MS 101
Constraints &
3D modeing in Fusion360

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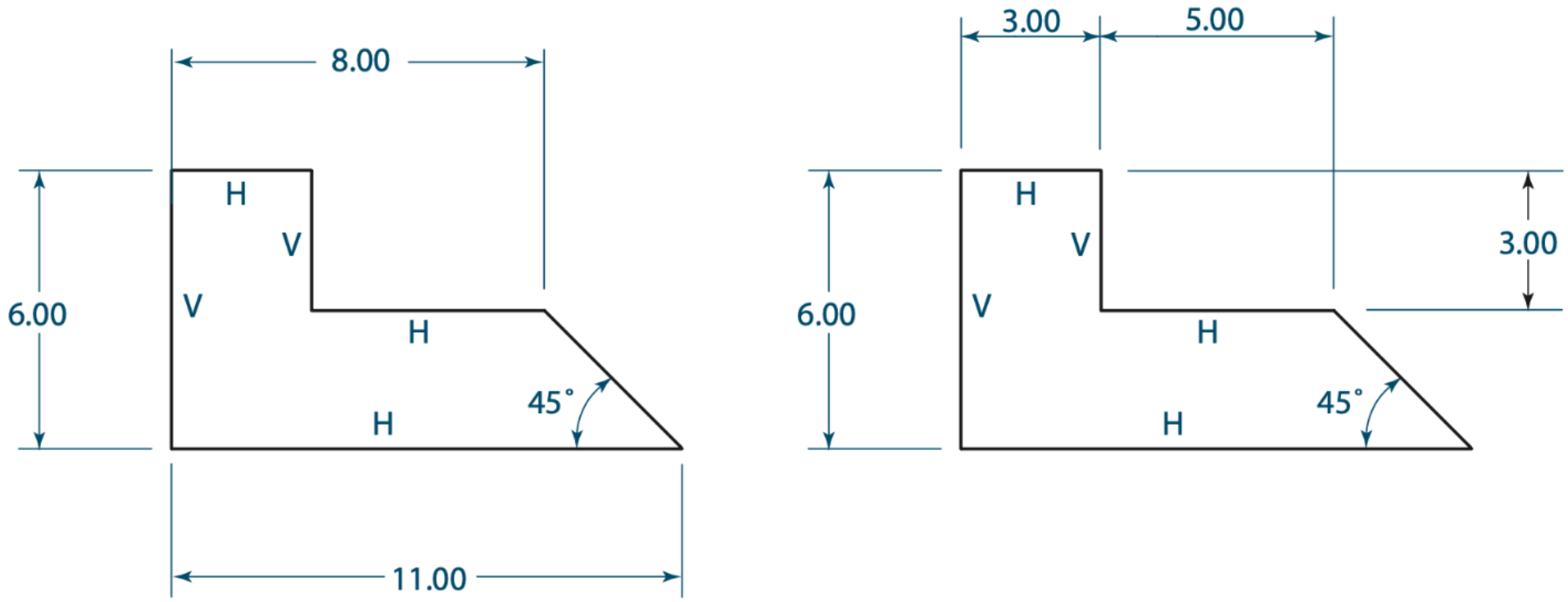
(**Constraint slides credit: Prof. Ankit Jain and Janani M.,**

Ref. Chapter 5, Lieu and Sorby book

3D modeling: Fusion 360 manual)

Dimensional Constraints

Measurements used to control the size and position of sketch entities



A set of constraints is not unique

There are three principal types of dimensional constraints:

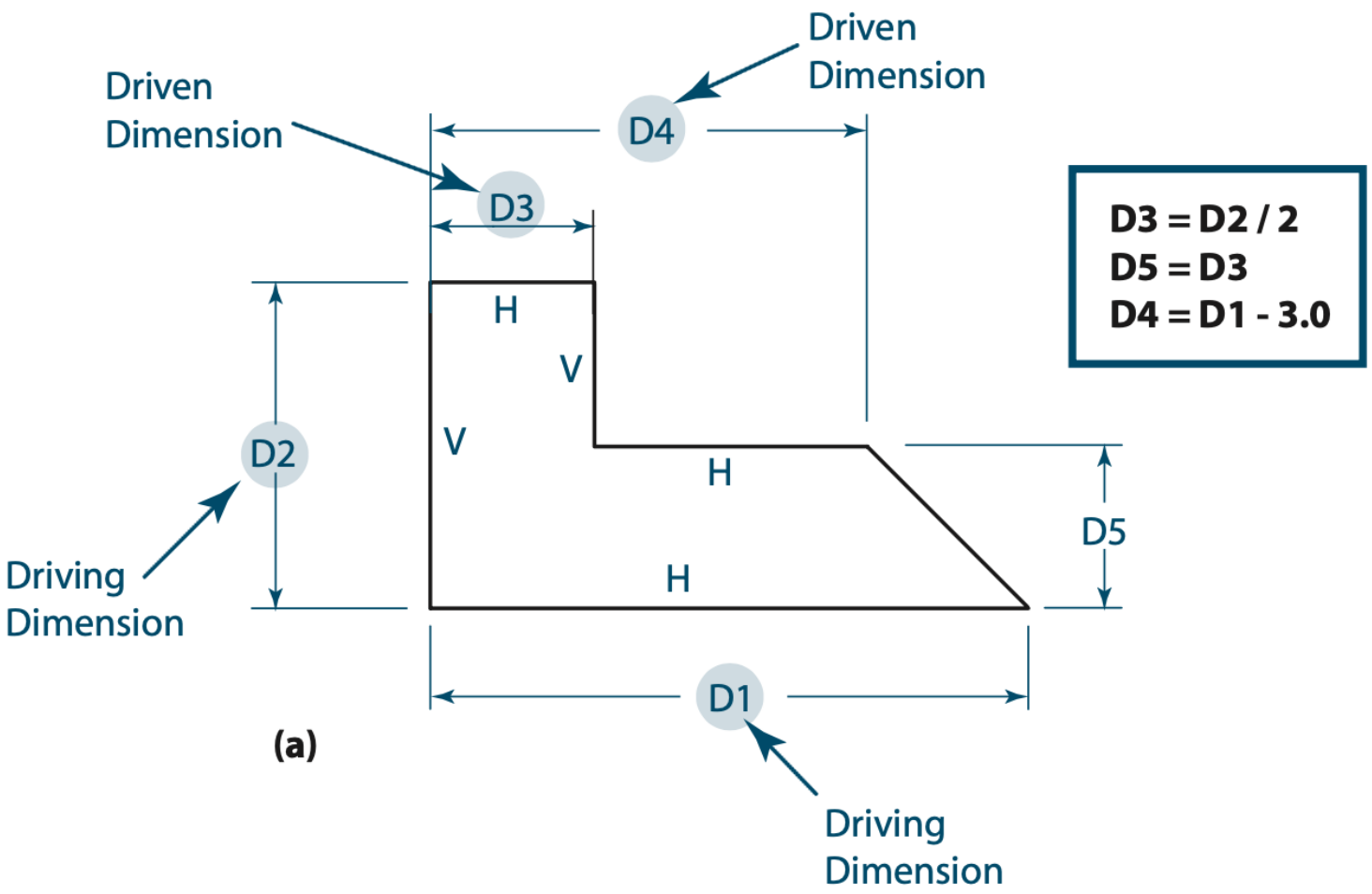
- Linear dimensional constraints define
  - the distance between two points,
  - the length of a line segment,
  - or the distance between a point and a line.
  - Linear dimensions can be measured horizontally or vertically or aligned with the distance being measured.

- Radial and diametral dimensional constraints specify
  - the radius or diameter of an arc or a circle.

- Angular dimensional constraints measure the angle between two lines.
  - The lines do not need to intersect, but they cannot be parallel.

Associative and Algebraic Constraints

relate one dimension to another using variables



Algebraic expressions consist of:

- constants and variables related to each other through the use of arithmetic functions
- +, -, *, absolute value, exponent, logarithm, power, square root, and sometimes minimum and maximum)
- trigonometric functions;
- and conditional expressions (if, else, or when) including inequalities comparisons (if A > B then . . .).

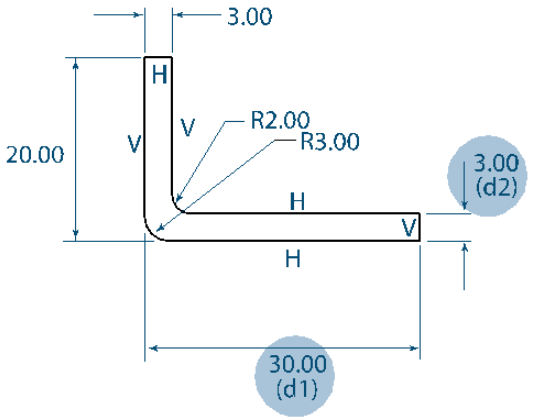
which set of constraints is preferred?

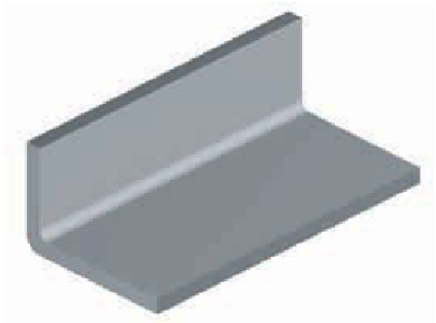
- what the function of the part and the design intent is?

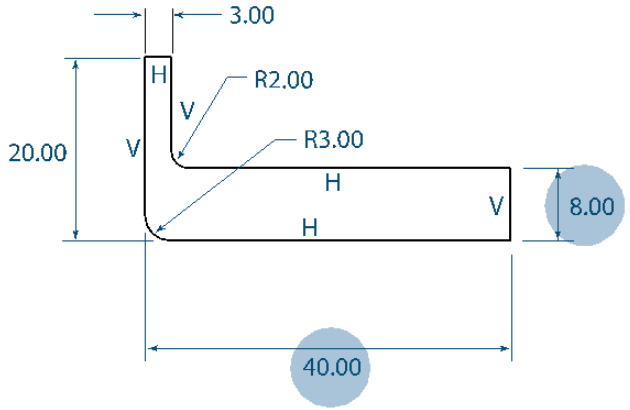
- or how the designer wants to be able to change the model.

