MS 101
Fusion360
Solid modification
&
Assembly of components

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**.

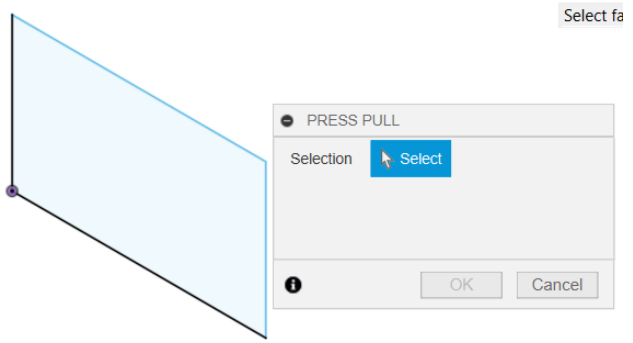
  The **Extrude** dialog displays.

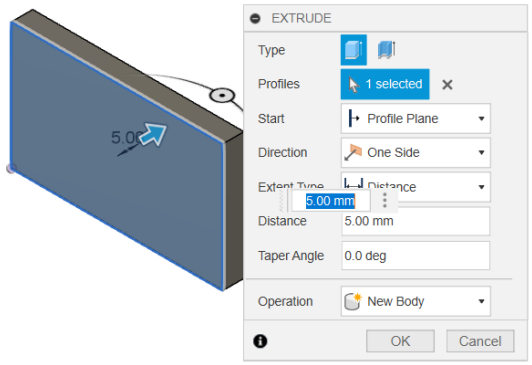
- **Edge**: **Round** the edges of the **solid body.**

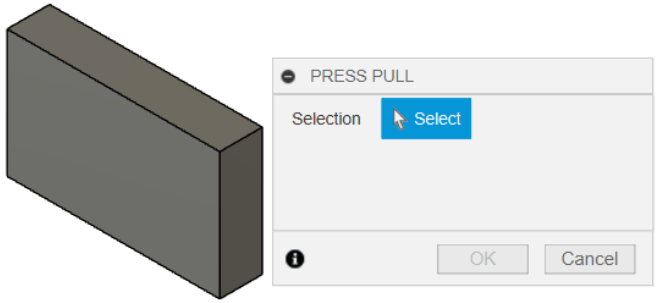
  The **Fillet** dialog displays.

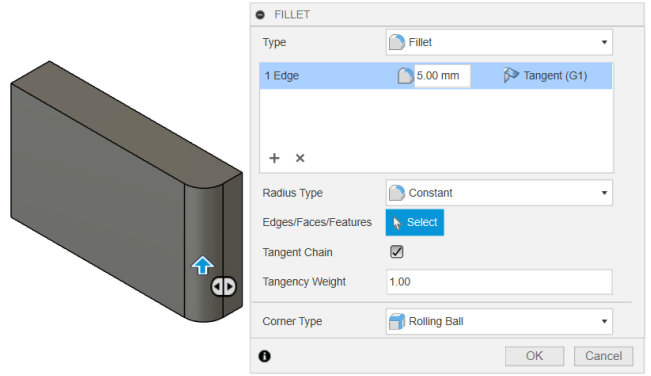
- **Face**: **Add or remove volume** from the solid body.

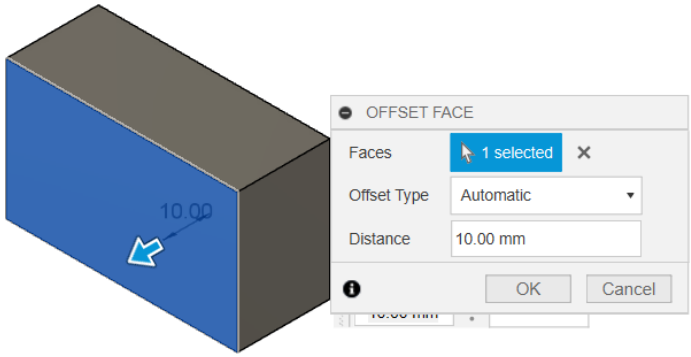
  The **Offset Face** dialog displays.





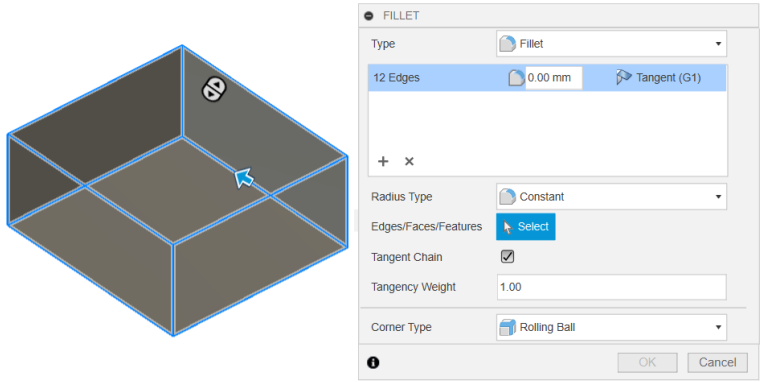


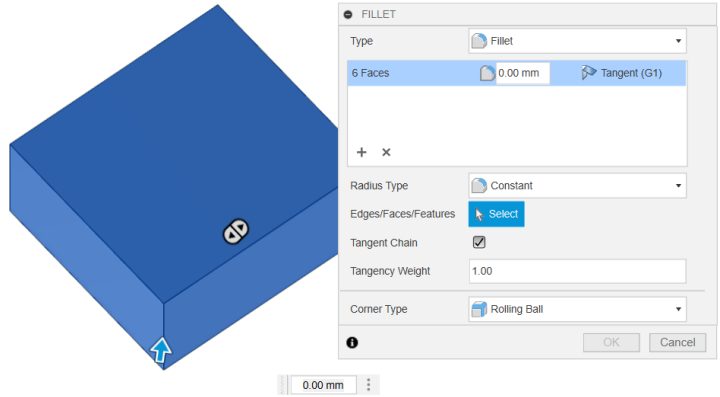


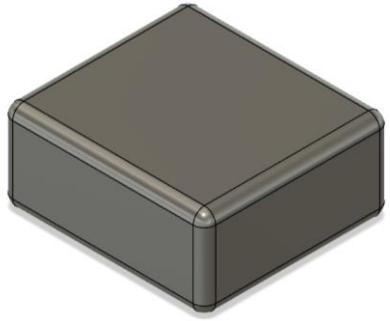


Create a fillet

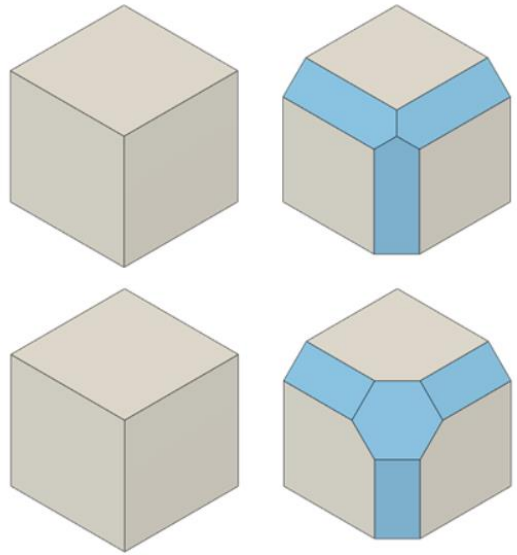
1. Click Design > Solid > Modify > **Fillet** .
2. In the canvas, select **edges, faces, or features** to fillet.
3. The **selection set displays as a row** in the selection box.
4. Adjust the settings associated with the selection set:
5. Optional: Click the **+ icon to add a selection set** to the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





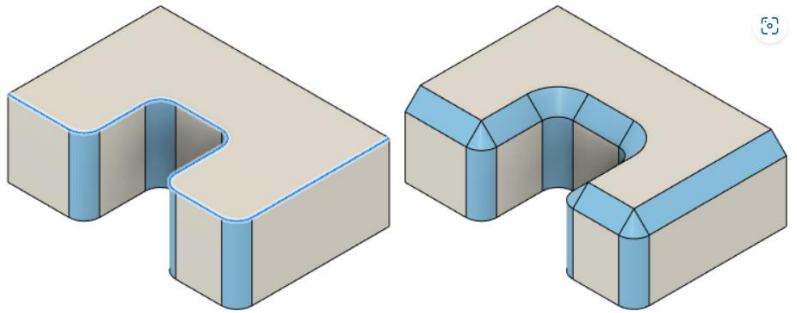


Create a chamfer



1. Click Design > Solid > Modify > **Chamfer** .
2. In the canvas, **select edges, faces, or features** to chamfer.

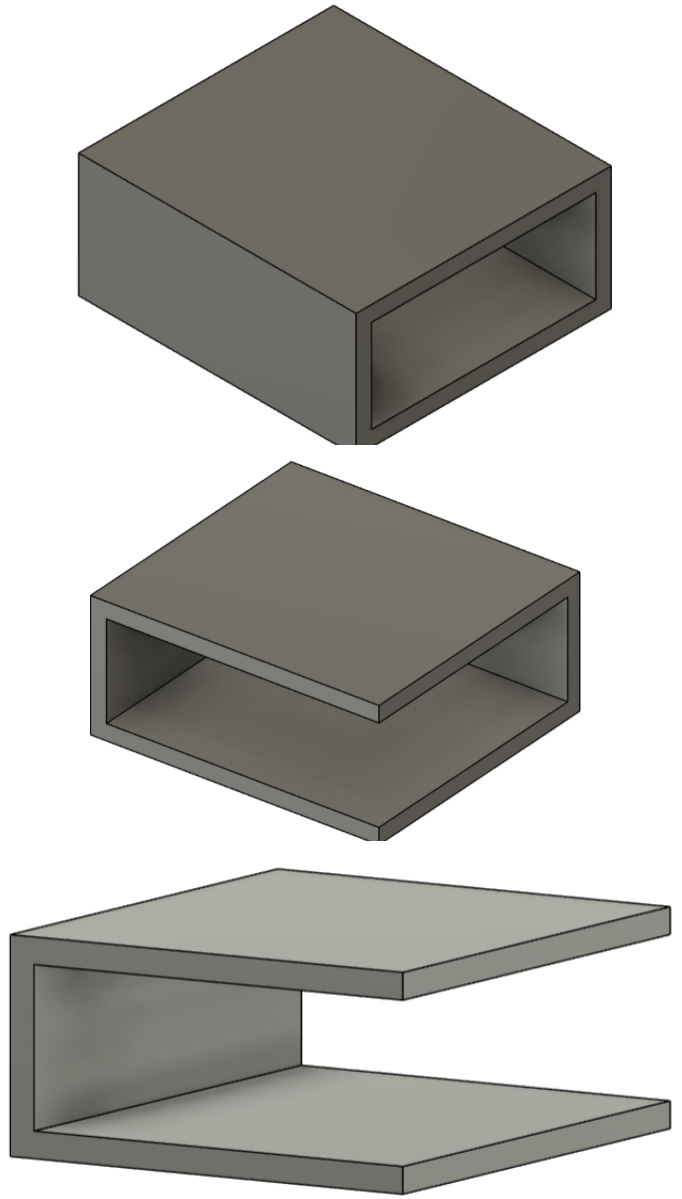
   The **selection set displays as a row** in the selection box.
3. In the dialog, select the chamfer Type:
4. Adjust the **Distance or Angle** values for the chamfer:
5. Optional: For the Two Distance chamfer type, click the Flip icon to flip the first and second sides.
6. Select a Corner Type:
7. Optional: 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.



