MS 101
Fusion360
Solid Modification
&
Assembly of components

(Fusion 360 manual)

Create a hole in a solid body

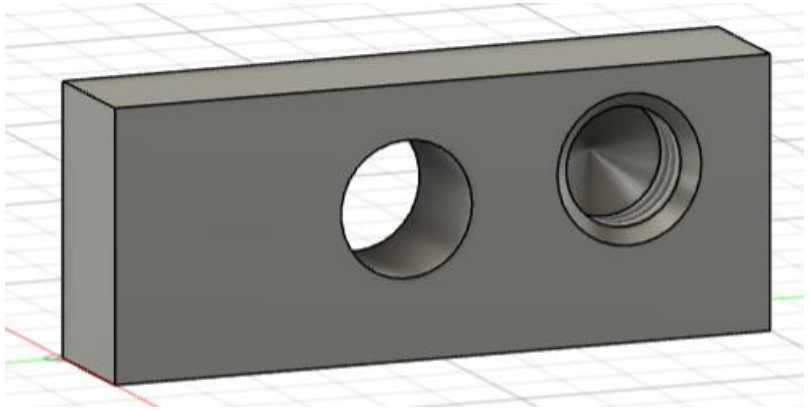
Click **Design > Solid > Create > Hole** .

The **Hole** dialog displays.

Select a **Placement** setting:

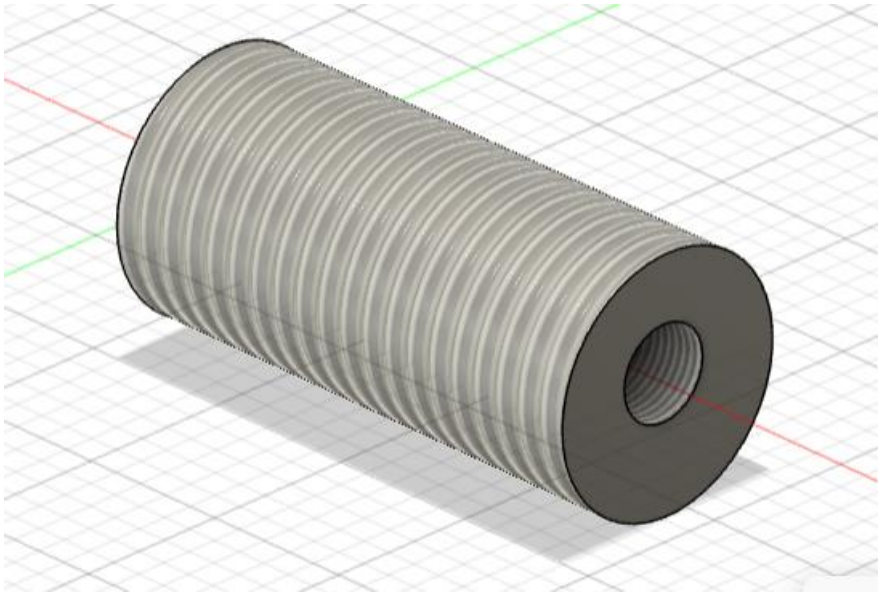
Select a face, plane, or sketch point to place the center of the **Hole**.

Adjust size, shape, and position settings as needed:



Add threads to holes or cylinders

In the **Design** workspace, **Solid** tab,
select **Create** > **Thread** and then select the face of the **Hole**.
Use the manipulator to change the size of the **Hole**, if desired.
In the **Thread** dialog, specify **Thread Type**, **Size**, and **Class**.



Create a fillet

1. Click **Design > Solid > Modify > Fillet** .

   The **Fillet** dialog displays.

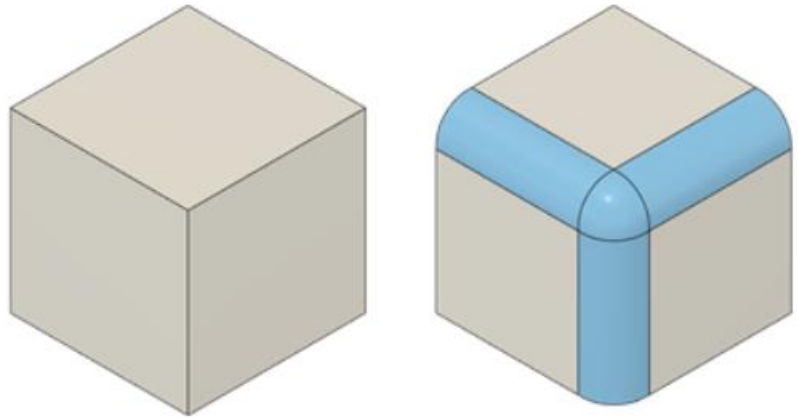
2. In the dialog, from the **Type** dropdown list, select **Fillet**.

