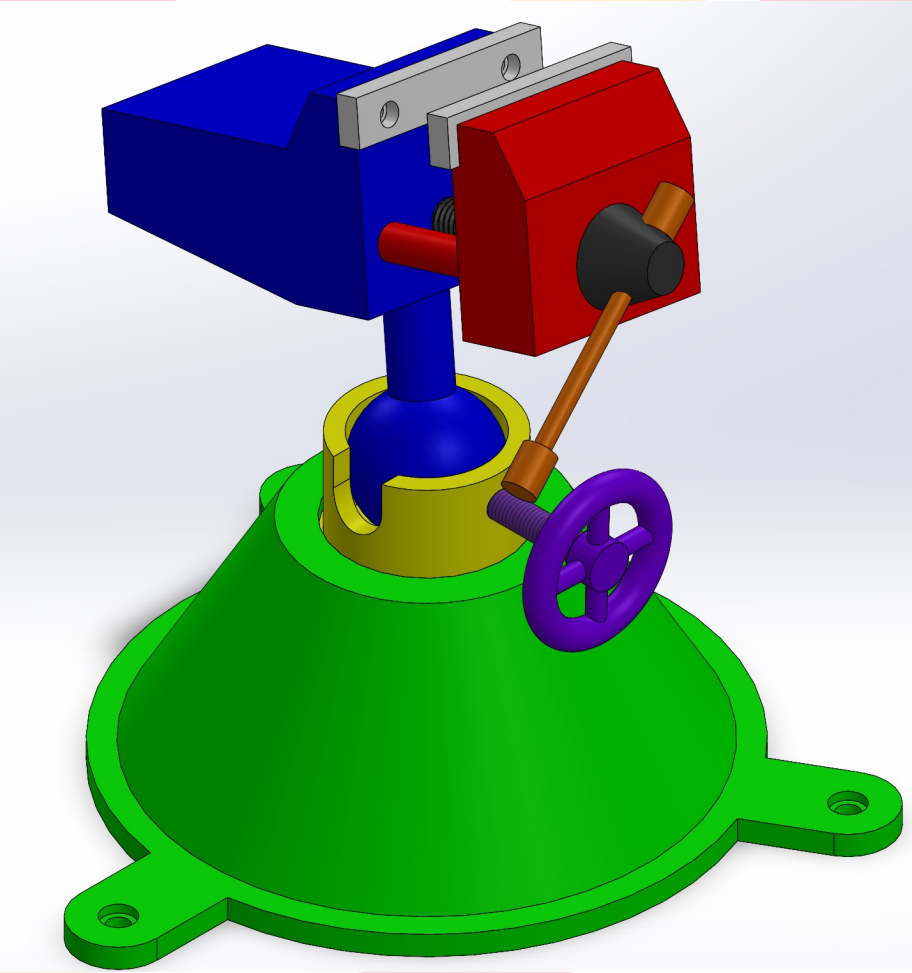
Assembly in SolidWorks

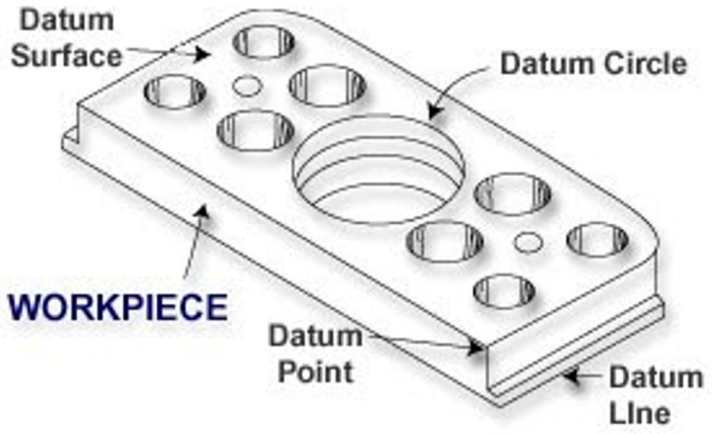


Design for Manufacturing

- Tolerances
- Manifold vs Non-manifold
- Part Influence on Assembly
- Assembly Steps