Create a thin-walled solid

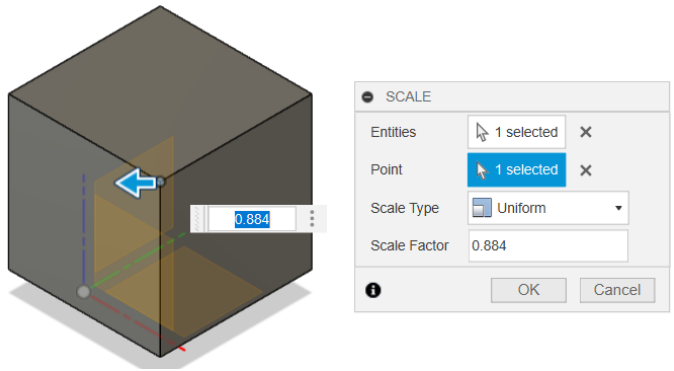
1. Click Design > Solid > Modify > **Shell** .
2. In the canvas or the Browser, **select faces or a solid body**.
3. In the dialog, select the **Direction**:
4. Specify **Inside** Thickness and **Outside** Thickness:

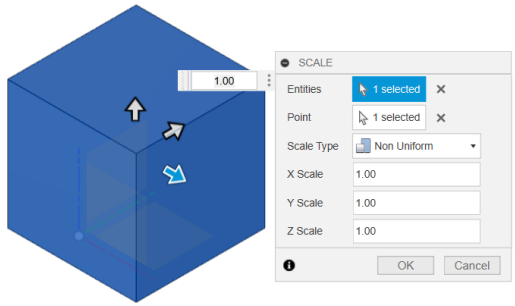
   Use the shell manipulator handles in the canvas, or enter exact values.



Scale components, bodies, or sketches

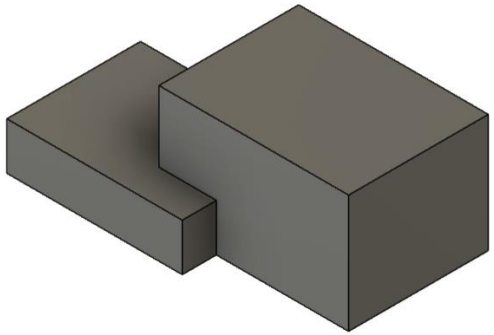
1. In the Design workspace, Solid tab, select **Modify > Scale**.
2. Select **the body or bodies** to scale, and pick a fixed anchor point for the scaling.
3. Choose a Scale Type from the dialog:
   - **Uniform**. Scale the body uniformly on all axes.
   - **Non Uniform**. Scale along the x, y, and z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

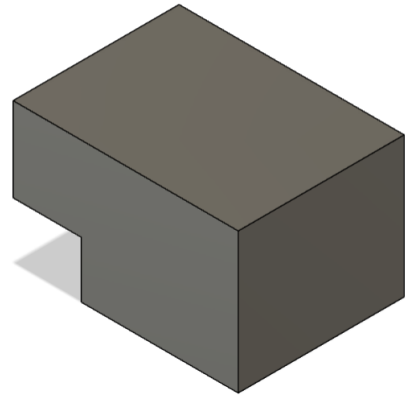




Combine solid bodies

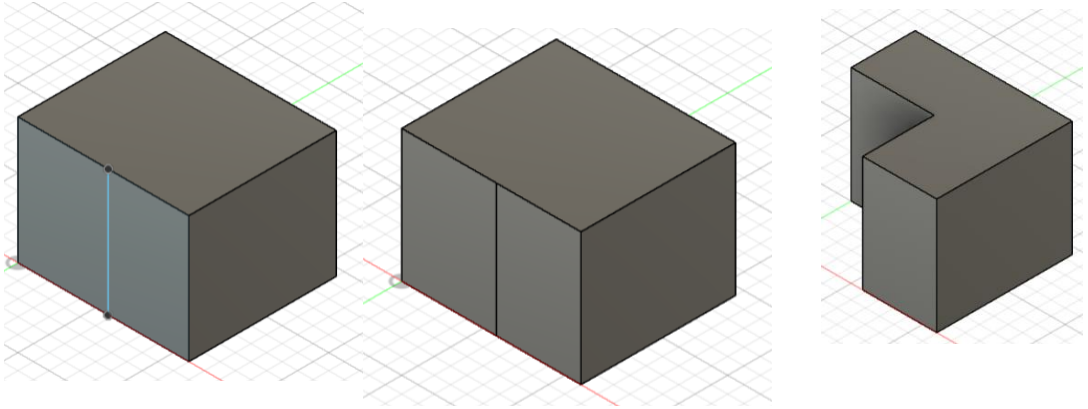
1. Click Design > Solid > **Modify > Combine** .
2. In the canvas, select the **Target Body**.
3. Select **Tool Bodies**.
4. In the dialog, select the **Operation**:
5. Optional: **Check New Component** to create a new component from the result.
6. Optional: **Check Keep Tools** to keep the Tool Bodies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



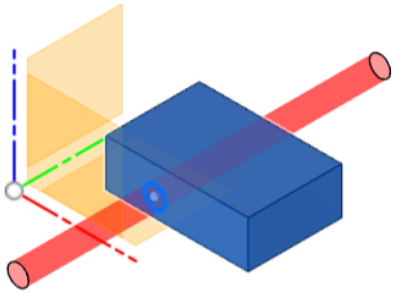


Divide a face/body into multiple faces/bodies

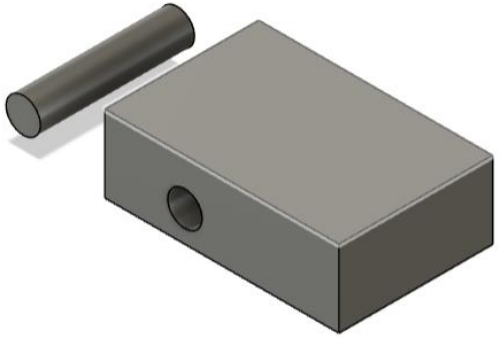
1. In the Design workspace, Solid tab, select Modify > **Split Face**.
2. Select a face to split. (Hold Ctrl (Windows) to select multiple faces.)
3. In the Split Face dialog, click the Splitting Tool field, and do one of the following:
   - Select a **surface or sketch** on the canvas.
4. If you select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



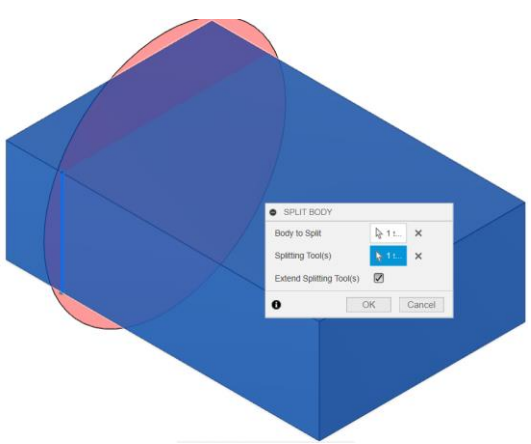
Line as face splitting tool Press pull to modify

