



TEXAS A&M UNIVERSITY
ROBOTICS TEAM & LEADERSHIP EXPERIENCE

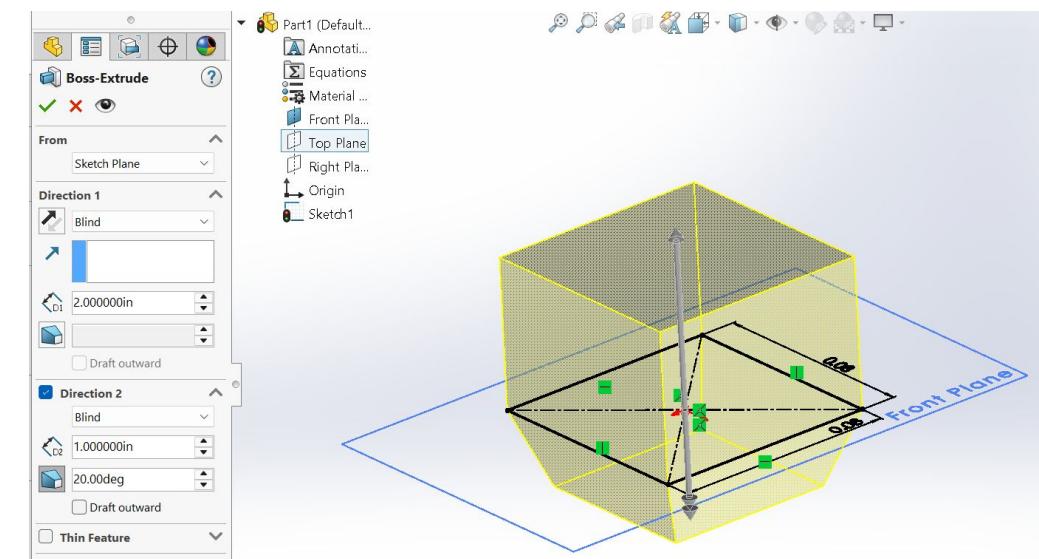
SolidWorks 3D

TURTLE Hatchling

The Basic CAD Journey



1. Create>Select a Plane (A face is a plane)
2. Create a Sketch
 - a. Draw general shape w.r.t the origin
 - b. Add relations
 - c. Dimension the rest from largest to smallest
3. Turn Sketch into a 3D Feature
4. Save
5. Repeat



Resilient Modeling Strategy



1. Add Material

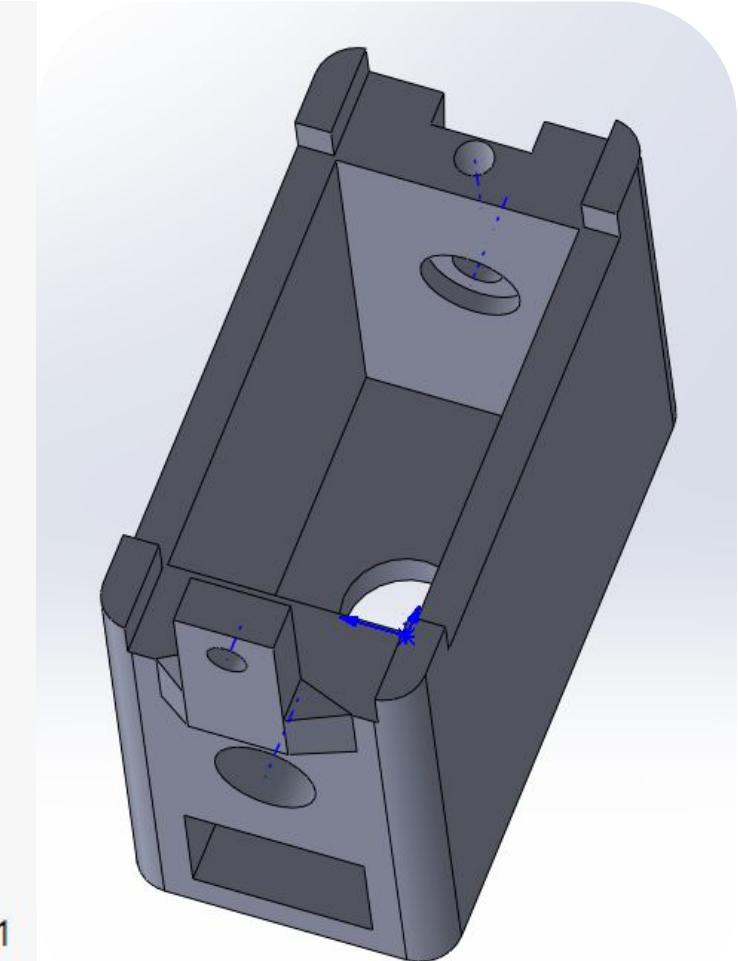
2. Subtract Material

3. Special Operations
 - a. Shell, Patterns, Mirror

4. Finishing Operations
 - b. Chamfer, fillets

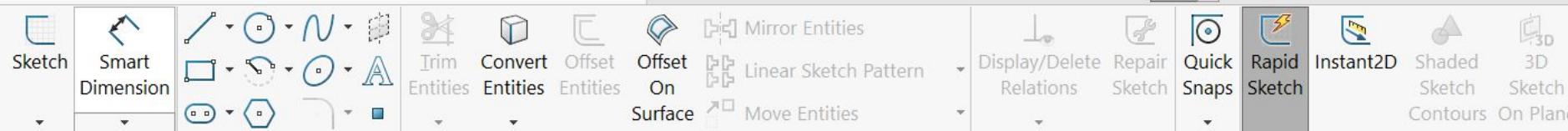
5. Repeat Loops
 - c. if necessary

- ▶ Floor
- ▶ Walls
- ▶ Stands
- ▶ Wire Hole
- ▶ Servo Mount Holes{ ->}
- ▶ Axe Heat Insert
- ▶ Chasis Servo Horn
- ▶ Chasis Servo Horn 2
- ▶ Chasis Screw CS
- ▶ Chasis Screw Hole
- ▶ Bottom Hole
- ▶ Sideways Connection{ ->}
- ▶ Sideways Connect Screw{ ->}
- ▶ Print Easy
- ▶ Sideways Connection Strength 1
- ▶ Sideways Connection Strength 2
- ▶ Corners



Renaming features was a personal preference

Sketch Tab



Drawing Tools

Special Drawing Tools

Patterns

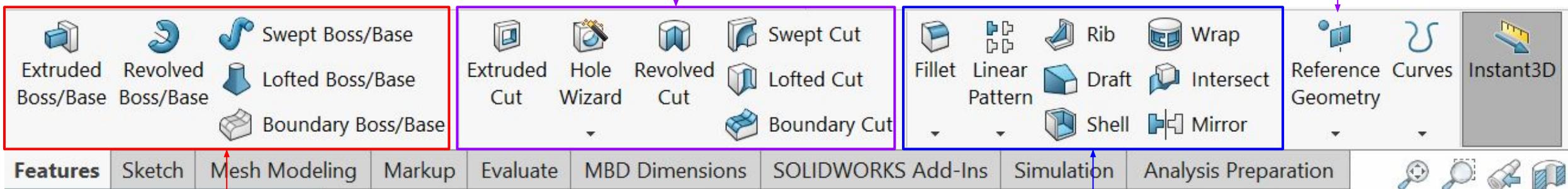
- Use “Drawing Tools” to make a general shape.
- Trim Tool is like an eraser
- Convert Entities projects selected geometry onto the plane
- Offset Entities does as it says
- Try not to use sketch patterns or fillets/chamfers

Features Tab



Cutting Tools for
removing volume

Extremely
Powerful



Boss/Base tools for
adding volume

**Notice:
Cutting and
Boss/Base
tools are
inverses**

Speciality Tools

Extruded - Expanding 2D Plane to 3D

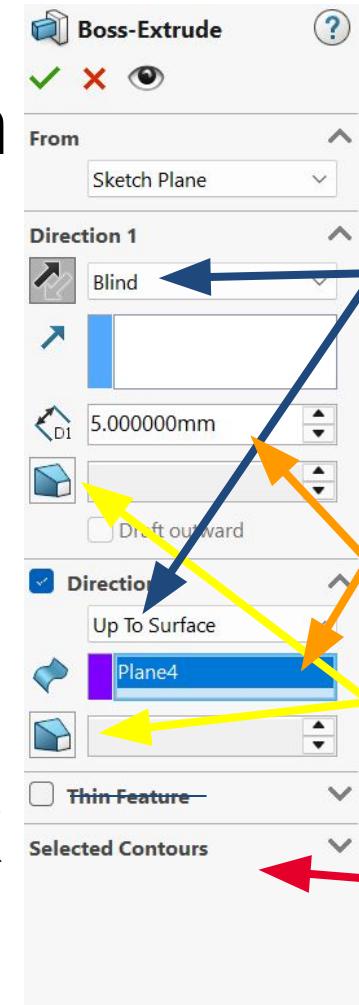


Requires:

- 2D Enclosed Region

End Conditions:

- Blind -> X distance
- Midplane -> X/2 distance in both directions
- Up to ... -> Goes to selection
- Offset From Surface -> X distance from selection



Starting Point

End Condition Type

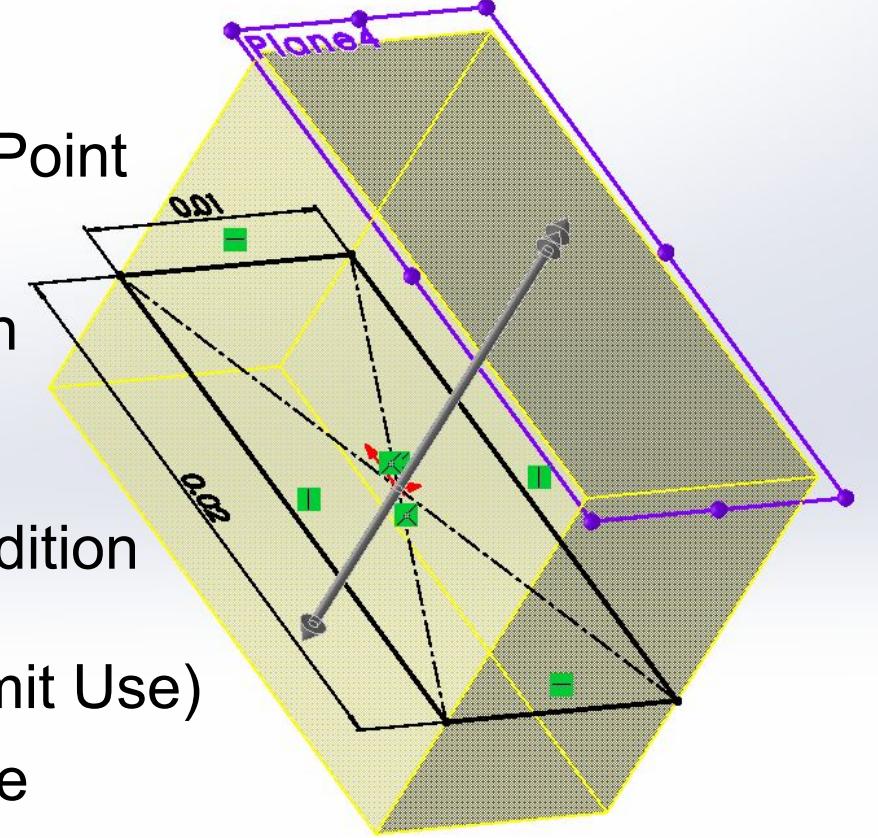
End Condition

Draft (Limit Use)

Don't Use

Select enclosed regions

Yellow is the preview



Revolved - Rotates around an Axis

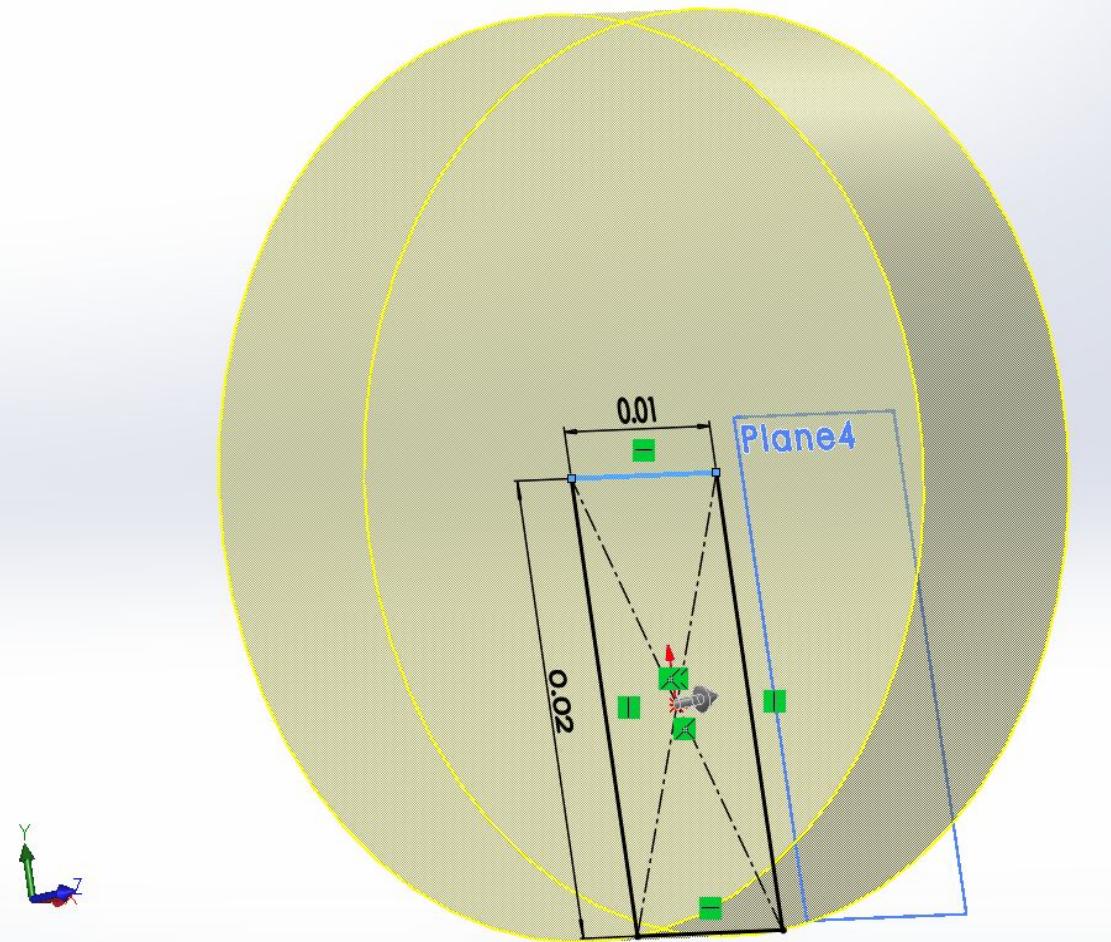
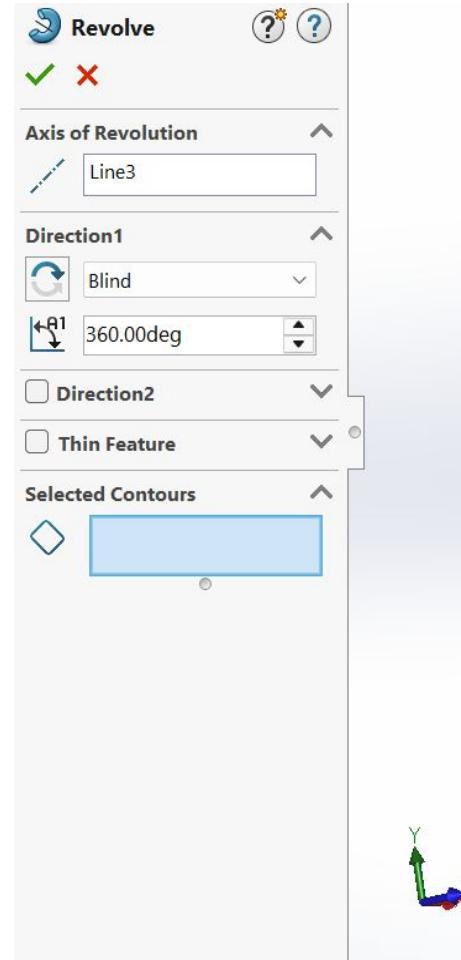


