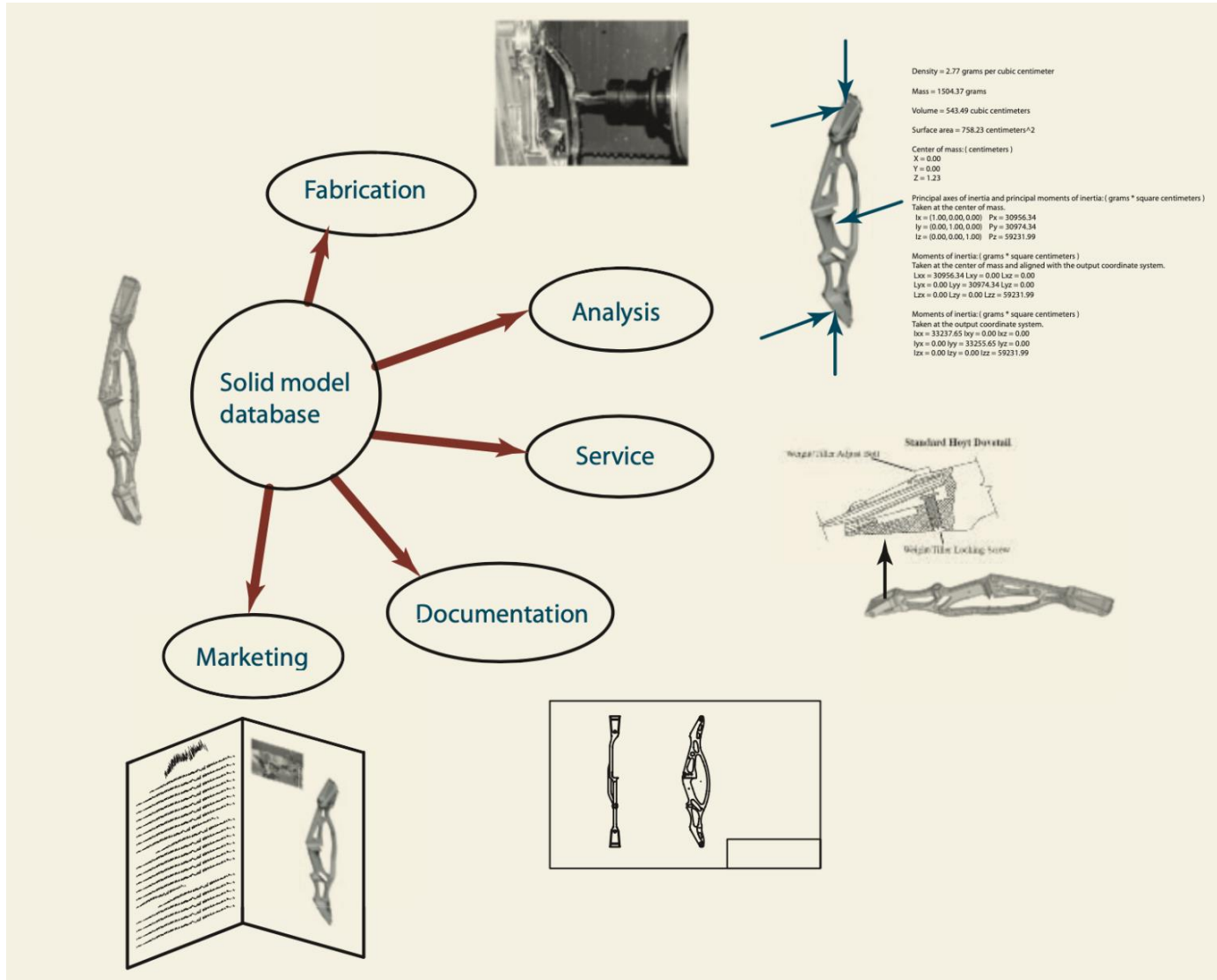# MS 101
Introduction to solid modeling
&
2D sketching in Fusion360

Prof. Rajneesh Bhardwaj,

Dept of Mechanical Engg, IIT Bombay
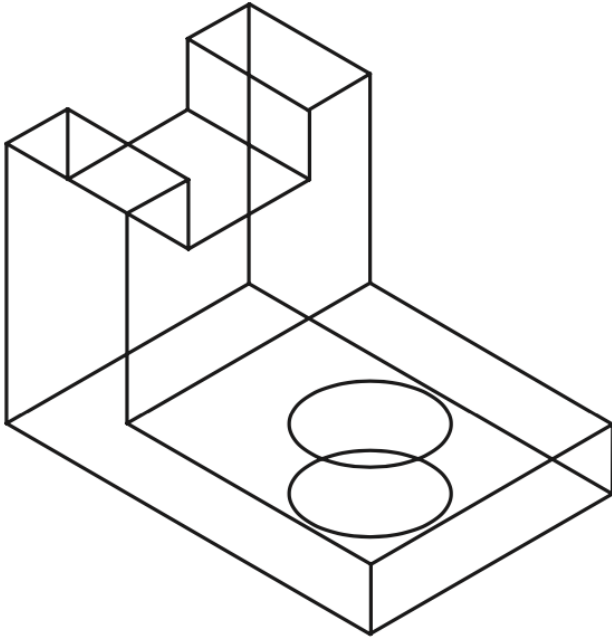
(Solid modeling slides credit: Prof. Ankit Jain and Janani M.,

Ref. Chapter 5, Lieu and Sorby book

2D sketching: Fusion 360 manual)

Solid Modeling

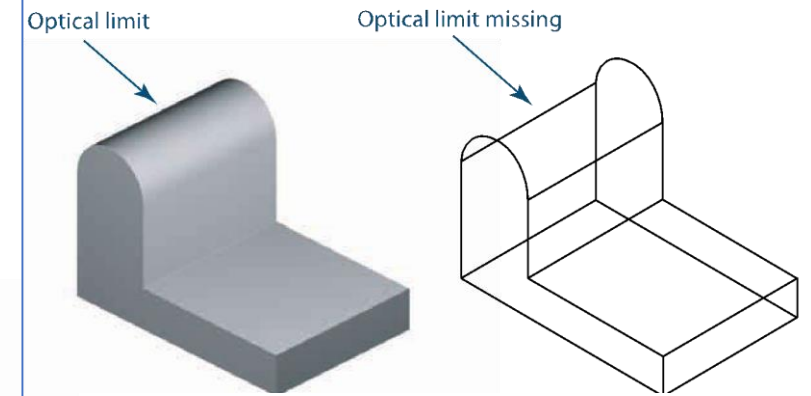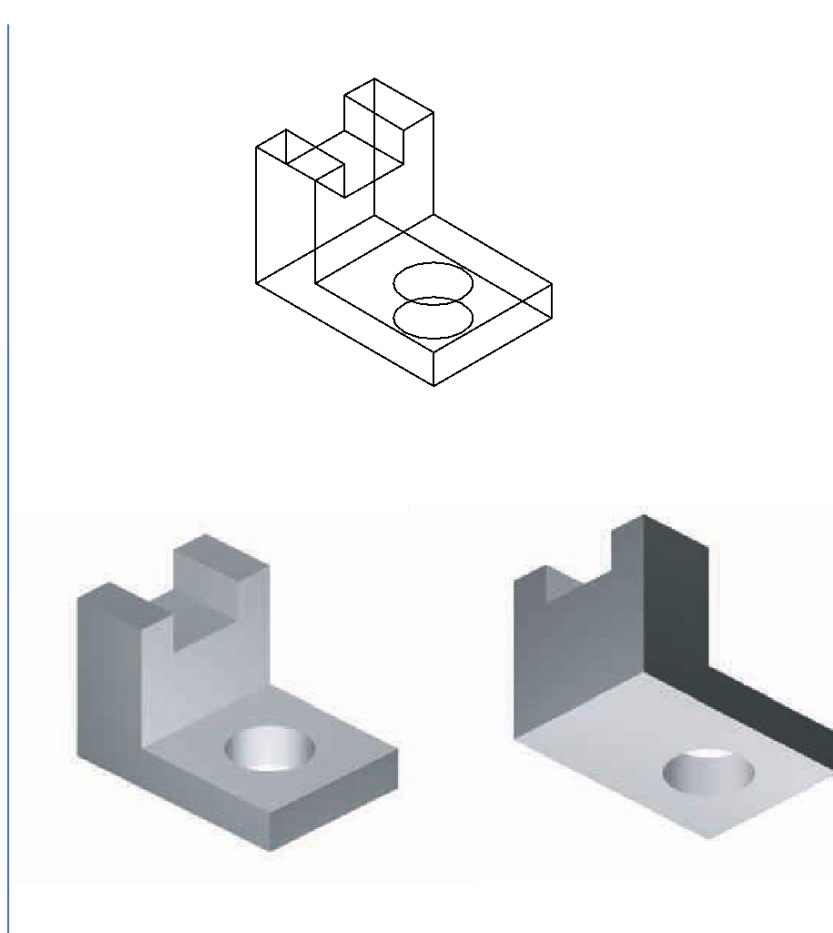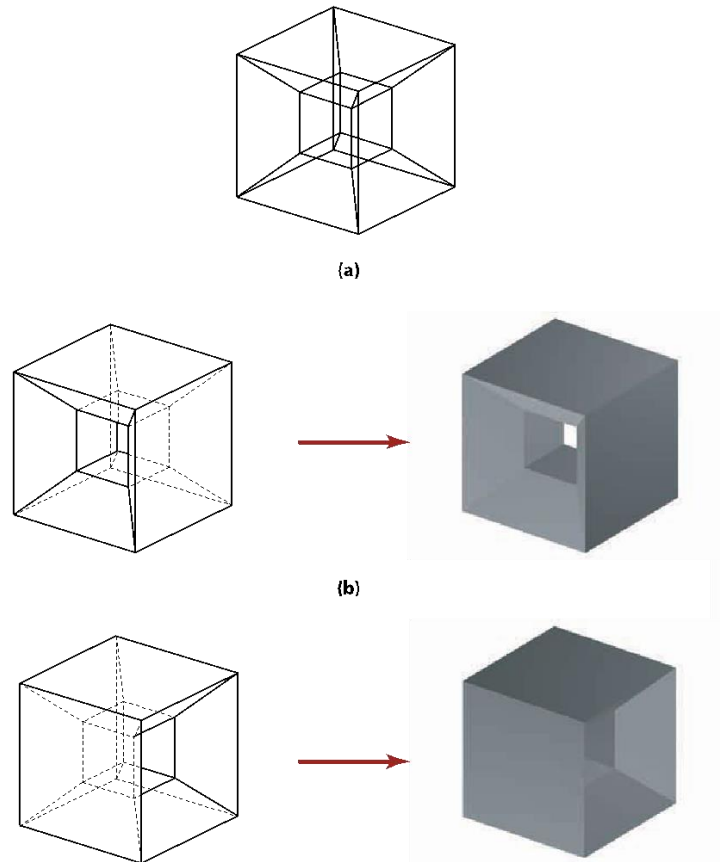Computer based simulation that provides a visual display of an object as if it existed in three dimensions
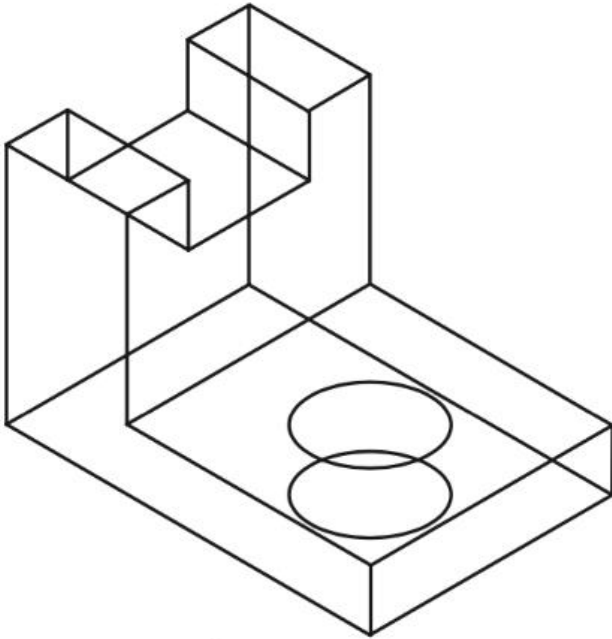
3D-Models

