Advanced Features and Assemblies

ITP 308

Week 3

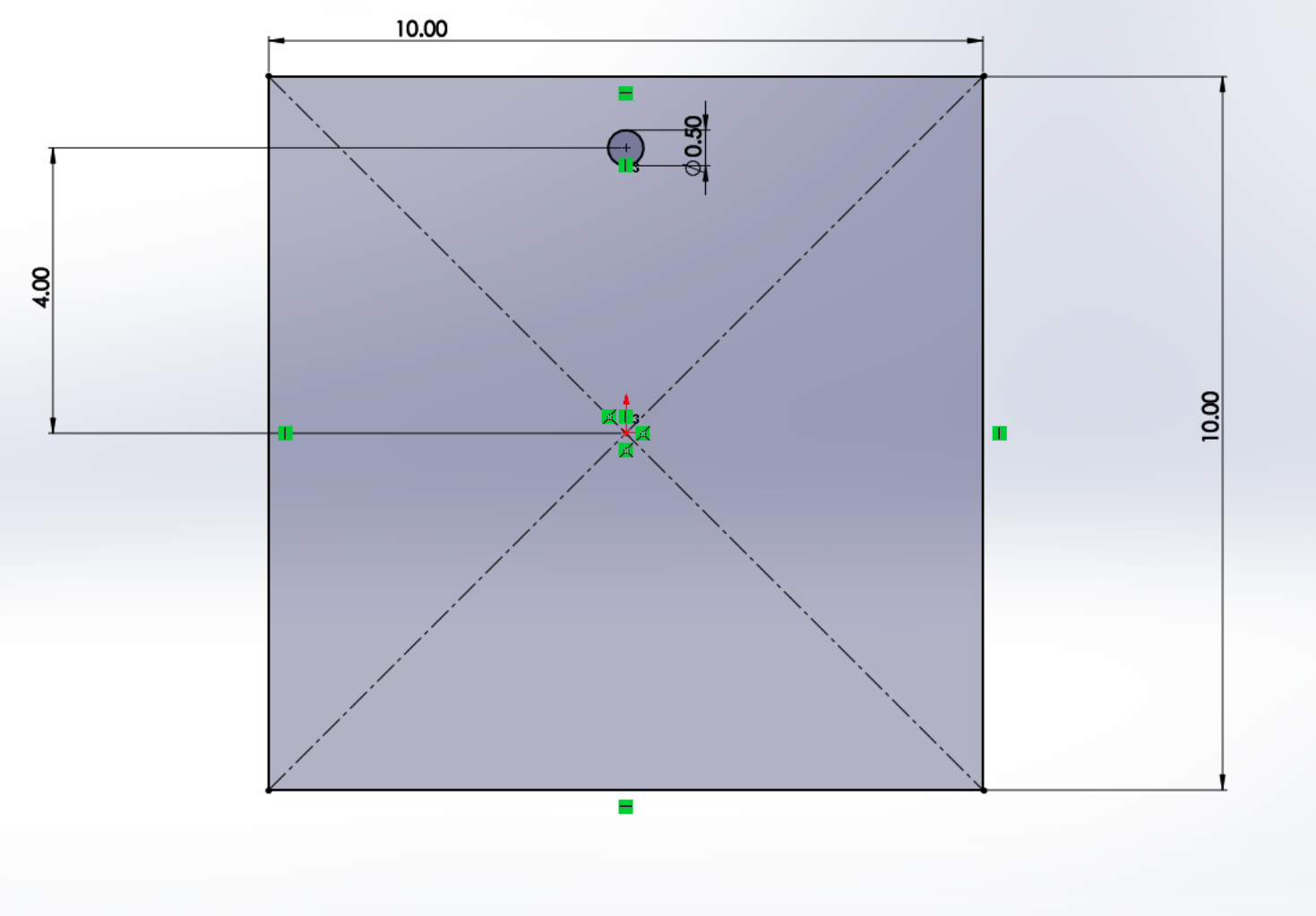
Goals

1. Revisit sketch patterning
2. Create hollow bodies using the shell command
3. Creating assemblies
4. Mating parts and components in assemblies
5. Adding sub-assemblies

Sketch Patterns

- Creating a bunch of holes in a pattern is a chore
- Better to define a singular entity and pattern
  - Circular pattern
  - Linear pattern
- Circular patterns around a central axis
- Linear patterns in 2 directions

