

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

How to Use Construction Lines, Patterns, Trim, and Extend in CAD

Chapter 1, Lesson 7

SendCutSend

Multiple Ways to Create Geometry

- In CAD there's always more than one way to do something. Even simple parts like a washer can be sketched in more than one way, you can choose the '**Two Circles**' or '**Offset**' method.

- **Two Circles Method:** Draw an outer circle (e.g., 100 mm diameter) and an inner circle (e.g., 60 mm diameter), then define both with dimensions. (see Figure 1.1)

- **Offset Method:** First, draw the outer circle. Next, select the '**Offset**' tool underneath '**Modify**' to create an inner circle at a fixed distance (see Figure 1.2). Then click the outer circle, drag it inward, and enter your desired offset distance (see Figure 1.3). In this method, it automatically creates the constraint of having them concentric.

- Both approaches yield the same washer profile, but the offset automatically maintains a concentric relationship

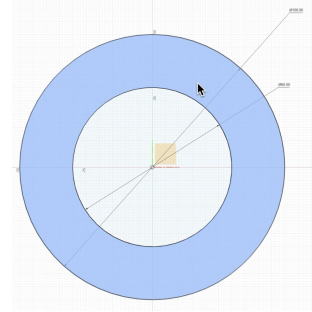


Figure 1.1

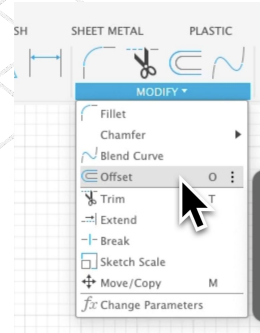


Figure 1.2

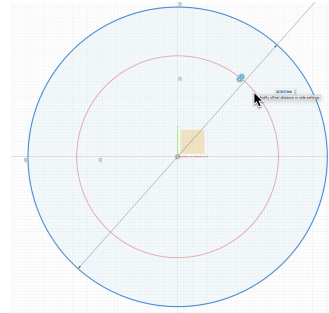


Figure 1.3

Construction Lines

- Construction lines are reference lines that do not affect profiles. They are displayed as dotted lines and are often used for alignment and symmetry.
- **Example:** Start by drawing a line down the center of the washer and apply two constraints: '**Vertical**' and '**Midpoint**'. This ensures the line is vertical, passes through the origin, and bisects the circle.
- To turn this line into a '**Construction Line**', you'll go to the Sketch Palette and select '**Construction**' under Linetype (see Figure 1.4). The line will change to a dotted style, indicating that it is now a '**Construction Line**' (see Figure 1.5).
- This line no longer affects the washer's overall geometry but can still be used as a reference for positioning and constraints.

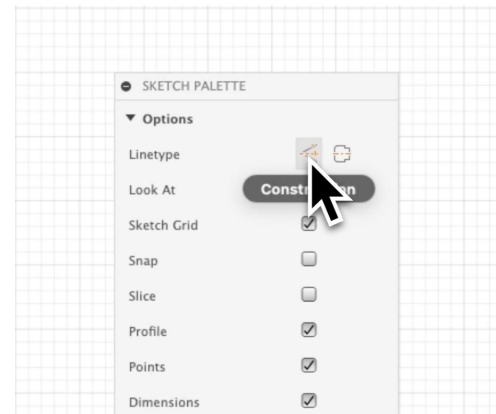


Figure 1.4

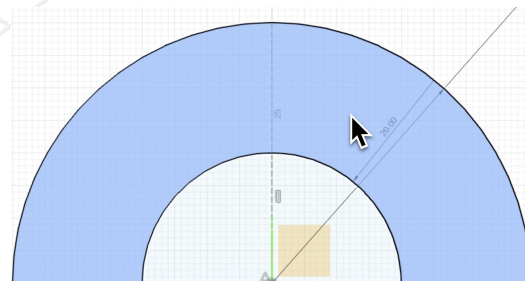


Figure 1.5

Creating Keys with Mirror

- **'Keys'** or **'Cutouts'** can be sketched on one side of a construction line and mirrored to the other side for symmetry.
 - First, click **'Line'** in the **'Create'** menu. Select your construction line, make sure it's perpendicular, and draw the two lines in (see Figure 1.6).
 - Next, define the sketch by selecting **'Sketch Dimension'** under **'Create'**. Add a dimension from the origin equal to half the width of your key (see Figure 1.7).
- Note:** This is important because the mirror tool will duplicate it to the other side.