Wireframe model

- The geometric entities were represented with x-, y-, and z-coordinates.

- Since simple curve or path entities were used to define the edges of an object
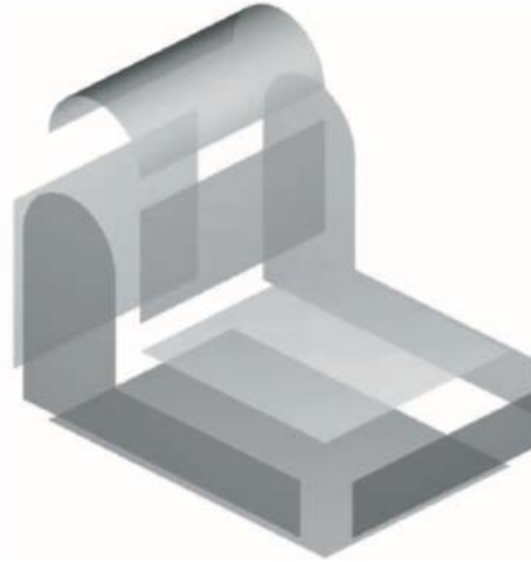
Wireframe Models

(a)

(b)
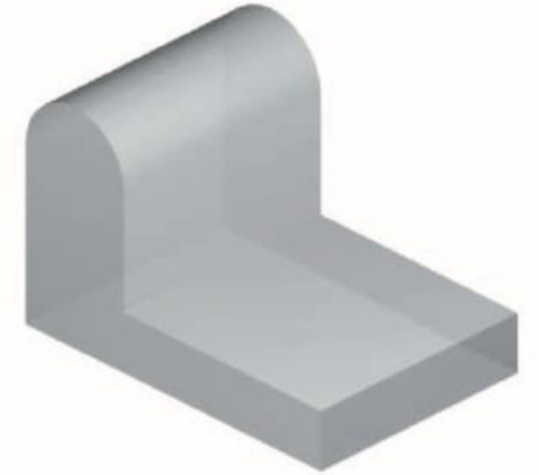
Optical limit

Optical limit missing
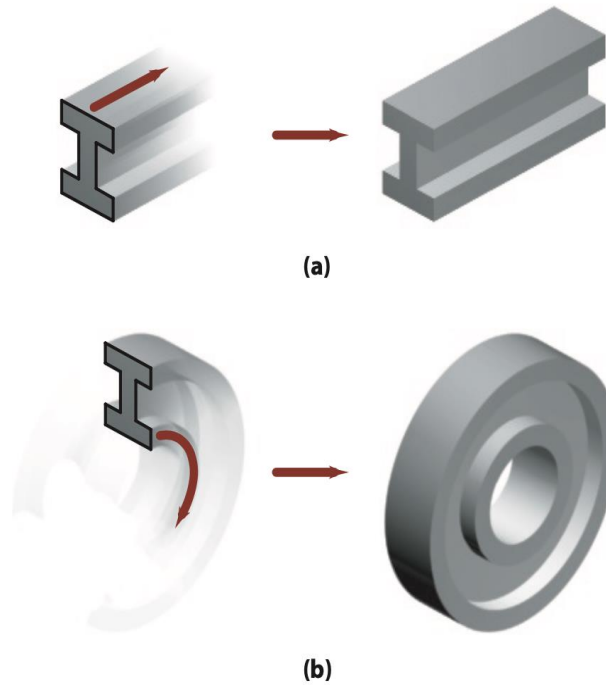
3D-Models

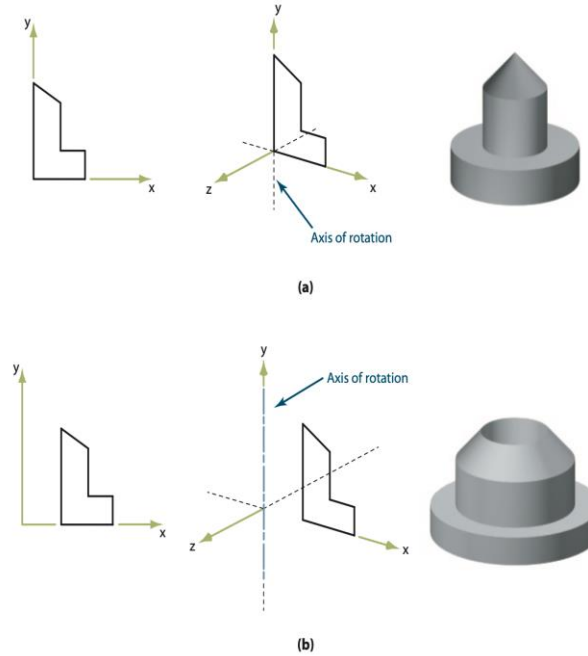Wireframe model

Surface model

Solid model

Solid models contain information about the outside as well as inside of a part

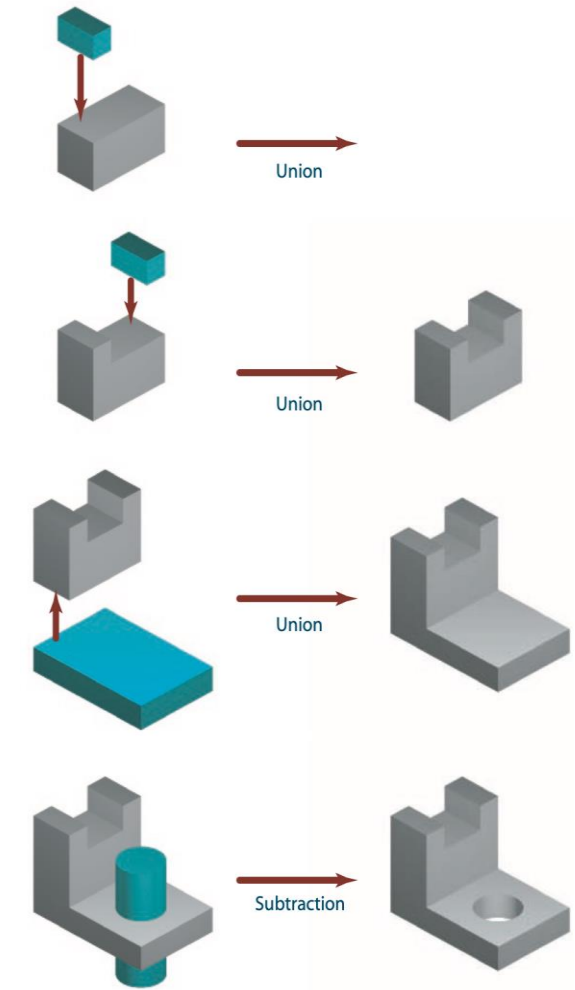But everything begins with a 2D sketch

Creating Complex 3D-objects
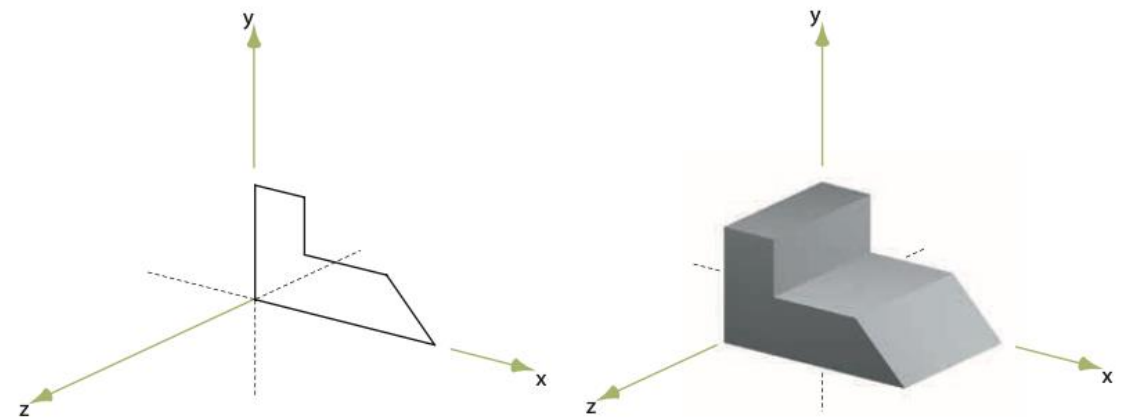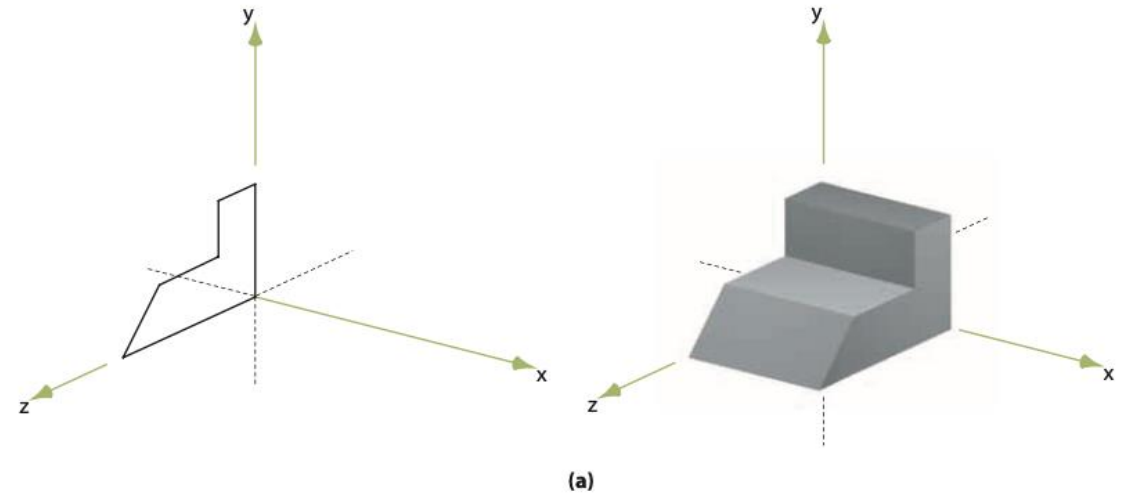

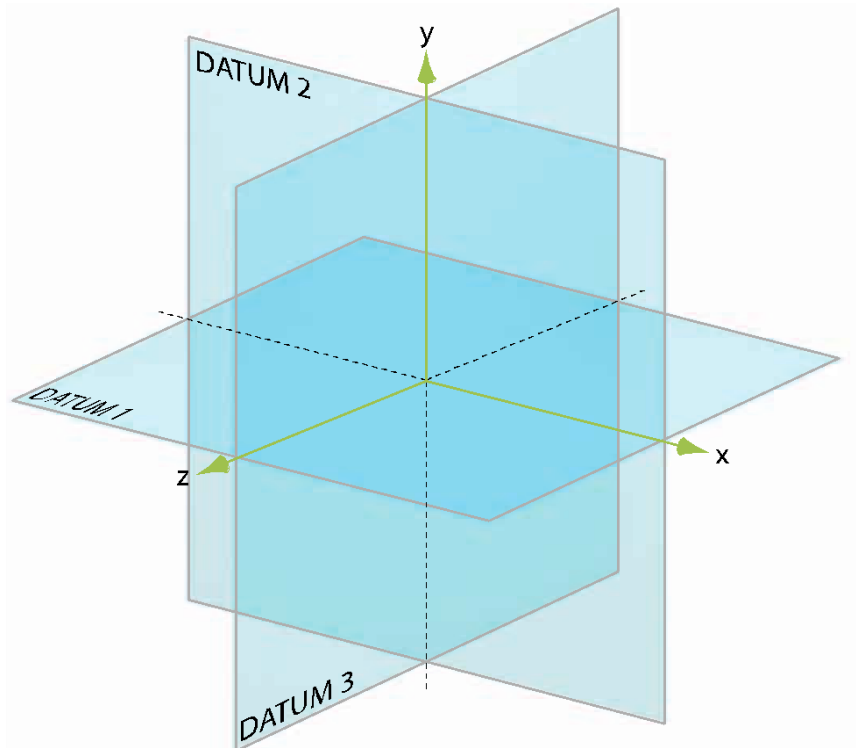Using 2D-sketches: (a) extrusion (b) revolve

Revolve
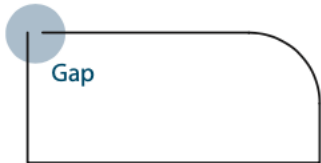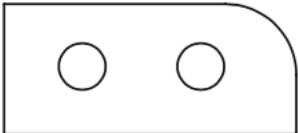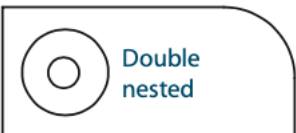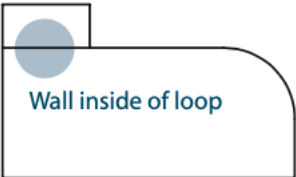
Combining existing objects

Orientation of the Sketch

(a)

Proper and Improper Profiles

Valid profiles must be closed loops

Proper

Closed loop

Nested loops

Multiple single nested loops

Multiple loops

Multiple loops

Simple revolved loop profile

Improper

Gap