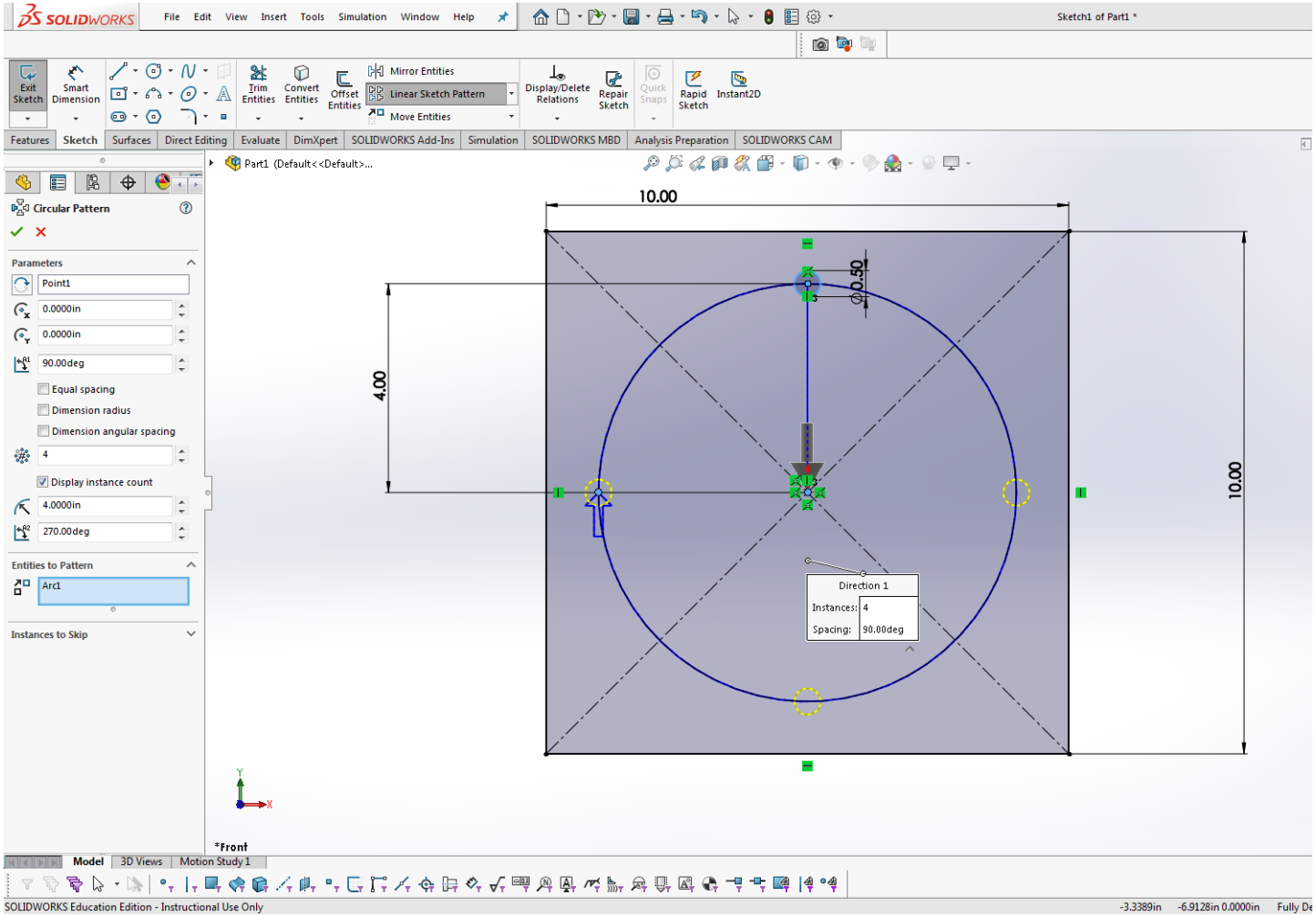
Circular Patterns



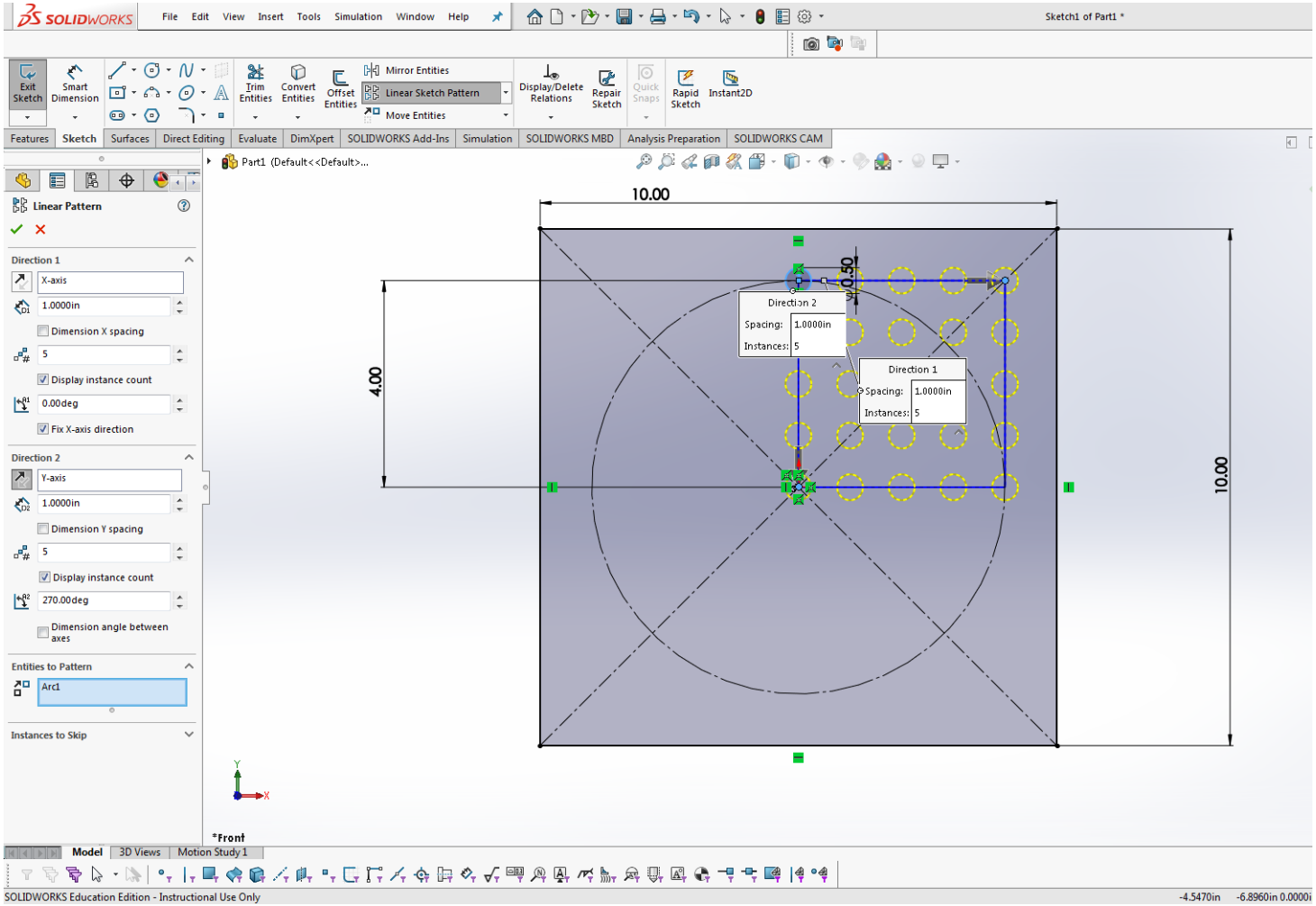
Pattern

- Choose “Circular Pattern”
- Select the vertex to pattern about (the “center” of the circle)
- Can shift the center left and right, or up and down
- Can choose to pattern up to a certain angular dimension
- Can choose to equally space pattern, or specify angular spacing

Pattern Options



Linear Pattern



Linear Pattern Options

- Can choose directions (don't have to be X and Y)
- Can choose spacing
- Can choose to "dimension spacing" to add the smart dimensions for spacing in that direction
  - Not always going to fully-defined the sketch though

