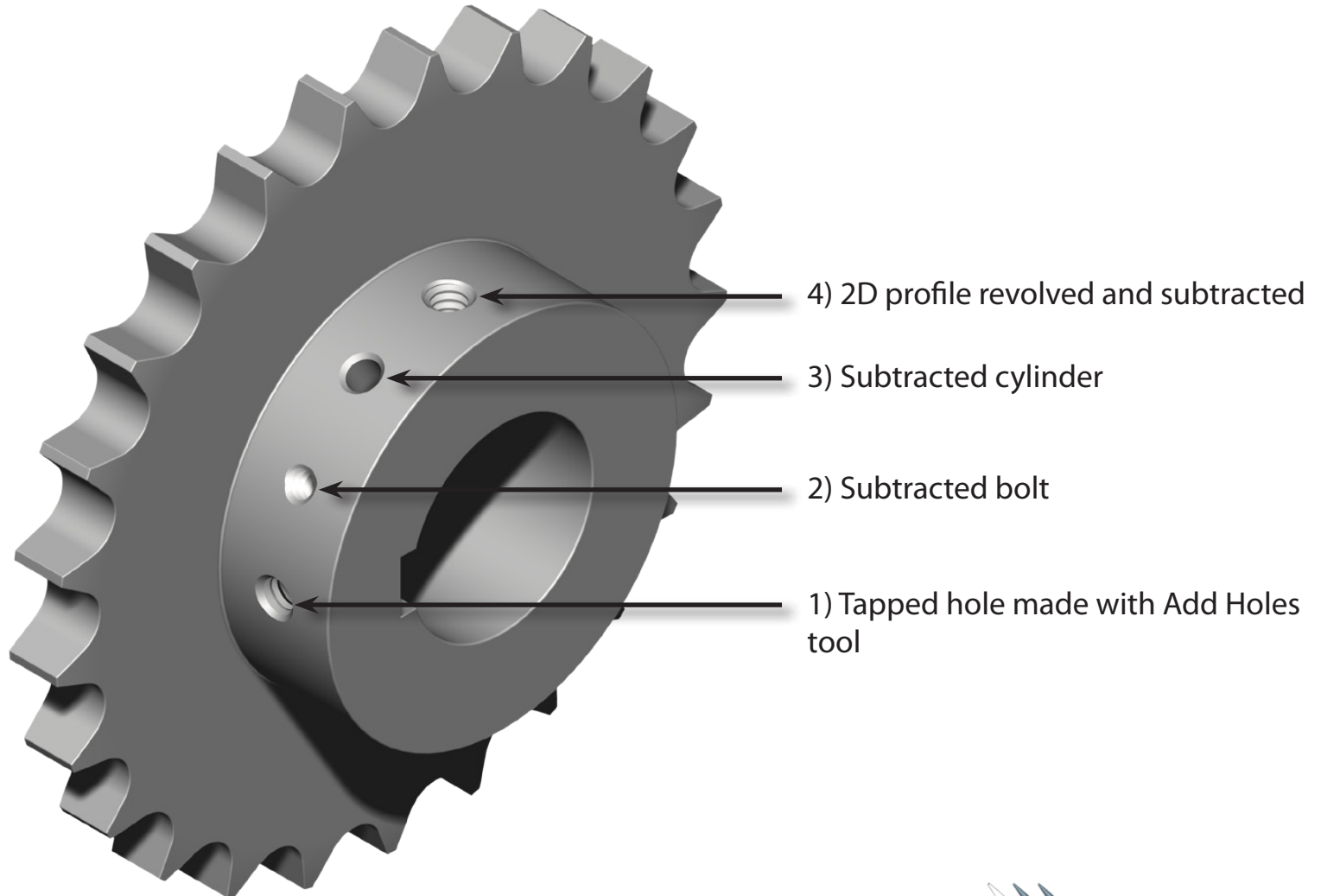


Finding a reliable method to place holes (tapped or otherwise) on a curved surface has perplexed many new users over the years.

Four methods come readily to mind when pondering the situation. They include:

- 1) Using the Add Holes tool, which is available only in the Platinum Edition.
- 2) Subtracting a bolt that has been created with the Thread3D Add-On tool.
- 3) Subtracting a simple cylinder.
- 4) Revolving a 2D profile and subtracting the result.



Using the Add Holes tool requires a bit of trick and this trick is not something that would readily occur to most users.