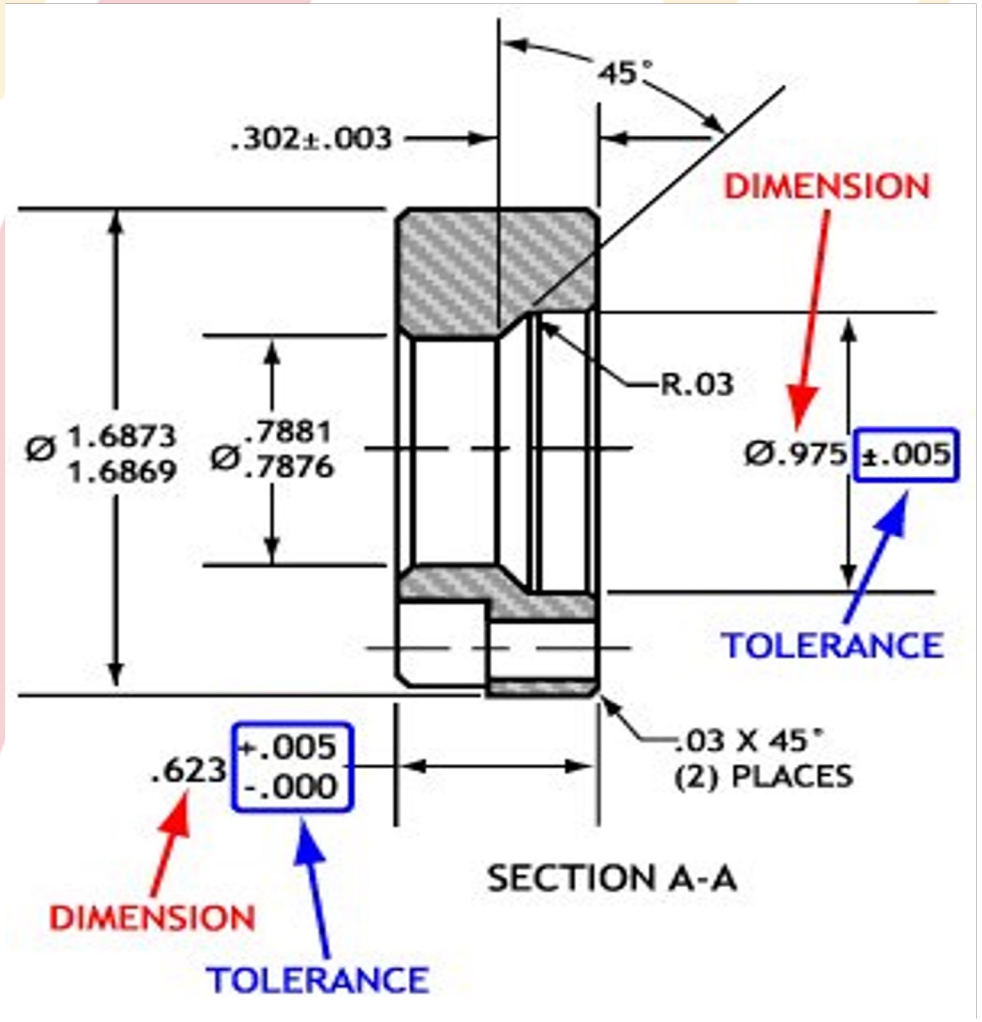
Geometric Dimensioning and Tolerancing

- Tolerances in a design tell the 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