Overlap, or extra segment

Double nested

Crossing

Wall inside of loop

Revolve profile overlapping axis

Making Precise Profiles – Constraints

Constraints are geometric relationships, dimensions or
equations that control the size, shape of entities in the profile sketch
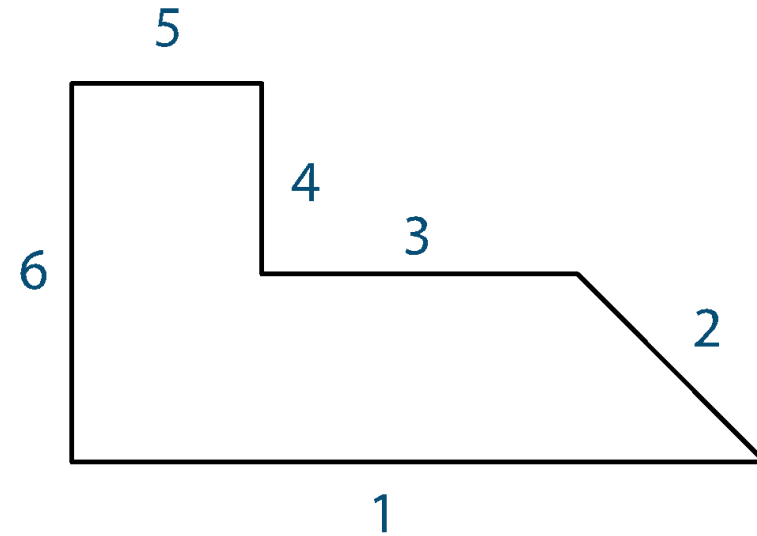
1. Geometric Constraints
2. Dimensional Constraints
3. Associative and Algebraic Constraints

GOAL : CONSTRAINT A PROFILE FULLY!

Geometric Constraints

Geometric constraints specify relationships between points, lines, circles, arcs, or other planar curves.

Examples of geometric constraints -
- ➢ Concentric
- ➢ Point on line
- ➢ Horizontal/Vertical
- ➢ Tangent
- ➢ Colinear
- ➢ Parallel
- ➢ Perpendicular
- ➢ Symmetric

Tangent

Colinear

Parallel
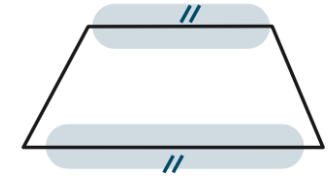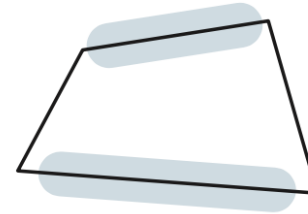
Perpendicular
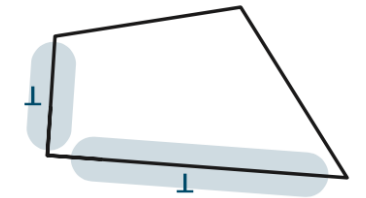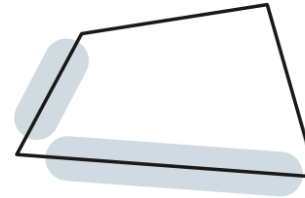
Equal length or equal radii
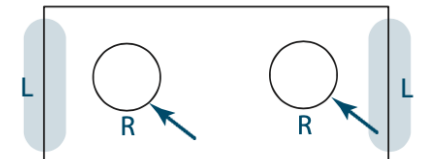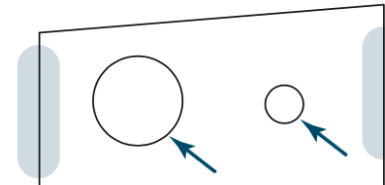
Concentric
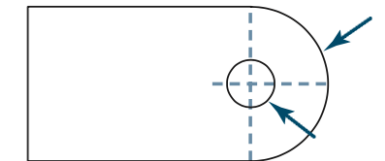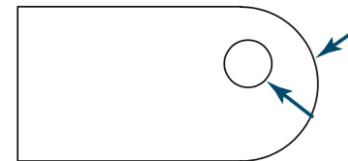
- Geometric constraints may be either implicitly defined (hidden from the designer) or explicitly displayed so you can modify them.

