

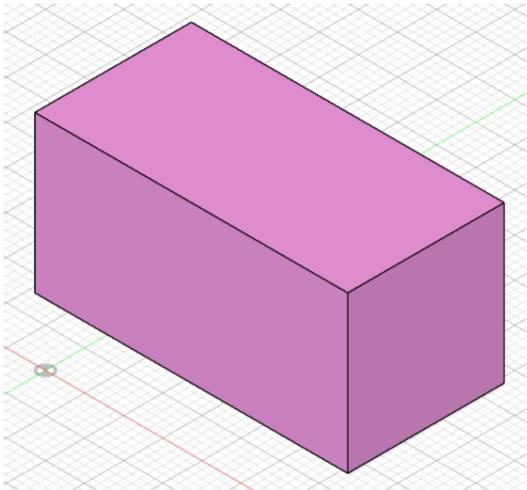
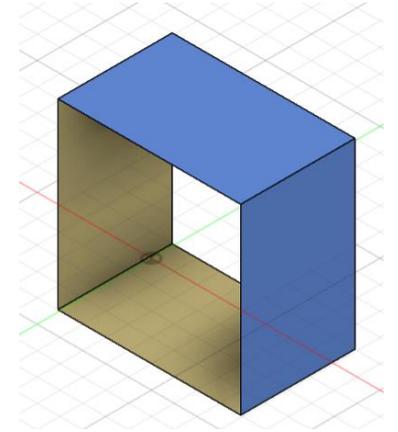
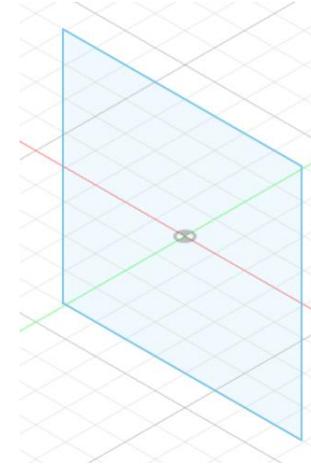
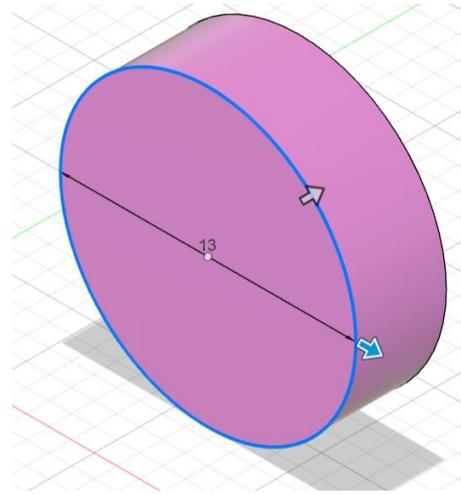
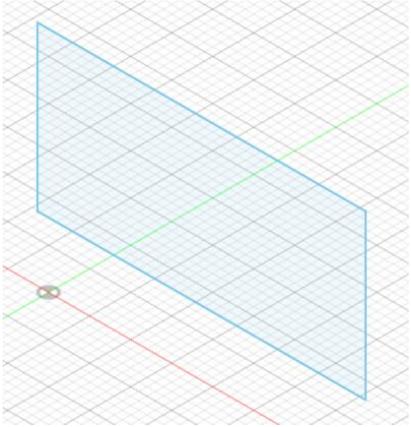
MS 101

Fusion 360

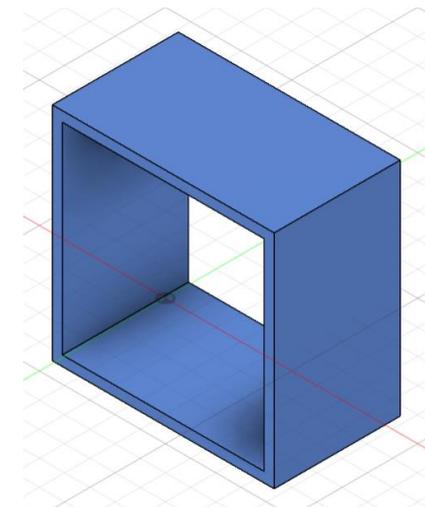
Solid Modelling

(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:

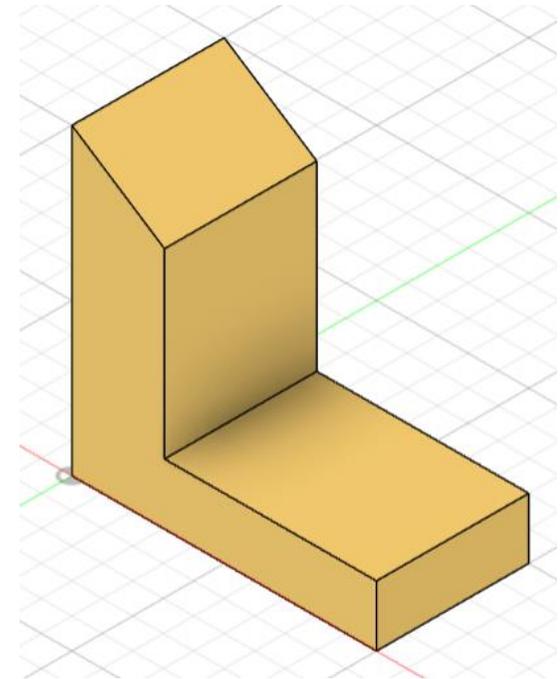
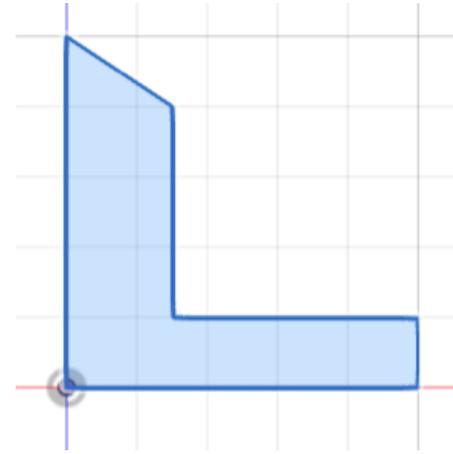
- Extrude
- Revolve
- Sweep
- Loft
- Rib
- Web
- Emboss