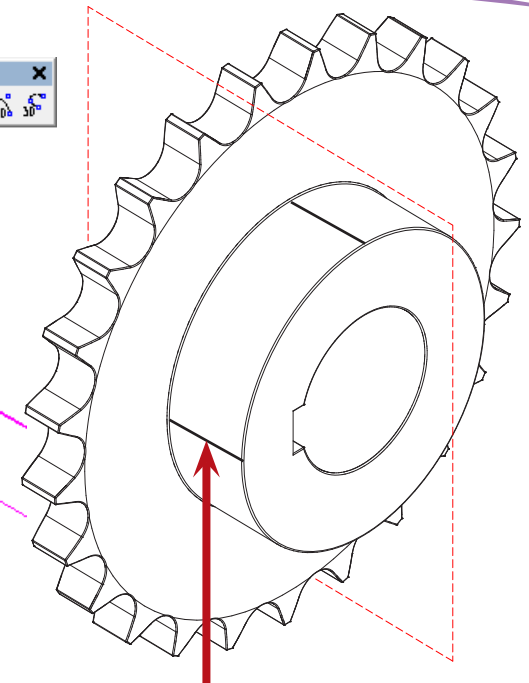
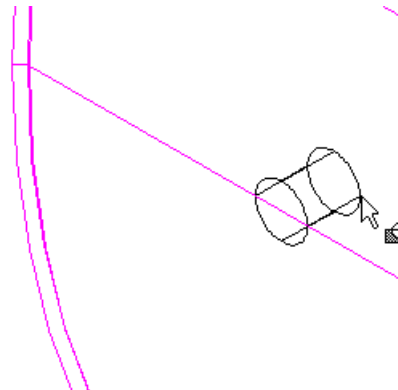
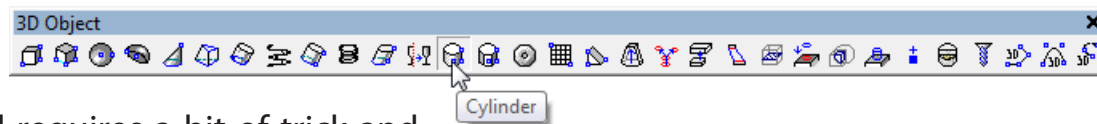
The Hole tool requires a flat plane and a point in order to work, so a user must create a flat plane on the side of the curve.

To do this a user must first align the Workplane parallel to the desired hole.

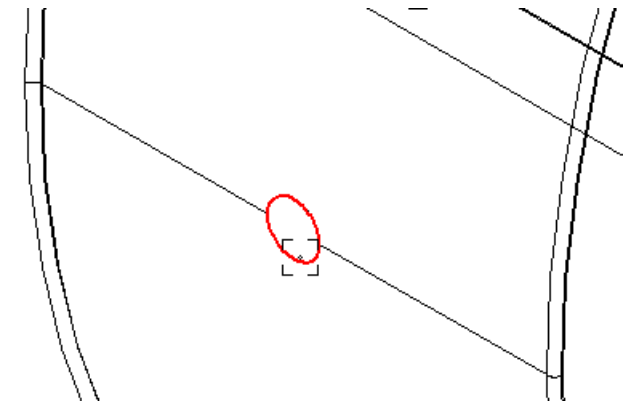
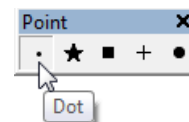
The user then creates a small cylinder that is tangent to the curve. In this example a user can Middle snap the line on the side of the boss. The cylinder must be made smaller than the final opening of the hole.

The small cylinder and the curved object (sprocket in this case) are then 3D Added, leaving a small flat circular face.

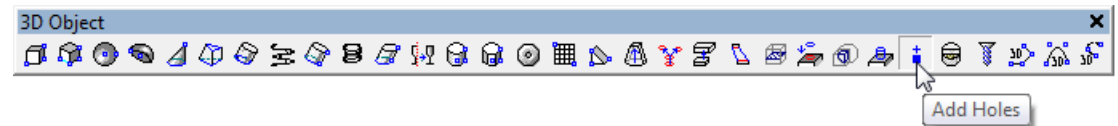
With the Auto Workplane by Face tool engaged and the Dot tool selected a user Center snaps a point to the highlighted facet. In progress to the right.



