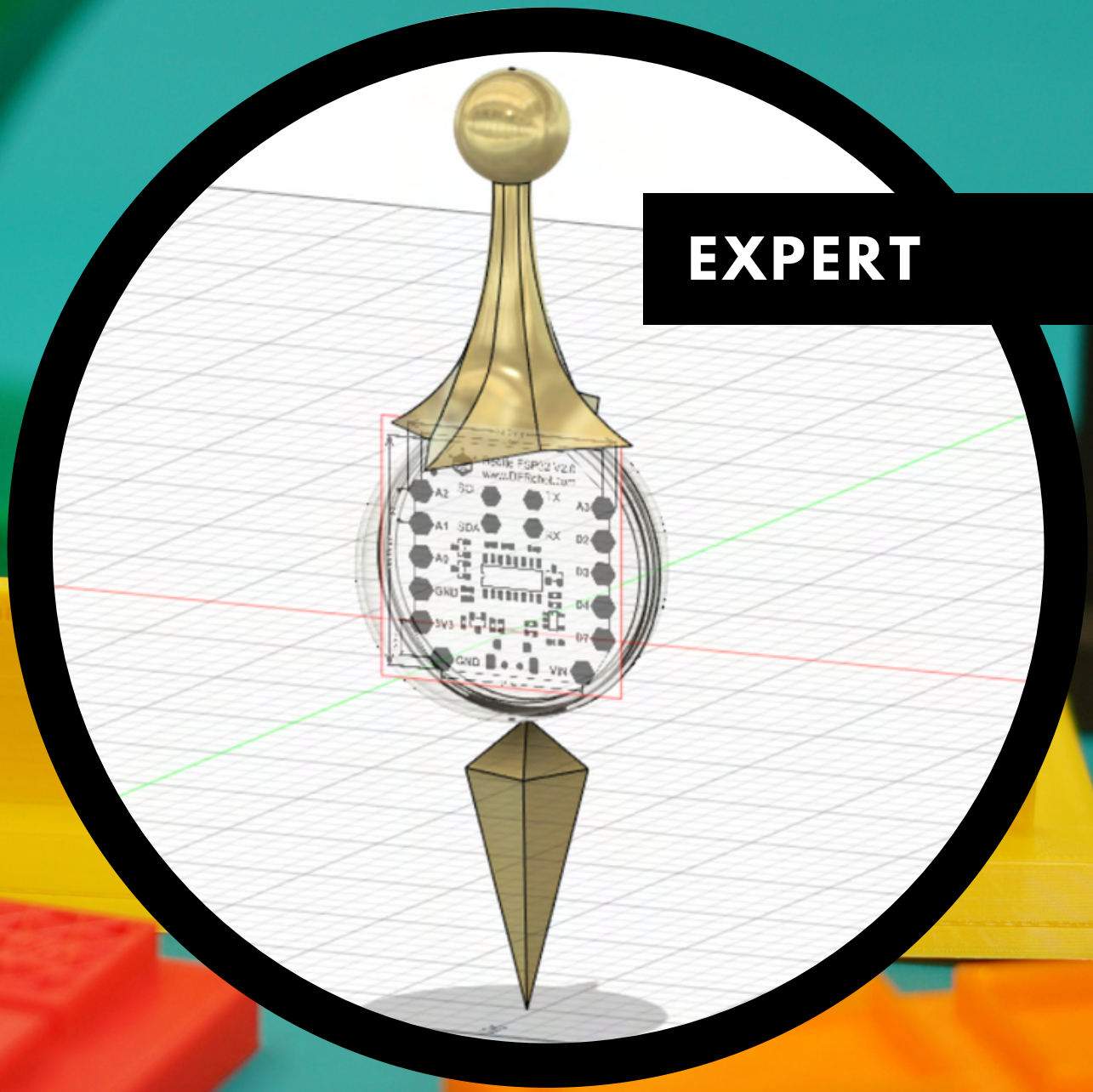
MAKER GUIDE
LIGHT UP JEWELLERY

CITY OF CANADA BAY LIBRARIES
THE LEARNING SPACE

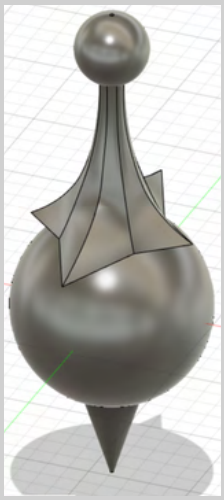


EXPERT

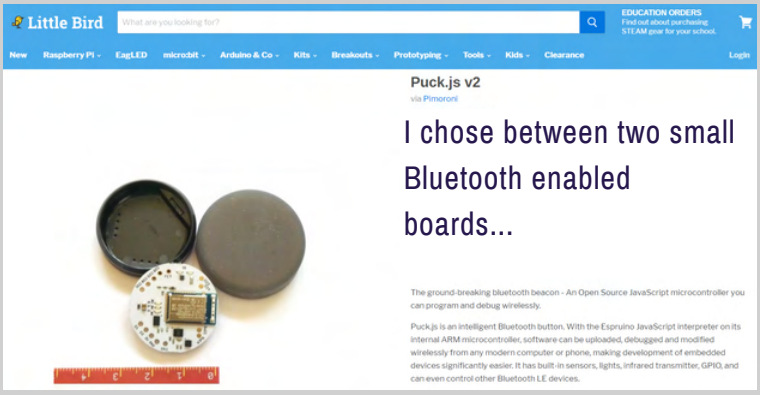


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LIGHT UP JEWELLERY



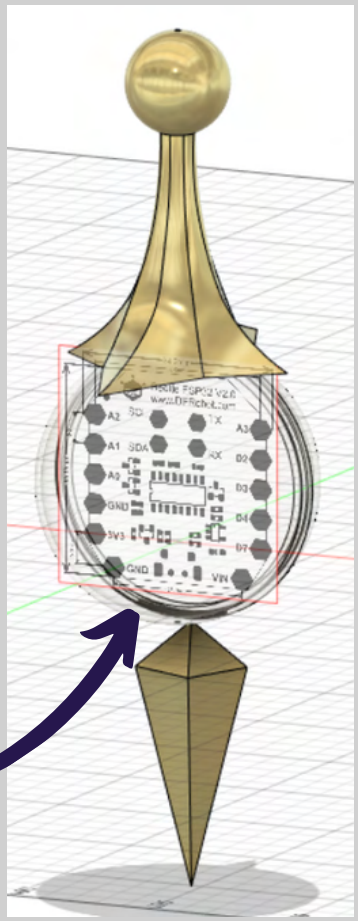
In this exercise we'll be making a piece of jewellery that lights up! I'll be using a Bluetooth board so I can coordinate my jewellery lights to change colour at the same time.

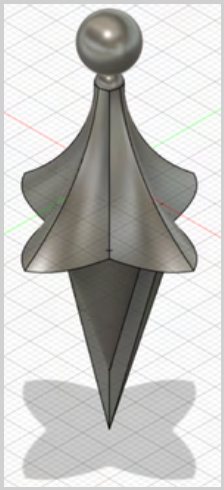


I chose between two small Bluetooth enabled boards...

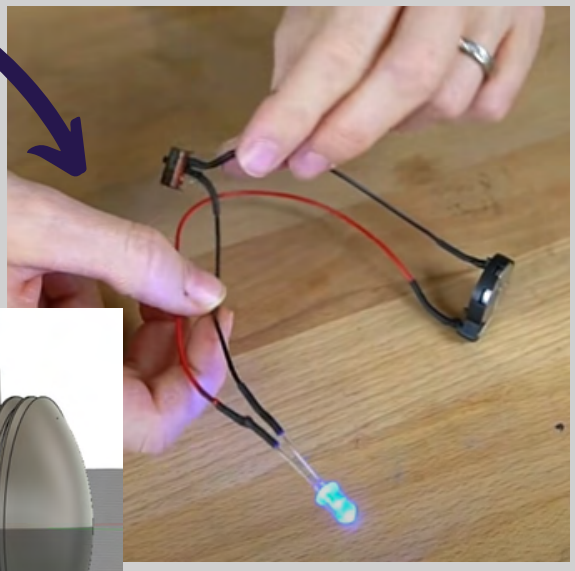


I went with the Beetle for this project and downloaded the measurement image for reference



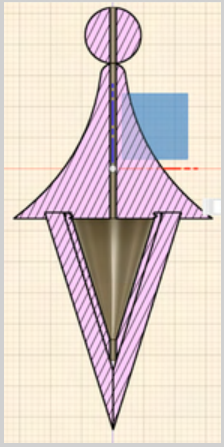


However this isn't necessary, if you would prefer you can just use a basic circuit like this one.

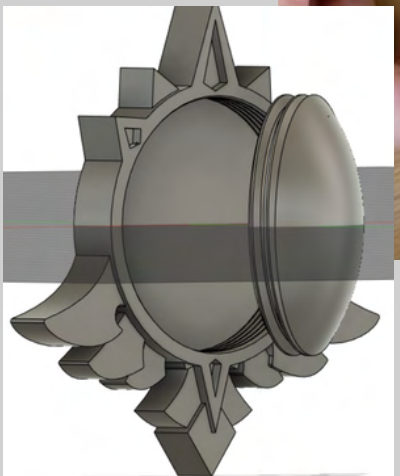


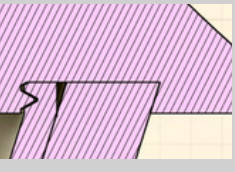
Getting started with LEDs
KamuiCosplay

Another thing to keep in mind while creating this piece of jewellery is that there's a ton of room for creating your own unique design. I've included some other example designs on this page if you decide you want to take your jewellery further.



Finally, how you attach the empty electronics case to the rest of the 3D print will depend on how your printing (FDM/SLA) and what materials you're using to print it (PLA/ABS/PETG/etc.) Later in the tutorial we discuss various methods for connecting the pieces.



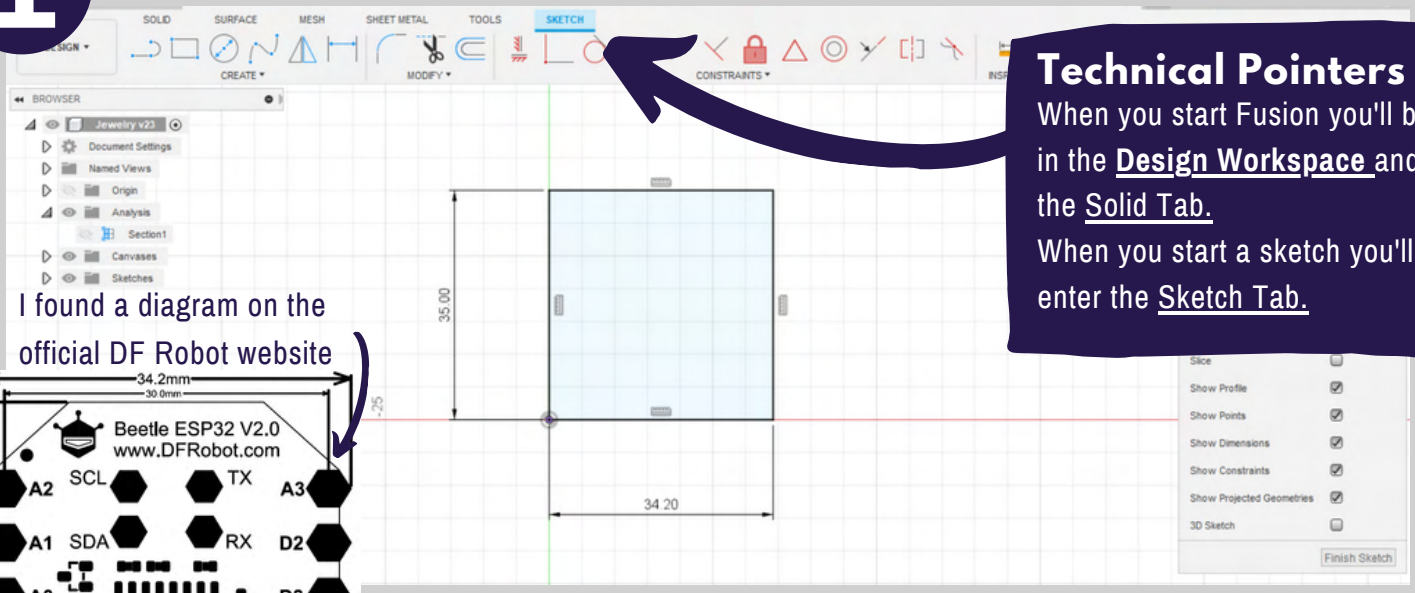




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MAKE IT TO SCALE

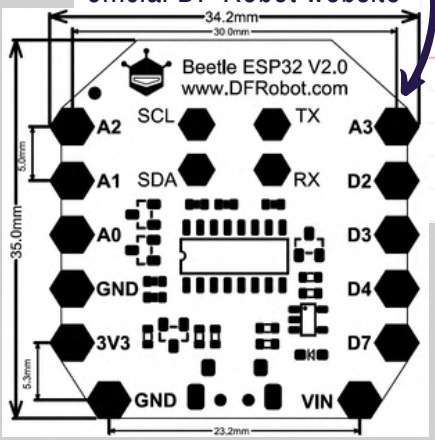
1 To start, I'm going to make myself a scale guide, this is optional but I found it very helpful. First jump into the 'Sketch' menu and make a square the size of your board



Technical Pointers

When you start Fusion you'll be in the **Design Workspace** and the Solid Tab.
When you start a sketch you'll enter the Sketch Tab.

I found a diagram on the official DF Robot website

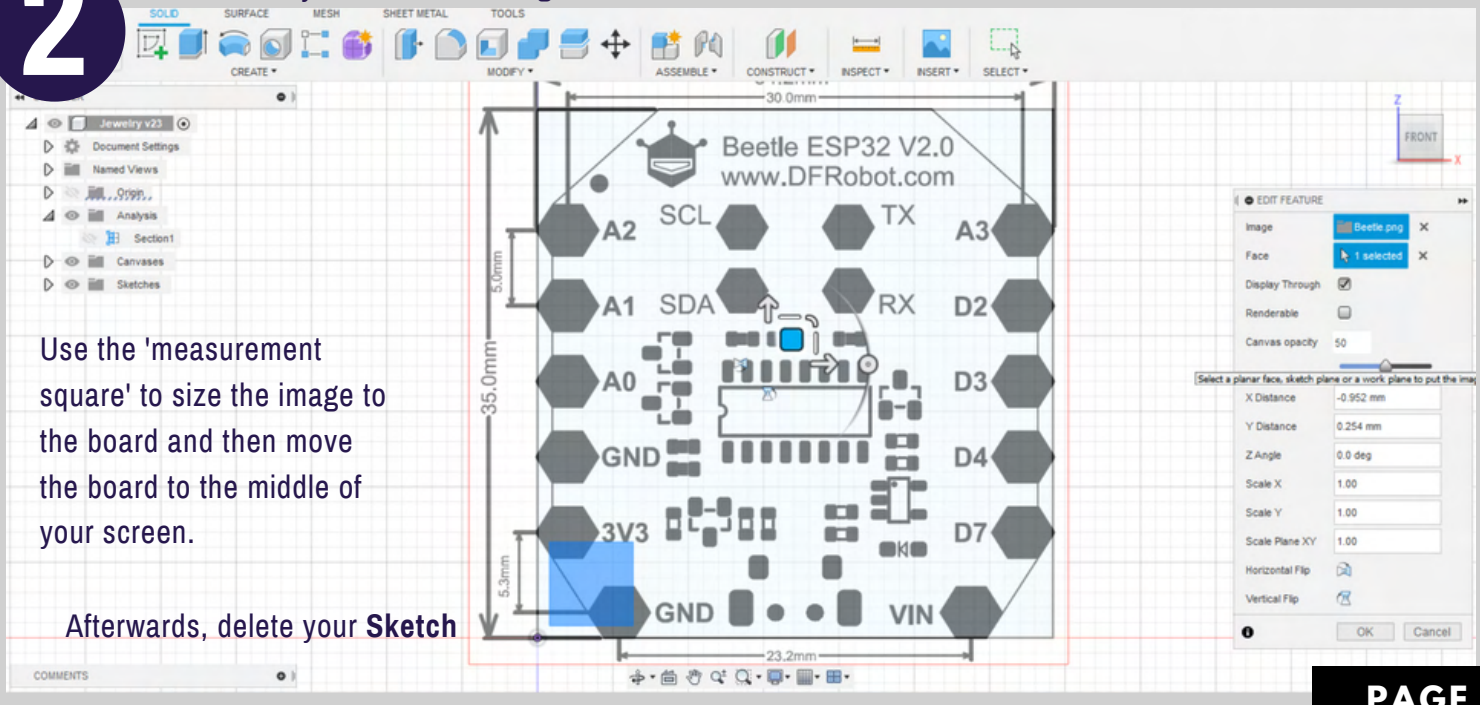


If you just make a basic circuit, then you can use a similar size to this board or measure your own circuit.

Technical Pointers

To zoom in Fusion roll the middle mouse button. To pan hold the middle mouse button

2 Next, insert your reference image as a **Canvas**



Use the 'measurement square' to size the image to the board and then move the board to the middle of your screen.