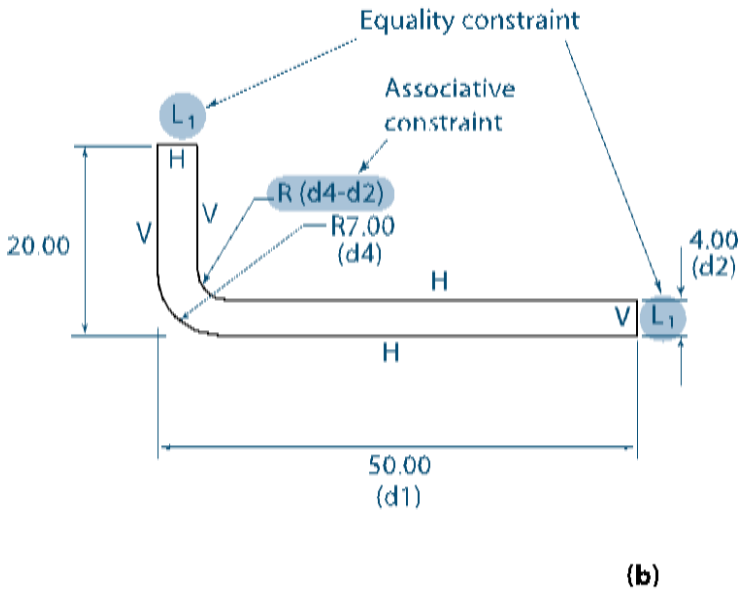
- You also should consider how the solid model will be used for analysis, manufacturing, and documentation when applying sets of constraints.

Constraint Strategy

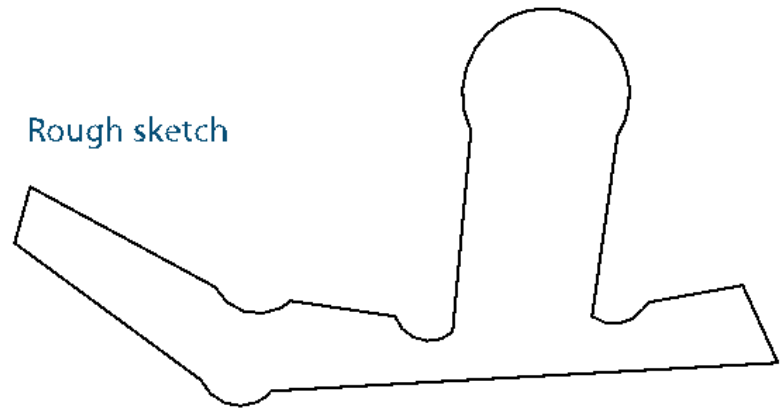




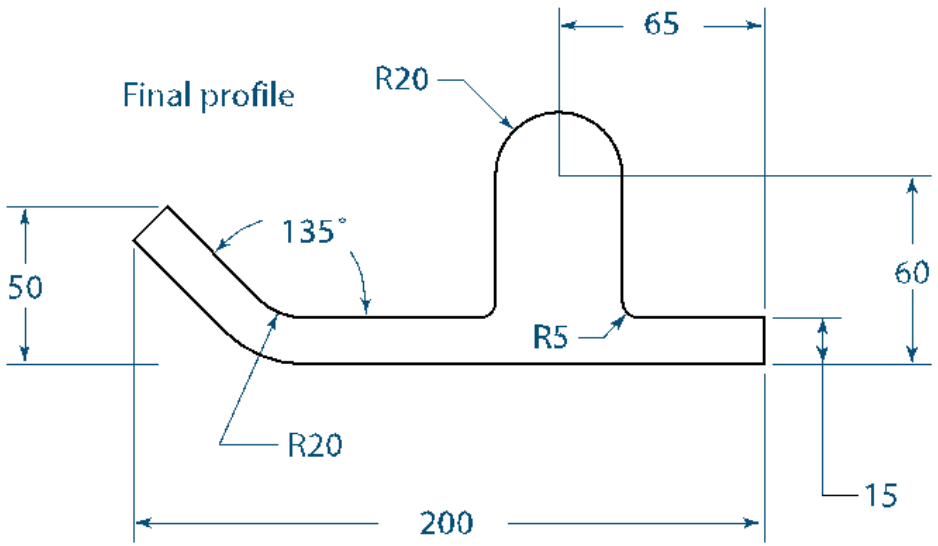




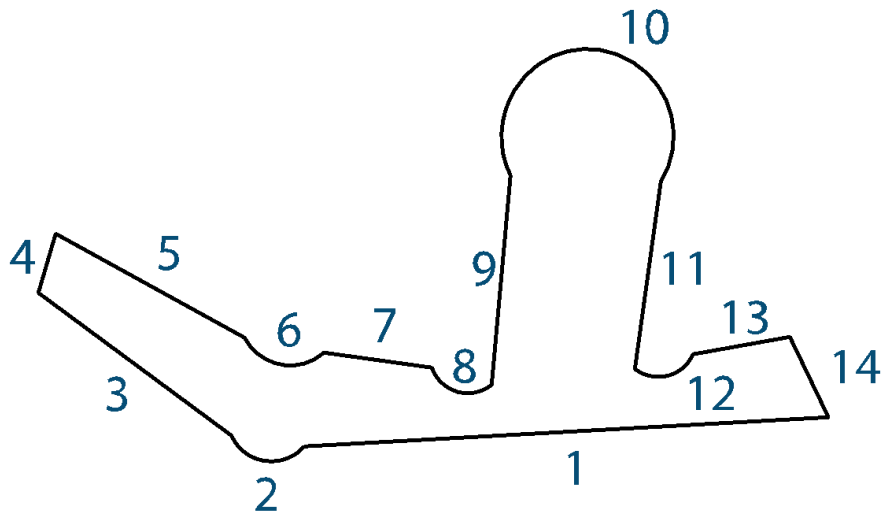
fully understanding the behavior of your model and the effects of your selection of dimensions and constraints is important.



