Adjustable Wrench

[1]

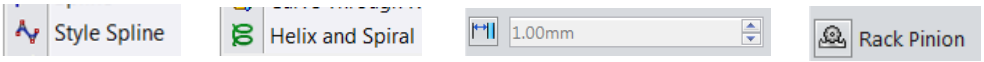
Introduction:

This lesson will focus on using the **Style Spline** tool, **Helix and Spiral** command and the **Rack Pinion** mate in assemblies to model an adjustable wrench.
https://www.youtube.com/watch?v=7Qn6oOAAsow

Learning Intentions:

This participants will be able to:

- Utilise the **style spline** tool to full define the spline outline of the wrench head.
- Create a helical screw thread using the **helix / spiral** and **swept boss/base** commands.
- Understand how to mate mechanical parts using the **distance** mate and **rack pinion** mate commands.



Prerequisite knowledge:

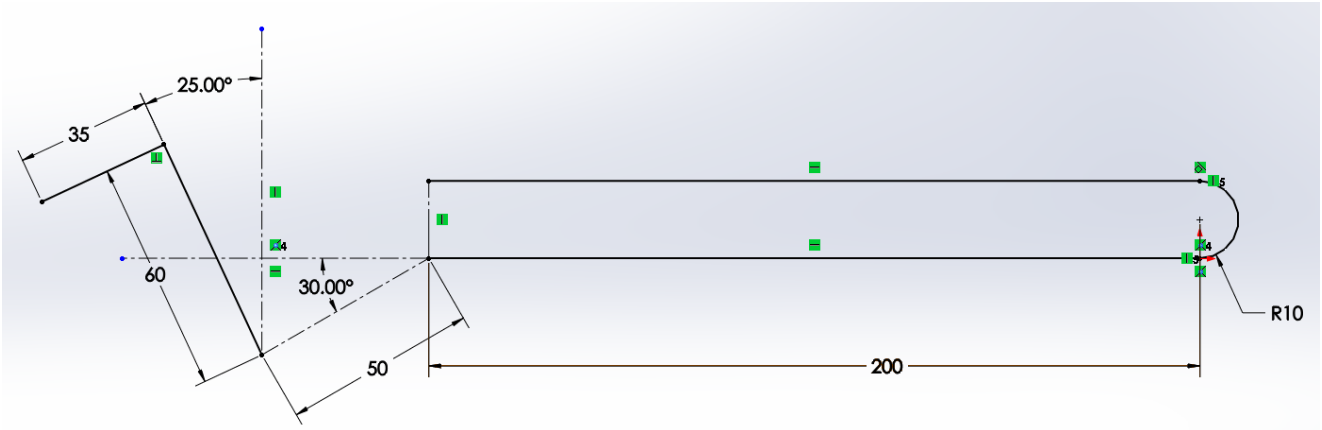
To complete this exercise you should have a working knowledge of SolidWorks 2009 and a previous knowledge of the following commands are required in this lesson: **sketching (spline, dimensioning), Extruded Boss/Base, Extrude Cut, Fillet, Adding Appearances.**