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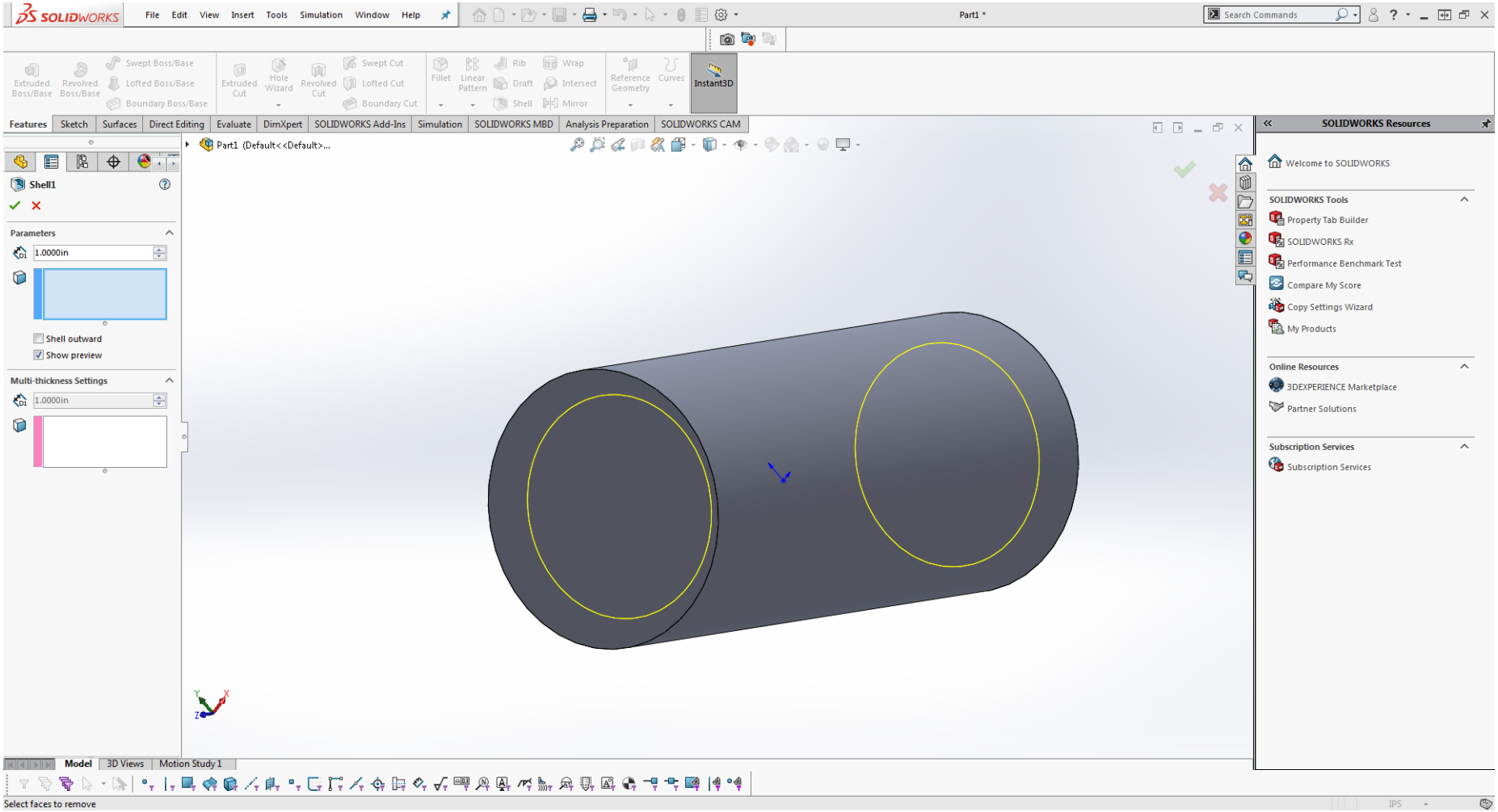
Shell

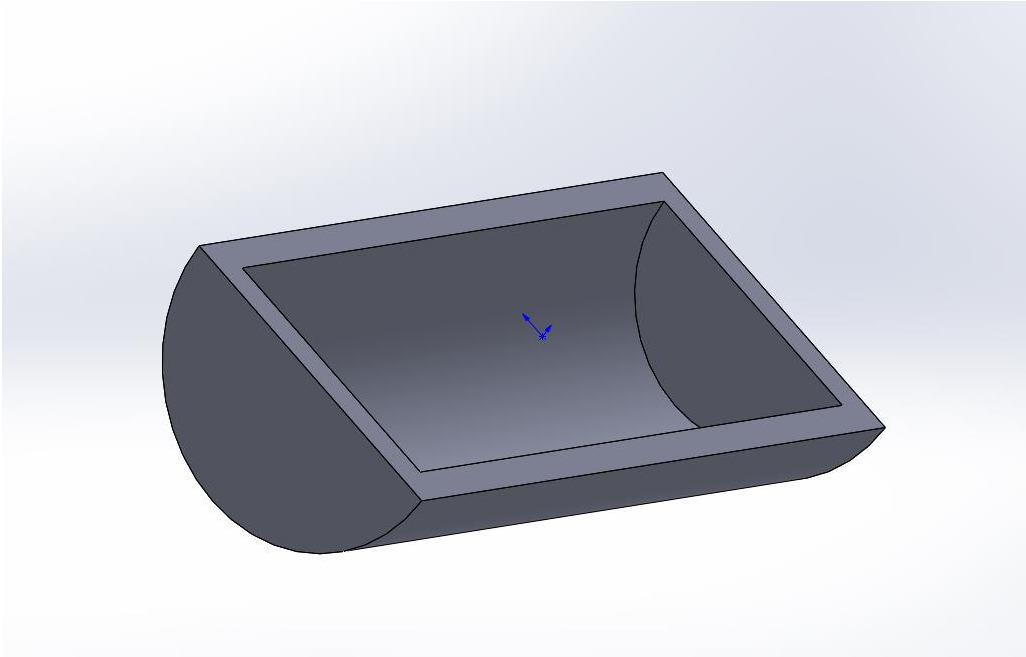
- The shell command creates hollow bodies
- Can create closed-hollow bodies
  - Hollows out the inside without opening a face
- Can create open-hollow bodies
  - Hollows out the inside and removes a face
- Can create different thicknesses for different faces
  - Limited to two different thickness values