Afterwards, delete your **Sketch**

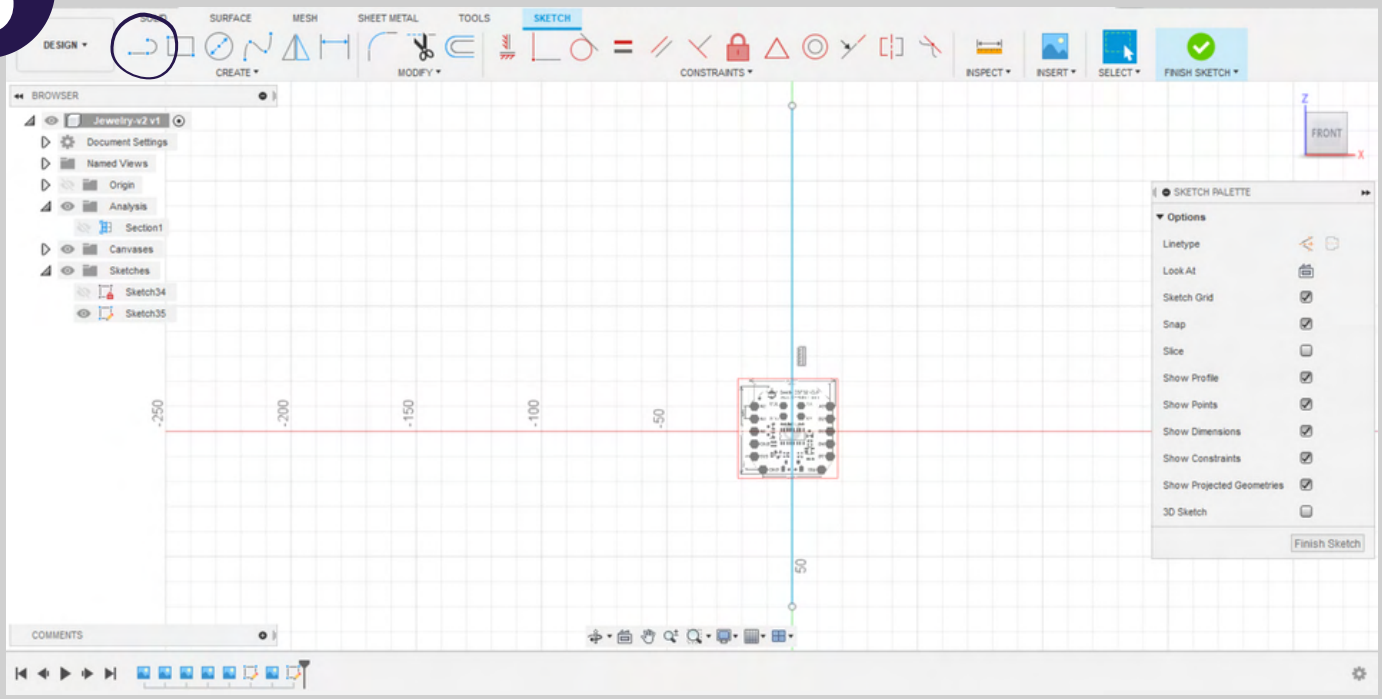


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CREATE THE MAIN BEAD

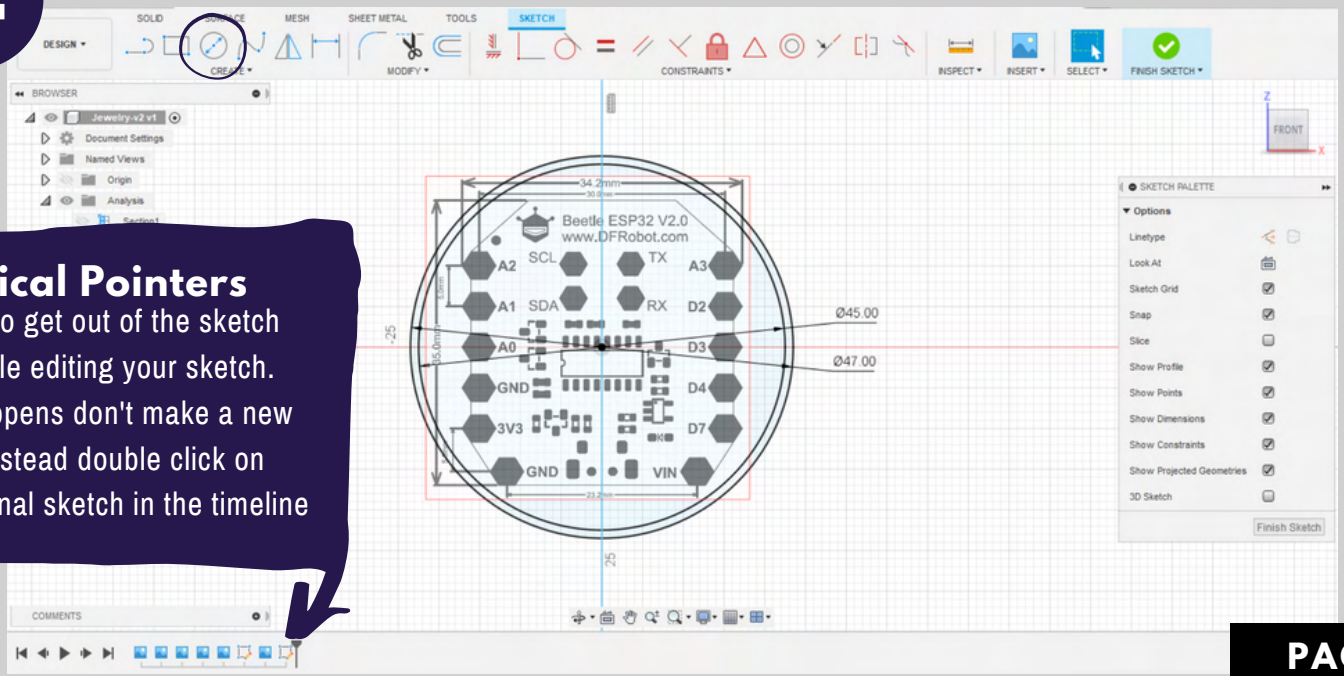
3

Now create a new **Sketch** and draw a **Line** down the center of your board.



4

Next draw two **Circles** one with a diameter of 45 cm and one of 47cm



Technical Pointers

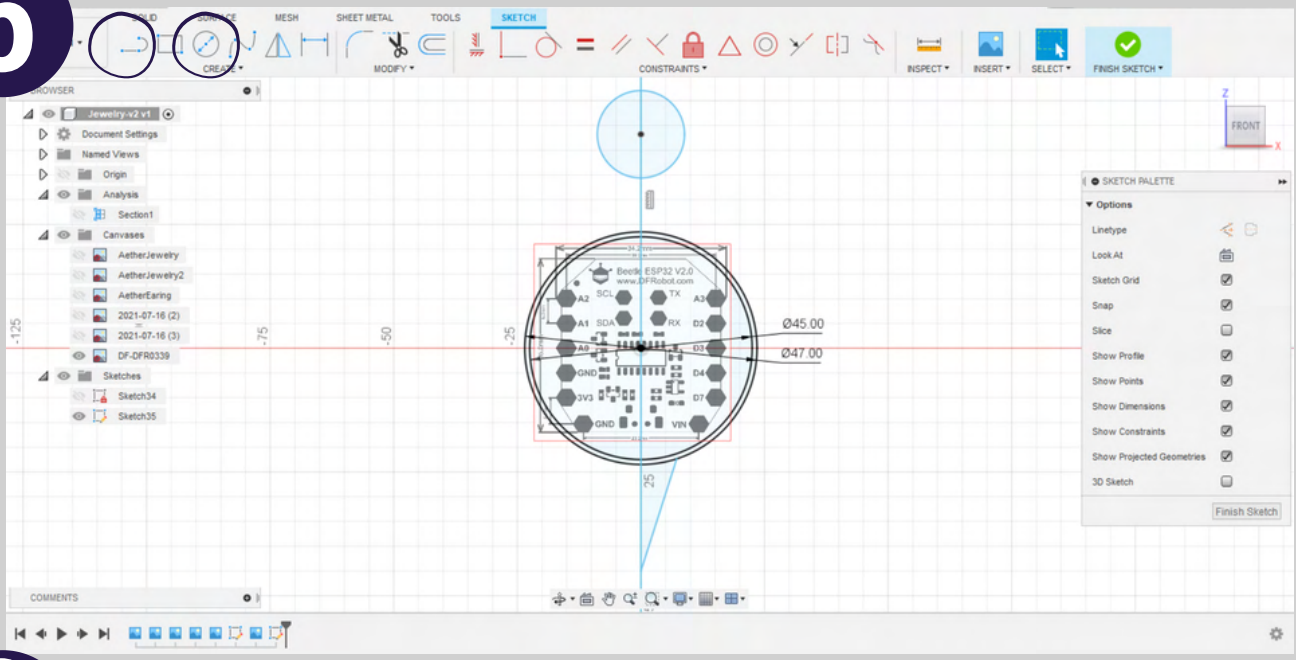
It's easy to get out of the sketch menu while editing your sketch. If this happens don't make a new sketch, instead double click on your original sketch in the timeline



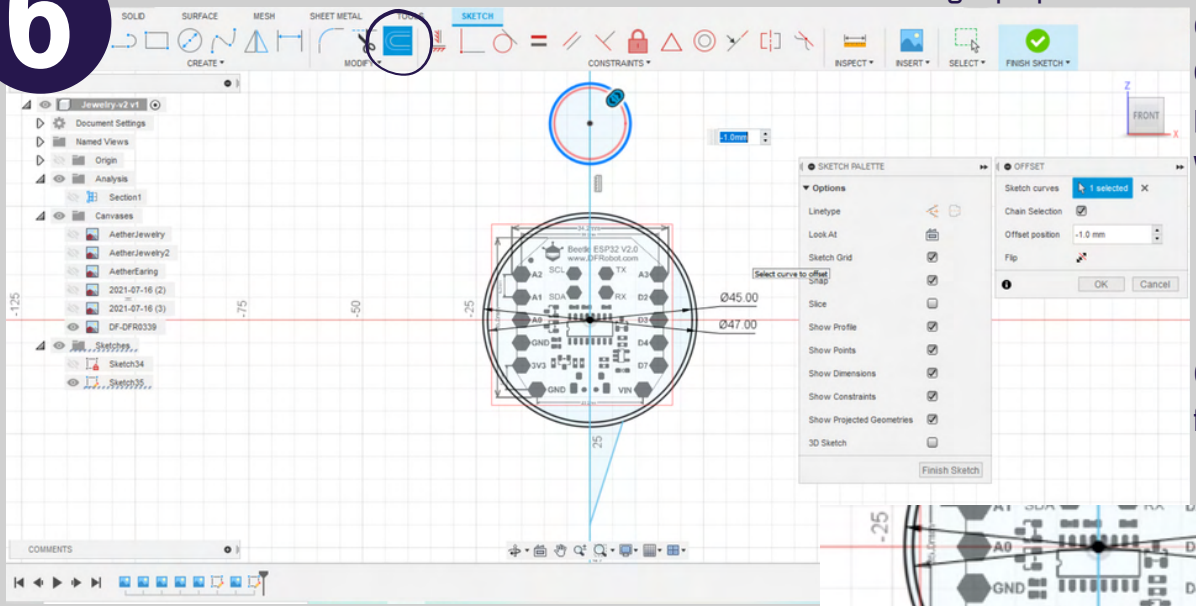
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CREATE THE DECORATIONS

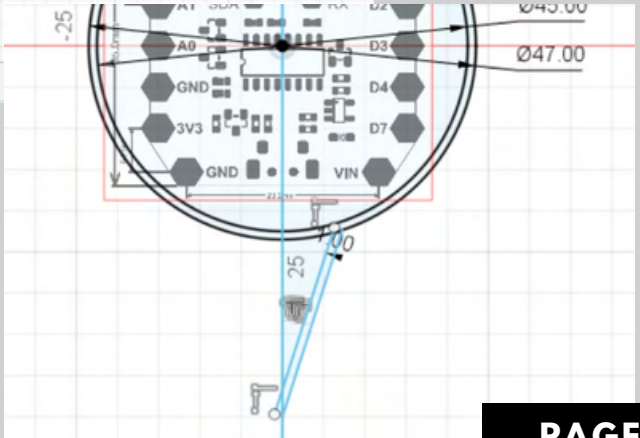
5 Next draw your jewellery accessories - I'm creating a circular bead and a cone shaped bead



6 Now create a **Offset** of 2mm - this is to hollow out the bead for weight purposes but it isn't essential, especially if the circuit in your piece of jewellery will be light.



Create an offset for both beads



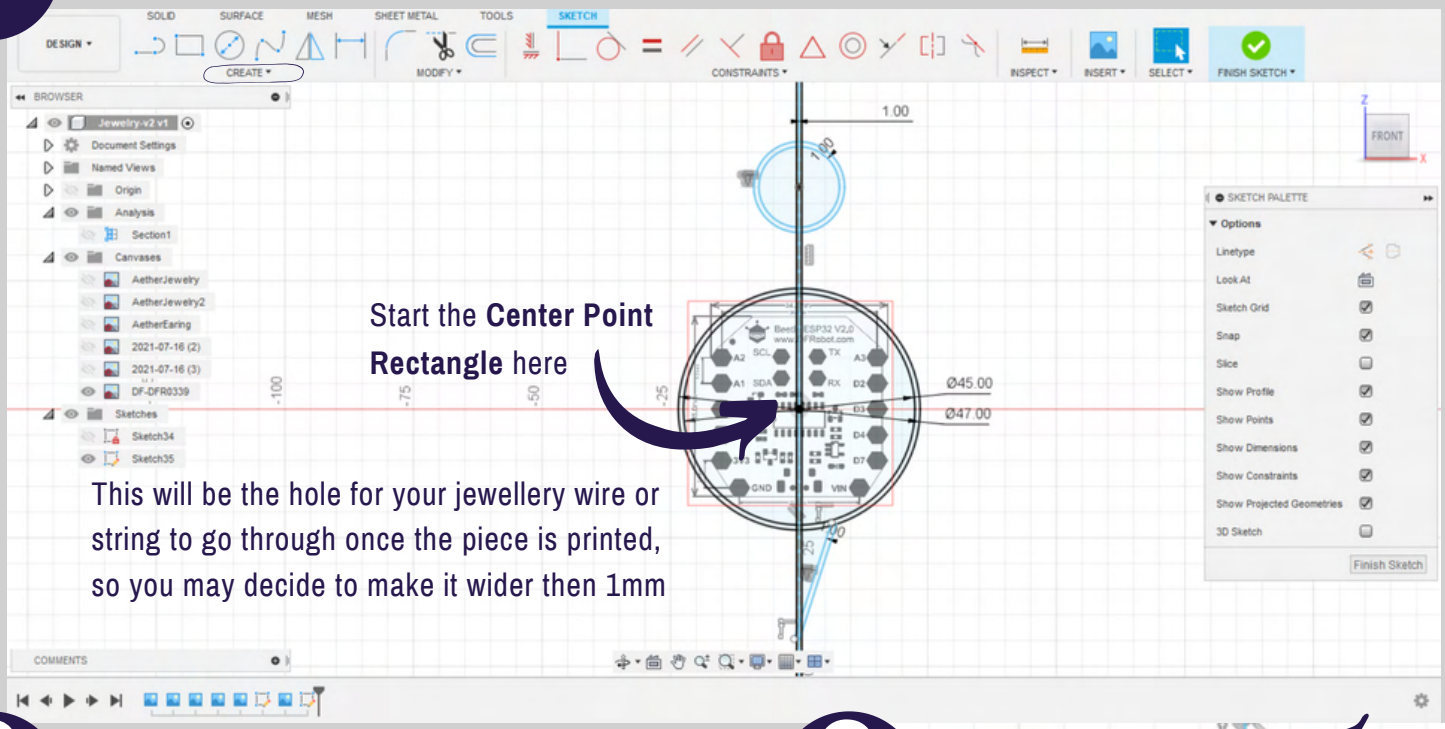


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CLEAN UP YOUR SKETCH

7

Now create a 1mm wide **Center Point Rectangle** (you'll find it in the Create menu under Rectangles) going the length of your piece.



