Fusion 360 Beginner Step-by-Step Guide: Making a Name Tag/Keychain
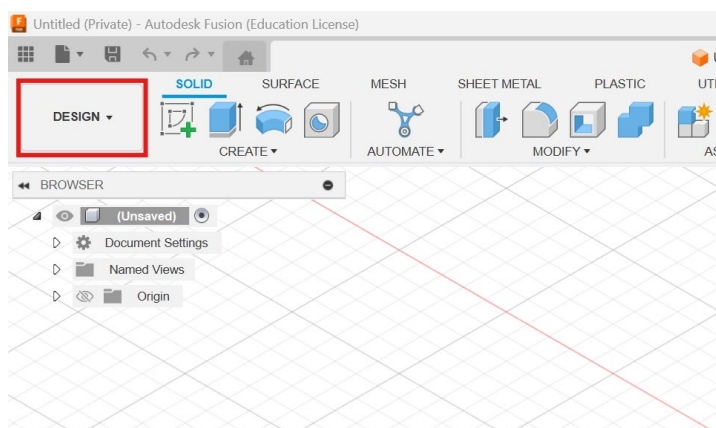
Objective:

Learn to navigate the Fusion 360 interface and use basic sketching, dimensioning, extruding, and filleting tools to create a custom name tag or keychain.

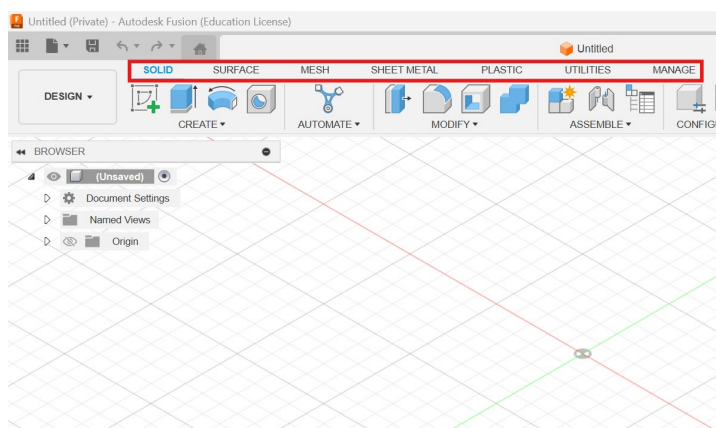## Step 1: Getting to Know the Interface

Open Fusion 360 and create a new project.

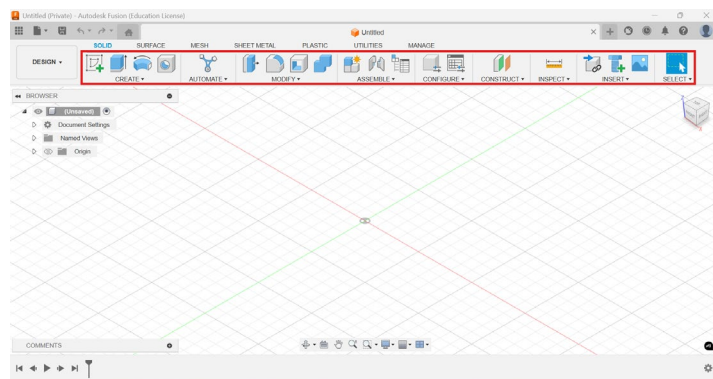Check that the workspace is set to 'Design'.
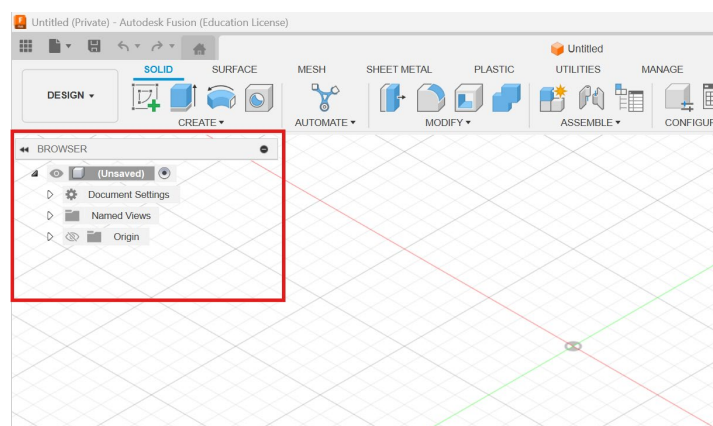