Extrude a solid body

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

The **Extrude** dialog displays.

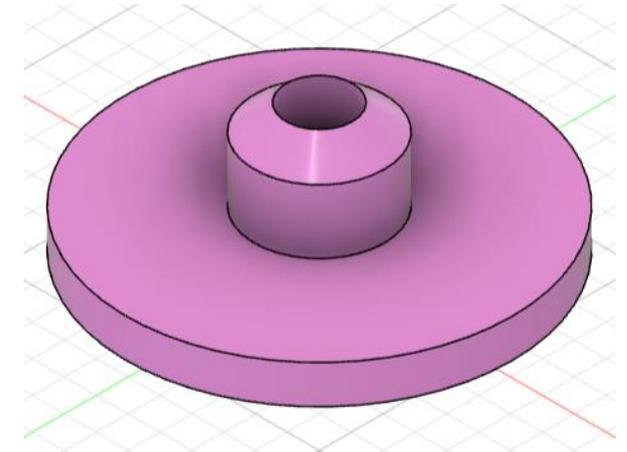
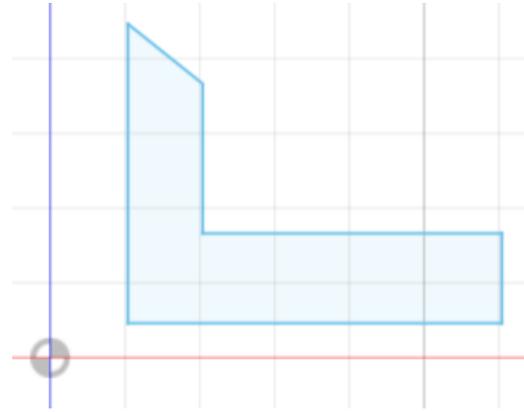
1. In the canvas, select one or more coplanar sketch profiles or planar faces to extrude.
2. In the dialog, select an extrude **Type**:
3. Select a **Start** setting, then adjust its associated settings:
4. Select a **Direction** setting, then adjust its associated settings:
5. Select an **Extent Type**,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



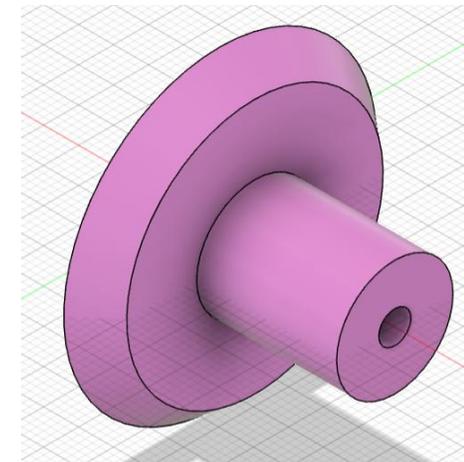
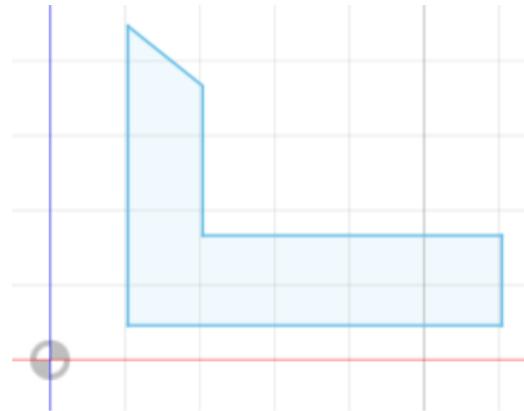
Revolve a solid body

