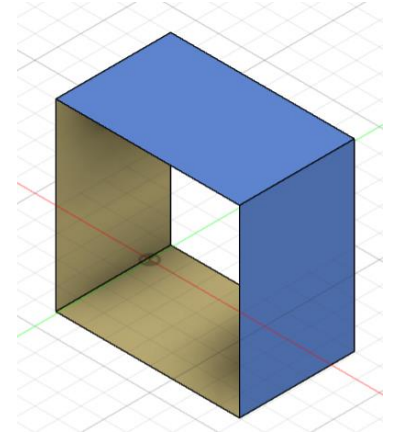
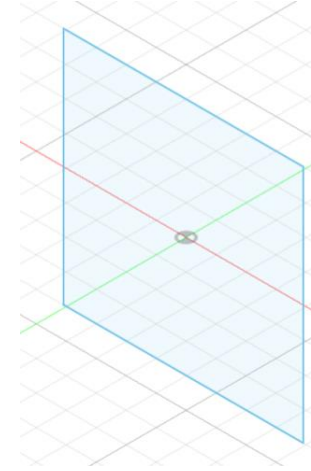
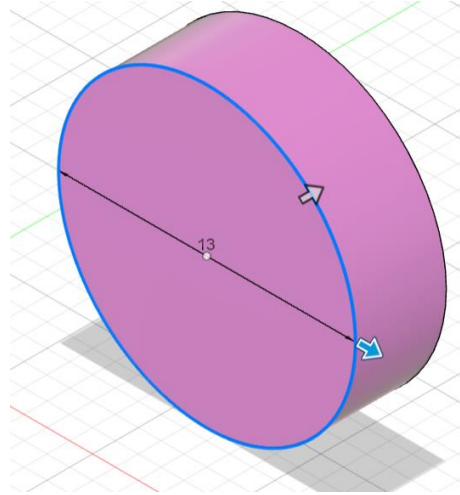
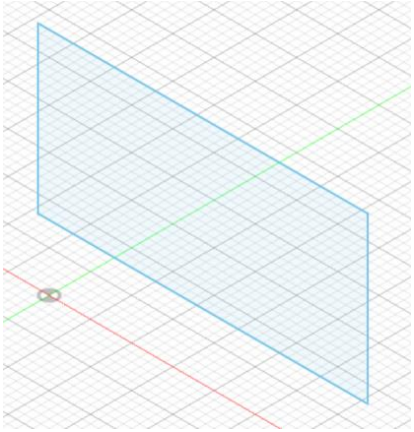


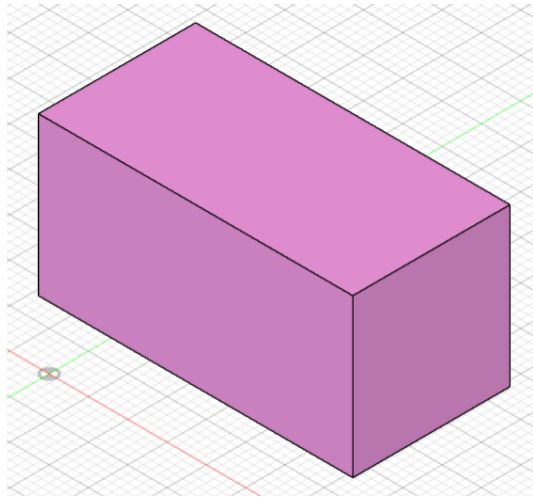
# **MS 101 Fusion 360 Solid Modelling & Modifications**

**(Autodesk: Product documentation)**

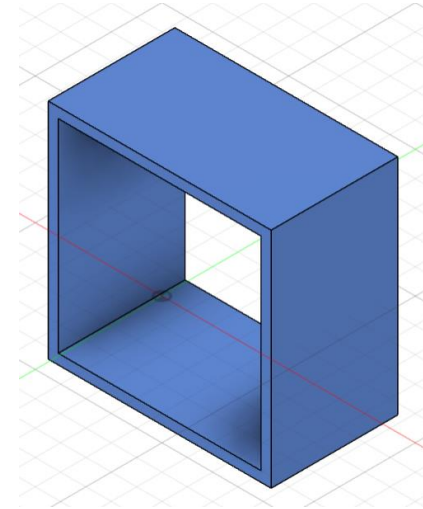
# Solid Modelling



**Solid from primitives**



**Solids from sketches**



**Solid from surfaces**

# Solid Modelling

A solid body from a closed sketch profile, open sketch curve, or planar face in Fusion can be created using the tools in the **Design** workspace, in the **Solid > Create** panel.

Tools to create a solid body from a sketch:

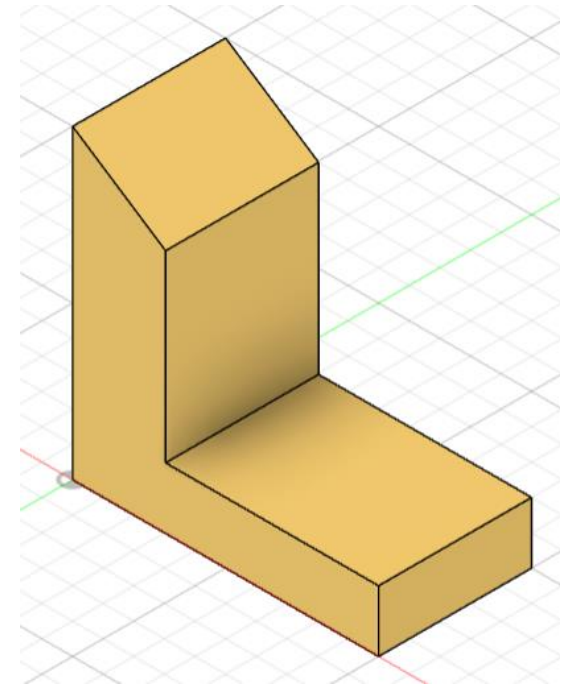
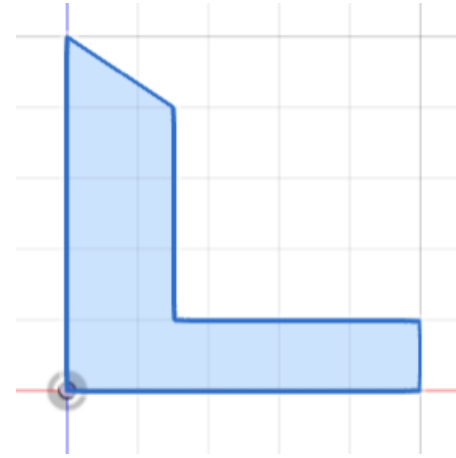
- Extrude
- Revolve
- Sweep
- Loft
- Rib
- Web
- Emboss

# Extrude a solid body

On the toolbar, click **Solid > Create > Extrude** .

The **Extrude** dialog displays.

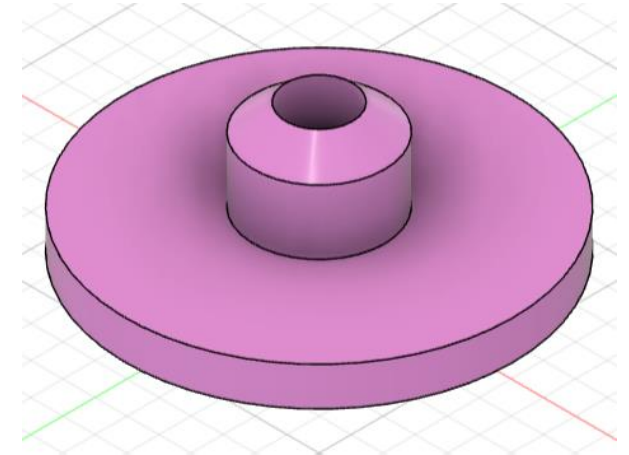
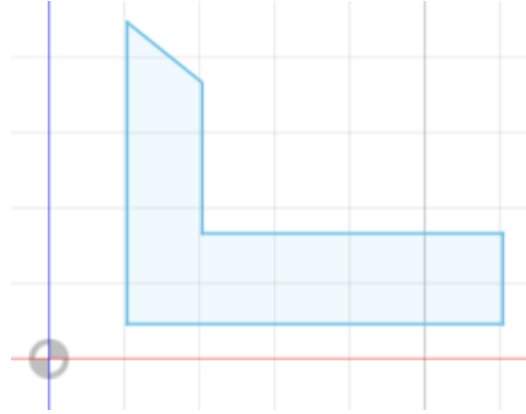
1. In the canvas, select one or more coplanar sketch profiles or planar faces to extrude.
2. In the dialog, select an extrude **Type**:
3. Select a **Start** setting, then adjust its associated settings:
4. Select a **Direction** setting, then adjust its associated settings:
5. Select an **Extent Type**, then adjust its associated settings:
6. Specify the **Taper Angle** to taper the extrusion.
7. Select an **Operation**, and adjust its associated settings



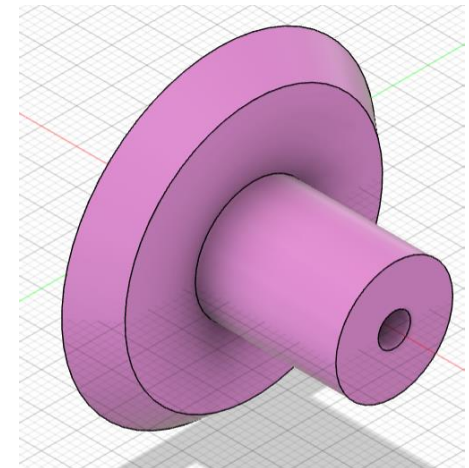
# Revolve a solid body

On the toolbar, click **Solid > Create > Revolve**.