Navigate the following areas:

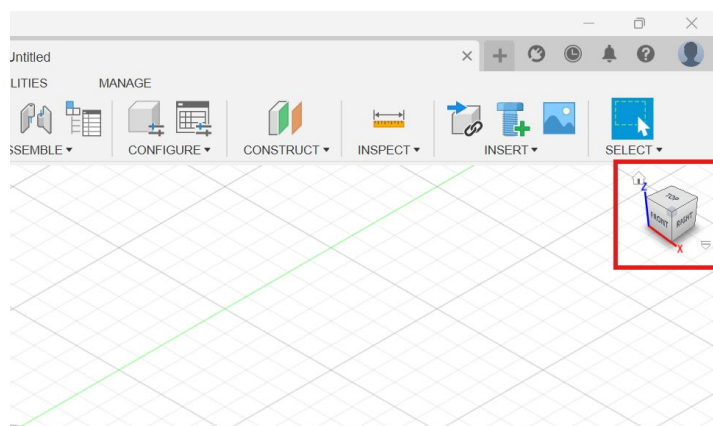- Top Bar: Access tools under tabs like Solid, Surface, Sheet Metal.

Tools Bar: Row of icons for tools and dropdown menus.

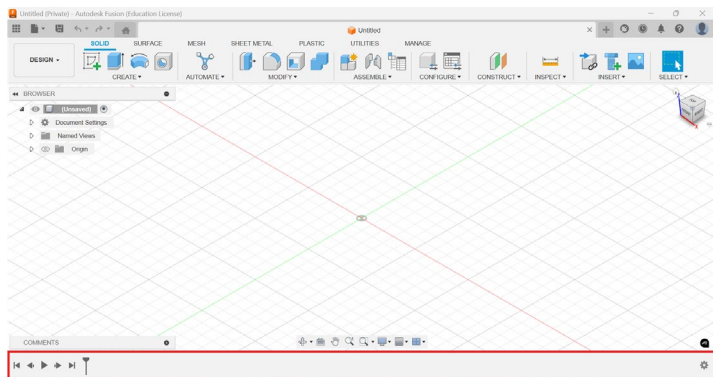- Project Browser (Left Panel): View and organize sketches, components, and bodies.

- ViewCube (Top-Right): Click or drag to change perspective. **Tip:** Press Ctrl + Middle Mouse to pan, Shift + Middle Mouse to orbit.
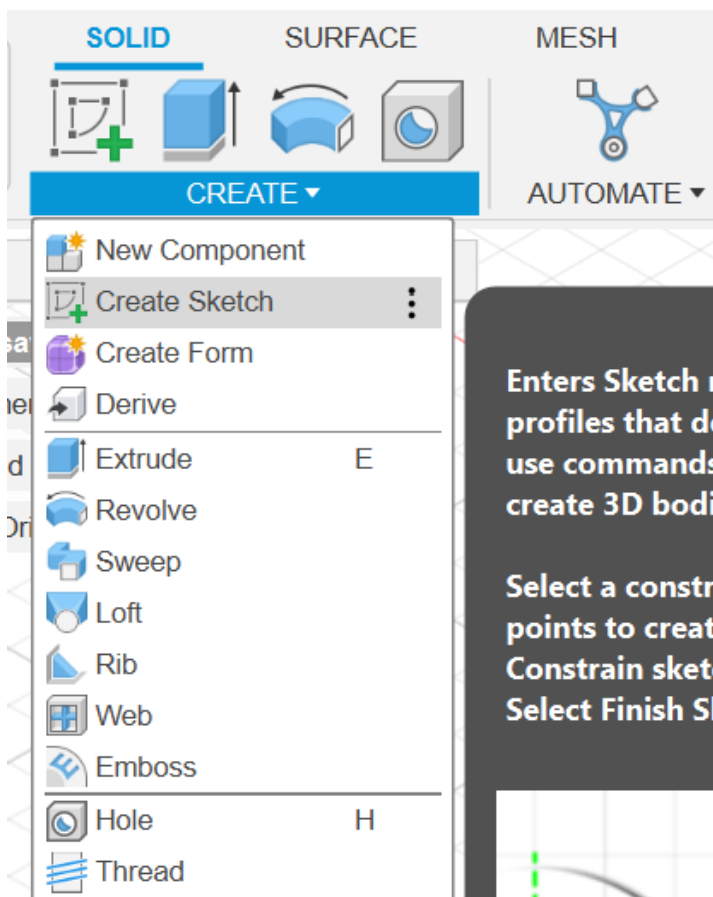