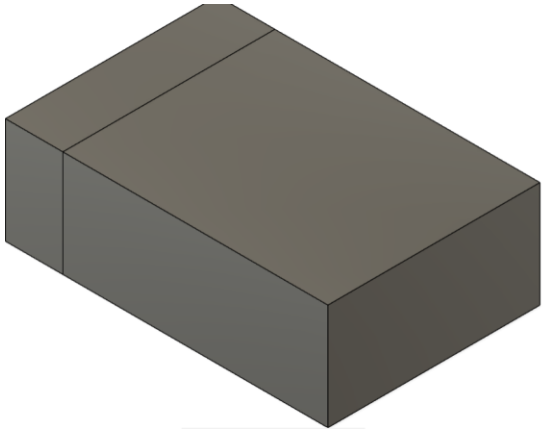


circle as surface for body splitting tool

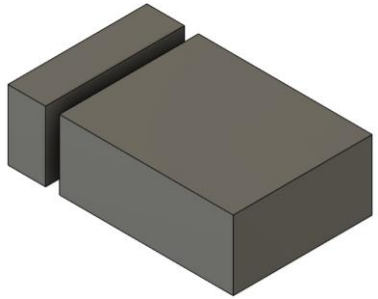


Move a body to separate





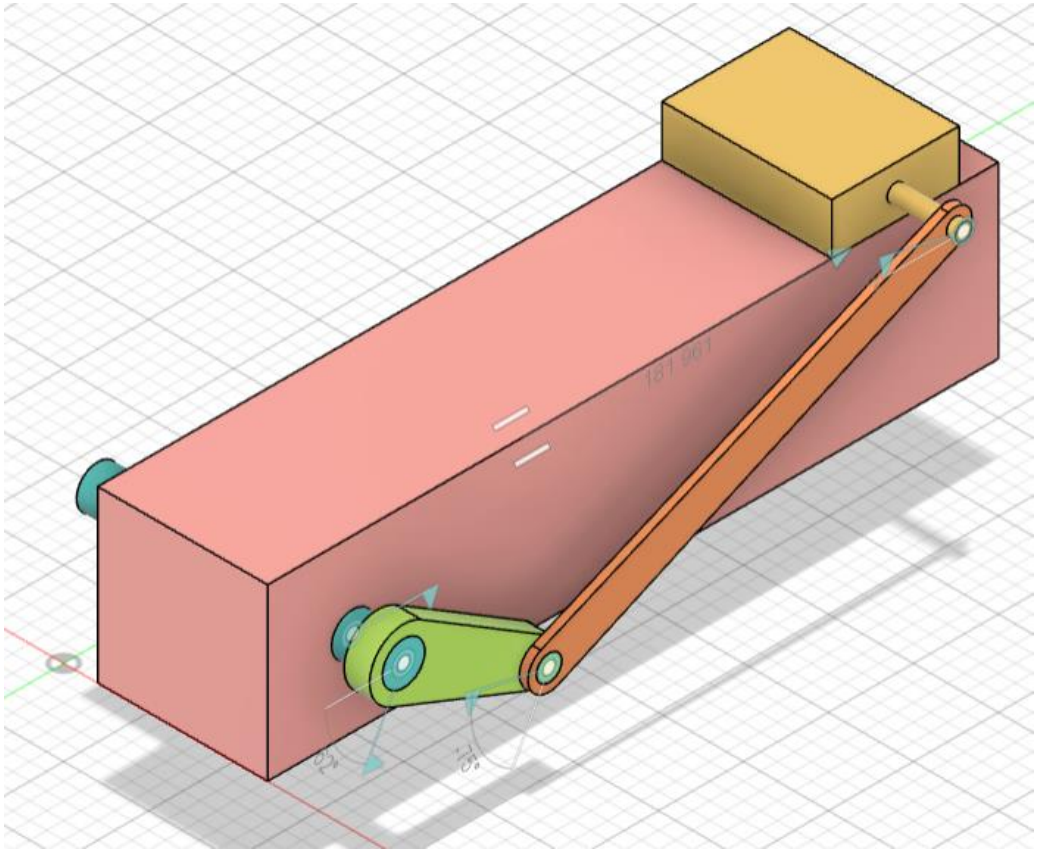
Line as body splitting tool



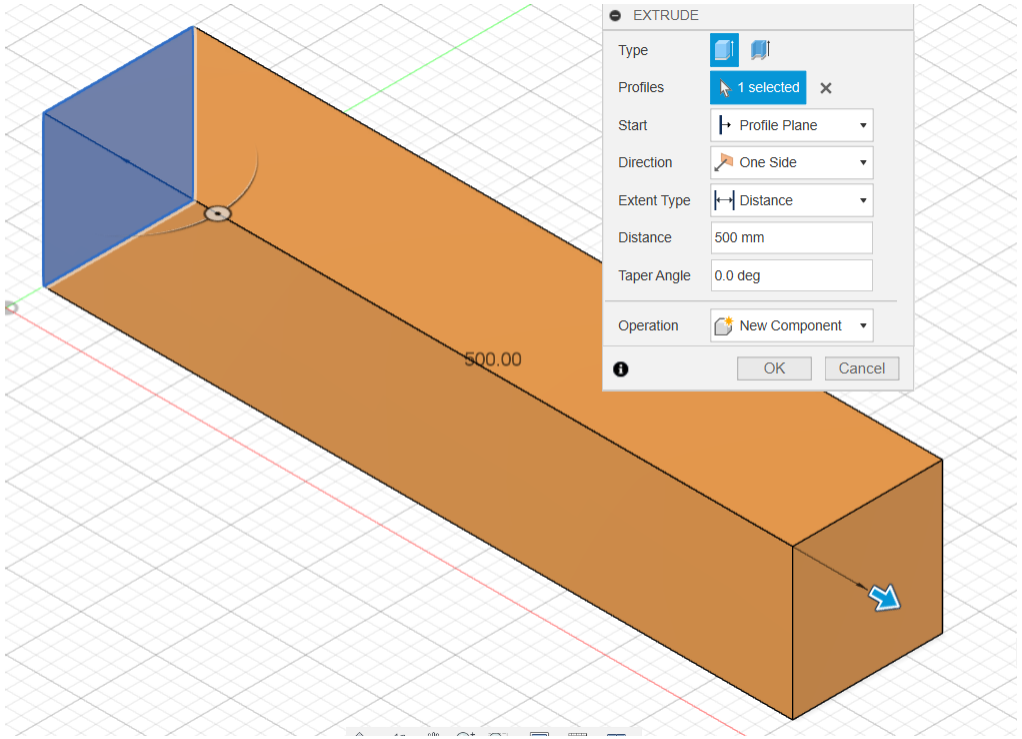
Move a body to separate

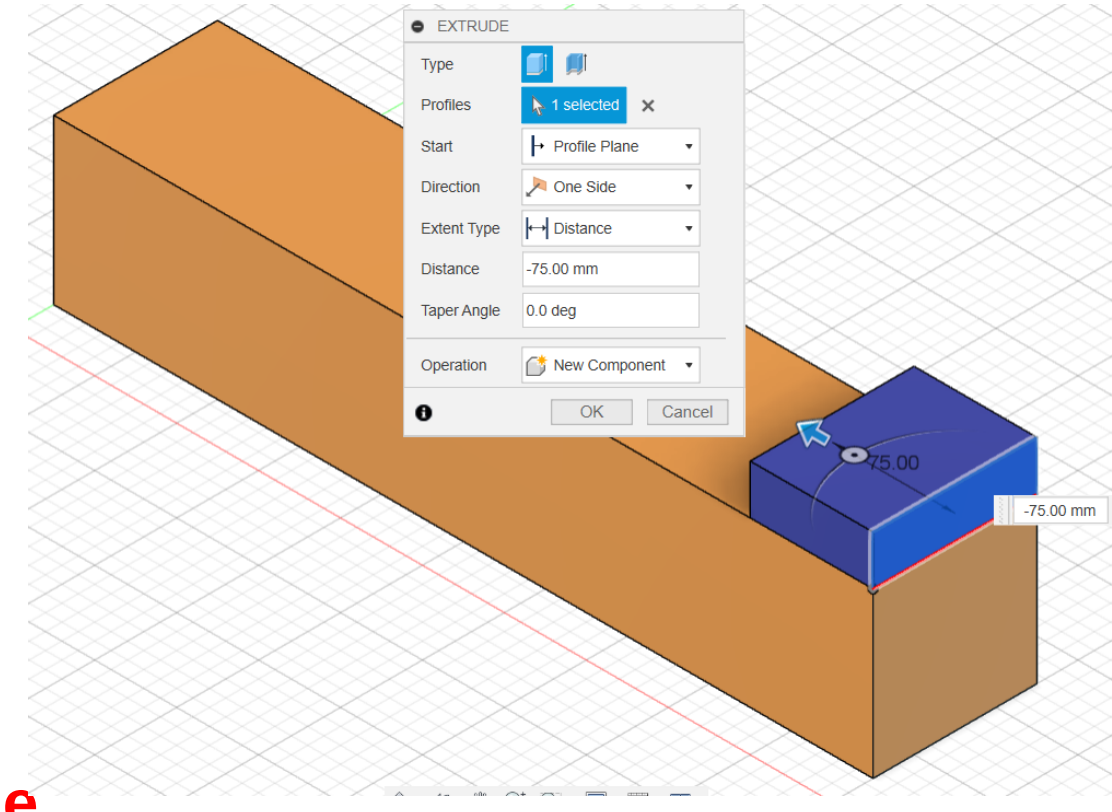
ASSEMBLY IN FUSION 360

Slider crank mechanism



Slider crank mechanism

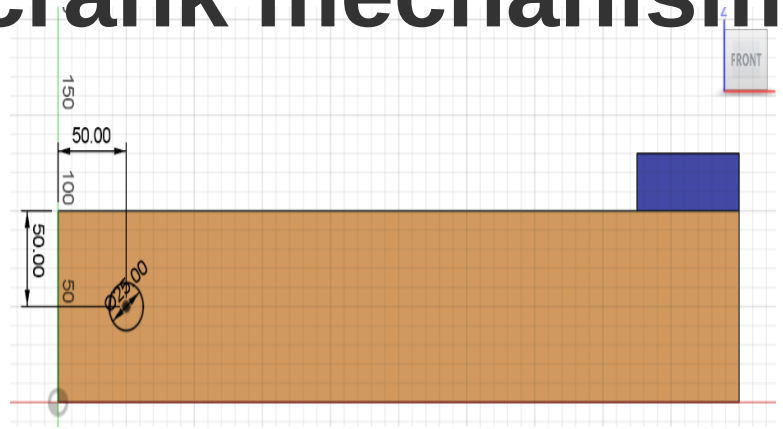


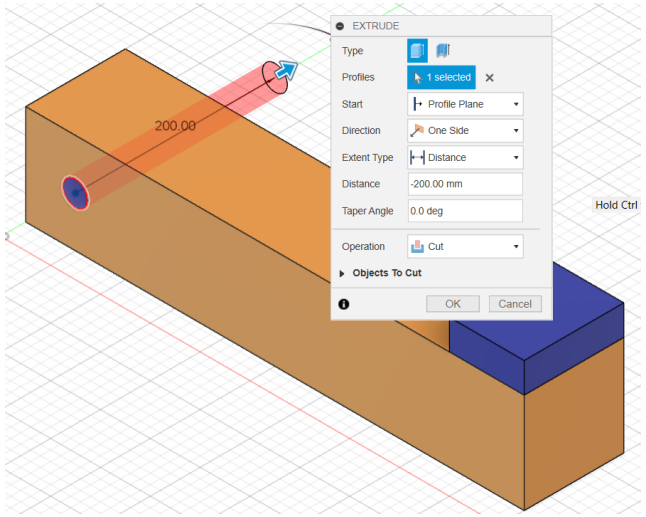


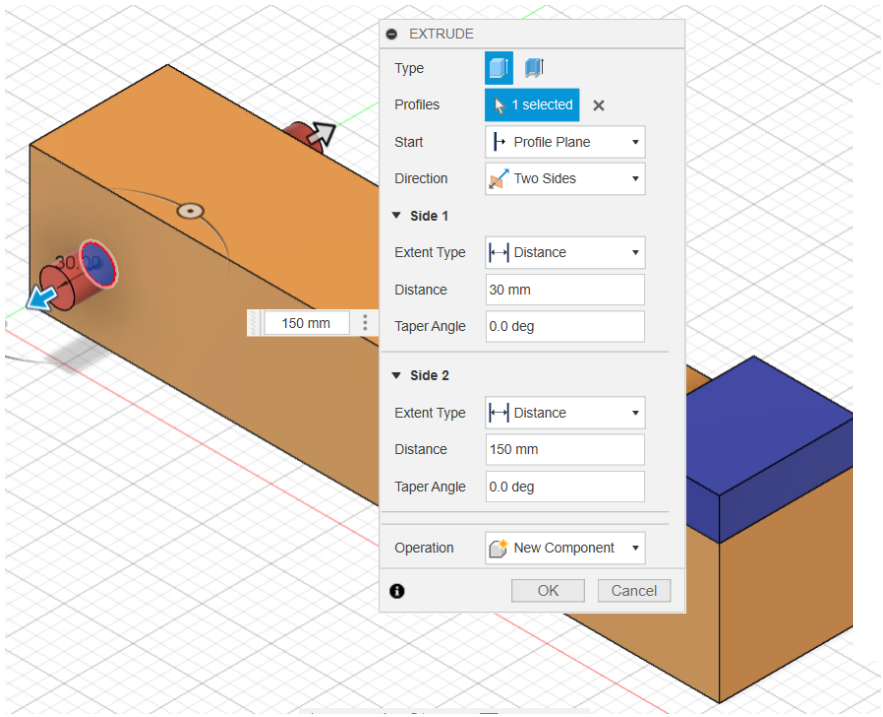
- **Sketch** a **100x100** **rectangle**.
- **Extrude** it to 500 mm.
- Save as **new component**.
- On the **right face sketch rectangle** 100x30.
- **Extrude** by -70 and save as **new component**.