- The **Revolve** dialog displays.
- In the canvas, select a **coplanar sketch profile** or face to revolve.
- In the canvas, select a **linear sketch curve, edge, cylindrical face, or axis** to revolve around.
  - **Partial:** Revolves the profile around the axis to an angle value that you specify.
  - **Full:** Revolves the profile 360 degrees around the axis.



Revolve around vertical axis



Revolve around horizontal axis

# Sweep a solid body

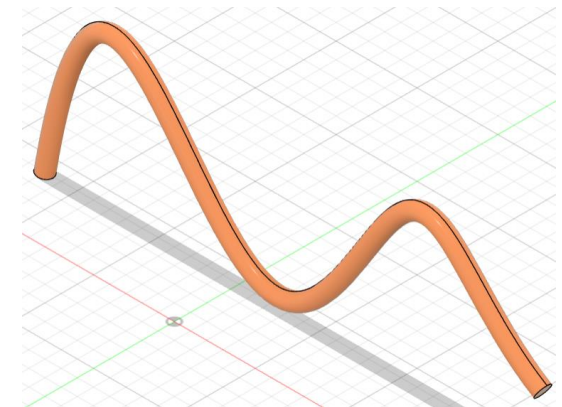
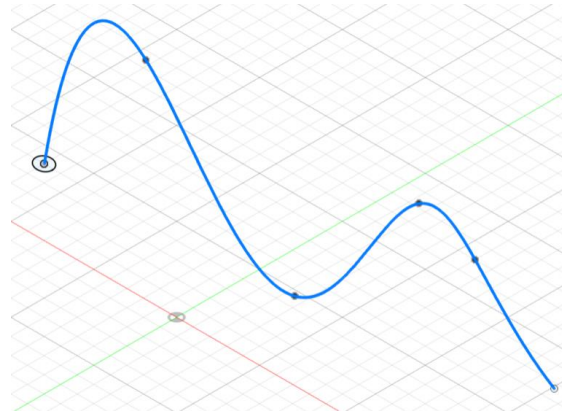
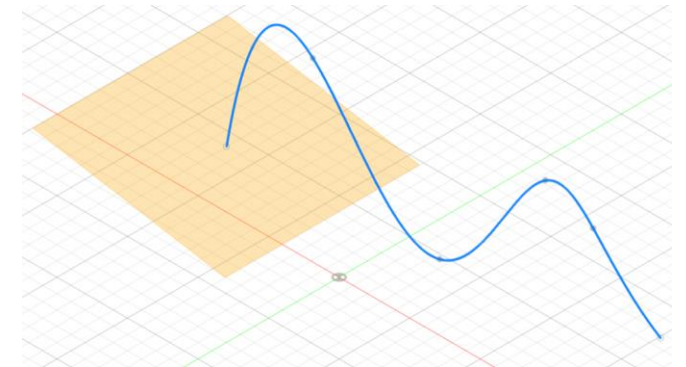
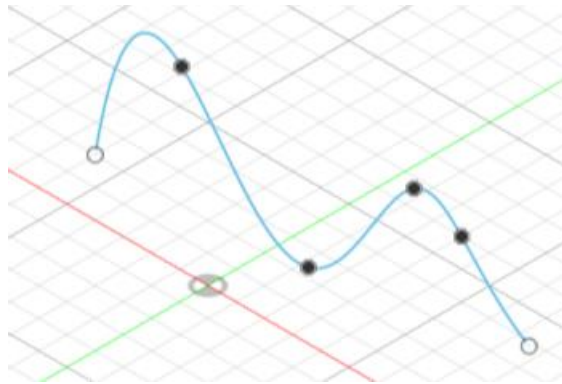
Sweep a profile along a path

1. On the toolbar, click **Solid > Create > Sweep**.

The **Sweep** dialog displays.

- Type
- Profile
- Path
- Distance
- Taper angle
- Twister angle
- Orientation
- operation

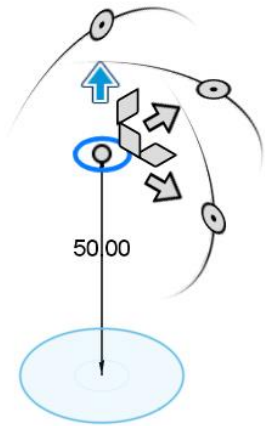
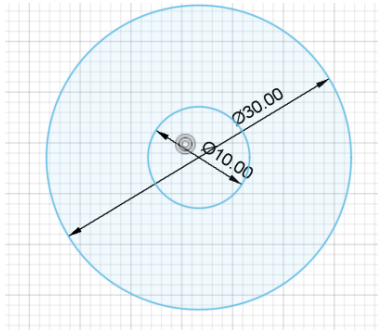
- Sketch a smooth line using 'fit point spline' in the front plane as the **path** for the sweep.
- 'Construct' a plane, 'plane along path'.
- Sketch the **sweep profile** on the above plane.





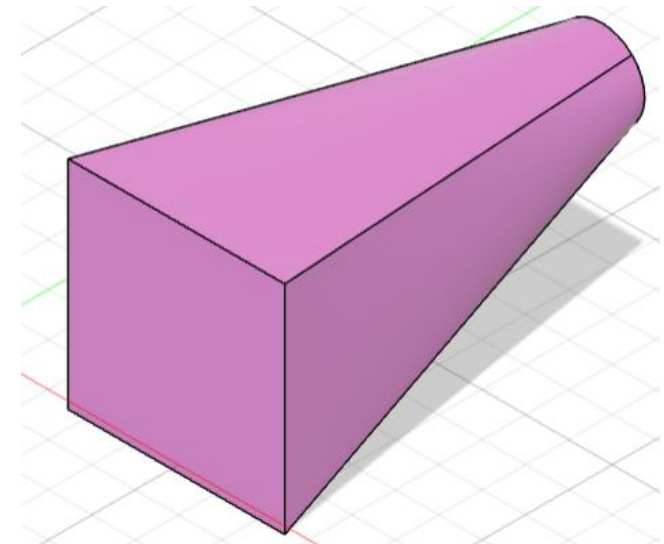
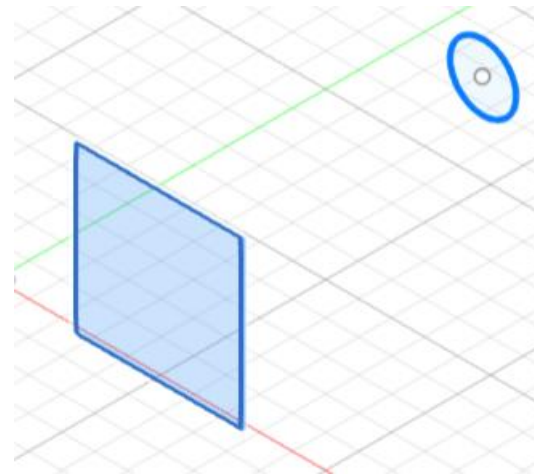
# Loft a solid body

- Sketch **1<sup>st</sup> 2D profile** in a plane.
- Sketch the **2<sup>nd</sup> 2D profile** on an **offset plane** of the first profile plane
- Or both the profiles can be in a single plane. **Remove constraints**
- **One of the profiles** can be moved away to **make the 2<sup>nd</sup> profile**



1. On the toolbar, click **Solid > Create > Loft** .

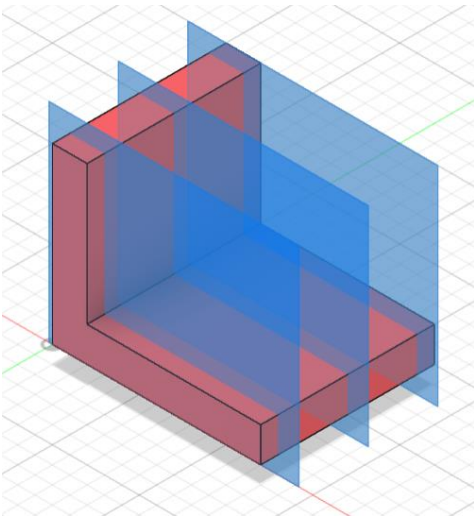
- The Loft dialog displays.
- Select two or more profiles
- Guide type
- Tangent edges
- operation



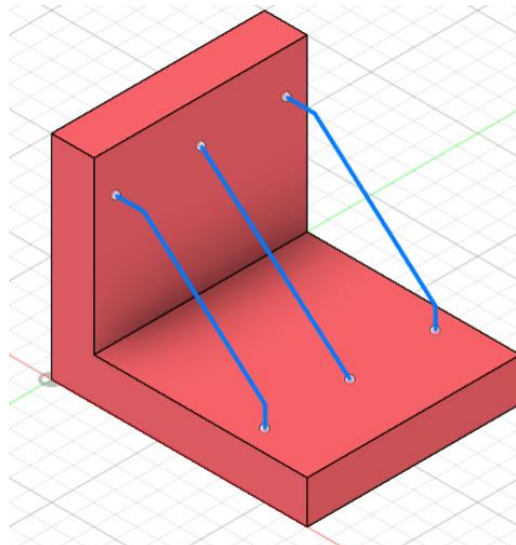
# Create a rib

1. On the toolbar, click **Solid > Create > Rib**.
2. In the canvas, select an open **sketch profile** to use as the Profile.
3. In the dialog, select a Thickness Direction:
4. Specify the **Thickness** value to extrude the rib, perpendicular to the sketch plane:
5. Select an Extent Type (distance), then adjust its associated settings:

