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

Welcome - SOLIDWORKS				? ×
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Arrow keys rotate the model. Ctrl + Arrow keys pan the model. Alt + Arrow keys rotate the model parallel to the viewing plane.				< >

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Inserting Parts



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The Assembly Tree

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

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Move Component



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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