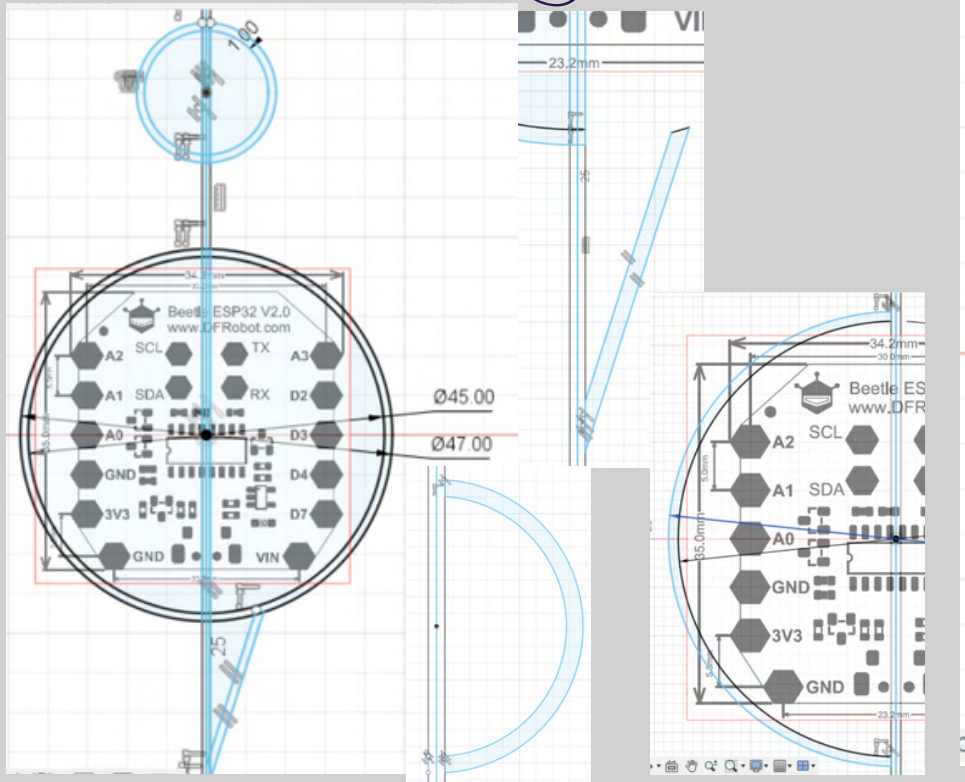
Start the **Center Point Rectangle** here

This will be the hole for your jewellery wire or string to go through once the piece is printed, so you may decide to make it wider then 1mm

8

Next **Trim** off the parts on one side of the sketch



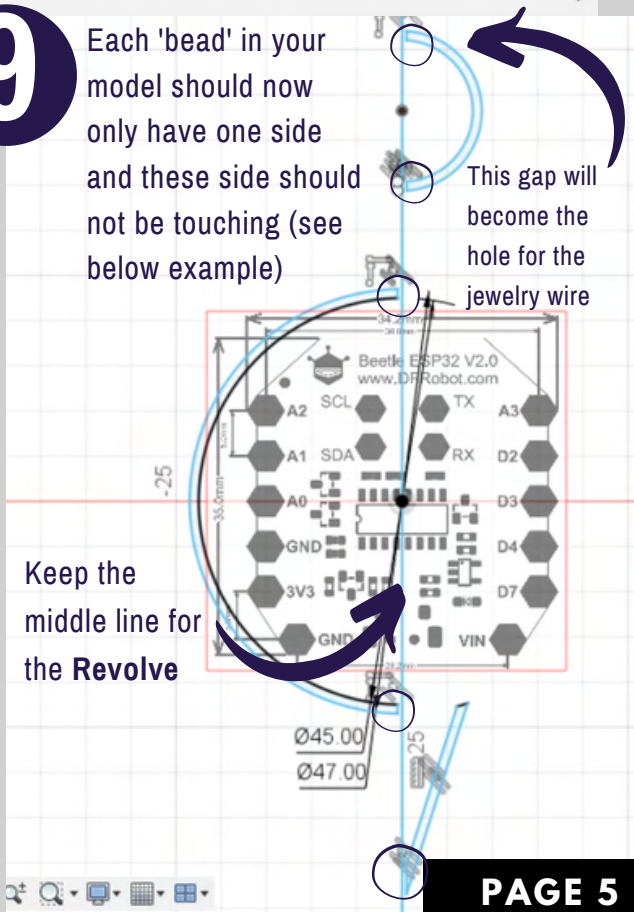


9

Each 'bead' in your model should now only have one side and these side should not be touching (see below example)

This gap will become the hole for the jewelry wire

Keep the middle line for the **Revolve**

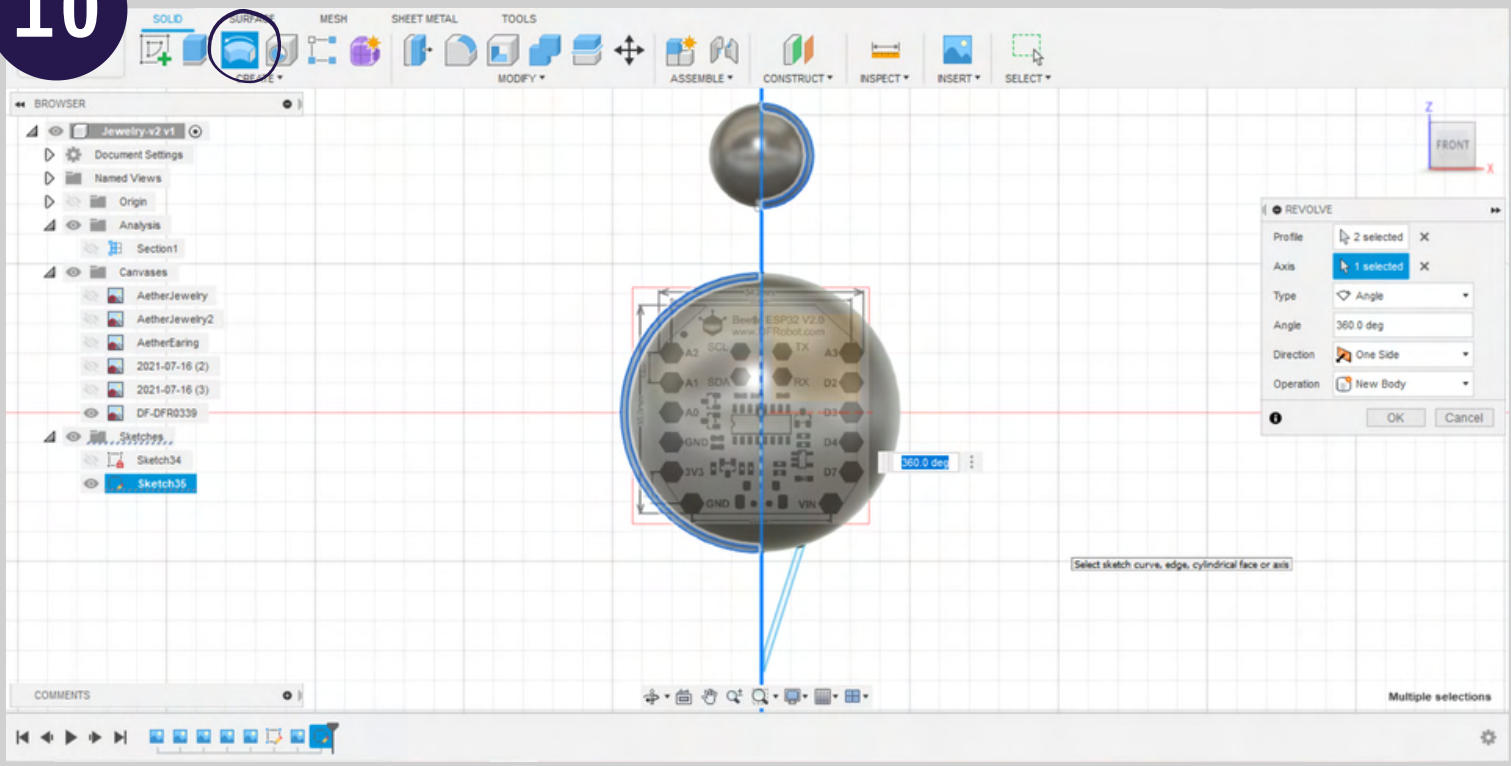




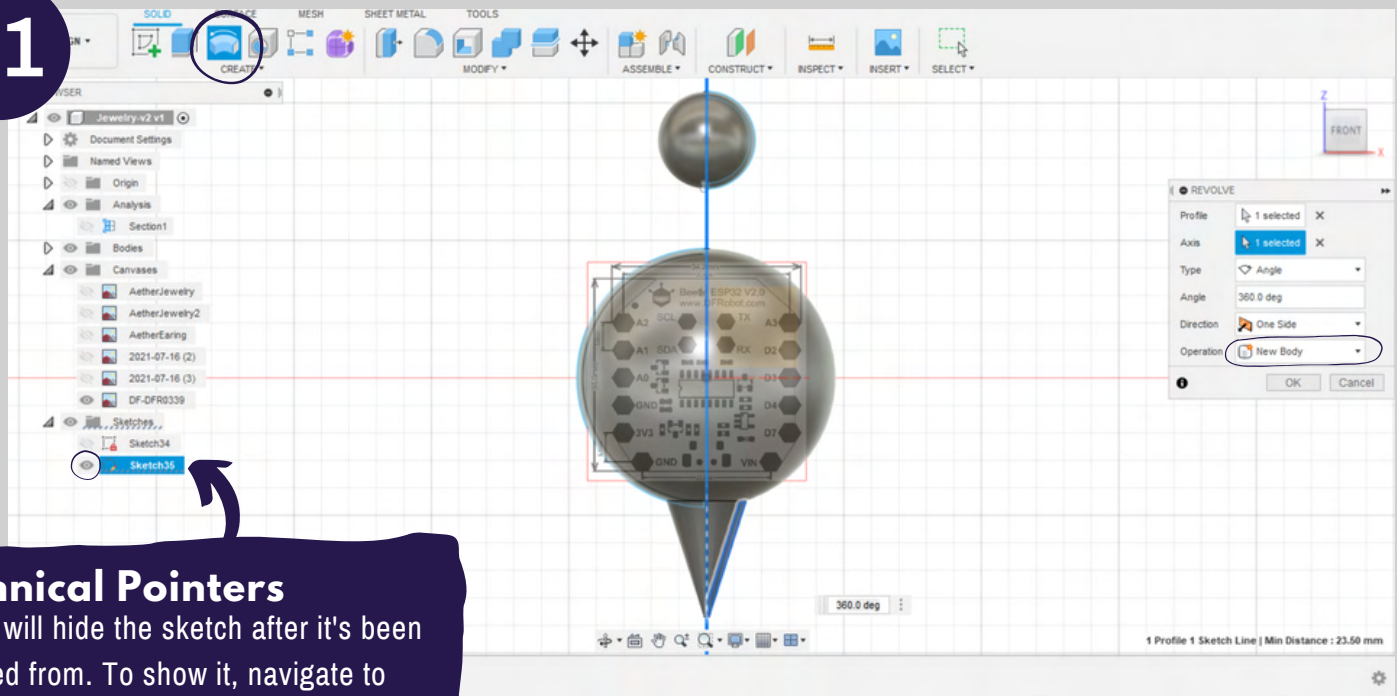
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REVOLVE IT

10 Now **Revolve** all the parts that are not connected to each other - you want to make separate bodies



11 Next **Revolve** that last body - making sure to make a new body instead of joining it to the other bodies.



Technical Pointers

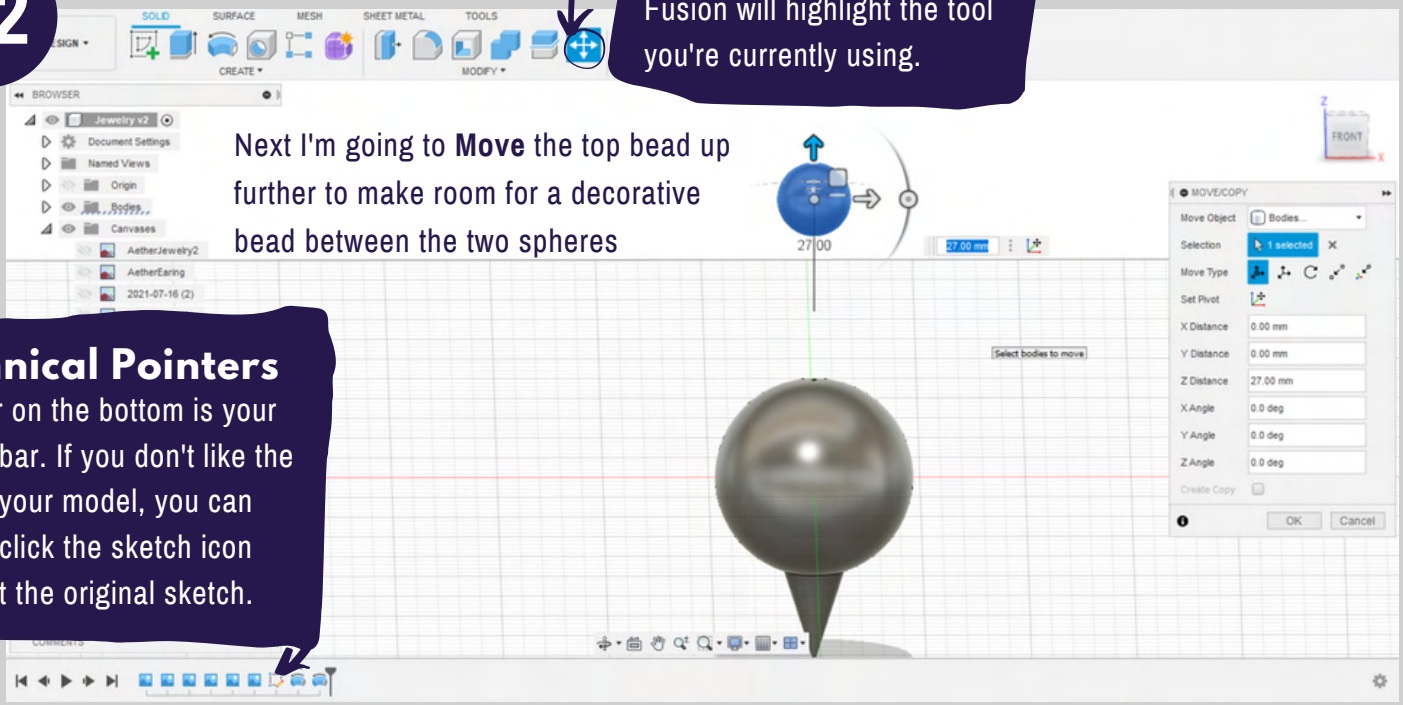
Fusion will hide the sketch after it's been extruded from. To show it, navigate to the Sketches Folder or the Browser and click the eye with the cross through it.



OFFSET IT

12

Technical Pointers
Fusion will highlight the tool you're currently using.

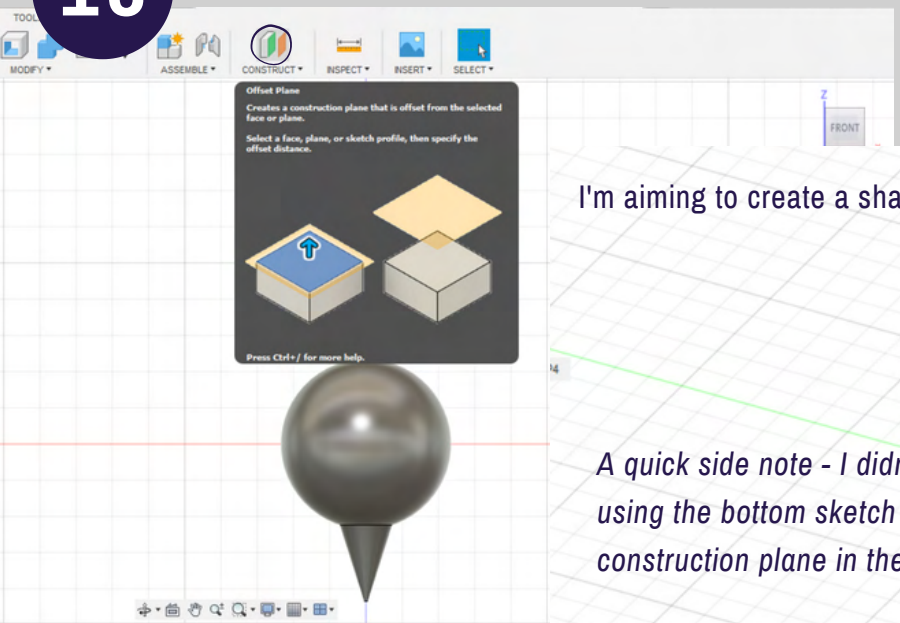


Next I'm going to **Move** the top bead up further to make room for a decorative bead between the two spheres

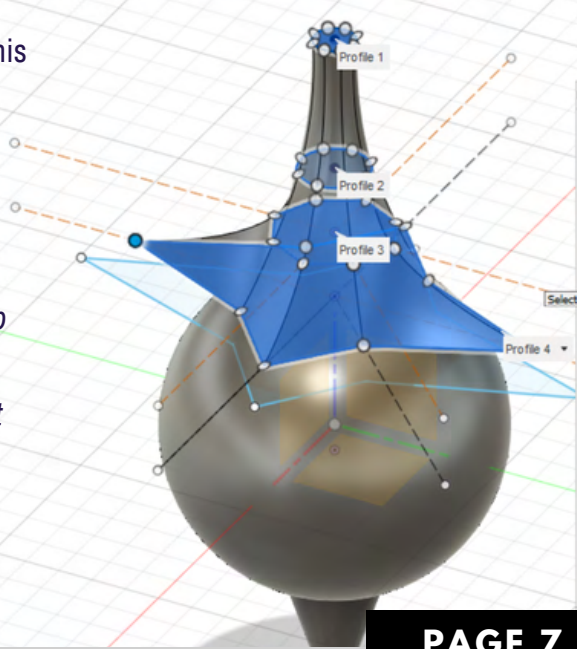
Technical Pointers
The bar on the bottom is your history bar. If you don't like the look of your model, you can double click the sketch icon and edit the original sketch.