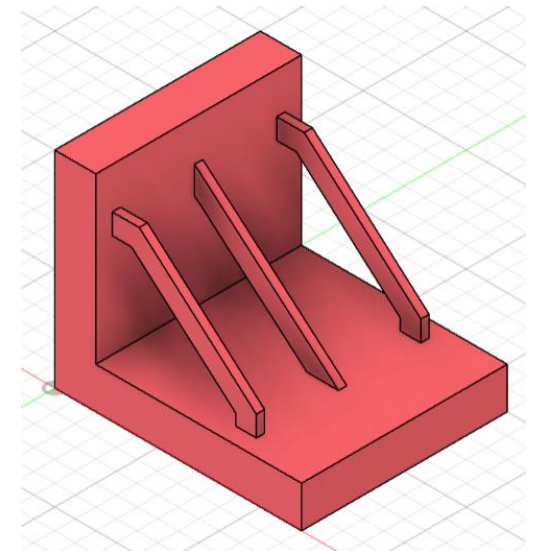
Construct offset planes



Construct 2D profiles



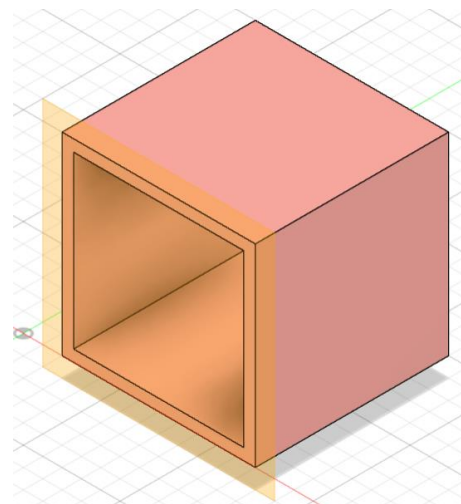
Create rib



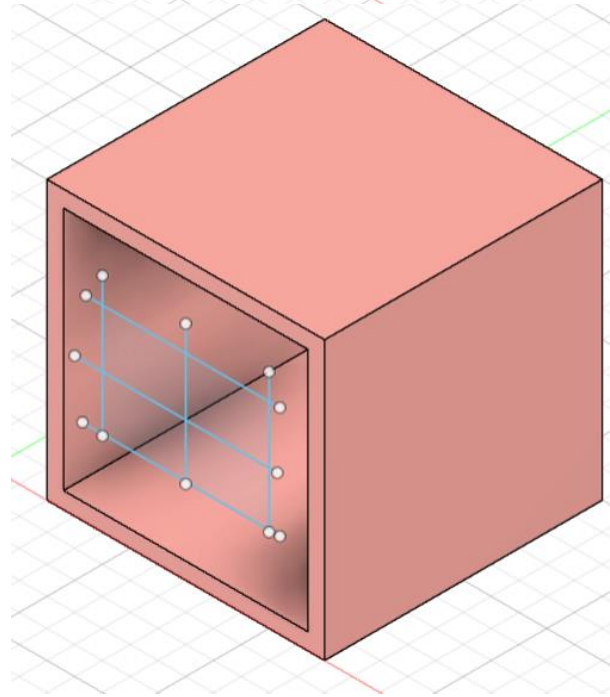


# Create a web

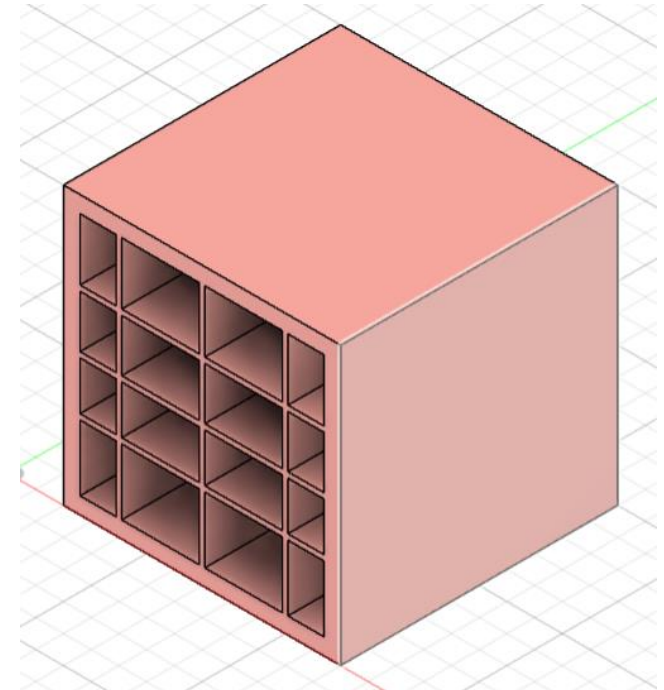
1. On the toolbar, click **Solid > Create > Web**.
2. In the canvas, select an open **sketch profile** to use as the **Profile**.
3. In the dialog, select a **Thickness Direction** setting:
4. Select an **Extent Type** setting, then adjust its associated settings:



Create solid and offset plane



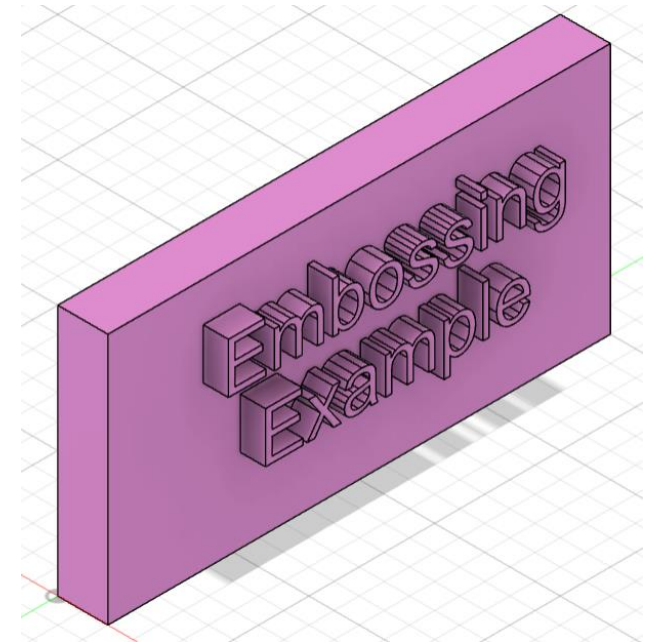
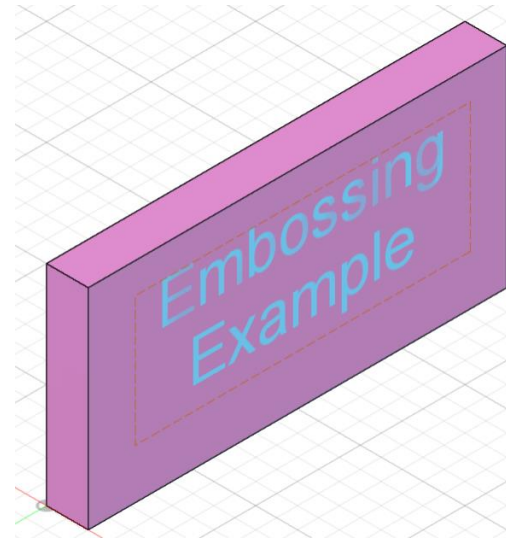
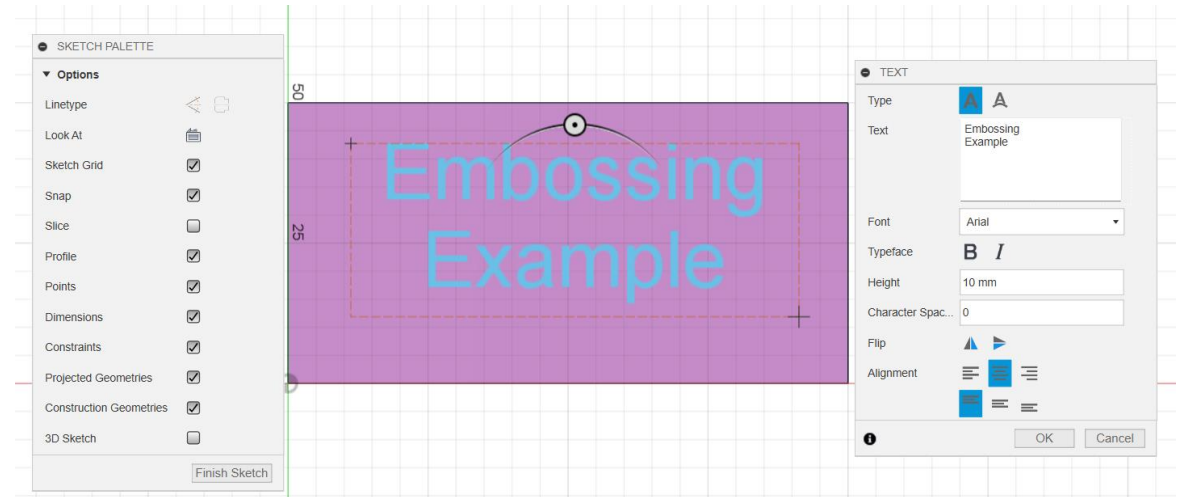
Create profile