3. In the canvas, select edges, faces, or features to fillet.

   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.



Create a chamfer

1. Click **Design > Solid > Modify > Chamfer** .

   The **Chamfer** dialog displays.

2. In the canvas, select edges, faces, or features to chamfer.

   The selection set displays as a row in the selection box.

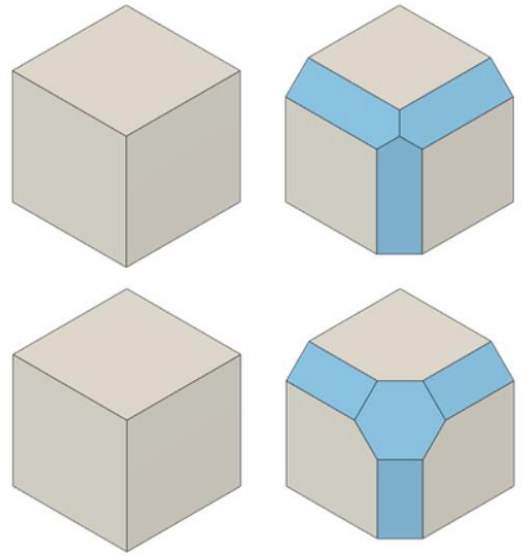
3. In the dialog, select the chamfer **Type**:

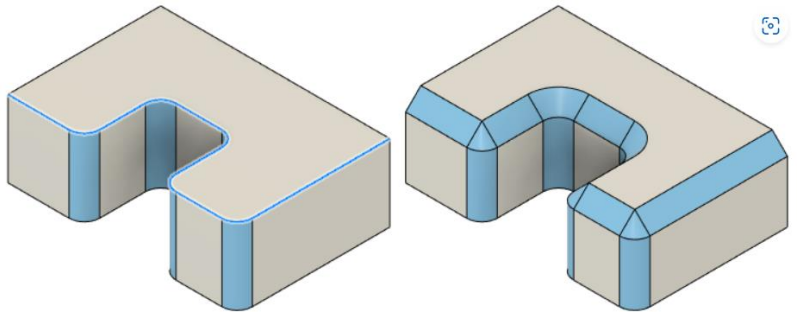
4. Adjust the **Distance** or **Angle** values for the chamfer:

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.





Create solids with Press Pull

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

The **Press Pull** dialog displays.

Select sketch profiles, edges, or faces:

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

  The **Extrude** dialog displays.

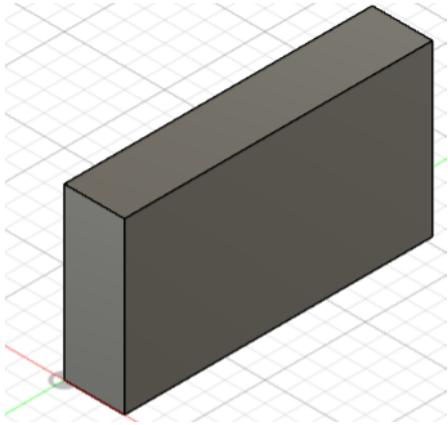
- **Edge**: Round the edges of the solid body.

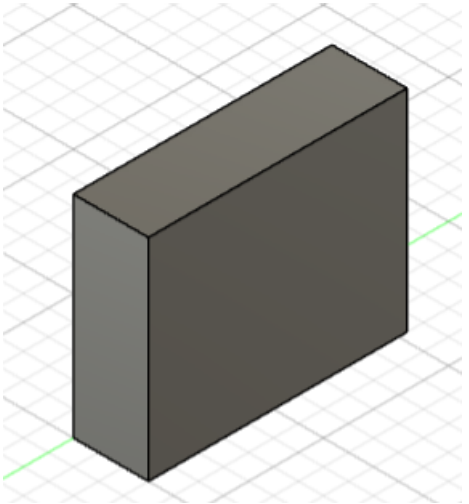
  The **Fillet** dialog displays.

