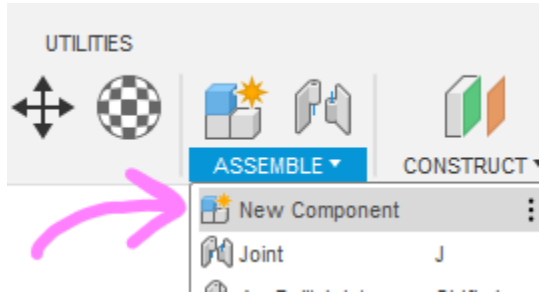Model a pushpin

Begin on the empty new workspace
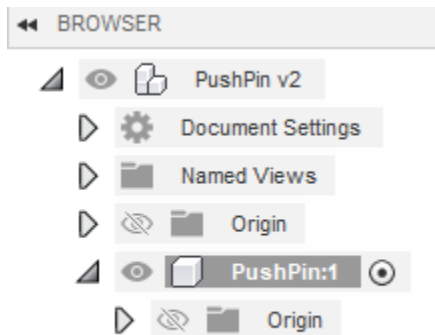First thing first!!! Always start with a new component!

UTILITIES

ASSEMBLE ▾   CONSTRUCT ▾
New Component
Joint   J

Rename the component to PushPin, then click OK.

NEW COMPONENT

Type	Standard
External	○
Internal	●
Name	PushPin
From Bodies	☐
Parent	▶ 1 selected ✕
Activate	☑

OK   Cancel
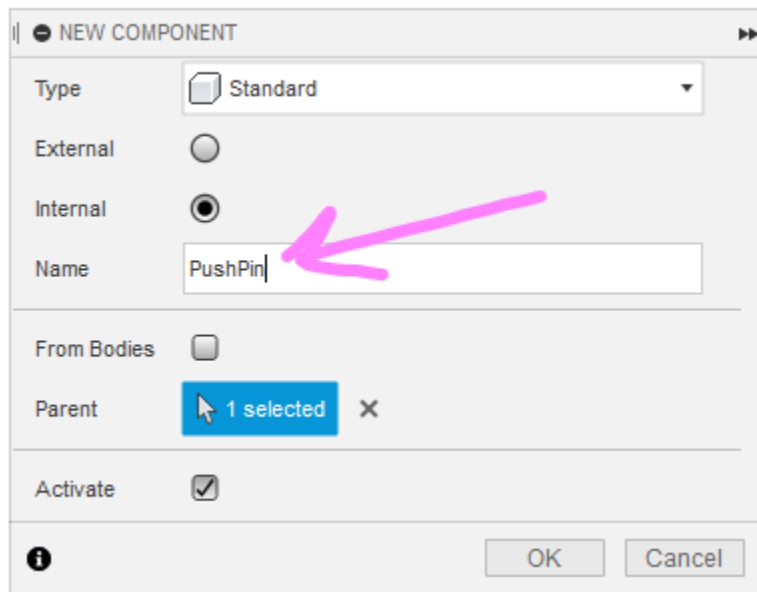
Our pushpin component is now located in the browser

◀◀ BROWSER

PushPin v2
  ▷ ⚙ Document Settings
  ▷ 🖼 Named Views
  ▷ 👁 🖼 Origin
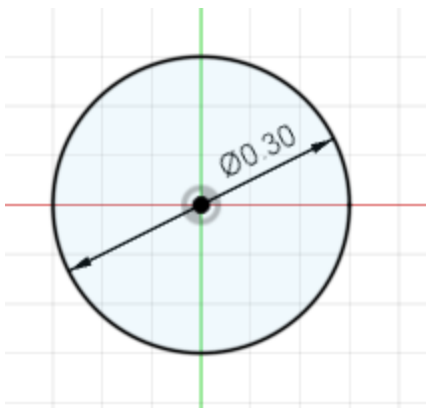 👁 □ PushPin:1 ⊙
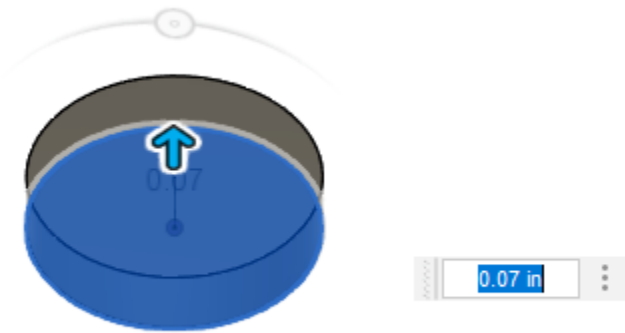   ▷ 👁 🖼 Origin

Create a new sketch

SOLID
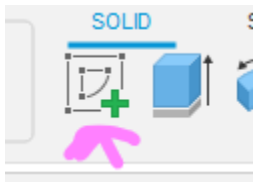
The view changes to "sketch" mode
Draw a circle in the sketch (representing the base of our pushpin). Give the circle a size of 0.3". The circle should be centered at the origin.

Press the "e" key to extrude. Notice that this automatically ends "sketch" mode and returns to the original workspace view. The circle can now be extruded. Extrude it by 0.07"