13

Next lets make a selection of **Offset Planes** for use in lofting



I'm aiming to create a shape like this



A quick side note - I didn't end up using the bottom sketch on the construction plane in the final loft

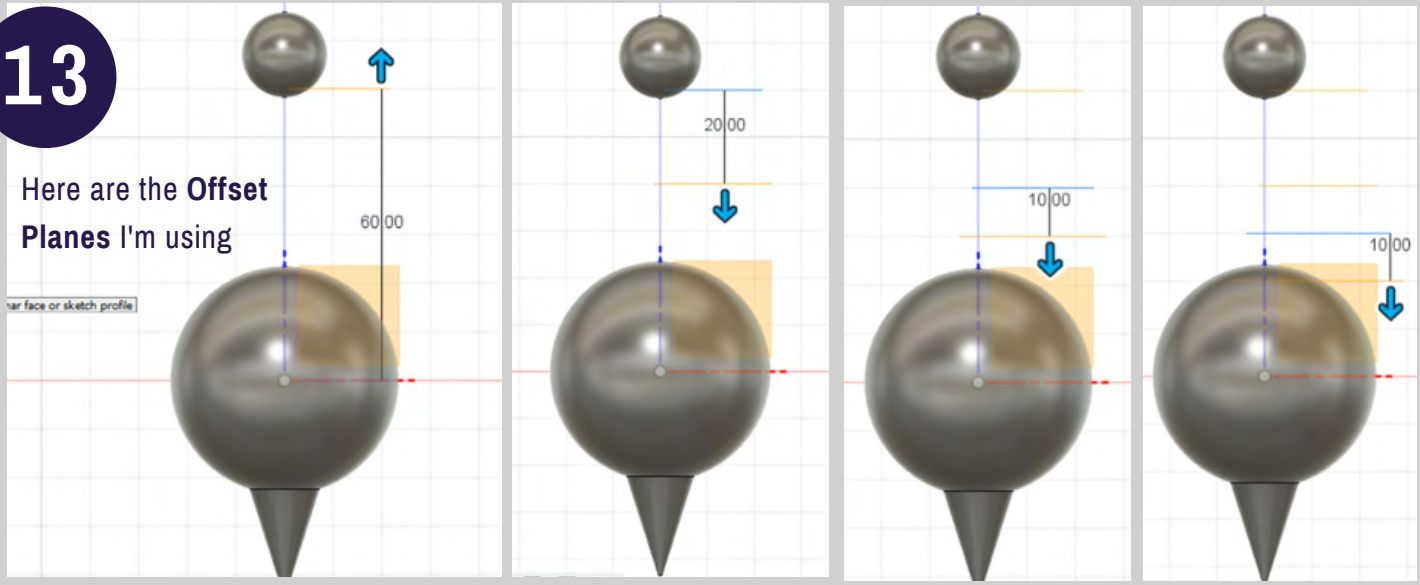


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PREPARING TO LOFT

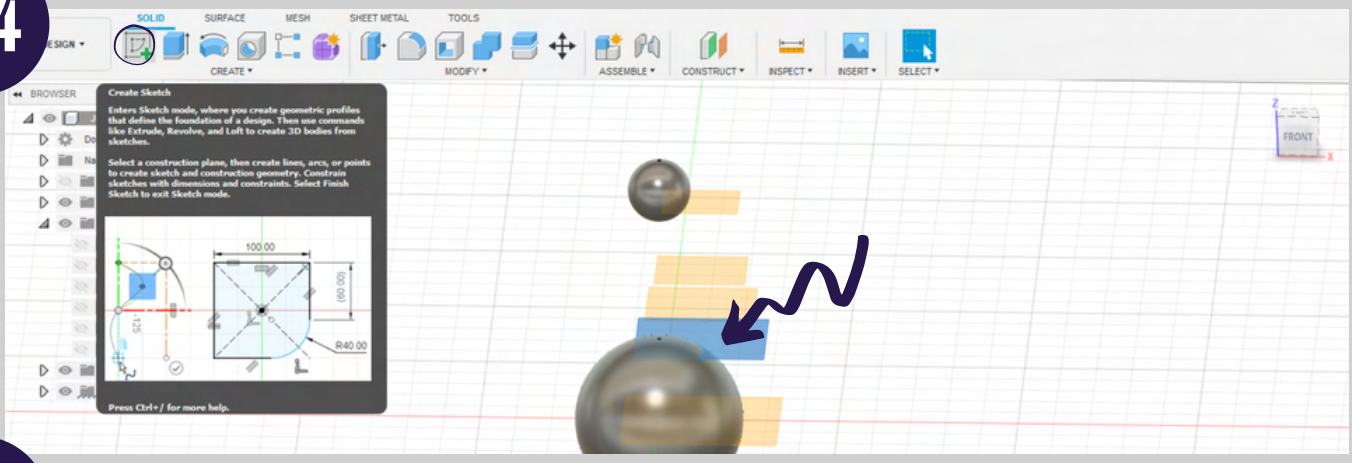
13

Here are the **Offset Planes** I'm using

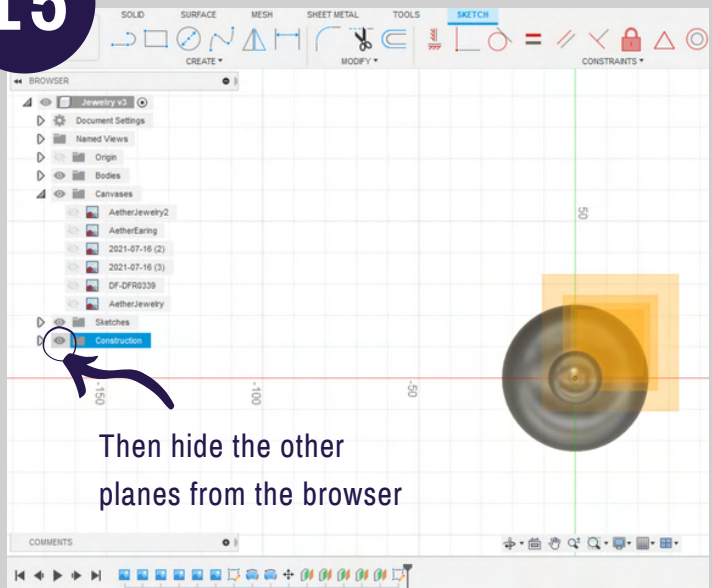


14

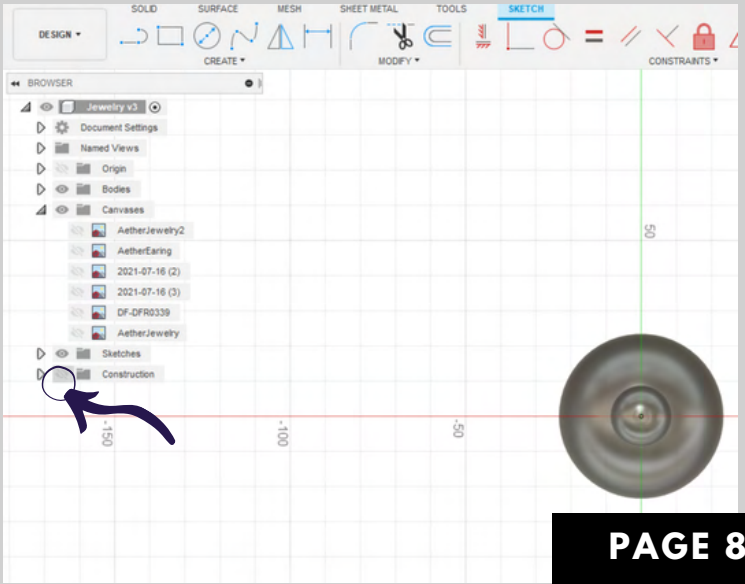
Next create a **Sketch** using the second to last offset plane



15



Then hide the other planes from the browser





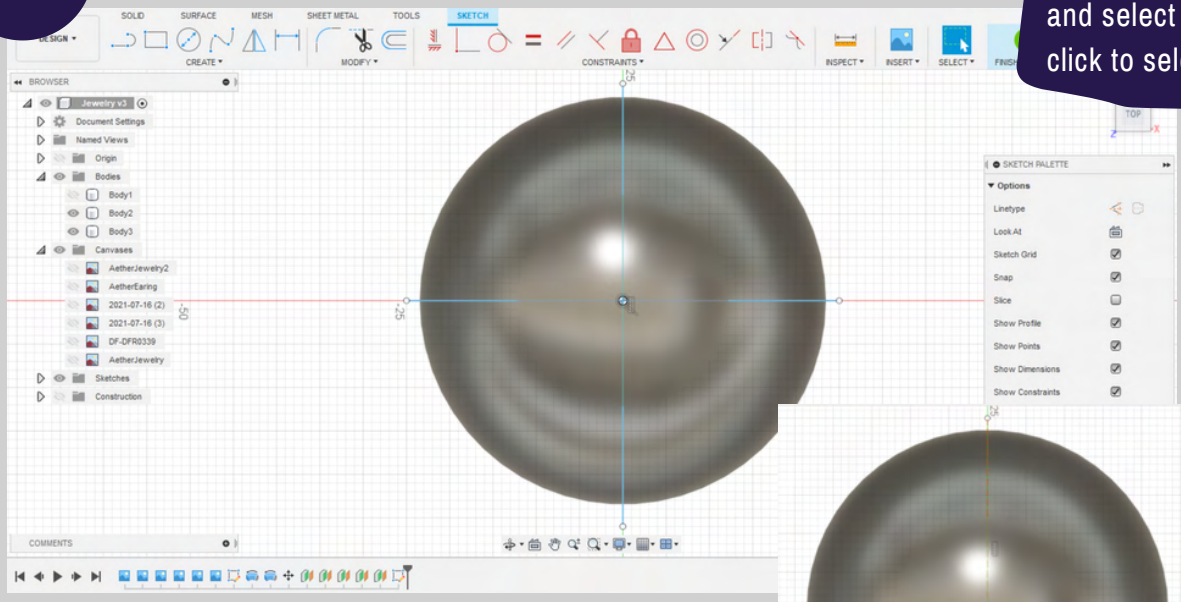
CITY OF CANADA BAY LIBRARIES
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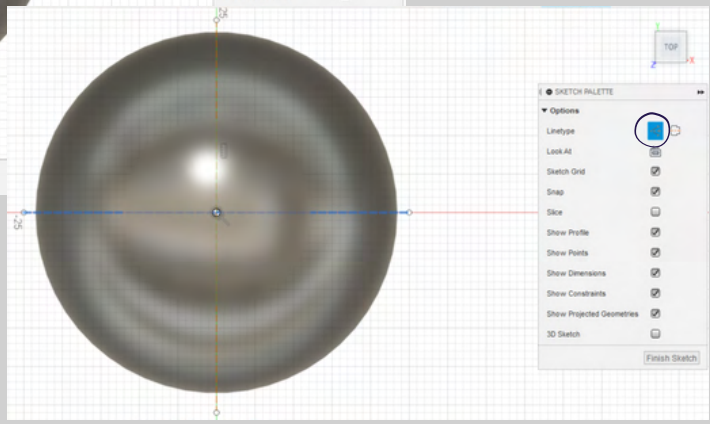
CREATE THE MAIN SHAPE

16 Next draw a cross through the center and turn those lines into construction lines by selecting them and typing X

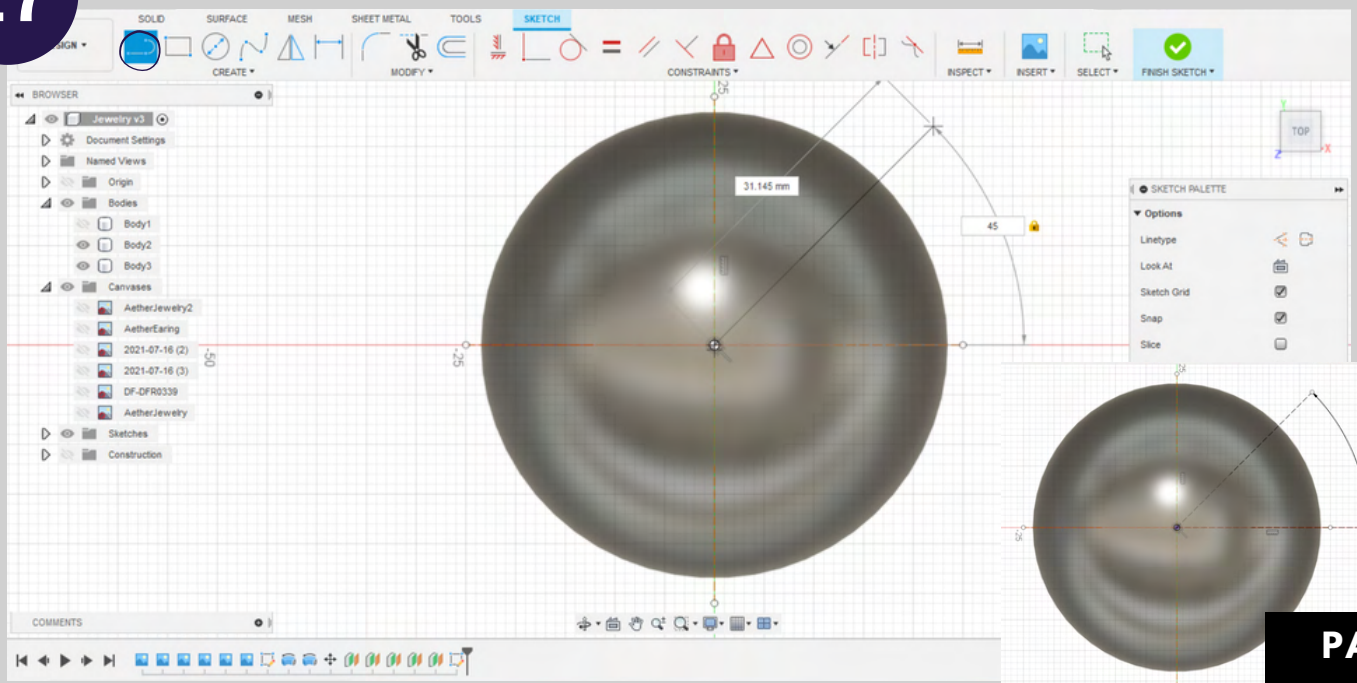
Technical Pointers

To select multiple lines hold shift and select each one OR double click to select all connected lines





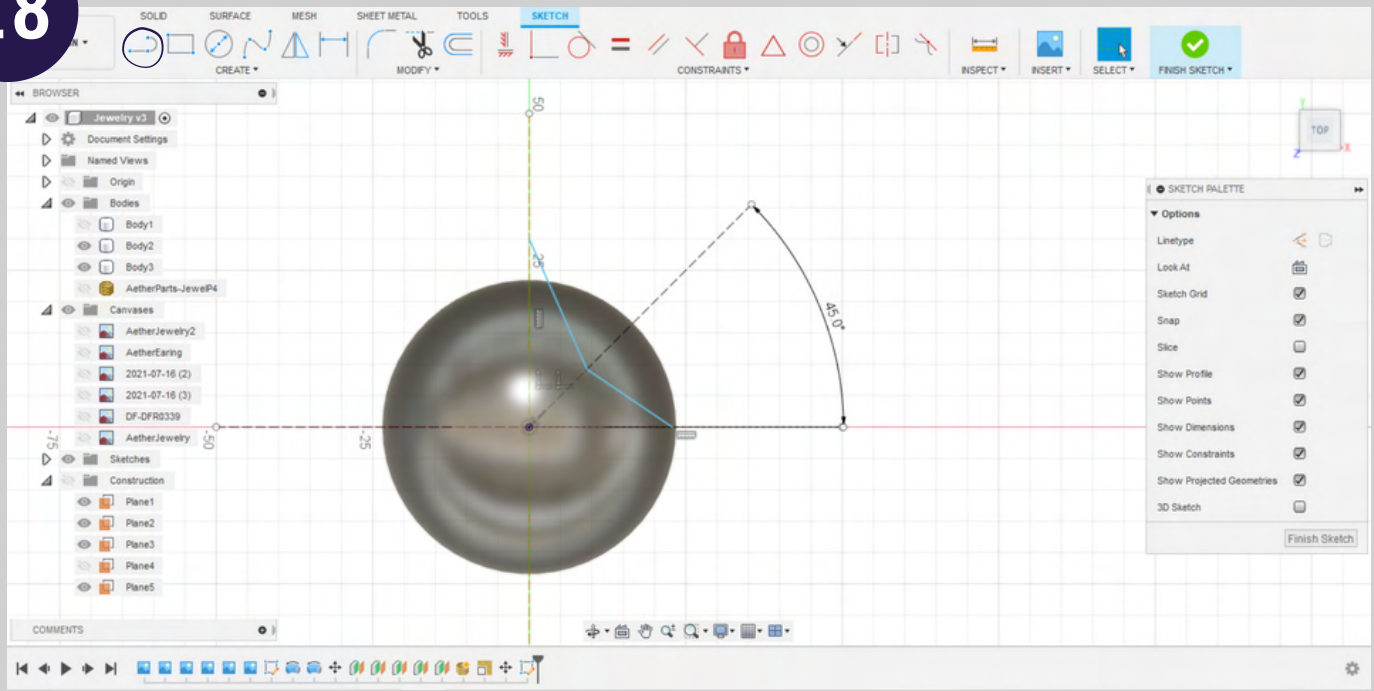
17 Next draw a line at a 45 degree angle in one of your quadrants and make it a construction line too



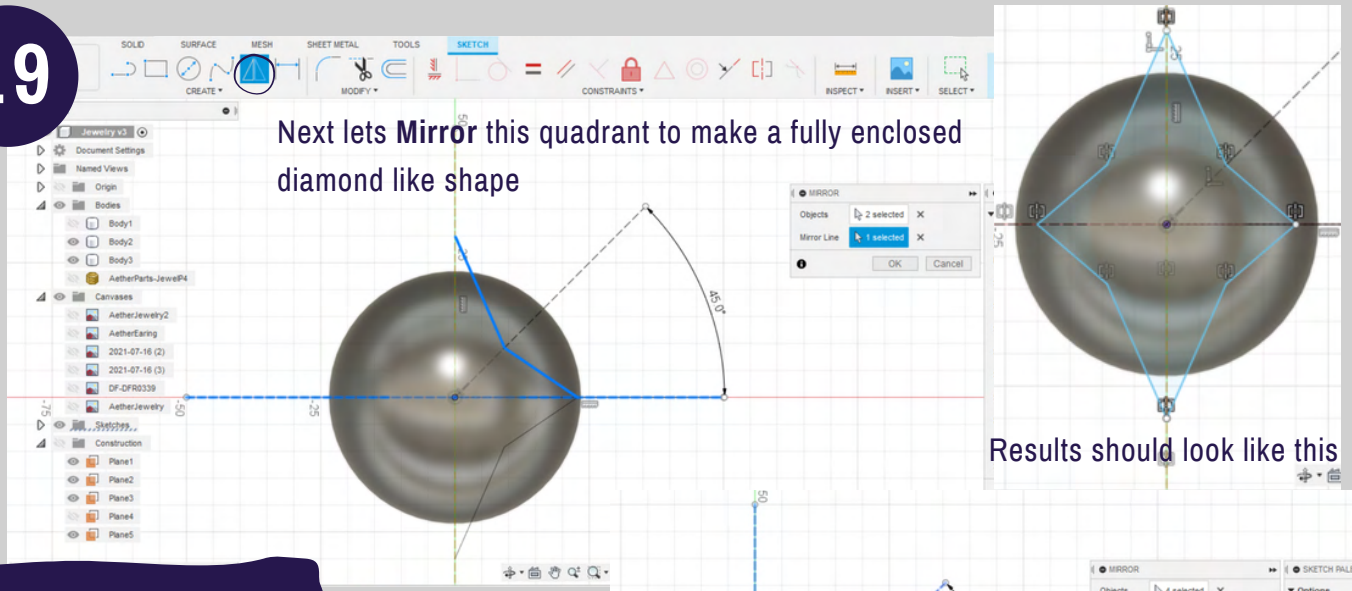


MIRROR IT!