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

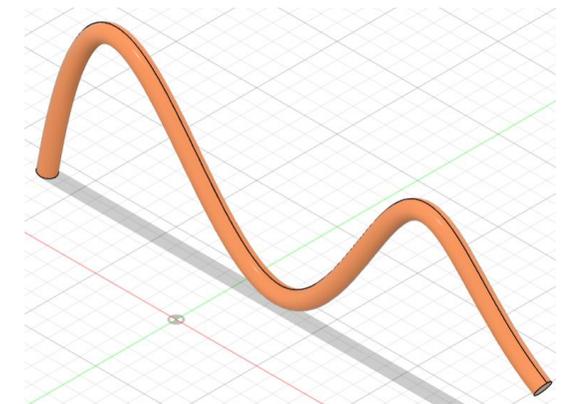
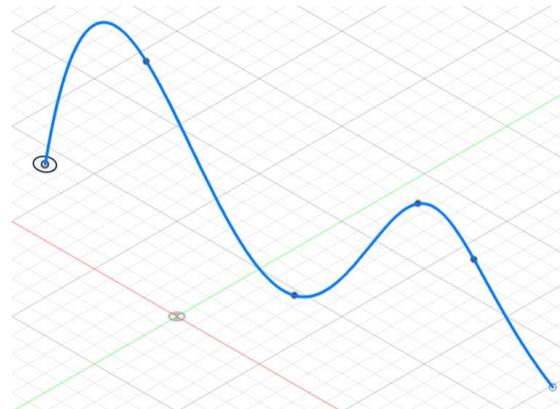
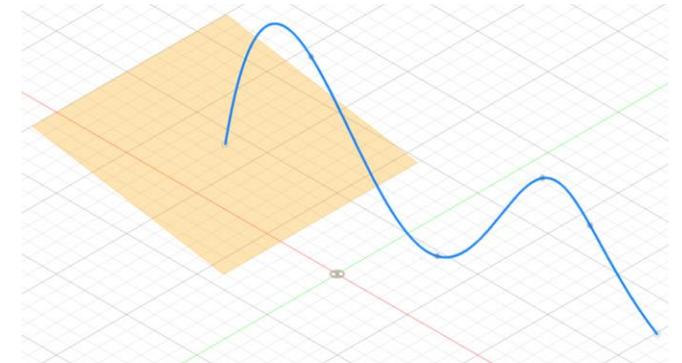
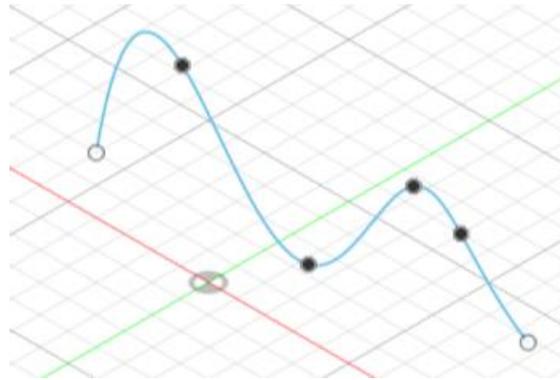
Sweep a profile along a path

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

The **Sweep** dialog displays.

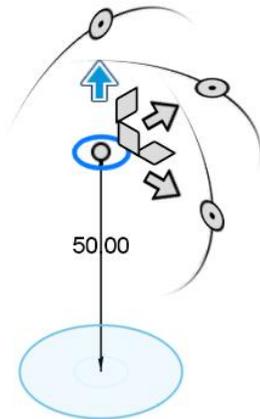
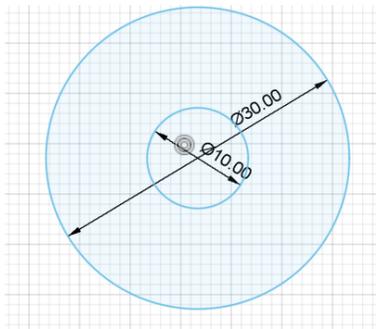
- Type
- Profile
- Path
- Distance
- Taper angle
- Twister angle
- Orientation
- operation

- 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.



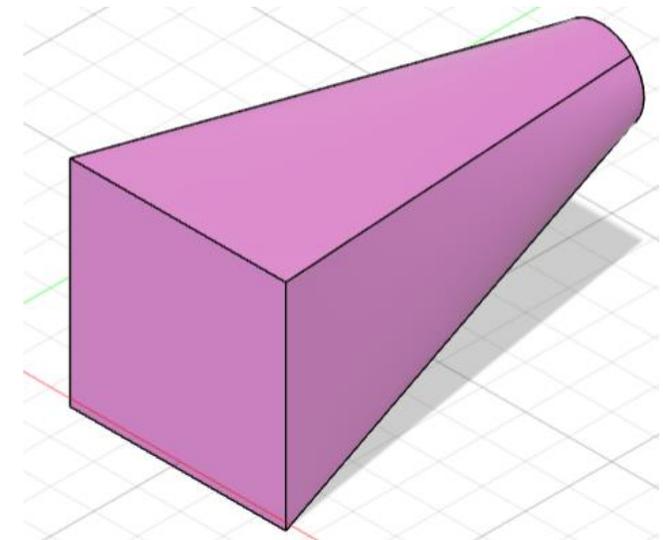
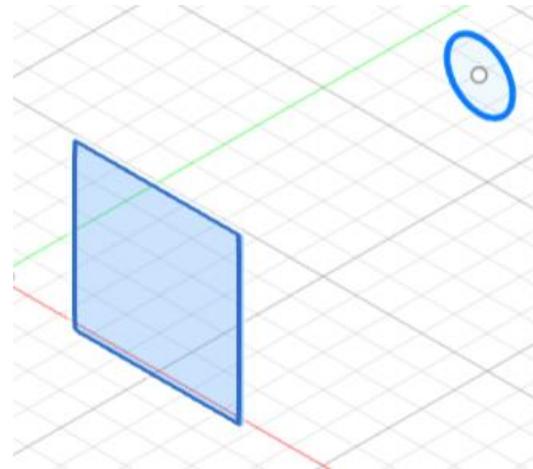
Loft a solid body