Lets draw a new sketch on the newly extruded circle surface. Click new sketch and select that surface face fr our sketch surface
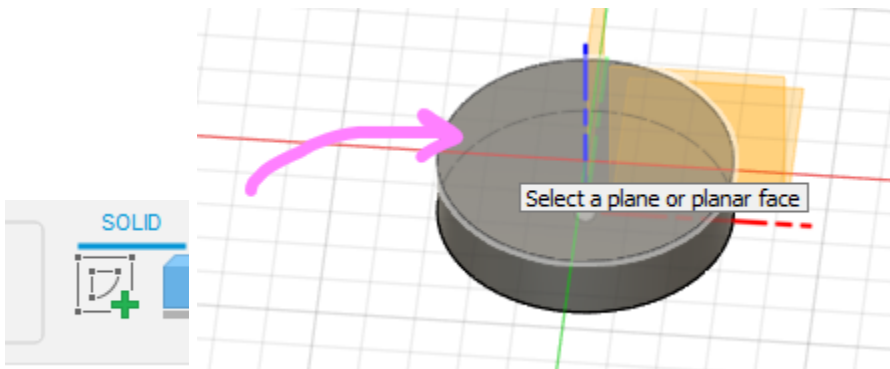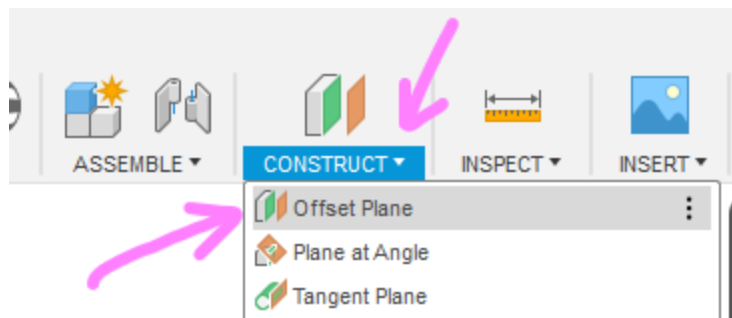

We are now transported into sketch mode again.
Draw a circle with radius 0.17" centered at the origin
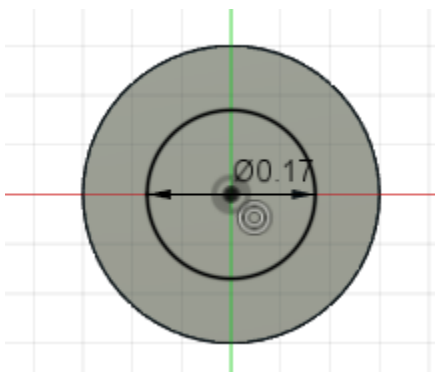

Click Finish Sketch to exit sketch mode
Next, we must create an Offset Plane (a plane that is offset from any particular surface by a given amount)
Click the Construct dropdown menu from above and choose the Offset Plane option

Next, we are asked to select a surface from which our plane will be offset from. Select the top of the cylinder surface first.

Then in the input box that appears, give the offset a value of 0.29". Click OK. Now our new plane (in yellow) is located at 0.29" away from the top of the surface. We will use this surface to draw our next circle.
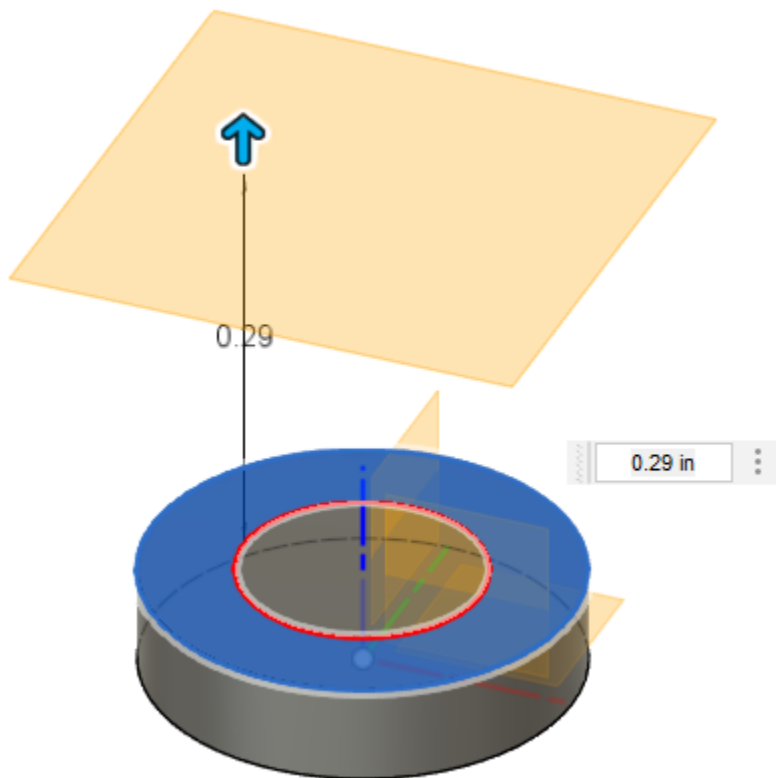
Create a new sketch

