Assembly in Solidworks
Design for Manufacturing

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

- Tolerances in a design tell the manufacturer and inspector how much variance or imperfection is allowable.

- 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.

- Top/Front/Right planes in parts and/or assemblies are often used 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
Creating an Assembly
Inserting and Mate Parts

- Insert Components
- Once part selected press Enter key
- Add Mates
- Most mates are the same as sketch relationship
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 and Rotate Component(s)

- Right mouse button to rotate parts
- Middle mouse button to rotate assembly
Activity

- Assemble the Vise
- Use parts provided on website
- Using your own 3D models is optional
- 1 Vise per pod
- 1 per person is optional