Advanced Parts

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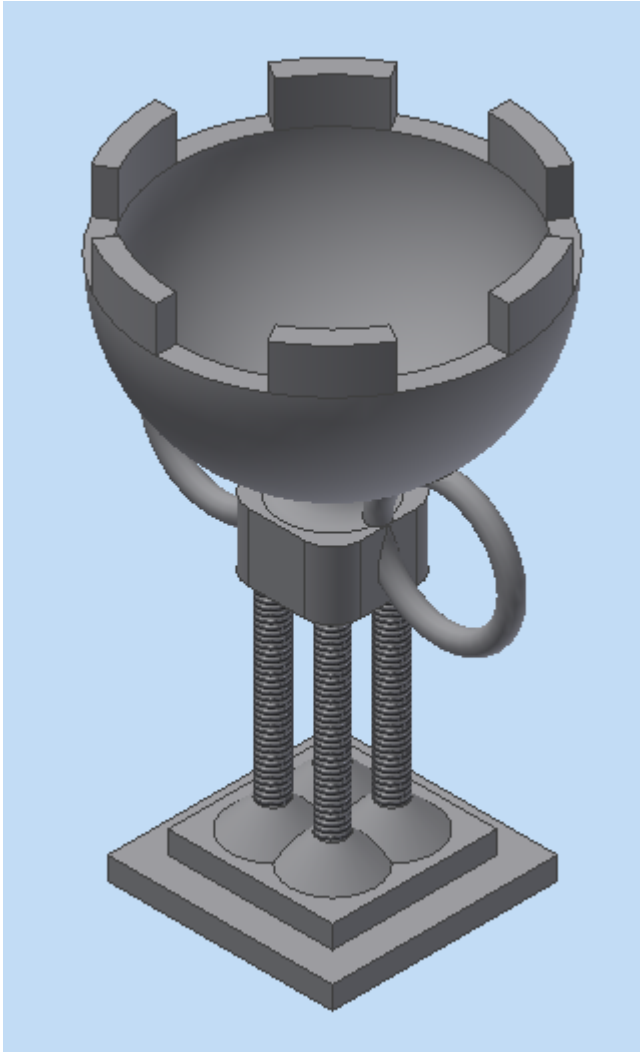
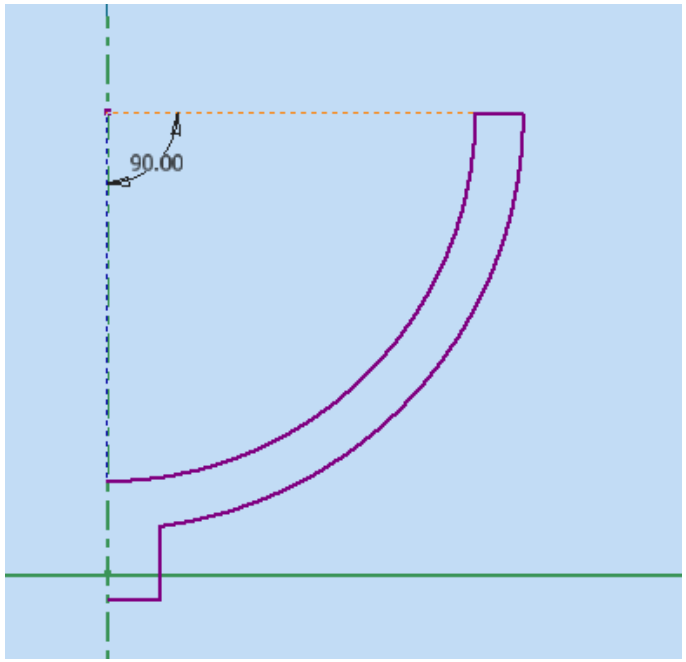
The purpose of this presentation is just to explore different features of inventor and have fun modeling!

Trophy



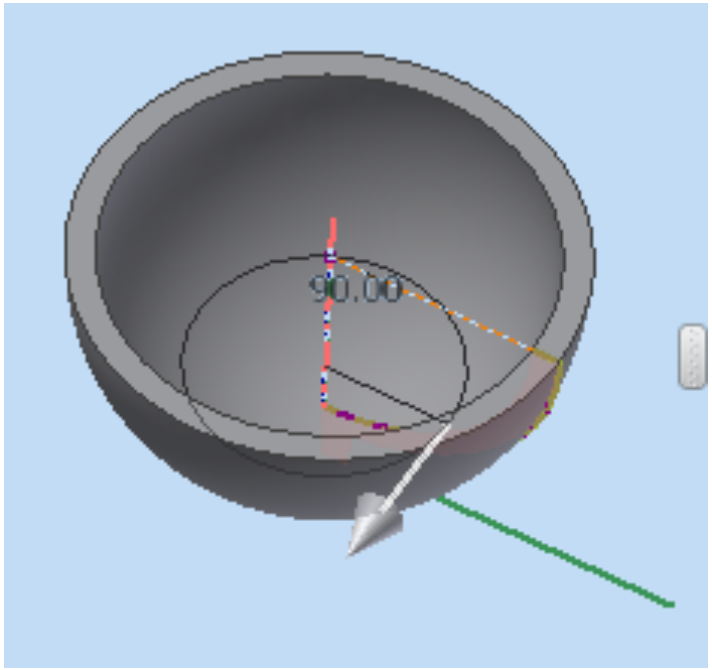
Make "Bowl"