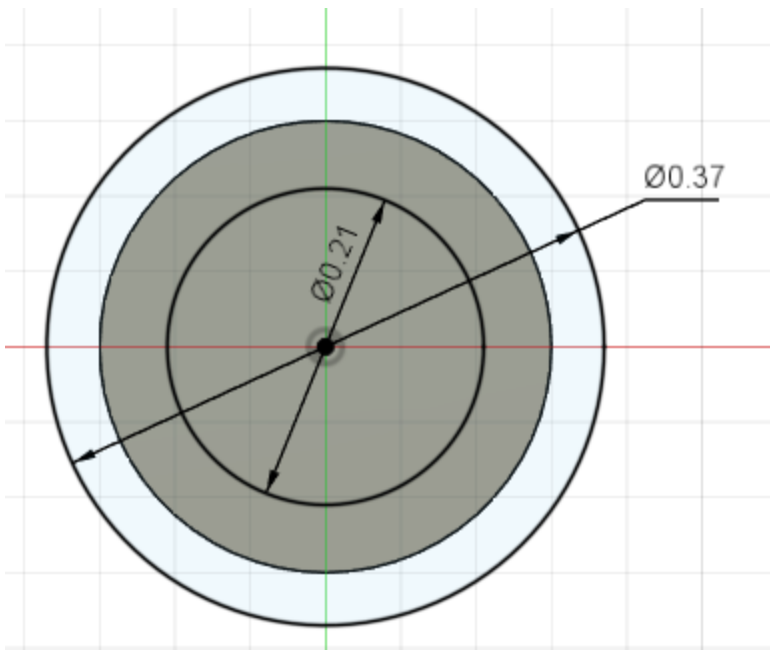Select this newly constructed offset plane as our surface to sketch on.
We are now in Sketch mode.
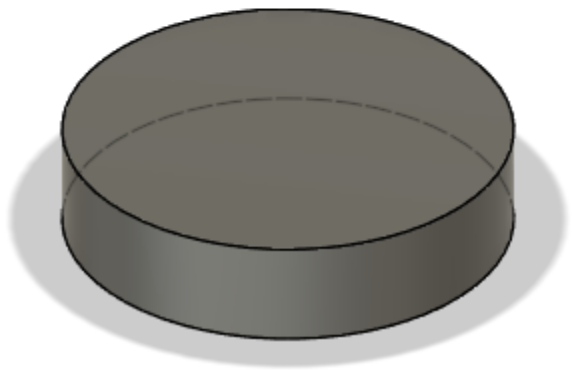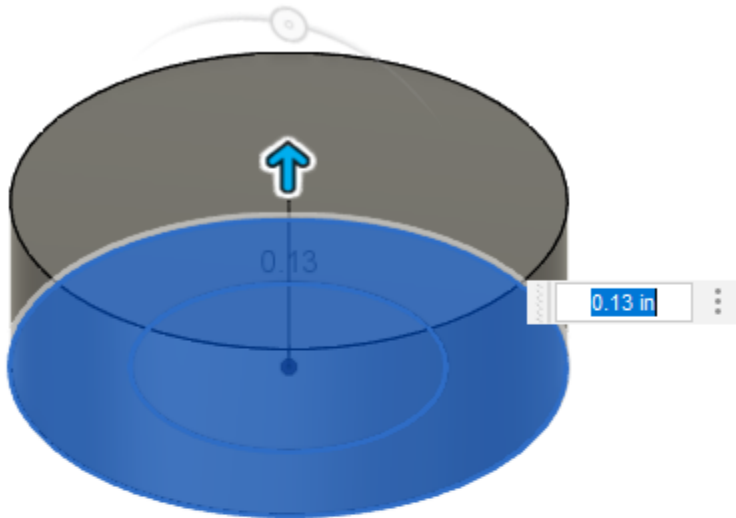We will draw two circles on this plane.
One circle will be 0.37"
One circle will be 0.21"
They should both be centered at the origin

Click Finish Sketch to complete the sketch and return to the default workspace
Click the "e" key to extrude. Select both circles to extrude a solid cylinder by 0.13"


Next. We will create a midplane (a plane located exactly between two surfaces)
From the Construct menu dropdown, select Midplane
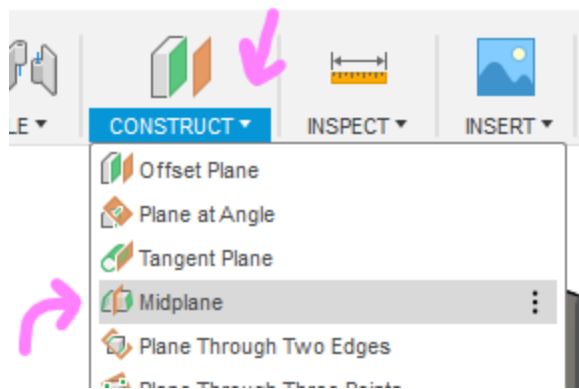
Next, click the two surfaces that we will use as our reference planes