Requires:

- 2D Enclosed Region
- Axis of Revolution

Note:

- Axis of Revolution can be any edge/line
- Axis of Revolution does not have to be connected to the enclosed region



Fillets/Chamfers



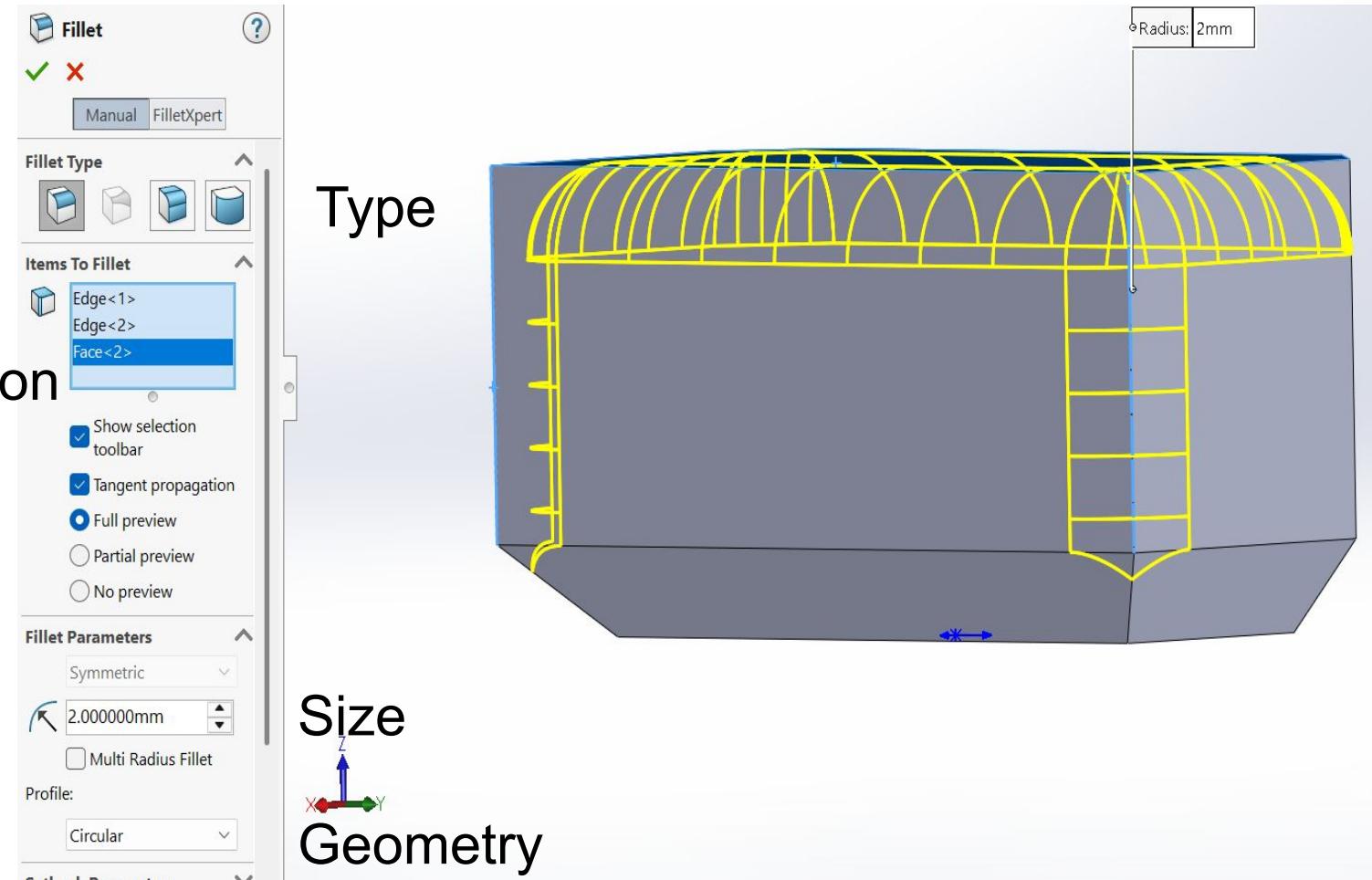
Fillet -> Curved

Chamfer -> Flat

Notes:

- Face selects all connected edges
- You can ignore the other options for now

Edge Selection



Importance



Understanding Extrude and Revolve Tools is fundamental to CAD

Proper sketch plane selection will enable you to CAD most parts with Extrude, Revolve, and Finishing Operations

How do you chose the origin location and planes?

Easiest Way is Symmetry

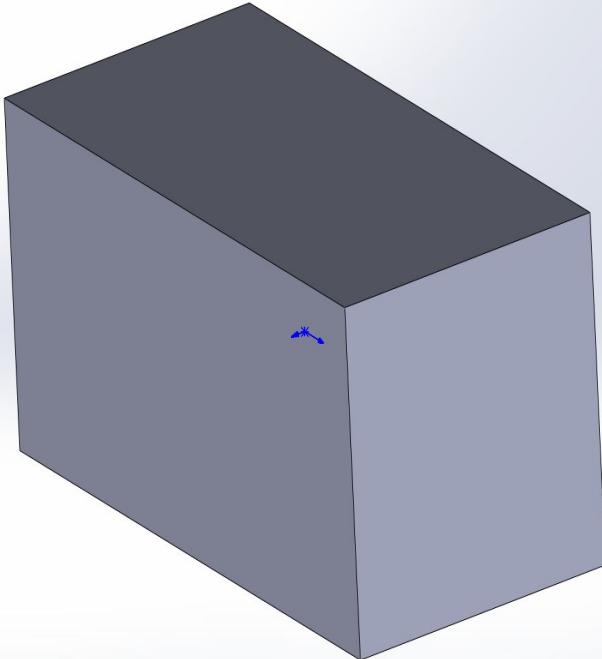


1. Identify Planes/Axis of Symmetry
 - a. Primary -> Entire Body
 - b. Secondary -> Features that can be mirrored or pattern
2. Pick the point that has the most intersections of step 1.
 - b. Prioritize the primary symmetric geometry
3. If step 2 has multiple points, chose one with a prominent feature

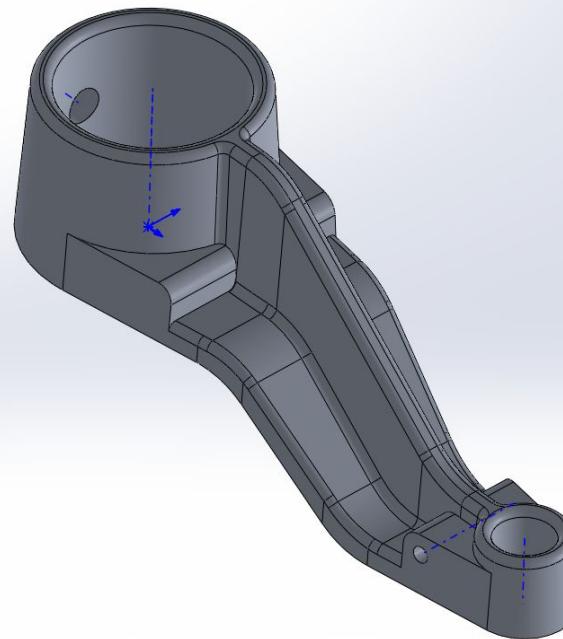
Planes of Symmetry - Practice



Rectangular Prism



Modified 2009
Model Mania



Nametag

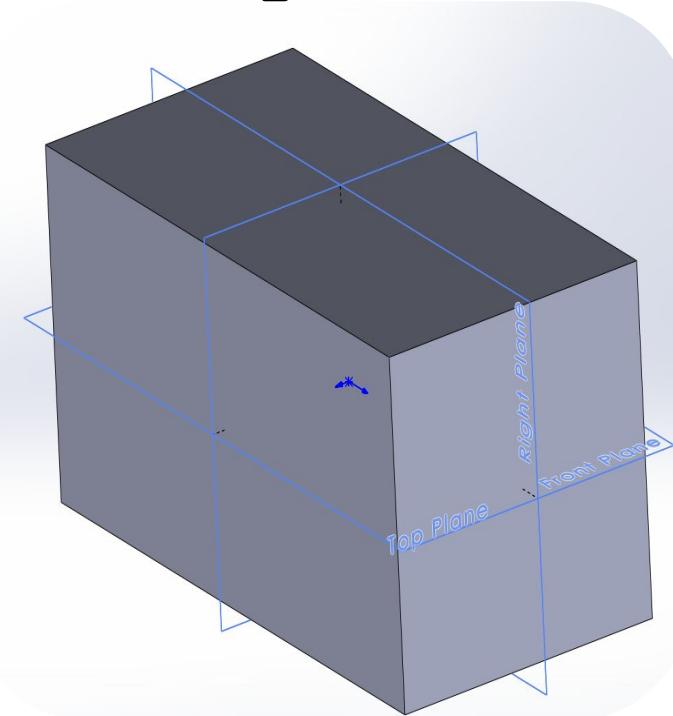


Planes of Symmetry



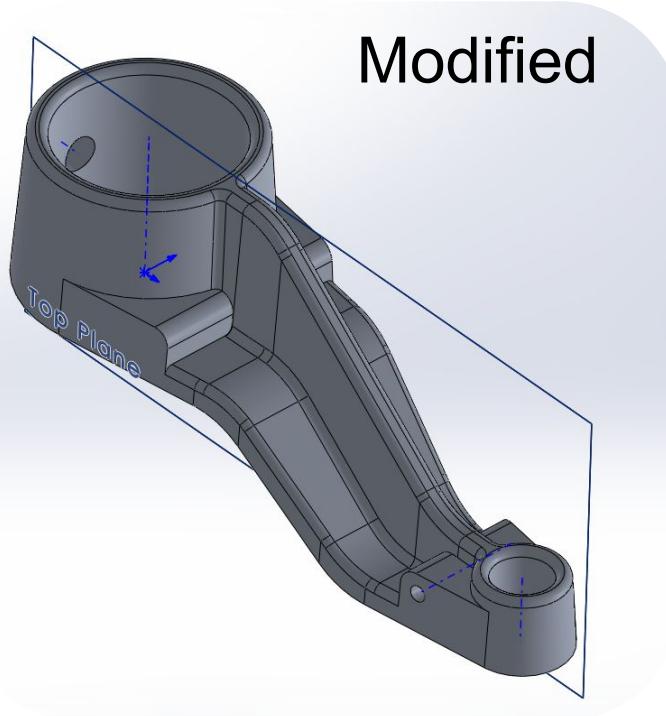
*** Often indicates extrude or mirror/linear pattern***