- new Part, sketch on xy-plane
- Project Axes
  - Make centerline
- Create **arc** in first quadrant
  - Center + 2 End Points
  - 90 degree arc
  - ~.8in for outside arc
- Connect the ends



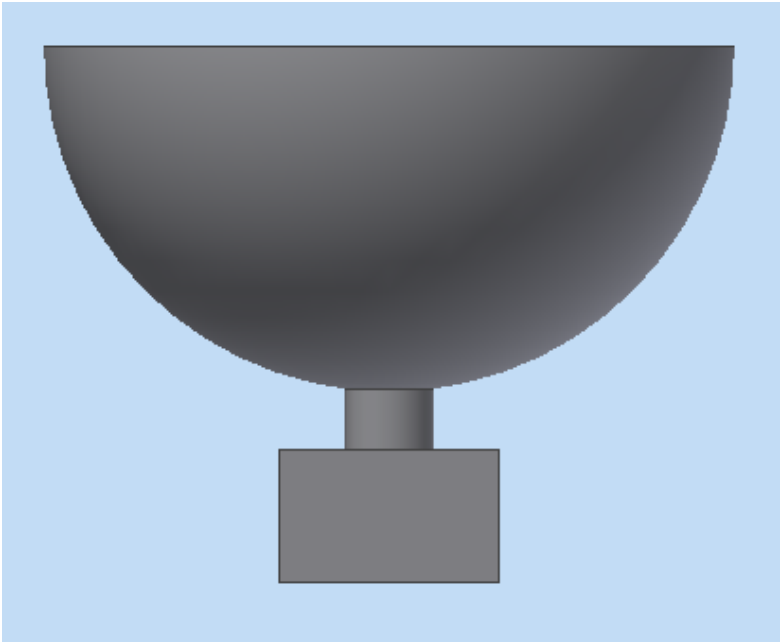
Revolve "Bowl"

- Finish Sketch
- Model --> Revolve
- Select
  - Profile
  - Axis
  - "Full" Revolve



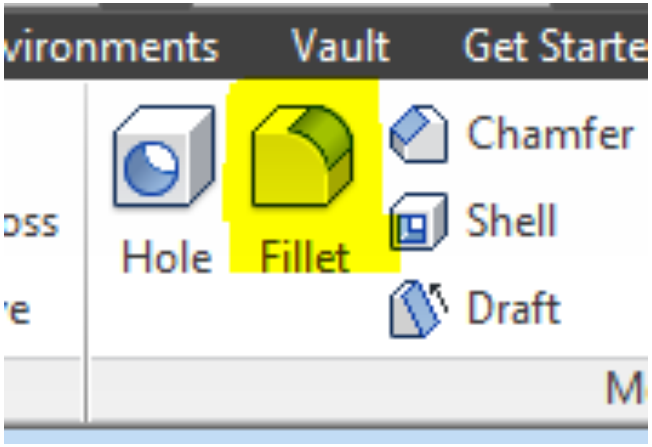
Create "Block"

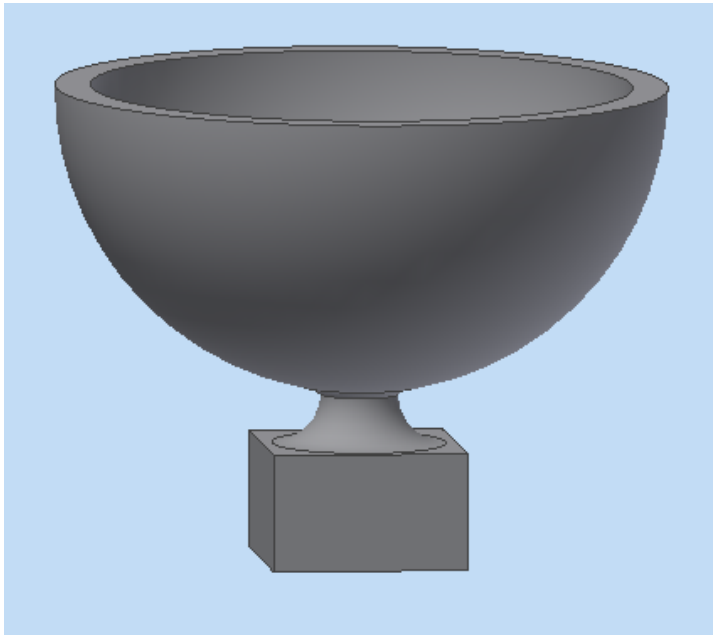
- Create new Sketch
  - Base of "Bowl"
  - Project center + axes
- Draw a square
  - Center
  - Width: .5in
- Finish Sketch
- Extrude