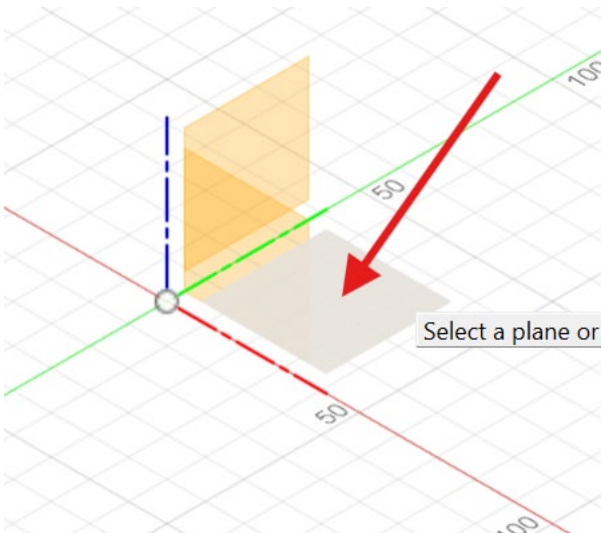- Timeline (Bottom): Tracks your design history for easy edits.

Step 2: Start a New Sketch

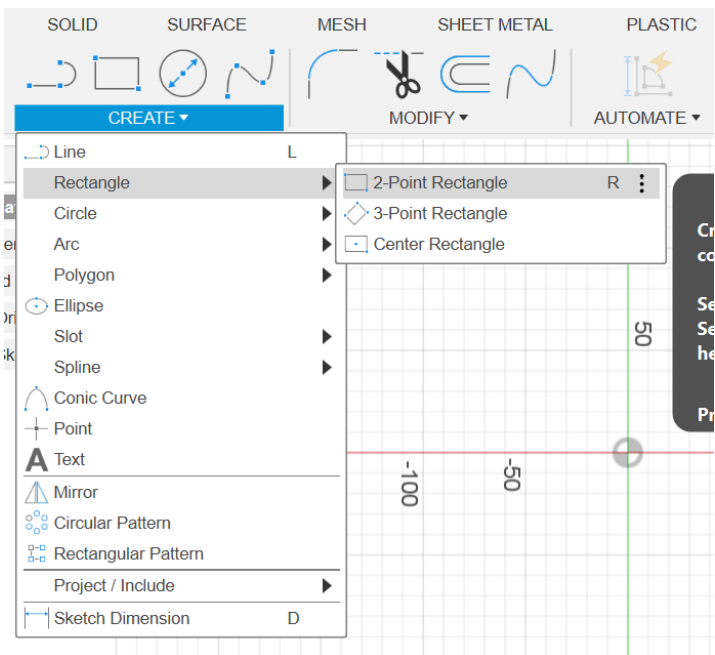Under Solid, go to Create > Create Sketch.

Choose the bottom plane as your sketch surface.