The two surfaces are the top of the lower cylinder and the bottom of the upper cylinder. Notice the new plane (in yellow) appears between these two surfaces. Click OK to complete the Midplane.
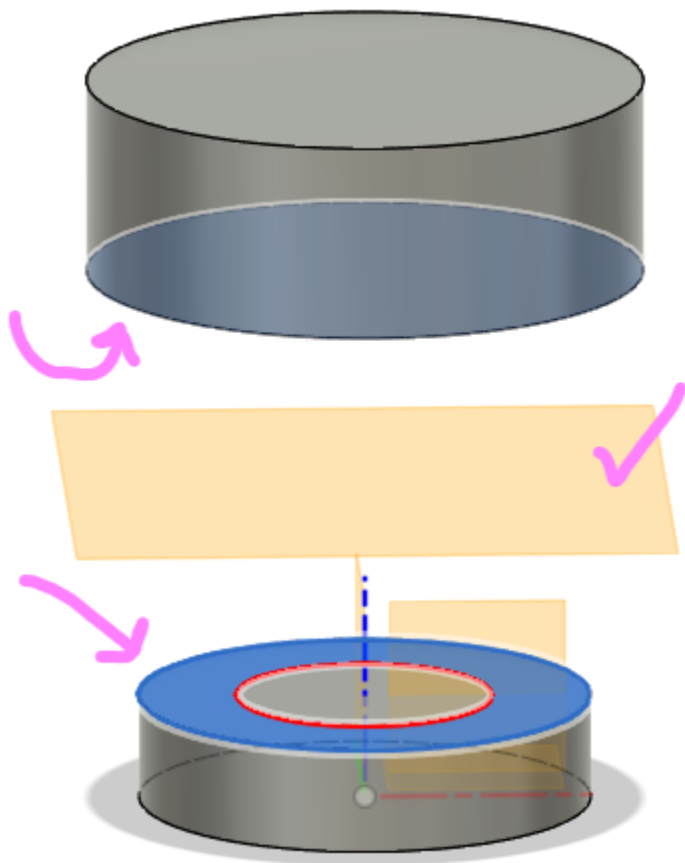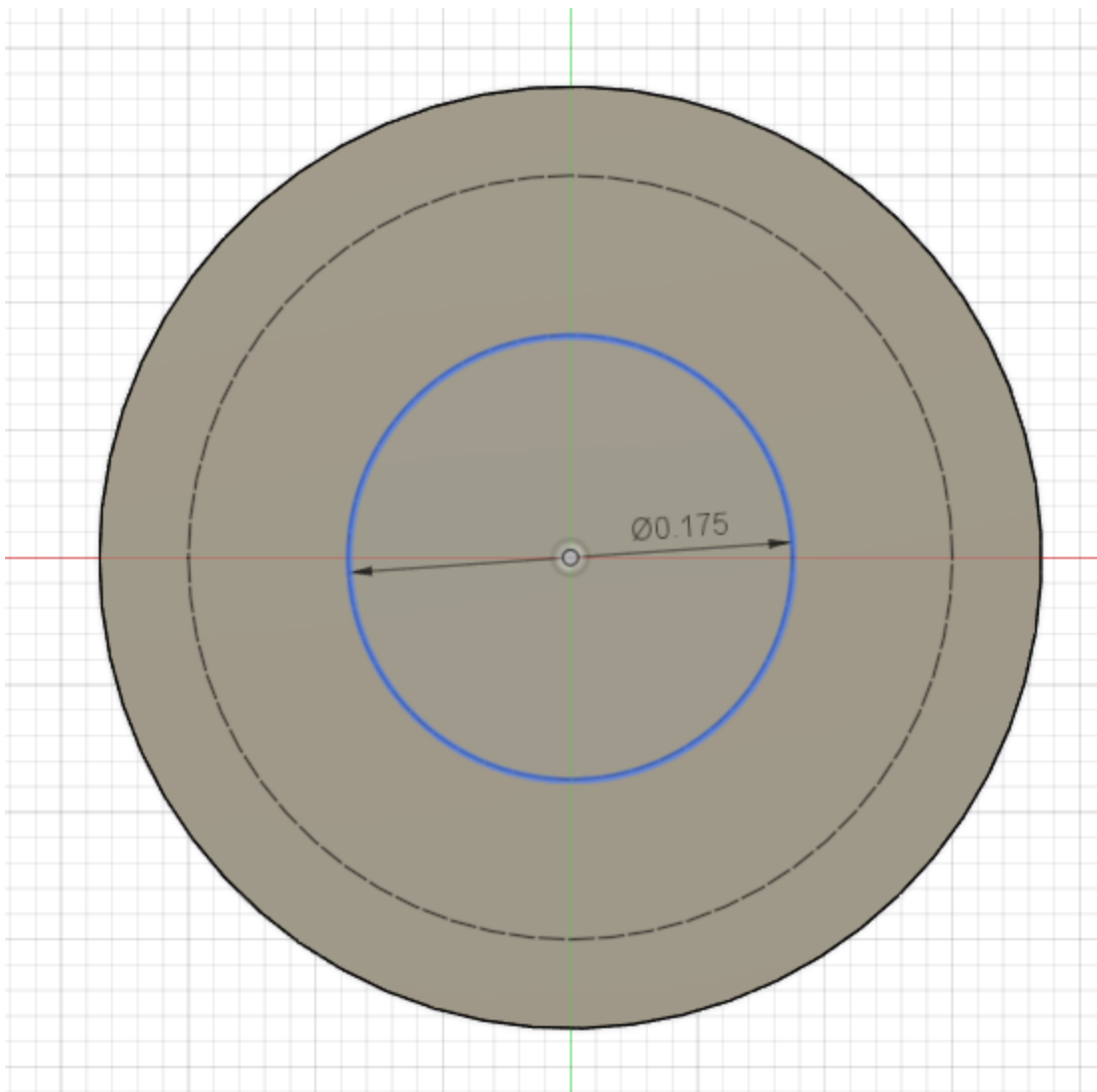

We will use this new plane to add curvature to our pushpin handle.

Lets draw a circle on the new midplane. Start by creating a new Sketch

And select the new midplane as out sketch surface.

Our view transition into Sketch mode

Lets draw the circle centered around the origin with a size of 0.175"

Click Finish Sketch to complete it.

We are now returned to the default workspace.

Lets loft a surface between the three curves that make up the handle.