Add "Fillet"

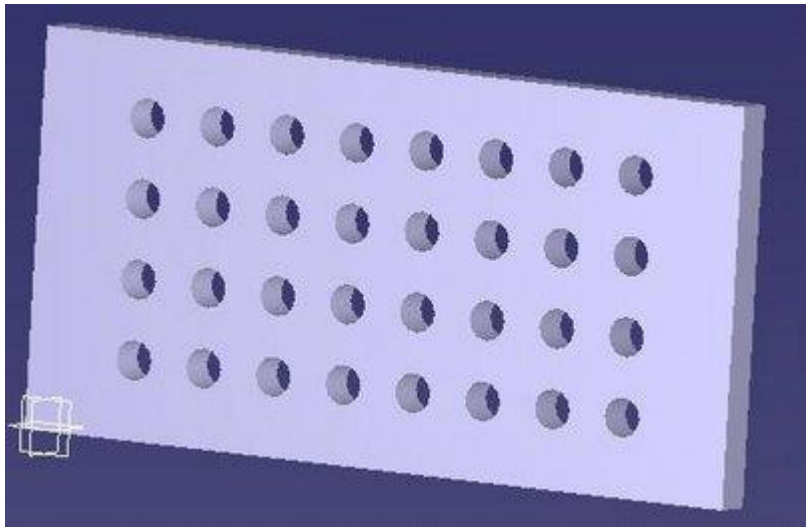
- **Fillet** makes round corner
  - 2D or 3D
- Start 3D Filet
- Select bottom and top edge of cylinder
- dimension: .125in

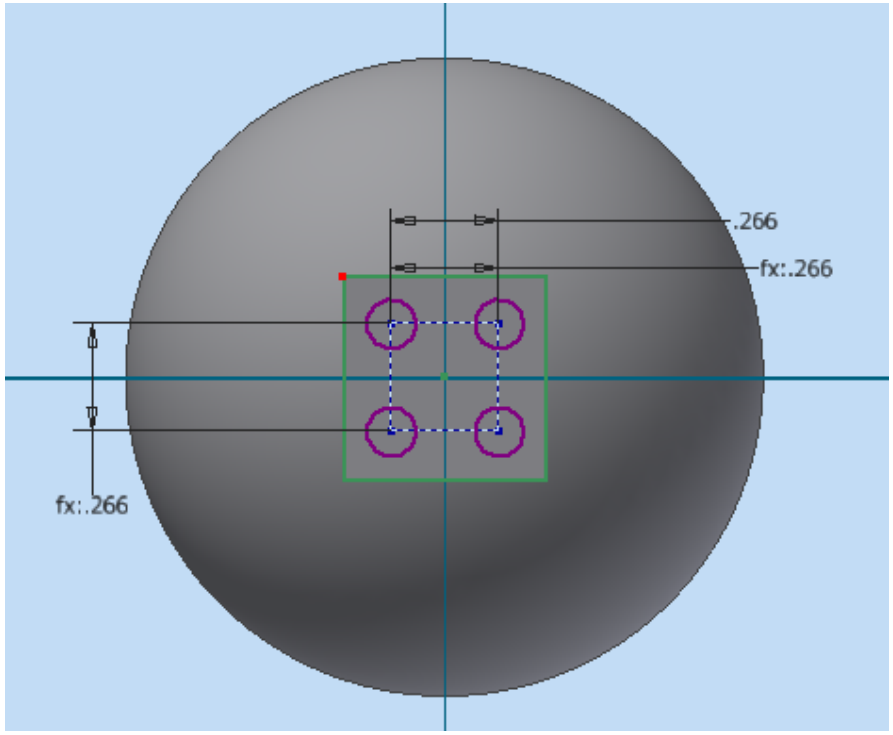




Add "Posts"