- Sketch **1st 2D profile** in a plane.
- Sketch the **2nd 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 2nd profile**



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

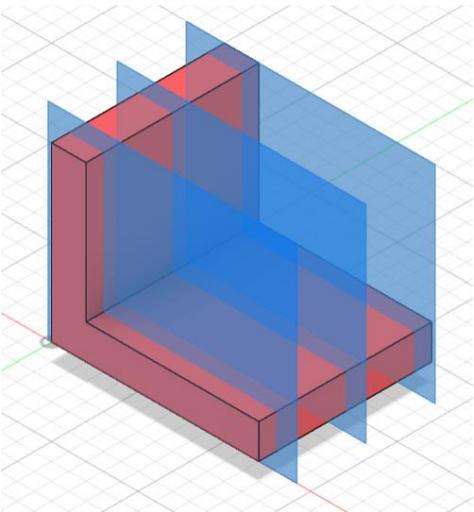
- The Loft dialog displays.
- Select two or more profiles
- Guide type
- Tangent edges
- operation



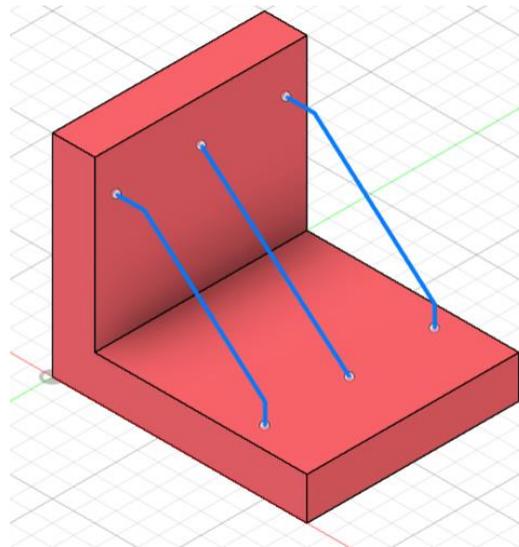
Create a rib

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:

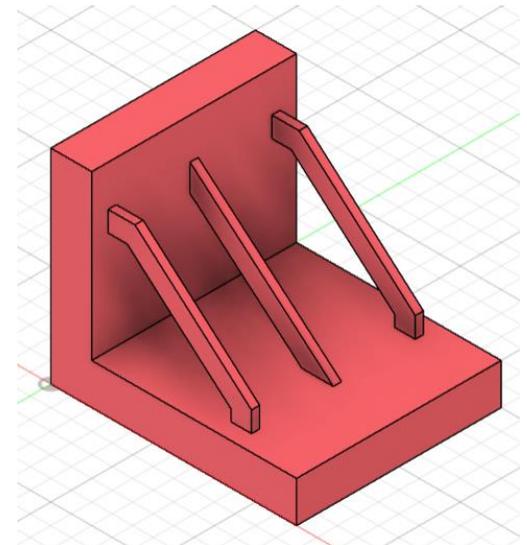
Construct offset planes



Construct 2D profiles

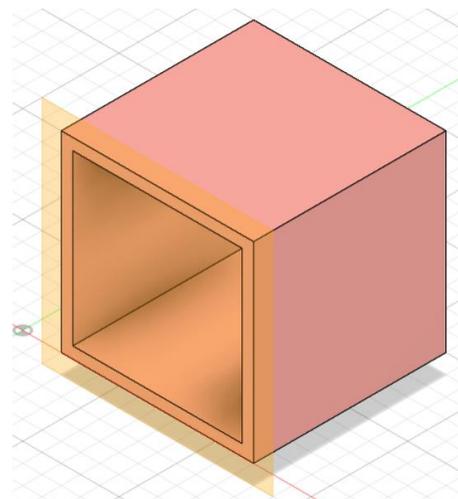


Create rib

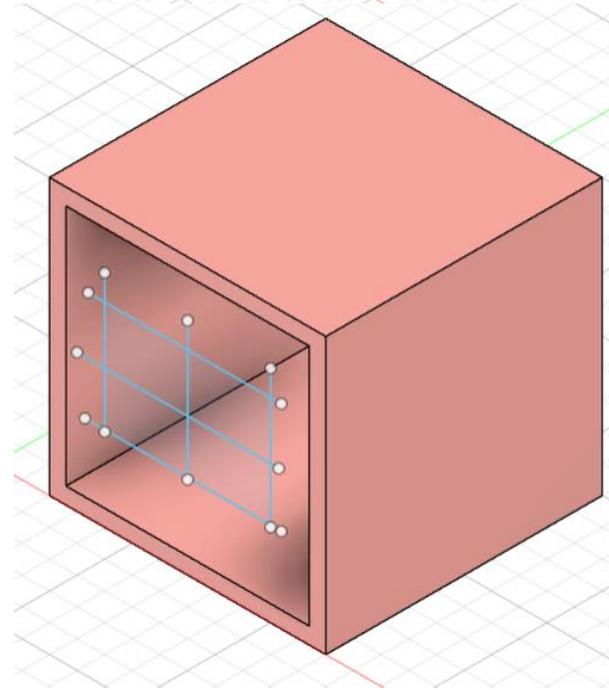


Create a web

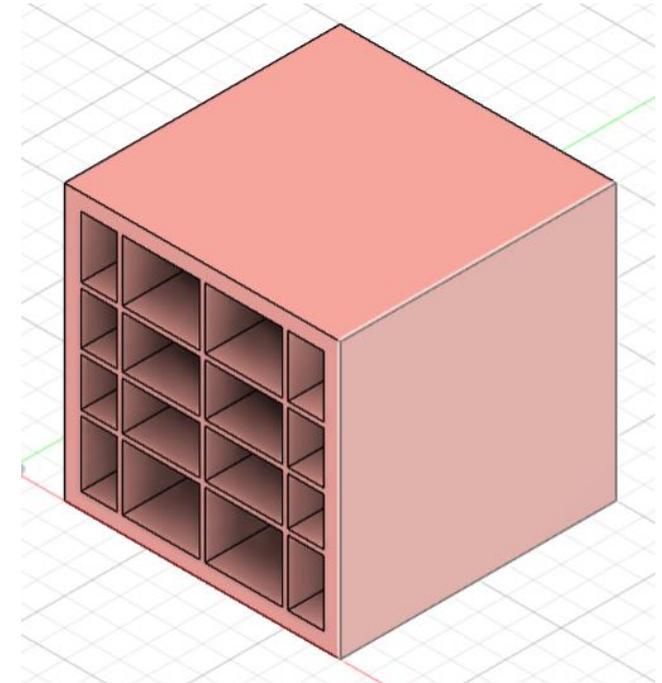
1. On the toolbar, click **Solid > Create > Web**.
2. In the canvas, select an open **sketch profile** to use as the **Profile**.
3. In the dialog, select a **Thickness Direction** setting:
4. Select an **Extent Type** setting, then adjust its associated settings:



Create solid and offset plane



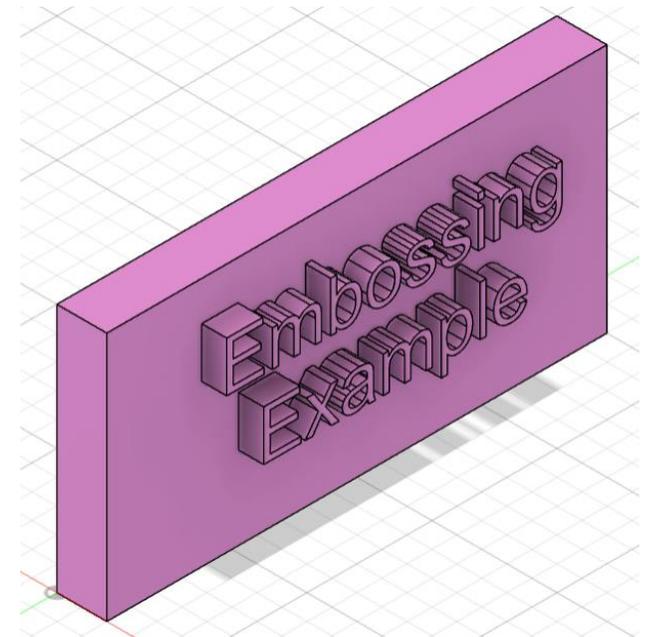
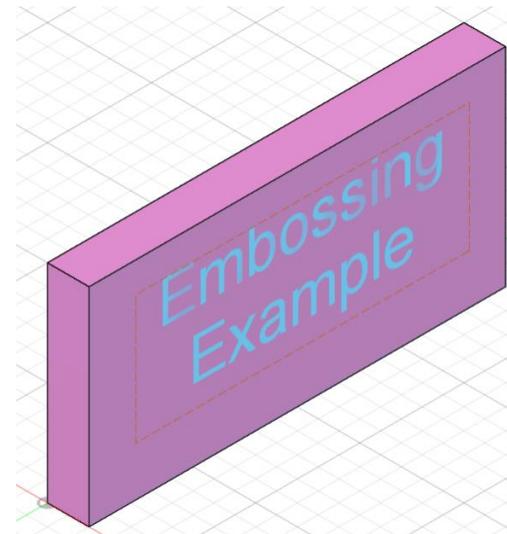
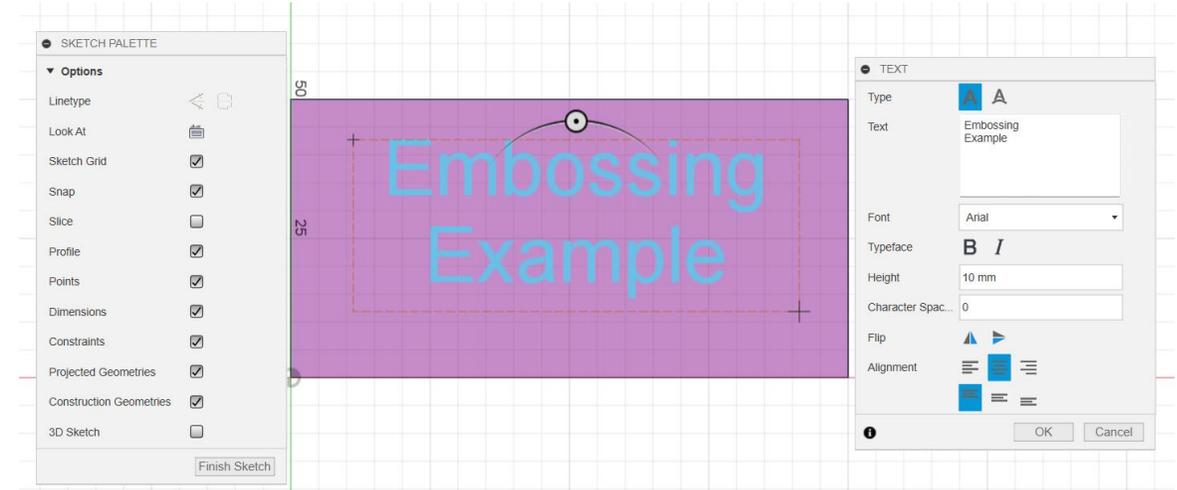
Create profile



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.