desired hole to go here



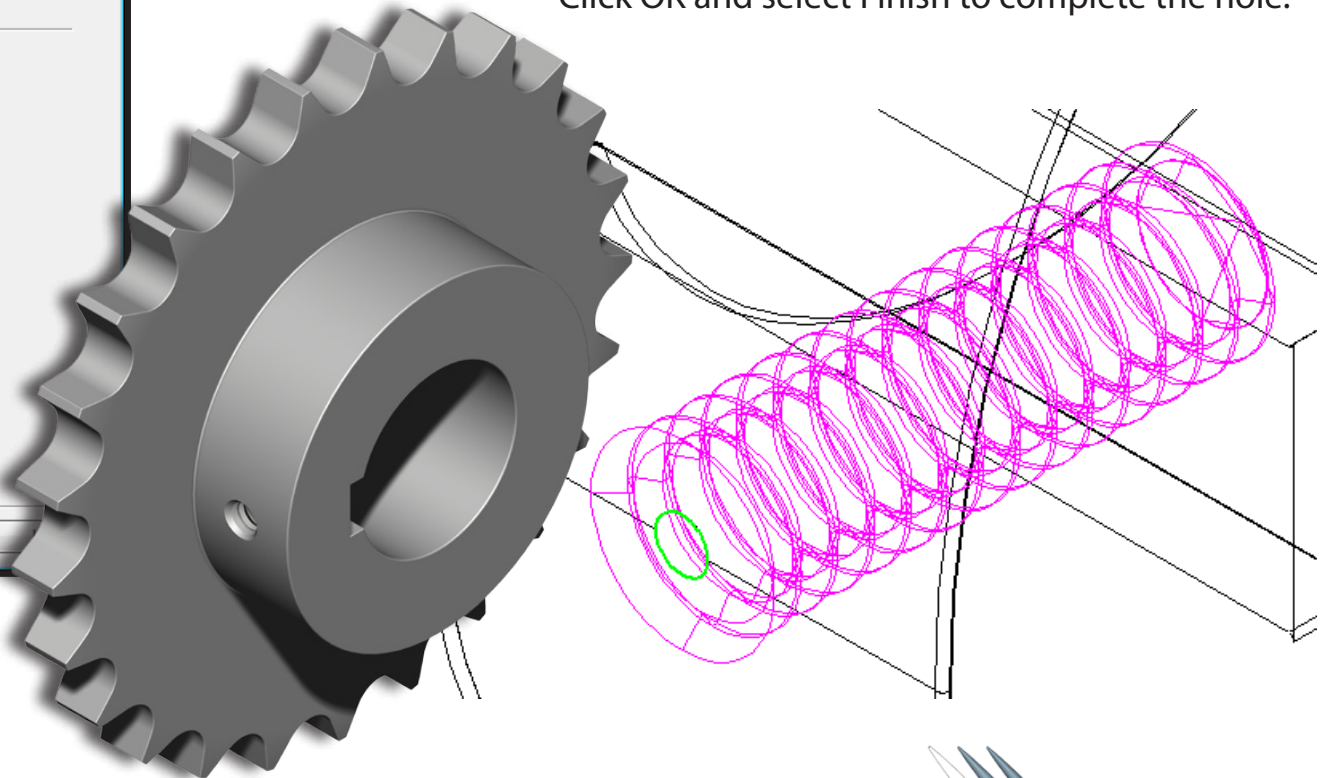
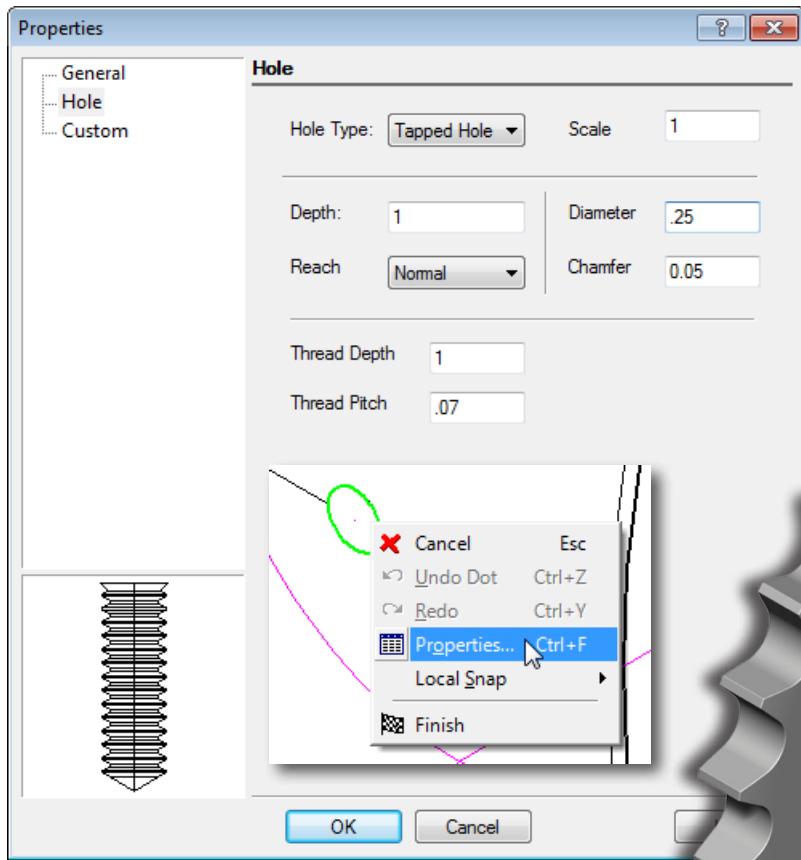
The Add Holes tool is selected from the 3D Object toolbar.