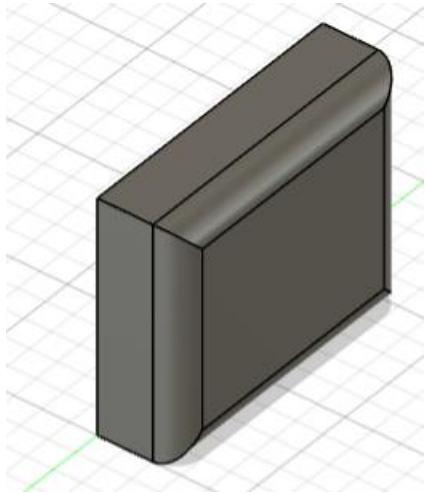
- **Face**: Add or remove volume from the solid body.

  The **Offset Face** dialog displays.

Use the manipulator handles to extrude, fillet, or offset geometry, or enter exact values in the dialog.







Create a thin-walled solid

1. Click **Design > Solid > Modify > Shell** .

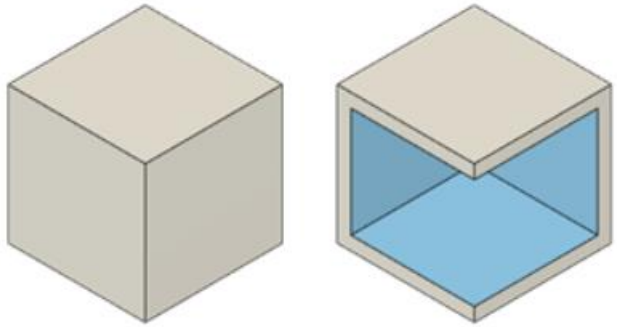
   The **Shell** dialog displays.

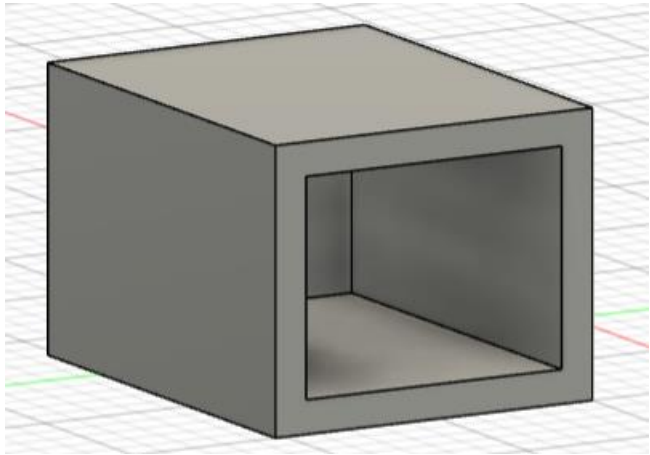
2. In the canvas or the **Browser**, select faces or a solid body.

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.





Create a draft

1. In the **Design** workspace, **Solid** tab, select **Modify > Draft**.

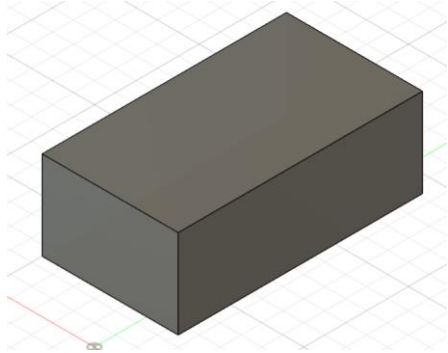
   The **Draft** dialog displays.

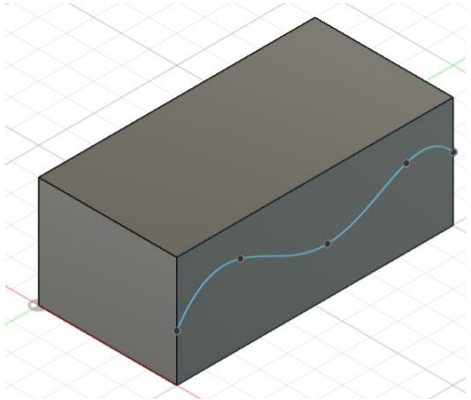
2. In the dialog, select the draft **Type**:
3. Select a plane or face to define the **Pull Direction**.

