

Practice Exercise – Dual Lever Sketch

Introduction to Sketching, Introduction to SOLIDWORKS



Summary:

In this exercise, you'll create a sketch using some common sketch tools, such as lines, circles, and arcs. Then you'll add dimensions and constraints to the appropriate sketch entities. As long the correct dimensions and constraints are added, the part should be fully-defined and the sketch elements will stay in place.

Reference Lessons:

- [Working With Sketch Tools](#)
- [Sketching Rectangles and Circles](#)
- [Geometric Relations](#)
- [Dimensions](#)

Instructions:

1. Open a new part file in SOLIDWORKS using millimeter dimensions.
2. Create the sketch profile on the XY plane as shown in the image below. Be sure to add the same dimensions and constrain the sketch elements as needed.