18 Now draw a quarter of a diamond-like shape, see below for an example...



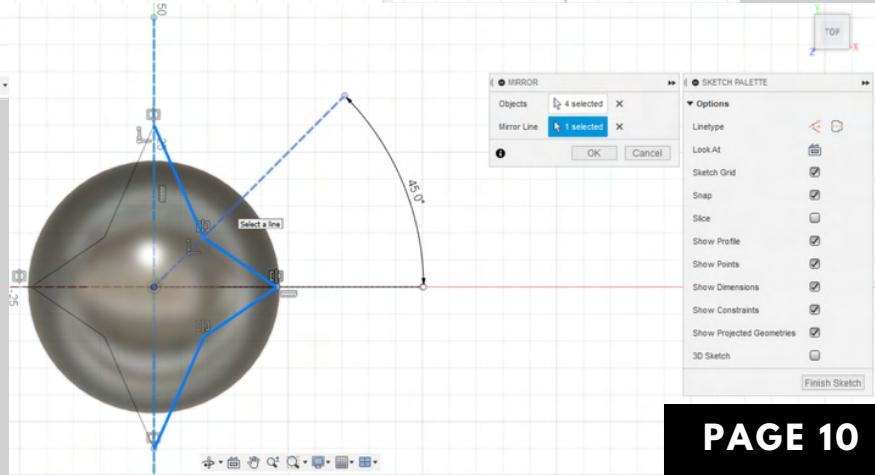
19 Next lets **Mirror** this quadrant to make a fully enclosed diamond like shape



Results should look like this

Technical Pointers

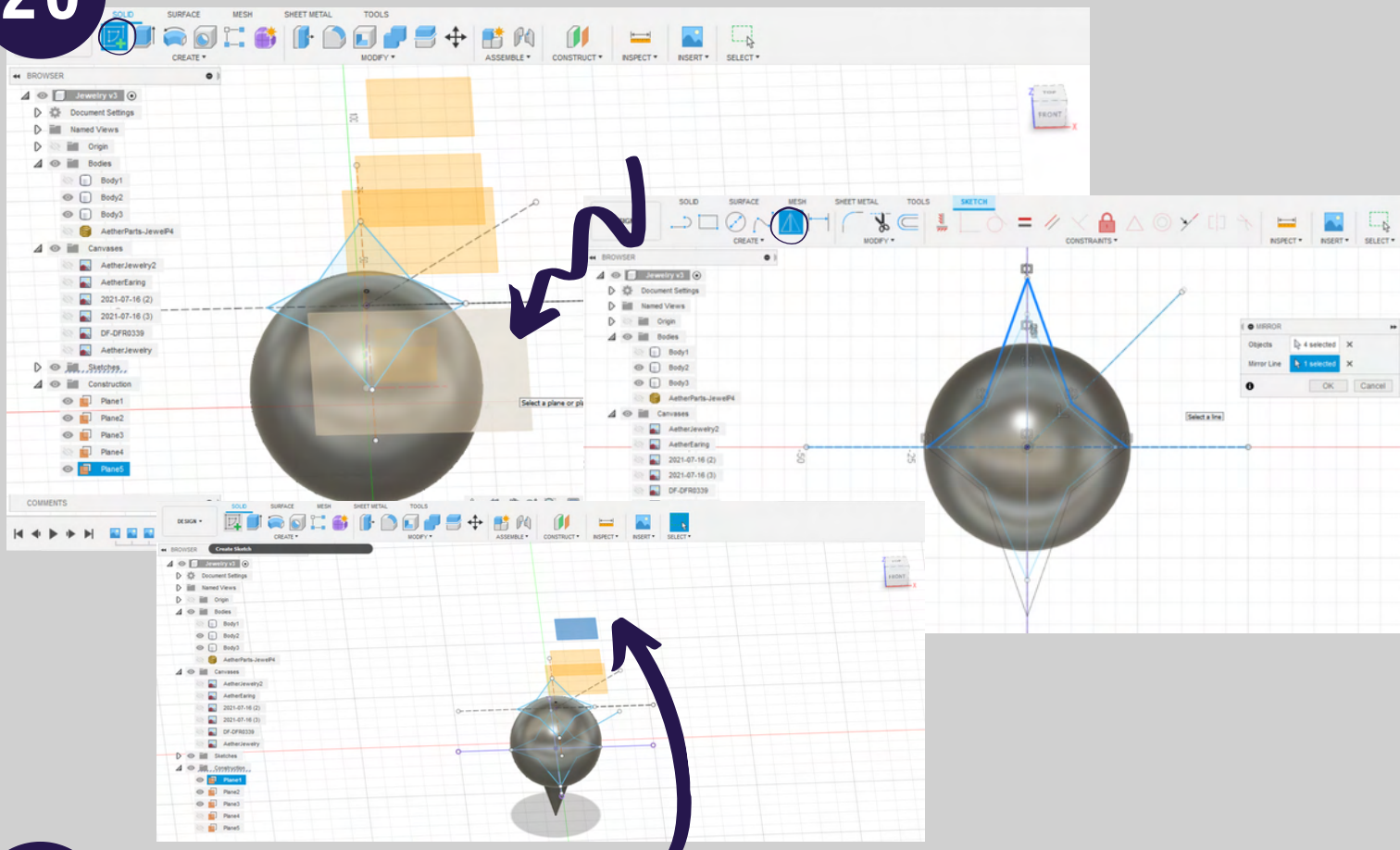
Fusion will show what the mirror will produce with black lines



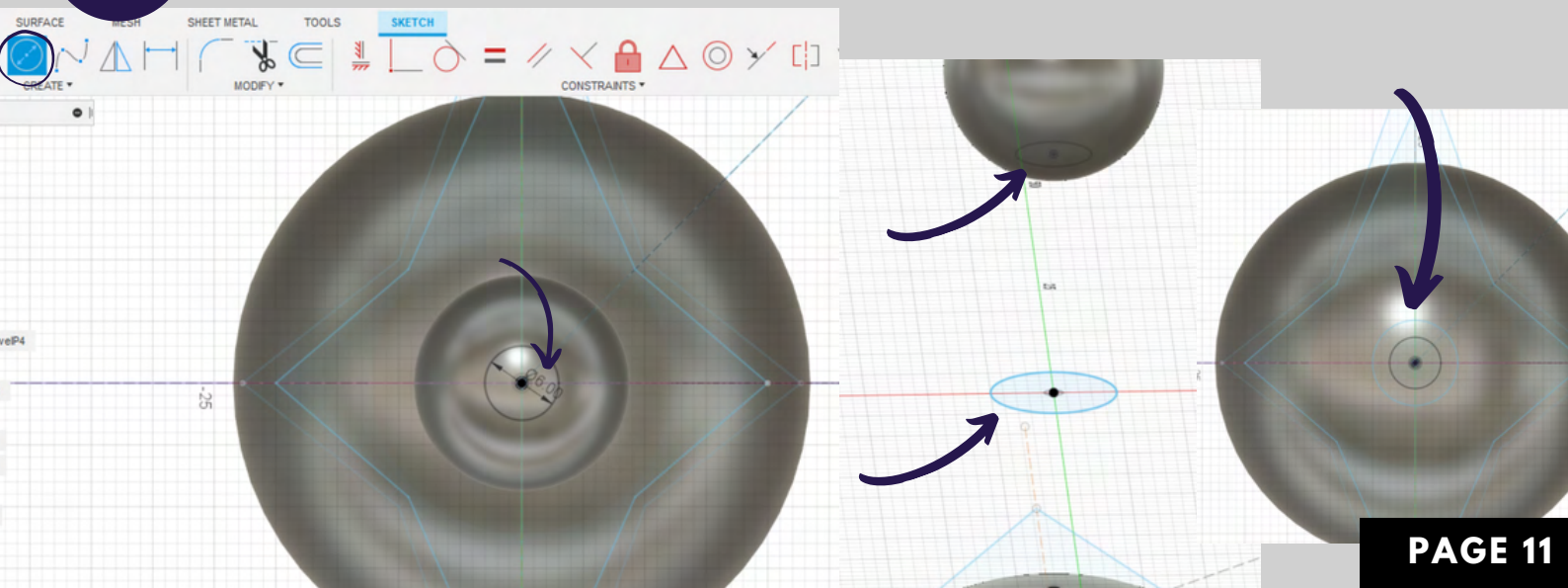


REPEAT THE LAST STEP

20 Next (after I made the construction planes visible again) I drew a similar diamond-like shape on the bottom plane. I did not use this shape in the final loft but you may want to experiment with yours



21 Then I drew two circles on the top two construction planes. The top one was 6mm wide and the second one was just a little wider to create a gently sloping neck for this loft

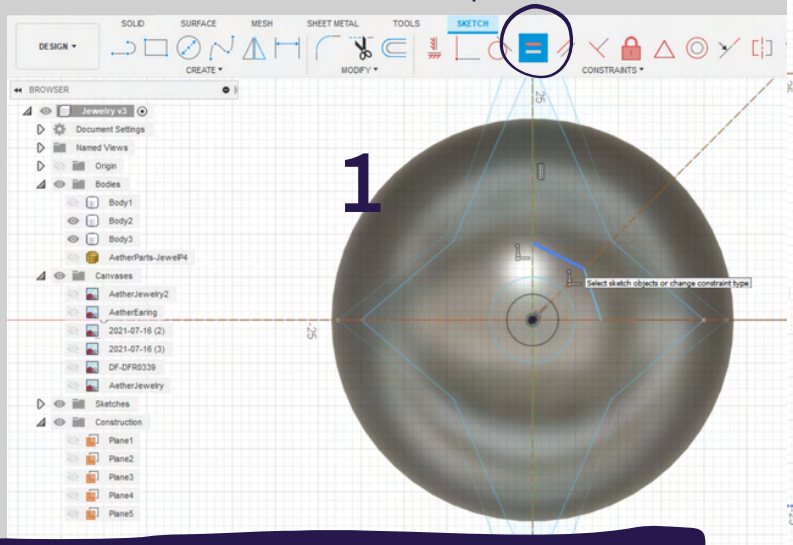


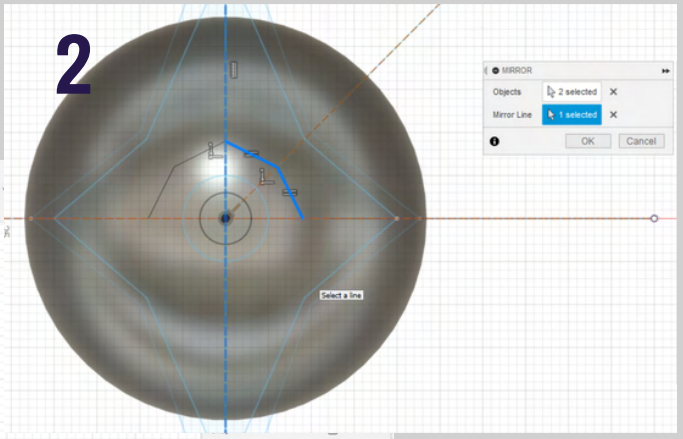


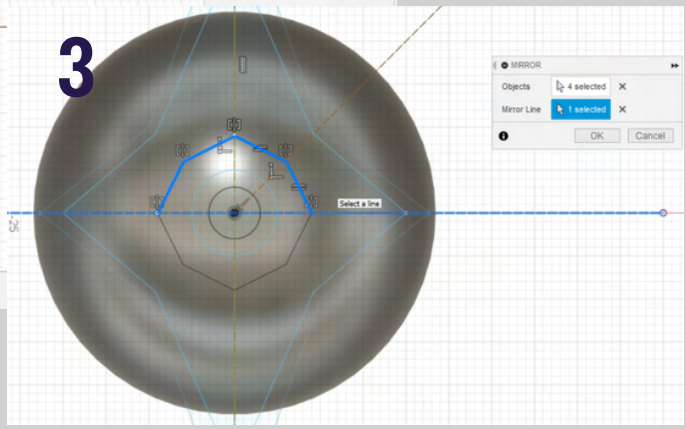
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MAKE THE TRANSITION

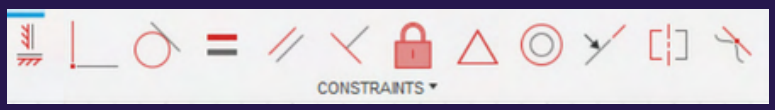
22 Finally, select the middle construction plane and create an octagon using the same method we used to create the diamond-like shapes. I used the **Equals** constraint to make sure the lines were equal





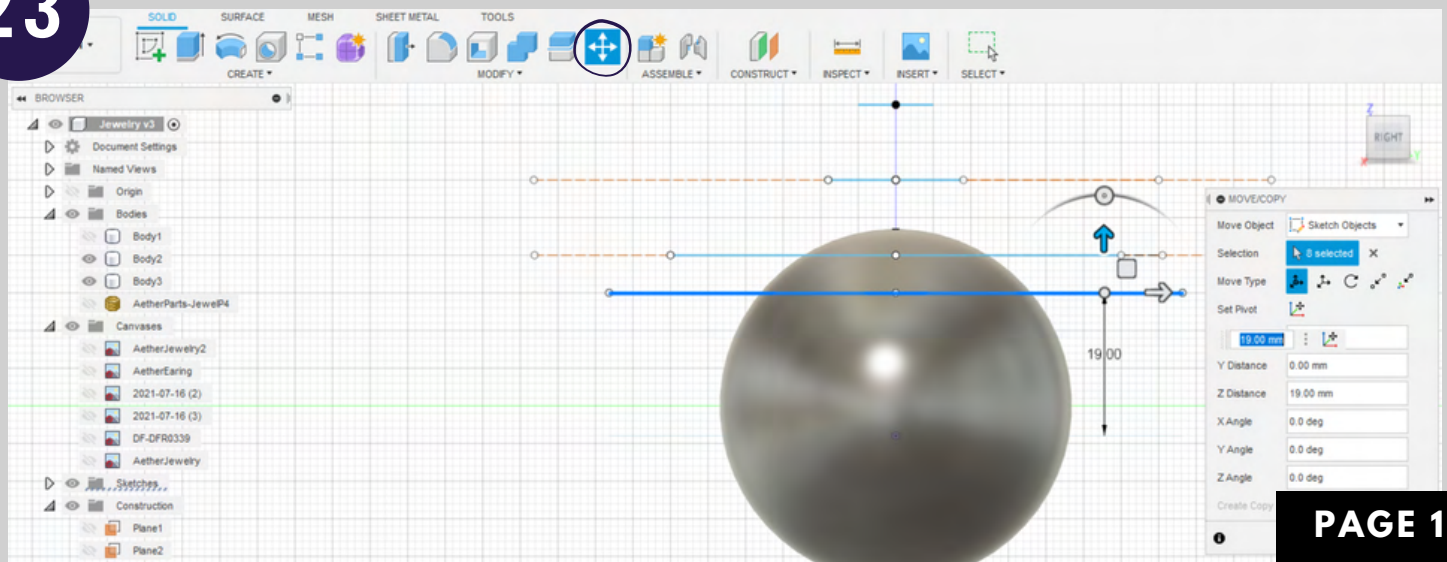


Technical Pointers



What are constraints?
Constraints are tools in Fusion that allow you to relate one sketch entity to another sketch entity.