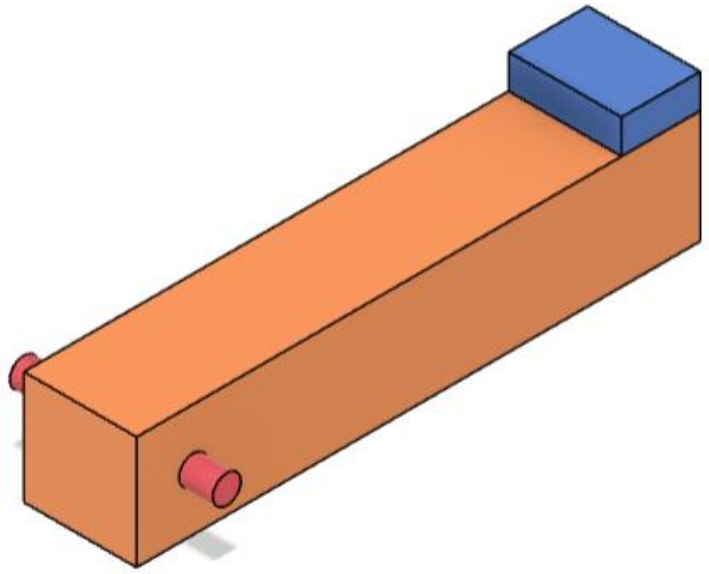
Slider crank mechanism

- **Sketch** a 25 dia **circle** on the **front face**, 50 from the **edges**.
- **Extrude** it to -150 to make hole, **operation: cut**
- Select the **same sketch** and **extrude** to **30** and **150** on **either side** and save as **new component** pin.



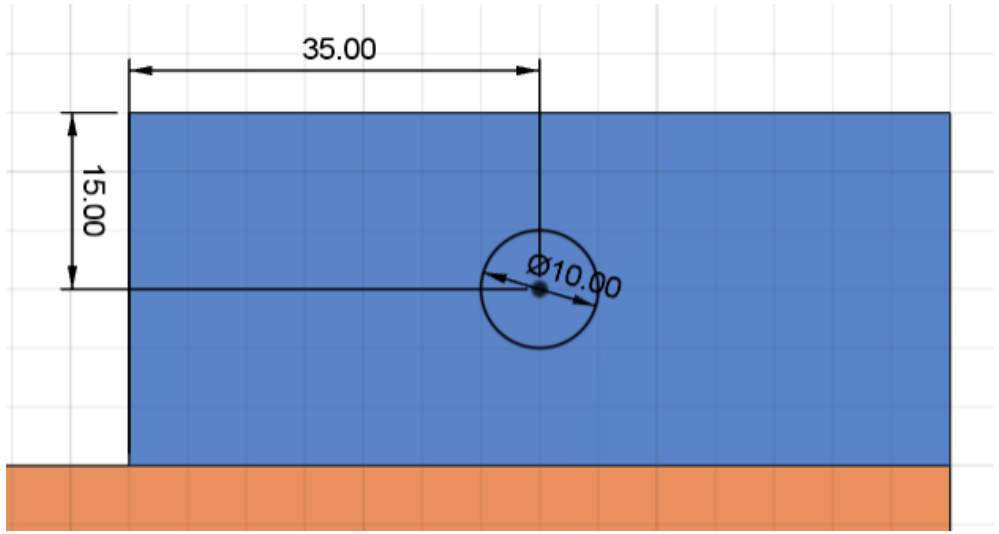


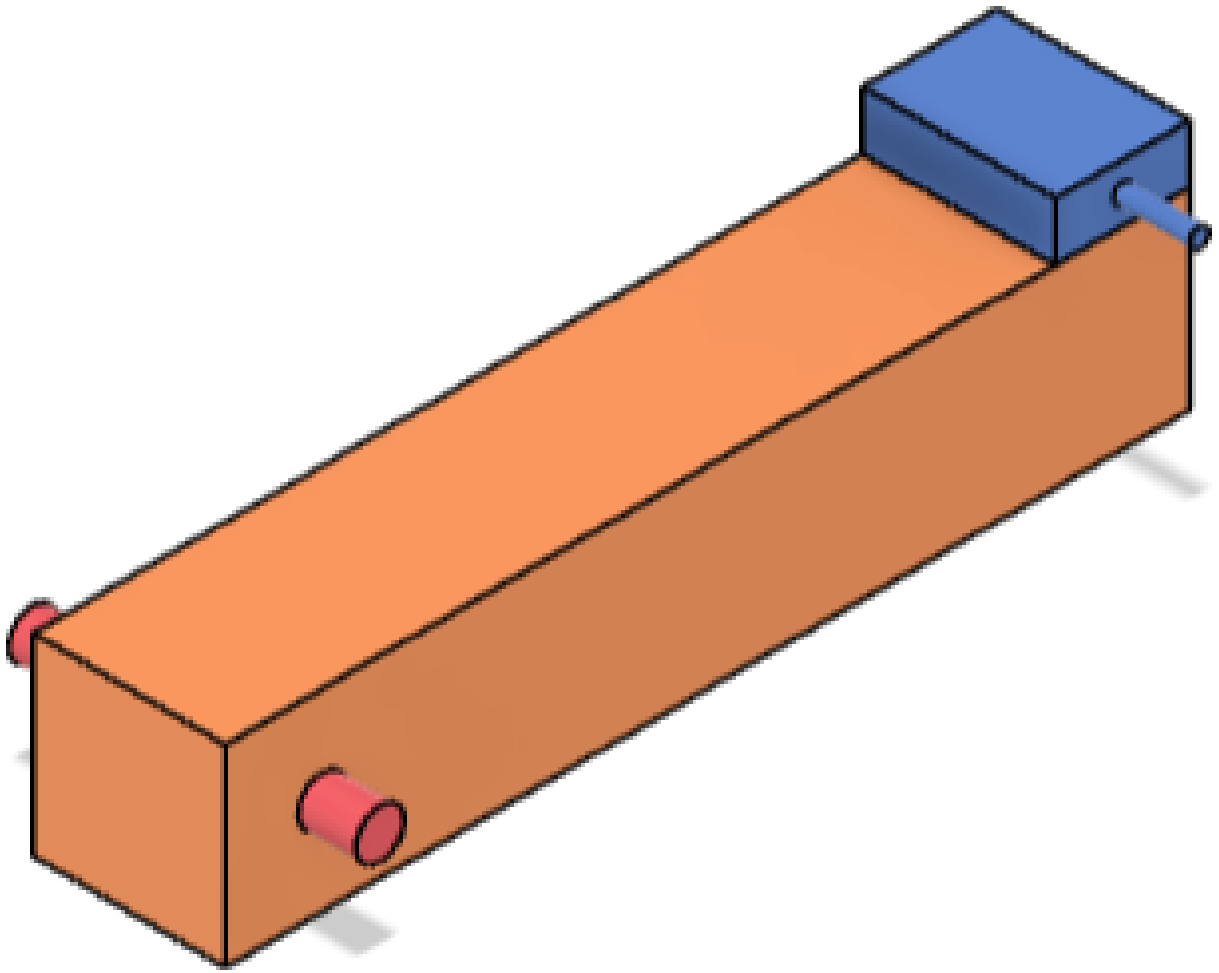




Slider crank mechanism

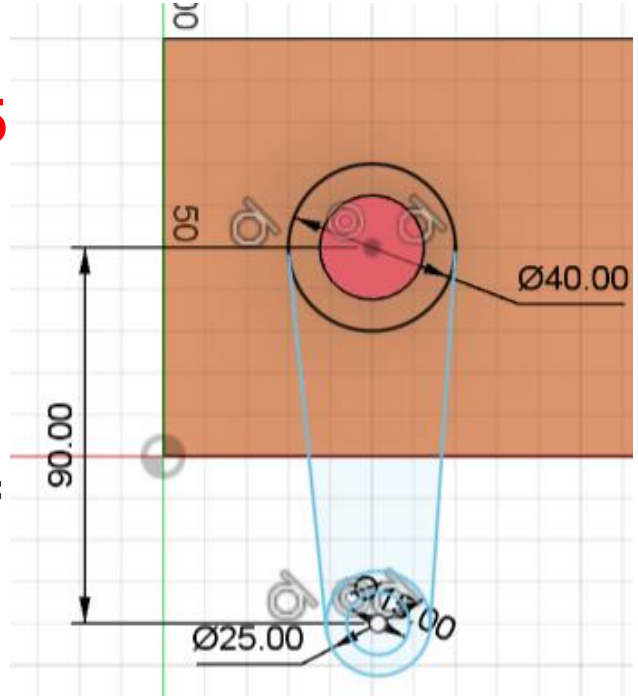
- Sketch a 10 mm dia. circle on the front face of the slider, 15, 35 from the edges.
- Extrude it to 35 to make a pin on the slider: operation: join.