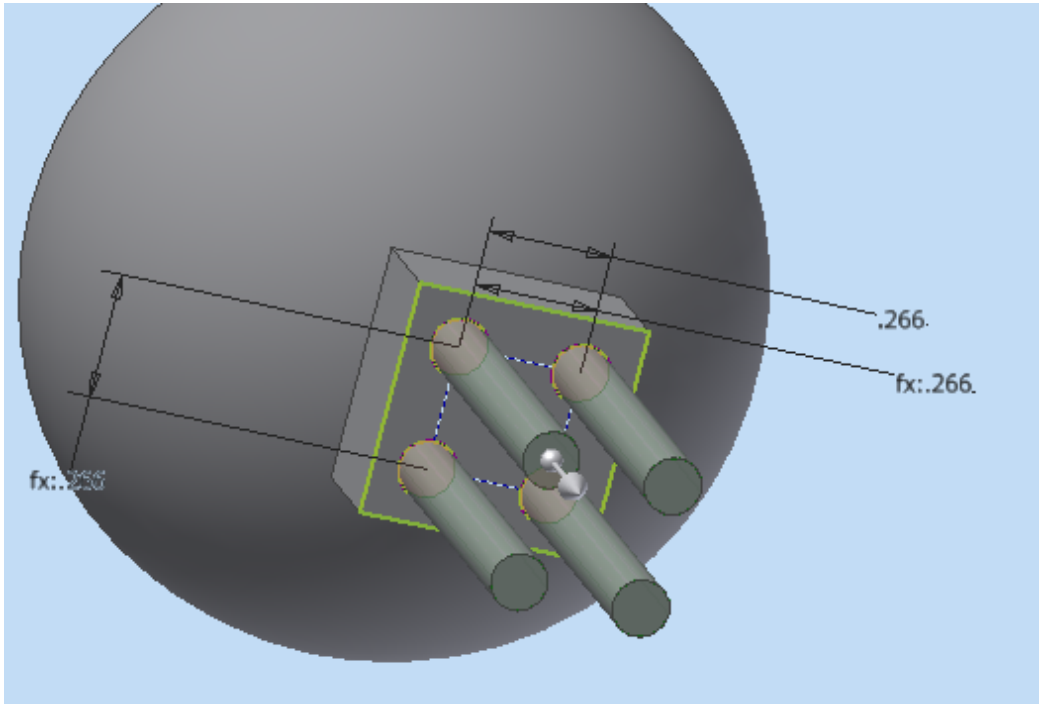
- Create new Sketch
  - Bottom of Block
  - Project Axes and Block
- Draw Circle
  - Offset square
  - **Construction line**
  - Center on diagonals
  - circle dia: 1/4in
- **Rectangular Pattern**
  - get dimension
  - 2 - Direction 1
  - 2 - Direction 2
  - change direction





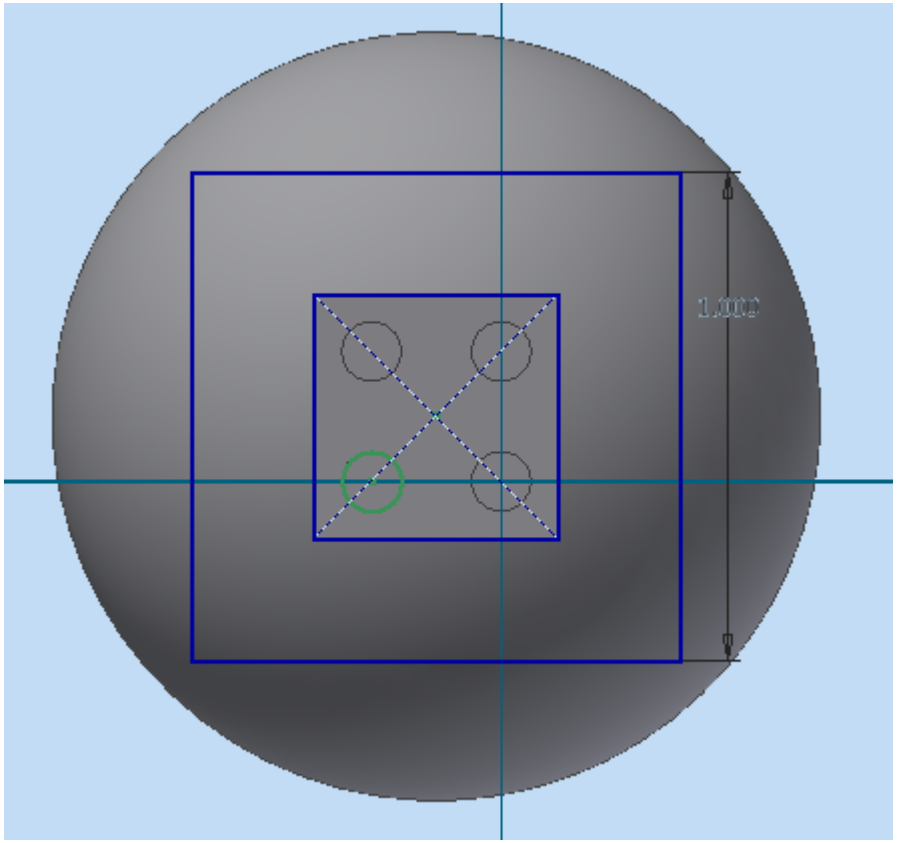
Extrude "Posts"

- Finish Sketch
- Extrude the 4 posts
  - 1in



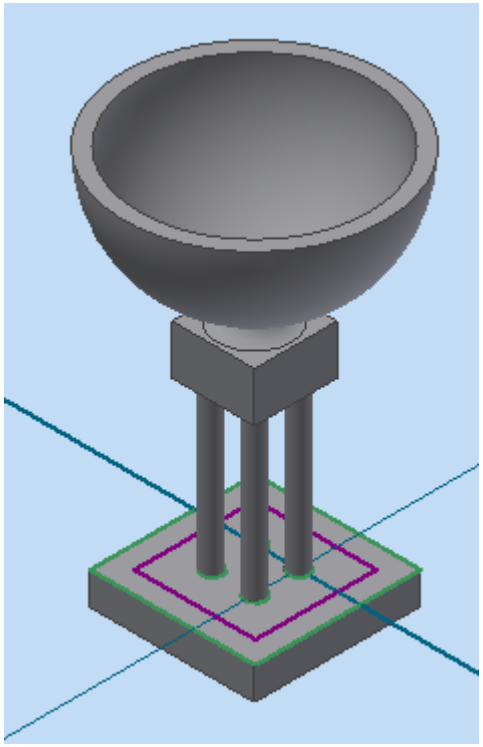
Create "Base"