Shell Options



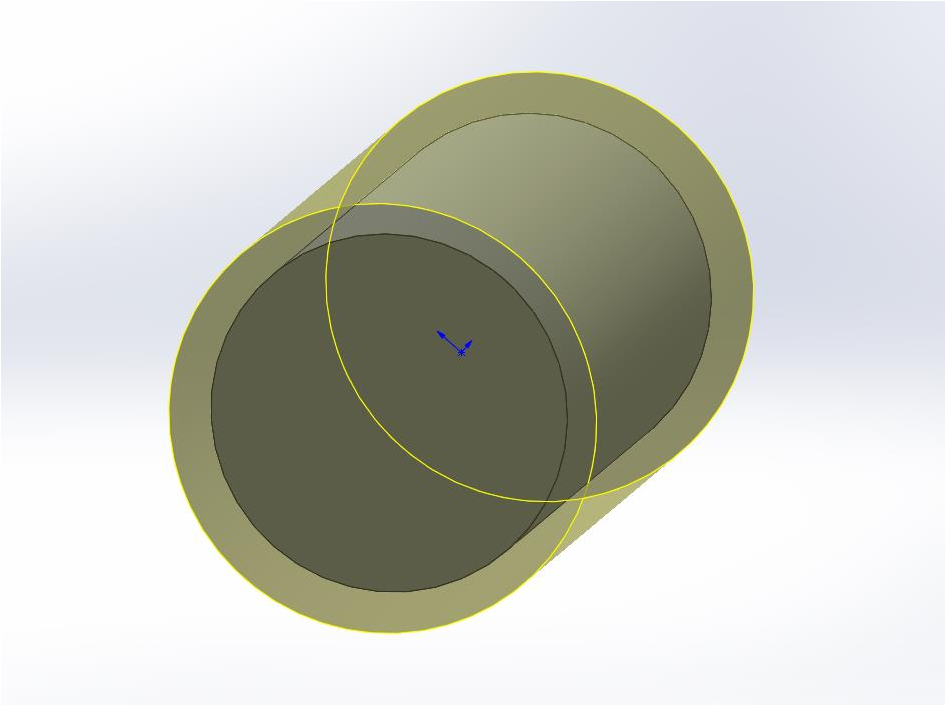
Shell Options

- Shelling will use the existing solid body as a boundary and remove material from within to create a uniformly thick hollow body
  - See the section view. 1in. thickness all around



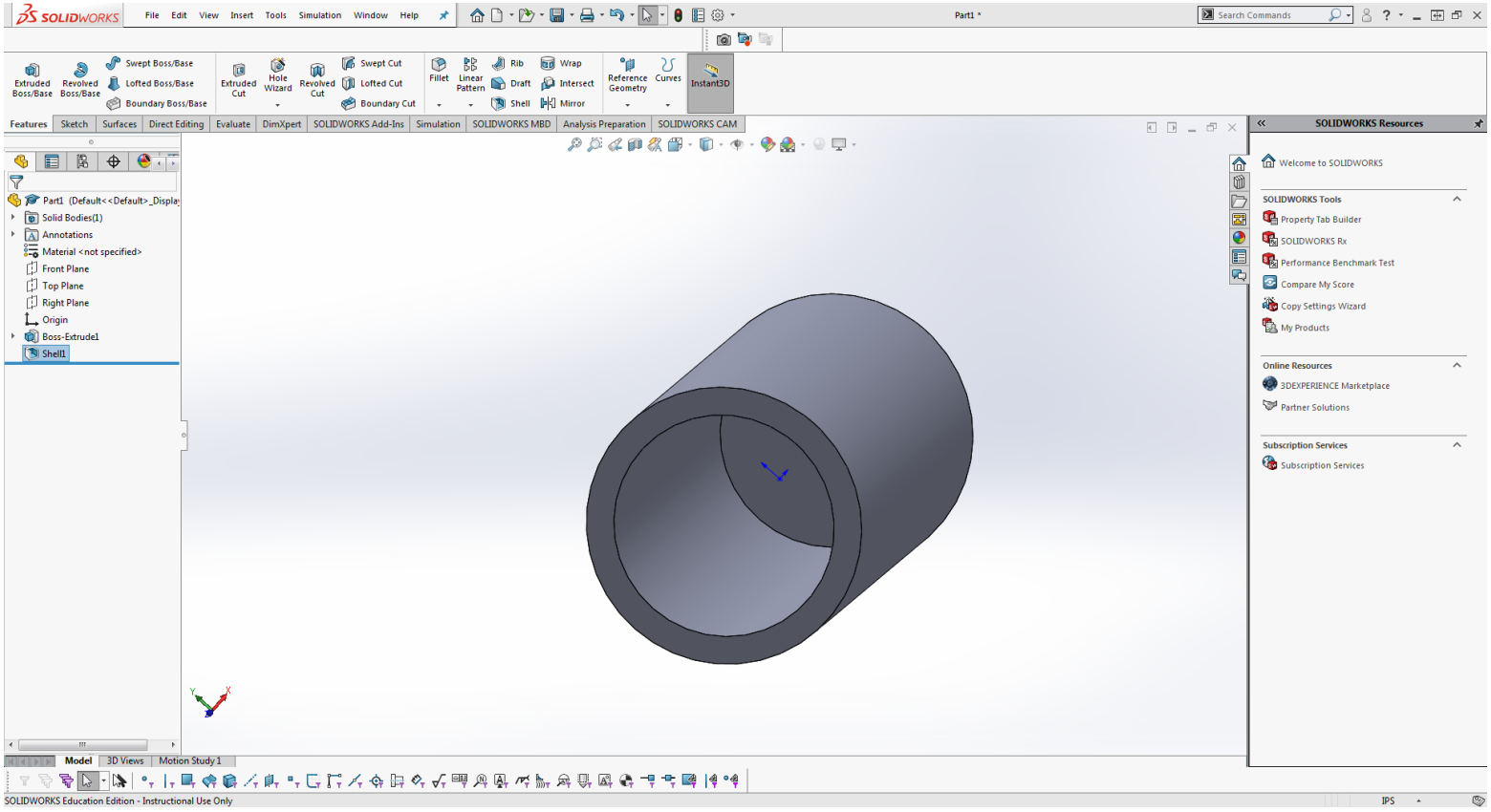
Shell Options

- Shelling outward will use the existing solid body and will create a uniformly thick layer around the outside of the body



Faces to Remove

- We can also create a shell feature that removes planar faces to expose the hollow insides