[1] http://autonixservice.ro/service

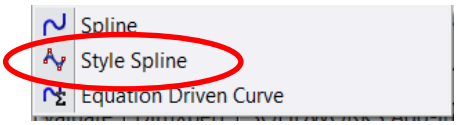
Wrench Handle

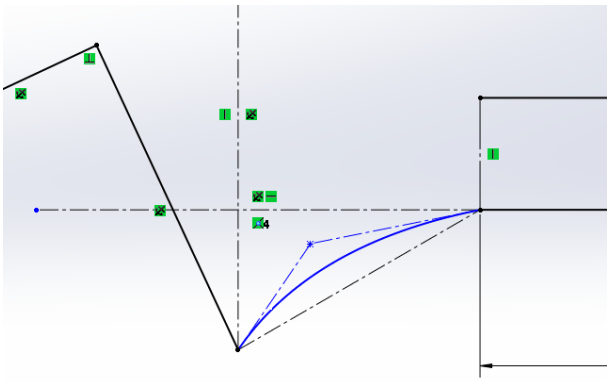
Handle outline

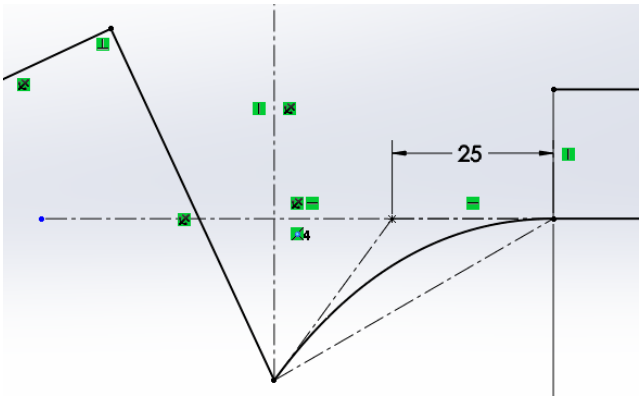
Create a sketch on the **Top Plane** and sketch the below sketch for the wrench handle outline.



Select **Style Spline** sketch tool and draw the style Spline shown using 3 points.