Step 3: Draw the Rectangle (Keychain Base)

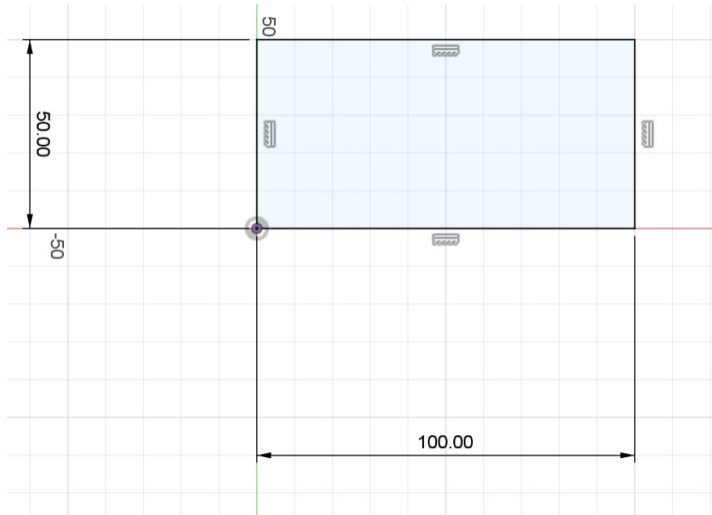Go to Create > Rectangle > 2-Point Rectangle.

Click the **origin** (0,0) and drag your mouse to the upper left.

Input:

- Enter the Height: 50 mm
- Press **TAB**
- Enter the Width: 100 mm
- Press Enter to place the rectangle.

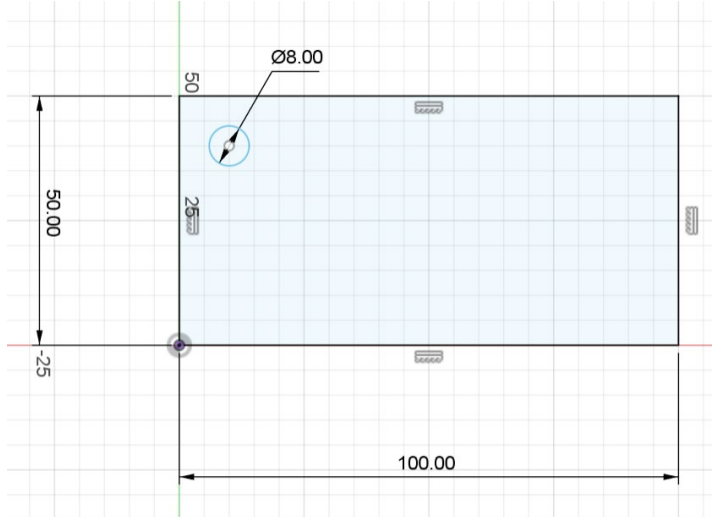# Fusion 360 Beginner Step-by-Step Guide: Making a Name Tag/Keychain

Make sure the rectangle turns black (fully defined). If it's blue, add Sketch Dimensions as needed.

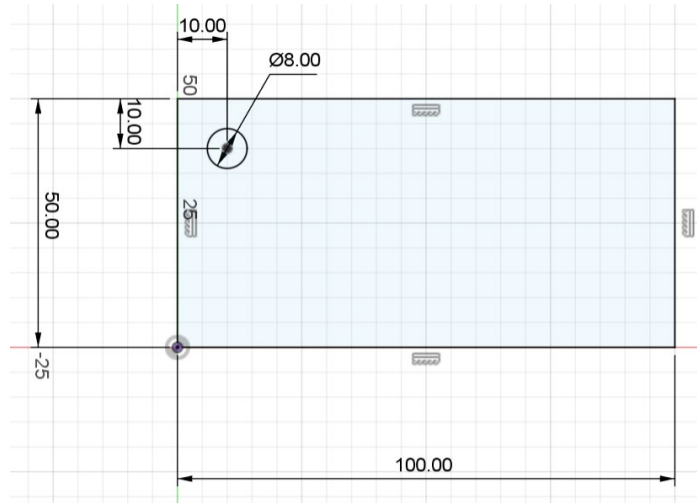## Step 4: Add a Hole (Circle)

Go to Create > Circle > Centre Diameter Circle.

Click inside the rectangle and set the diameter to 8 mm.