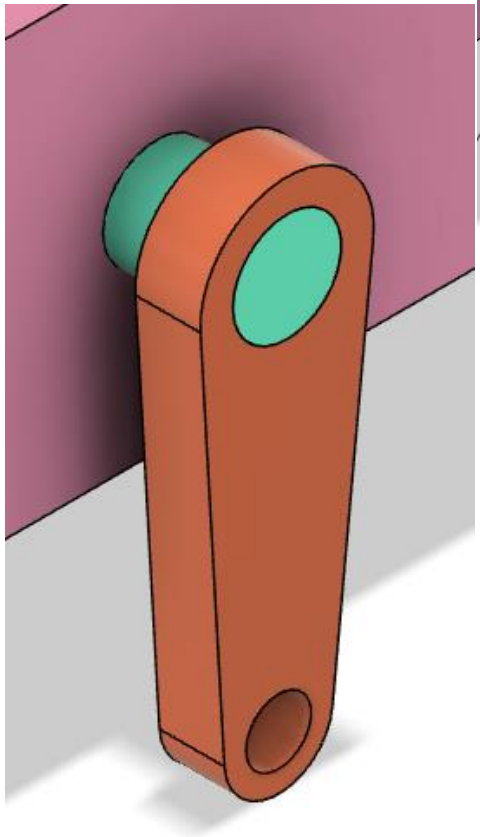


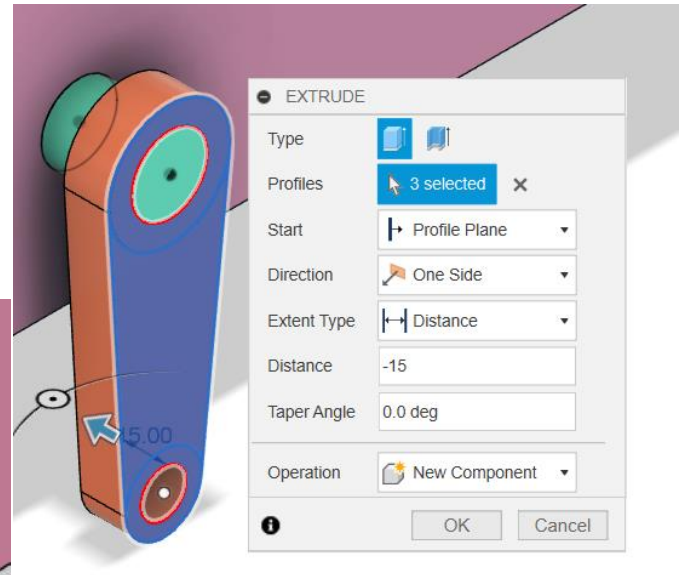


Slider crank mechanism

- On the face of the 25 dia pin
- Sketch concentric circle of 40 dia.
- Below it sketch concentric circles of 25, 15 dia.
- Sketch tangent lines exterior to the circles.
- Center to center distance 90.
- Extrude it to -15 as new component crank.

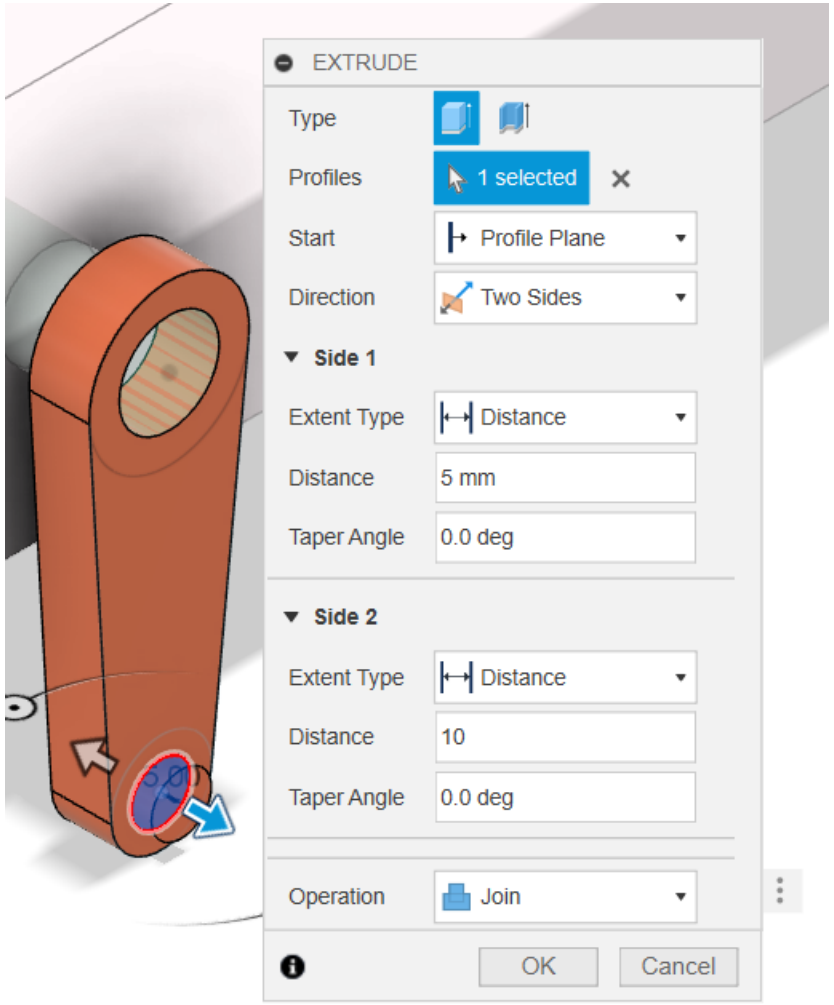


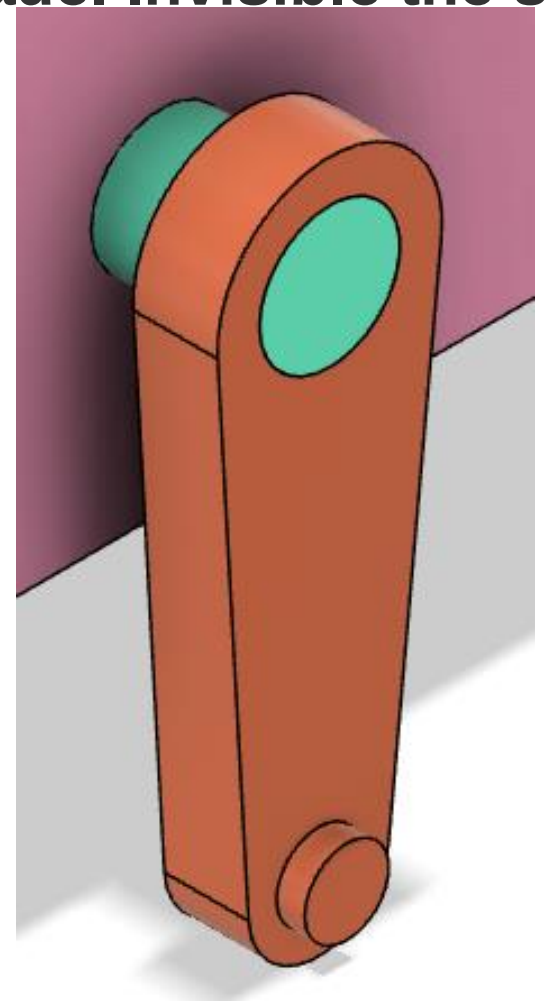




Slider crank mechanism

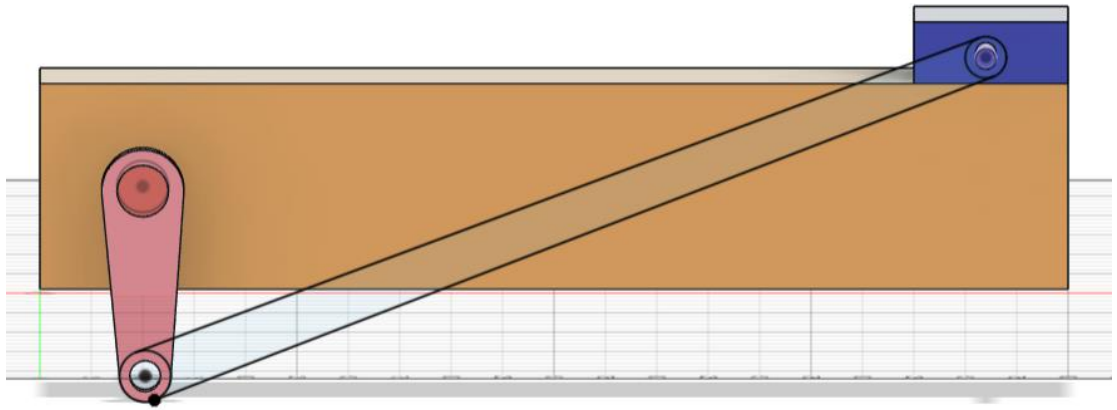
- Activate **crank sketch**
- Select the **bottom inner circle of the crank hole** and **extrude** 5 and 10 on either side. Operation: **join**: **bar pin** is made. Invisible the sketch.