Create web & select thickness

# Emboss a solid body

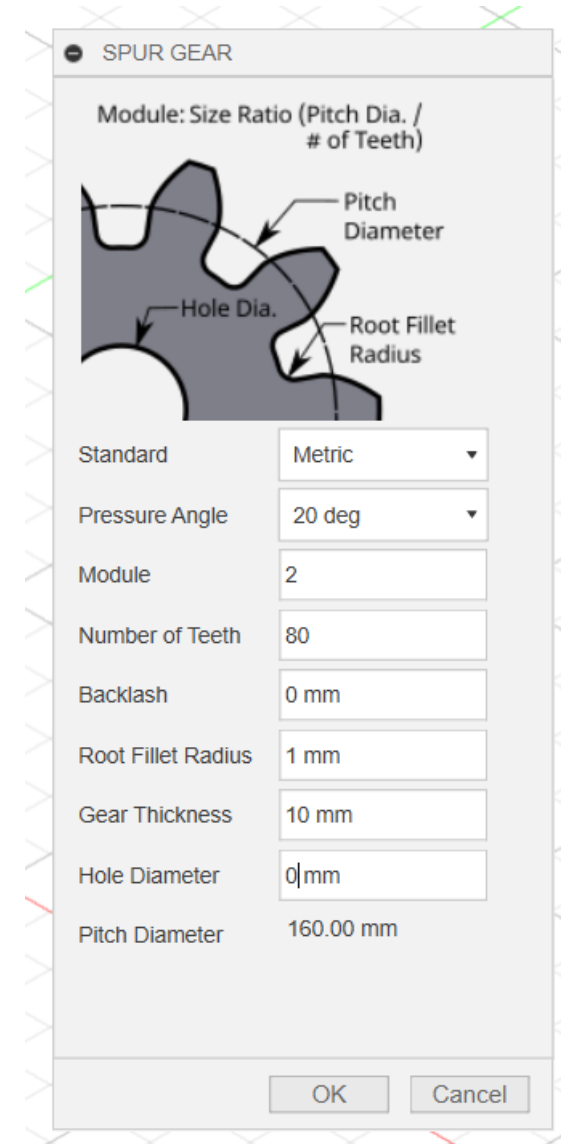
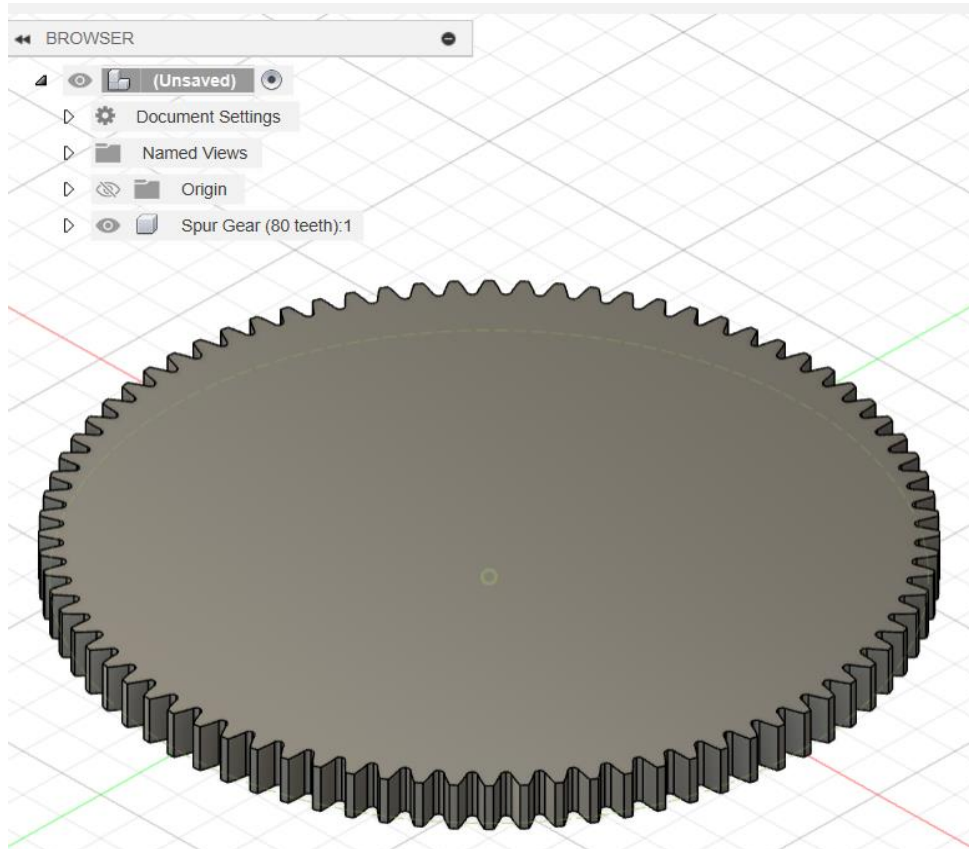
1. On the face of a body select **sketch**
2. Create text; create **text frame**; type texts, select fonts, height, alignment etc.
3. On the toolbar, click **Solid > Create > Emboss**.
2. In the canvas, select the **Sketch Profiles** you want to emboss.  
You can select any **2D sketch profile**, including text.
3. On a **solid body**, select the **Faces** you want to emboss.
4. In the dialog, select the Effect:
  - **Emboss** : Add material.
  - **Deboss** : Remove material.
5. Adjust the **Depth value** for the emboss feature.



# Spur Gear

## Spur gear creation

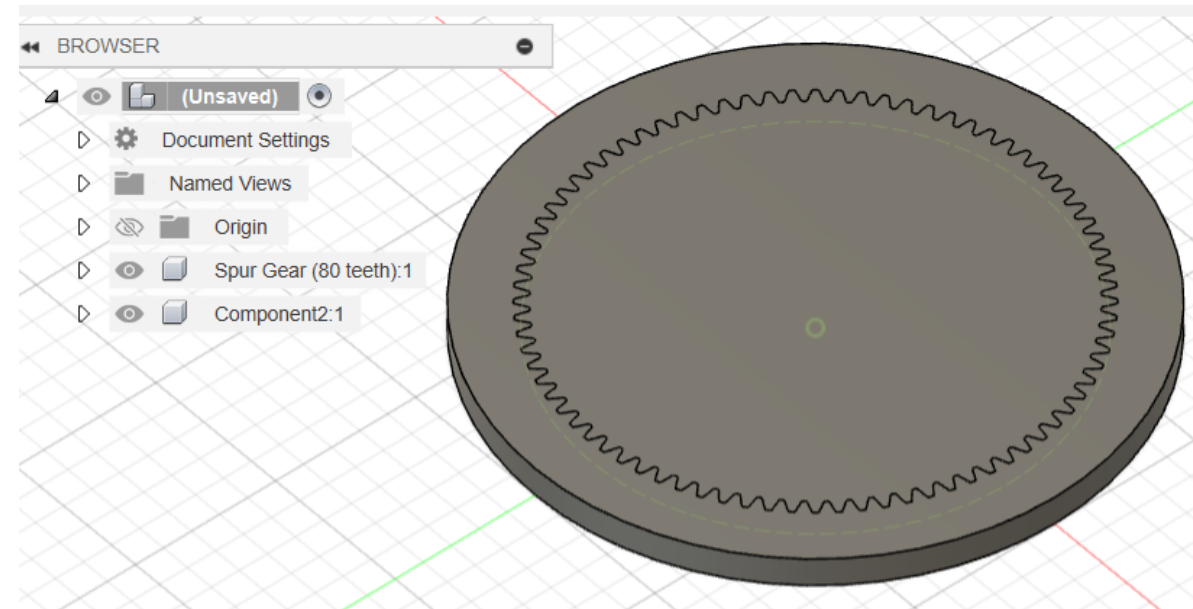
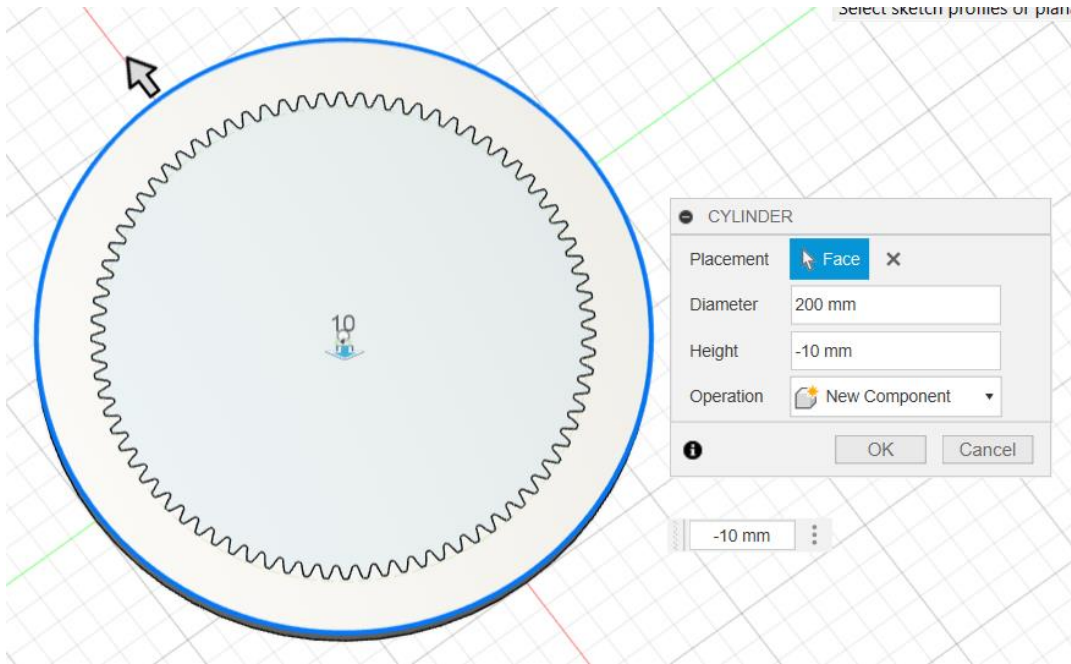
UTILITIES → ADD-INS → Scripts and Add-Ins → SpurGear → run



