

Practice Exercise – Dual Hose Fitting

Drawings, Introduction to SOLIDWORKS



Summary:

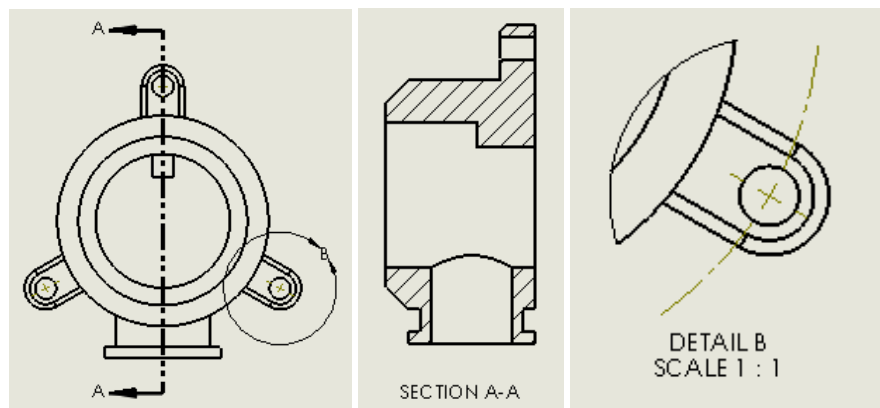
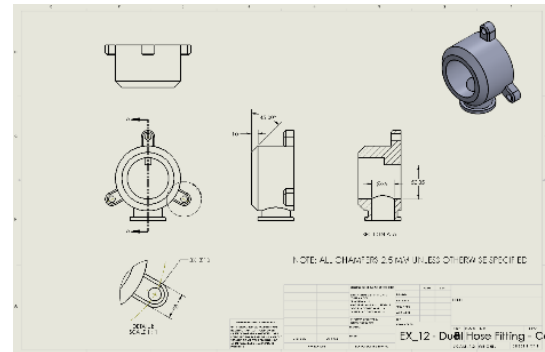
In this exercise, you'll use an existing drawing and add a section view and a detail view. Those views will contain dimensions as well as some annotations.

Reference Lessons:

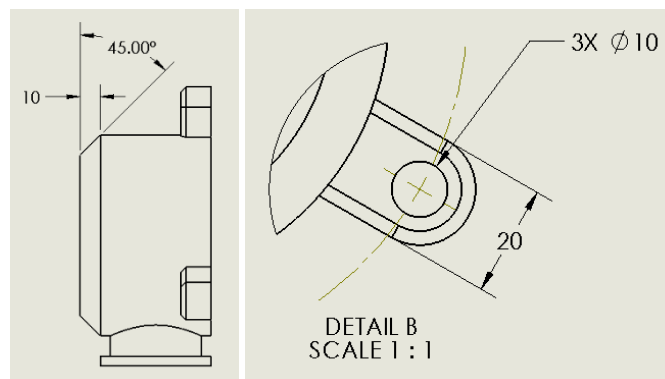
- [Section View](#)
- [Detail View](#)
- [Annotations](#)
- [Dimensions](#)

Instructions:

1. Download and unzip the files found [here](#).
2. Open the file "Dual Hose Fitting.SLDDRW".
3. Create the section view and detail view as shown in the images below:



4. Add a couple dimensions to the Right view and the Detail view and add an annotation to the drawing as shown in the images below:



NOTE: ALL CHAMFERS 2.5 MM UNLESS OTHERWISE SPECIFIED