If your curves are not visible, then that means that the sketch is hidden (sketches are automatically hidden from view after doing things like extruding the curve)
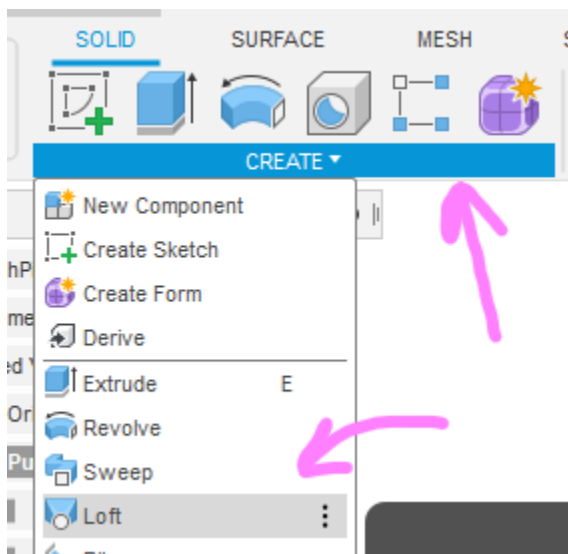
In the browser hierarchy, enable the correct sketch layers
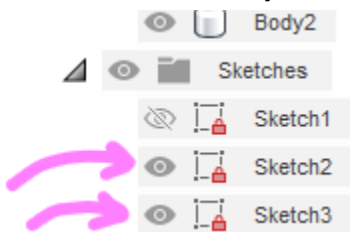
Your view should look like this with three visible curves to make the stem of the handle
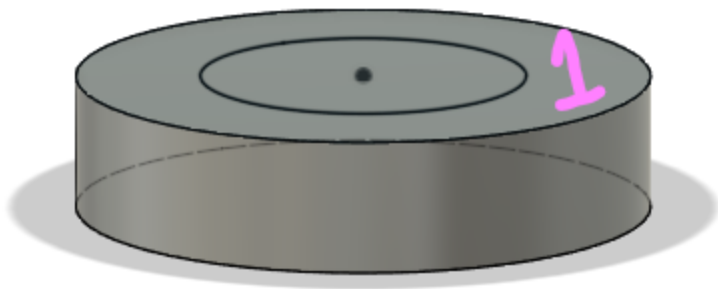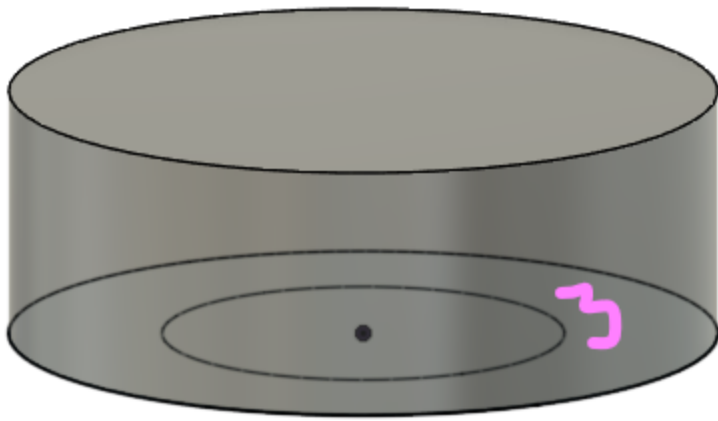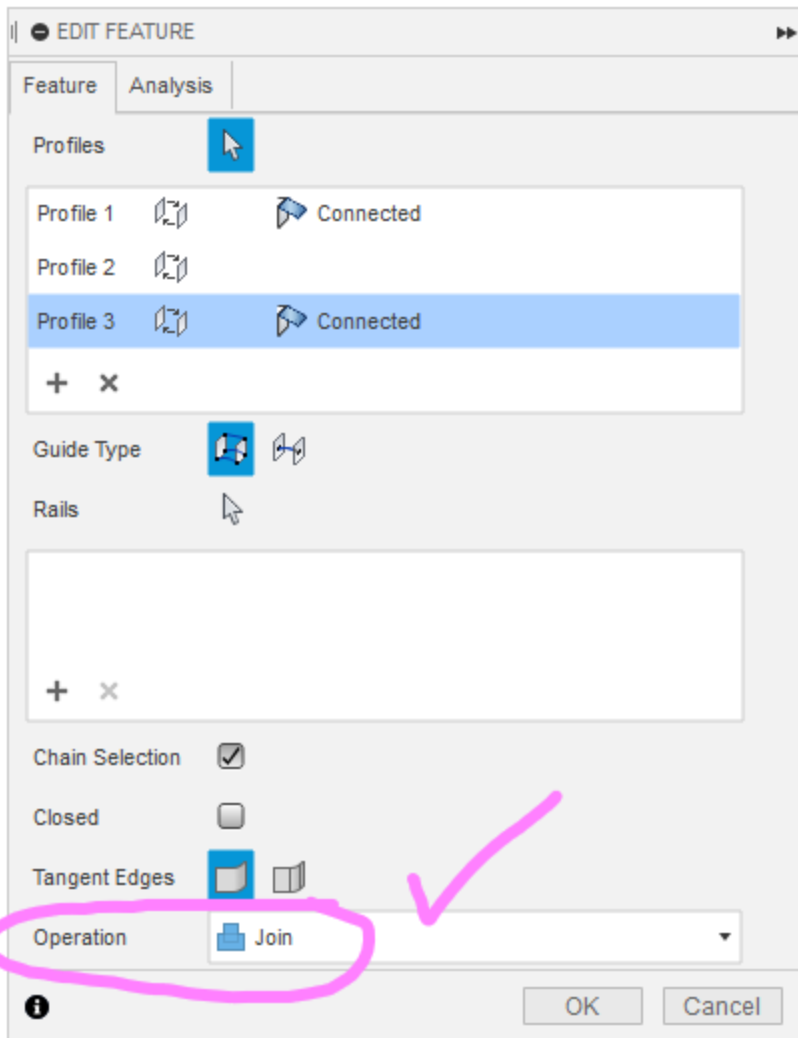
We will now Loft (connect them with a surface/solid) between the three curves.
From the Create menu above, select the Loft option

Fusion asks us to select the curves profile that we want to Loft. **Select them in order** from bottom to top.
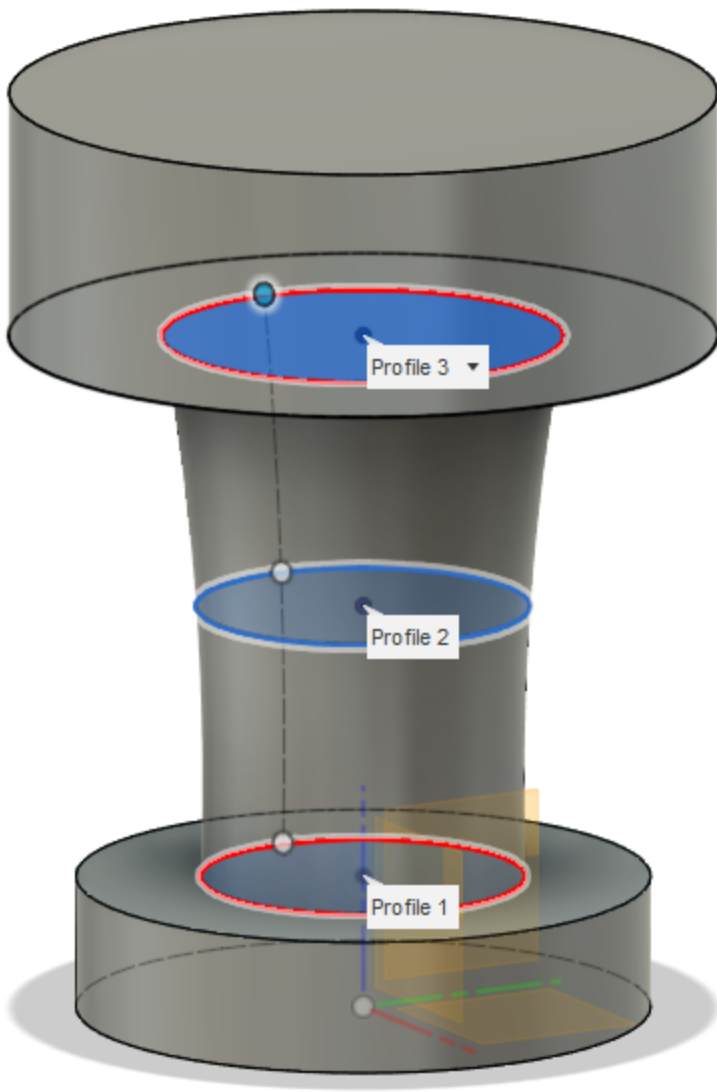
We want the newly lofted surface to Join with the existing geometry, therefore the Join operation should be selected
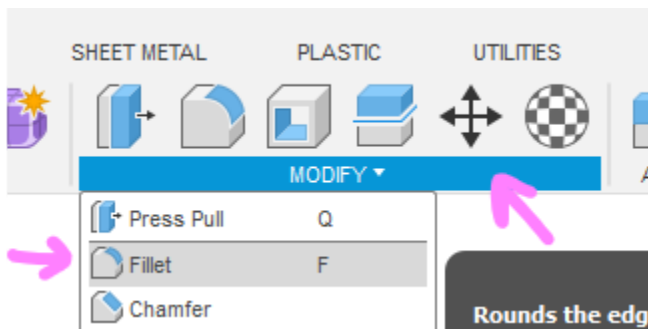
Click OK to confirm the Loft action.