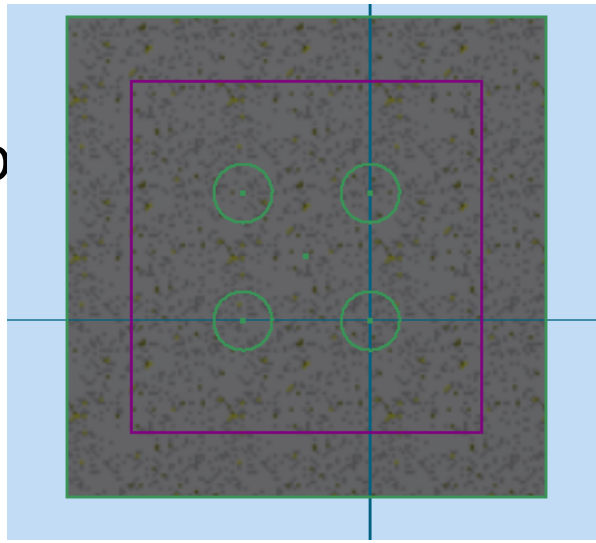
- Create new Sketch
  - Bottom of Posts
  - Project Posts and Axes
- Make Rectangle
  - Center about origin
  - Side = 1in
- Finish Sketch
- Extrude



Add "Step"

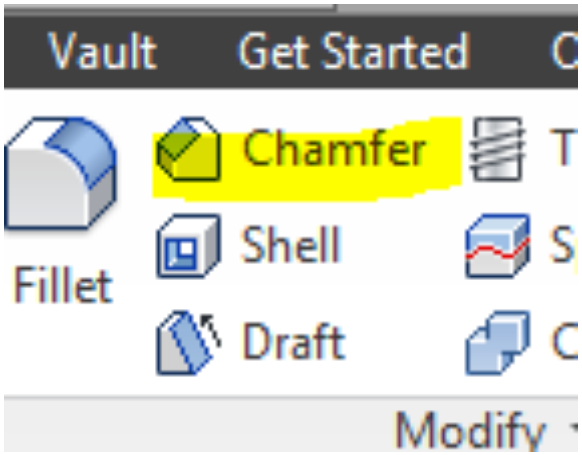
- Create new sketch
  - On top of base
  - F7 --> **Slice Graphics**
  - Project Axes and Base
- Draw a smaller 2nd rectangle
  - Offset
- Finish Sketch
- Start Extrude Command
  - "Cut" away the outside to make a step

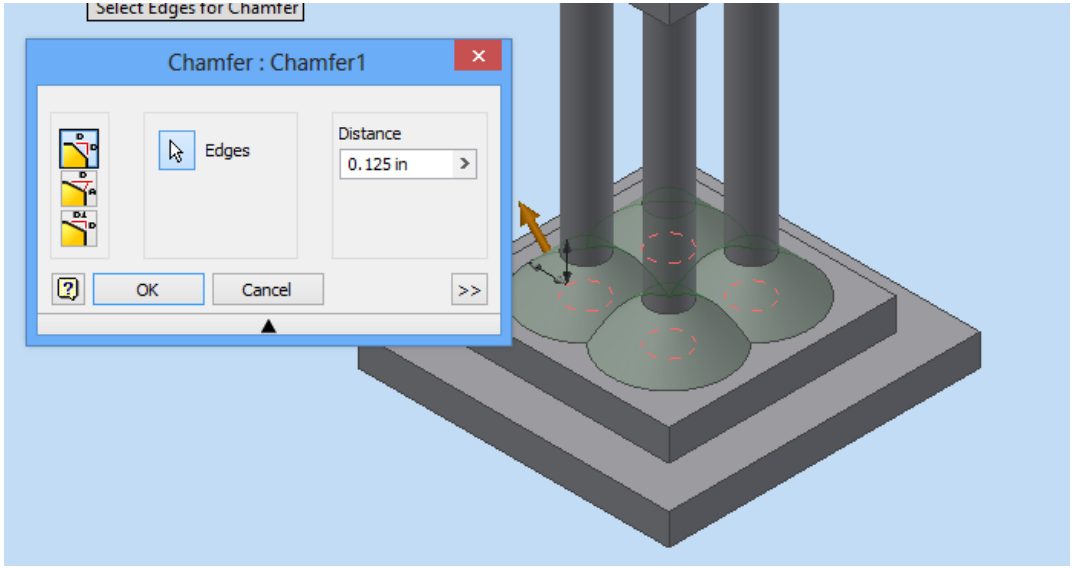




Add "Chamfer"