(a)



(b)

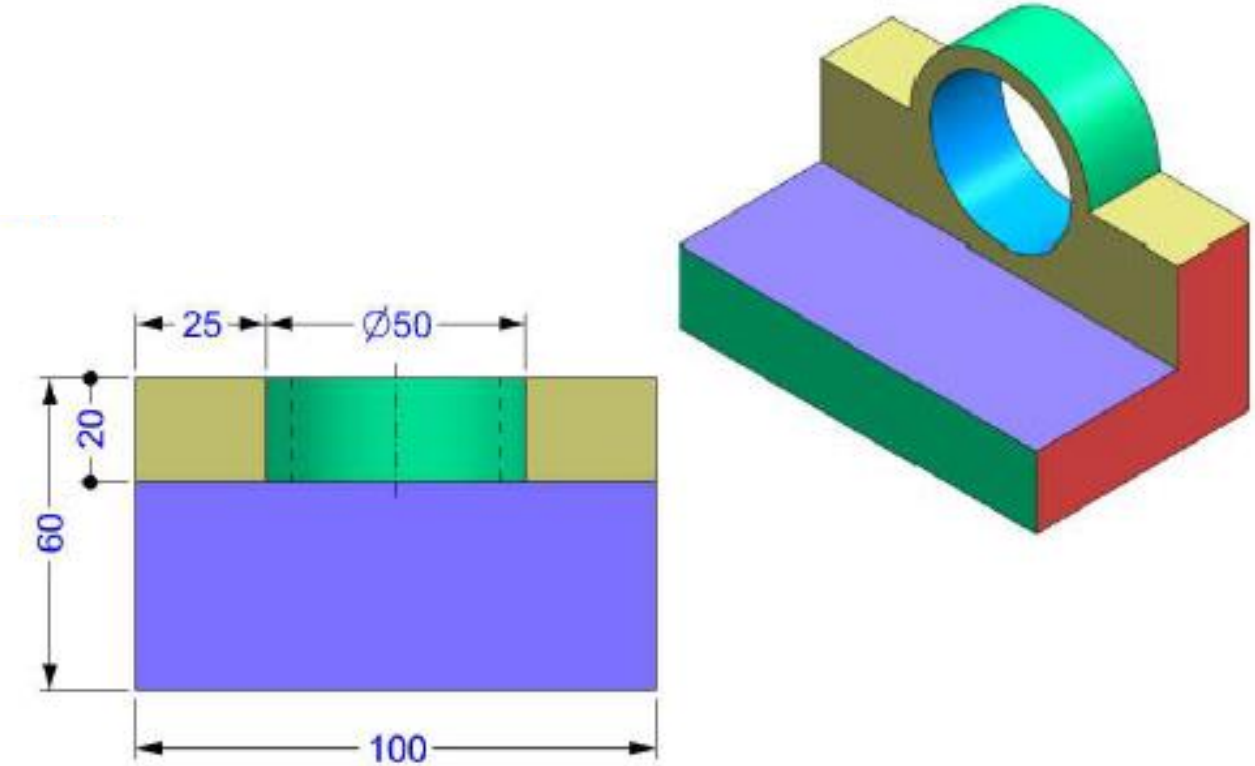


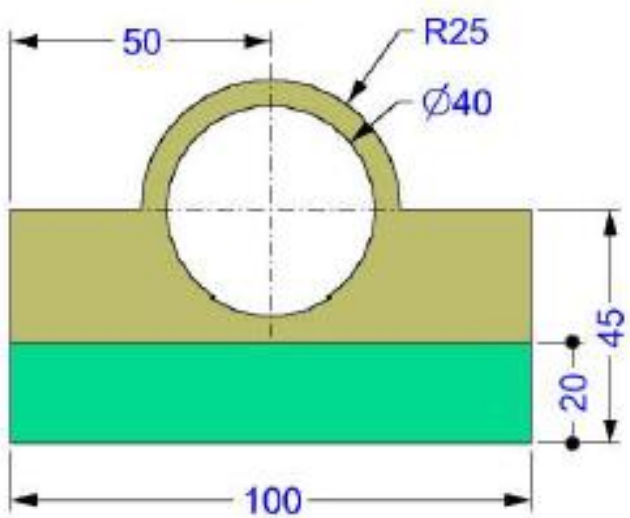
Segment	Constraint
1	Horizontal, Tangent to 2
2	Concentric with 6, Tangent to 1, Tangent to 3
3	Perpendicular to 4, Parallel to 5, Tangent to 2
4	Equal Length to 14, Perpendicular to 3
5	Parallel to 3, Tangent to 6
6	Concentric with 2, Tangent to 5, Tangent to 7
7	Horizontal, Tangent to 6, Tangent to 8
8	Tangent to 7, Tangent to 9
9	Vertical, Tangent to 8, Tangent to 10
10	Tangent to 9, Tangent to 11
11	Vertical, Tangent to 10, Tangent to 12
12	Tangent to 11, Tangent to 13
13	Horizontal, Tangent to 12
14	Vertical, Equal Length to 4

Constraint Strategy

- If a profile is over-constrained or under- constrained, it may not be possible to create a solid feature from the profile.
- The first constraint usually applied to a new sketch is a ground constraint. Ground constraints serve as anchors to fix the geometry in space.
  - a geometric entity such as a line or point on the profile having been made coincident with one of the basic modeling planes or with the origin of the coordinate system
- Next, geometric and dimensional constraints should be added and/or changed until the profile is fully constrained.

