EDU[**CAD**]TION **STUDY GUIDE**

How to Fully Define CAD Sketches with Dimensions

Chapter 1, Lesson 6

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What are Dimensions?

- Dimensions are measurement rules that define the size, distance, or angle of sketch elements.

- They complement constraints by adding exact numerical values, such as millimeters, inches, or degrees.

- Together with constraints, they help turn blue (undefined) sketches into black (fully defined) ones.

How to Apply Dimensions

- You can apply dimensions by going to the **'Create'** dropdown and selecting 'Sketch Dimensions' or set your 'Sketch Dimension' tool (shortcut: D).

- Click on a line, circle, or angle to apply a dimension. **Note:** *Dimensions are automatically applied based on the type of object, circle, half-circle, or line, and will follow the current dimensioning rule you've set. (see Figure 1.1)*

- Enter a numerical value (e.g., 100 mm or 4 in).

- You can change units by going to Document Settings → Units → Change Active Units. A new window will appear where you can select the unit system you want to use. (see Figure 1.2)

- Dimensions can also perform math (e.g., typing '2+2' gives 4 in). To edit a dimension, simply double-click the numerical value with the left mouse button (see Figure 1.3).

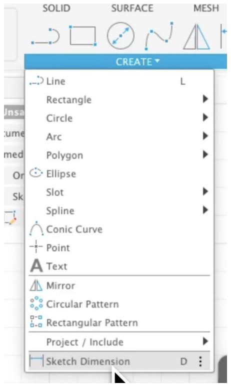

Figure 1.1

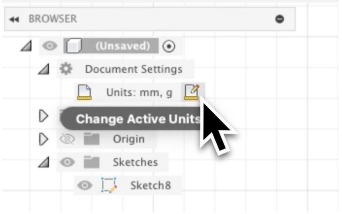

Figure 1.2

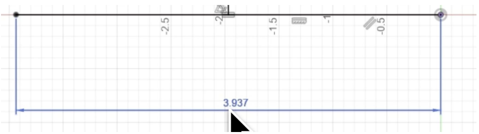

Figure 1.3

Key Features of Dimensions

- Define exact lengths, radii, diameters, and angles.
- Can be edited anytime by double-clicking on them.
- Allow switching between units (mm, in, cm) with automatic conversion.
- Work with constraints to fully lock geometry in place.
- Help identify what parts of a sketch are still undefined (blue). **Note:** *The easiest way to check is to click on a blue line and move it; this will reveal what hasn't been fully defined yet.*

Profiles

- A profile is a fully enclosed area of a sketch, such as the interior of a circle or a closed polygon.

- Profiles are essential because they can be selected and extruded to create 3D features. Dimensions help ensure profiles are accurate and robust for later modeling steps.

- We've added circles to our drawing to show profiles. You'll see that the enclosed area of the circle appears blue; this indicates a profile. (see Figure 1.4)

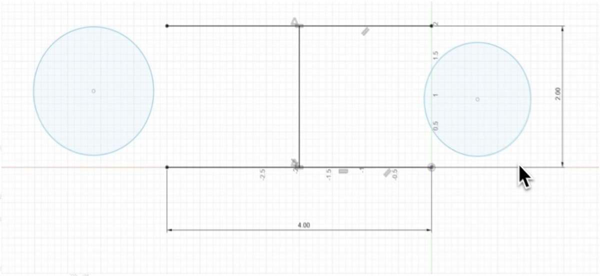

Figure 1.4

Advanced Tools (Tangent)

- **Tangent:** Ensures that a curve and another object touch at exactly one point without crossing each other. This creates a smooth transition between the two objects. You can apply this constraint by selecting **'Tangent'** under the '**Constraints**' menu, then choosing the two objects you want to make tangent(see Figure 1.5)

- When applied, the line will adjust its angle, or the curve will shift slightly, so they meet at a single point.

- If the curve and line are already close to tangent, applying the constraint will "lock in" that relationship.

Example: *As shown in Jake's drawing, the line touches the circle at only one point, forming a tangent.* (see Figure 1.6)

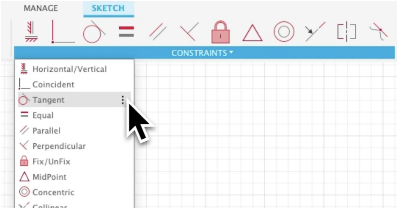

Figure 1.5

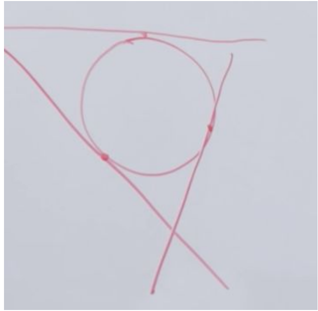

Figure 1.6

Advanced Tools (Mirror)

- **Mirror:** Allows you to create symmetrical sketches quickly by duplicating geometry. It works by reflecting selected sketch curves across a chosen sketch line, which acts as the "mirror line." You can apply the '**Mirror**' option by selecting Mirror from the '**Create**' menu. (see Figure 1.7)

- After selecting the '**Mirror**' option, a new box will pop up where you can select which object you want to mirror. (see Figure 1.8)

- The mirrored geometry is fully constrained to the original, so any changes to the original will also update the mirrored copy.

- This tool is especially useful when designing symmetrical parts, as it saves time and ensures accuracy.

Example*: If you draw one half of a part and then mirror it across a centerline, the other half is created automatically.* (see Figure 1.9)

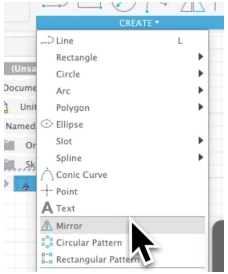

Figure 1.7

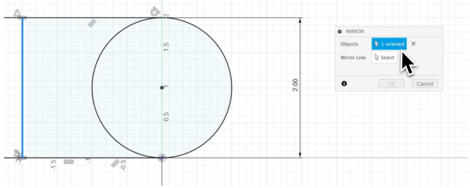

Figure 1.8

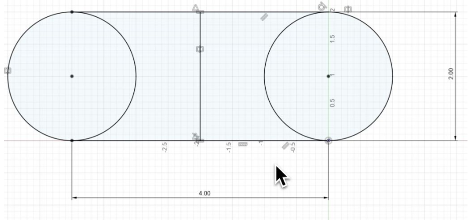

Figure 1.9

Summary

Without dimensions, sketches may remain unstable and imprecise. Dimensions lock geometry with exact values, ensuring models remain editable and reliable. They allow changes to propagate consistently across the sketch, creating flexible and scalable designs. The goal is always a robust, fully defined sketch where all lines and shapes turn black.

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