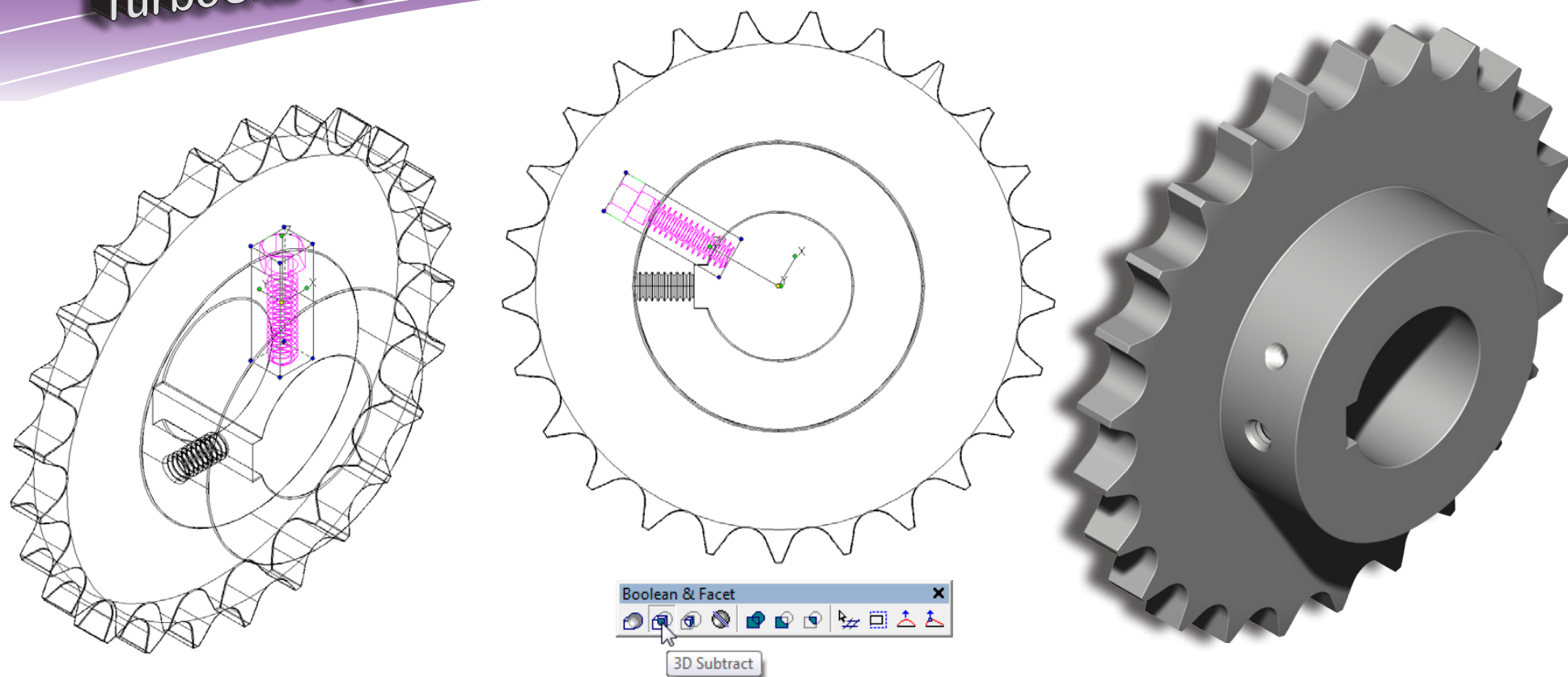


The flat plane is selected and then the dot.