Solid Primitives

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

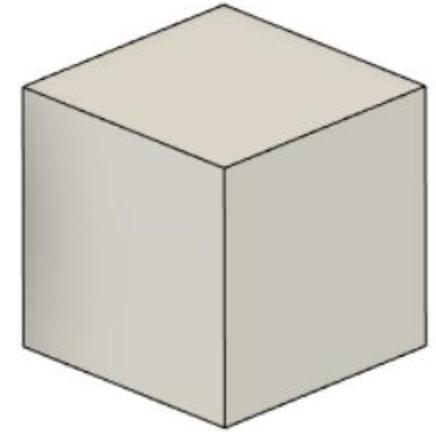
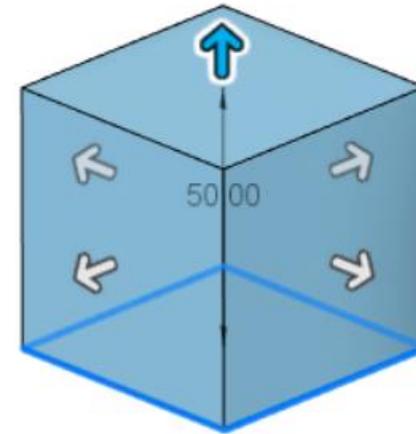
Use the following commands to create a solid body from a primitive shape:

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

Box

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

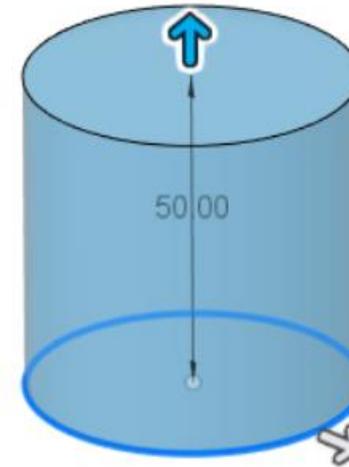
Select a **plane or planar face**, place the first corner, specify the **length and width**, then specify the **height** of the box.



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**.

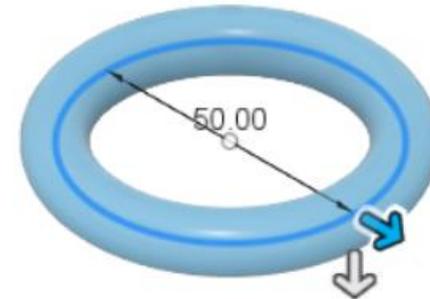
Select a **plane** or **planar face**, place the **center point**, then specify the **diameter** of the sphere.



Torus

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

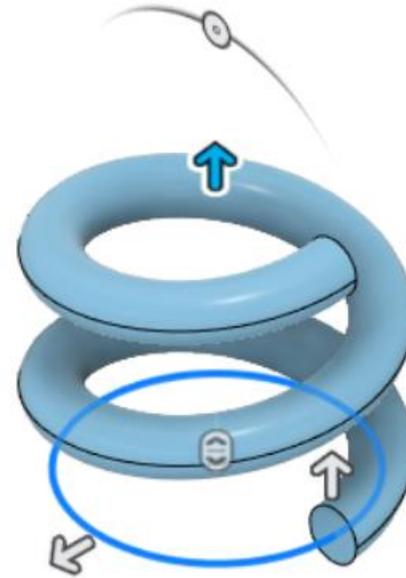
- Select a **plane** or **planar face**, place the **center point**, specify the **inner diameter**, then specify the **torus diameter**.
- Can also select the **position** of the torus relative to the **inner diameter**.



Coil

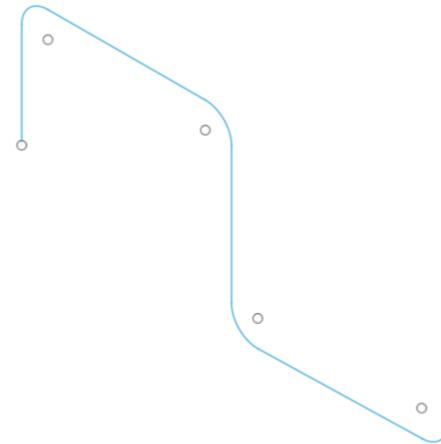
The **Coil** command creates a solid body in the shape of a **primitive coil**.

- 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.