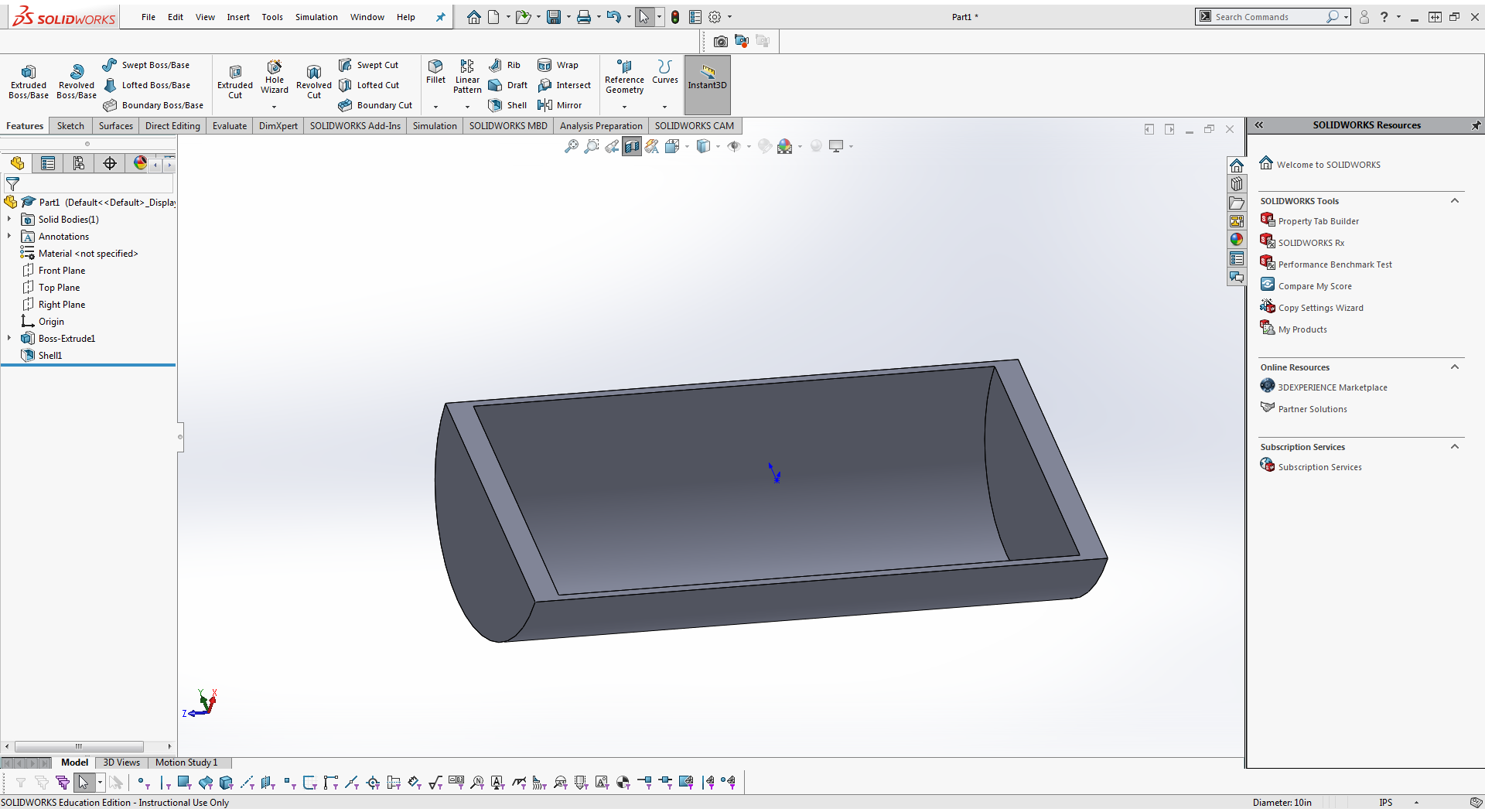


Multi-Thickness Parameters

- We can change the thickness of certain faces using the multi-thickness parameter setting

- We can only have two different thickness values

Multi-Thickness



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Assemblies

- Now that we can model parts, putting them together to form fully functional, articulating assemblies is the next logical step

- Assemblies have:
  - Their own Feature Design Tree
  - Their own origins
  - Their own cardinal planes (front, top, right)

- You can have:
  - Parts from external files
  - Assemblies from external files
  - Parts internal to the assembly file (in-context parts)
  - Assemblies internal to the assembly file (in-context assemblies)