2D sketching in Fusion: First angle projections


25
Ø50
20
60
100


50
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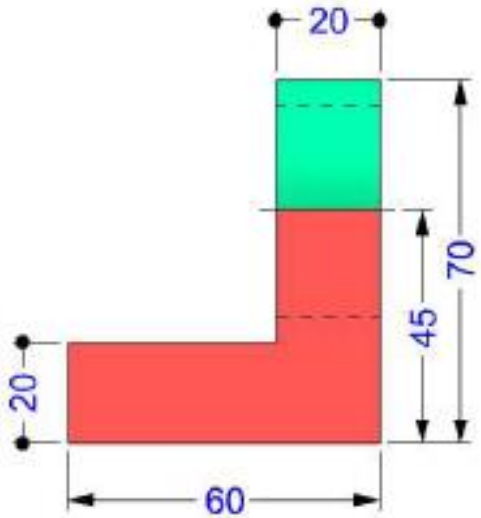
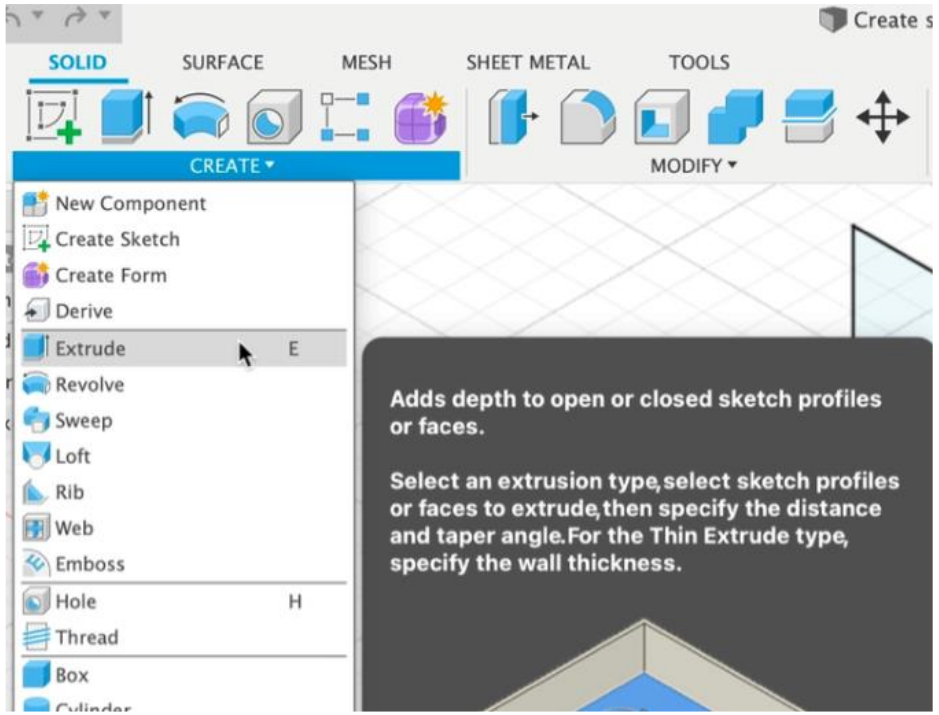

20
70
45
20
60

3D solid bodies in Fusion 360

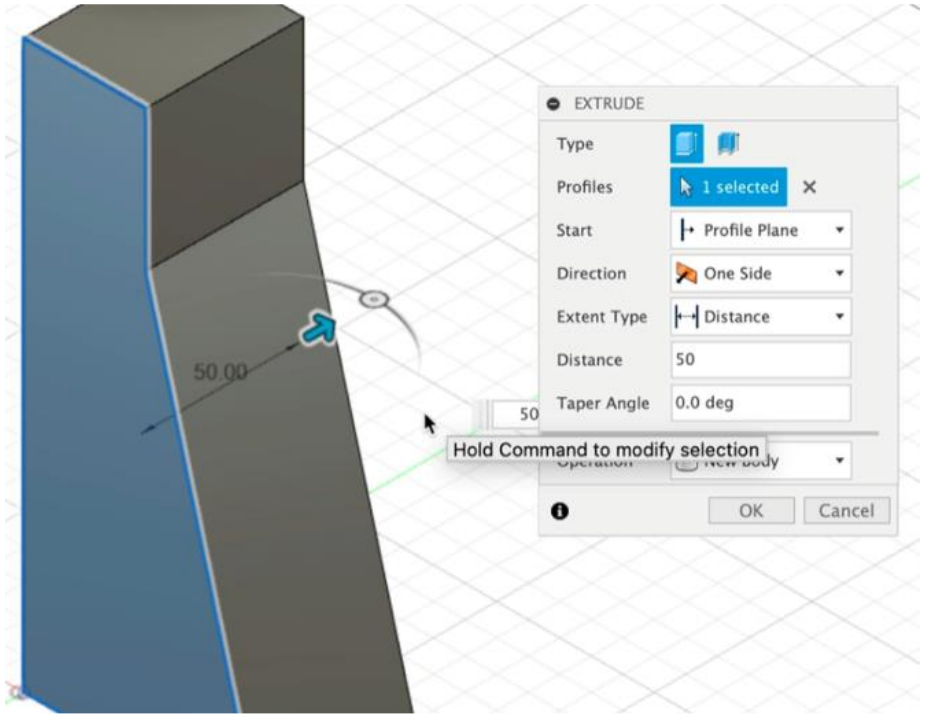
Learning objectives

- Turn a 2D sketch into a 3D body.
- Use the Extrude and Revolve commands.

