Assembly in SolidWorks
Creating an Assembly
Inserting Parts

- Click “Browse if parts do not show in dialog box”
The Assembly Tree

- The design tree stores all information regarding the parts, mates, materials, and history of the assembly.
- Very useful for manipulating parts.
Move Component
Design for Manufacturing

- Tolerances
- Manifold vs Non-manifold
- Part Influence on Assembly
- Assembly Steps
Geometric Dimensioning and Tolerancing

- Tolerances in a design tell the inspector how much variance or imperfection is allowable before the part must be considered unfit for use.

- Tolerance is the difference between the maximum and minimum limits on the dimensions of the part.

- Since parts are never perfect, a datum feature is used during inspection, to substitute for the perfect datum of the drawing.

- Datum features are simply referred to as datums.
Plus / Minus Tolerancing

- When the part is produced in a manufacturing process, there will be errors.
- Even though most errors are undetectable to our eye, the variations can be picked up using precise measurements such as a CMM.
Manifold vs. Non-Manifold

๏ Think of it as “Manufacturable” vs “Non-Manufacturable”

๏ Can this part be manufactured?
Part Influence on Assembly

- Minimize total parts
- Use as many standard components as possible
- Custom parts increase manufacturing cost
- Design parts in a modular fashion
- Integrate common parts across product lines
Assembly Design

- **Bottom-up Design**
  - Traditional method
  - Parts are designed first then inserted assembly and mated into position
  - Ideal for previously constructed and standard parts

- **Top-down Design**
  - Referred as “in context design”
  - Parts’ shapes, sizes, and locations designed can be designed in assembly
Activity

๏ Complete the Lesson 2: Assemblies tutorial

๏ Assemble the Vise
  - If your parts do not fit correctly, use the parts that are shared with you to make the assembly