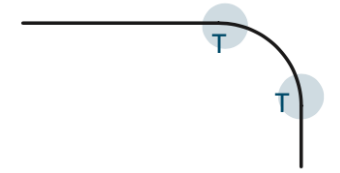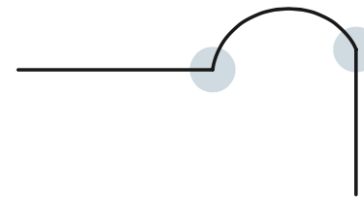 - nearly horizontal or vertical lines into true horizontal or vertical lines, if two lines are nearly perpendicular or parallel or an arc and a line are nearly tangent at the common endpoint, the sketching editor will impose the assumed geometric relationship.
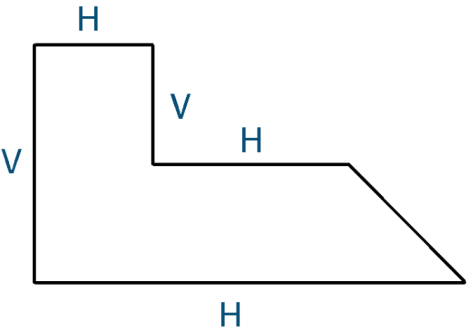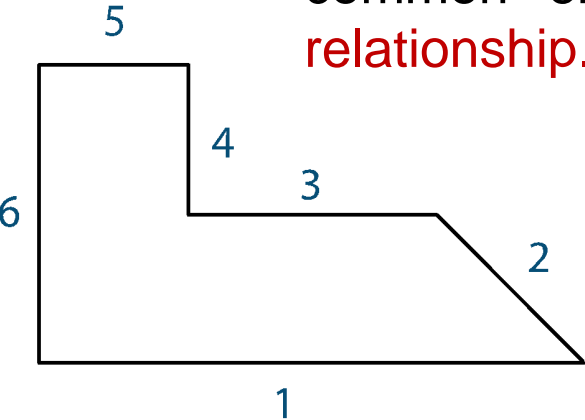
Segment	Constraint
1	Horizontal
3	Horizontal
4	Vertical
5	Horizontal
6	Vertical

- A set of geometric constraints is not unique,
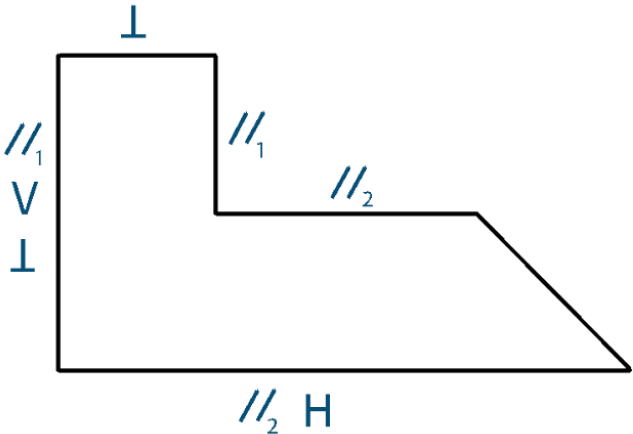
Segment	Constraint
1	Horizontal, parallel to 3
3	Parallel to 1
4	Parallel to 6
5	Perpendicular to 6
6	Vertical, parallel to 4, perpendicular to 5

2D sketching in Fusion 360

First angle projections of a 3D object

RHSV

FV

LHSV

180

140

75°

Ø150

TV
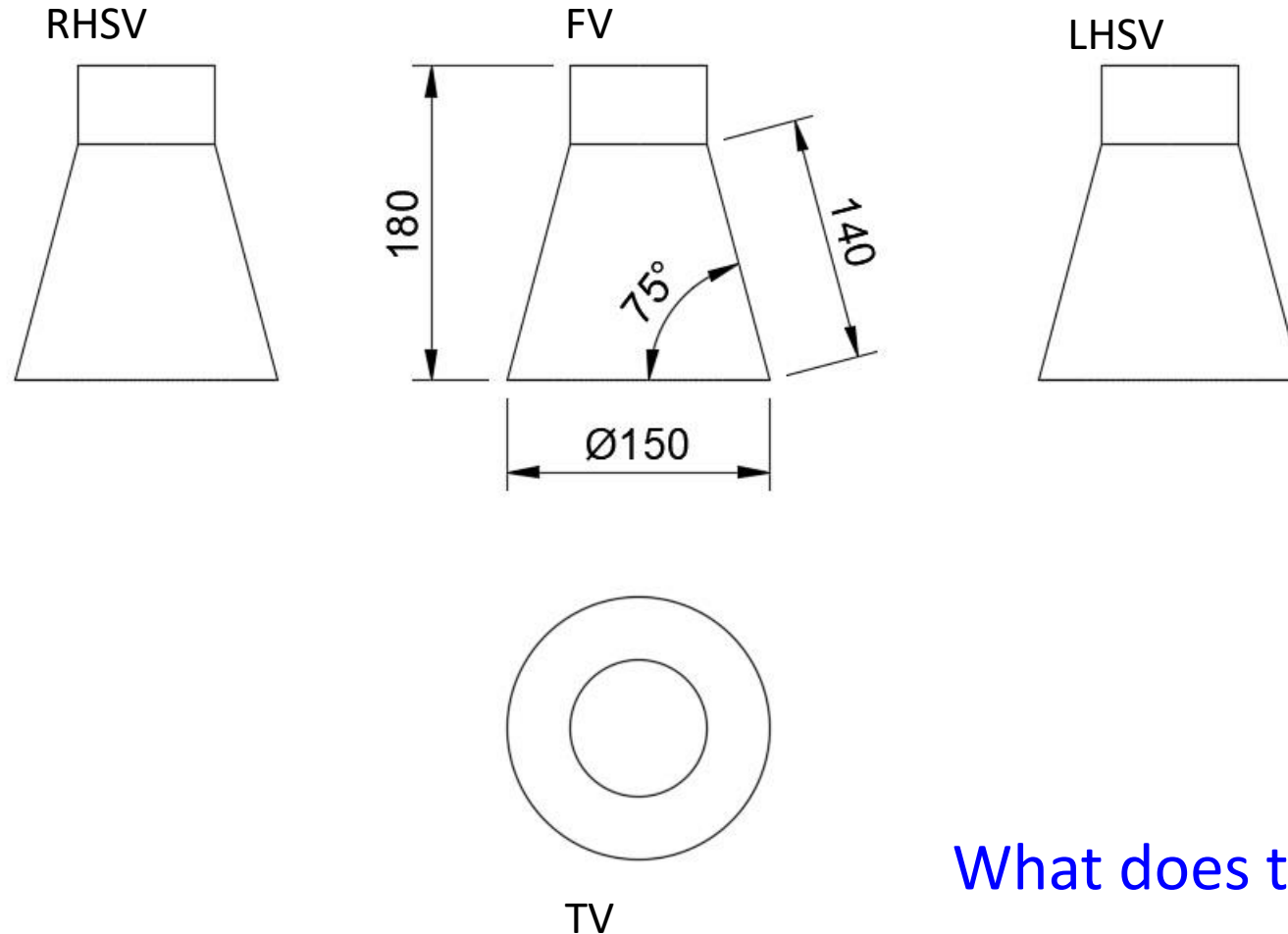
What does this object look like?

Sketch 2D profile

- Axisymmetric object
- Say, a tea kettle
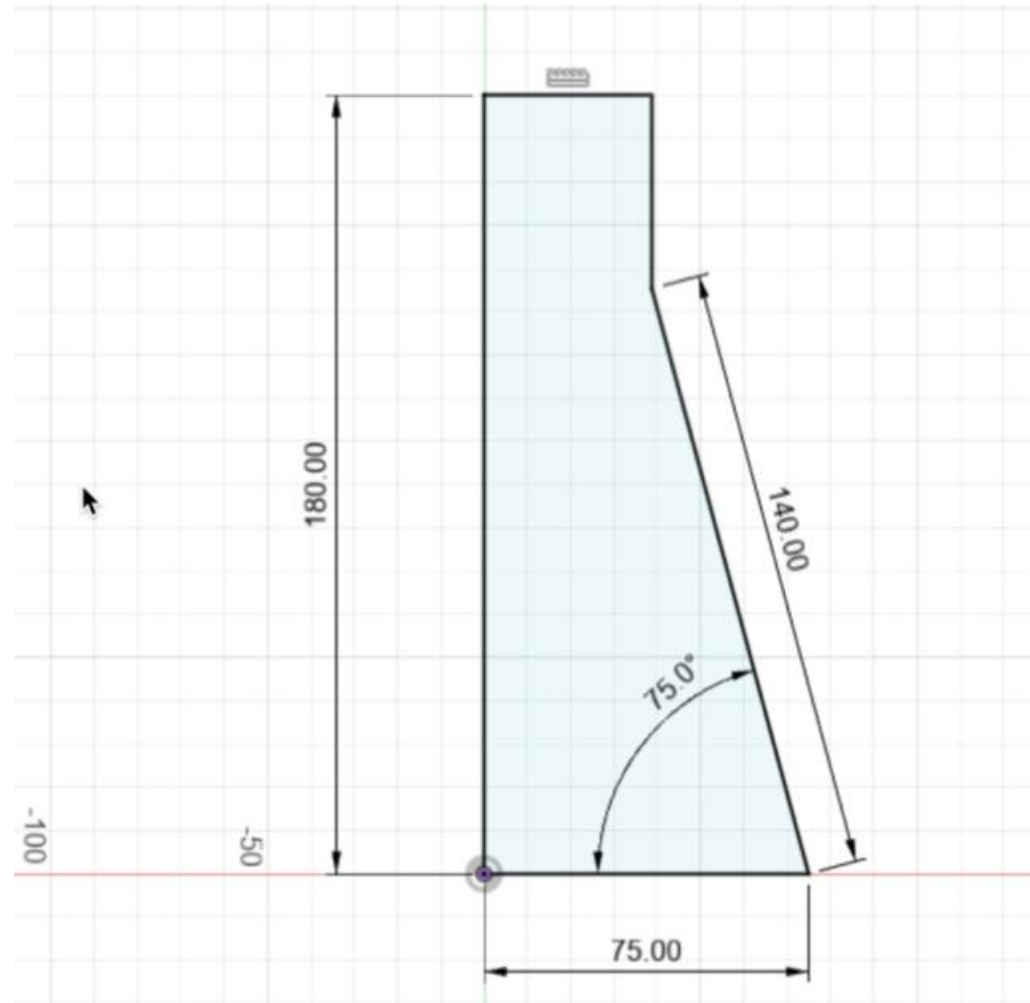
Units