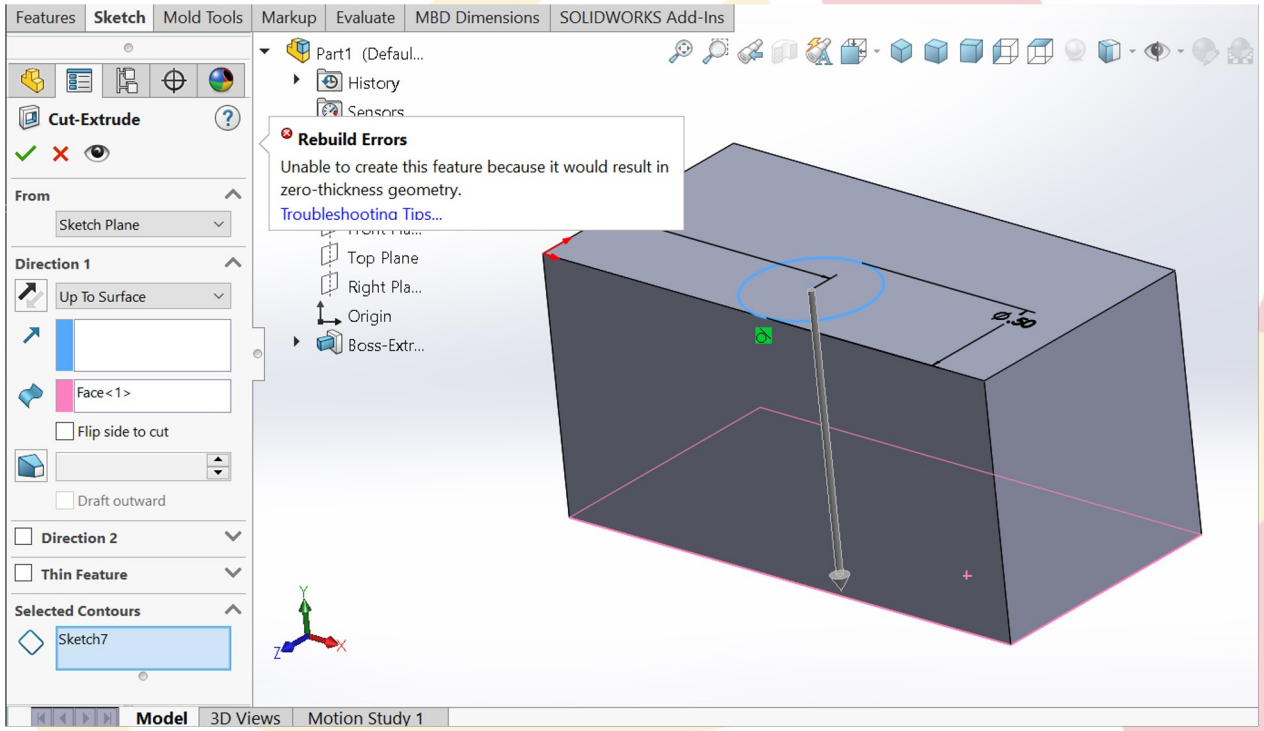
Plus / Minus Tolerancing

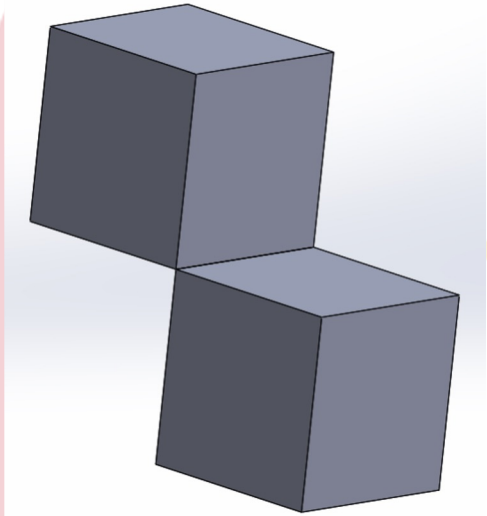
- When the part is produced in a manufacturing process, there will be errors.
- Even though most errors are undetectable to our eye, the variations can be picked up using precise measurements such as a CMM.