Now we have the stem of our PushPin.

Lets round off the edges.

Select the Fillet option from the Modify menu above

Next select the edge ring of the upper cylinder to give it curvature

Click the arrow to pull the fillet radius and to see the intended curvature. Looking at the pushpin model in your hand, try to match the curvature to the best of your abilities.

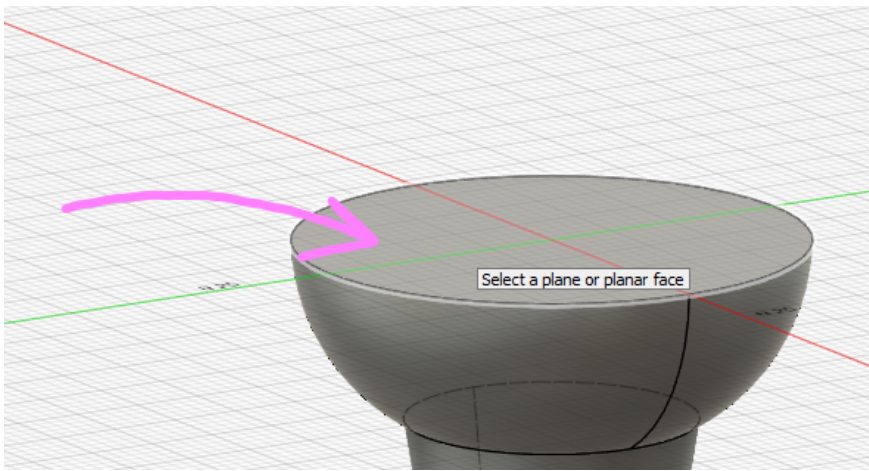When ready, click OK to apply the fillet.
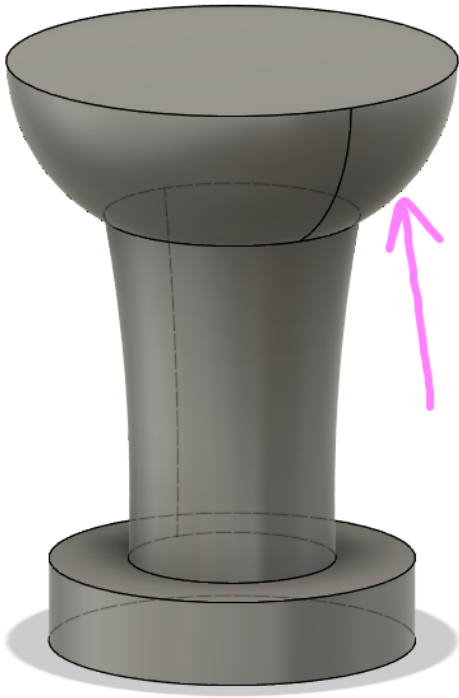
Great so far!
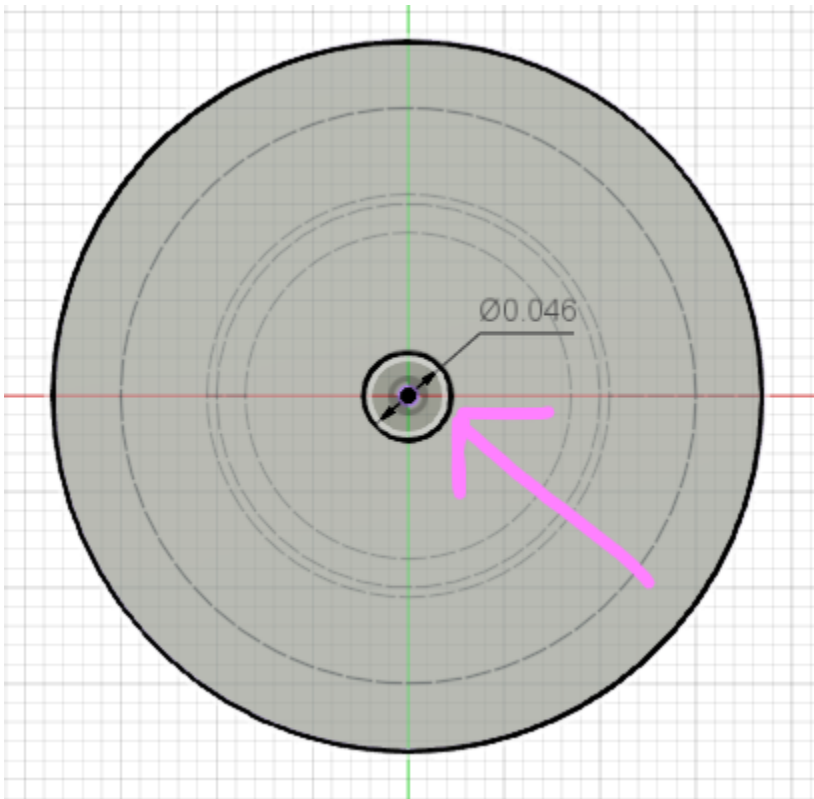Now let's repeat the same Fillet action on the bottom cylinder edge