Rectangular Prism



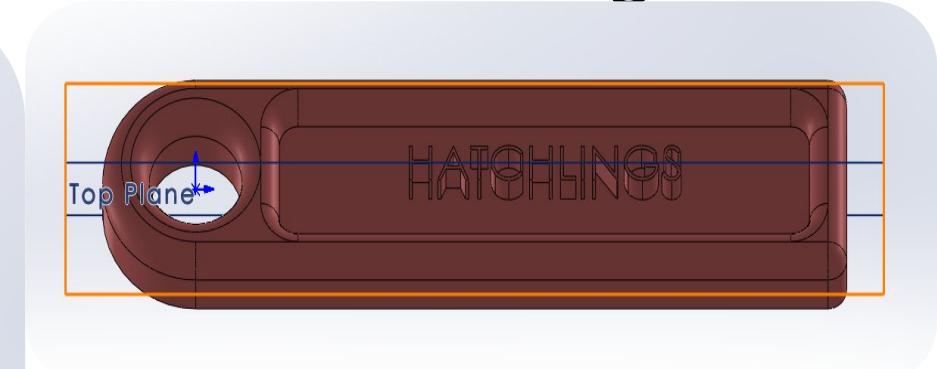
3 P-POS

2009 Model Mania



1 P-POS

Nametag

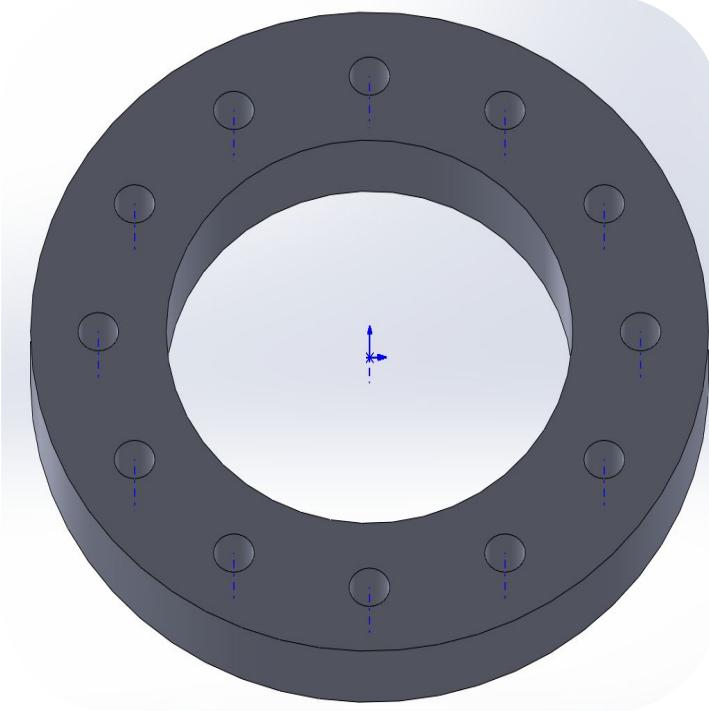


1 P-POS
1 S-POS
(before
indent/name/fillet)

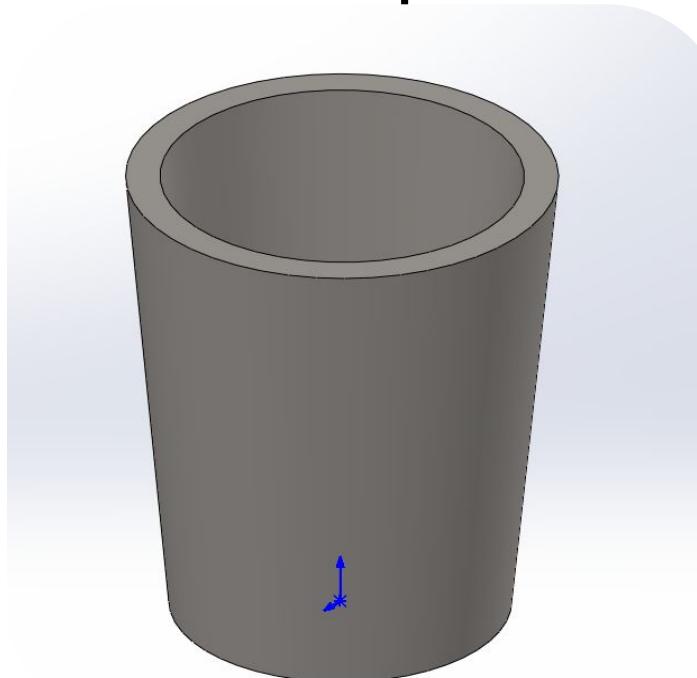
Axis of Symmetry - Practice



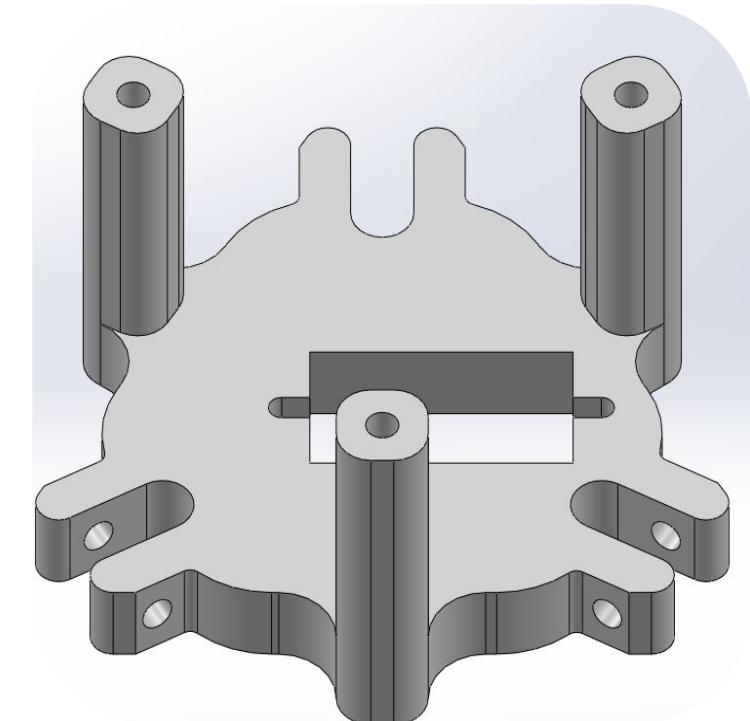
Gasket



Cup



Base of a claw

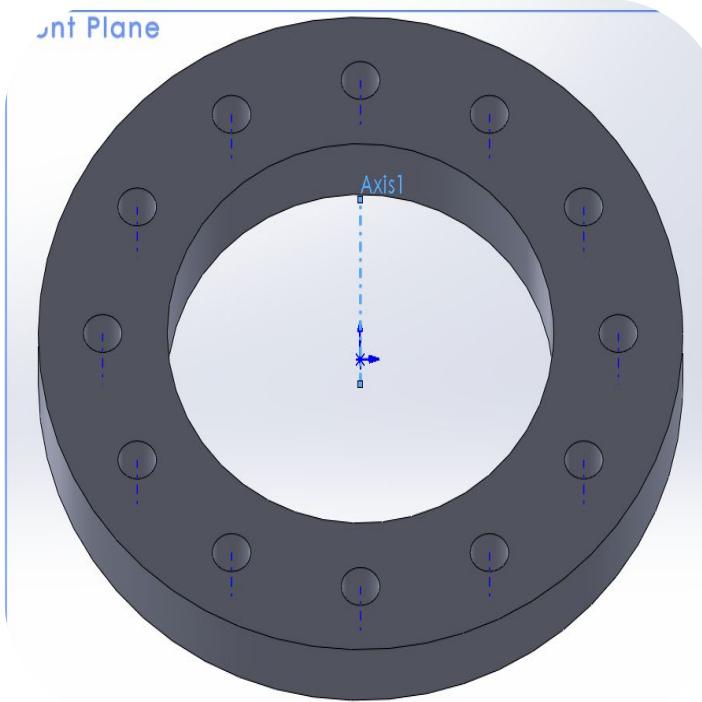




Axis of Symmetry

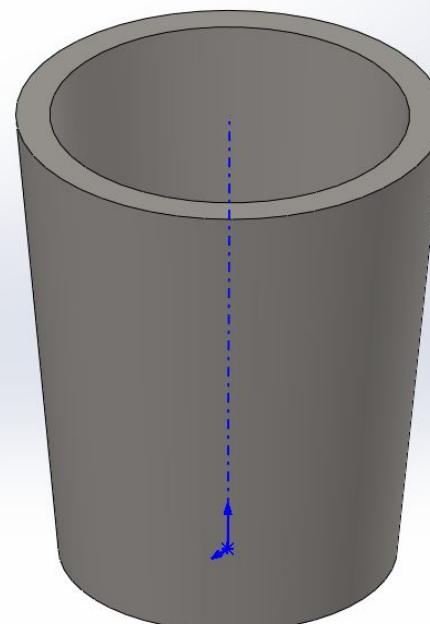
*** Often indicates revolve or circular pattern***

Gasket



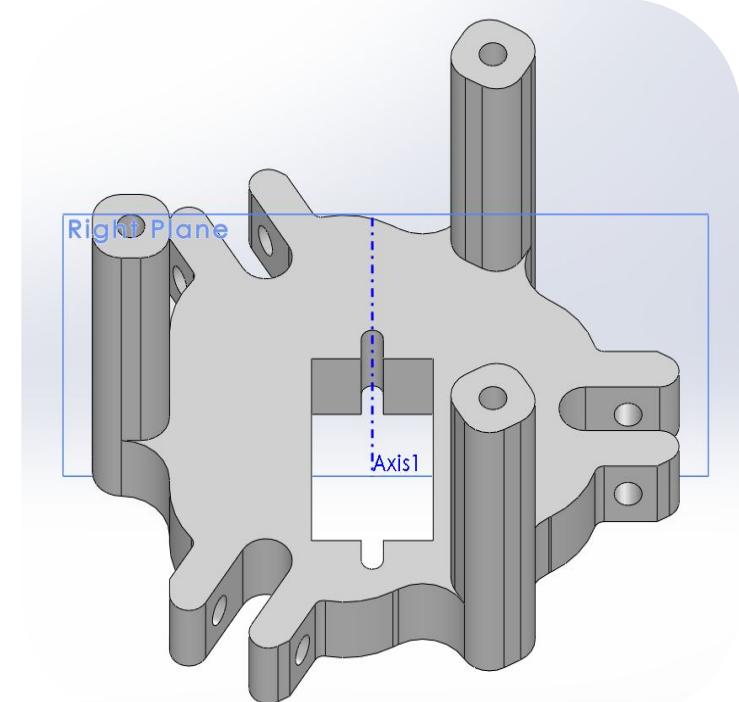
1 P-AOS
1 S-AOS
1 P-POS

Cup



1 P-AOS

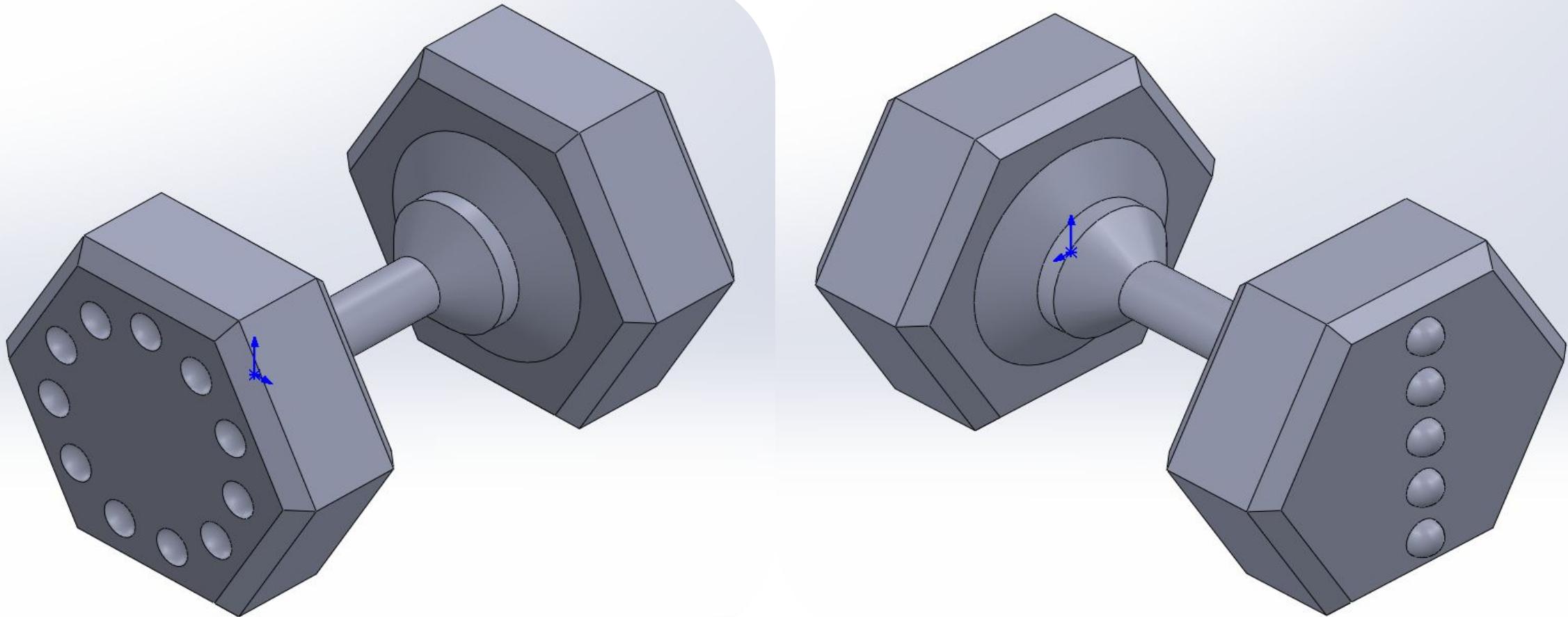
Base of a claw



1 S-AOS
1 P-POS

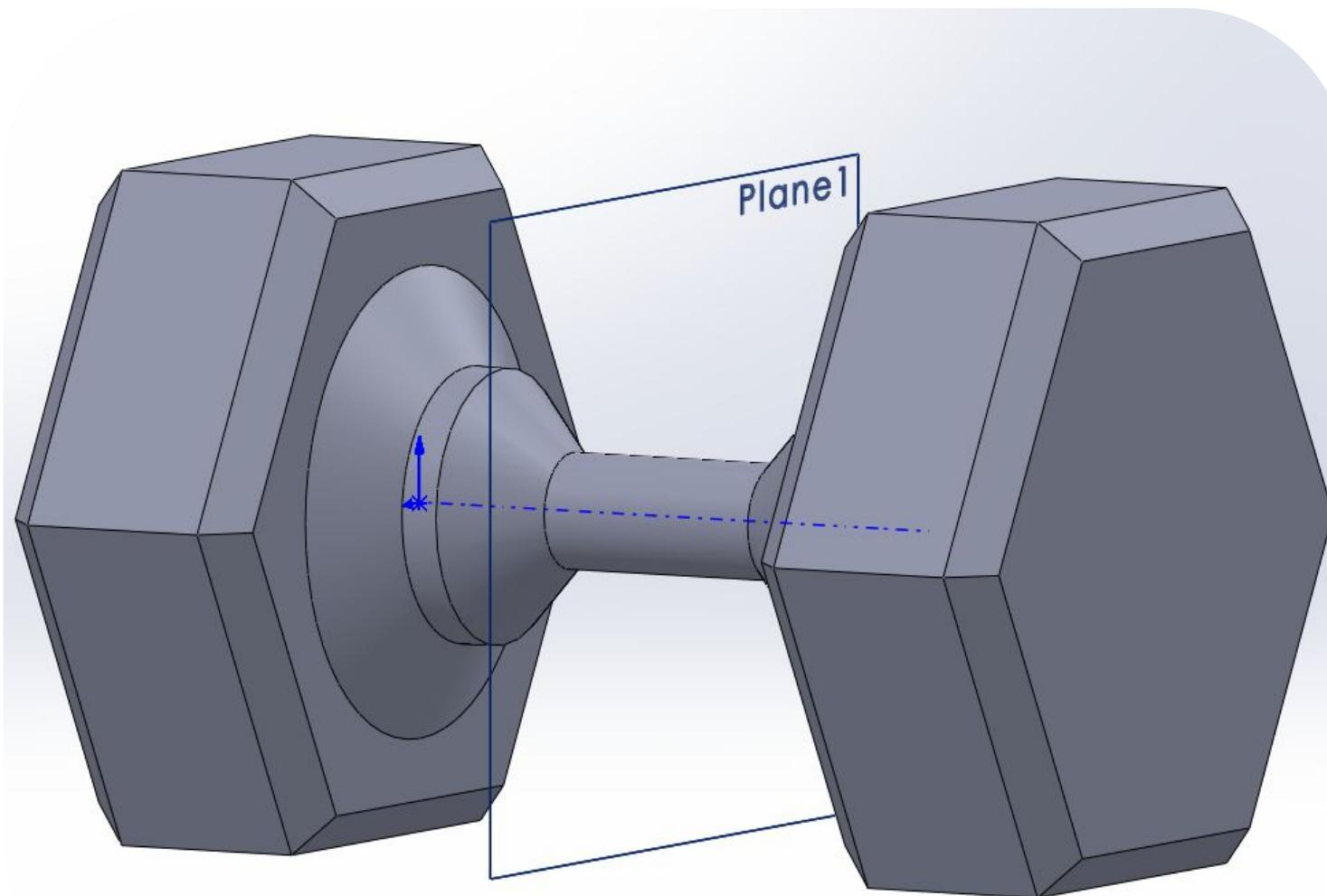
(before servo cutout)

Let's make a Dumbbell





Ignoring the end patterns...