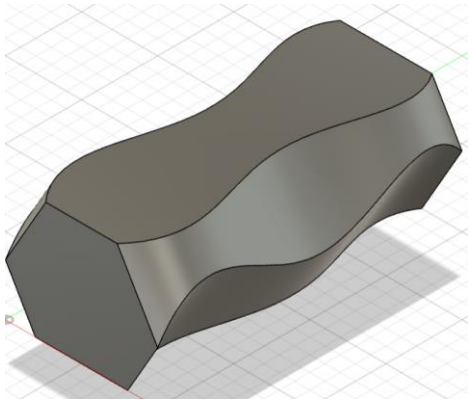
   To flip the pull direction, click the pull direction manipulator in the canvas.

4. Select the faces to draft.
5. For the **Fixed Plane** draft type, select **Draft Sides**:
6. For the **Parting Line** draft type, select a plane, face, edge, or sketch curve on the solid body to use as the **Parting Tool**.
7. For the **Parting Line** draft type, select the **Parting Line Type**, and adjust the associated settings:
8. Adjust the **Angle** of the draft:

   Use the angle manipulator handle in the canvas, or enter an exact value.:





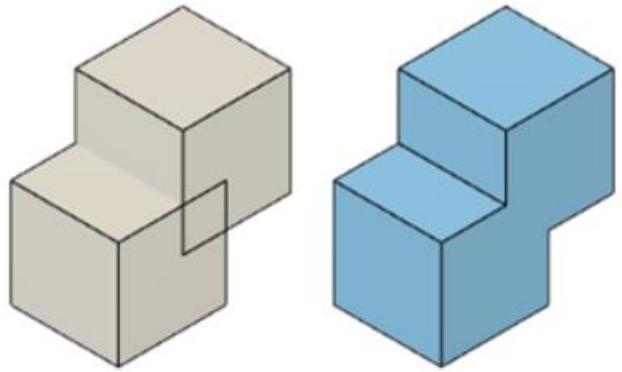


Scale components, bodies, or sketches

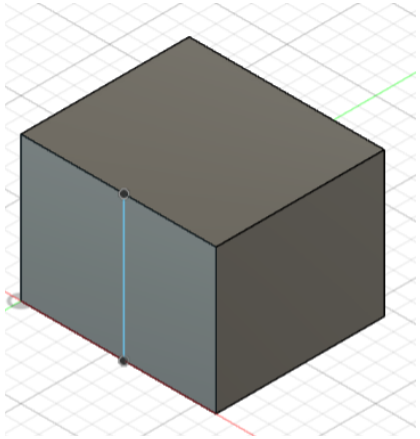
1. In the **Design** workspace, **Solid** tab, select **Modify > Scale**.
2. Select the body or bodies to scale, and pick a fixed anchor point for the scaling.
3. Choose a **Scale Type** from the dialog:
   - **Uniform.** Scale the body uniformly on all axes.
   - **Non Uniform.** Scale along the x, y, and z axes separately.
4. Use the manipulators or the dialog to set the scale distance (-0.5 to halve the size or 2 to double the size, for example).

Combine solid bodies

1. Click **Design > Solid > Modify > Combine** . The **Combine** dialog displays.
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.