The body of our pushpin is now complete
Lets make the needle part.
Create a sketch on the top surface of the pushpin
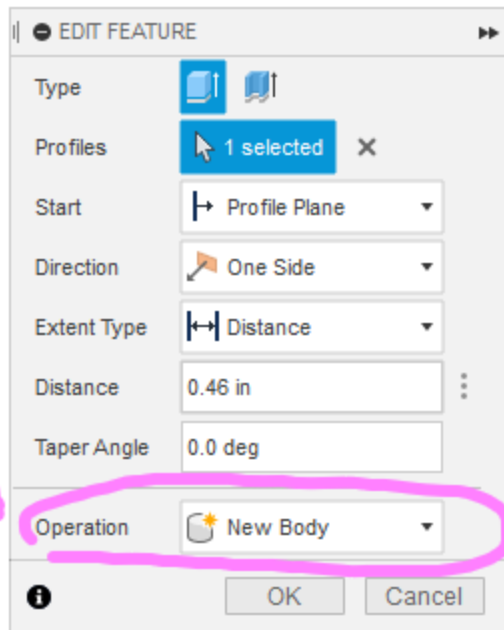
We are transported into Sketch mode
Lets draw a circle with a radius of 0.046 centered at the origin

Click Finish Sketch.
Next we will extrude the new small circle to make the needle.
Press the "e" key and select the circle to extrude it. Give it an extrusion distance of 0.46". Note that this time, instead of the Join operation we are going to use the New Body operation. Be sure to switch it to New Body before clicking OK

EDIT FEATURE

Type	
Profiles	1 selected ✕
Start	⊢ Profile Plane ▾
Direction	One Side ▾
Extent Type	↔ Distance ▾
Distance	0.46 in
Taper Angle	0.0 deg
Operation	New Body ▾

OK   Cancel

0.46 in

0.46

This created the needle portion as a new body in our hierarchy browser, which will help for rendering purposes