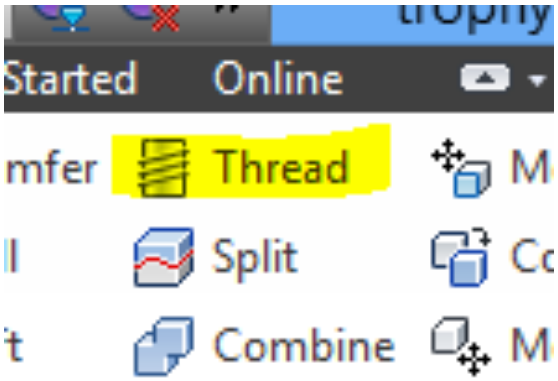
- Start Chamfer Command
  - Cuts the corner off of a meeting point
- Select edges where posts meet the base
  - 0.125 Radius

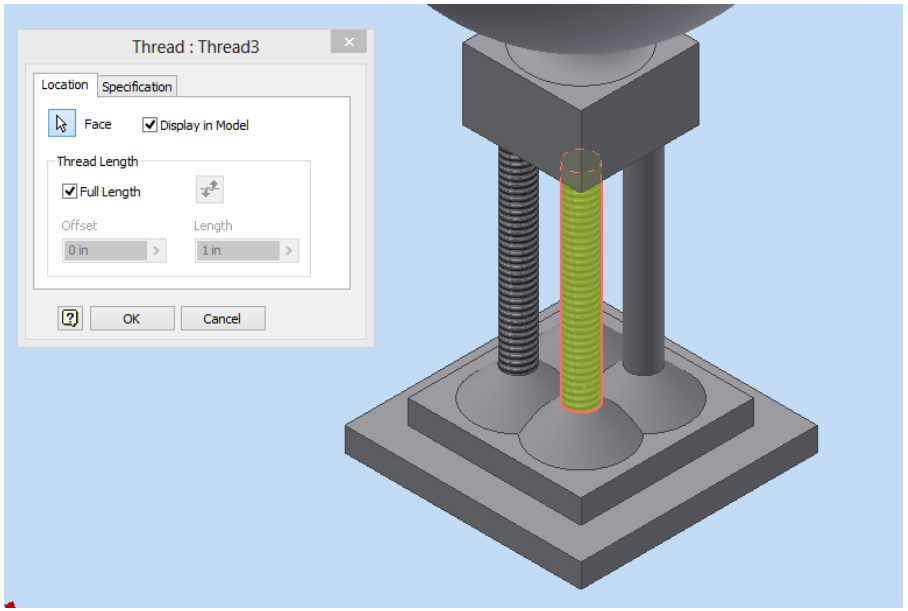




Add "Thread"

- Start Thread Command
  - Select the posts
  - View "Specifications" tab
  - 5-44 UNF - What does it mean?





Thread specification - How threads are notated / designated:

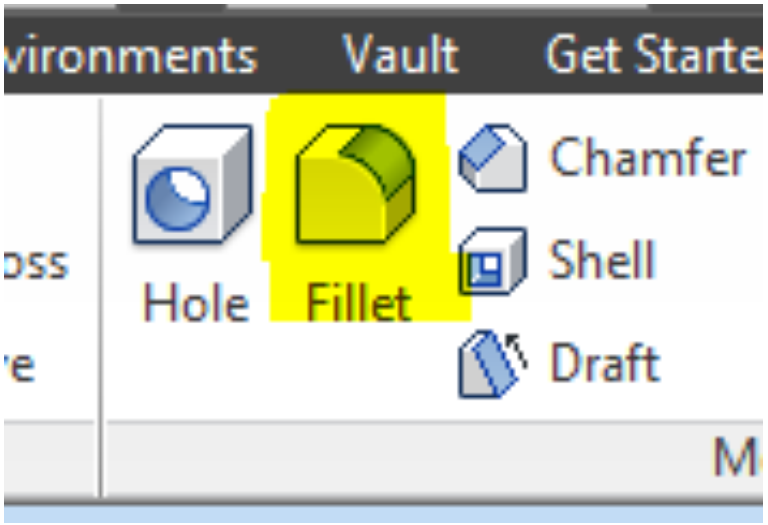
example Unified thread designation:

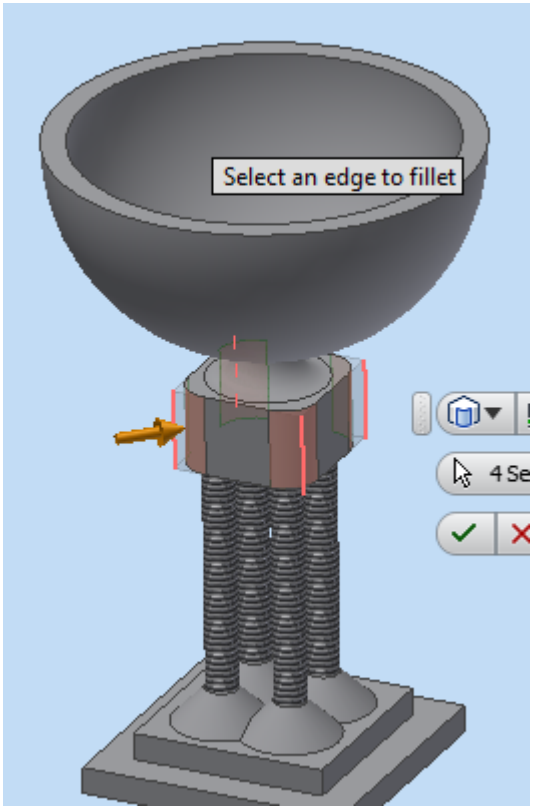
1/4-20 UNC-2A

- **1/4** - the nominal diameter, also the major / largest diameter
- **-20** - the number of threads per inch
- **UNC** - UNC = Unified Coarse, UNF = Unified Fine. Uou may also see UNRC or UNRF. These refer to an external Unified Rounded thread (there is no internal rounded thread). UNRC's and UNRF's are interchangeable with their non-R counterparts. The only difference is that the vallies (roots) of external R threads have a mandatory rounded shape, whereas with the UNC and UNF threads the roundness is optional.
- **-2A** - This represents the tolerance / fit of the thread. There are 6 common options, 1A, 2A, 3A, 1B, 2B, and 3B. A=external, B=internal. 1 is the loosest fit, 3 is the most precise and tightest fit with potentially zero clearance. If the tolerance isn't specified, chances are it's the more common 2A or 2B designation. 1 is hardly used, and only in cases where frequent re-assembly is needed or the threads need to work even with significant damage. Class 3 have slightly greater stripping resistance, and are common in the aerospace industry.

Add "Fillet" (again)

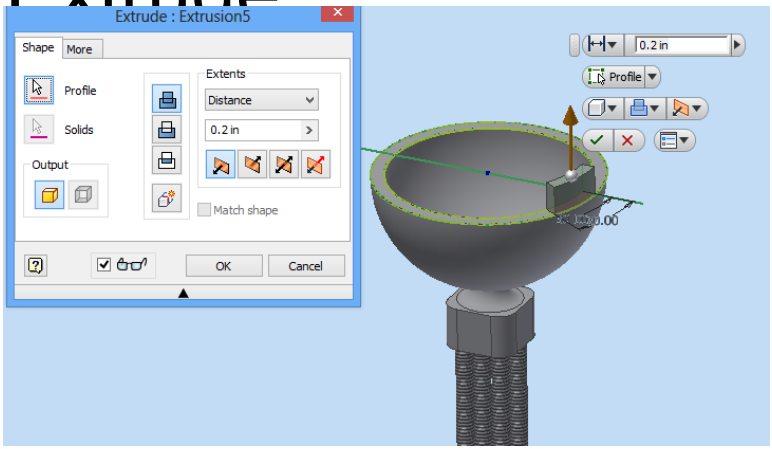
- Start 3D fillet
- Select corners of the "block"

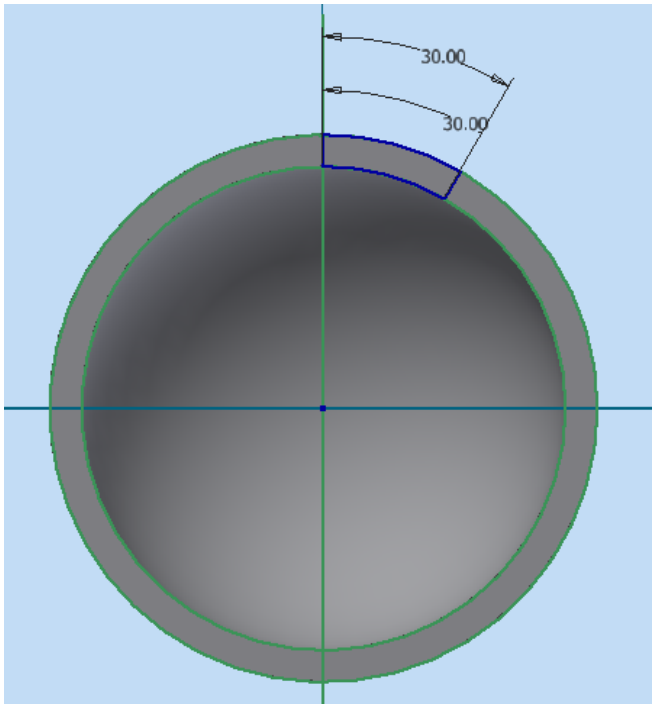




Add "Castle"