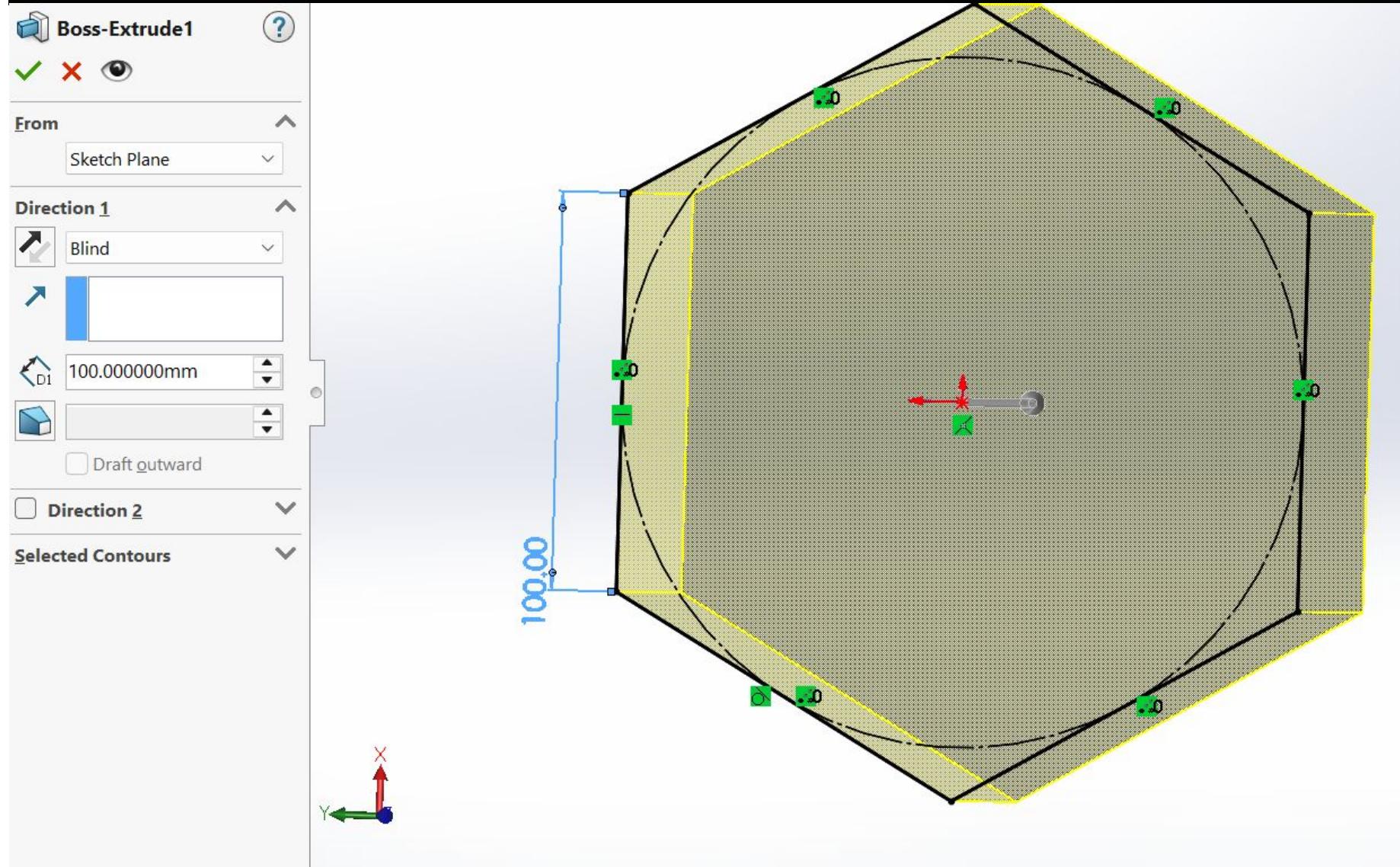


1 P-POS -> Mirror
1 S-AOS (Handle) ->
Revolve

Flat Edges ->
Chamfer

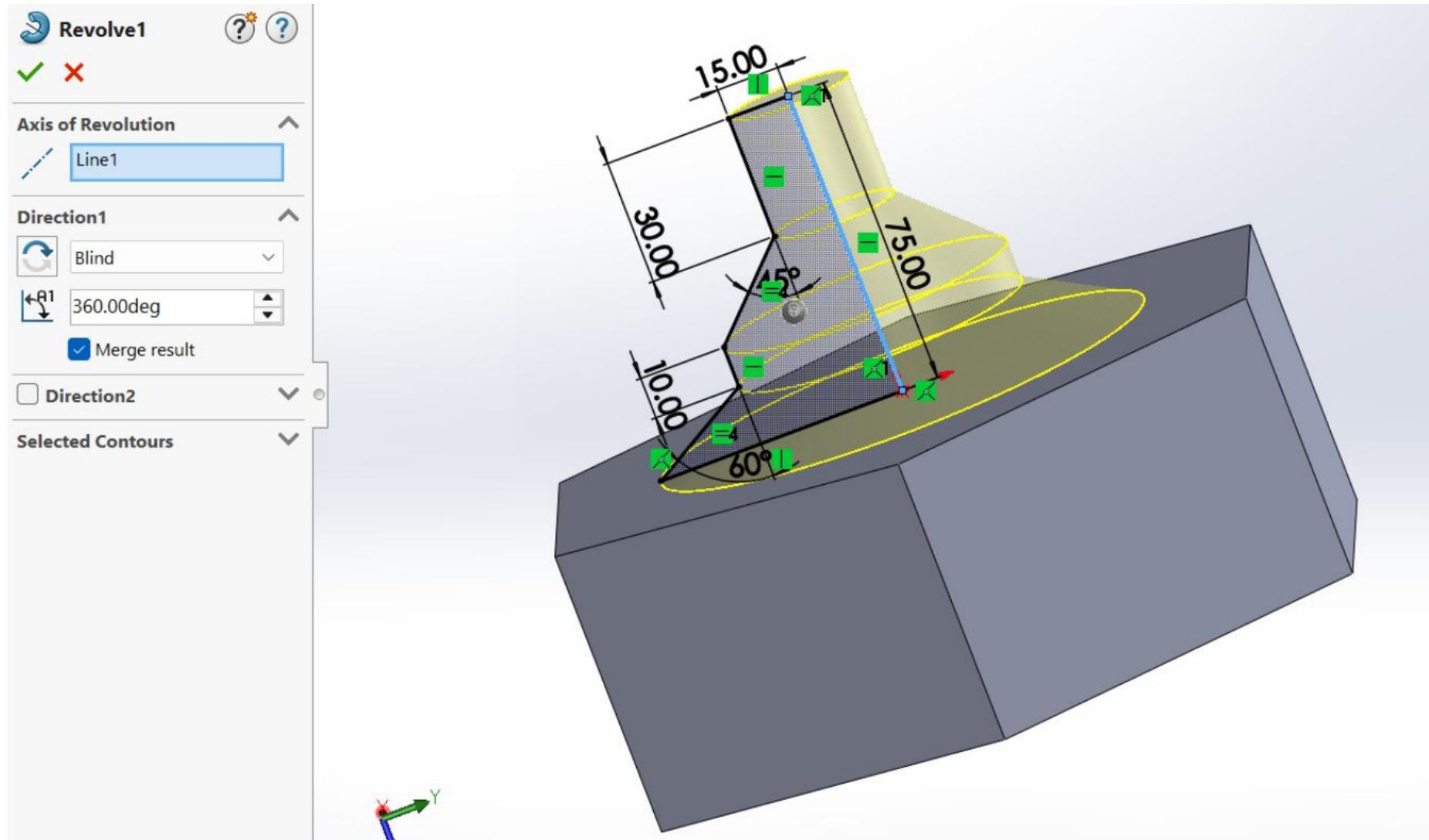
End piece -> The
view from “Plane 1” is
an extruded hexagon

Hexagon End Cap



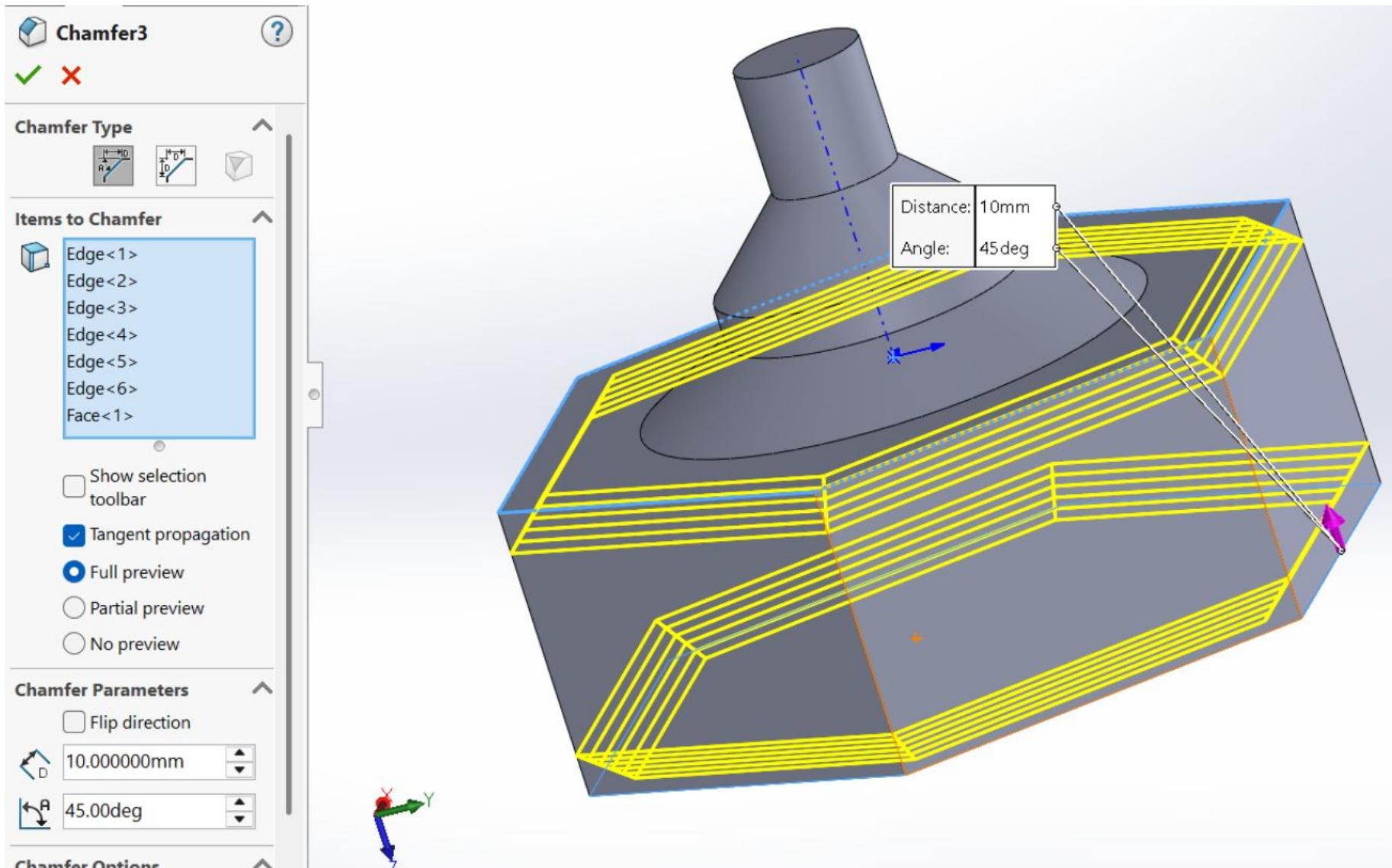
- 100 mm side hexagon
- 100 mm boss extrude