- Finally, use the **'Mirror'** tool under **'Create'** (see Figure 1.8). Select the shape and the construction line as the mirror line, then click OK in the mirror dialog box (see Figure 1.9). This ensures symmetry and saves time by reducing repetitive sketching.

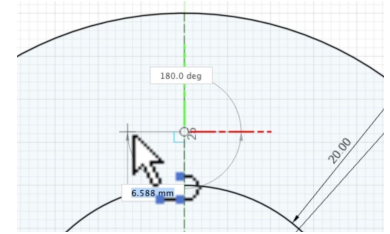


Figure 1.6

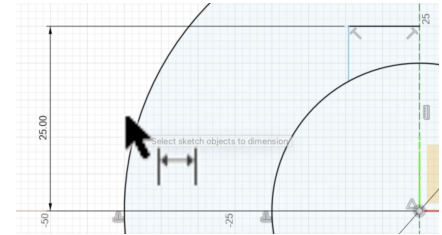


Figure 1.7

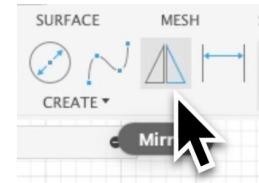


Figure 1.8

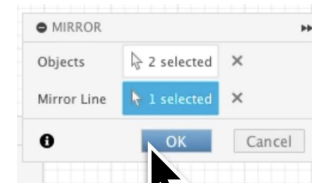


Figure 1.9

Patterns

- Now that we've created our first key, we still need two more. There are multiple ways to do this, but we'll use the Pattern tool, which efficiently replicates sketch geometry.
 - **Circular Pattern:** Duplicates features around a center point (e.g., creating three evenly spaced keys in a washer) (see Figure 1.10).
 - **Rectangular Pattern:** Creates arrays of features in rows and columns. Patterns save time and ensure design consistency (see Figure 1.11).
 - For our washer, we'll use the Circular Pattern. Go to **'Create' → 'Circular Pattern'**
 - Select the lines you want to duplicate, then select the origin as the center point. Choose the distribution and quantity. (see Figure 1.12)
- Note:** the quantity always includes the original geometry.

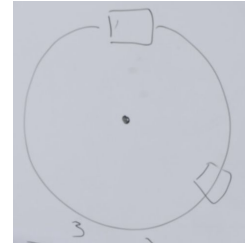


Figure 1.10



Figure 1.11

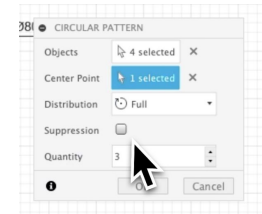


Figure 1.12

Trim and Extend

- **Trim Tool:** Removes unnecessary parts of sketch geometry, such as cutting away circle segments inside a profile. You'll find it under '**Modify**' → '**Trim**'. Click the line you want to '**trim**', and it will highlight in red to show what will be removed (see Figure 1.13).
- After trimming, you may notice the offset is lost and the inner circle turns blue. To fix this, select '**Dimension**' and re-add the reference to the outer circle.
- **Extend Tool:** The opposite of Trim. It lengthens geometry so it meets another line or curve.
- These tools help clean up sketches, keeping profiles simpler and more reliable for extrusion into 3D parts.

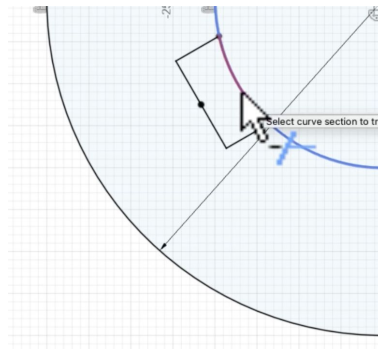


Figure 1.13

Summary

Learning multiple methods for sketching builds flexibility and problem-solving skills in CAD. Different approaches may be better suited depending on design goals, complexity, or ease of modification. Tools like construction lines, patterns, trim, and extend help create cleaner, more robust sketches that are easier to edit later.

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