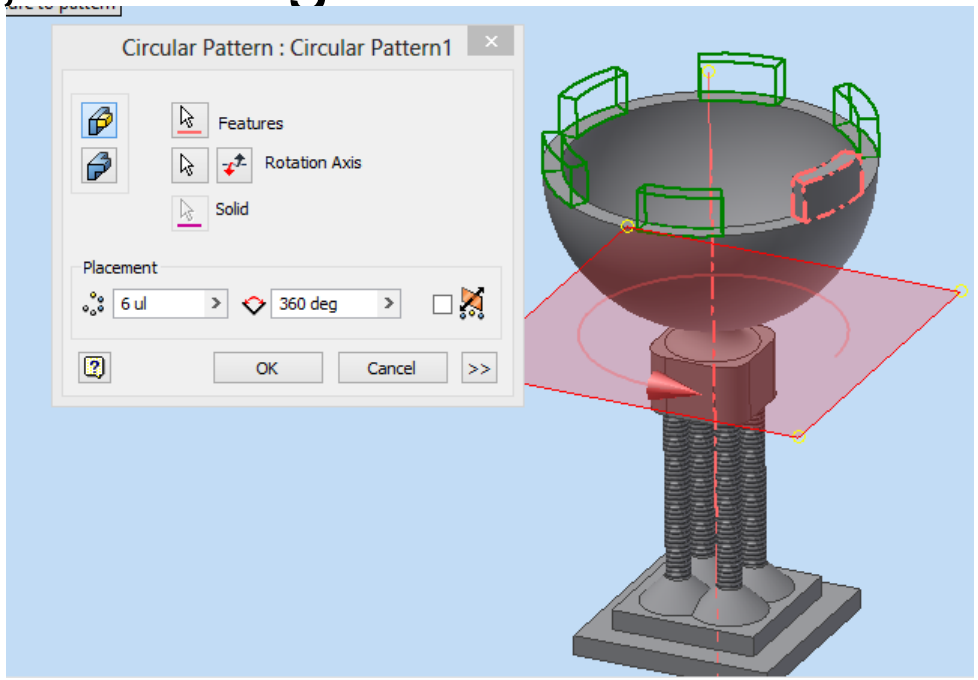
- Create new sketch on the top of the "bowl"
  - Project Geometry
- Draw 2 arcs
  - 30 degrees
- Connect the ends with lines
- Finish Sketch
- Extrude





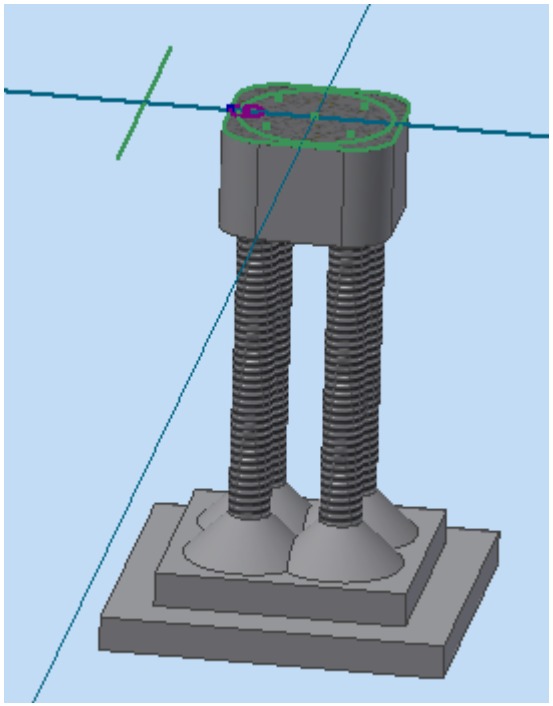
Circular Pattern

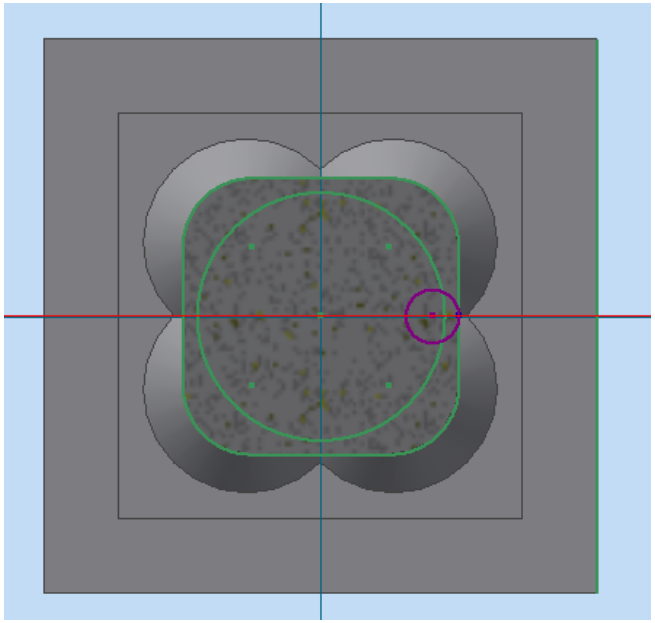
- Start Circular Pattern
- Select Feature
- Select Center Axis
- 6 times, 360 degrees



Mirror

- New Sketch above mid block
- Create a circle tangent to the side
- Create a new line outside for revolve axis





Then...

- Revolve the circle around the axis
- Mirror
  - select feature and the plane