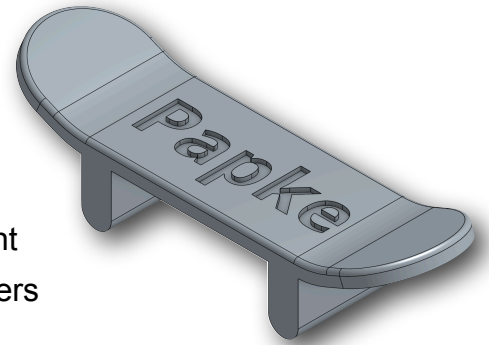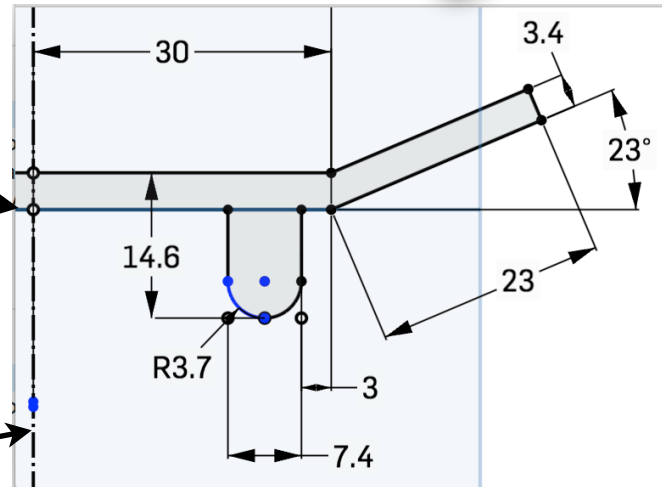# Tech Deck - *Onshape*

Create a new part in Onshape
1. Create > Document > call it "tech deck" > Create public document
   a. Make units millimeters: ☰ > Workspace units ... > Millimeters

Create the **tech deck sketch**
1. Sketch > click on the "Front" plane (rotate view so "Front" is facing you)
2. Sketch and dimension the tech deck outline as shown in picture (first point is the origin)
3. Round the bottom of the "wheel" by using the fillet tool with a 3.7 radius ⟶

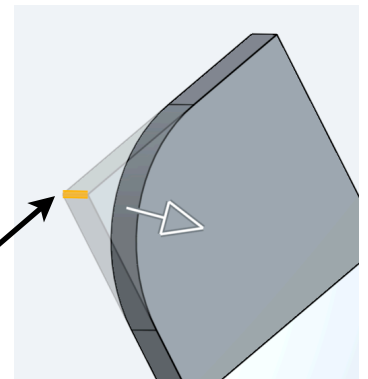Create the **other half** of the tech deck
1. Click on the "Mirror" tool ⟶
2. "Select a mirror line" means to click on the center vertical line ⟶
3. "Select entities to mirrored" means to click on each line of the sketch. You should see the lines appear on the opposite side to create the completed tech deck
4. Finish sketch by clicking on the green checkmark

Extrude the tech deck by 28 mm
1. Click on the "Extrude" tool
2. Click anywhere on both sides of the tech deck and type 28 for "Depth"
3. Click the green checkmark to finish extrude

Round some of the **edges** with the "Fillet" tool ⟶
1. Click on the fillet tool > click on each of the four corners of the tech deck
2. Enter 14mm for the Radius
3. Use the fillet tool at 20mm to round the transition from the board to the tail
4. Click the green checkmark to finish
5. Use the fillet tool at 1mm to round the edges of the entire board - click fillet > click the top of the board > 1mm

Add your name to personalize
1. Create new sketch: Click "Sketch" > click on top of tech deck
2. Click "Text" tool and drag a rectangle filling the top of the tech deck
3. Type your name (select bold so name shows up better)
4. Resize name (dimension tool) and center so name fits > Extrude name 1mm