- Decide the units of measure you'd like to design in.

- From the Browser, expand **Document Settings**, where you'll see the Active Units for this design.

- Click on **Change Active Units**, followed by selecting the desired units in the dialog
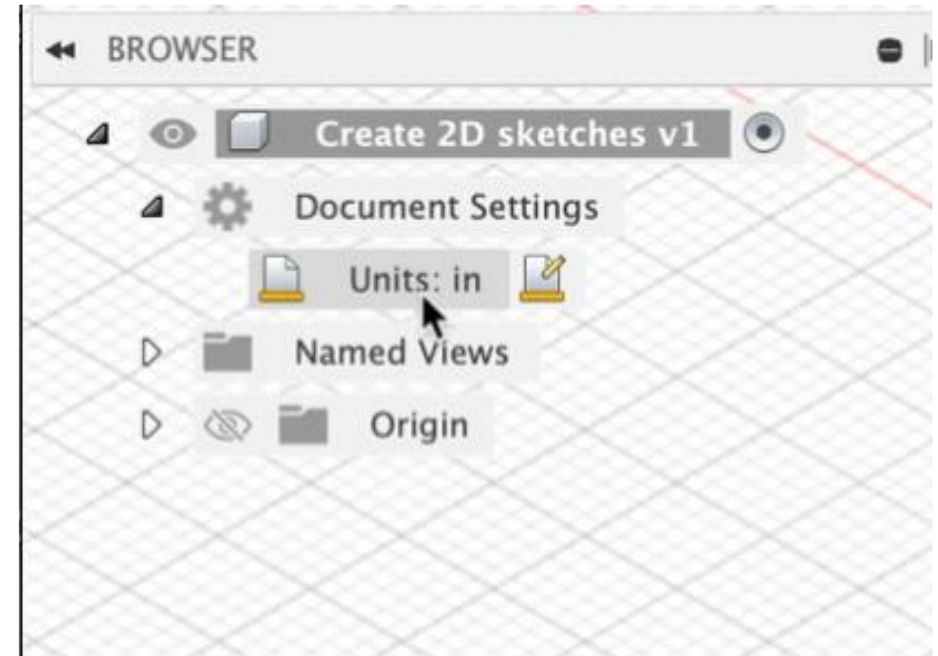
Sketching mode

- To access the Sketch tools, you first need to select the **Create Sketch** feature
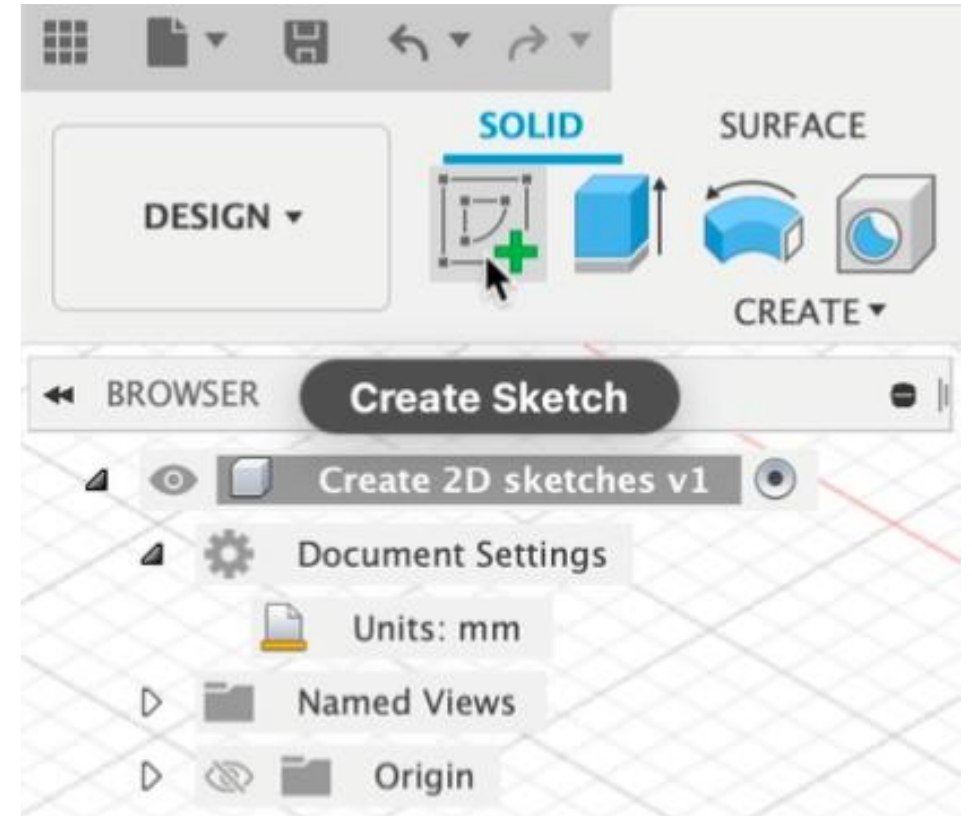
Selecting plane

- To enter sketching mode, select a 2D face or plane. The easiest way to do this is to select one of the existing original planes in the Canvas.

- Consider the orientation of your sketch and how the part will rest. To select the side profile of a tea kettle, select the front or **XZ** origin plane.

Select a plane or planar face
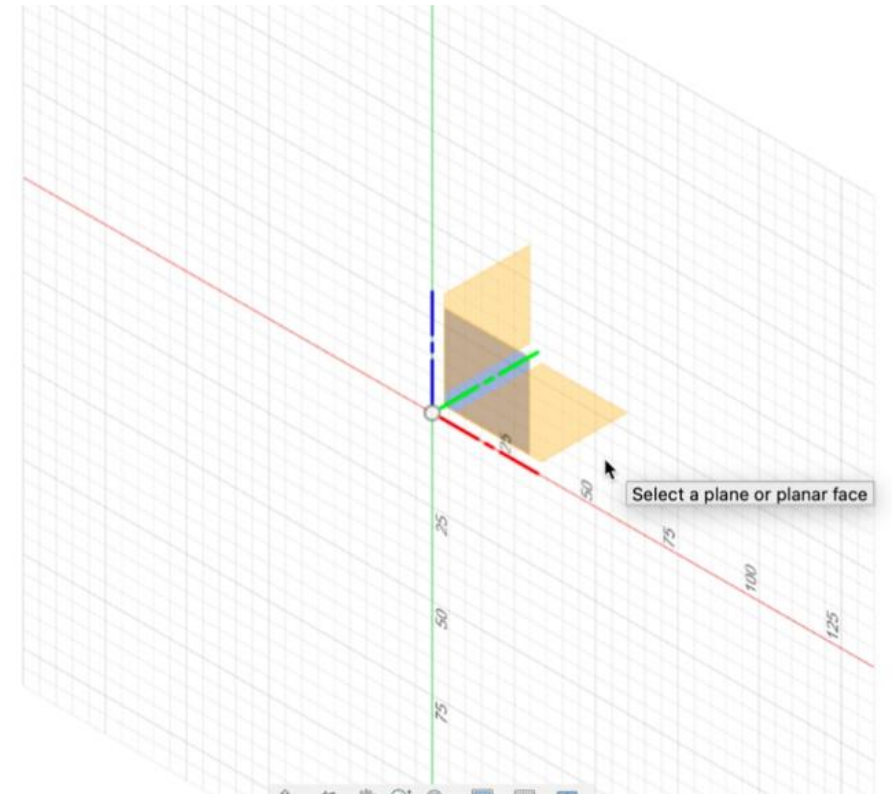
Sketch tools

- Sketch tools now appear in the Toolbar.

- Activate the **Line** tool to start sketching the profile of a tea kettle. It's symmetrical, so you can sketch half of the shape and use the 3D Revolve command to turn it into a solid body.