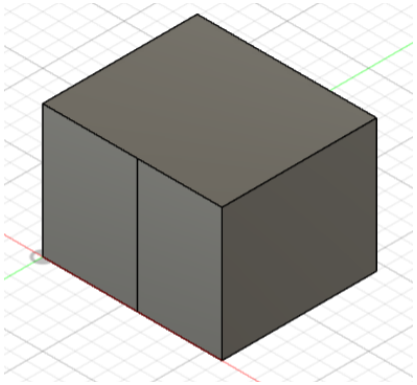


Divide a face into multiple faces

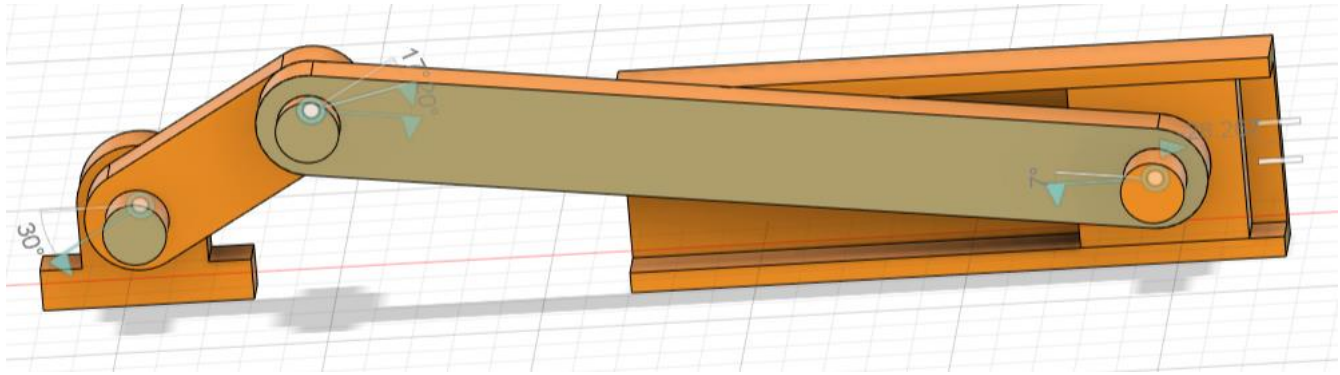


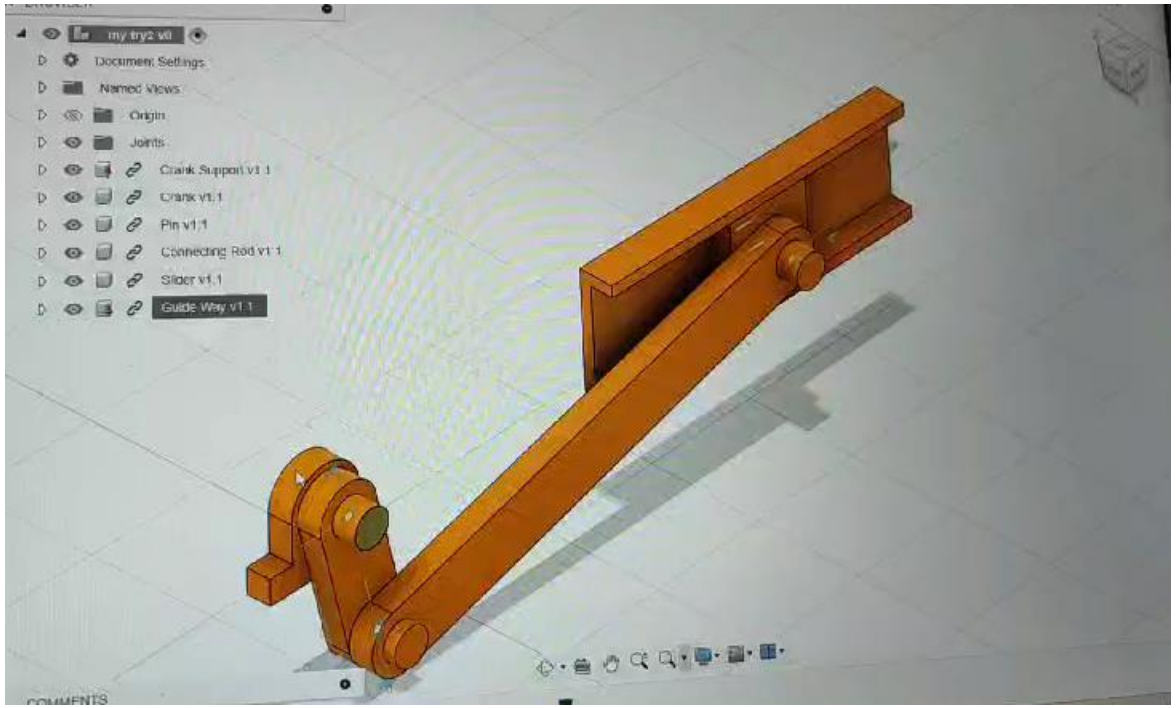
1. In the **Design** workspace, **Solid** tab, select **Modify** > **Split Face**.
2. Select a face to split. (Hold Ctrl (Windows) to select multiple faces.)
3. In the **Split Face** dialog, click the **Splitting Tool** field, and do one of the following:
   - Select a surface or sketch on the canvas.
   - Select a workplane.
4. If you select a sketch or surface, make sure that **Extend Splitting Tool** is selected.

   This setting ensures that your cutting tool completely intersects your faces.