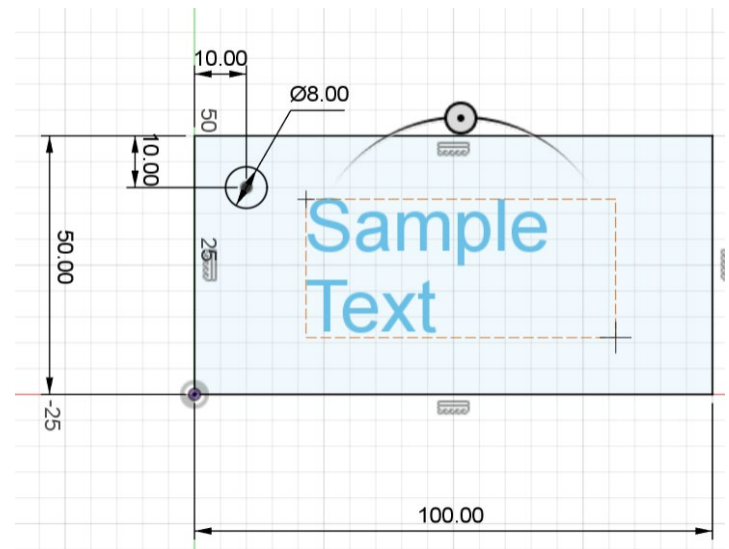Use the Sketch Dimension tool to fully define the circle. Create > Sketch Dimension:

- Click the centre of the circle → then click the left vertical edge → set to 10 mm.
- Then click the centre → then click the top horizontal edge → set to 10 mm.

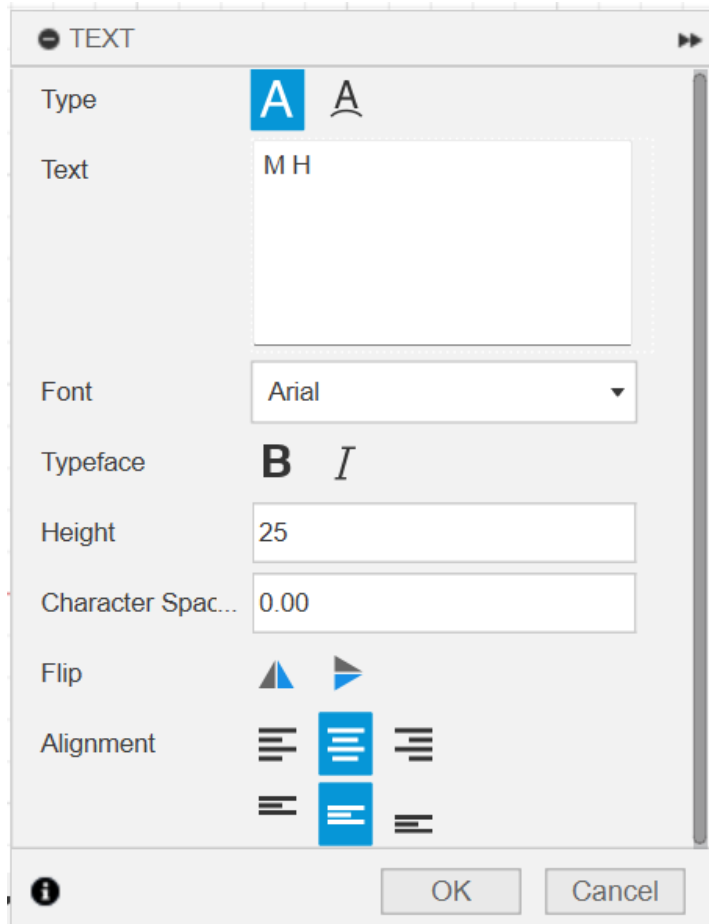## Step 5: Add Text

Go to Create > Text.