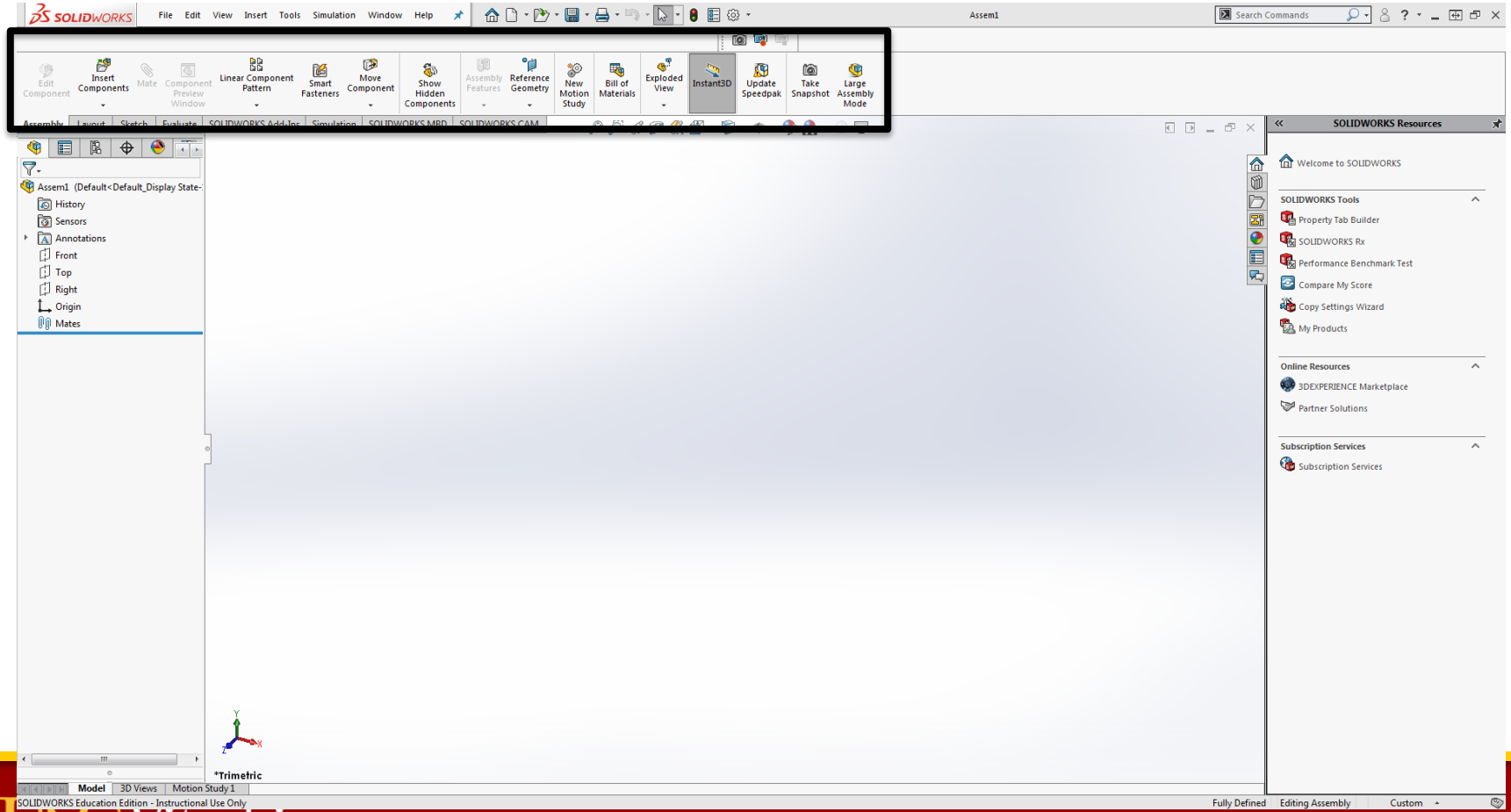
Assemblies

- IMPORTANT!

1. When working with assemblies, place ALL files in the same folder
2. When submitting homework and lab assignments with assemblies, submit all the part files along with it
    1. An assembly is just instructions on how to put it together, you still need the parts
3. Do not change the name of parts once you've started the assembly
    1. This causes a nightmarish hell-scape of link errors when I try to open your files and I don't get enough sleep as it is.

Creating Assemblies

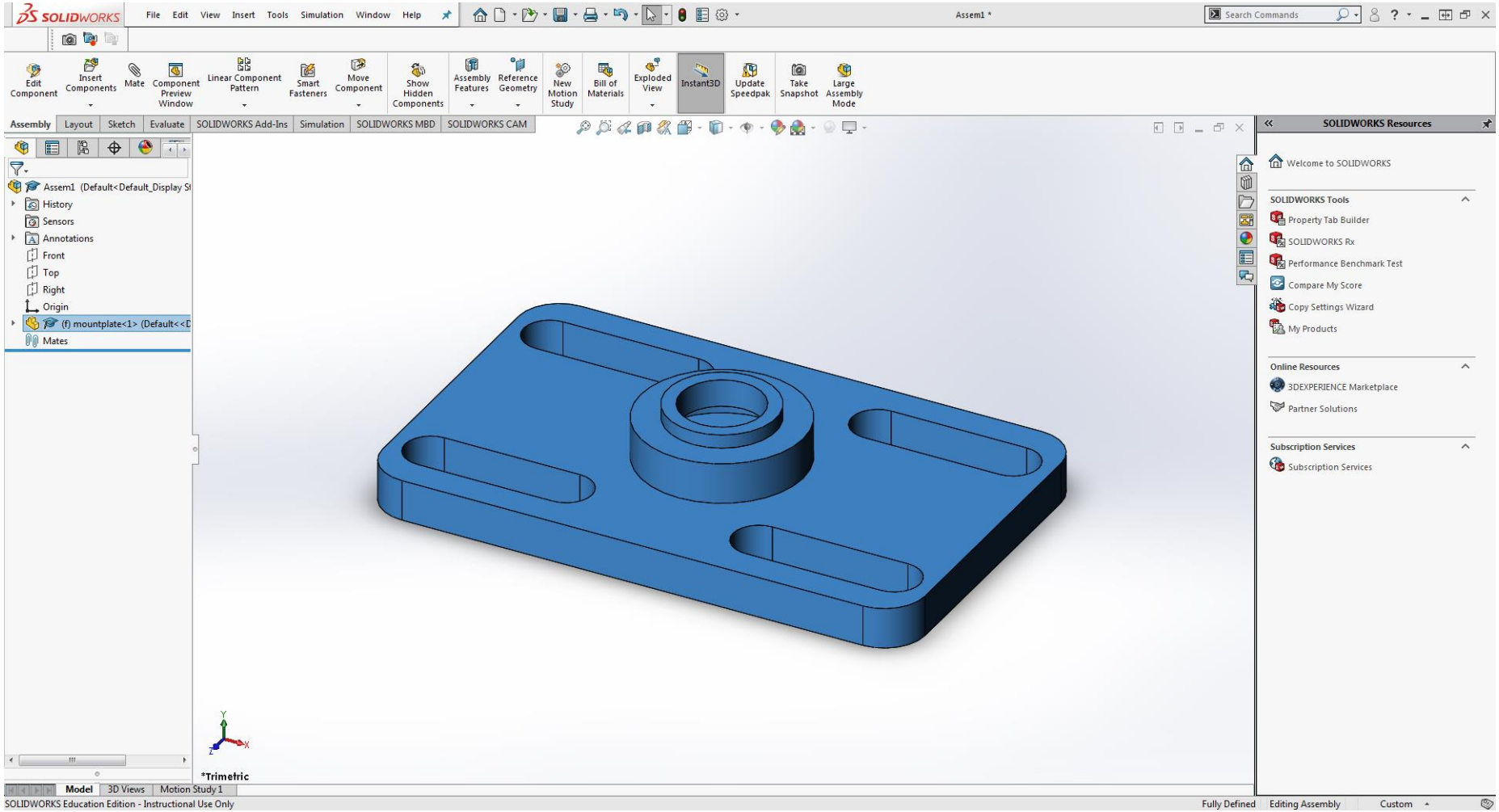
- The assembly commands are slightly different from the part's commands



Adding items

- Using the “Insert Components” tool, we can import external parts and assemblies to our assembly stage
- Usually the first part is placed at the origin of the assembly file
- The first part is also “fixed” in place by default
  - Don’t want this behavior
  - Would rather move and position it ourselves
  - First break the “fixed” then use mates