Origin
Bodies
  Body1
  Body3
Sketches
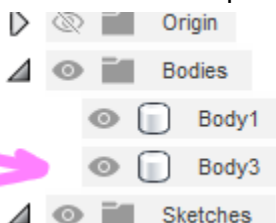
Next lets give the needle some sharpness by using the Chamfer tool
Select the Chamfer option from the Modify menu above
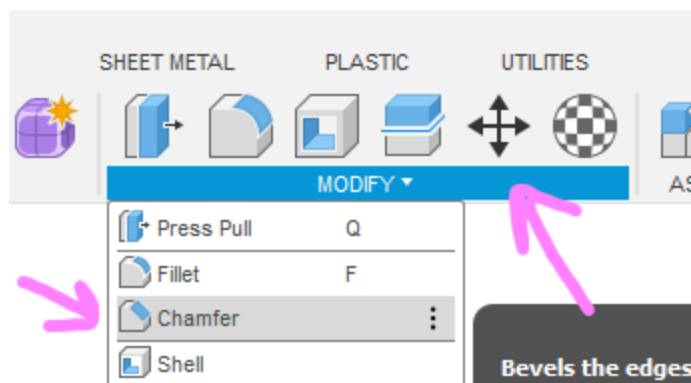
In the chamfer dialog box, change the type from EquiDistance to "Two Edges"
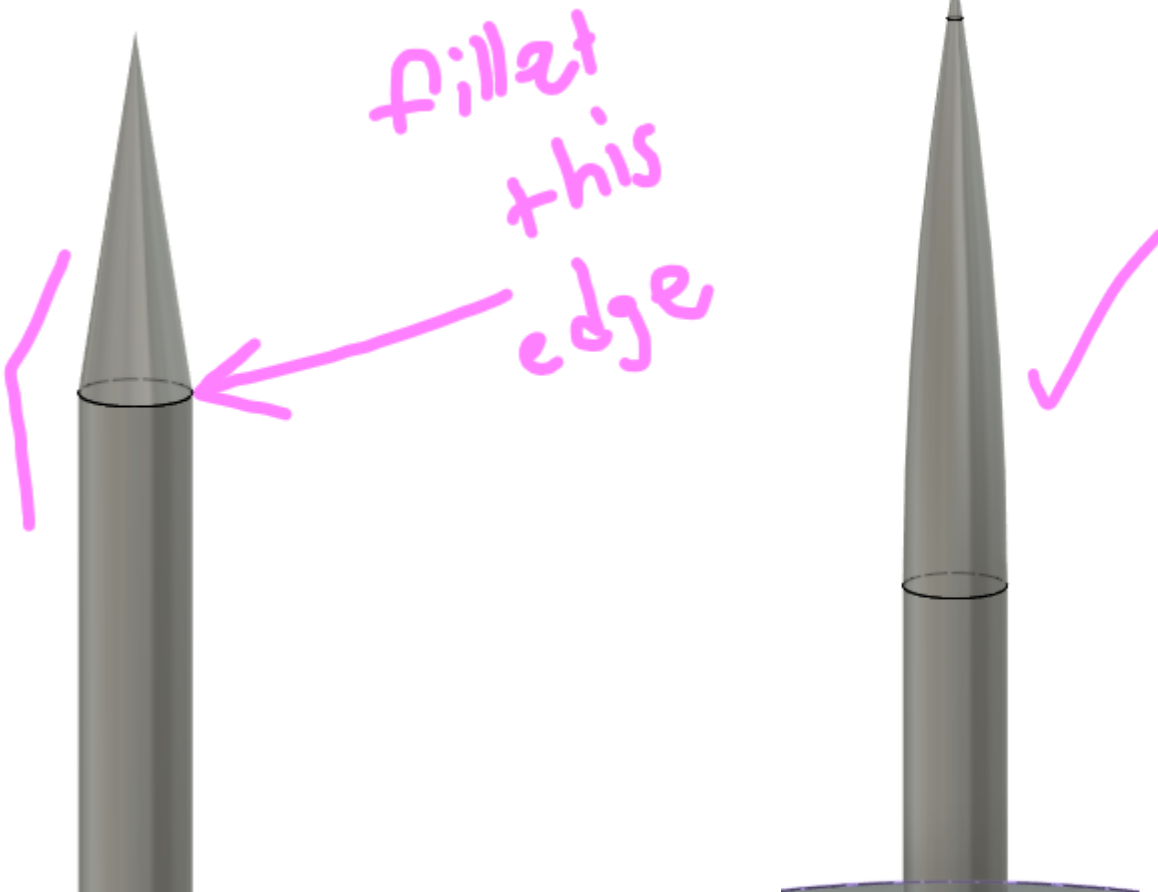This allows us to create a sharper point using two arrow handles rather than one.
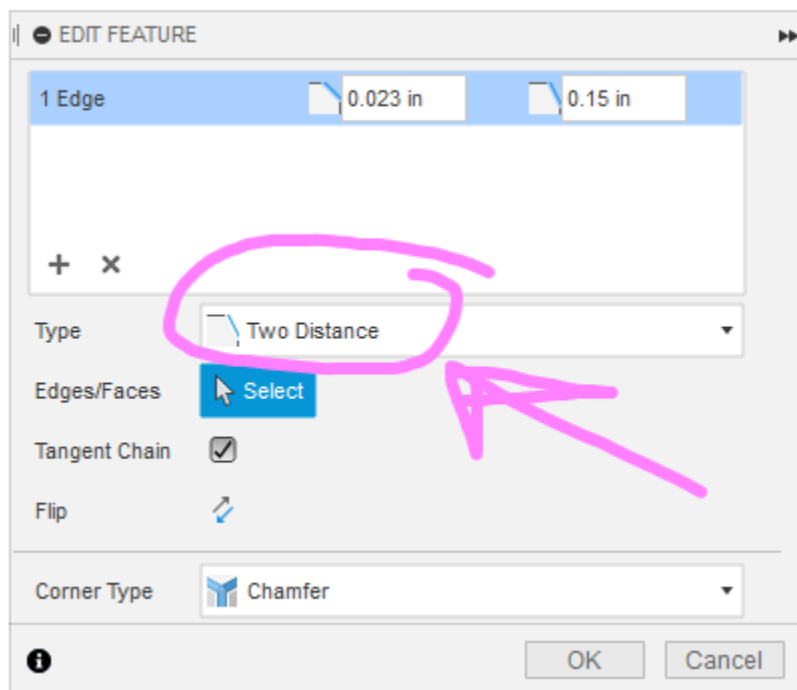
Give the point a nice and sharp chamfer by pulling the arrow handles. You will have to use your best judgment for this.
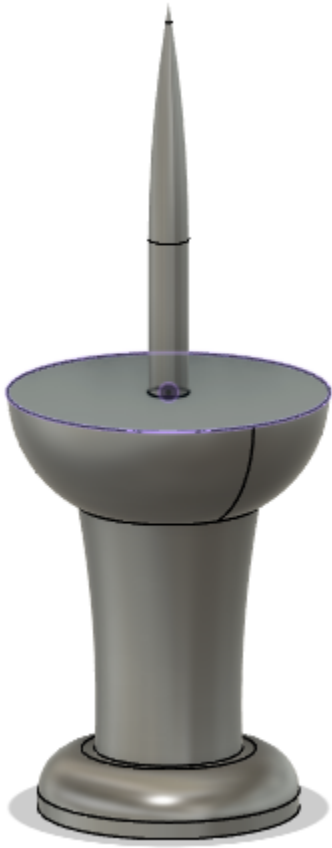
0.023 in
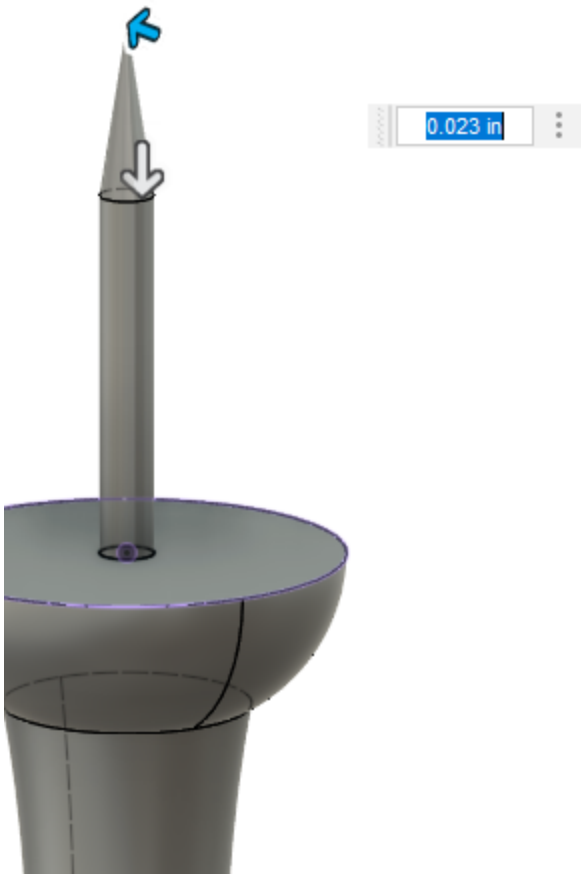
Click ok when ready.
Finally, lets give the harde needle edge some roundness using the Fillet tool

fillet
this
edge

Our PushPin is now complete

In class we will give it some color and render it out. We will also construct an architectural drawing from it.