Handle



- Angled lines are equal in length
- 10 , 30, 15, 75 mm line lengths

Chamfer



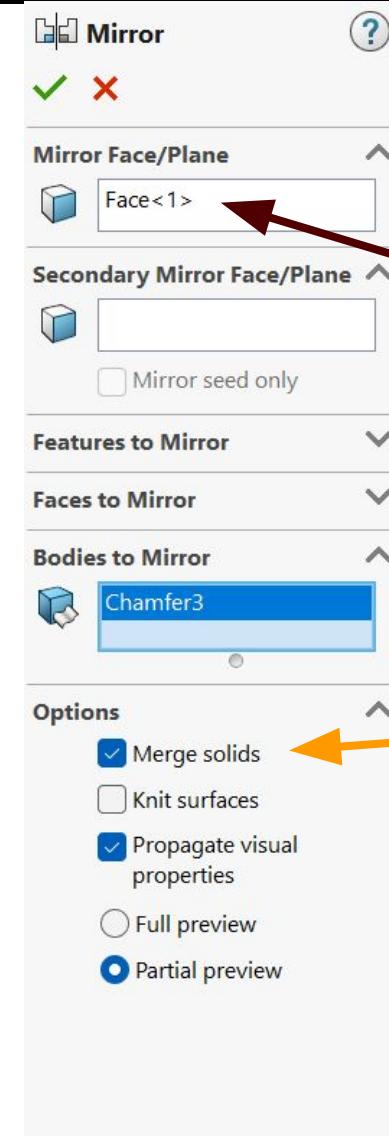
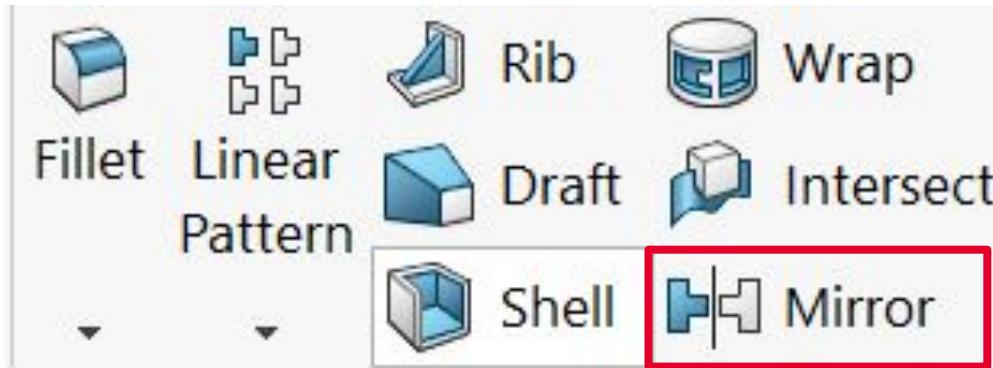
- Top six outside edges are selected
- Bottom face selected

Mirror

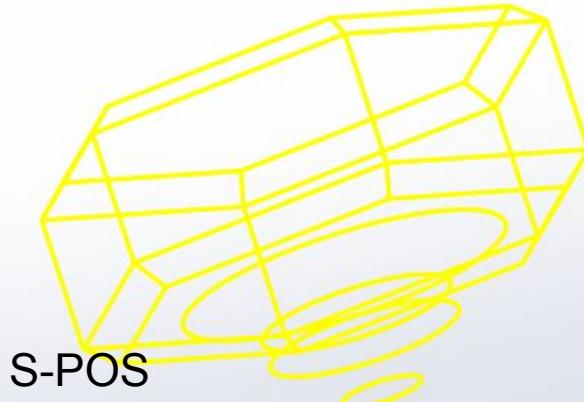


This is a very powerful tool

Let's you CAD half of the model if you have a plane of symmetry



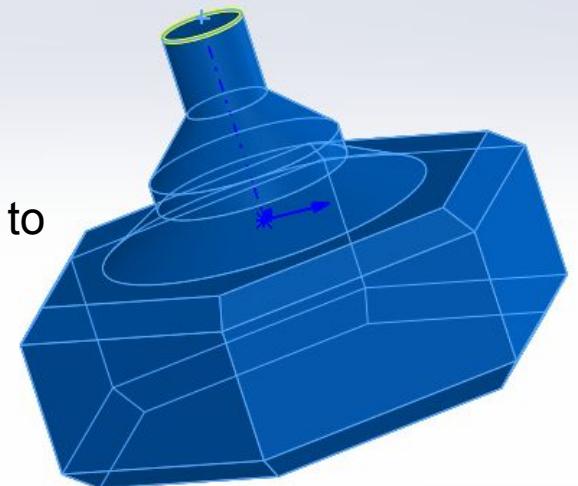
POS



Good for S-POS

Good for P-POS

Uncheck if you want to shell only one side



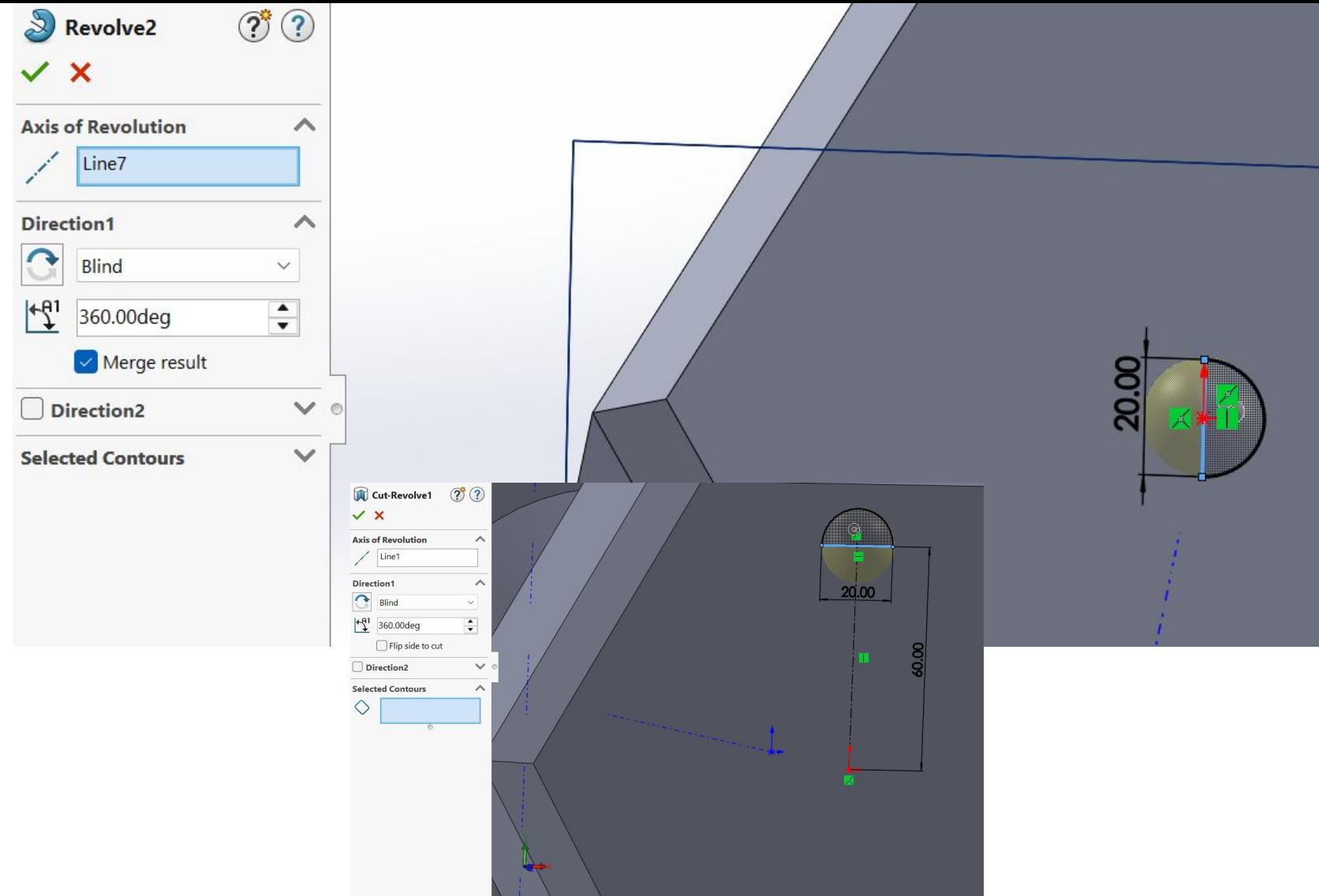
Spheres



Technically infinite
AOS. Just pick 1

Note:

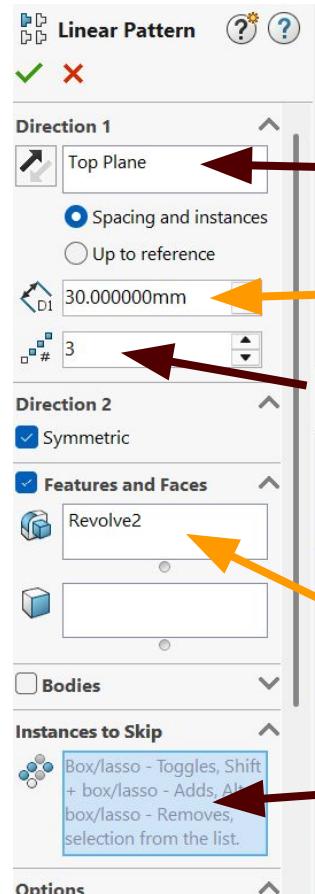
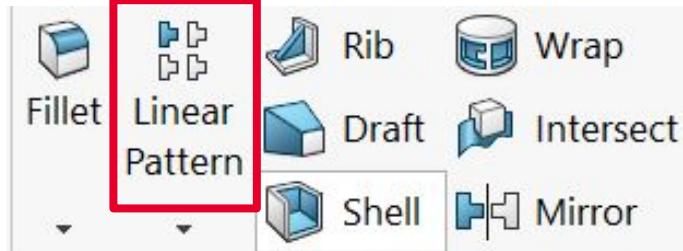
- SolidWorks doesn't like a full circle for making spheres
- Make a half circle and revolve about the straight line



Linear Pattern



Please use
when possible

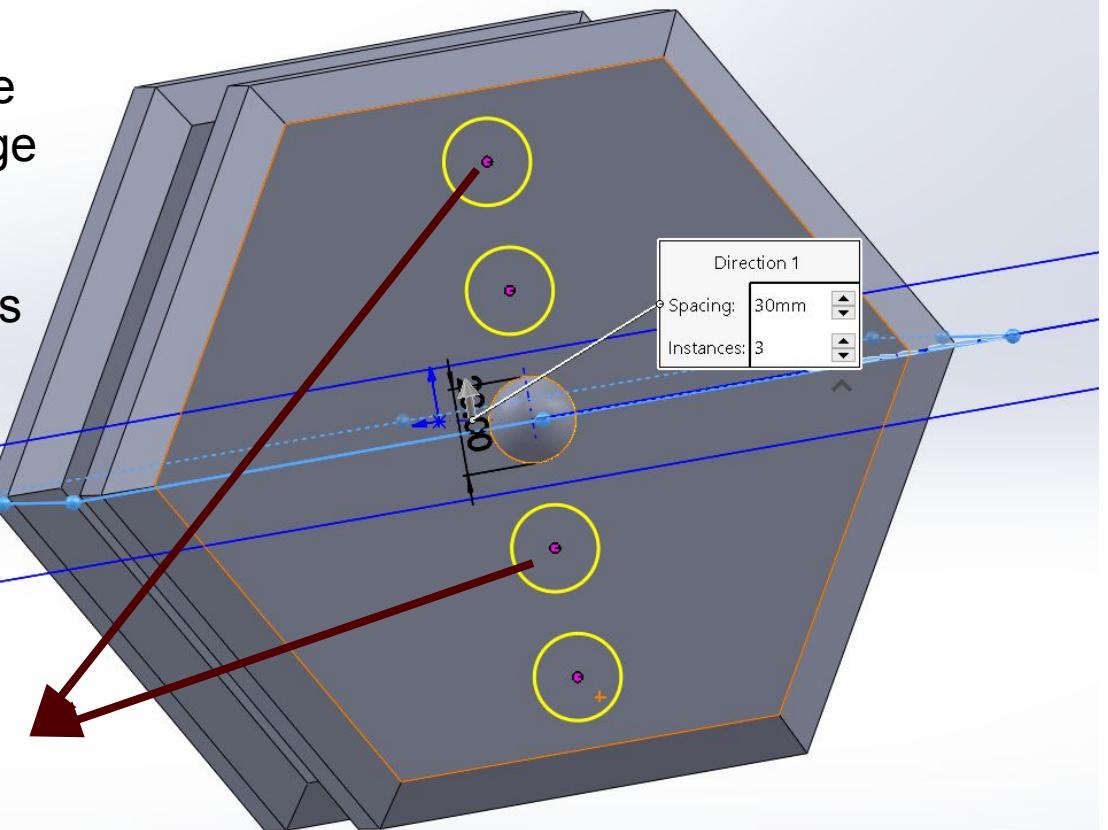


Normal to face
Parallel to edge

Spacing
Count, includes
original

Feature(s)