Adding the Mountplate



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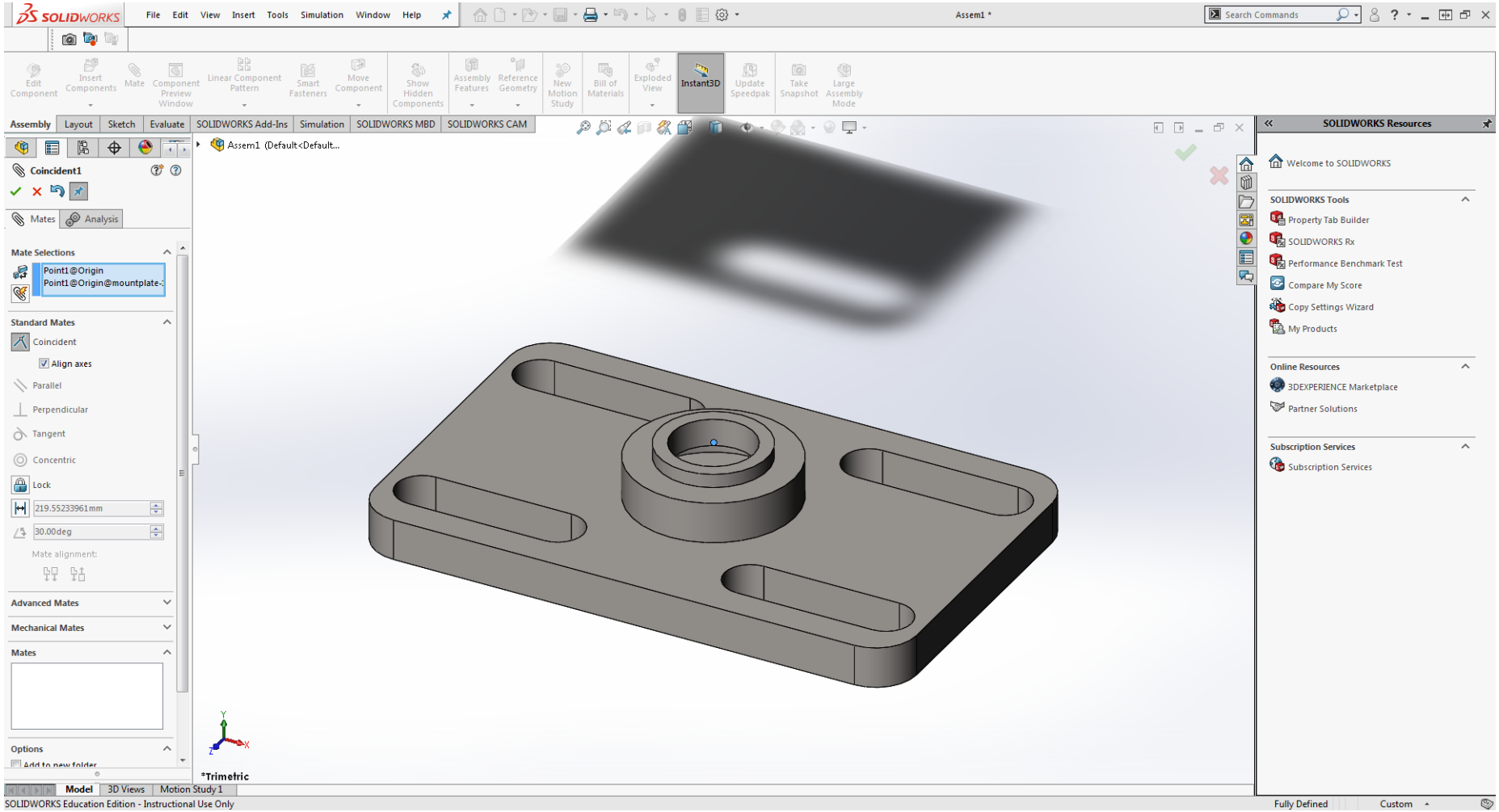
Mating to the origin

- We will “Mate” the origin of the mountplate to the origin of the assembly

- Mating in assemblies is the same as relationships in sketches

- Use mates to define the relationship between parts and sub-assemblies so they:
  - Don’t move
  - Move in very specific fashion

Mates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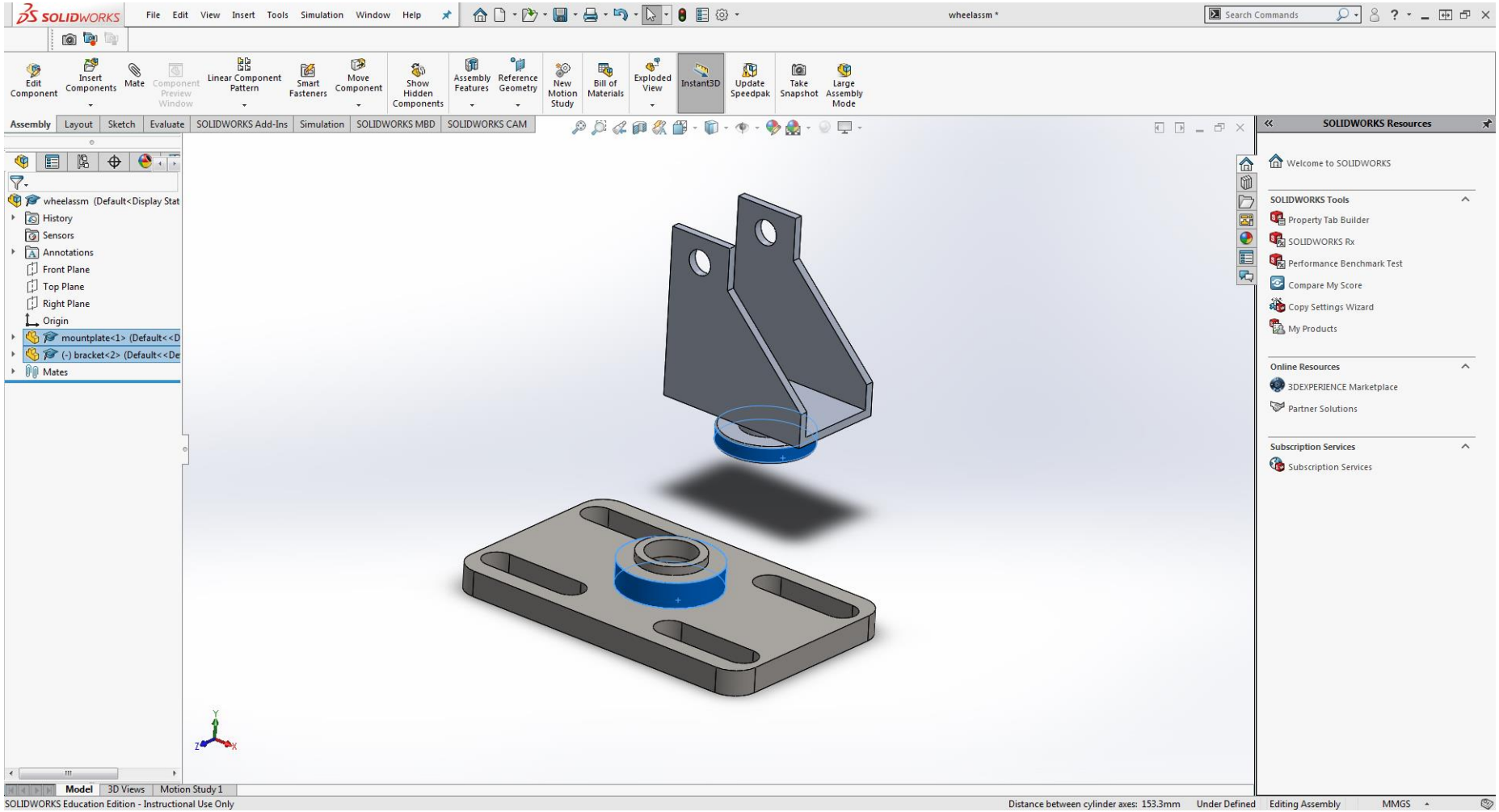
Fully Defined
Custom