# Ring Creation

## Ring creation

Select gear-plane ➡ SOLID ➡ CREATE ➡ Cylinder

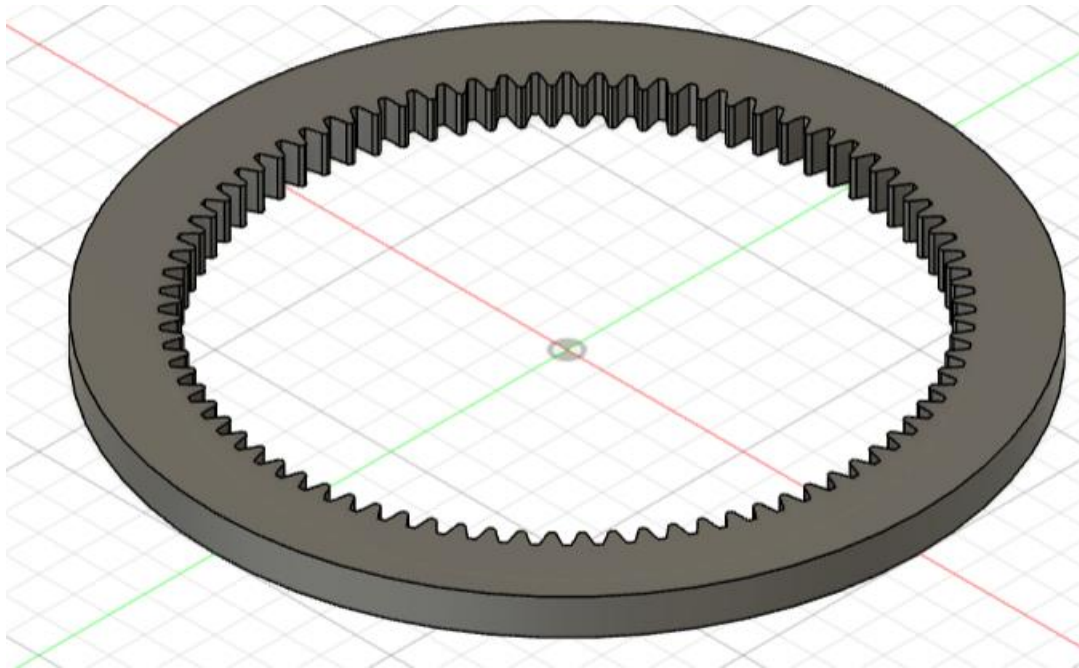




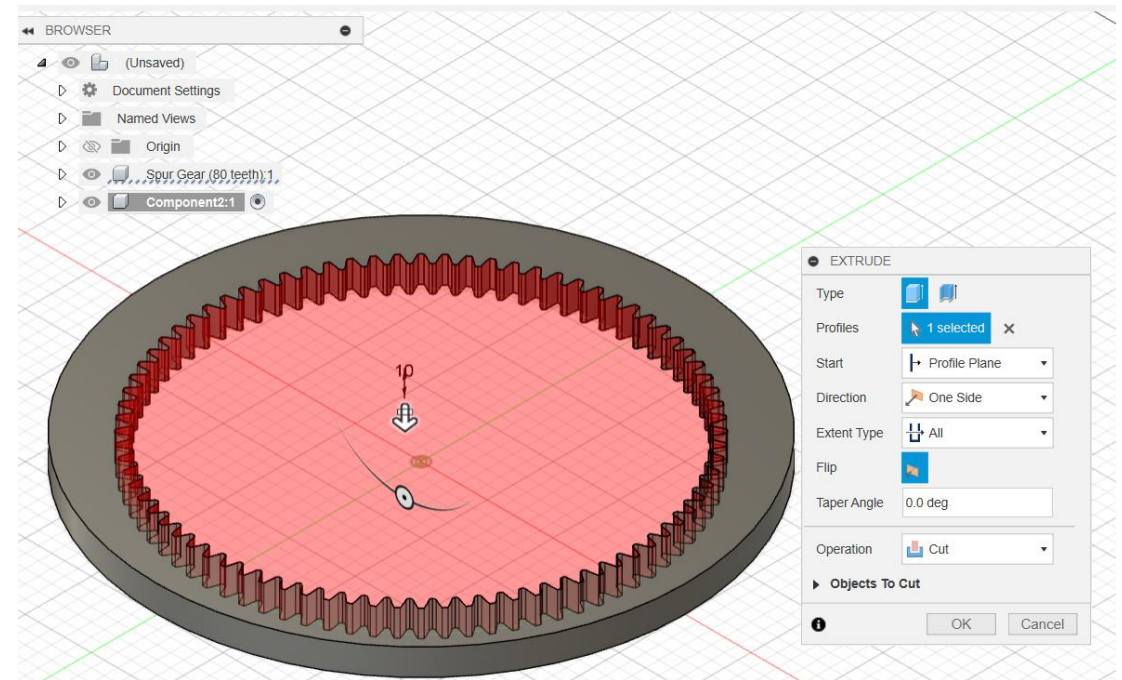
# Ring Creation

## Ring creation

Activate component2:1 → Extrude → Select face of gear as profile → Cut:



Ring Gear (80 teeth)



# Solid Primitives

In the Design workspace,  
in the **Solid >**  
**Create** panel, let to create  
a solid body from a  
primitive shape in Fusion  
360.

Use the following commands to create  
a solid body from a primitive shape:

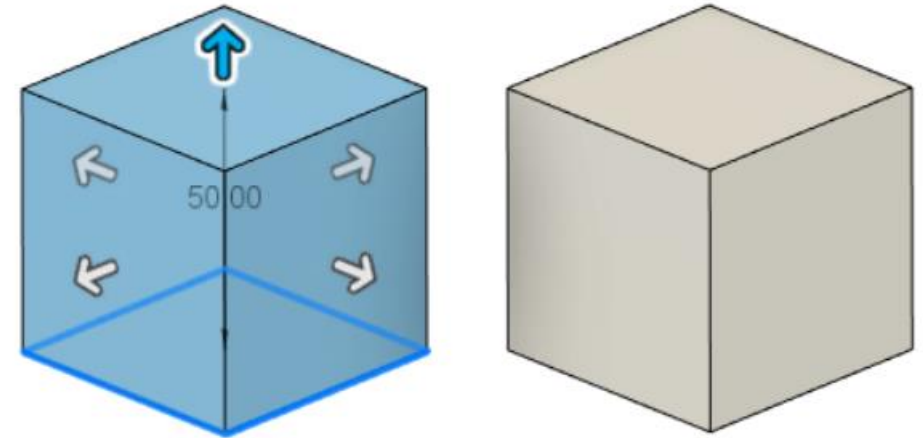
- . **Box**
- . **Cylinder**
- . **Sphere**
- . **Torus**
- . **Coil**
- . **Pipe**



# Box

The **Box** command creates a solid body in the shape of a **primitive box**.

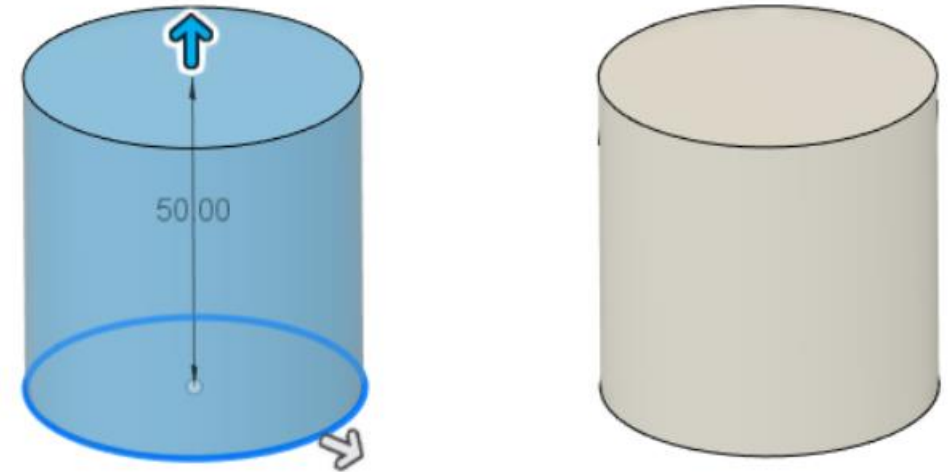
Select a **plane or planar face**, place the first corner, specify the **length and width**, then specify the **height** of the box.



# Cylinder

The **Cylinder** command creates a solid body in the shape of a **primitive cylinder**.

Select a **plane or planar face**, place the **center point**, specify the **diameter**, then specify the **height** of the cylinder.



# Sphere

The **Sphere** command creates a solid body in the shape of a **primitive sphere**.

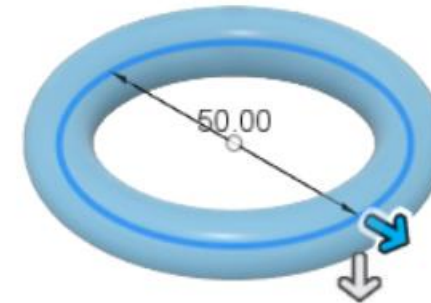
Select a **plane** or **planar face**, place the **center point**, then specify the **diameter** of the sphere.



# Torus

The **Torus** command creates a solid body in the shape of a **primitive torus**.

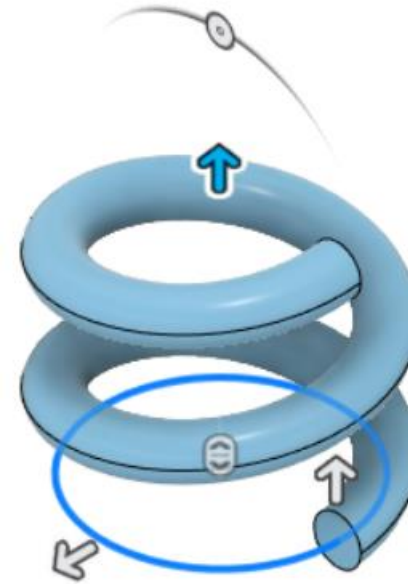
- Select a **plane** or **planar face**, place the **center point**, specify the **inner diameter**, then specify the **torus diameter**.
- Can also select the **position** of the torus relative to the **inner diameter**.