The Properties dialogue is then opened and the desired settings established.

Click OK and select Finish to complete the hole.



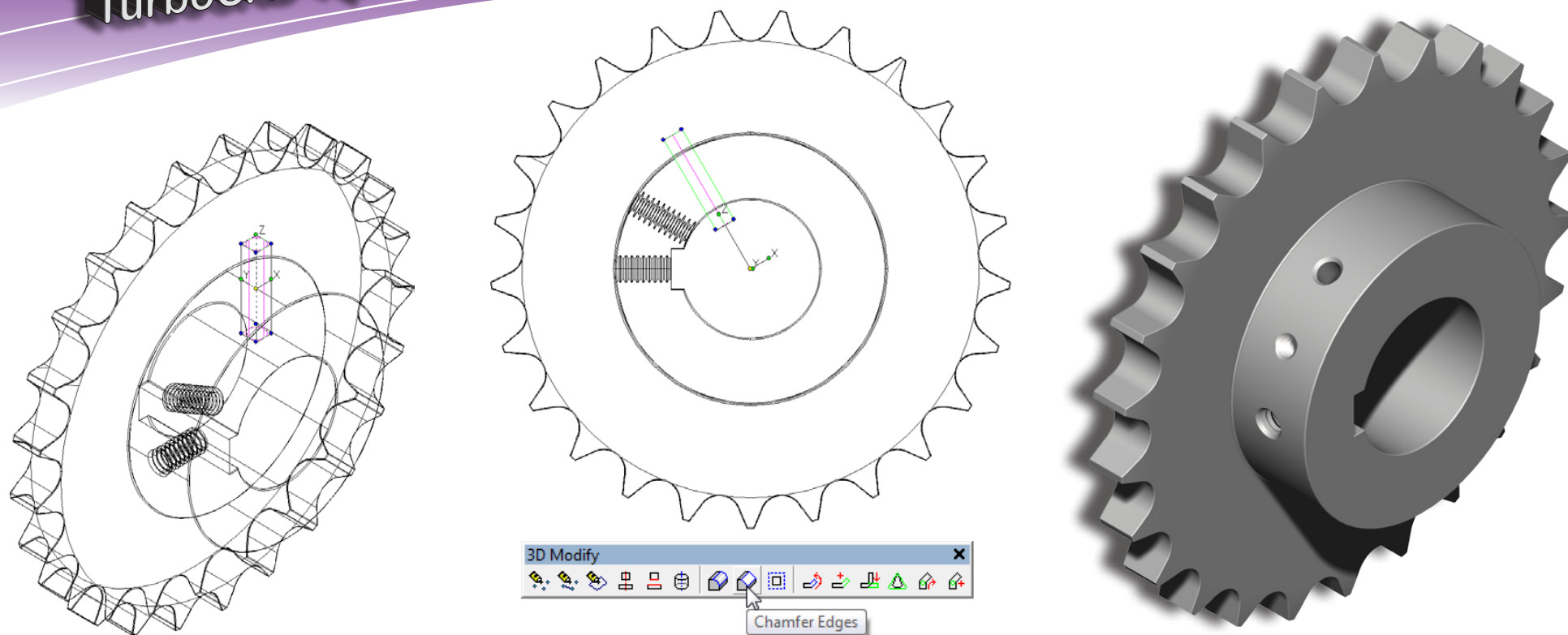


Subtracting a bolt from the curved surface is probably the least appealing method as the user cannot chamfer the entry area to clean it up.

A bolt is created with the Thread3D tool from the Special Tools toolbar.

Once the bolt is created it can be snapped to the top quadrant point and moved up or down to ensure that the threads clear the desired thickness. A user can also relocate the bolt's reference point so that the bolt can be rotated to the desired angle.

The bolt is then subtracted from the curved surface to create the final tapped hole.



Subtracting a simple cylinder can provide adequate results, even for a tapped hole. Tapped holes are frequently illustrated without threads in all manner of CAD drawings as long as details are provided.