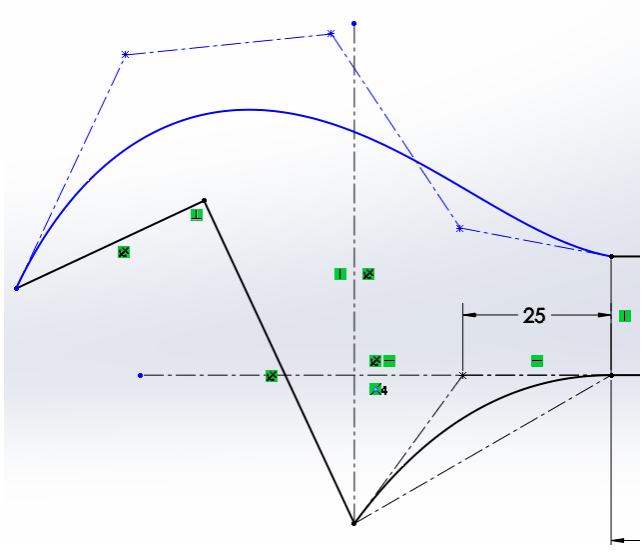




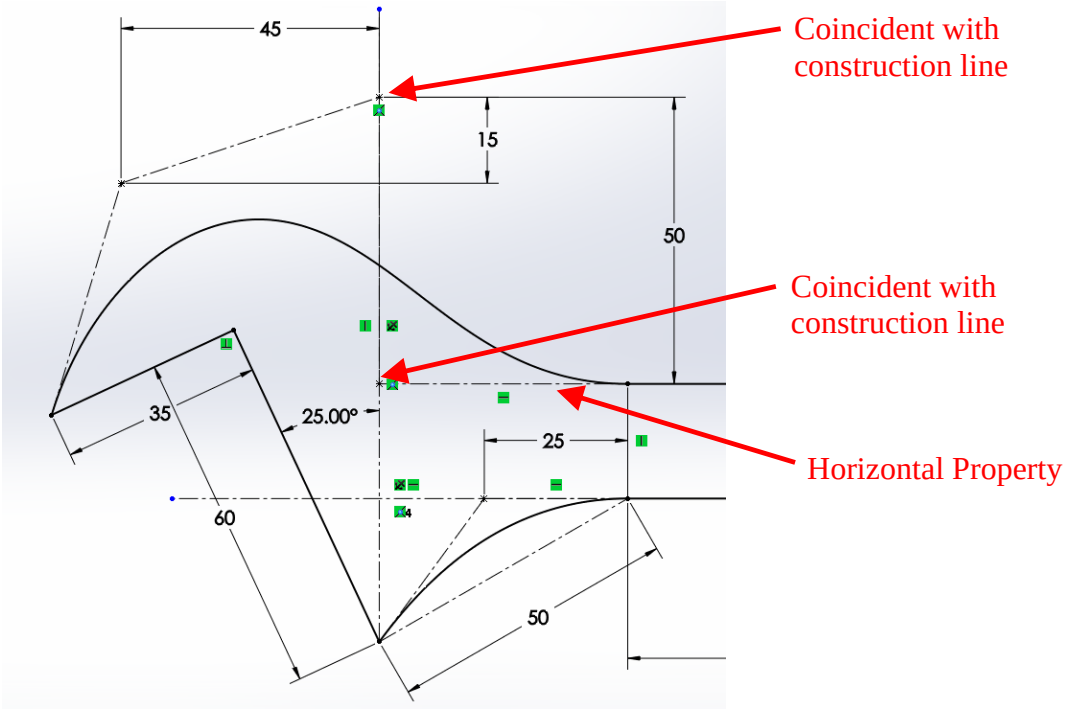


Add a **horizontal** property to the **inference line** of the spline and dimension the vertex as shown.

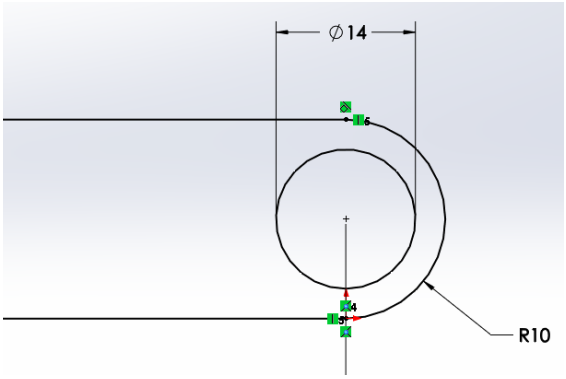
Create the **style spline** shown to complete the head of the wrench.

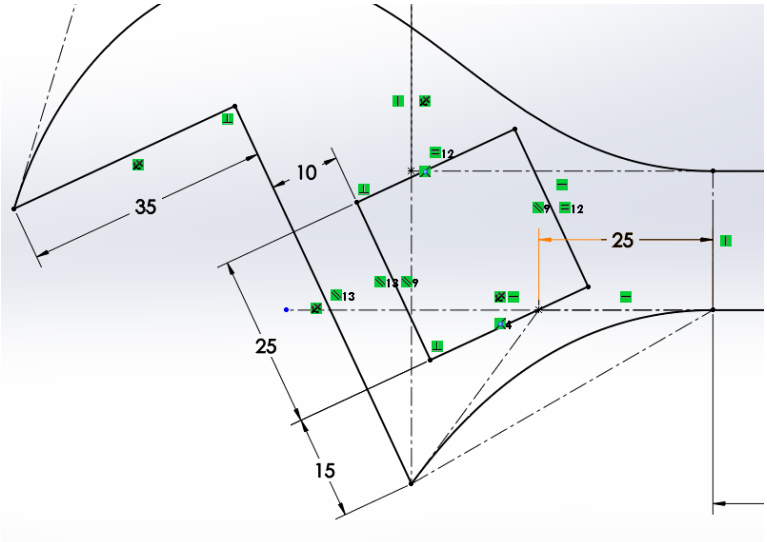


Add the properties and dimensions shown to the style spline inference line and vertices.



Draw a concentric circle at the end of the handle.