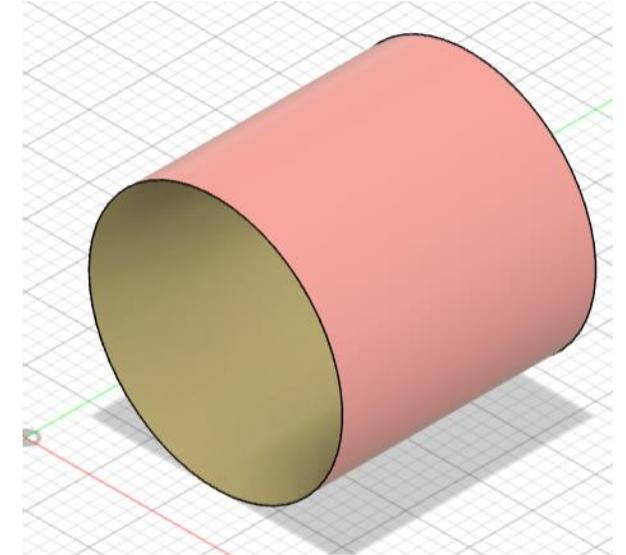
Solid from surfaces

Thicken a surface

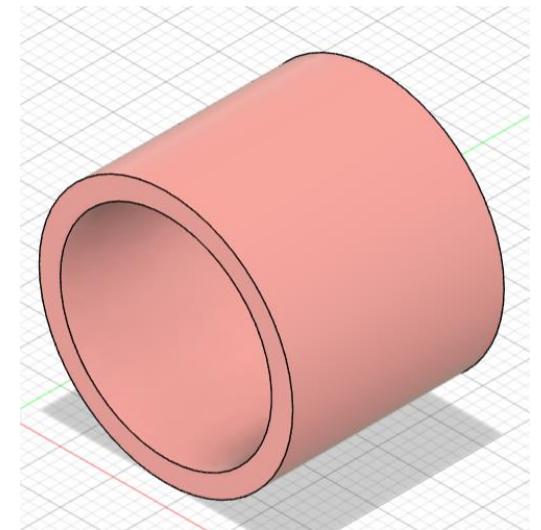
Thicken cannot remove a face, so the maximum offset value is set at the distance where a face disappears.

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**).

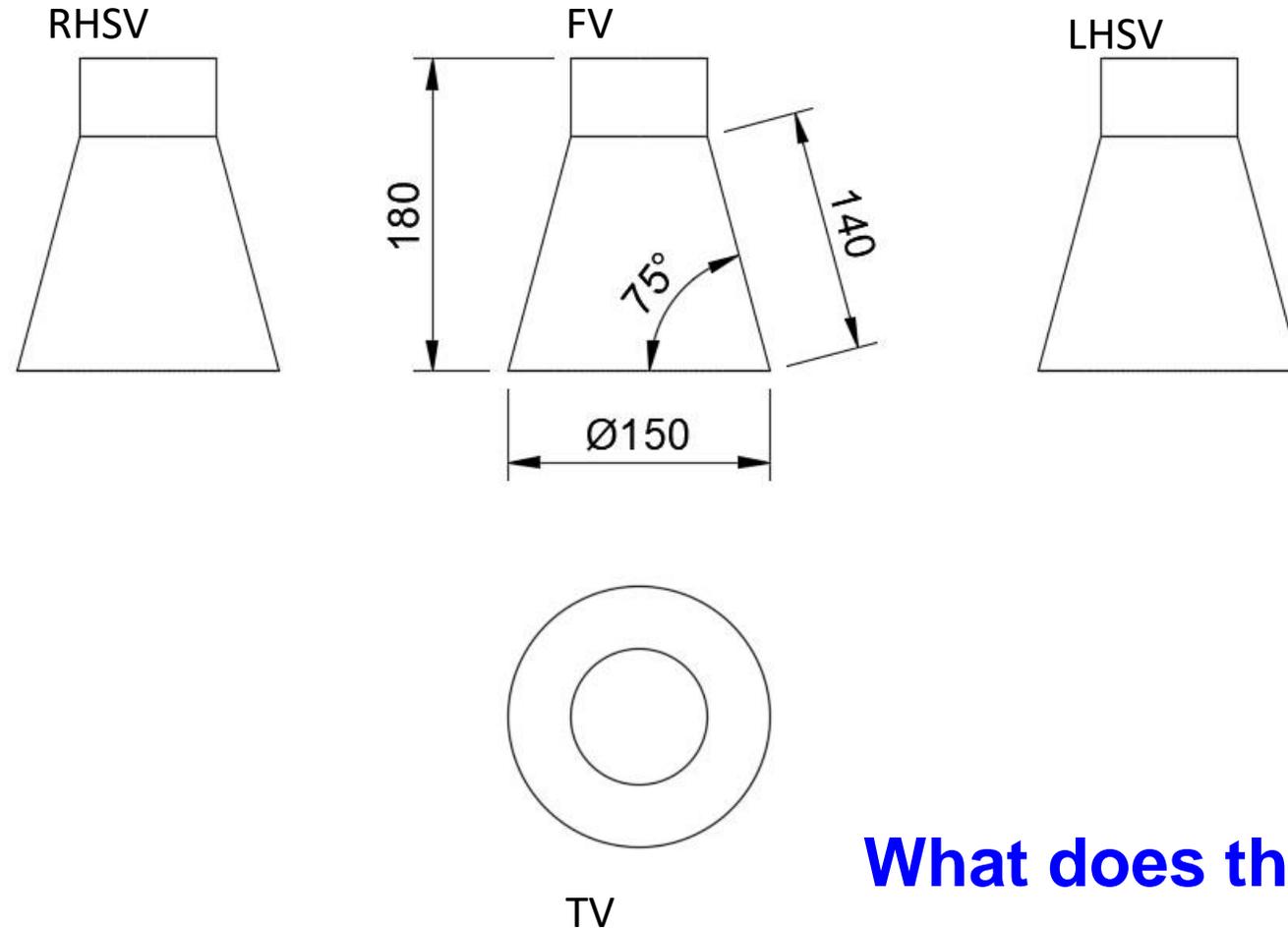
Under **surface**
extrude circle 2D
sketch



