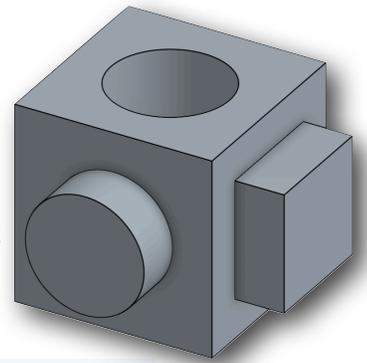


Practice cube - Onshape

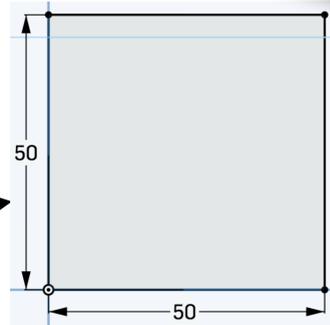


Create a new part in Onshape

1. Create > Document > call it "practice cube" > Create public document
2. Make units millimeters  > Workspace units ... > Millimeters
3. Click the green checkmark to accept settings

Create the practice cube sketch

1. Click "Sketch"  > click on the "Front" plane
2. Rotate view cube so "Front" is facing you
3. Click on "Rectangle" tool 
4. Make square > Enter 50 for the height and width

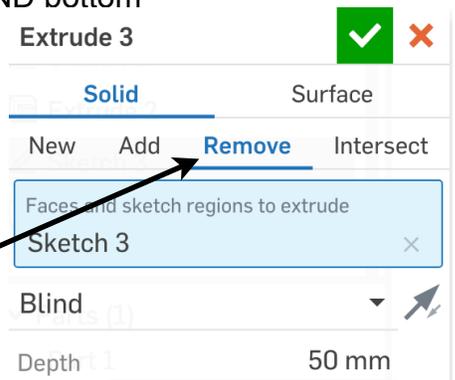
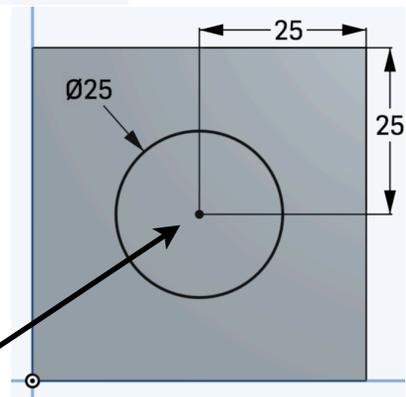


Extrude square into a cube

4. Click on the "Extrude" tool 
5. Enter "50" for the "Depth"
6. Finish sketch by clicking on the green checkmark

Create circle on the front of cube

1. Click on the "Front" of the view cube
2. Click "Sketch"  > click on front face of cube
3. Click on the "Circle" tool 
4. Create a circle near the center with 25mm diameter
5. Click on the "Dimension" tool 
6. Dimension the center point of the circle 25mm from the top AND bottom
7. Extrude the circle 12.5 mm
8. Click the green checkmark to finish extrude



Create hole through the top

1. Create Sketch on "Top" plane > repeat steps above for creating a circle
2. Extrude circle 50mm
3. Be sure to click on "Remove" to make the hole
4. Click the green checkmark to finish

Create square on the right plane

1. Create "Sketch" on right plane
2. Click on "Rectangle" tool > create 25mm x 25mm square
3. Dimension each side 12.5mm from outside of square
4. Extrude square 12.5mm
5. Click green checkmark to finish