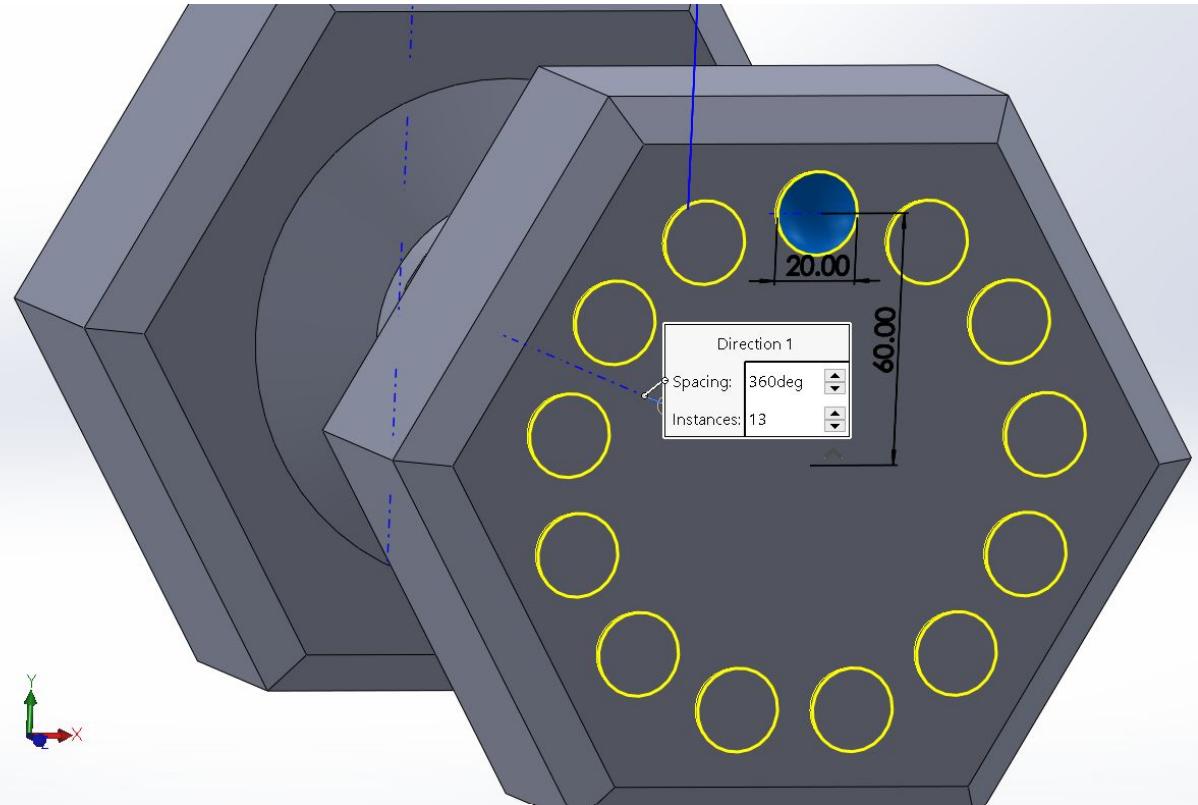
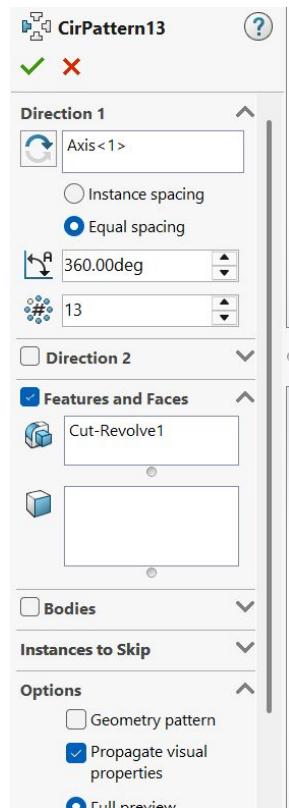
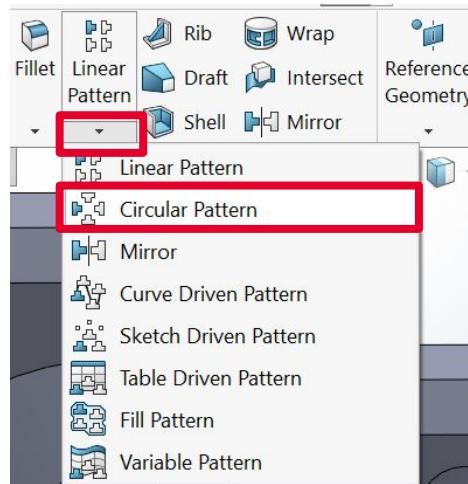
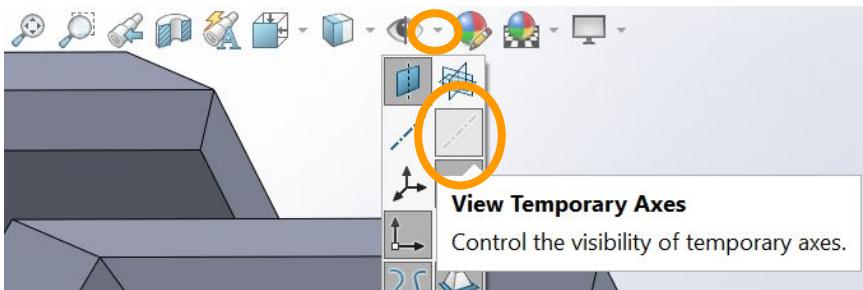
Click the
pink dots



Circular Pattern



Activating temporary axes is typical helpful



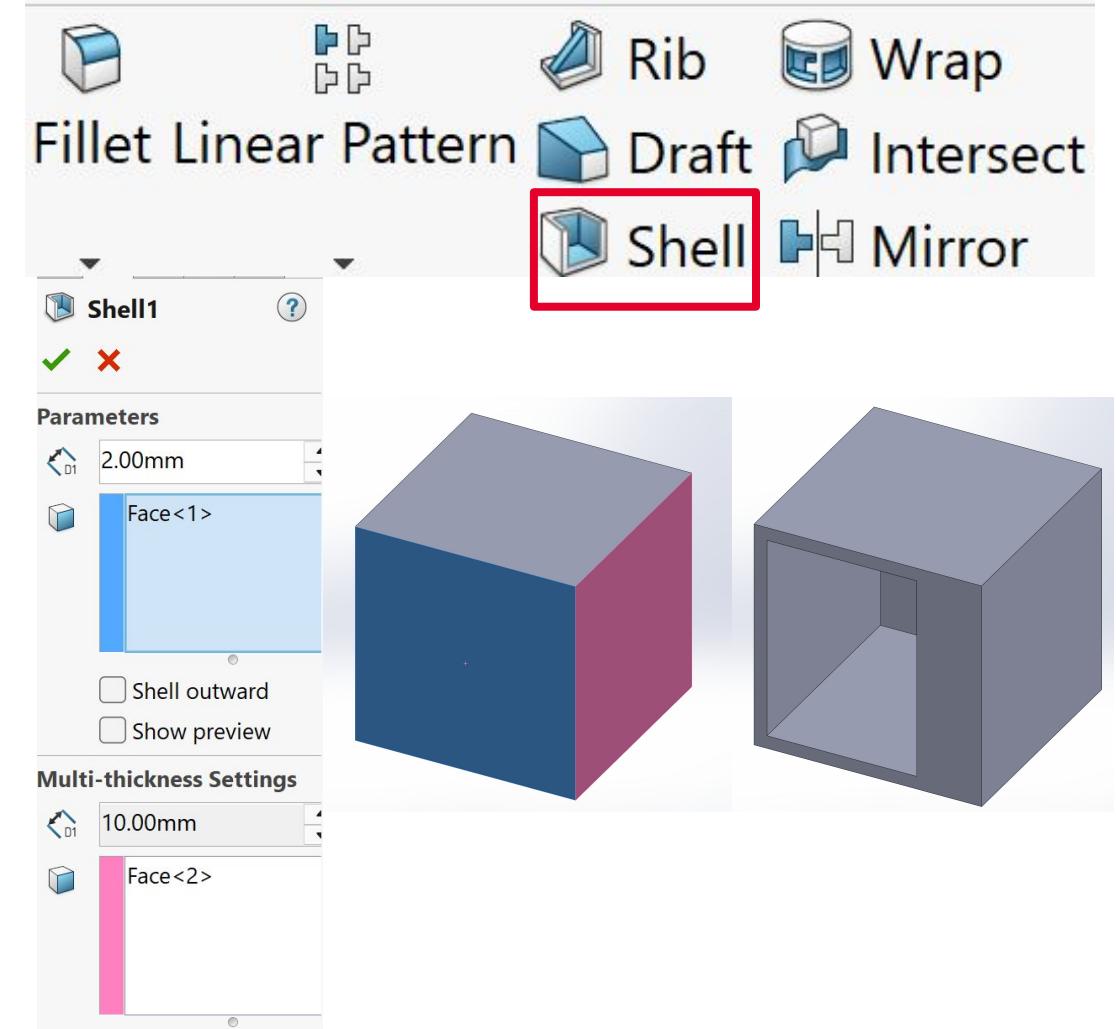
Shell



Hollows out a part with a specific thickness

Key Points:

- Selected faces are opened
 - If none, a closed hollow shape is created (Almost impossible to manufacture)
- Multi-thickness can be used to apply varying thickness to specific faces



Sweep



Involves two components:

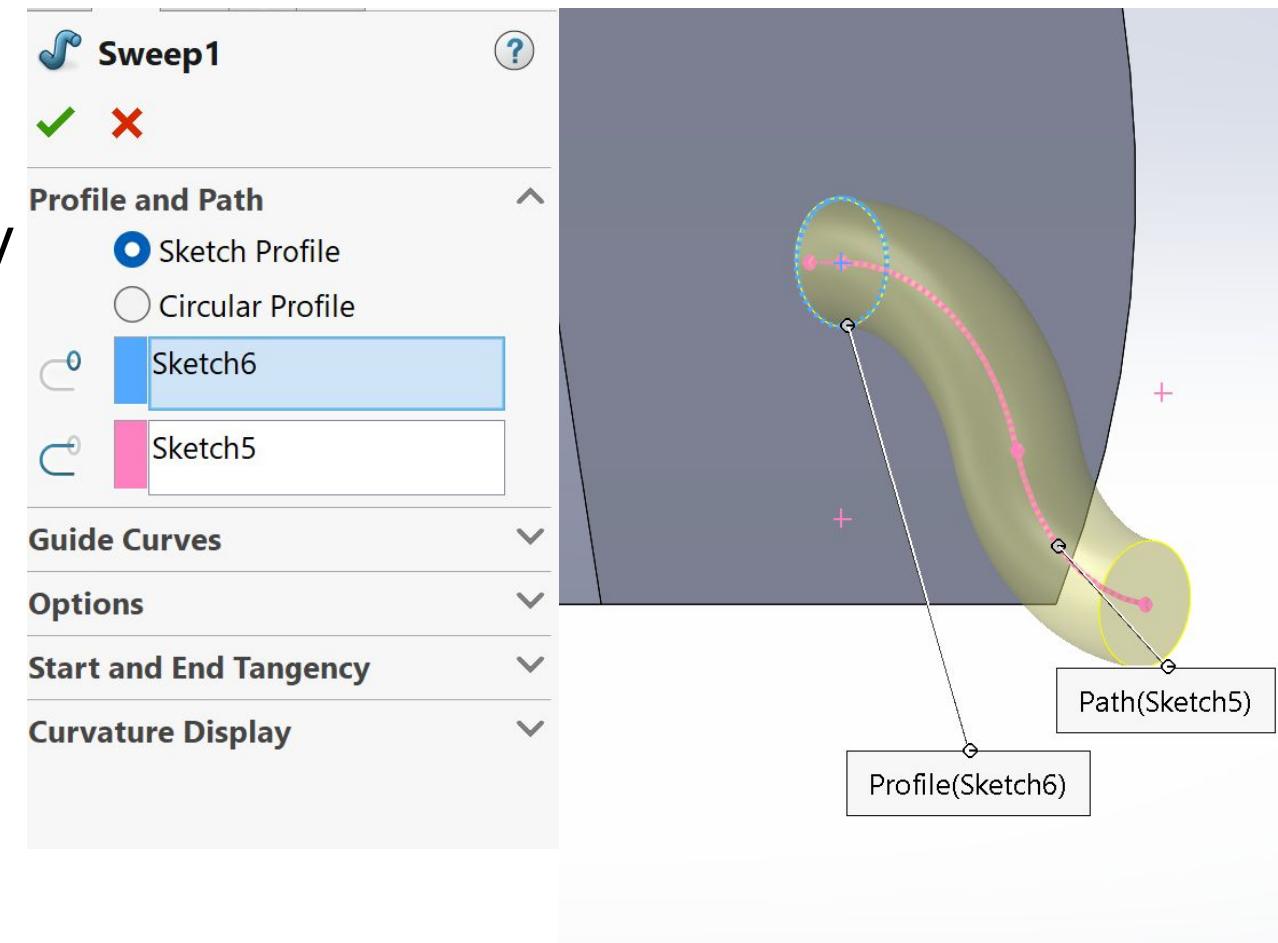
Profile: Defines cross-sectional shape

Path: Guides the profile's trajectory

Both are **usually** sketches

Alternatives:

- Edges can be used instead of sketches for the path.
- A circular profile can be used with a path, eliminating the need of a profile sketch.

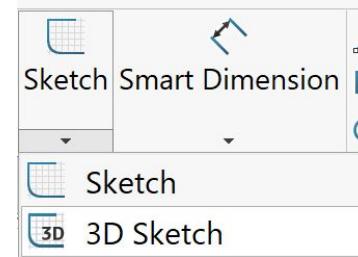


3D Sketches



Allows for sketches to be created in 3D space which is useful for complex geometries like frames and paths.