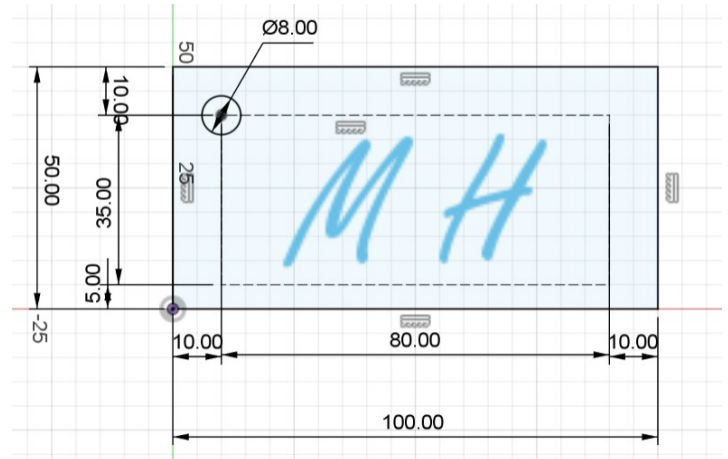Click and drag a text box inside the rectangle (roughly from the top-left to the bottom-right area).

In the text dialog:

- Replace sample text with your initials or name.
- Set Height to 25 mm.
- Set the Font to a **Basic Font**.
- Set Horizontal Alignment to Centre, Vertical Alignment to Middle.
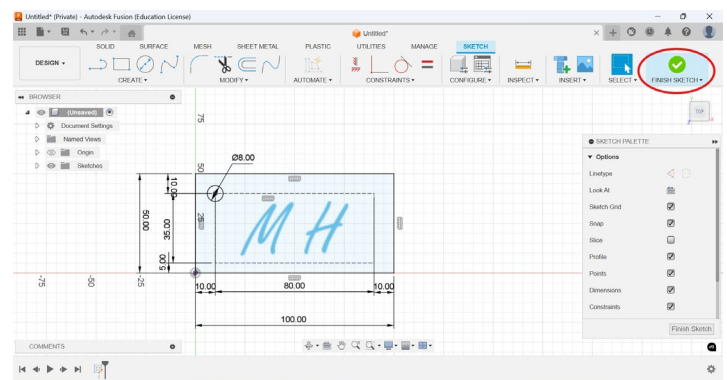- Click OK to place the text.

- Top edge of text box: set height to 80 mm
- Side of text box: set width to 35 mm
- Confirm all text lines are black (fully defined).
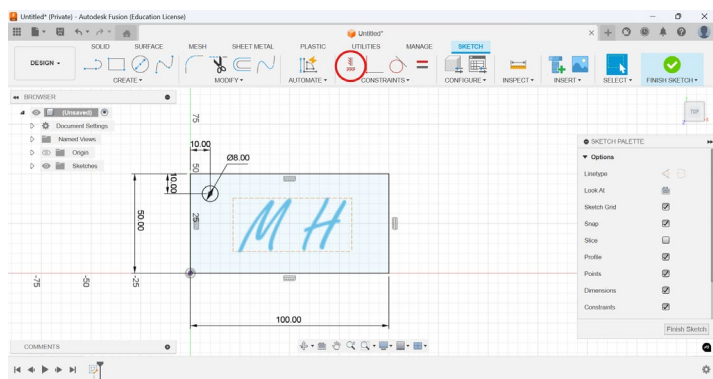

Click Finish Sketch (top-right corner).

Step 6: Constrain the Text

Use the Horizontal/Vertical Constraint:

Click on the top edge of the text box to make it horizontal.

Use Sketch Dimension to define position:

- Bottom of text box → Bottom of rectangle: 5 mm
- Right of text box → Right edge of rectangle: 10 mm
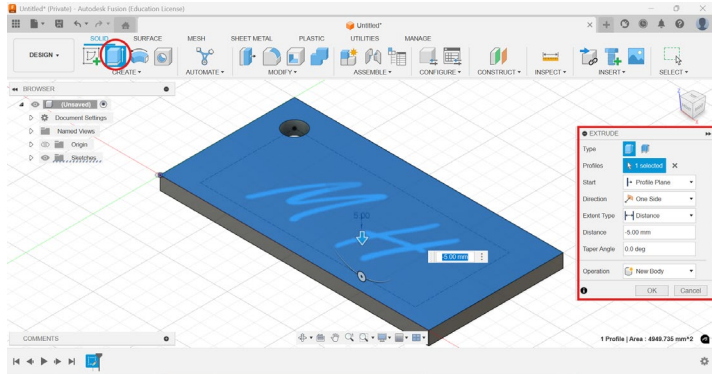
Step 7: Extrude the Keychain Body

Go to Solid > Create > Extrude.