# Coil

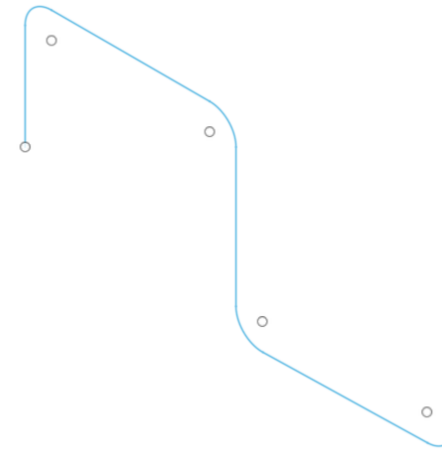
The **Coil** command creates a solid body in the shape of a **primitive coil**.

- Select a **plane** or **planar face**, place the **center point**, specify the **diameter**, then adjust the coil settings.
- Can adjust the coil **type**, **rotation**, **diameter**, **number of revolutions**, **height**, **angle**, and **section shape**.



# Pipe

- **3D sketch the path** of the pipe on **top plane**.
- Finish the sketch, **Pipe** command creates a solid body in the shape of a **primitive pipe** that follows a **path**.
- Select a **path for the pipe to follow**, then specify **the distance**, **section shape**, and **section size**.
- Can also choose to **hollow** the pipe.



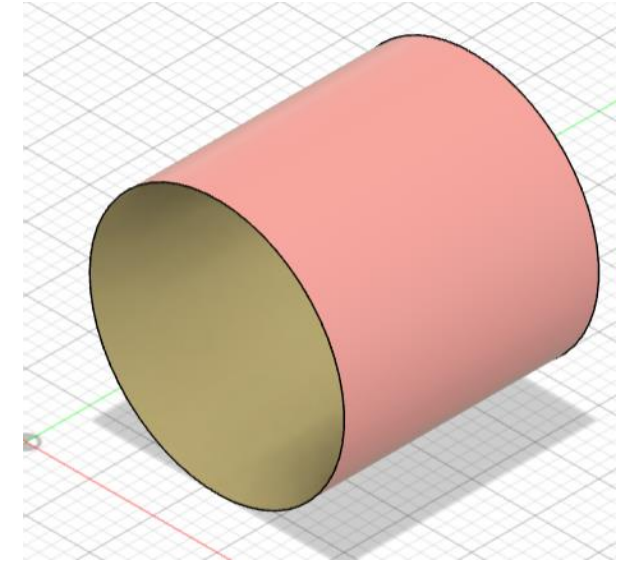
# Solid from surfaces

## Thicken a surface

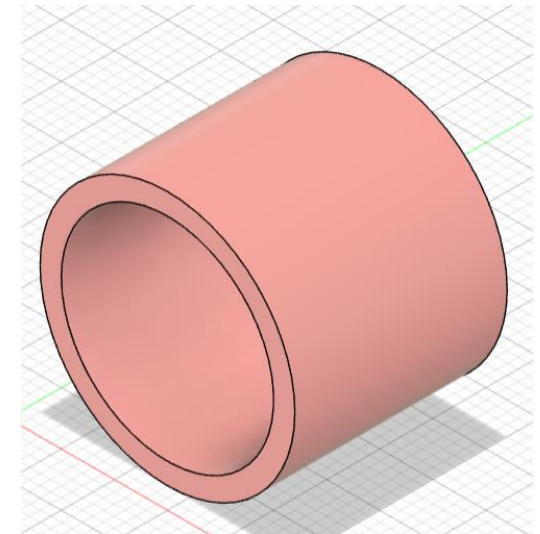
**Thicken** cannot remove a face, so the maximum offset value is set at the distance where a face disappears.

1. In the Design workspace, **Solid** tab, select **Create > Thicken**.
2. In the **Thicken** dialog, **Chain** selection only if you want to select specific **faces or surfaces**.
3. Select a **face or faces**.
4. Use the manipulator or enter a value for the thickness (**positive** values thicken the **exterior surface**, and **negative** ones thicken the **interior surface**).

Under **surface**  
extrude circle 2D  
sketch



**Thicken** the surface  
to make the  
cylindrical **hollow**  
body



# Solid Modifications

## Create solids with Press Pull

Click Design > **Solid > Modify > Press Pull**.

Select **sketch** profiles, **edges**, or **faces**:

- **Sketch Profile**: **Extrude** a new solid body from the **sketch profile**.

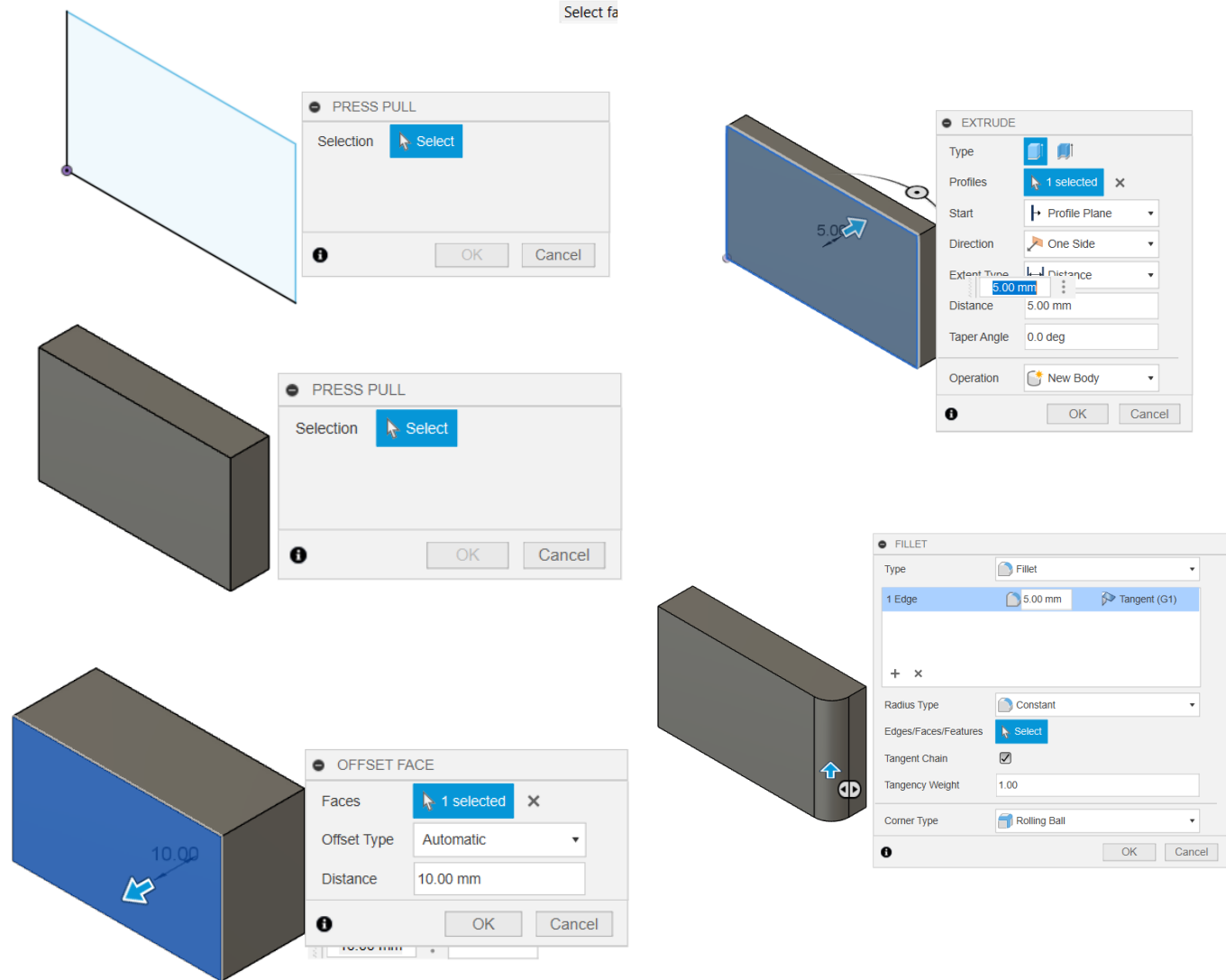
The **Extrude** dialog displays.

- **Edge**: **Round** the edges of the **solid body**.

The **Fillet** dialog displays.

- **Face**: **Add or remove volume** from the solid body.