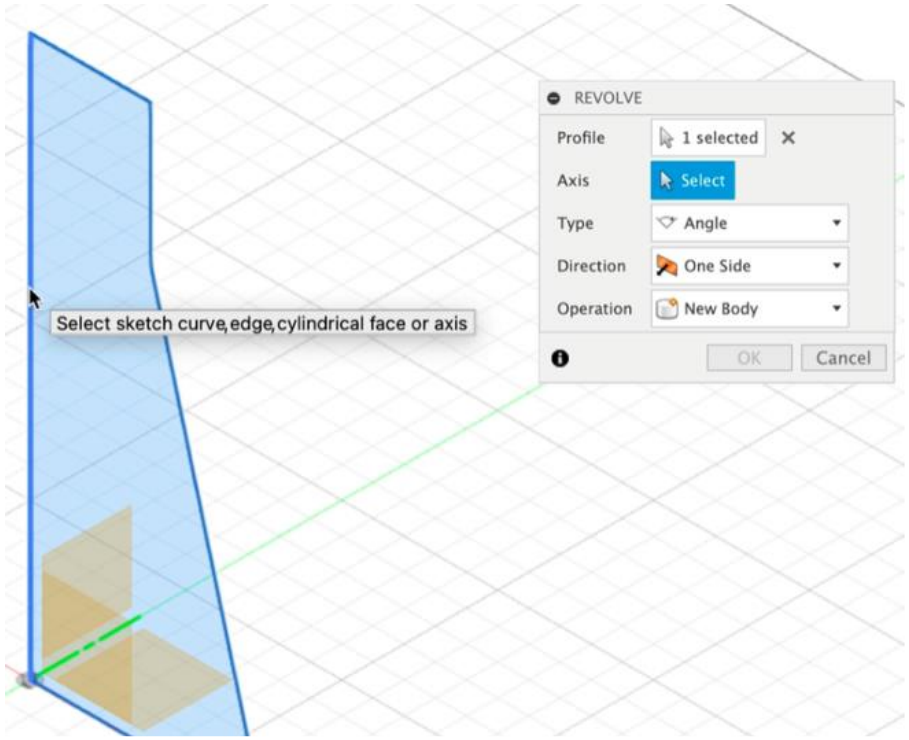
Select **Extrude** from the Toolbar to activate it, or press **E** on your keyboard.



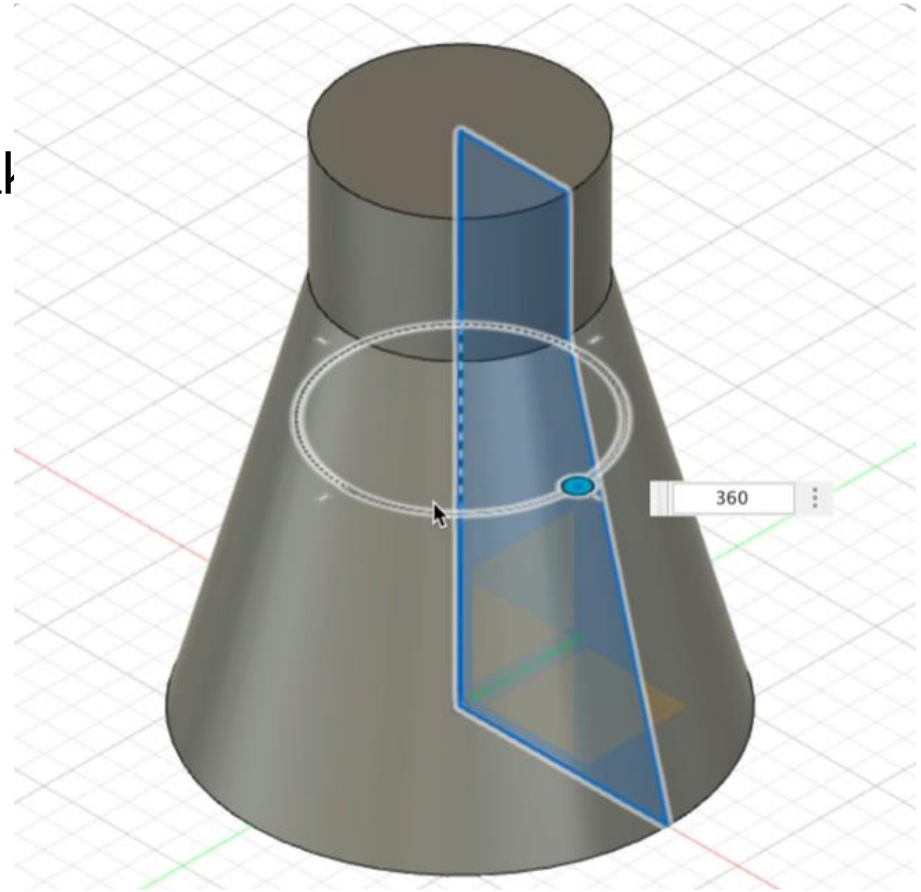
Extrude will automatically detect your closed sketch profile. You can then use the directional arrow or define the Extrude distance in the dialog. Notice how the Extrude command turns a 2D sketch into a solid 3-dimensional body.