Adding the Bracket

- Next we will add the bracket
- When building an assembly, it is ALWAYS best to build one piece at a time.
- Having too many pieces at once is ALWAYS a recipe for disaster
- Be sure to fully define each part within the assembly space, and then only mate to fully defined parts

Concentric


SOLIDWORKS
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Linear Component Pattern
Smart Fasteners
Move Component
Show Hidden Components
Assembly Features
Reference Geometry
New Motion Study
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Welcome to SOLIDWORKS
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SOLIDWORKS Rx
Performance Benchmark Test
Compare My Score
Copy Settings Wizard
My Products
Online Resources
3DEXPERIENCE Marketplace
Partner Solutions
Subscription Services
Subscription Services
Model 3D Views Motion Study 1
SOLIDWORKS Education Edition - Instructional Use Only
Distance between cylinder axes: 153.3mm
Under Defined
Editing Assembly
MMGS

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Sub-Assemblies

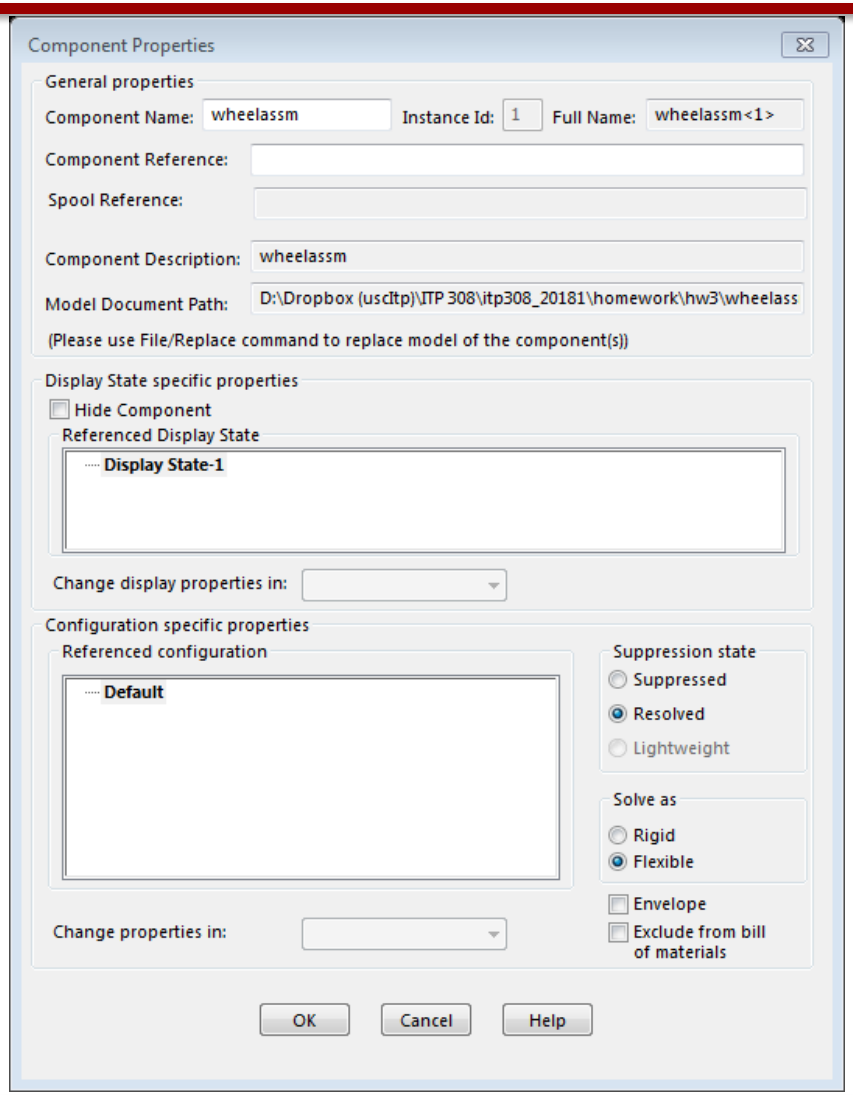
- Adding an assembly to an assembly is a common practice
- Often times easier to work on small assemblies and bring them into the larger assembly context
- If your sub-assembly needs to move (within the larger assembly context) you need to modify the properties

Solving as Flexible

- In the properties of the sub-assembly
  - Right click on the assembly and choose the component properties

- Select “Flexible” under the “Solve As” options

Flexible


Component Properties
General properties
Component Name: wheelassm
Instance Id: 1
Full Name: wheelassm<1>
Component Reference:
Spool Reference:
Component Description: wheelassm
Model Document Path: D:\Dropbox (uscltp)\ITP 308\itp308_20181\homework\hw3\wheelass
(Please use File/Replace command to replace model of the component(s))
Display State specific properties
Hide Component
Referenced Display State
Display State-1
Change display properties in:
Configuration specific properties
Referenced configuration
Default
Suppression state
Suppressed
Resolved
Lightweight
Solve as
Rigid
Flexible
Envelope
Exclude from bill of materials
Change properties in:
OK
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Help