Practice Exercise – Swivel

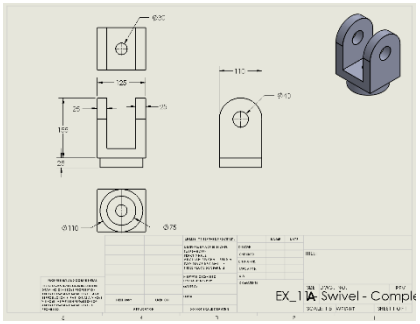
Drawings, Introduction to SOLIDWORKS

Summary:

In this exercise, you'll create a Swivel part using the drawing views supplied in the instructions. Once the part is created, you'll create a drawing using a Base view and specific projected views. Then dimensions will be added to those views in the drawing.

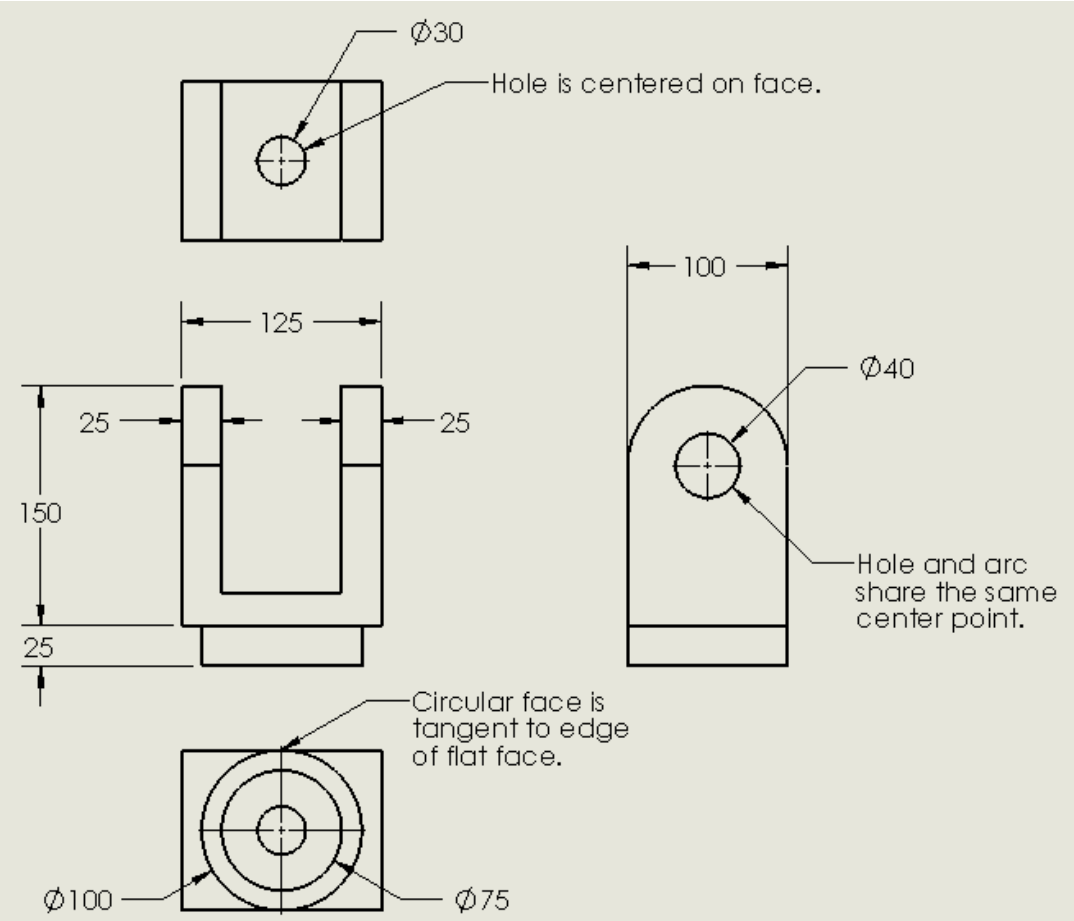
Reference Lessons:

- Drawings Overview - Starting a New Drawing
- Model Views
- Projected Views
- Dimensions

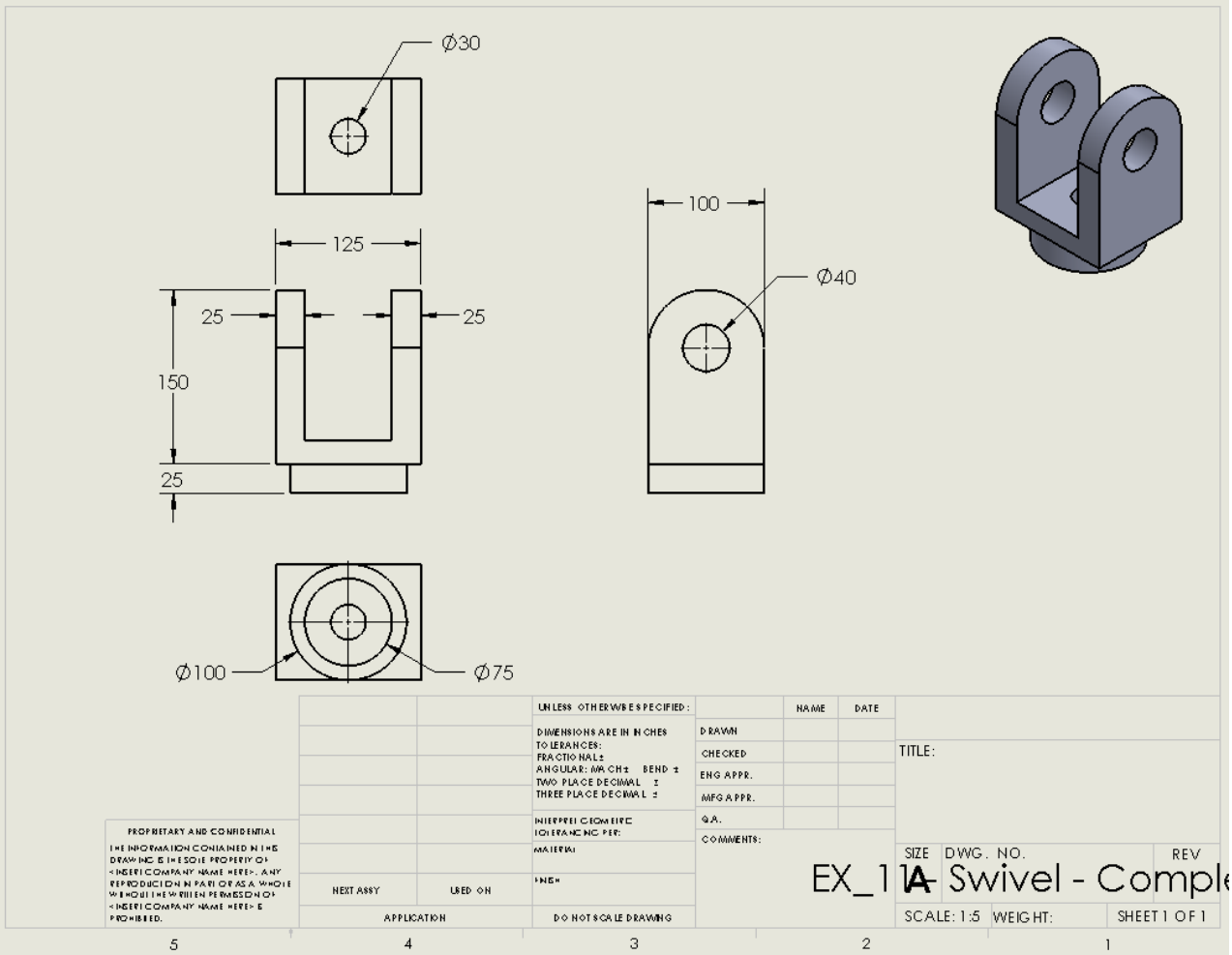


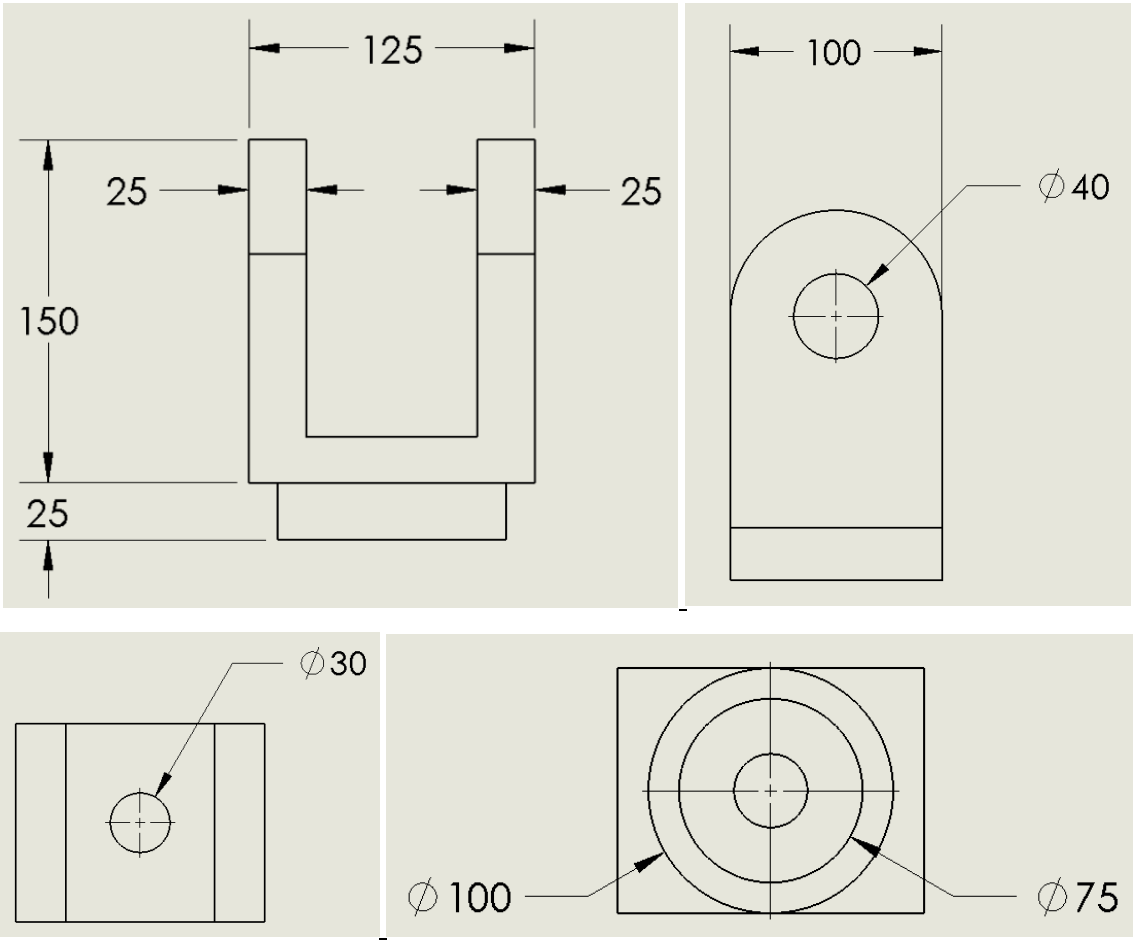
Instructions:

1. Open a new part file in SOLIDWORKS using millimeter units.
2. Create the part file as shown in the image below:



3. Create the drawing with all the same dimensions and views as shown in the images below (units are in millimeters):




125
25
25
150
25
100
∅40
∅30
∅100
∅75