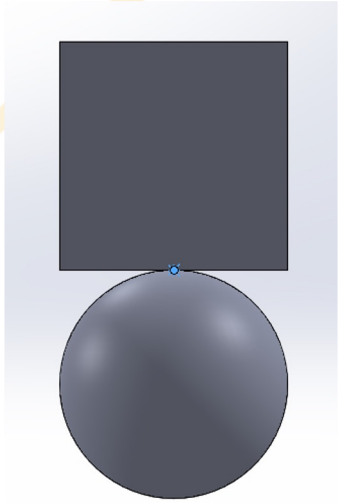


Manifold vs. Non-Manifold

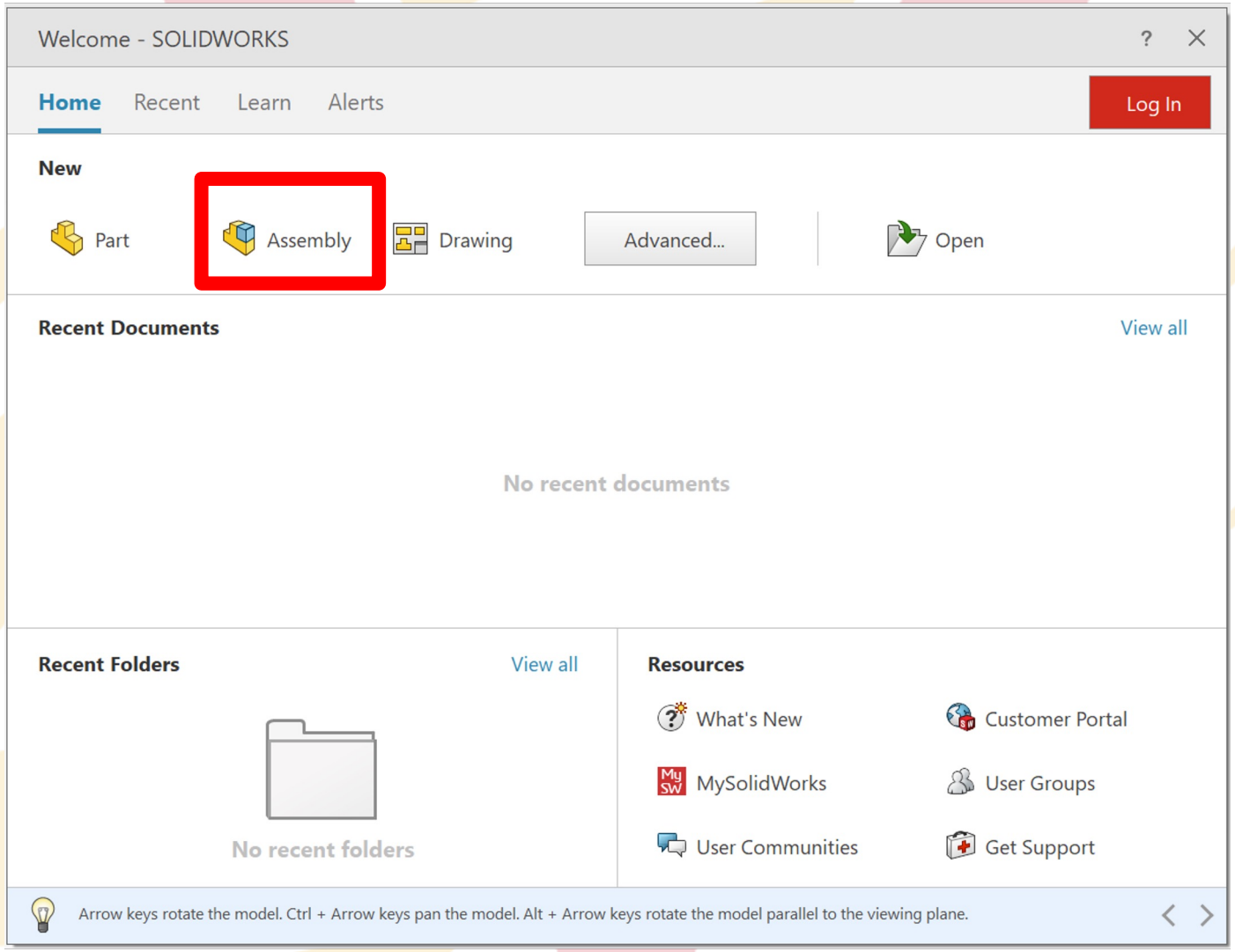
- Think of it as “Manufacturable” vs “Non-Manufacturable”
- Can this part be manufactured?