Select the rectangle (base shape).

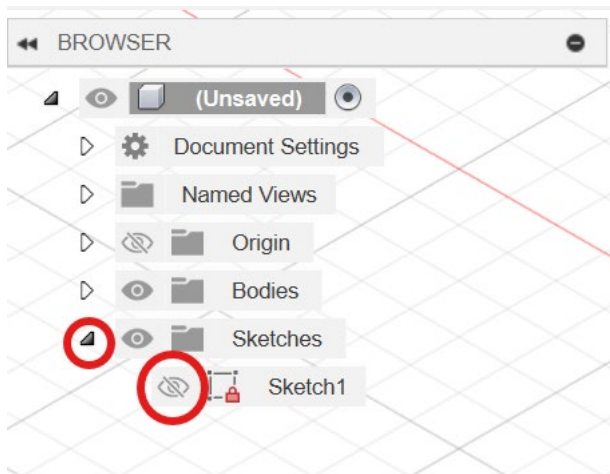Set the extrusion Distance to -5 mm (extrudes downward).
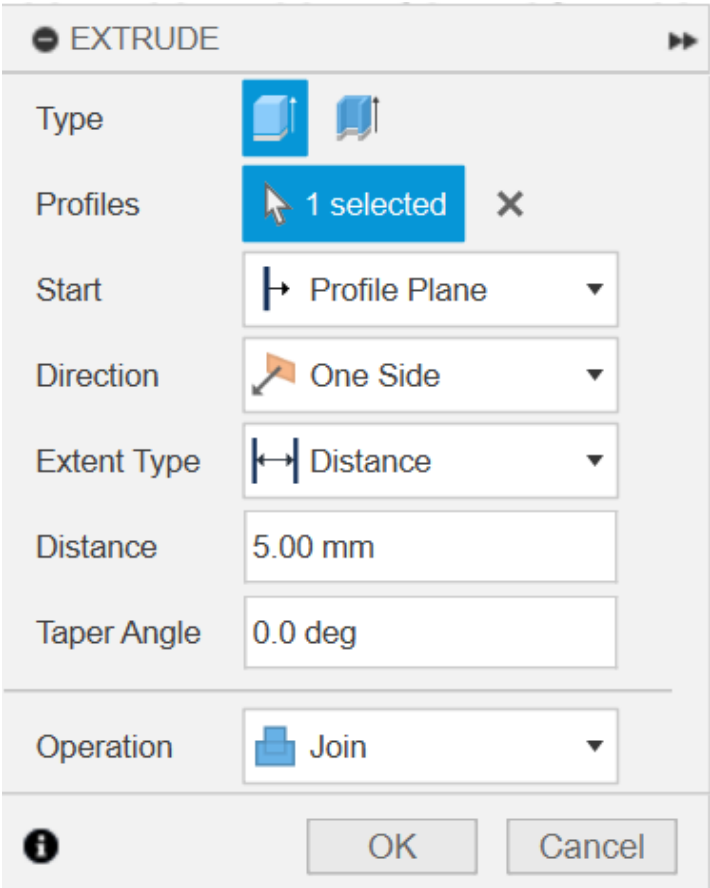
Click OK.

Step 8: Extrude the Text

In the Browser panel (left side):

- Expand the Sketches folder.
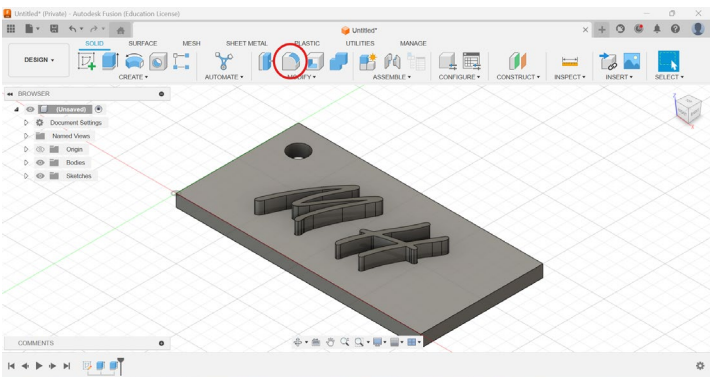- Click the eye icon next to your sketch to make it visible again.