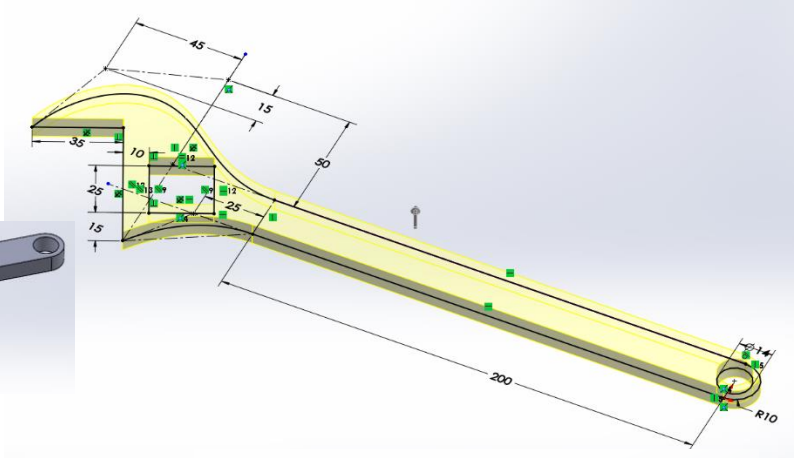


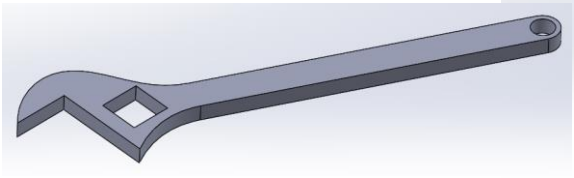


Sketch the square shown using the **3 point corner rectangle tool.**

Add the parallel properties and dimensions shown.

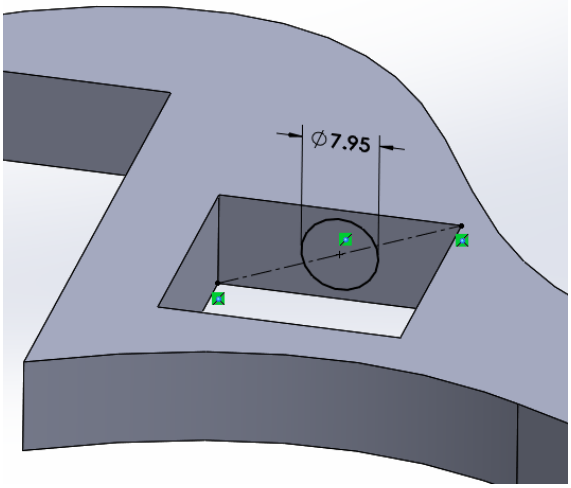
Extrude Mid-Plane **10mm**.

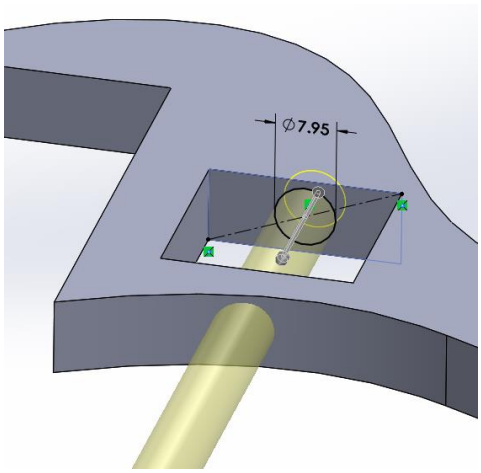




Remove hole

Sketch the circle shown on the inner surface of the adjustment position.