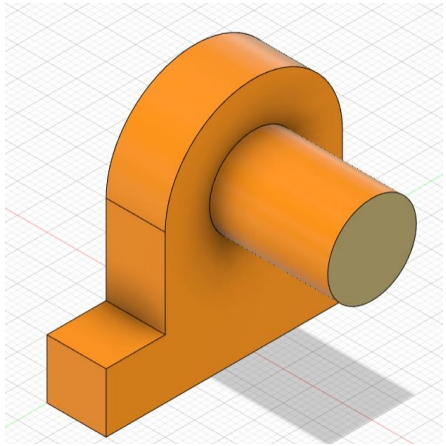


Slider Crank Mechanism

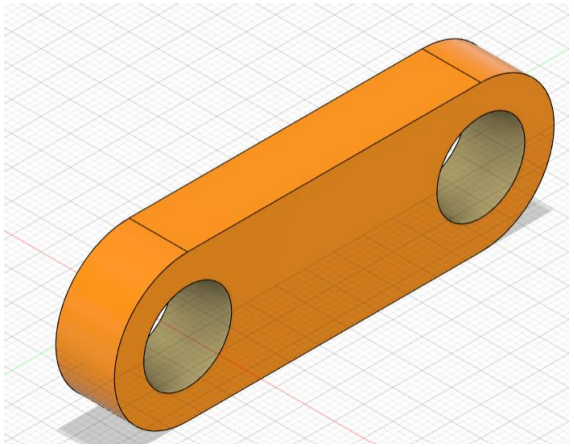




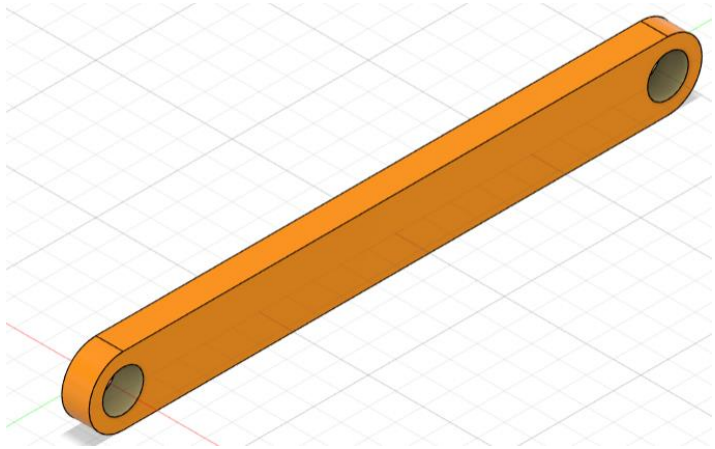
Slider Crank Mechanism



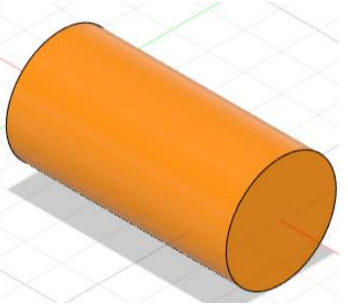
Crank support



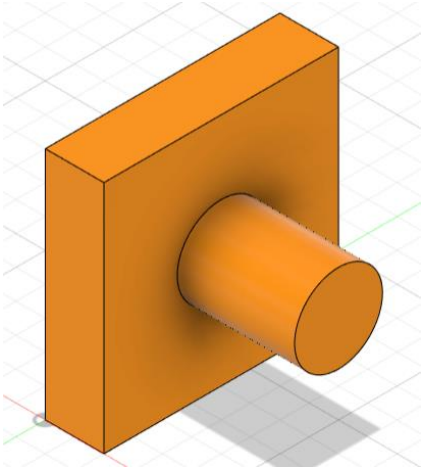
Crank



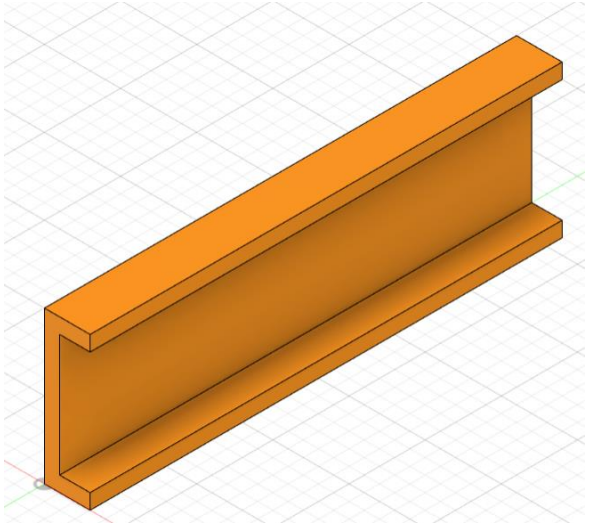
Connecting rod



Pin



Slider



Guide way