- Go to Solid > Create > Extrude again.
- Select the text only.
- Set the extrusion Distance to 5 mm.
- Set Operation to Join.
- Click OK.

Hide the sketch again by clicking the eye icon. Now Collapse the Sketches folder.

Step 9: Round the Corners (Fillet)
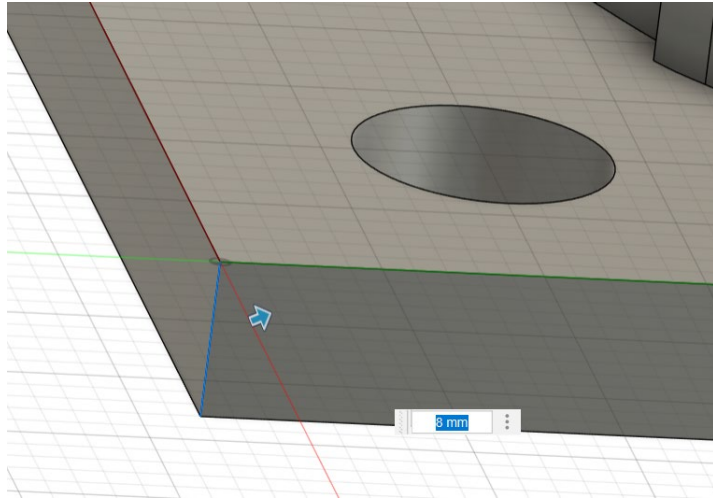
Go to Solid > Modify > Fillet.

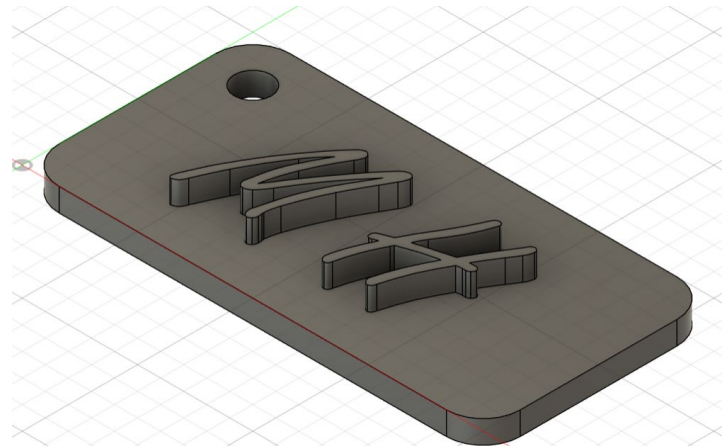# Fusion 360 Beginner Step-by-Step Guide: Making a Name Tag/Keychain

Select the four outer corners of the keychain.
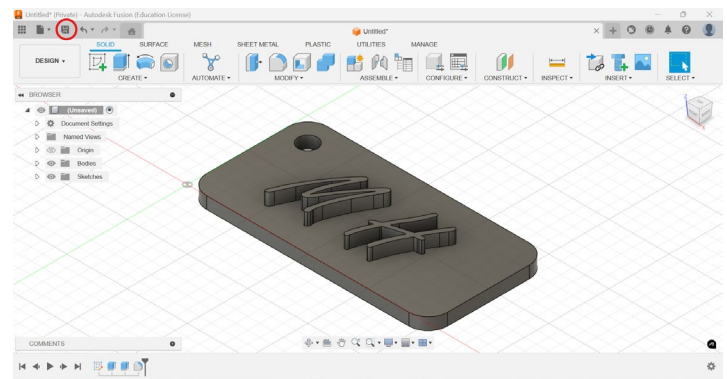
Set Radius to 8 mm.

Press Enter to apply.

Your name tag should now look something like this:

Step 10: Save Your Design

Click the Save icon in the top-left corner.

Name your file "Keychain_v1".

Choose a project folder if needed.

Click Save.