- As a best practice, start your sketch from the center origin point. This will help you fully define the sketch.
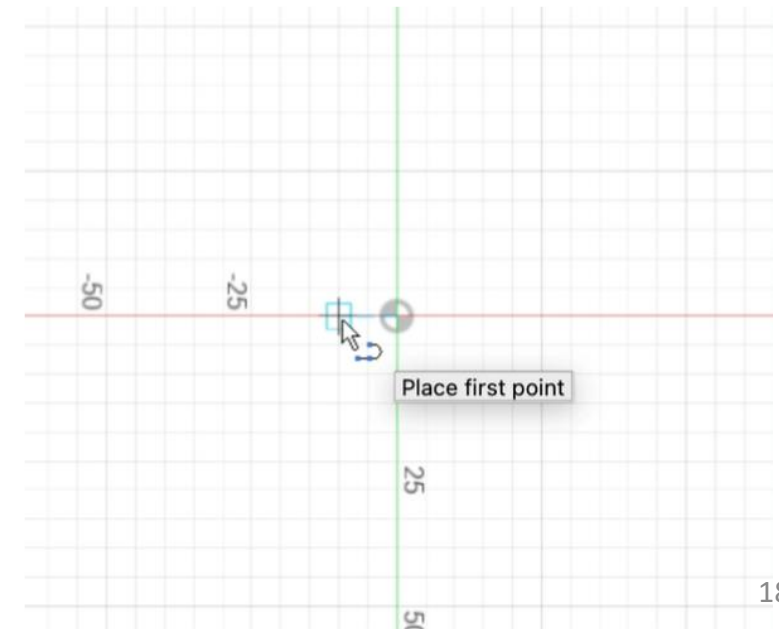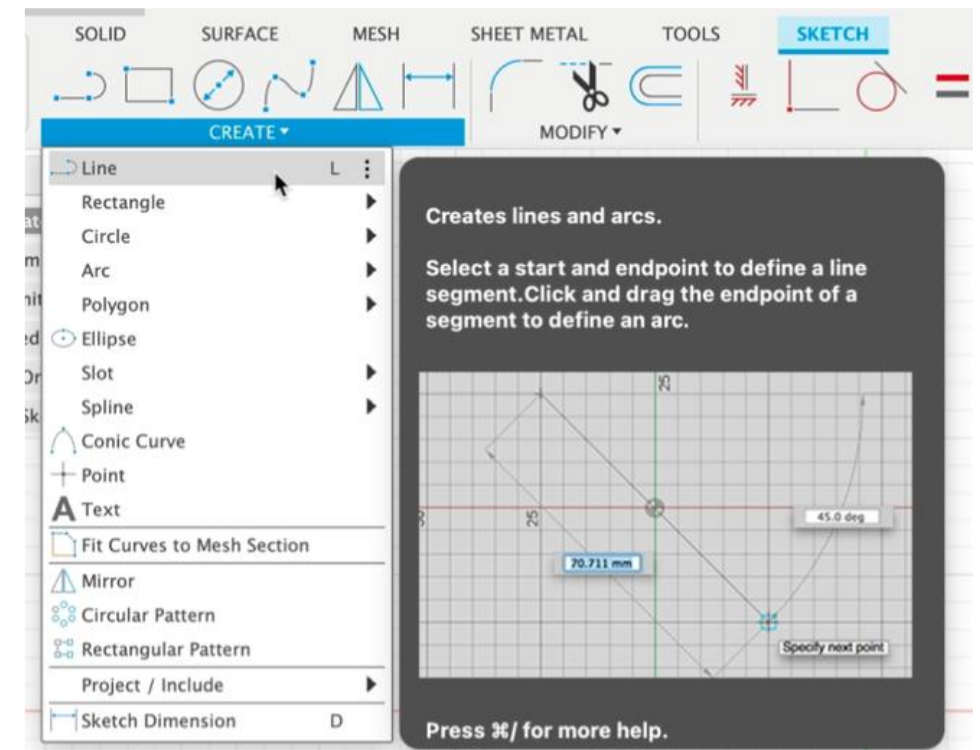
Draw a line

- Select the origin point, and notice how the start of the line snaps into place. As you move the cursor away from the origin, the line will extend, and two different dimension inputs will appear.
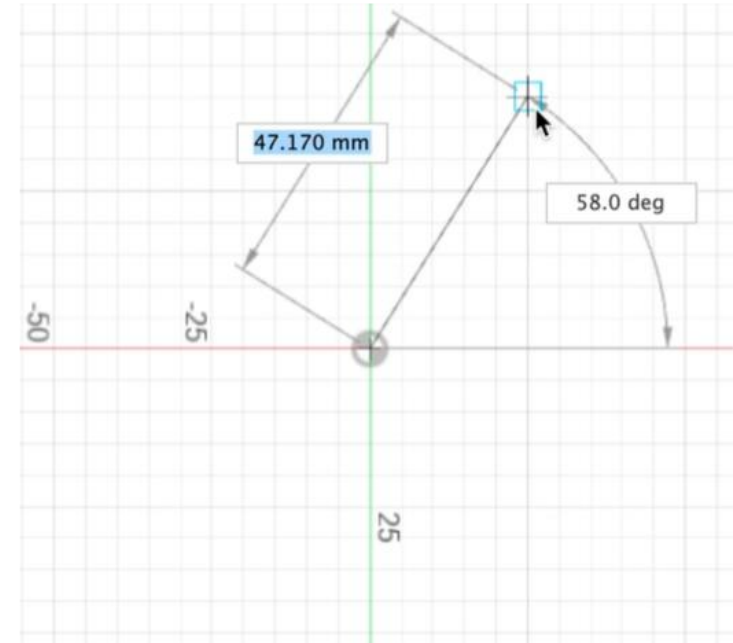
Assign dimensions

- To define the height of the tea kettle, draw a vertical line equal to 180 millimeters by typing 180 on your keyboard. Fusion 360 uses the unit you set in the Document Settings. Then press the tab key to lock the dimensions in place.
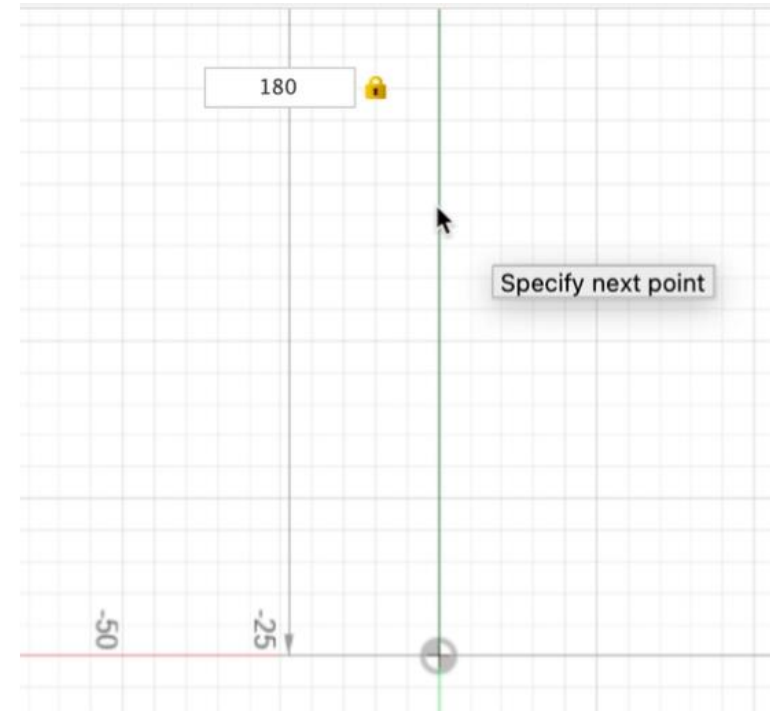
Snap the line

- To place the line, click on the Canvas to snap the line vertically when the vertical icon appears
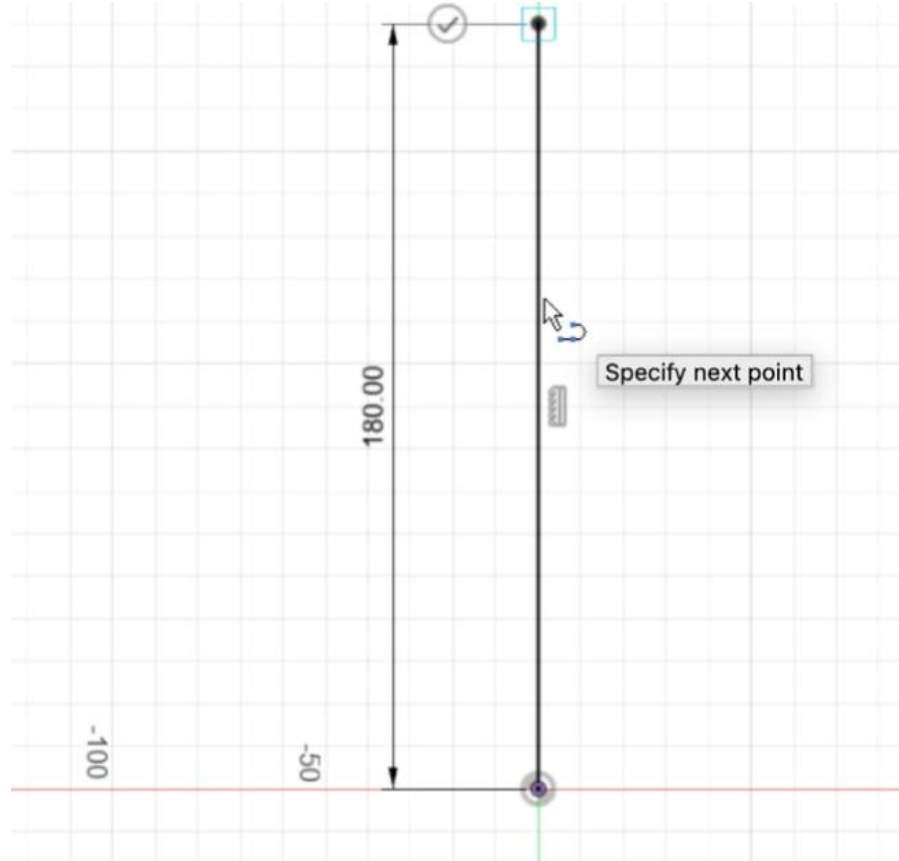
Draw another line

- Now press the **Escape** key to clear the line command and start another line from origin.