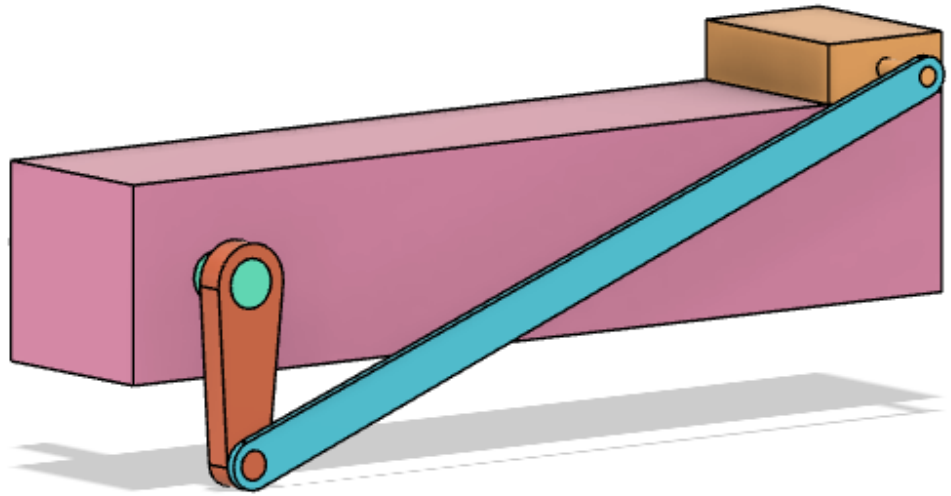


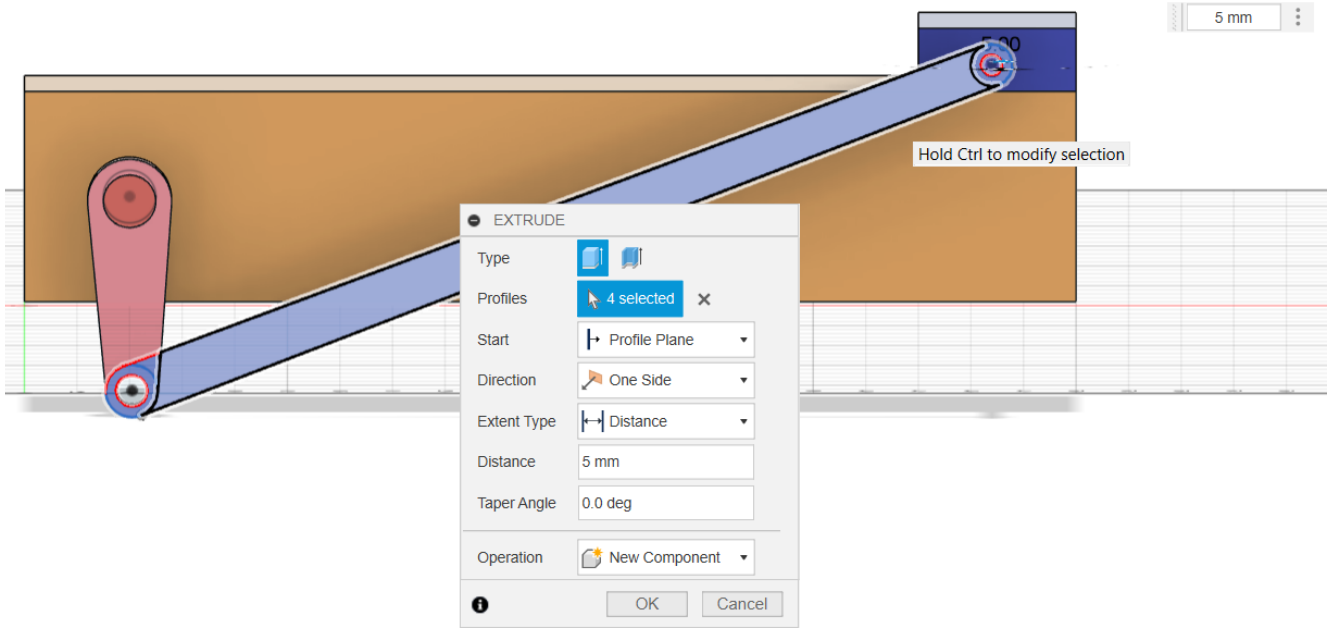


Slider crank mechanism

- Select the **crank pin face** plane Sketch concentric **circle** of 25 dia on the **crank** and 20 dia on the **slider pin**.
- Sketch **lines tangent** to the **outer circles**.
- **Extrude** it by -5 and save as **new component**.

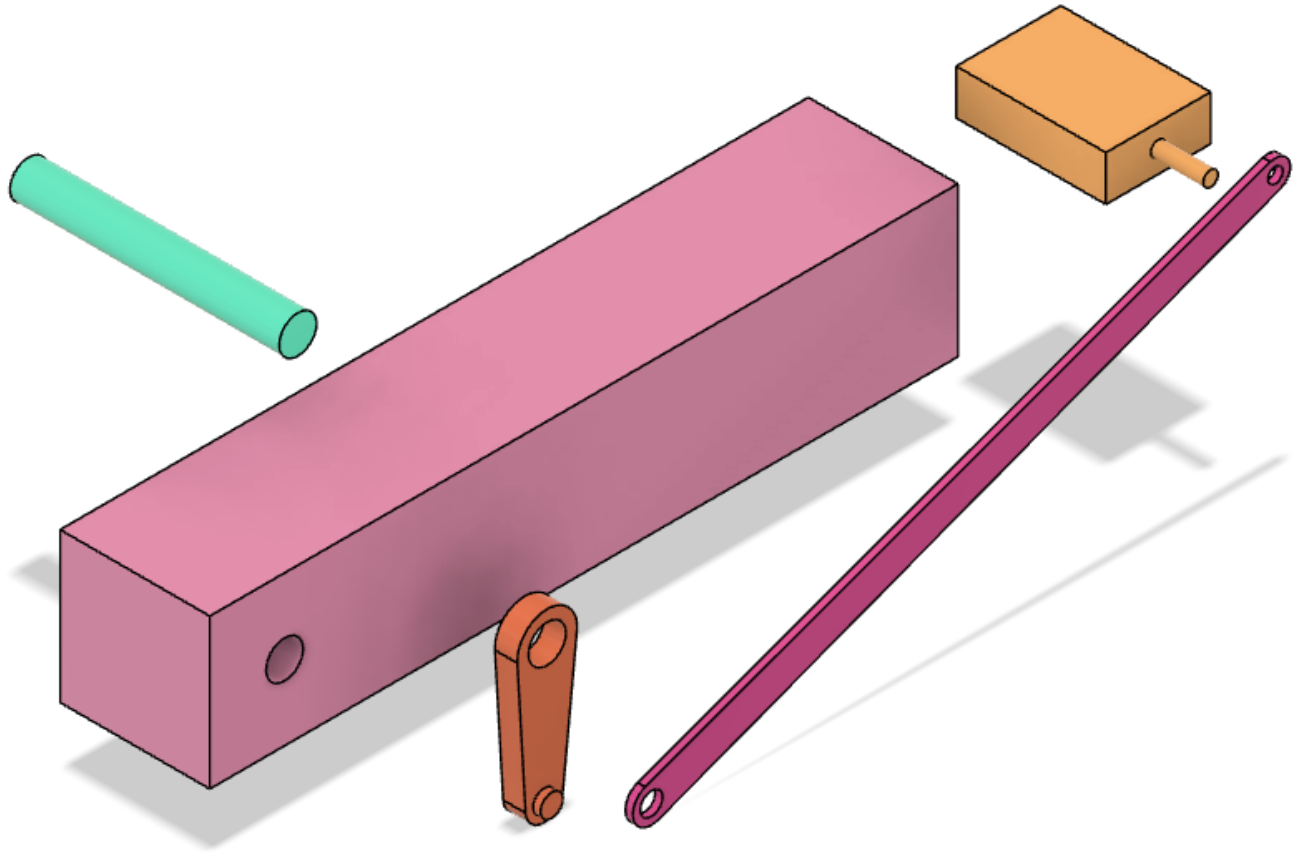






Slider crank mechanism

- Right click **component1** and **ground** it.
- All other **components** can be **separated**.

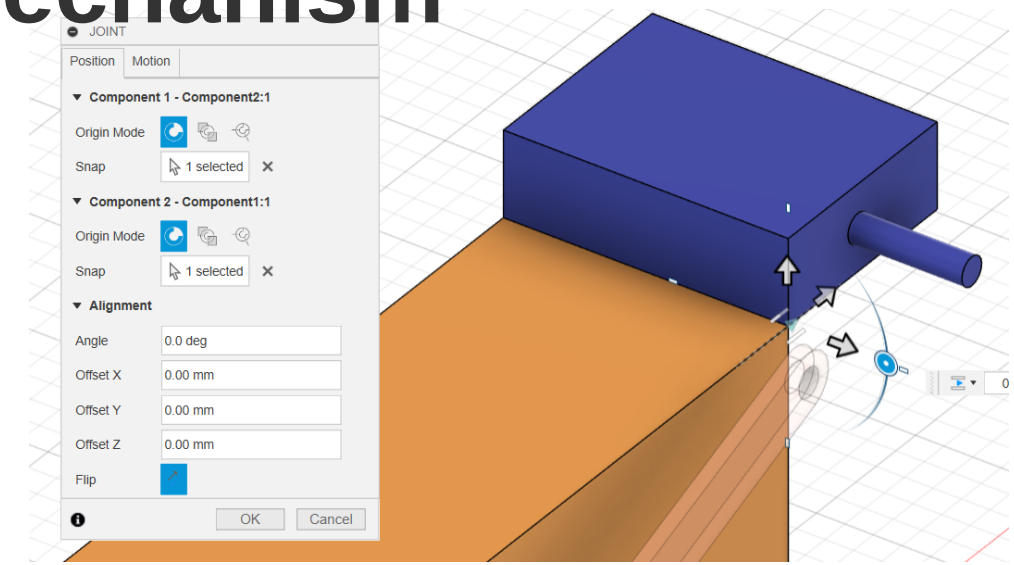


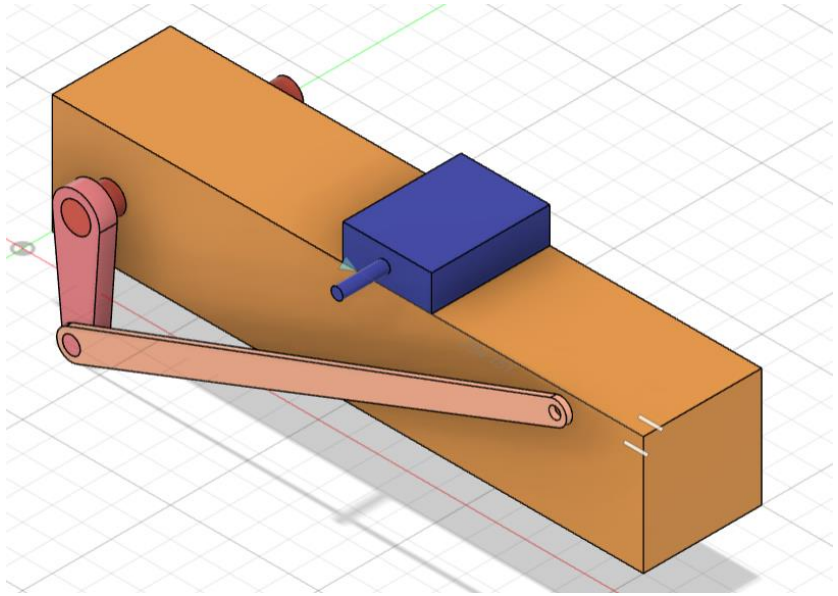
Joints

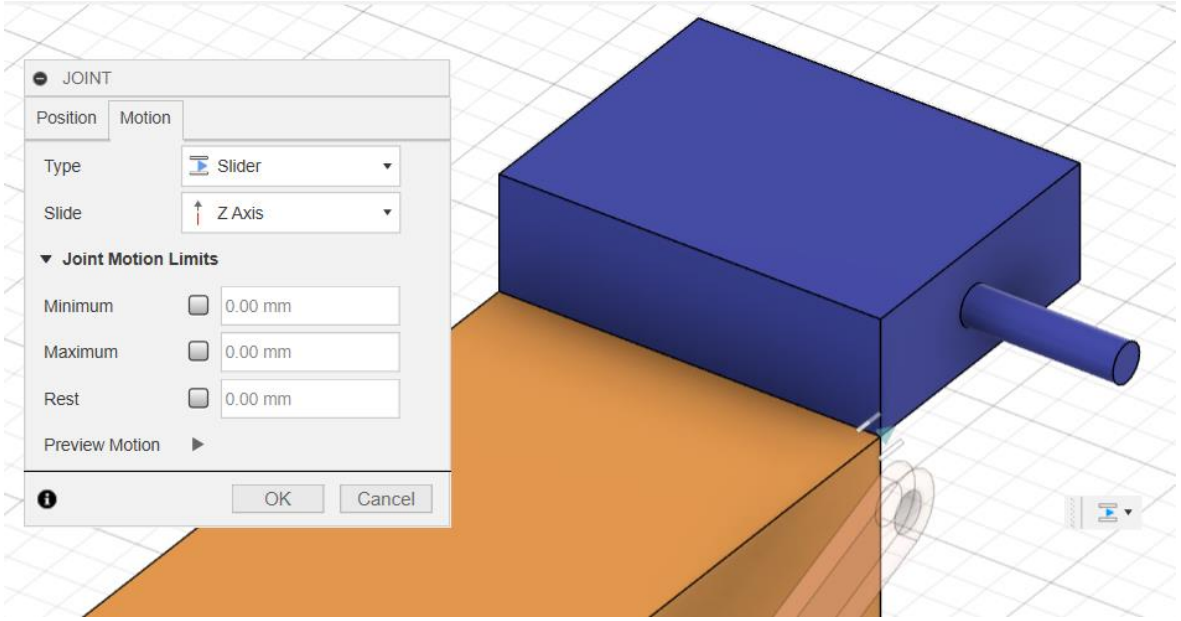
1. Slider(slide) & body.
2. Pin(revolute) & body.
3. Cam(revolute) & pin
4. Bar (revolute) & slider pin
5. Bar (revolute) & cam