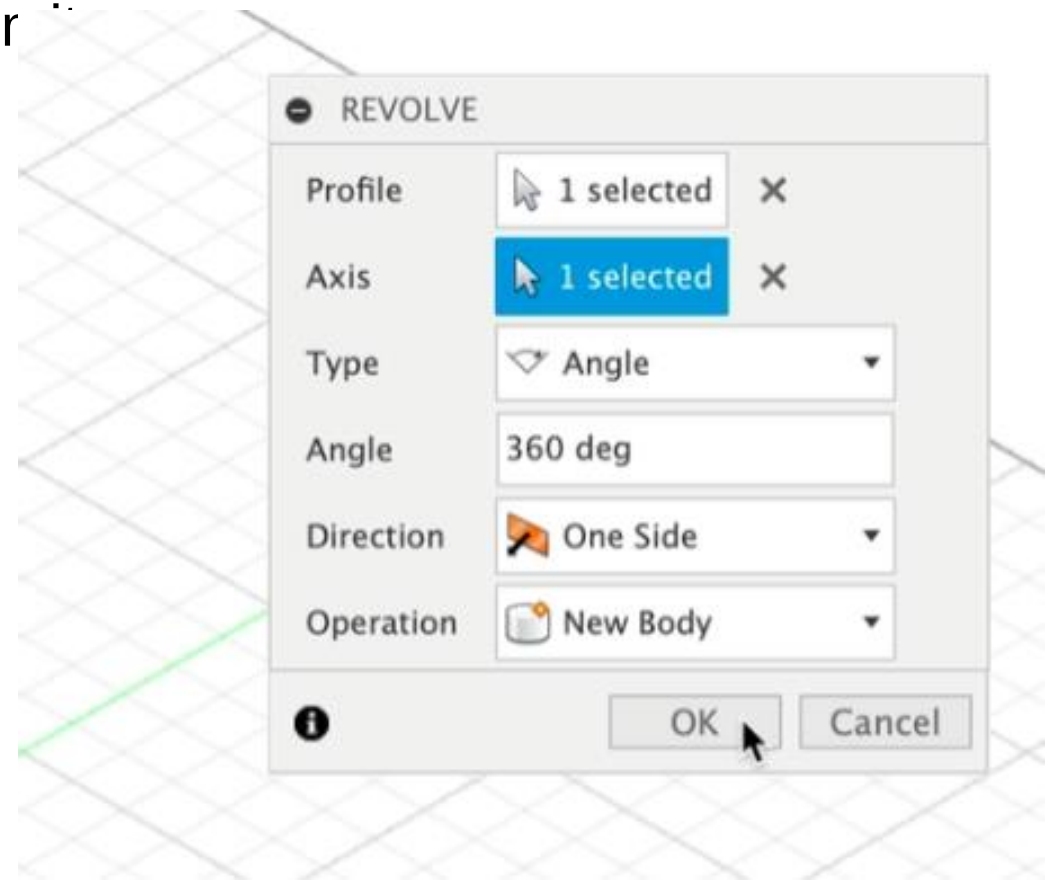
To make the tea kettle base symmetrical, cancel the **Extrude** command and activate the **Revolve** tool in the Toolbar. In the **Revolve** dialog, you can see you need to define the axis after defining the Profile. Select the tallest vertical line on the left.



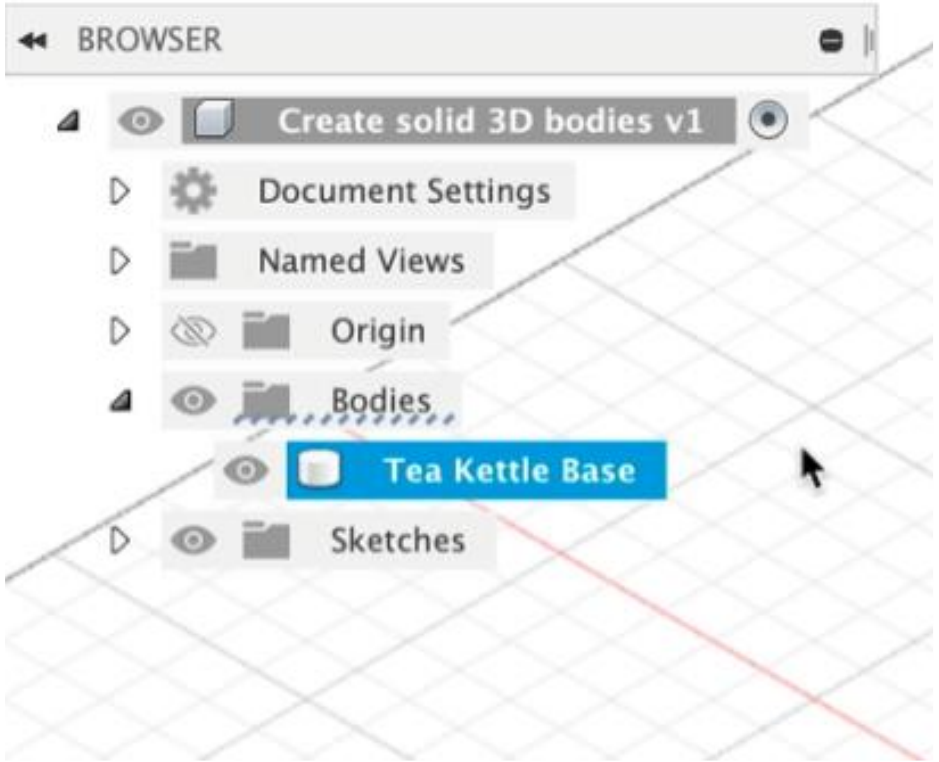
A typical tea kettle is symmetrical, so mak
sure to set the angle to **360 degrees**.