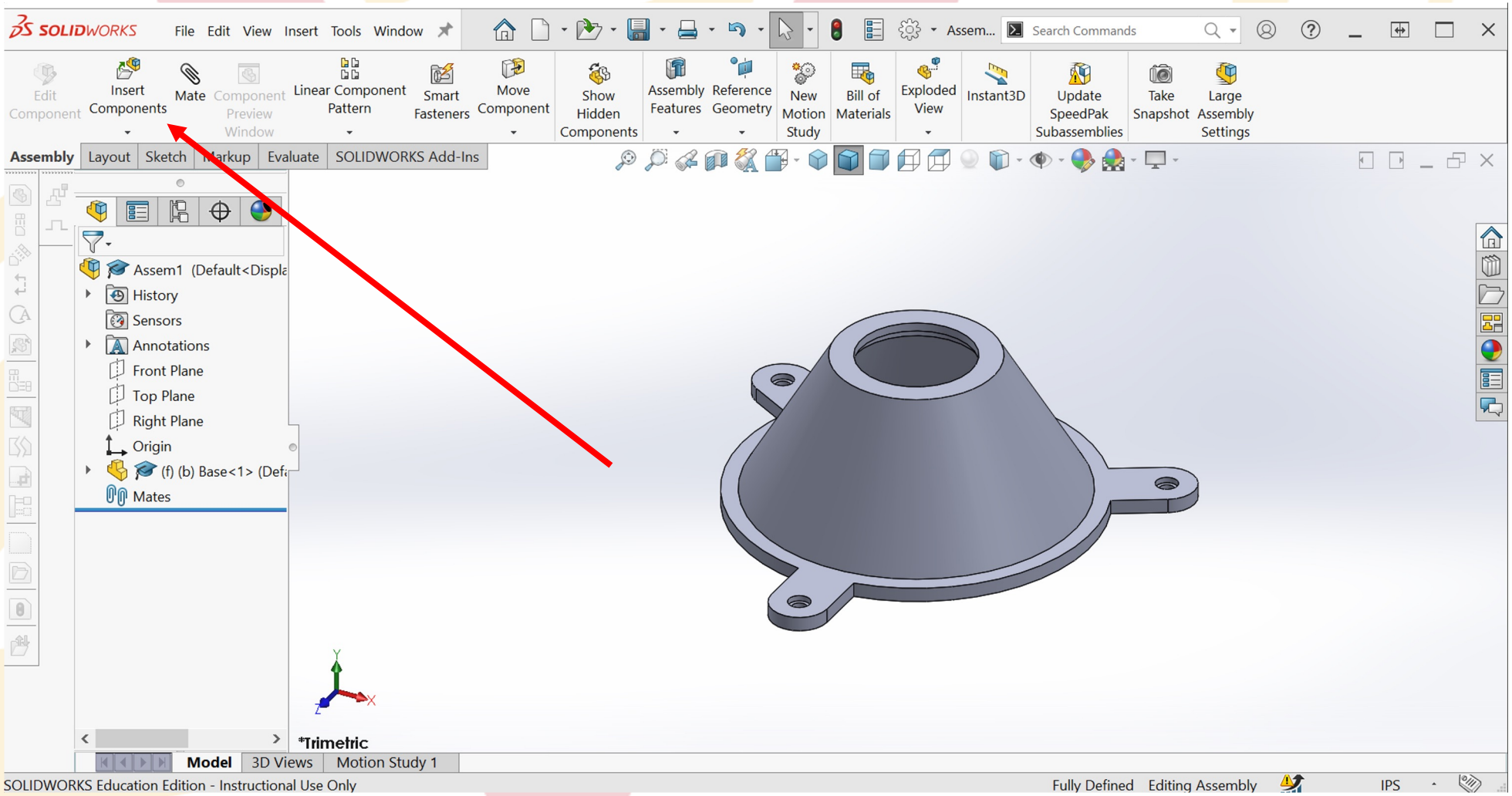


Creating an Assembly



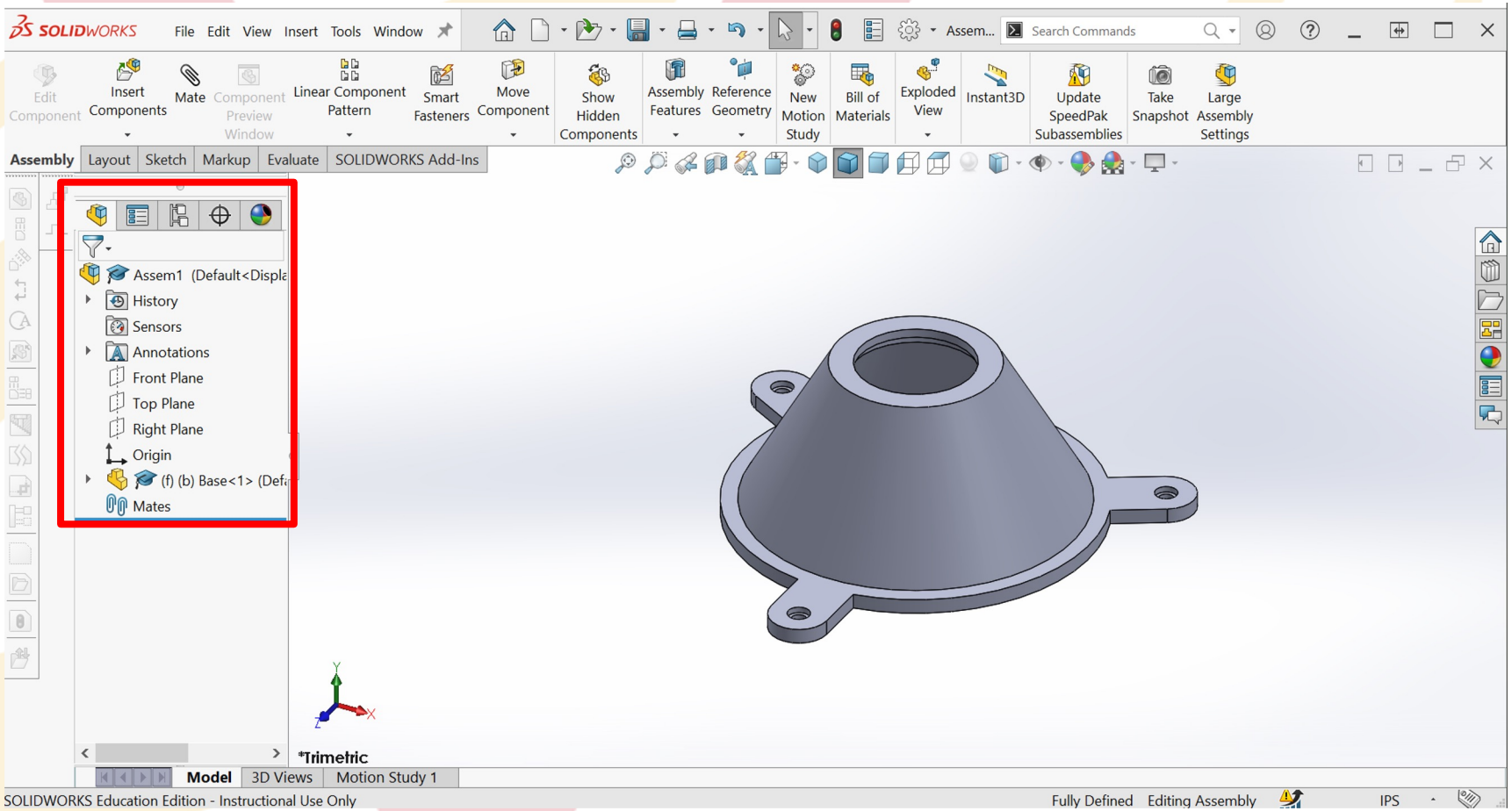
Inserting Parts