Slider crank mechanism

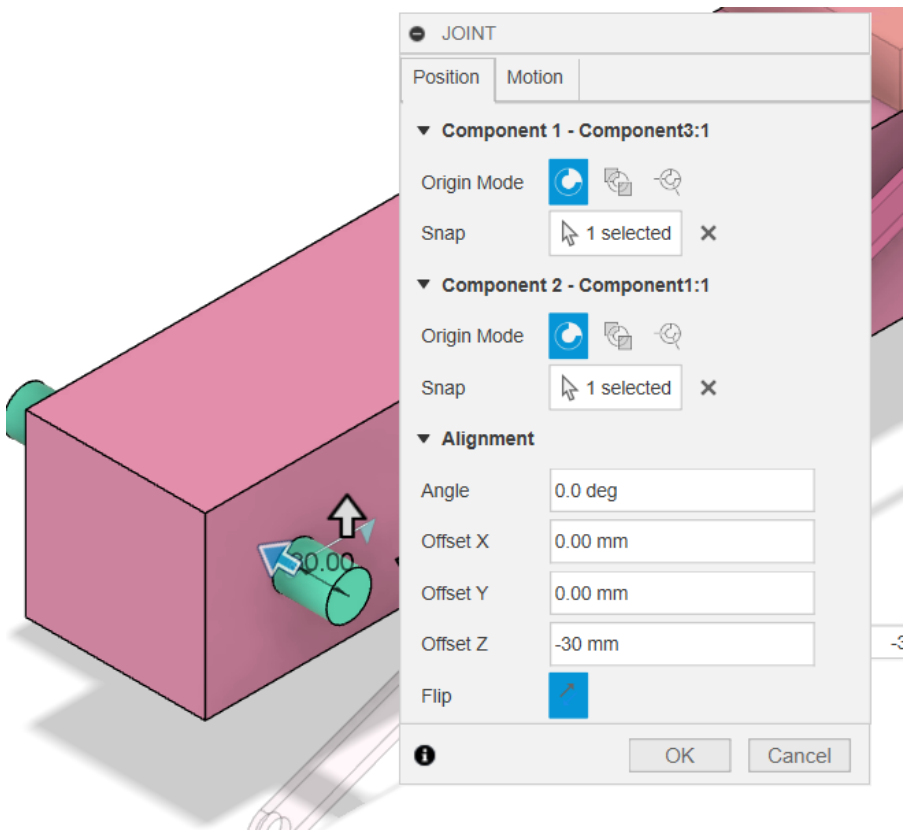
- Joint from assemble.
- Snap parallel edges of component2 (slider) & 1.
- Motion as slider.
- Click the slider, move the mouse to slide on the block.