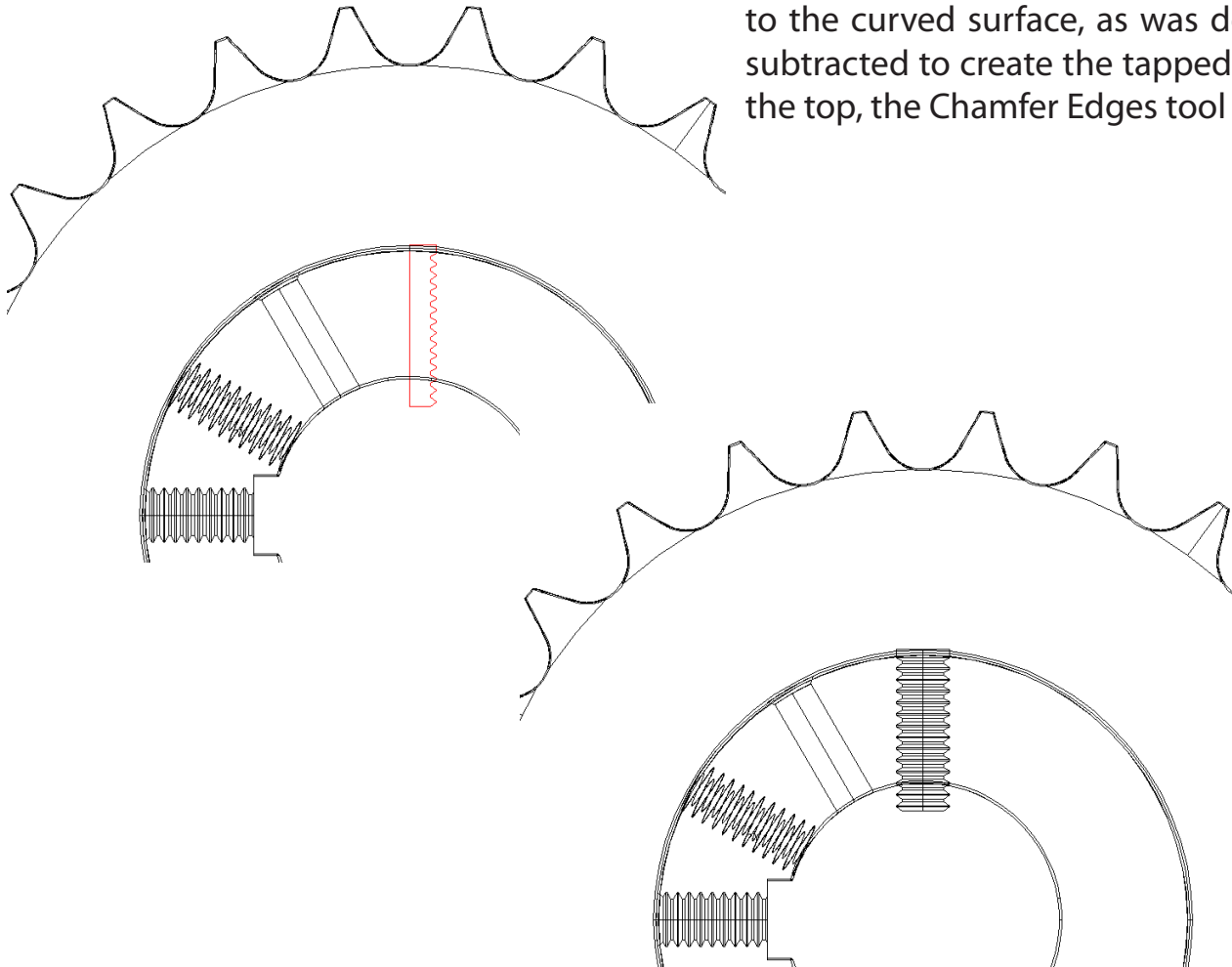
A simple cylinder is created with the Cylinder tool from the 3D Objects toolbar.

Once the cylinder is created it can be snapped to a quadrant point and moved up or down to ensure it clears the desired thickness. As with the bolt, the user can also relocate the cylinder's reference point so that it can be rotated to the desired angle.

The cylinder is then subtracted from the curved surface to create the hole.

A user can then use the Chamfer Edges tool to clean up at the entry.

Like the last two methods, the fourth requires creating a new object that gets subtracted from the curved surface. In this case a mock 2D thread profile is created and then revolved into a 3D object. The new object is then snapped to the curved surface, as was done with the bolt and the cylinder, and then subtracted to create the tapped hole. Providing the user has left a good lip at the top, the Chamfer Edges tool can be used to clean up the opening.



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