- Click “Browse if parts do not show in dialog box

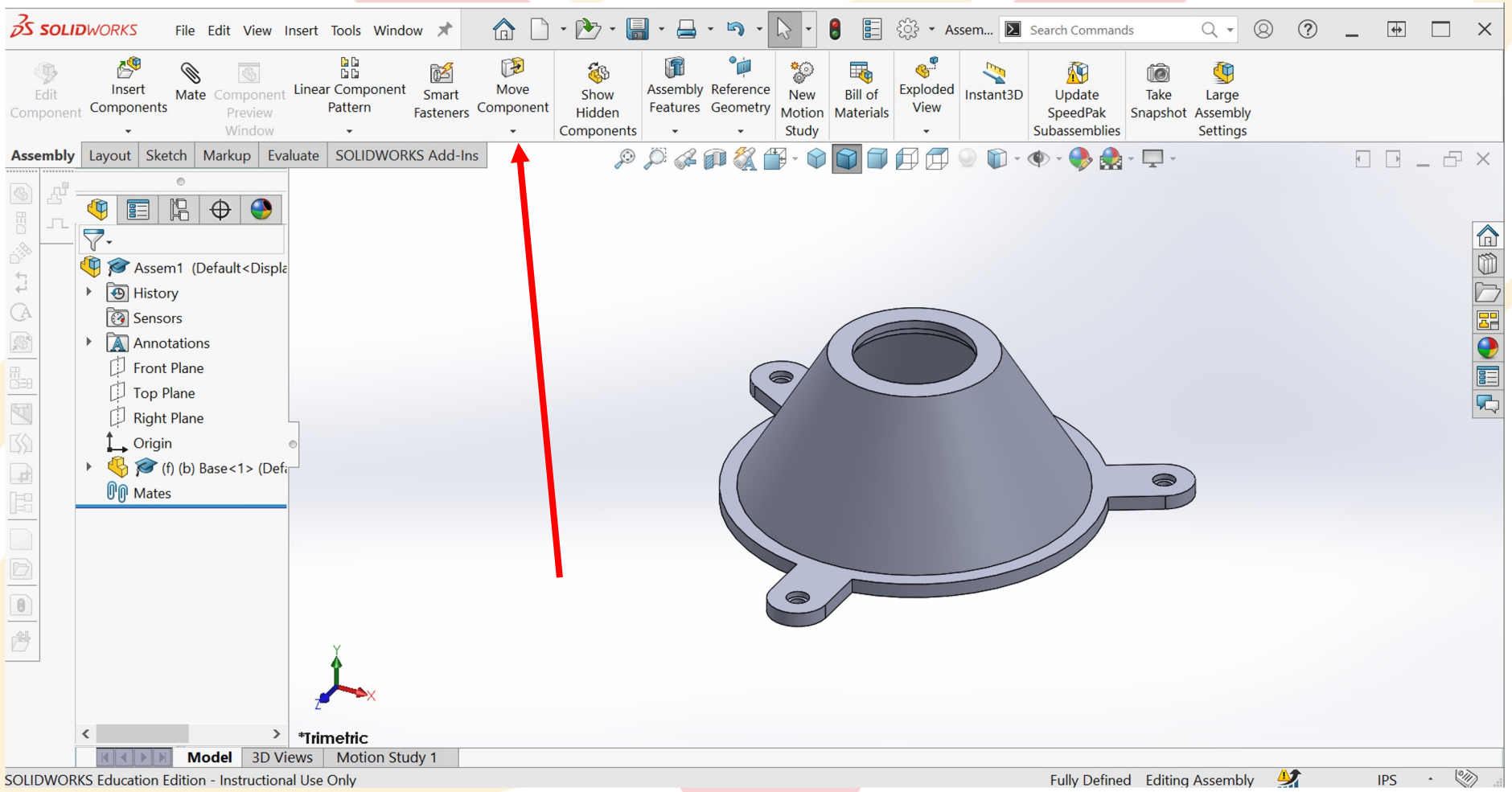


The Assembly Tree

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



Move Component



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

Vice Assembly