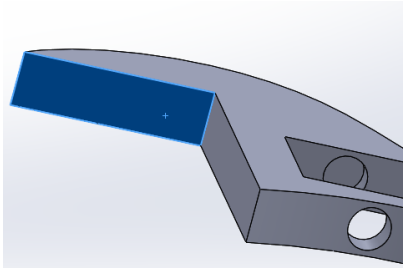


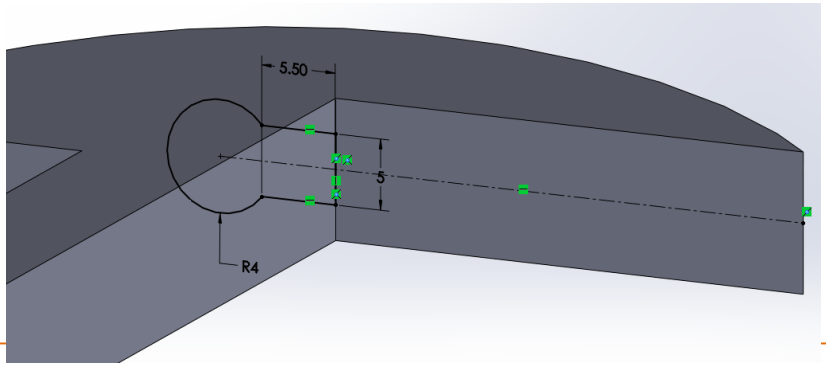


Extrude cut in direction 1 - **5mm** and direction 2 **Through All**.

Slider Slot

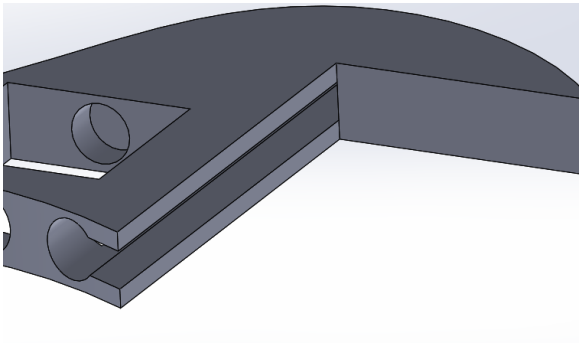
Create a sketch on the inner surface of the wrench jaw.

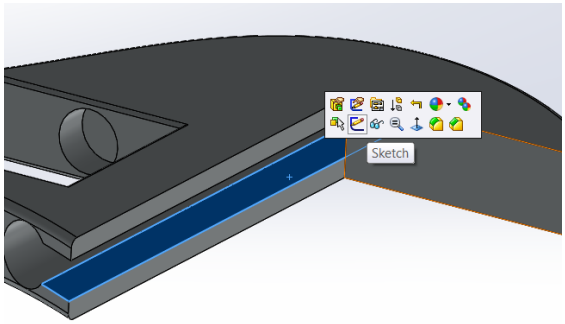




Sketch the slot design shown.

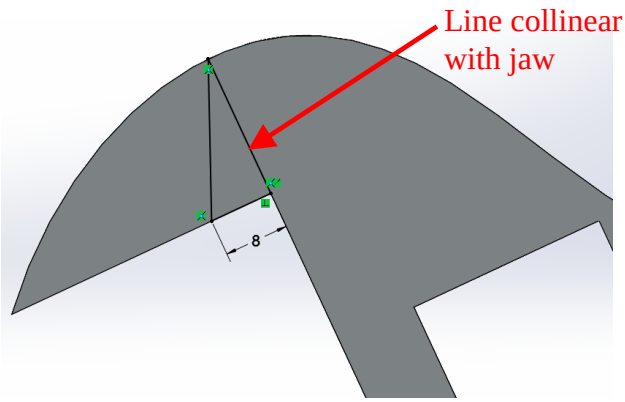
Extrude Cut the slot design **Through All** in both directions.





Complete the slot for the wrench jaw by creating a sketch on the inner surface.

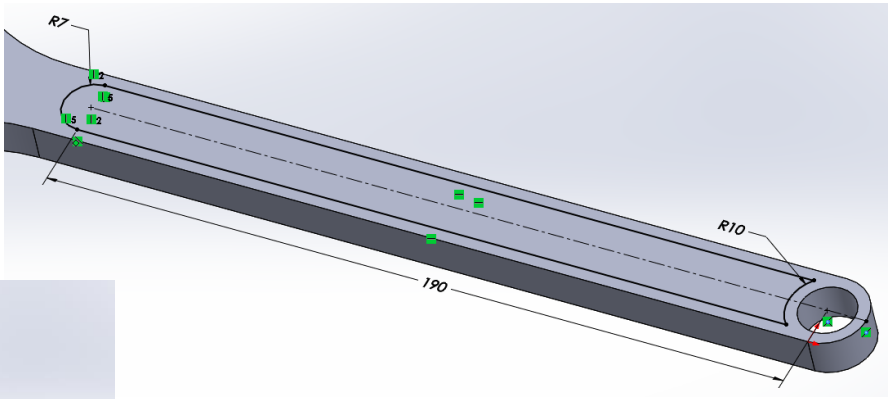
Create the triangular sketch show.