Select **OK** in the Revolve dialog to comm
the values.



In the Browser you'll see a bodies folder. Toggle the folder open to see your newly created 3D body. Rename each 3D body to keep track of them later on. This is important when working with large assembly models. To do this, select the body once, followed by a second time to edit the name. Press **Enter** to confirm.



Sweep and loft

Sweep a solid body

Sweep a profile along a path

1. On the toolbar, click **Solid > Create > Sweep**.

   The **Sweep** dialog displays.

   Type

   Profile

   Path

   Distance

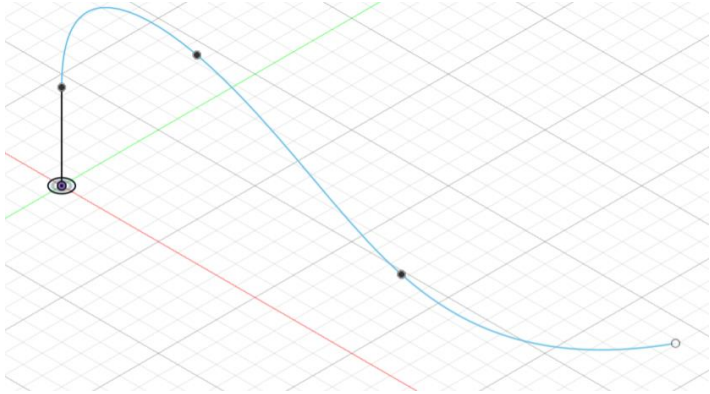
   Taper angle

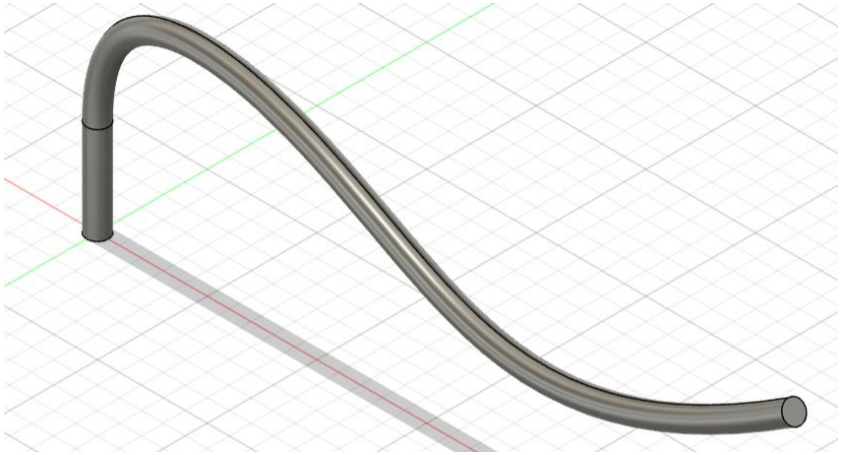
   Twister angle

   Orientation

   operation

Sketch a smooth path line in the front plane.
Sketch the sweep profile on the top plane.





Loft a solid body

1. On the toolbar, click **Solid > Create > Loft** .

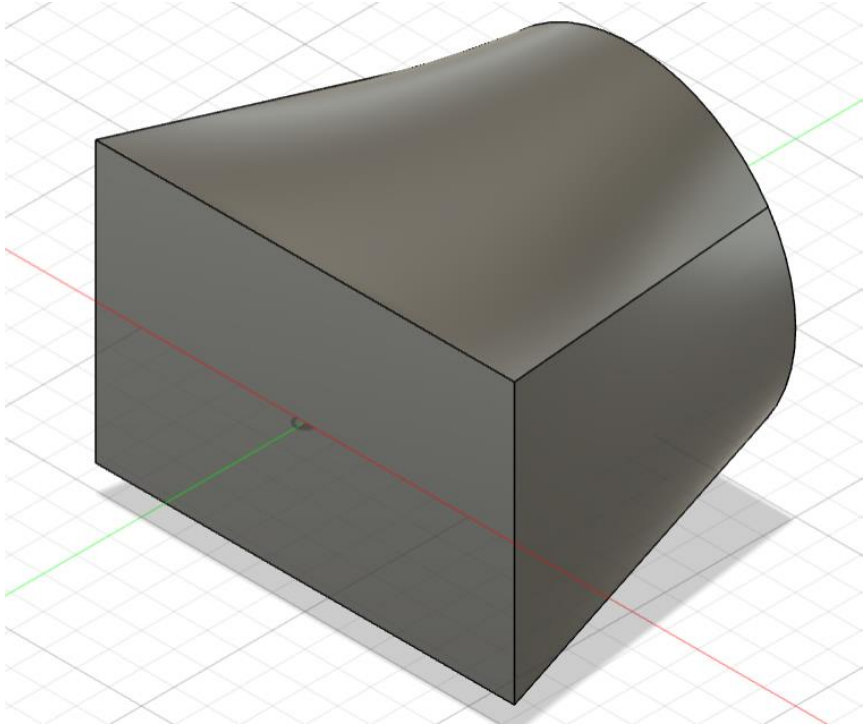
   The **Loft** dialog displays.

   Select two or more profiles

   Guide type

   Tangent edges

   operation



Sketch the second profile on an offset plane of the first profile plane