23 Then I moved up my bottom shape using the **Move** tool



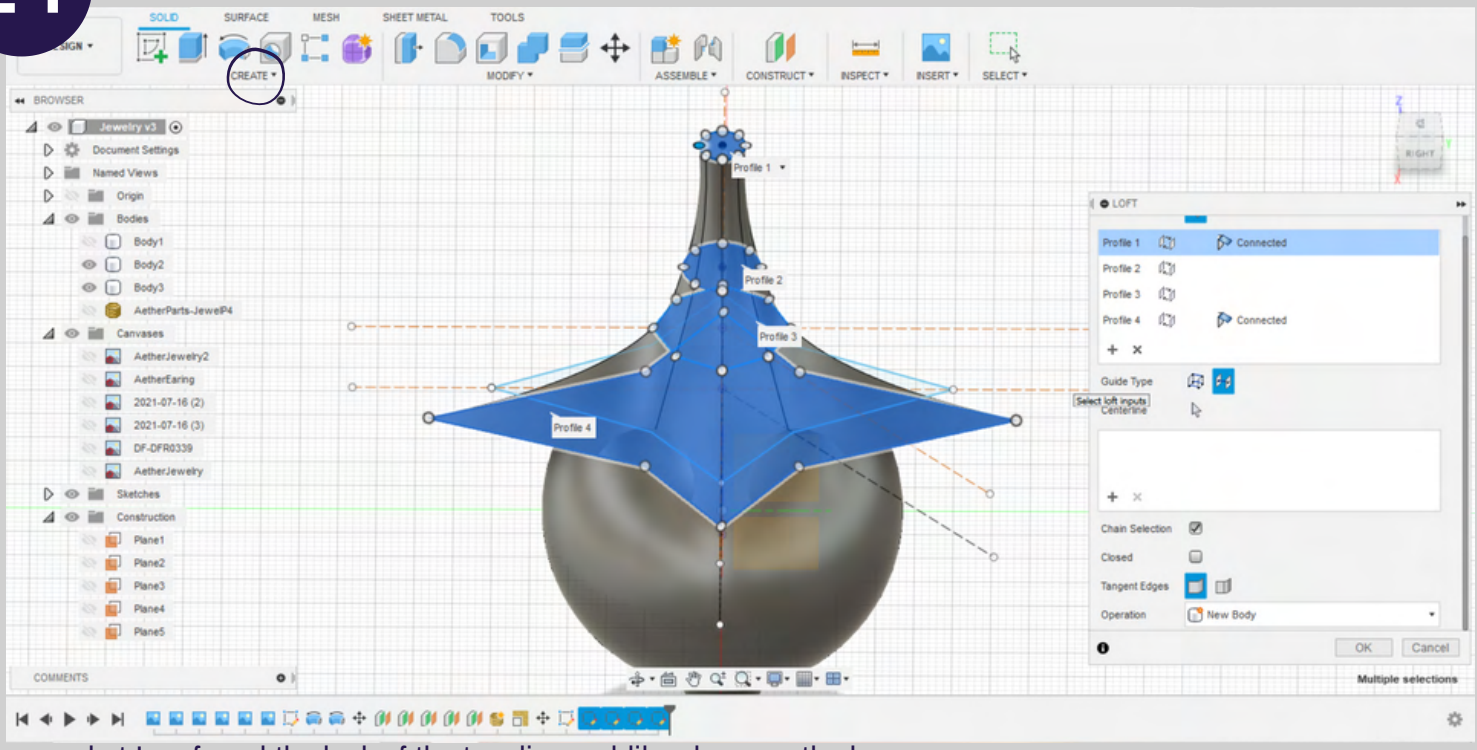


LOFT IT!

24

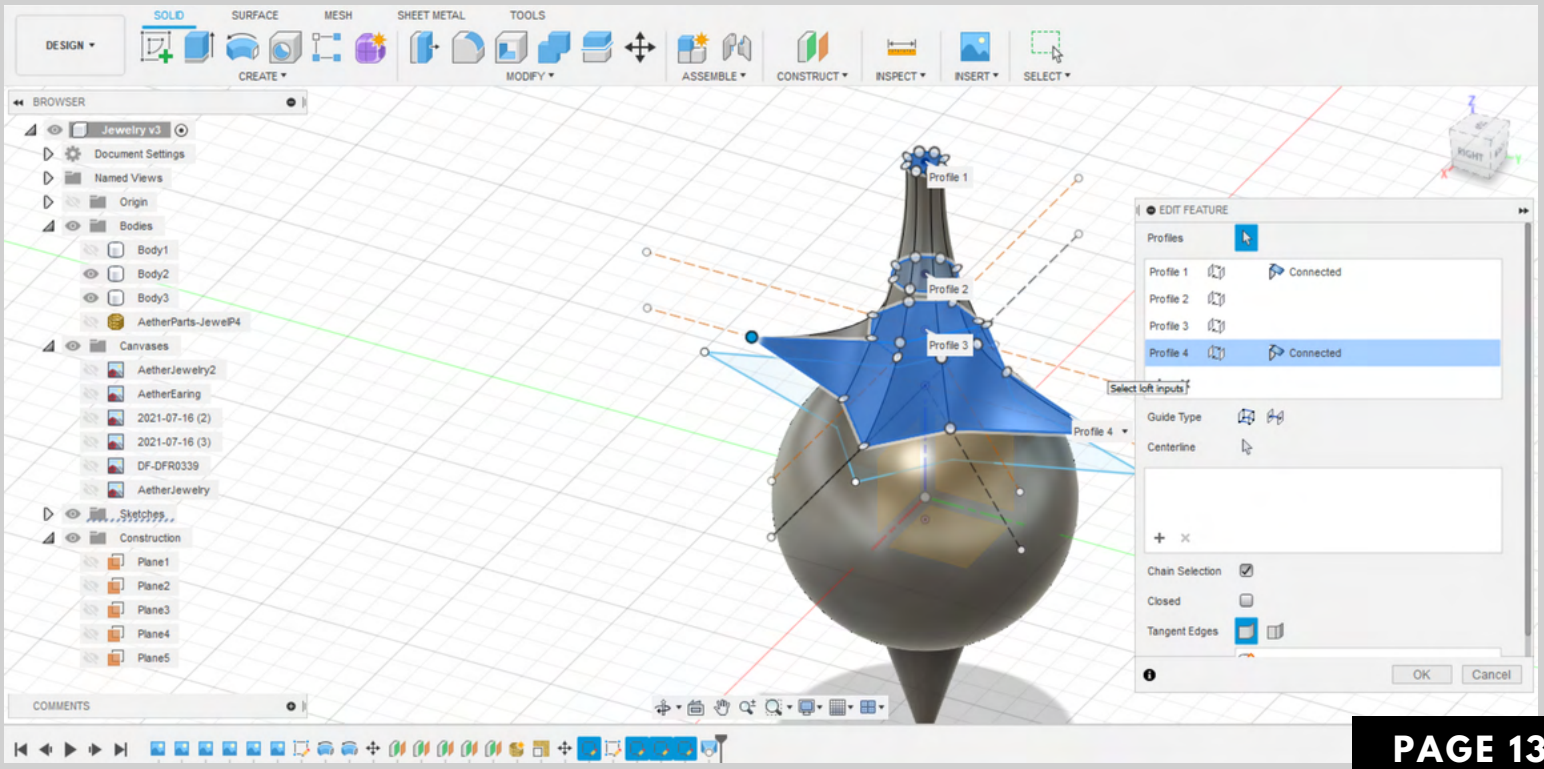
Now to experiment with the **Loft** function (found in the Create menu).

At first I used the bottom diamond like shape in the loft...



...but I preferred the look of the top diamond-like shape as the base.

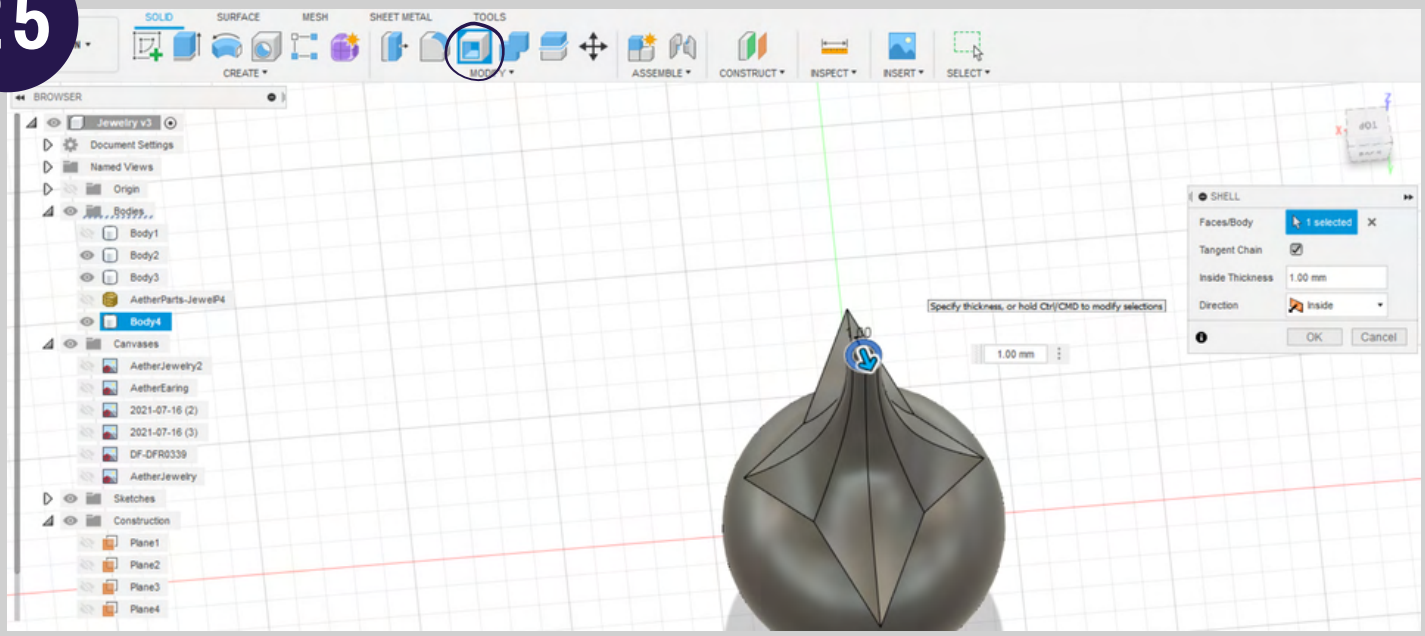
You may want to experiment as well to see what shape you like for your jewellery design.



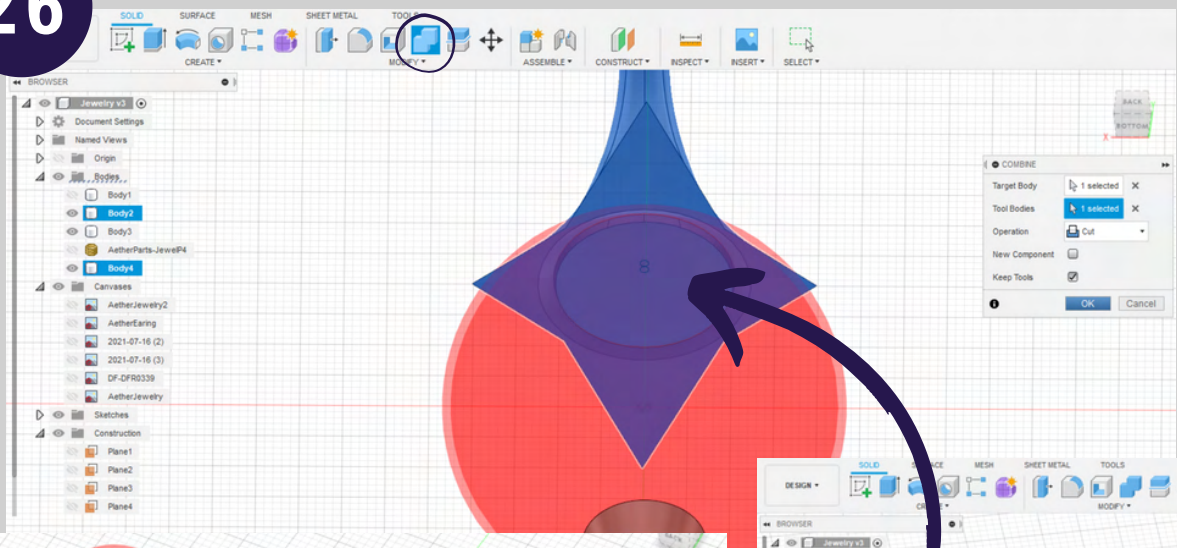


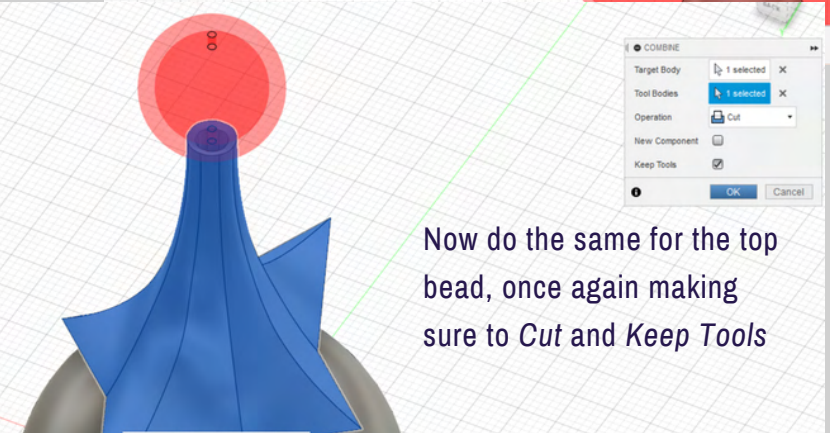
MAKE IT FIT

25 Next I hollowed out the body using the **Shell** tool and specifically selecting the top face

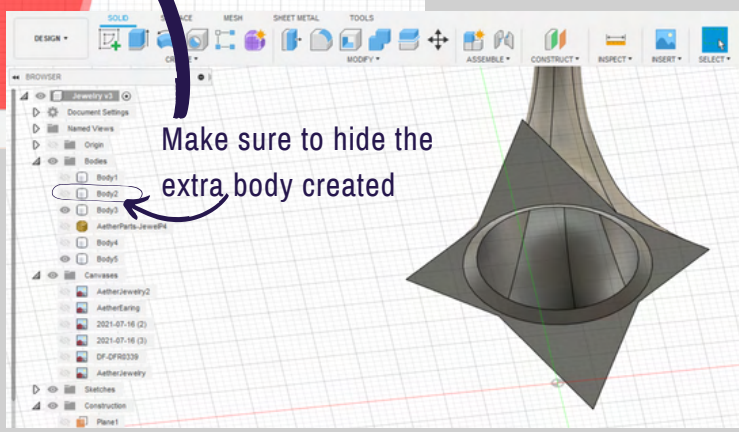


26 Now use the **Combine** tool to cut the new body so it fits to the other beads. Start by choosing the new body as the *Target Body* for the combine and choose the large sphere as the *Tool Body*. Make sure its set to cut and you select '*Keep Tools*' before clicking the ok button





Now do the same for the top bead, once again making sure to *Cut* and *Keep Tools*



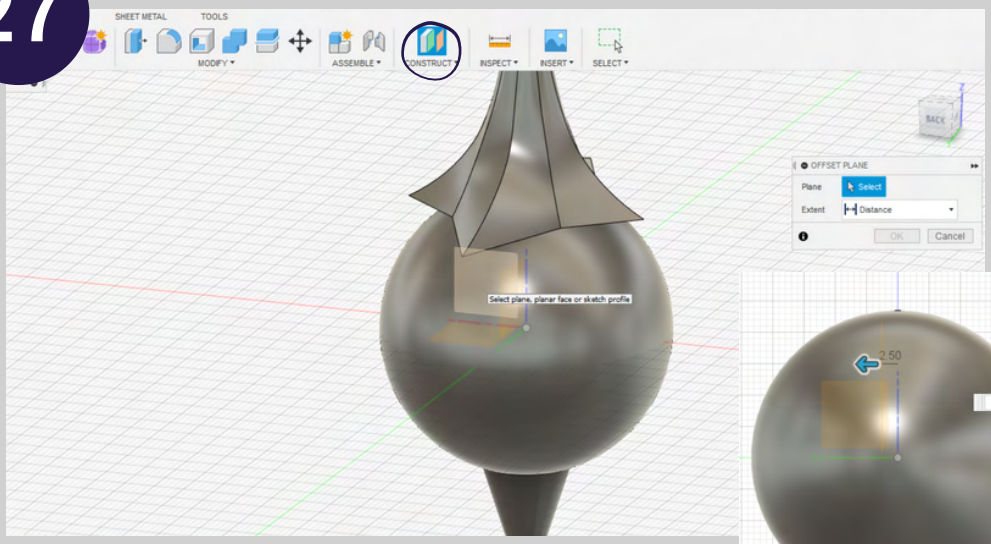
Make sure to hide the extra body created



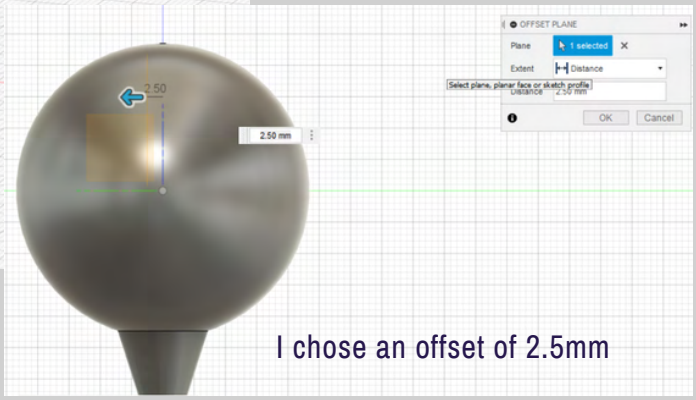
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MAKE AN OPENING

27 Next lets create an **Offset Plane** to act as a cutting tool for the main bead.

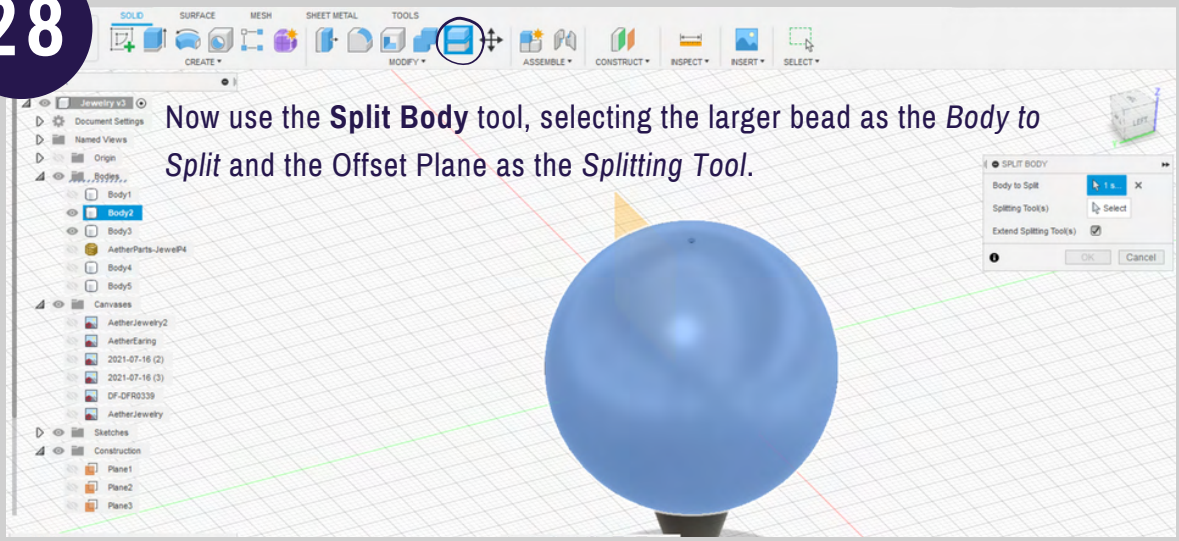


We need to offset the plane enough to allow a reasonable area for thread or a latch to go and to clear the jewellery wire hole, but if it's too far the board won't fit in.