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

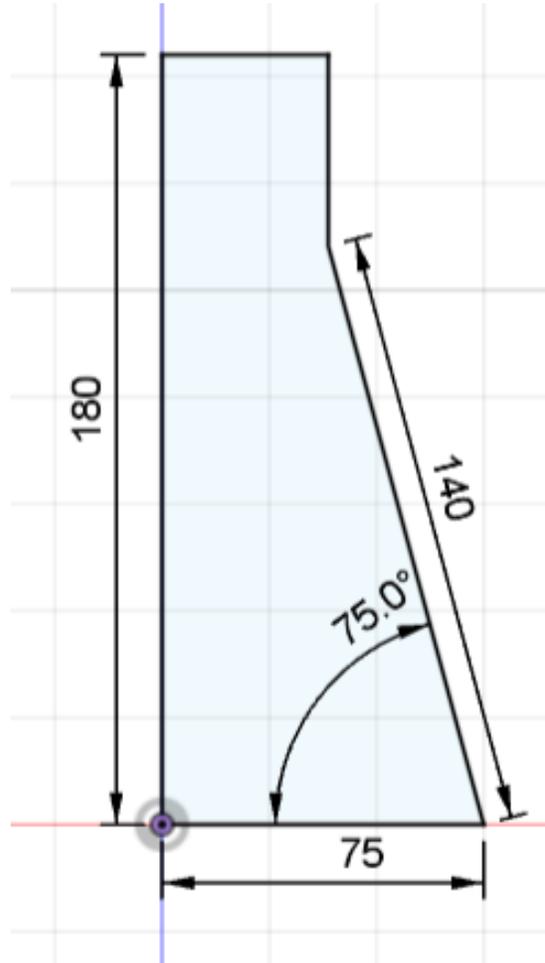


First angle projections of a 3D object

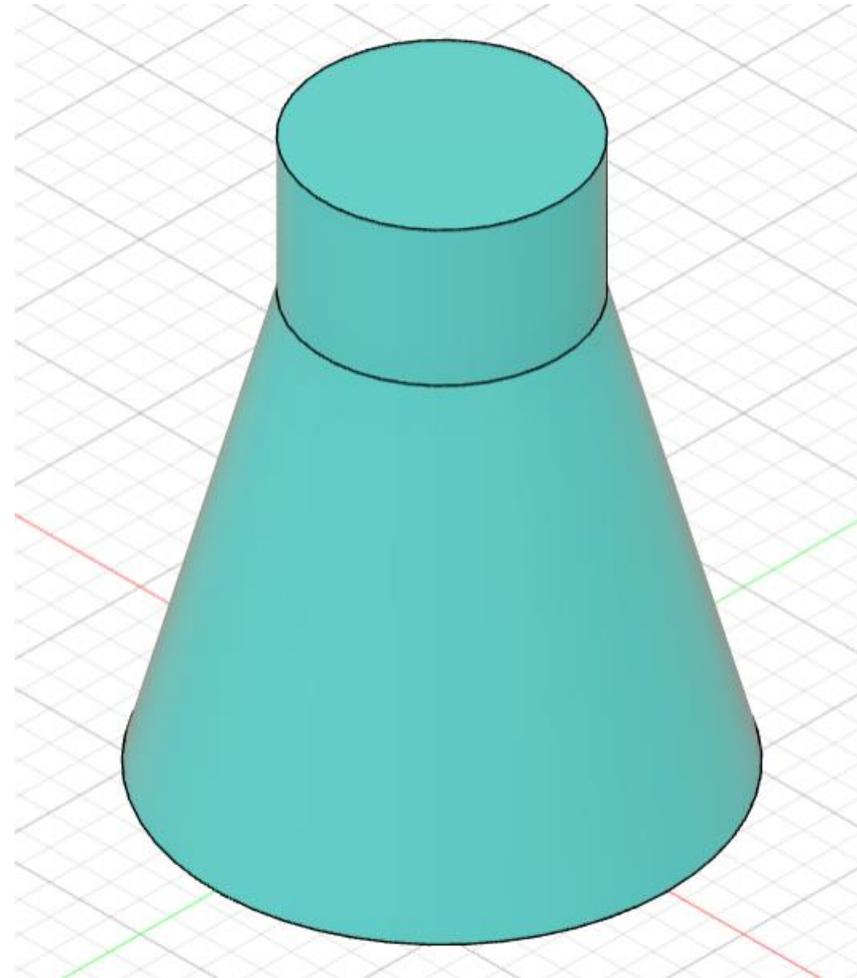


What does this object look like?

Solids from sketches



Axisymmetric 2D sketch



Solid by revolving