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

How to Use Constraints in CAD to Define and Control Sketches

Chapter 1, Lesson 5

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What are constraints?

- Constraints are rules in CAD that control the behavior and relationships of sketch geometry. They ensure that sketches are not chaotic by defining how lines, points, and shapes interact.
- A fully defined sketch turns from blue (undefined) to black (defined).
- If the sketch is blue, you are able to move it anywhere on the page, change the orientation, or angle. (see Figure 1.1)
- To fully define your sketch, you'll need to apply constraints, which can be found in the **'Toolbar'**. (see Figure 1.2)
- The best way to learn constraints is to read through their descriptions so you can understand what each one does.

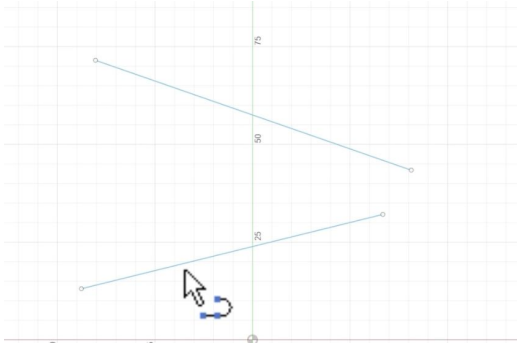

Figure 1.1

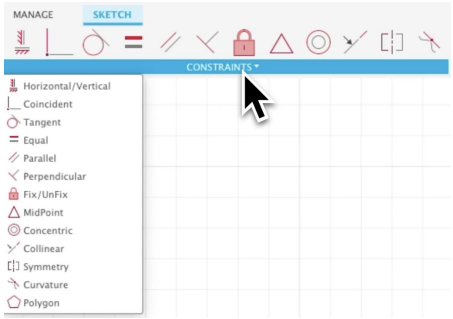

Figure 1.2

Types of Constraints

•**Parallel:** Constrains two lines so that they extend in the same direction and never intersect. (see Figure 1.3)

• **Horizontal/Vertical:** Forces a line to align horizontally or vertically, depending on its angle. **Important Note:** If the line's angle is greater than 45°, the constraint will make it vertical. If the line's angle is less than 45°, it will make it horizontal. (see Figure 1.4)

• **Coincident:** Attaches a point (e.g., line endpoint) to another point or the origin.
Note: *You'll see that once we've attached the sketch to the origin, the only thing that can still be adjusted is its length, since that dimension hasn't been defined yet. (see Figure 1.5)*

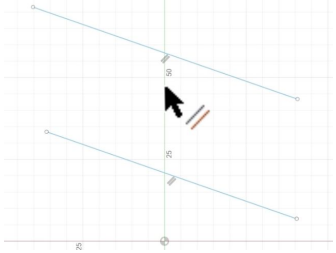

Figure 1.3

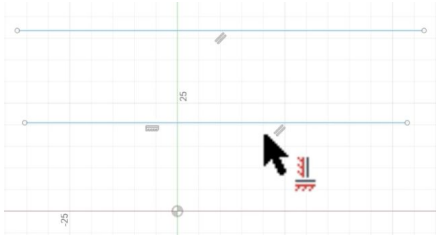

Figure 1.4

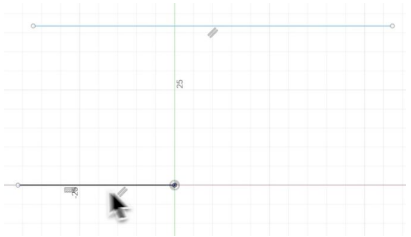

Figure 1.5

Types of Constraints Contd.

• **Equal:** Forces two lines (or shapes) to always be the same length/size. When you adjust one line, the other will adjust to match. (see Figure 1.6)

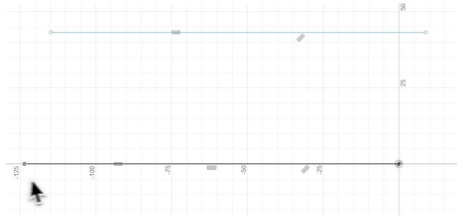

Figure 1.6

• **Midpoint:** Snaps a point to the exact middle of a line. In our lesson, we've added a line to add to midpoint. (see Figure 1.7)

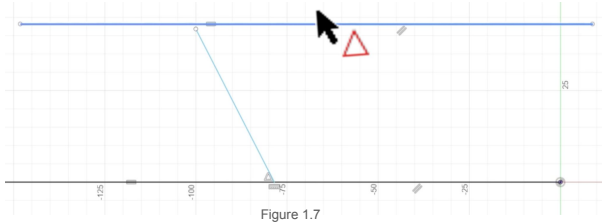

Figure 1.7

• **Perpendicular:** Forces two lines to meet at 90 degrees. (see Figure 1.8)

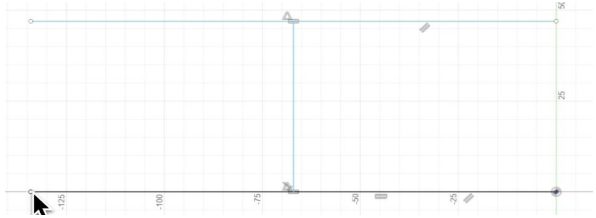

Figure 1.8

Note: *Each rule (or constraint) you add further limits how the sketch can move or change, tightening up its behavior.*

Important Note: *Adding constraints alone does not automatically make a sketch fully defined. The sketch can still move freely within the limits of the constraints you've applied. When applying a constraint, the first object you select becomes the reference, and the second object will adjust and orient itself to match the first.*

Summary

Without constraints, sketches are unstable and geometry floats freely (blue). Constraints add rules, ensuring models behave predictably and remain editable. They are a critical step before applying dimensions, which complete the process of fully defining sketches (turning all lines black). Mastering constraints prevents errors, saves time, and ensures reliable CAD designs.

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