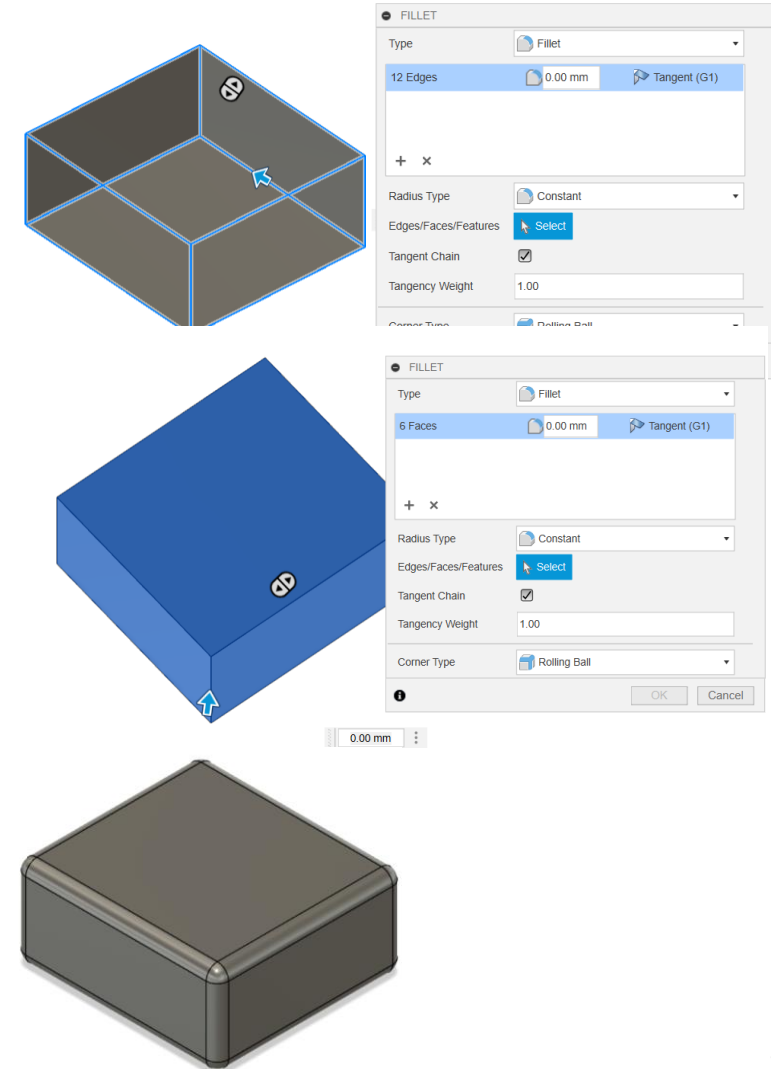
The **Offset Face** dialog displays.



# Solid Modifications

## Create a fillet

1. Click Design > Solid > Modify > **Fillet**.
2. In the canvas, select **edges, faces, or features** to fillet.
3. The **selection set displays as a row** in the selection box.
4. Adjust the settings associated with the selection set:
5. Optional: Click the **+** icon to add a **selection set** to the list. Repeat steps 3-4 to create **fillets with different settings** than the first selection set.

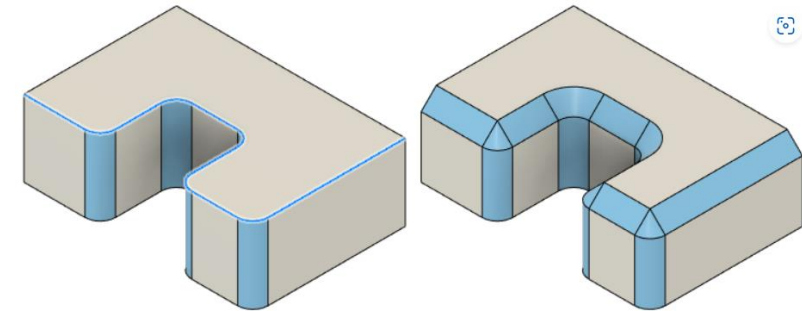
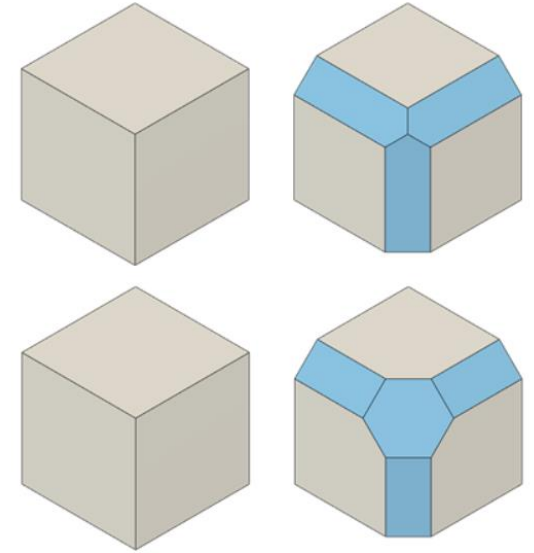




# Solid Modifications

## Create a chamfer

1. Click Design > Solid > Modify > **Chamfer** .
2. In the canvas, **select edges, faces, or features** to chamfer.  
  
The **selection set displays as a row** in the selection box.
3. In the dialog, select the chamfer Type:
4. Adjust the **Distance or Angle** values for the chamfer:
5. Optional: For the Two Distance chamfer type, click the Flip icon to flip the first and second sides.
6. Select a Corner Type:
7. Optional: In the selection box, click the **+ icon to add a selection set to the list**. Repeat steps 2-6 to create fillets with different settings than the first selection set.

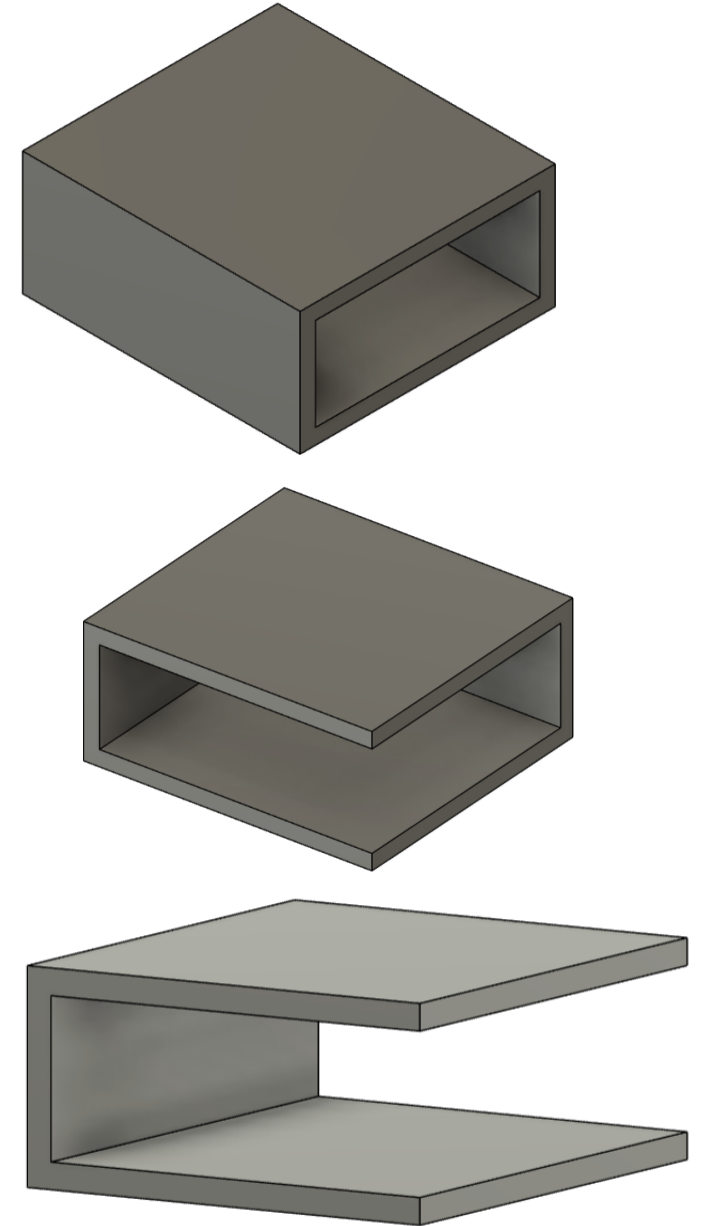


# Solid Modifications

## Create a thin-walled solid

1. Click Design > Solid > Modify > **Shell** .
2. In the canvas or the Browser, **select faces or a solid body**.
3. In the dialog, select the **Direction**:
4. Specify **Inside** Thickness and **Outside** Thickness:

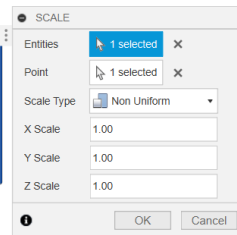
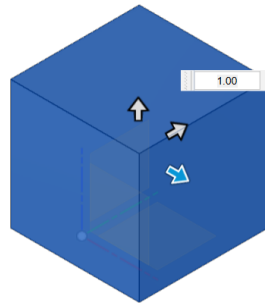
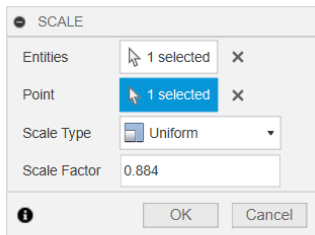
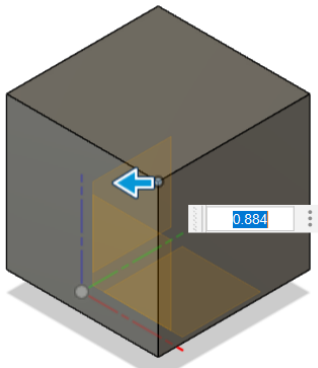
Use the shell manipulator handles in the canvas, or enter exact values.