- Quickly reactivate a feature by right-clicking to access the Marking Menu. From here, select **Repeat Line** to reactivate the last active command (Or simply select the **Line** tool again).
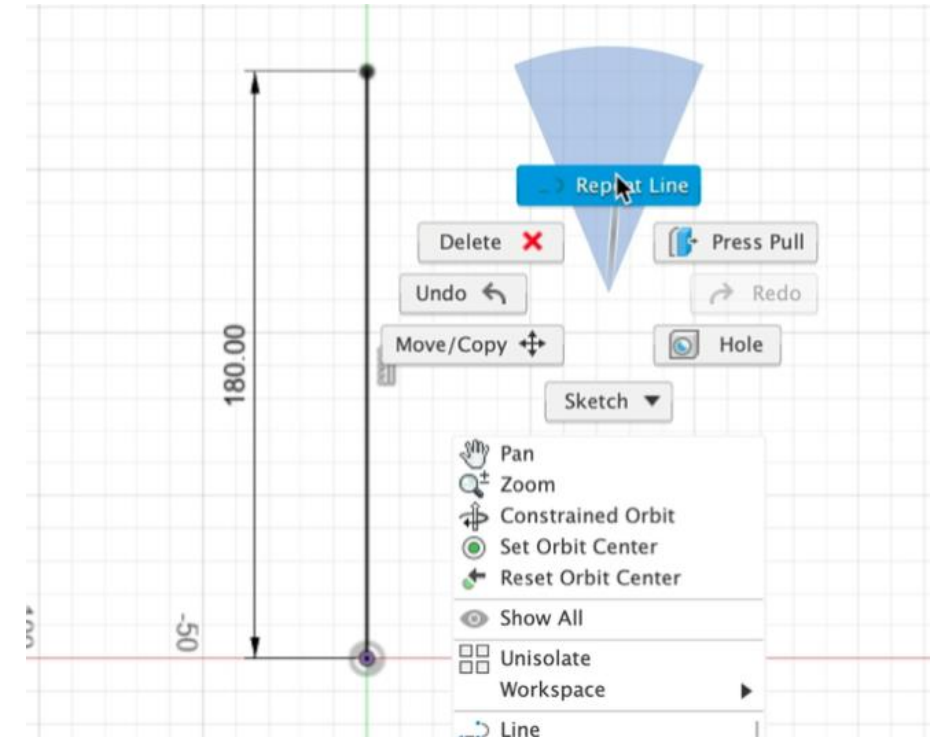
Snap the line

- Half of the width of the tea kettle will be 75mm. Start this line from the center origin point, moving toward the right. To ensure the bottom of the tea kettle is flat, click to snap the line in place when the right-angle icon appears in the corner.

Create line at an angle

- You can place line segments without yet defining the length or degrees, and add dimensions later.

- Create one line for the angled based, followed by a second line that runs vertically.

- Then create a third line that connects to the top of the center line.
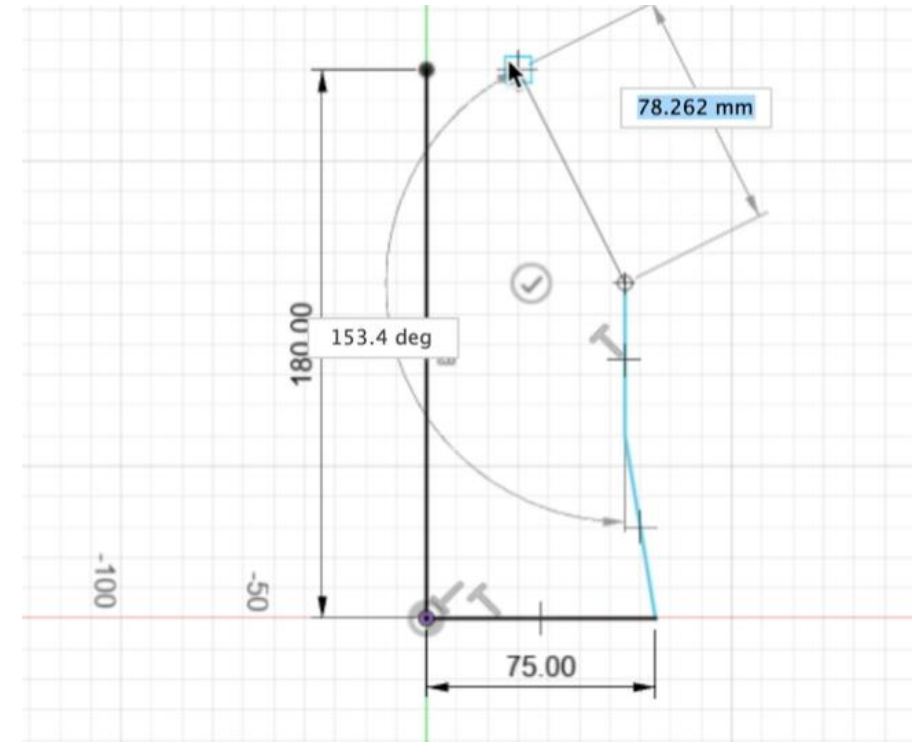
Save sketch

- Toggle open the **Sketches** folder in the Browser to see all design sketches.

- Rename the sketches by selecting them and then clicking a second time. Type the desired name, such as "Side Profile."
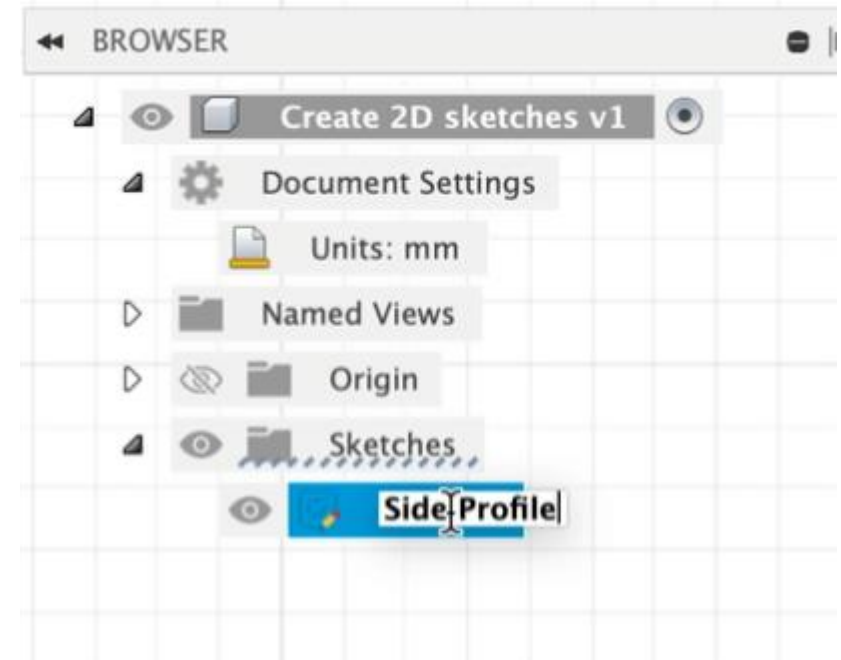
Add constraint

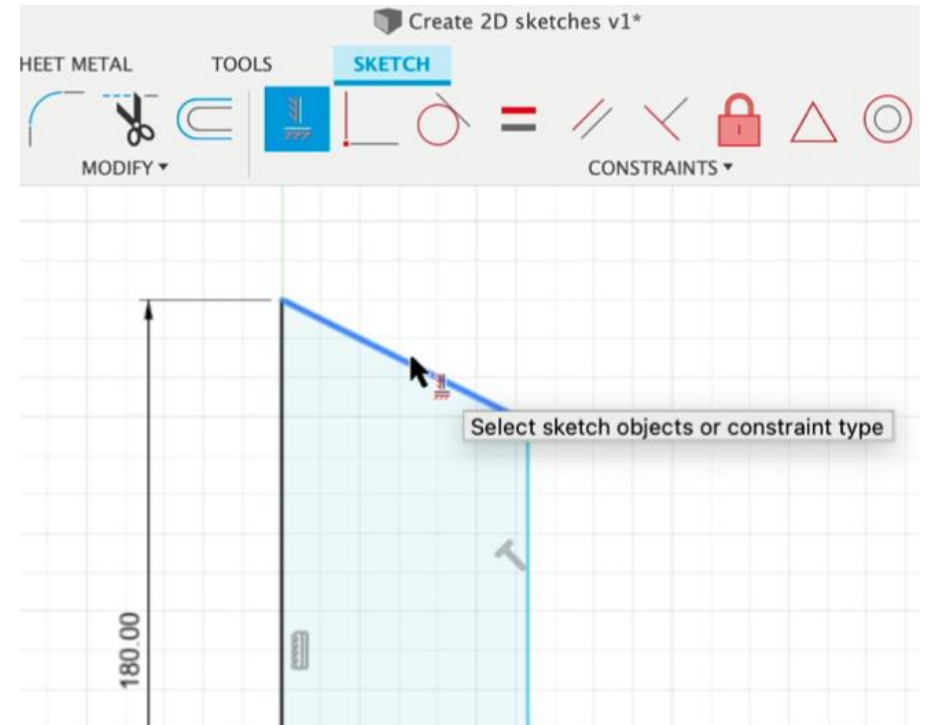- Finish defining your sketch. First, make sure the top of the tea kettle is flat so a lid can rest on top. Select the horizontal/vertical constraint in the toolbar, and then select the top line to force it to remain horizontal.