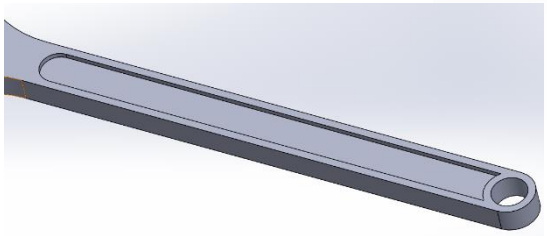


Extrude Cut this sketch to the same width as the slot – **5mm**.

Handle Design

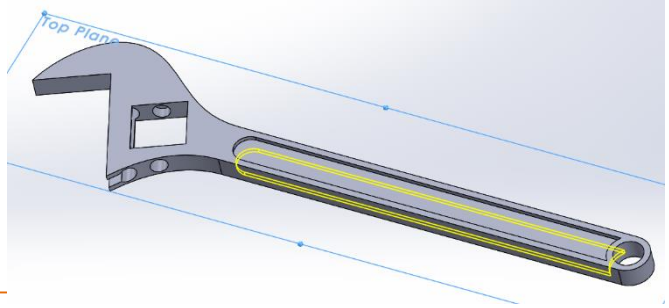
Sketch the design onto the top surface of the wrench



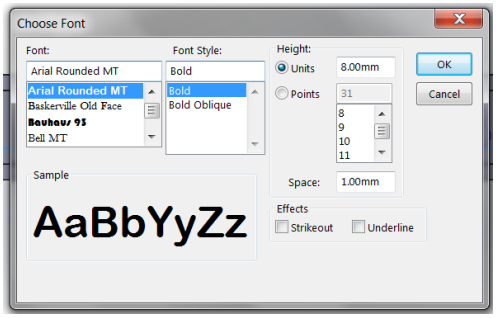


Extrude Cut **2mm**.

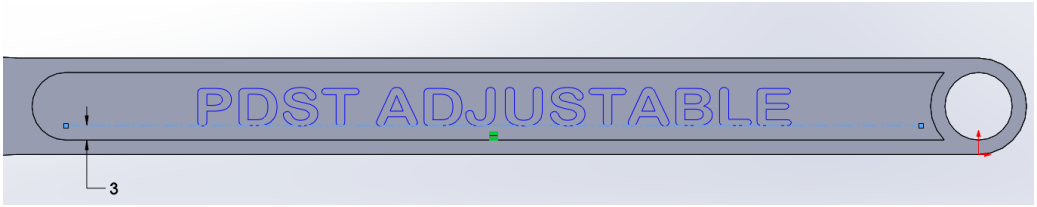
Mirror the design cut in the **Top Plane**.