# Solid Modifications

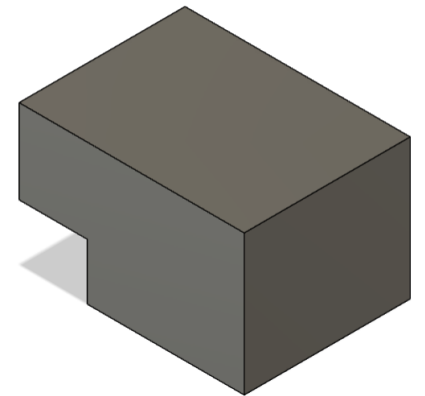
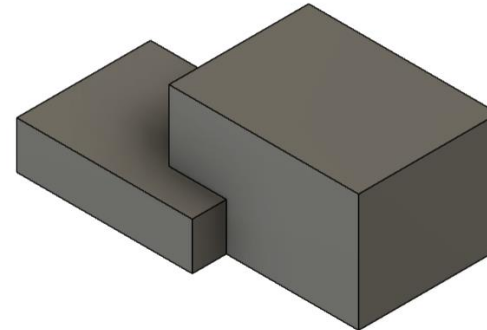
## Scale components, bodies, or sketches

1. In the Design workspace, Solid tab, select **Modify > Scale**.
2. Select **the body or bodies** to scale, and pick a fixed anchor point for the scaling.
3. Choose a Scale Type from the dialog:
  - **Uniform**. Scale the body uniformly on all axes.
  - **Non Uniform**. Scale along the x, y, and z axes separately.

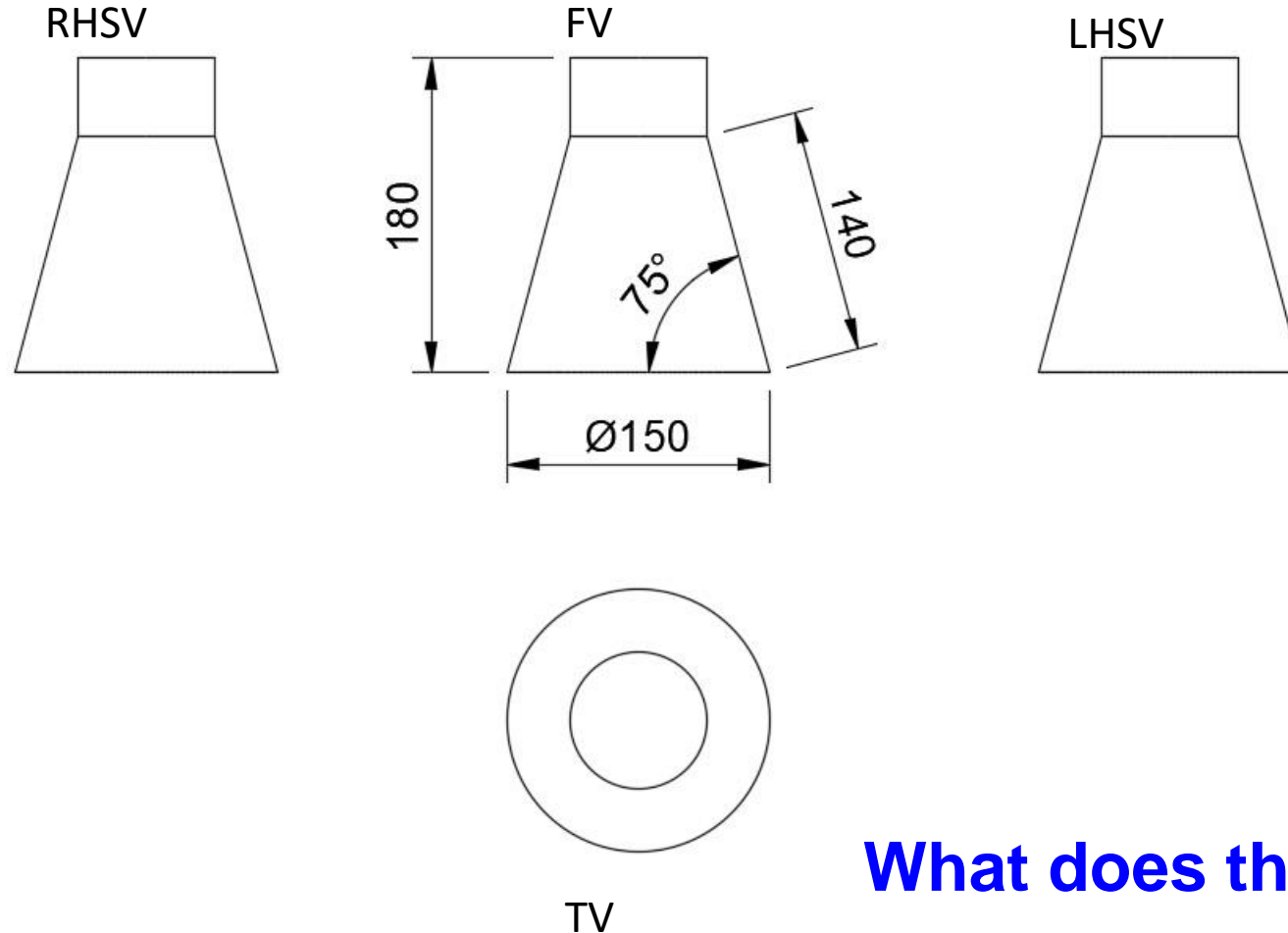


## Combine solid bodies

1. Click Design > Solid > **Modify > Combine**.
2. In the canvas, select the **Target Body**.
3. Select **Tool Bodies**.
4. In the dialog, select the **Operation**:
5. Optional: **Check New Component** to create a new component from the result.
6. Optional: **Check Keep Tools** to keep the Tool Bodies after the solid bodies are combined.

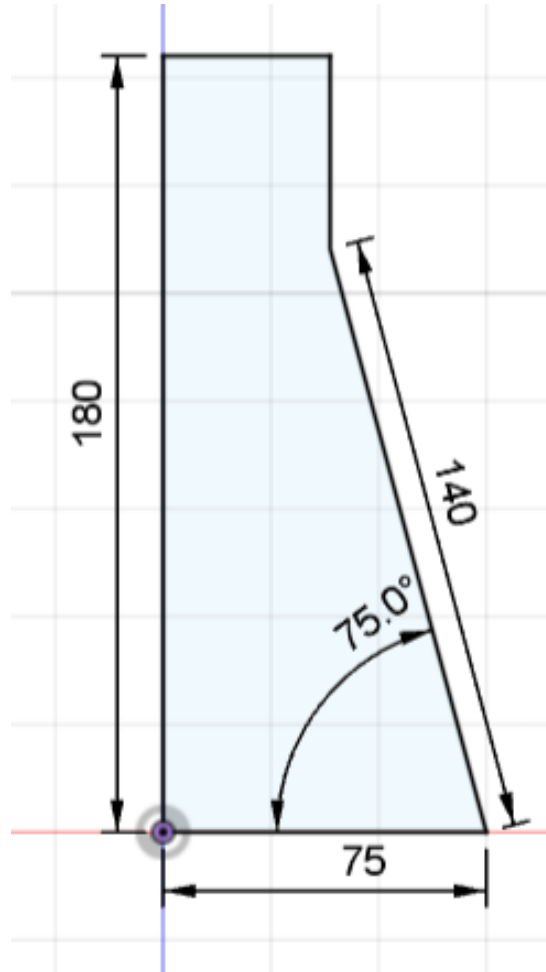


# First angle projections of a 3D object

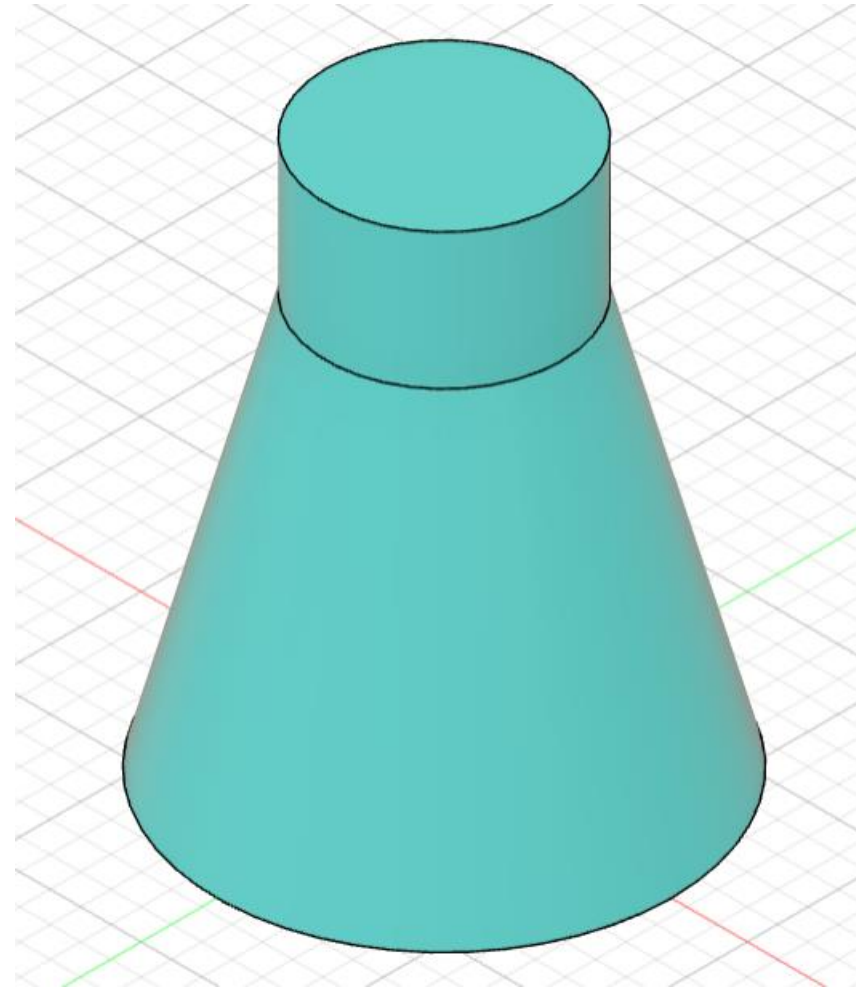


**What does this object look like?**

# Solids from sketches



**Axisymmetric 2D sketch**



**Solid by revolving**