Can change dimensions at this stage

- You can apply sketch dimensions to the geometry at any time throughout the sketching process. Activate the **Sketch Dimension** tool from the Create menu or by typing **D** on the keyboard.
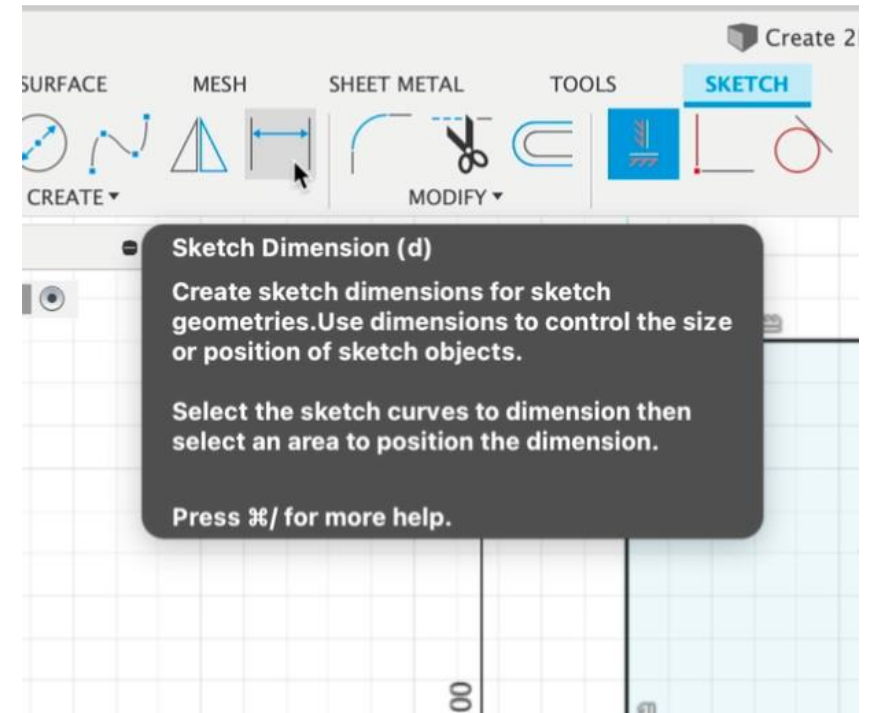
Assign angle

- You can select two entities to dimension the angle between them. Click to place the dimension, and then type 75 to make the angle 75 degrees. Press **Enter** to save it.
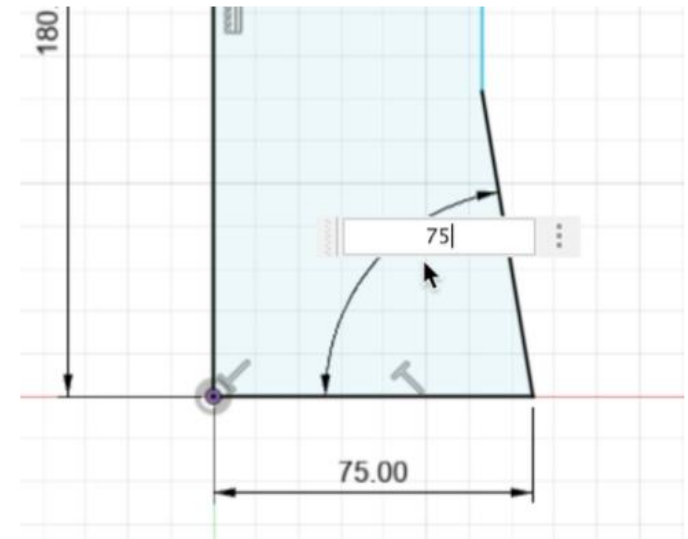
Assign dimension to line at an angle

- Define the remaining line. Add a final dimension of 140 millimeters to the bottom angled line, being careful to place the dimensions for the length and not the vertical distance between endpoints.
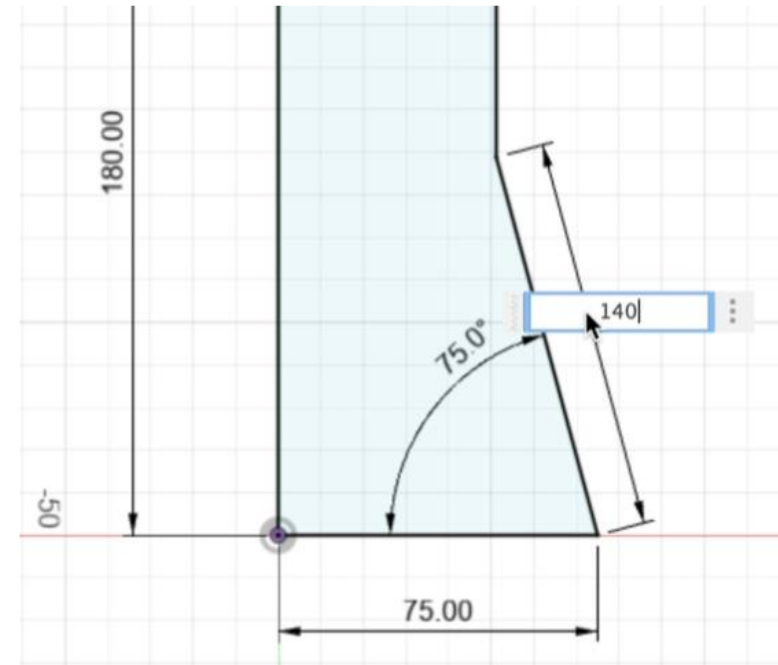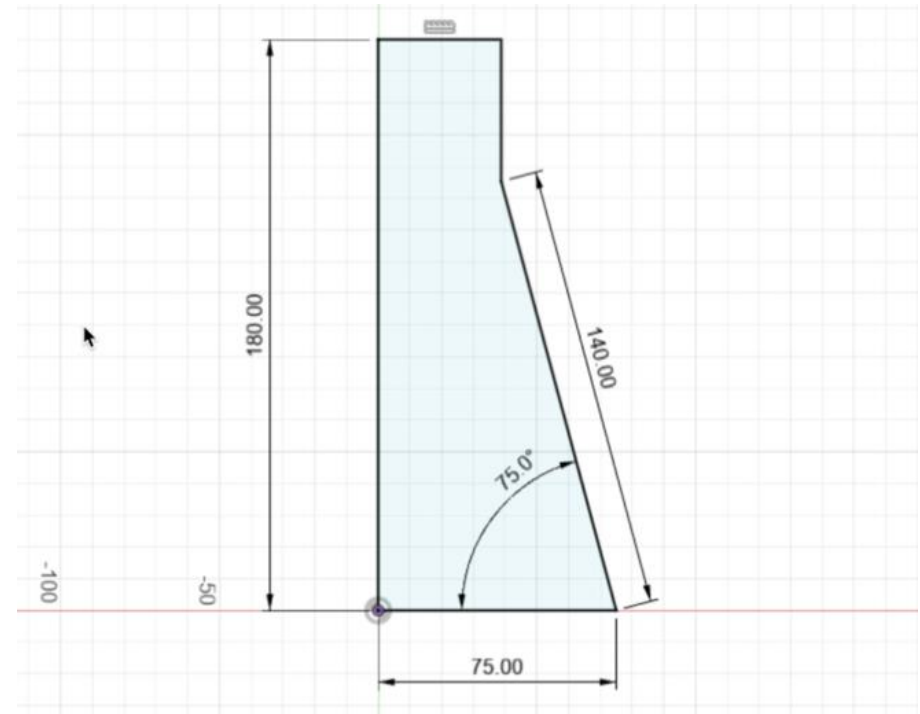
Red lock icon

- All the sketch geometry now appears black and fully defined. Look for a red lock icon in the Browser, which lets you know your sketch is fully defined.
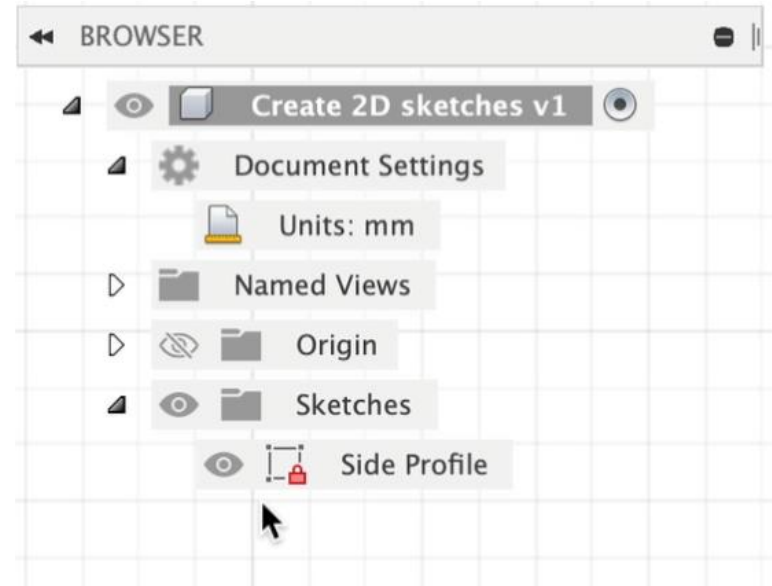
Complete the sketch

- The sketch is now complete and can be turned into a 3D body using solid modeling tools. Select **Finish Sketch** to exit the sketch mode.

Isometric view (say, a tea kettle)