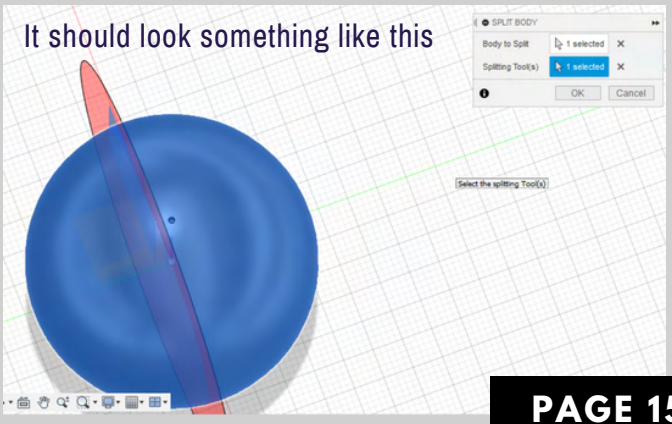


I chose an offset of 2.5mm

28 Now use the **Split Body** tool, selecting the larger bead as the *Body to Split* and the Offset Plane as the *Splitting Tool*.



It should look something like this

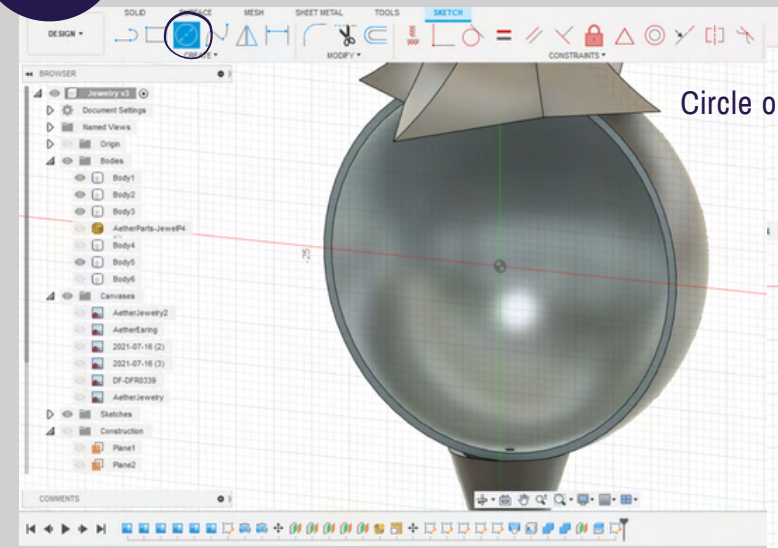


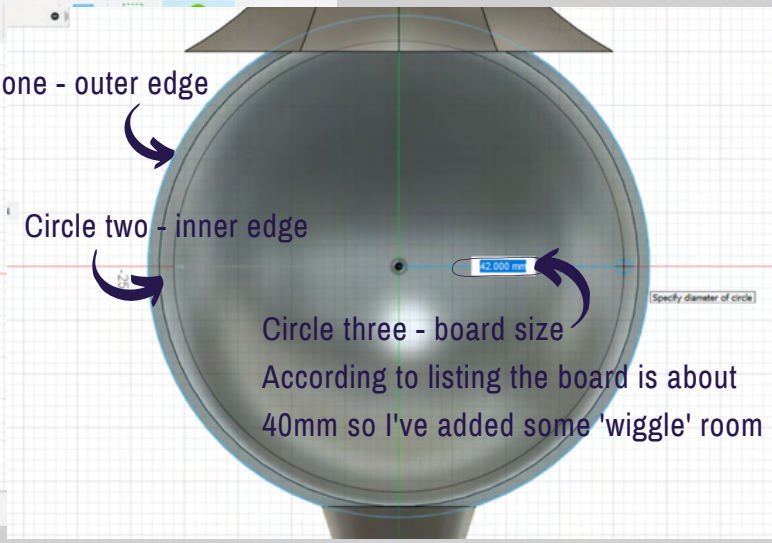


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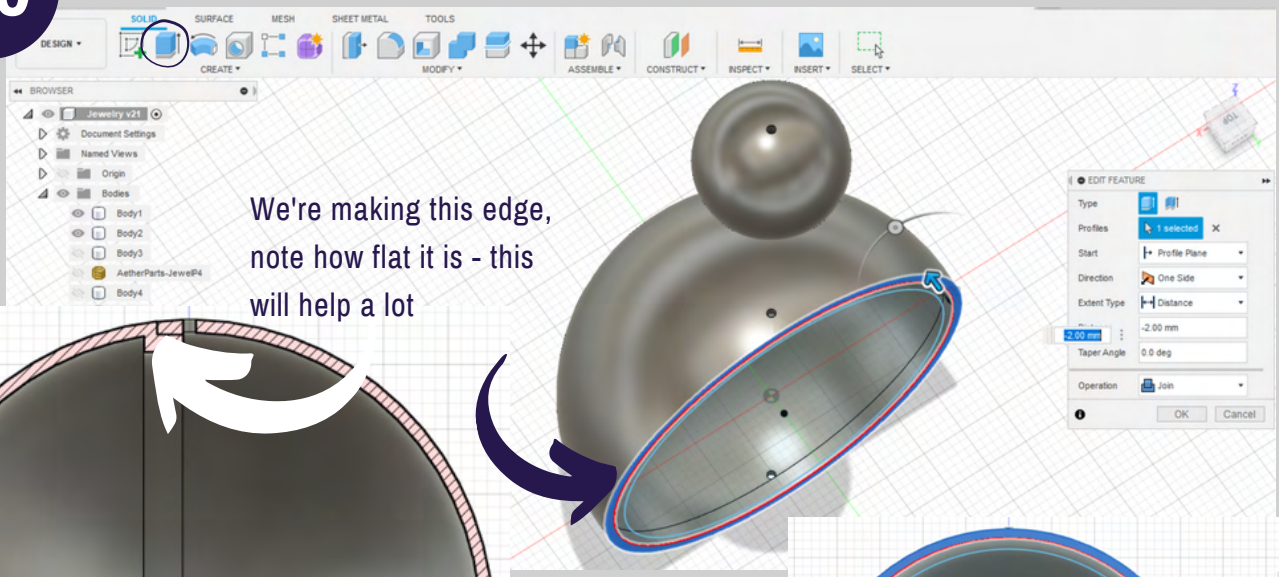
CREATE A LIP

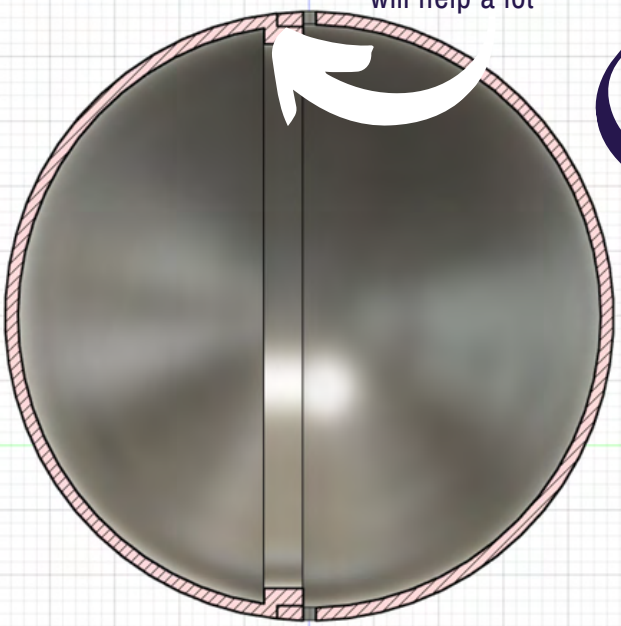
29 Next, we want to create a lip for each side of the bead, open the Sketch menu and hide one half of the bead. Then draw 3 circles, one the size of the outer edge of the bead, of the size of the inner edge of the bead and one the size of your board.



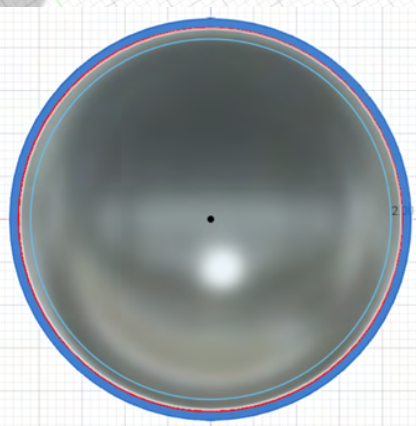


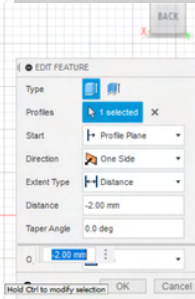
30 Start with the slightly larger half bead with the hole in it and **Extrude** the outer circle in -2mm





The aim is to make a good surface for a thread or fit to go on

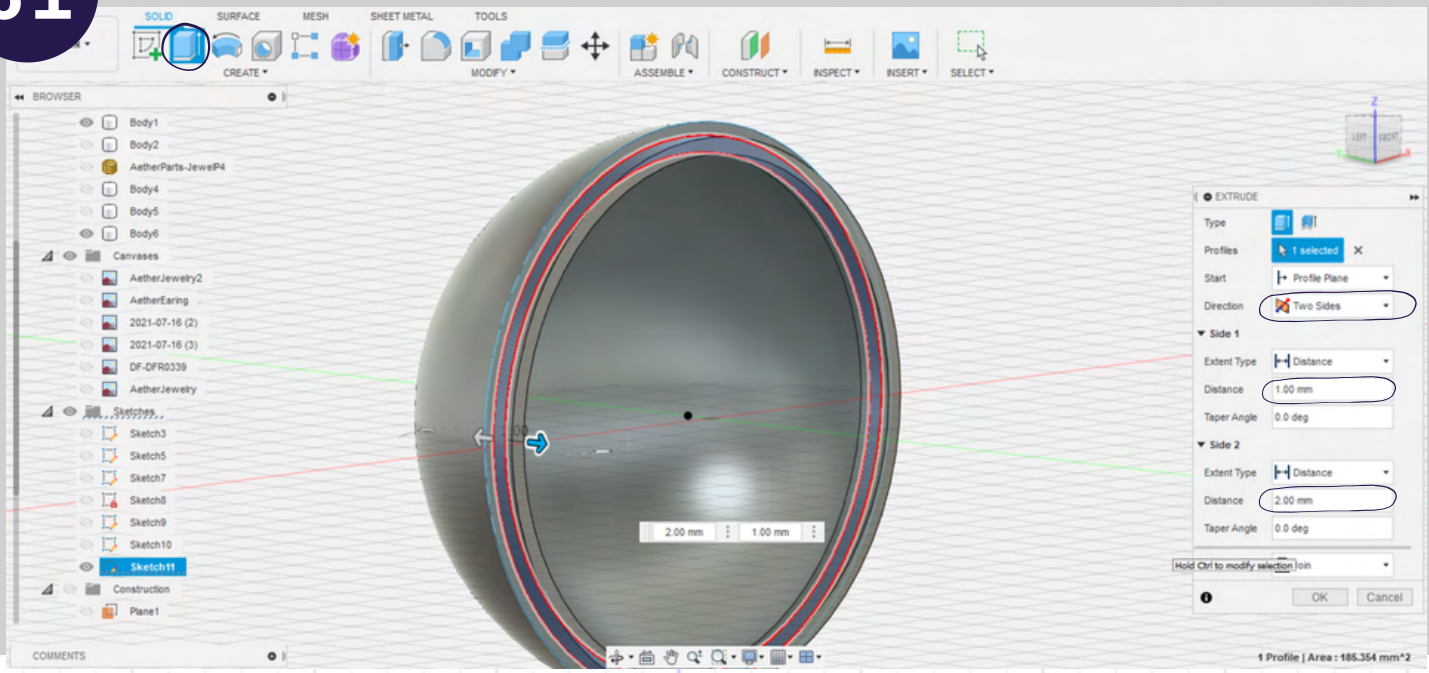


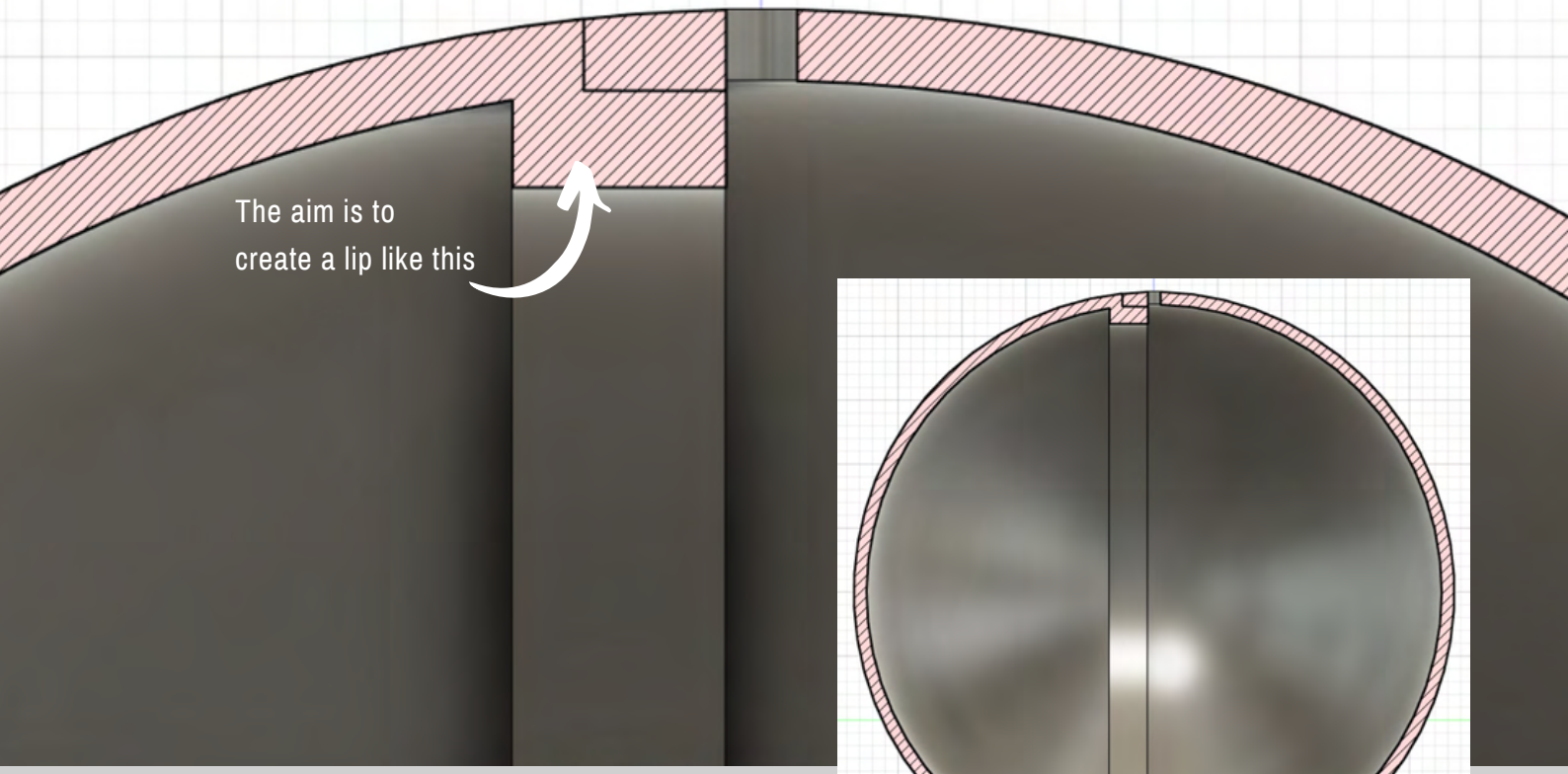




FINISH THE LIP

31 Next, hide the larger part of the bead and show the smaller part. Then **Extrude** the middle ring back into the bead and forward 2mm.