Hit the drop down menu under Sketch and then 3D Sketch to start one

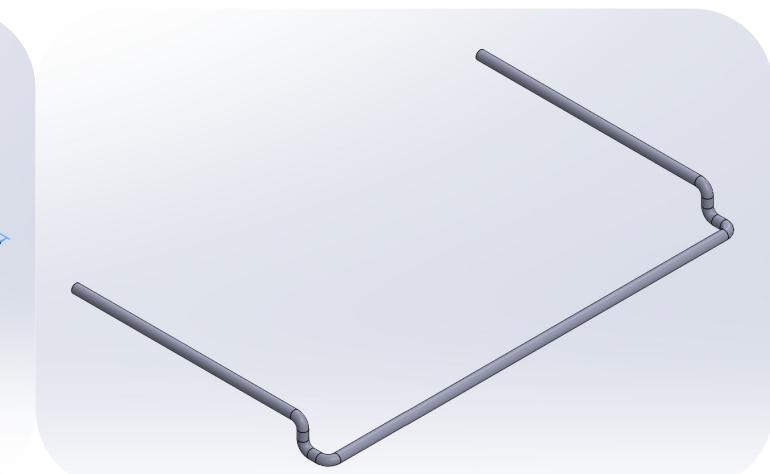
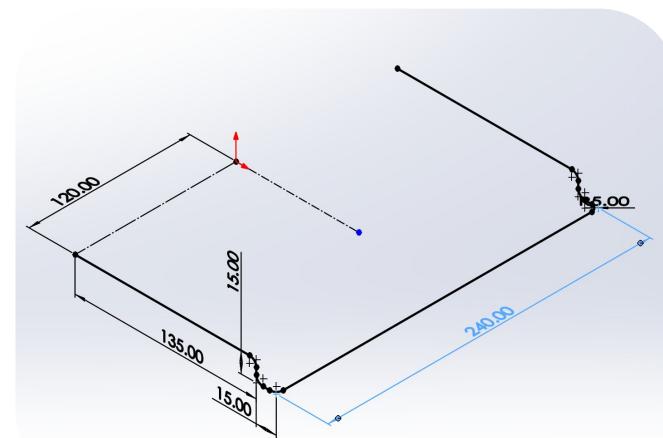


Key Features:

- No plane needs to be selected
- Tab key to switch between sketching on the XY, YZ, and XZ planes easily

Remember to use the **spacebar** to help navigate

Use a 2D Sketch when possible

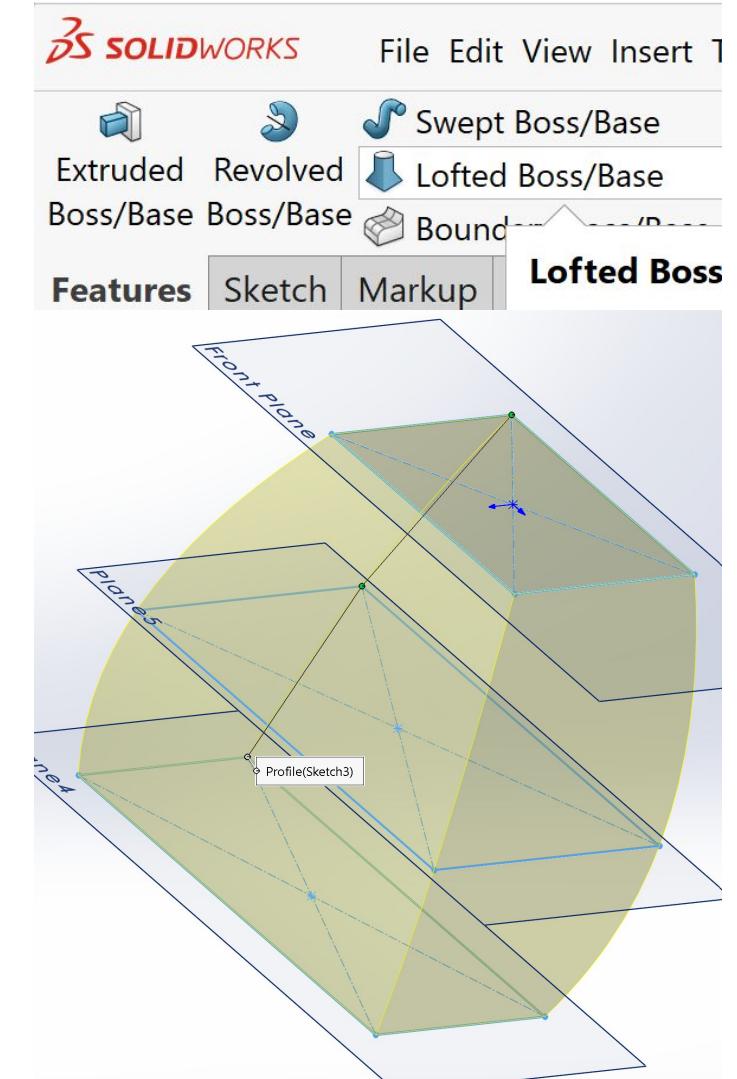


Lofts



Creates smooth 3D shapes by transitioning between two or more profiles. These profiles can be sketches or faces.

Guide curves can be used to control the transition path to refine the shape.

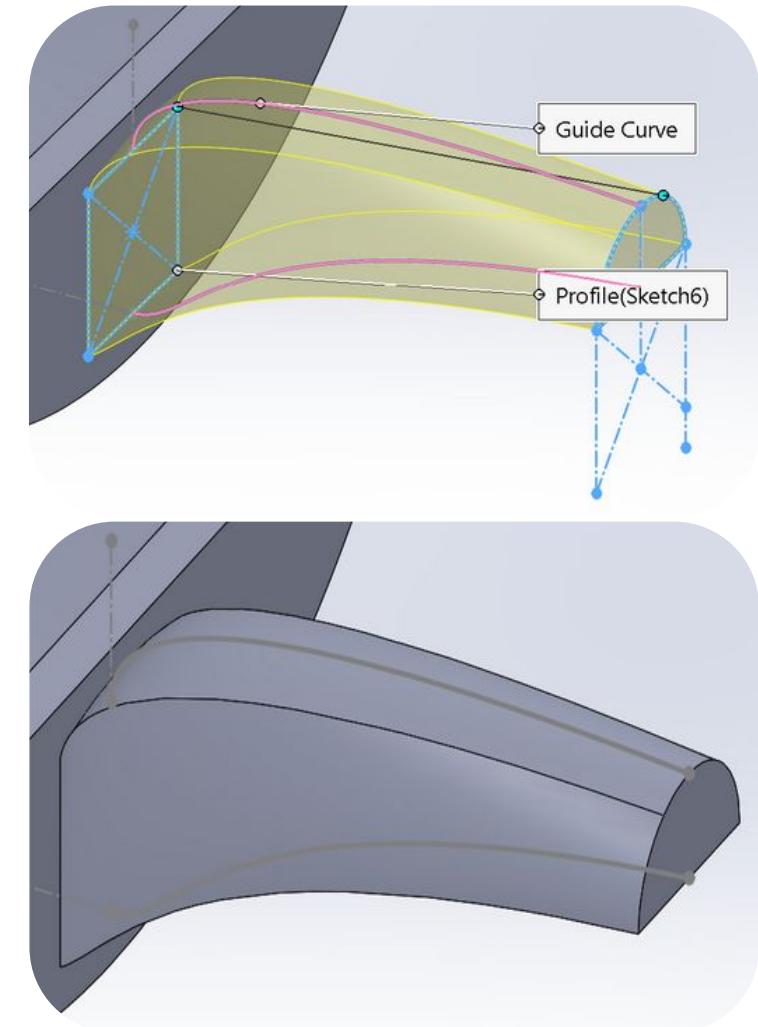


Guide Curves



Guide curves are paths that control the shape of lofts and sweeps by defining intermediate transitions between profiles or along paths.

They can be either: sketches, model edges, or reference curves



Measure Tool

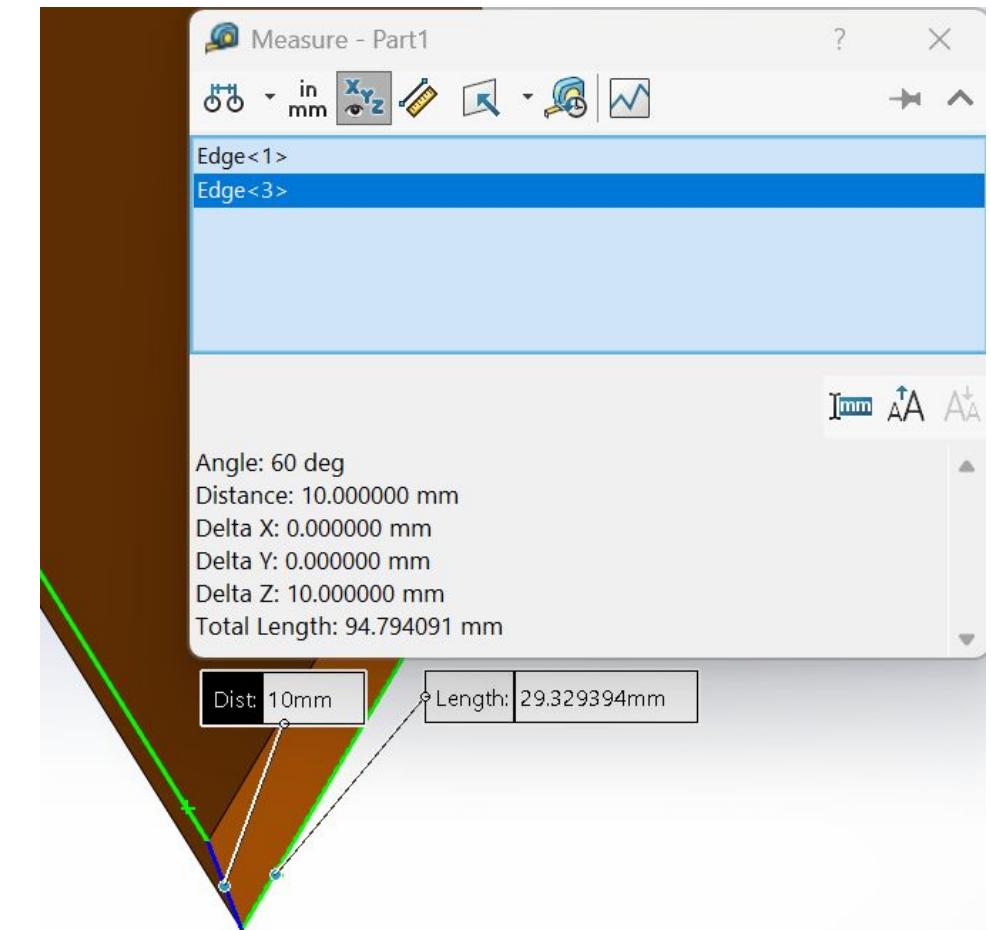


1. Click “Measure” within “Evaluate” tab
2. Select geometry to measure

Mostly used for:

- Determining various distances
- Angle between faces/edges
- Length of edges
- Area / Perimeter of faces

*Works in parts and assemblies



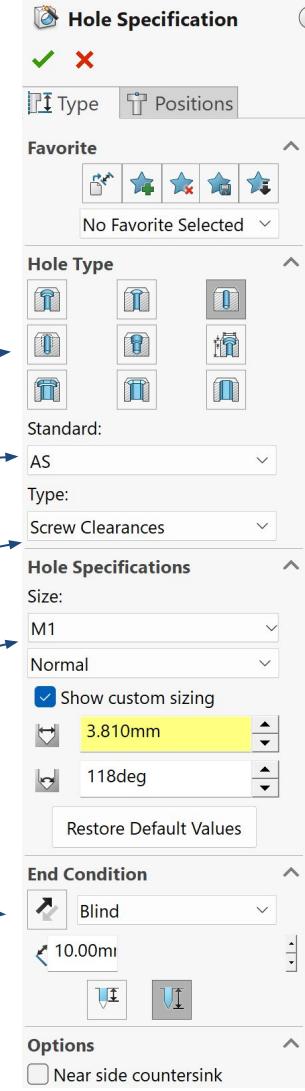
Hole Wizard



Allows for the creation of standardized holes, and is a Sketch and Feature creation tool in one.

To use:

- Click on Hole Wizard in the Features tab
- Select the hole type
- Specify the unit standard
- Type - provides standard sizing for drill bits, dowels, tap sizes, and screw clearances
- Size - includes normal and fits for screw clearances (loose, normal, close)
- Specify end conditions



Hole Wizard - Hole Types



Counterbore



Countersink



Hole



Straight Tap



Tapered Tap



Legacy Hole



Counterbore Slot



Countersink Slot



Slot

Global Variables



Equations, Global Variables, and Dimensions



Filter All Fields



Name	Value / Equation	Evaluates to	Comments
Global Variables			
"Value"	= 50	50.000000	
"Equation"	= "Value" * 3 / 2 + 10	85.000000	
"Angle"	= 60deg	60.000000deg	
"Equation2"	= sin ("Angle")	0.866025	
Add global variable			
Features			
Add feature suppression			
Equations			
Add equation			

OK

Cancel

Import...

Export...

Help

Automatically rebuild

Angular equation units:

Degrees

Automatic solve order

Link to external file:

- Global means the values apply to all configurations
- Variable is a changeable value

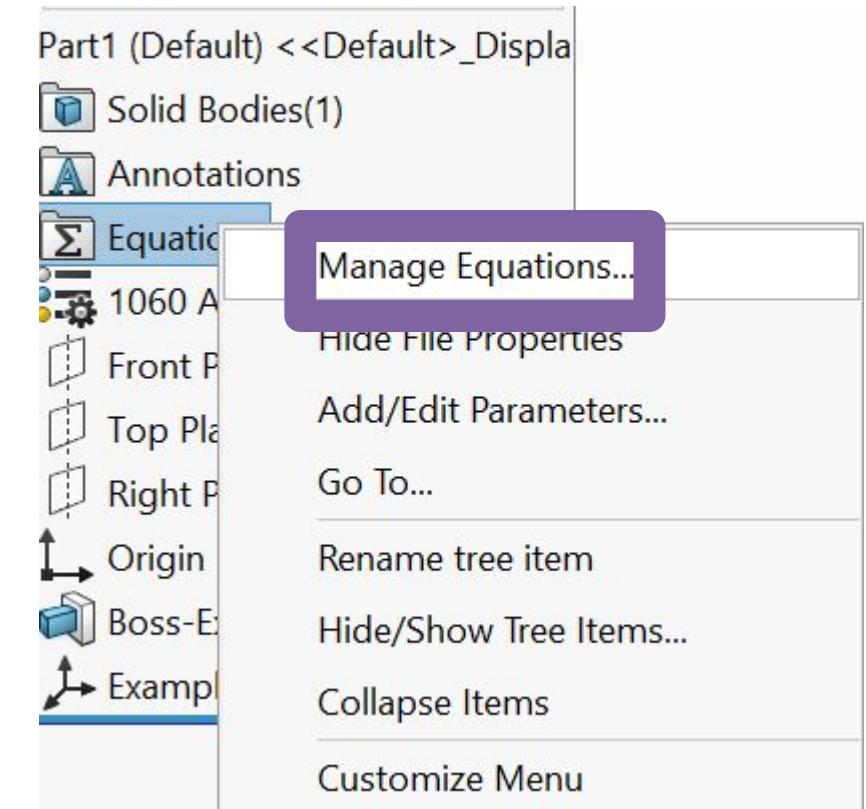
Creating Global Variables



1. Right click “Equations” in the feature tree
2. Select “Manage Equations...”
3. Create a variable name in first column
 - a. Avoid function names
4. Add a value or equation in second column

The first major benefit of parametric design by allowing:

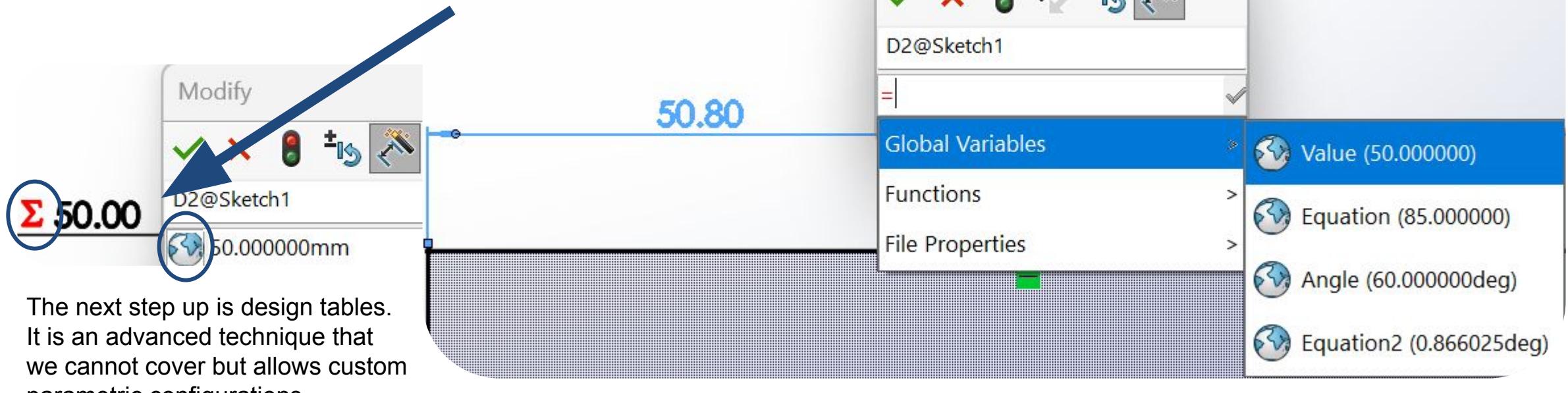
- Modification of multiple dimensions in one spot
- Equation based dimensions
- Limit repeated smart dimensions



Using Global Variables



1. Enter a sketch and use “Smart Dimension” as normal
2. Enter the “=” sign. (This is easy to forget)
 - a. The “=” sign is what allows the value to change
3. Enter the “<Var>”
4. Verify the link. Should look like



Material Selection

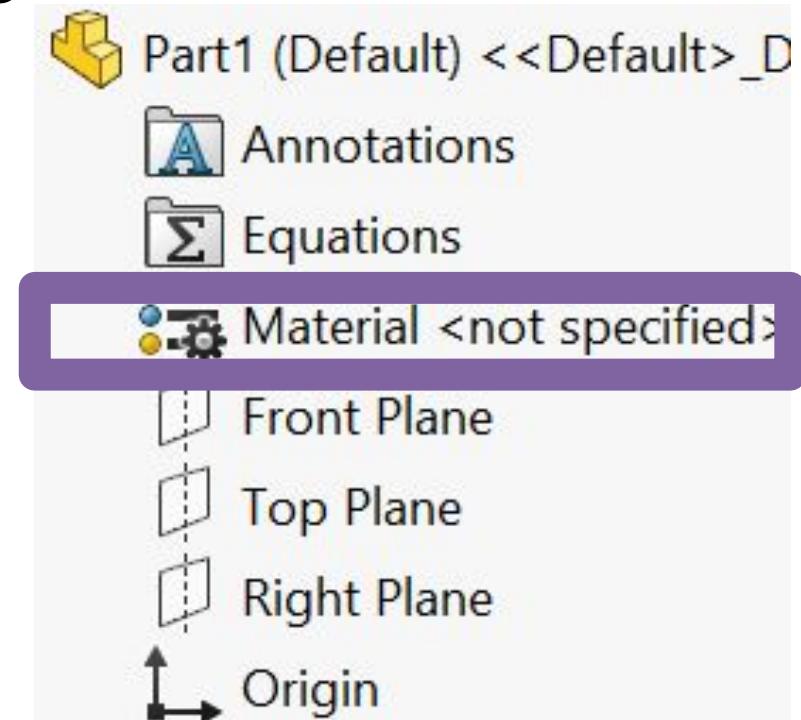


1. Right click “Material” in the feature tree
2. Click “Edit Material”
3. Find correct material and click “Apply”

*This is important for accurate mass and inertia property calculations

**There is no default PLA/PETG material

***Part specific

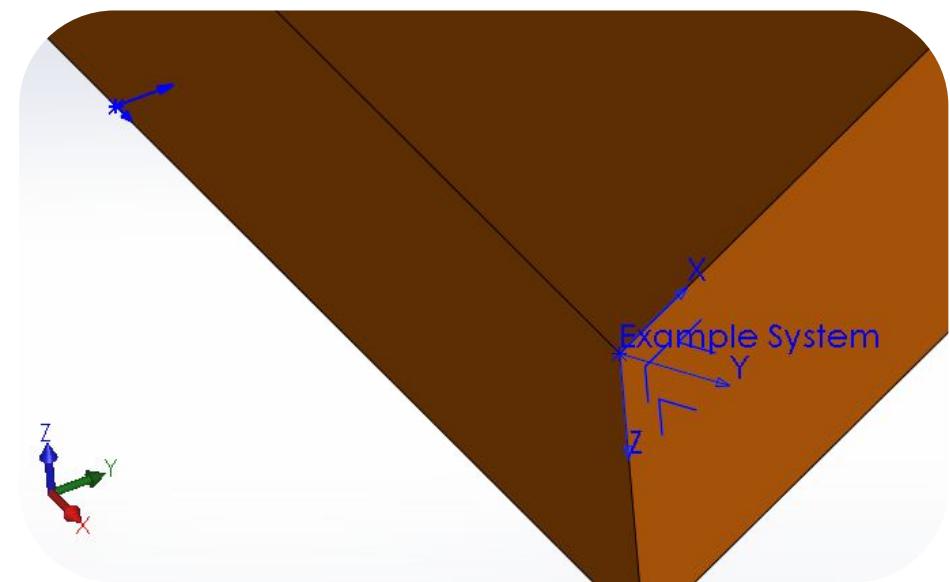


Reference Coordinate System



1. “Position” is the origin location
2. Orientation determines axis rotation
 - a. Edges/Points go in that direction
 - b. Faces are normal to plane
 - i. Prioritizes the second face

*Great for determining properties in assemblies



Mass Properties

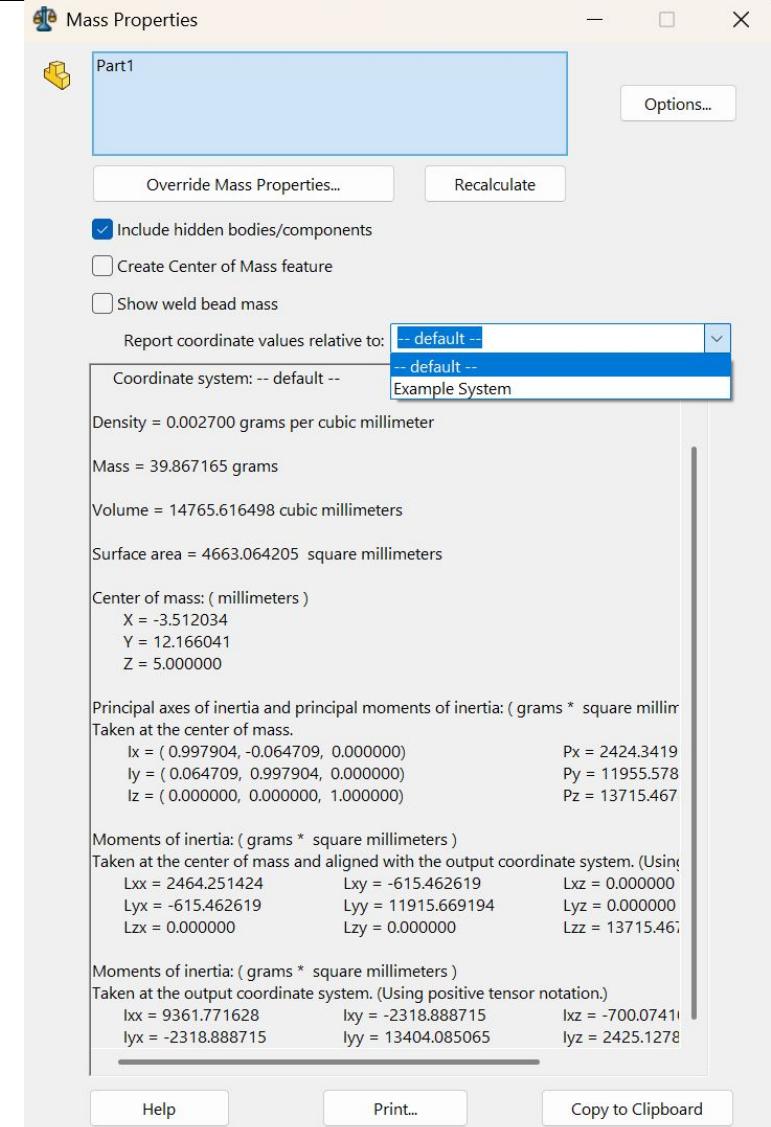


1. Click “Mass Properties” within “Evaluate” tab
2. Select coordinate system

*Make sure to select the correct coordinate system

Gives:

Mass, Volume, Surface area, CoM,
Inertia Axis, and Inertia



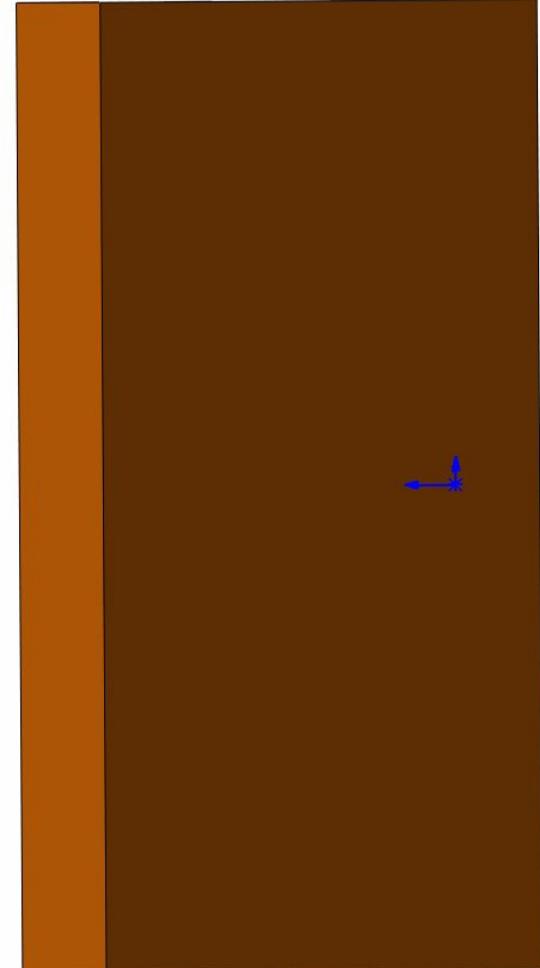
Part Appearance



1. Select the four color ball in the View (Heads Up) bar
2. Select intended scope (Part, Face)
3. Select a color or appearance

* There is no functional purpose outside of renders

** Typically used to identify between parts in an assembly

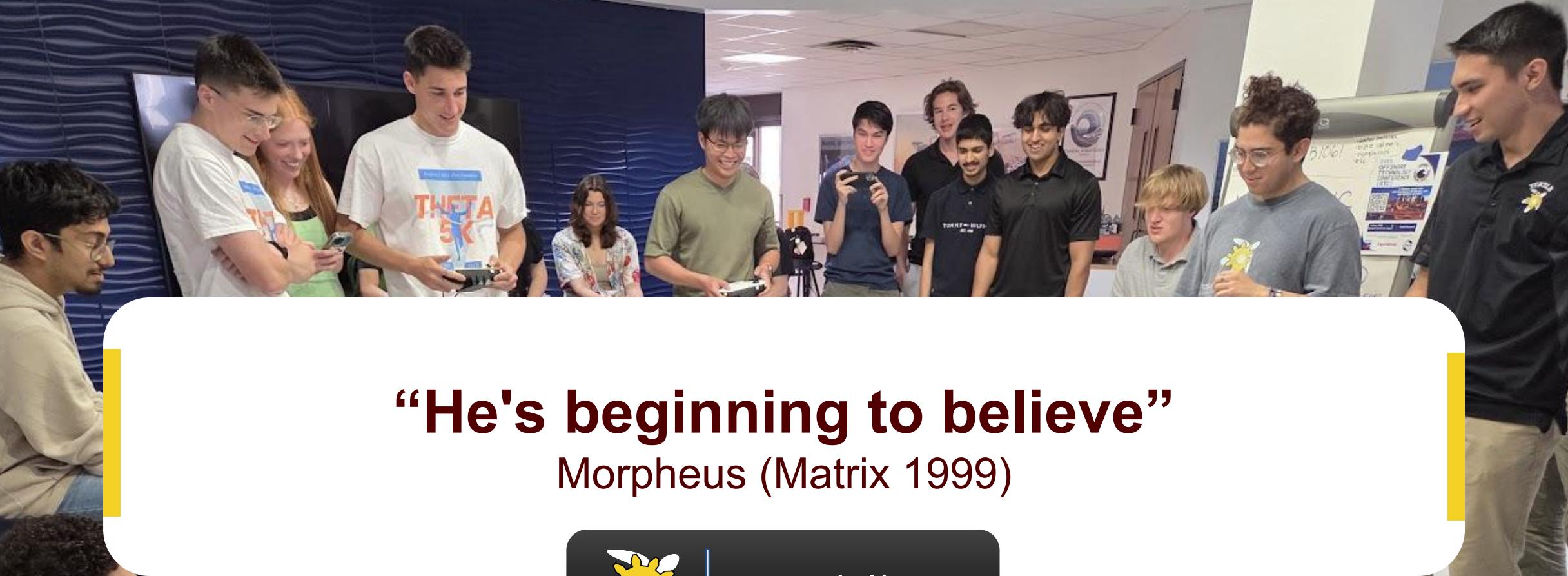




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Tools, Project and Process

Next Week



“He's beginning to believe”
Morpheus (Matrix 1999)



Hatchling