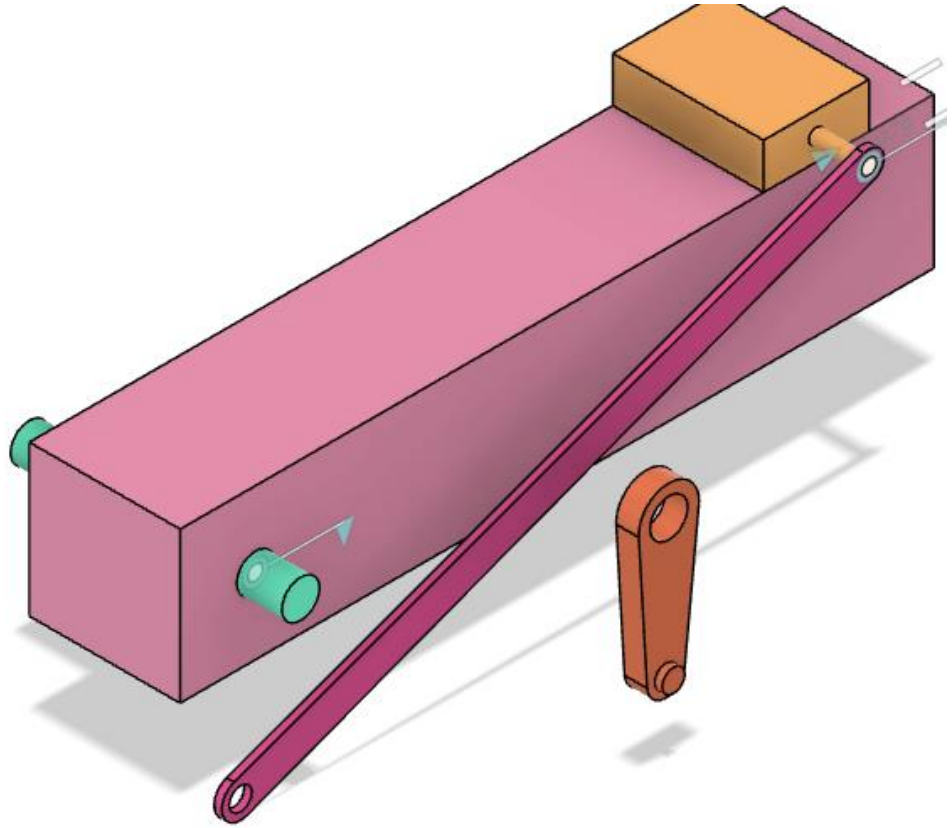






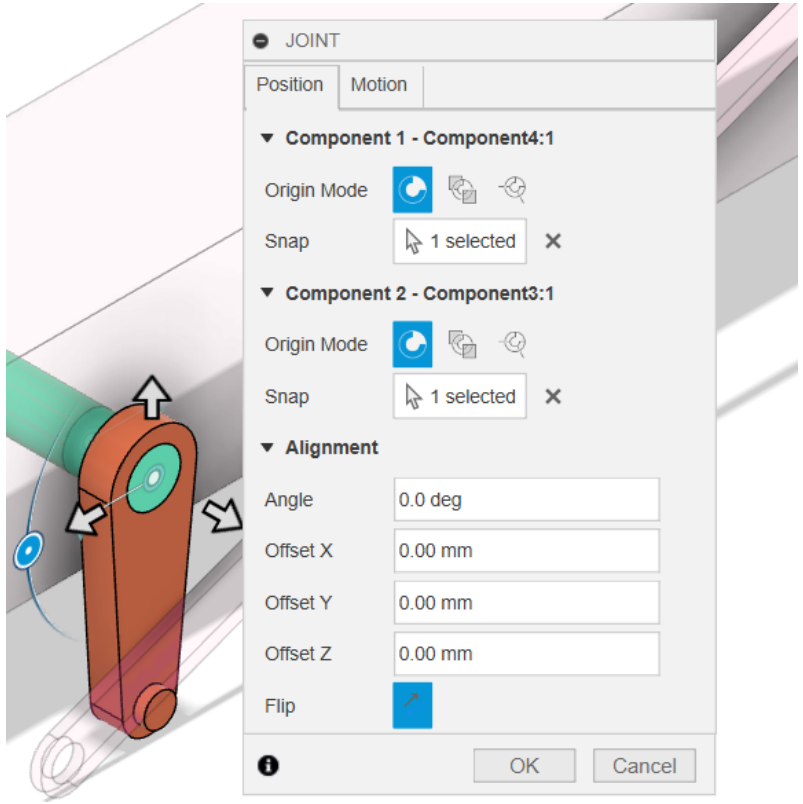
Slider crank mechanism

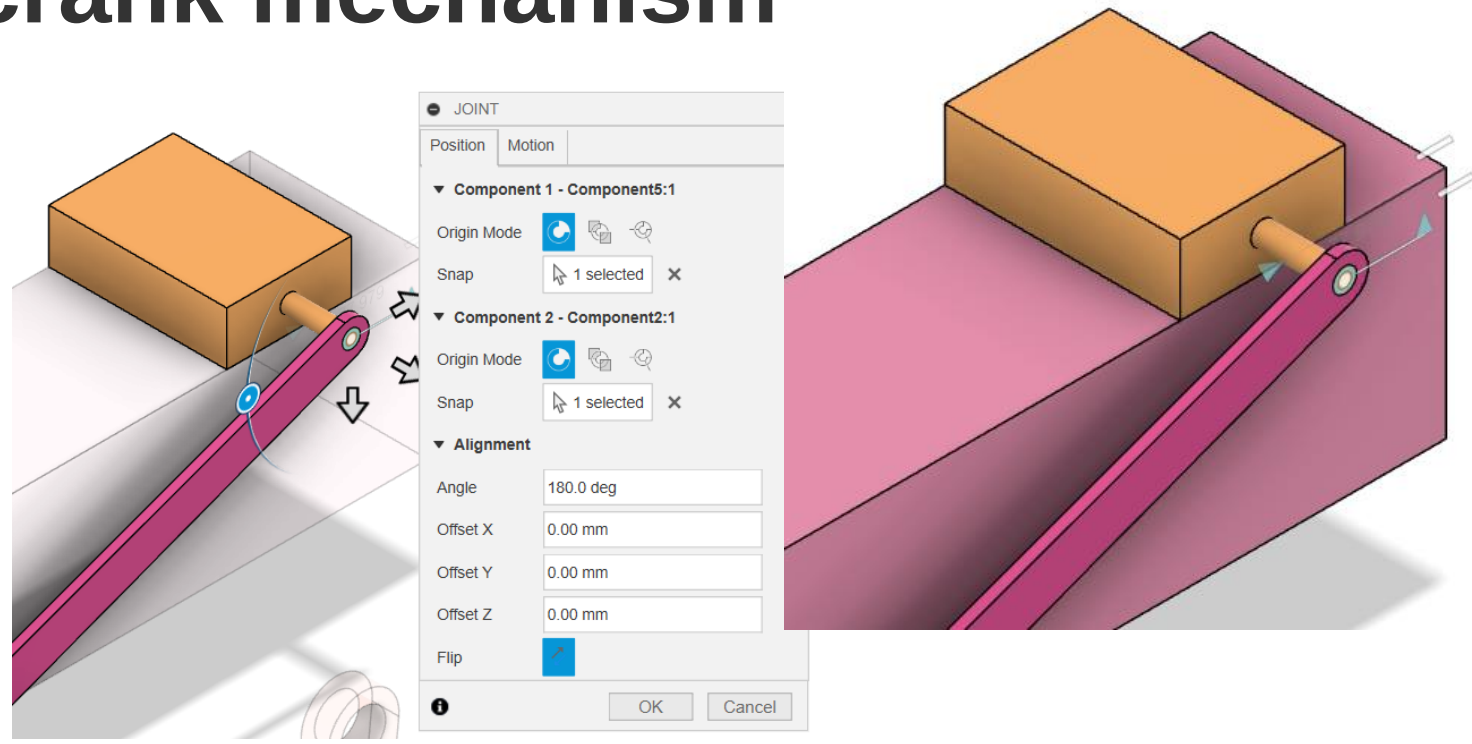




- Select **outer edge of the pin** and **edge of the circle** on the block.
- Select motion **revolute** and **offset z by -30mm**.

Slider crank mechanism

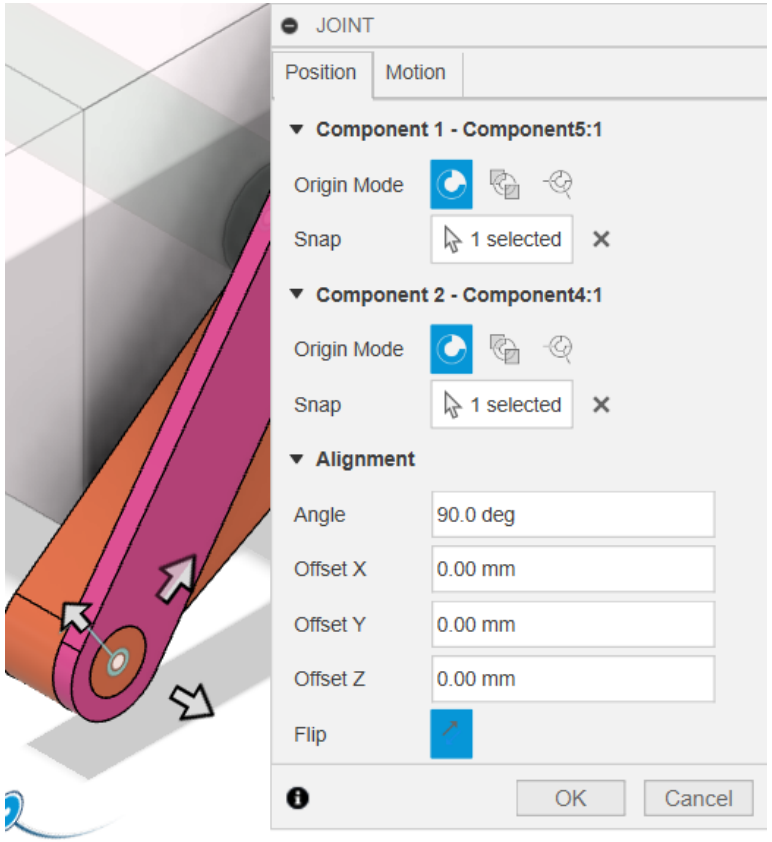




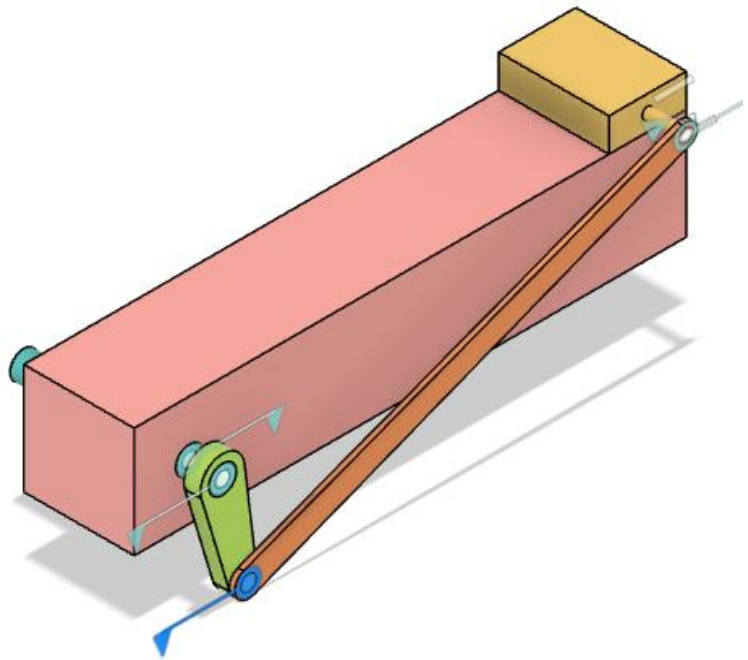
- **Joint** between **crank & pin** select **crank circle outer edge** and **outer edge of the pin.**
- Motion: **revolute**

- **Joint** between bar & slider pin.
- Snap the **outer edge of circle** on the **bar** and **outer edge of slider pin**
- Select motion: **revolve**.

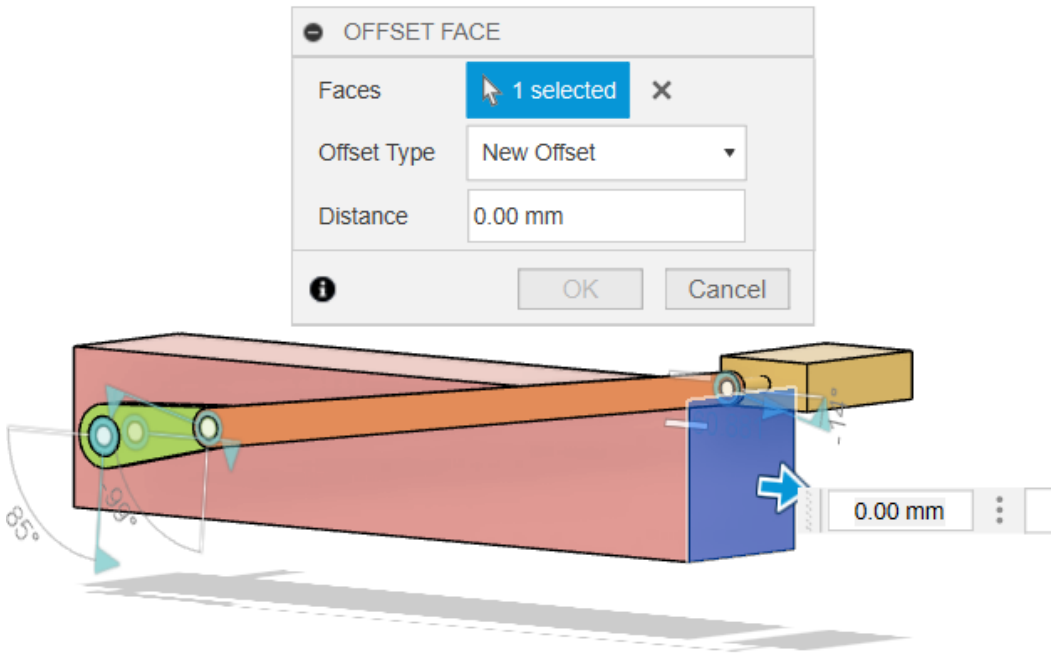
Slider crank mechanism

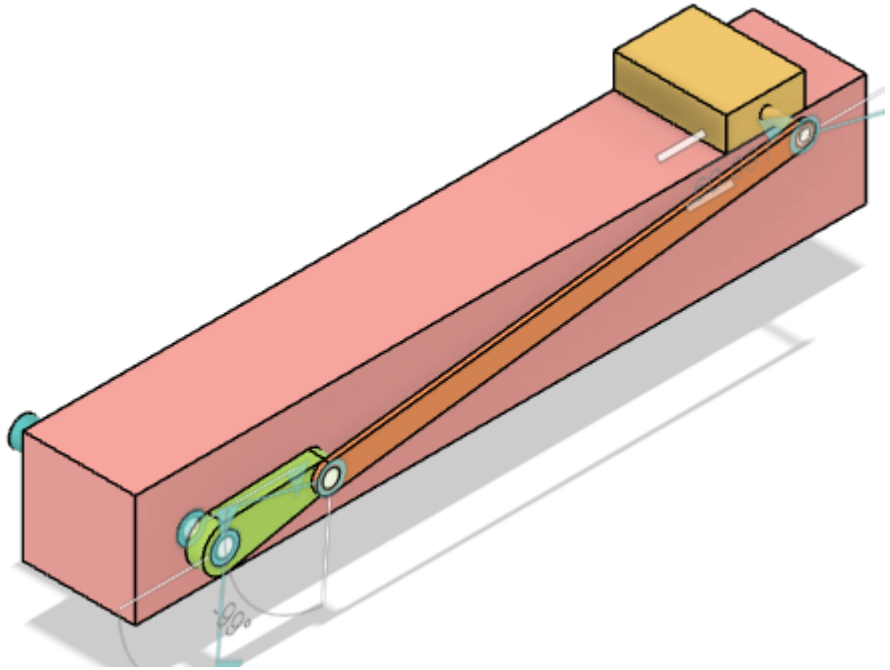


- **Joint** between **crank pin** and the **bar**.
- Select **outer circle edge of the bar** and **outer edge of the crank pin**. Motion: **revolute**



Slider crank mechanism





- **Press pull the face to increase the body length to cover the slider for the extreme position**

Exercise

- **Geneva Mechanism**
- **Quick return mechanism**
- **Parallelogram mechanism**
- **Angular transmission mechanism**
- **scotch-yoke mechanism**
- **Oldham coupling**

Thank you