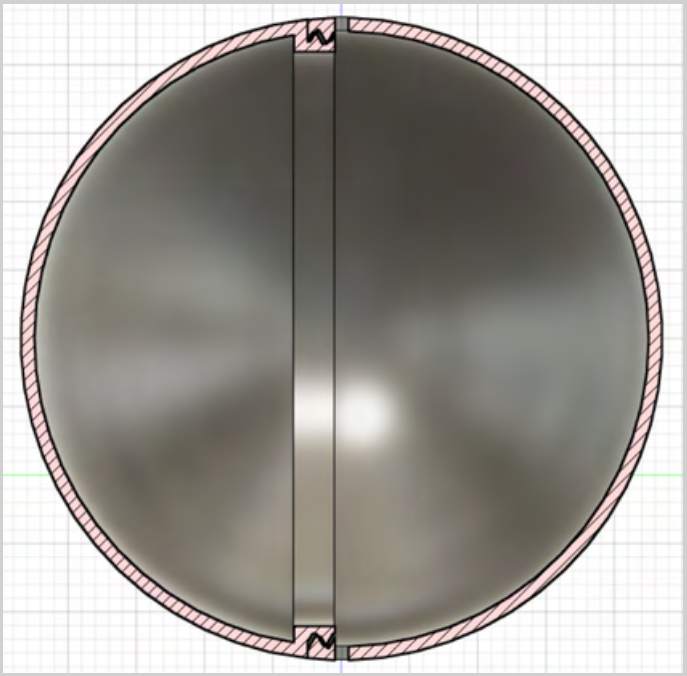


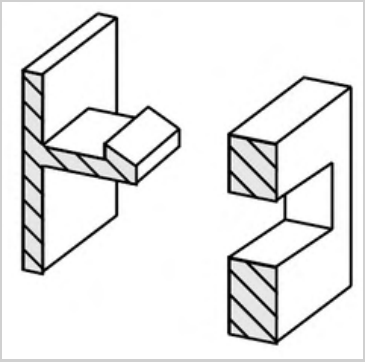


CONNECTING THE TWO HALVES

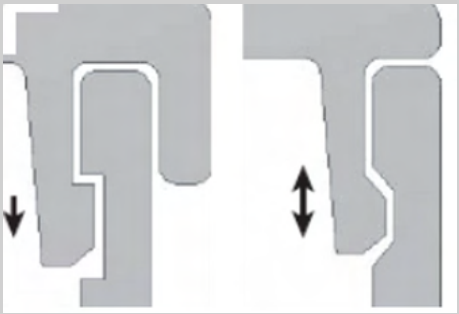
So, there are a few things to think about when considering how you will connect these two halves together.

First, how are you printing this piece?
I'm using SLA printing so the resolution available to me is quite fine compared to an FDM printer. Therefore I'm not concerned about the printer wrecking my threads.
However, your FDM printer might be well-calibrated and able to handle small details. The great thing about Fusion is that you can delete the threads from the **History Bar** if they didn't work and try something different.



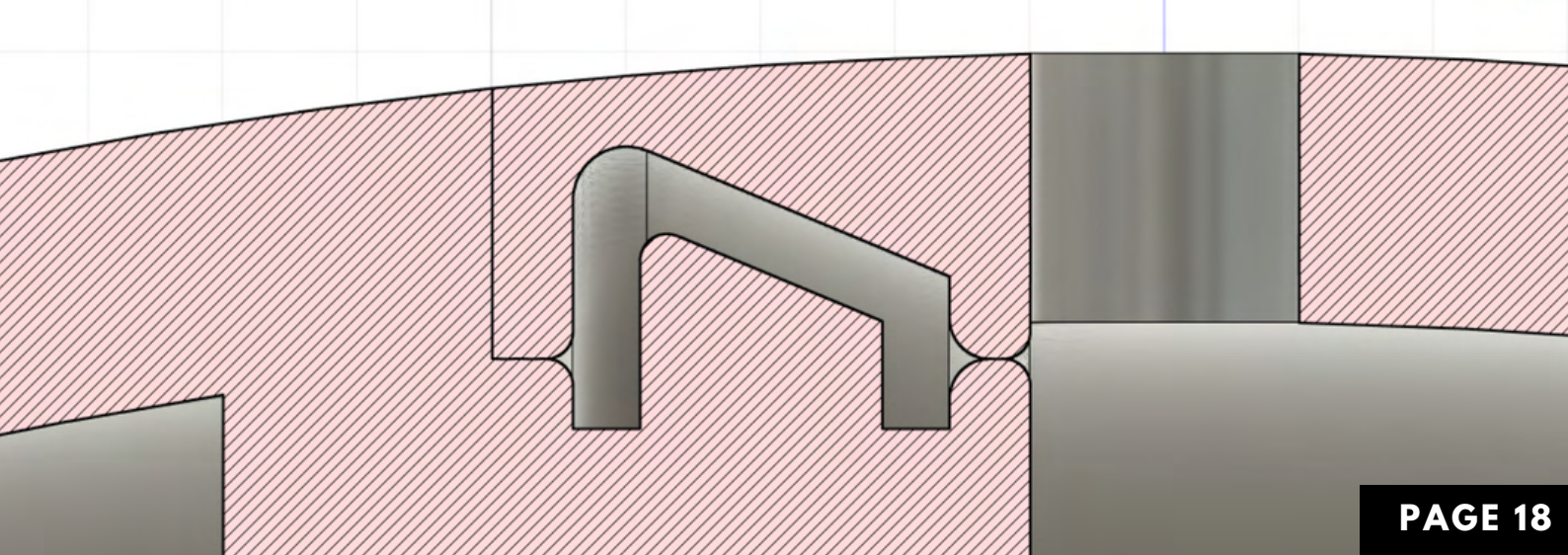


Cantilever snap-fits
Hub.com



Backup snap fit design that holds the fitting into place
infocenter.3dsystems.com

Another option is a snap-fit design. This design would work better with a more flexible filament like ABS or PETG as the model will need a certain amount of 'give' to make such a small snap-fit work. However, there's more space to put in a printing tolerance.
If you go with this design you may want to print off a few test prints to see what your printer and filament can handle.



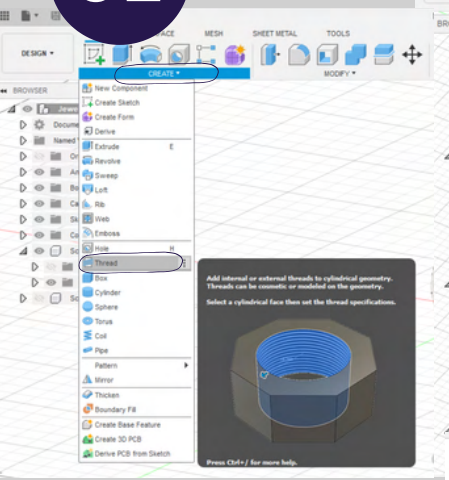


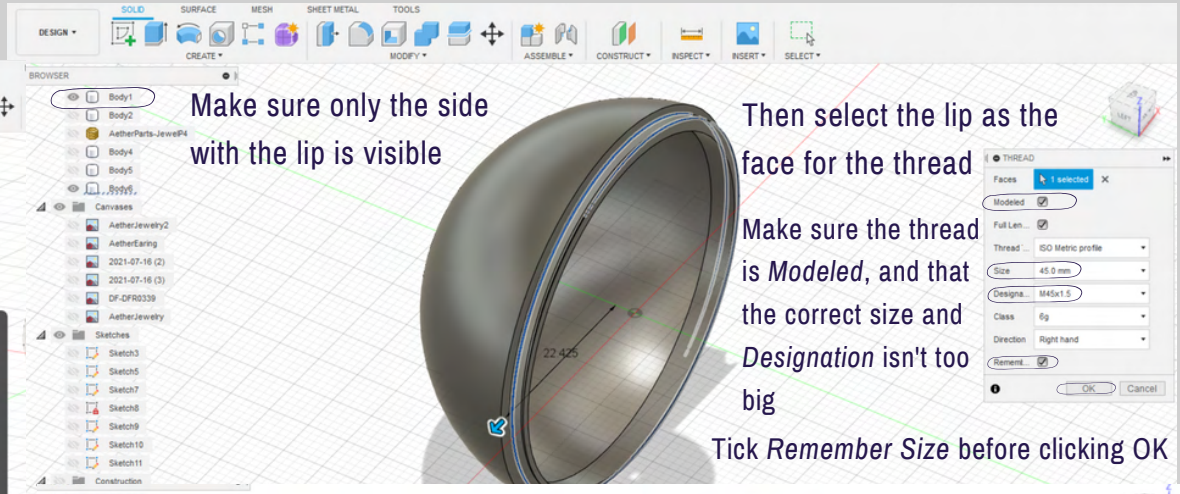
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OPTION 1: THREAD

32

Applying a thread is easy enough, head to the Create menu and select the **Thread** tool



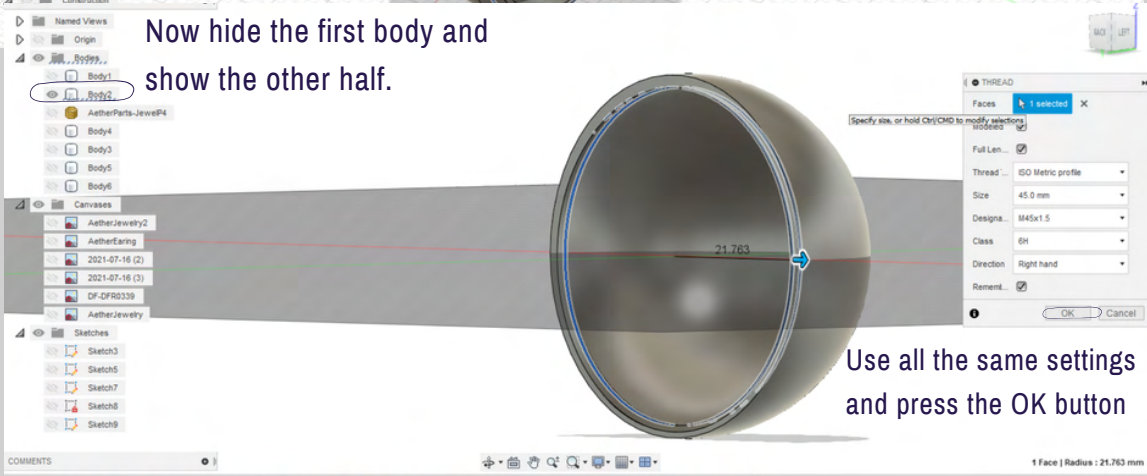


Make sure only the side with the lip is visible

Then select the lip as the face for the thread

Make sure the thread is *Modeled*, and that the correct size and *Designation* isn't too big

Tick *Remember Size* before clicking OK

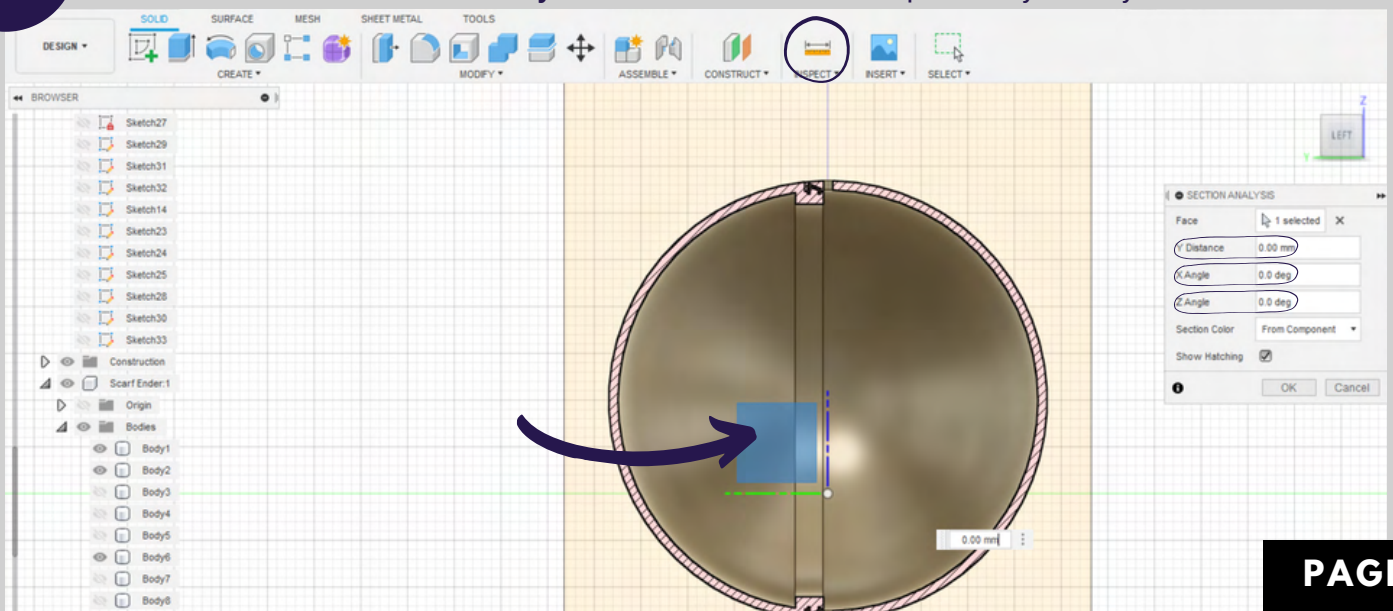


Now hide the first body and show the other half.

Use all the same settings and press the OK button

32

Option 2: The snap-fit. This will be easier to model if you can cut the model in half so go to the Inspect menu and choose the **Section Analysis** tool. Choose the middle plane so you cut your model in half.





OPTION 2: SNAP FIT

Now navigate up to where the two halves meet and draw a shape like you see below...

32

I created the bigger shape first then used the Offset tool to create the smaller shape

Then **Mirror** the shape to the bottom as well

Next **Revolve** the larger shape to make a hole...

...and **Revolve** the smaller shape to make a tab

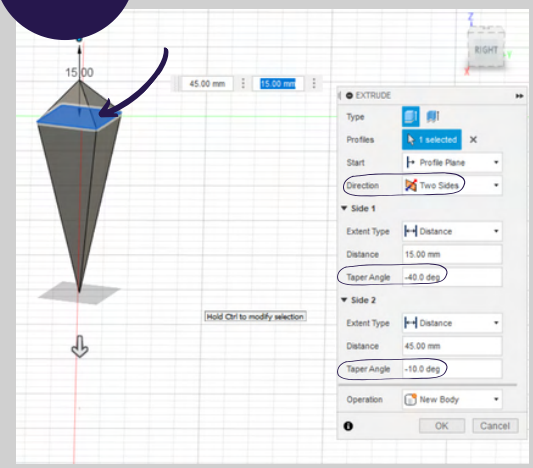
Finally **Fillet** any hard corners which might cause unnecessary friction

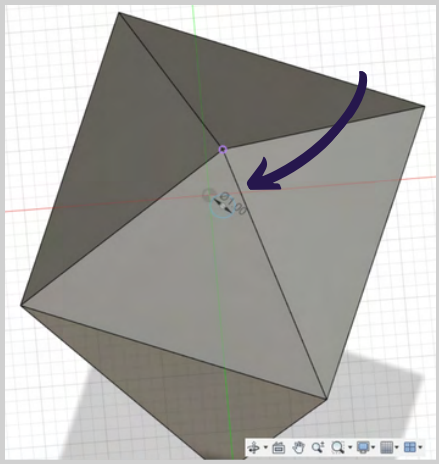


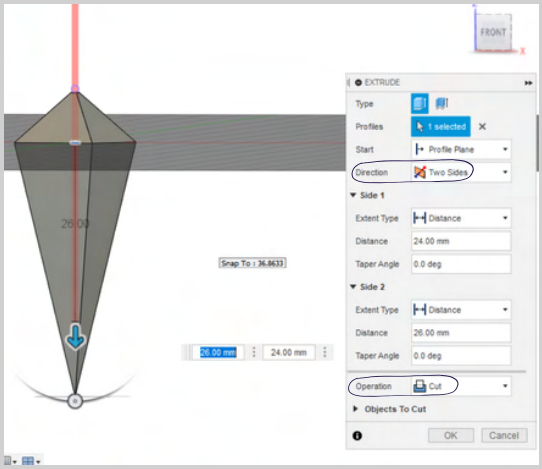
PRINT IT OUT

33

A quick bonus bead idea, I created this octahedron by sketching a square and extruding it with a different *Tapper Angle* on each side. Then I sketched a circle in the middle of the shape and just cut it out.







34

And here's what the final jewellery piece should look like! The middle should be printed with transparent resin or filament to let the light shine through. The other beads would look good spray-painted gold once they've been sanded and primed.