Add the **text** shown to the handle

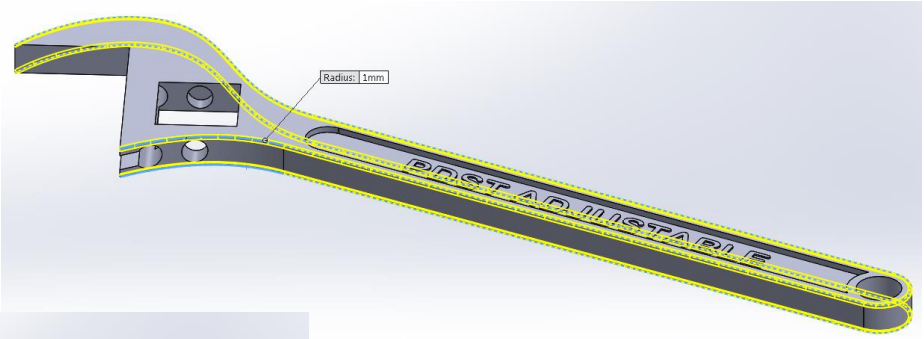


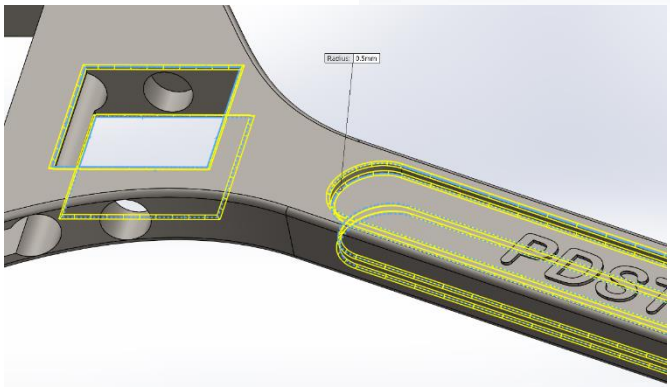
Extrude the text **0.5mm**



Fillets

Add **1mm** fillet





Add **0.5mm** fillet to inner edges of handle

Materials and Appearances

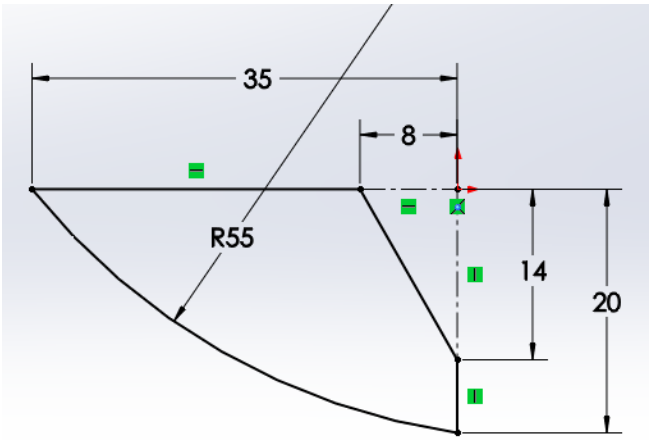
Apply **Tool Steel** as the material

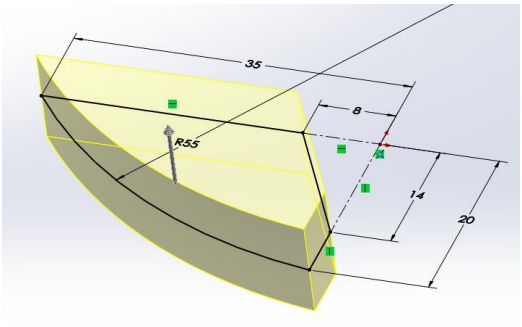
Apply a **Chromium Plate** as the appearance to the part.

Wrench Jaw

Jaw Outline

Create the sketch over on the **Top Plane**

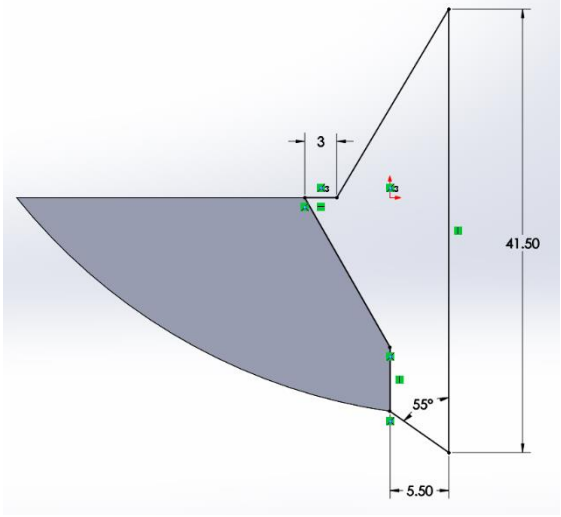


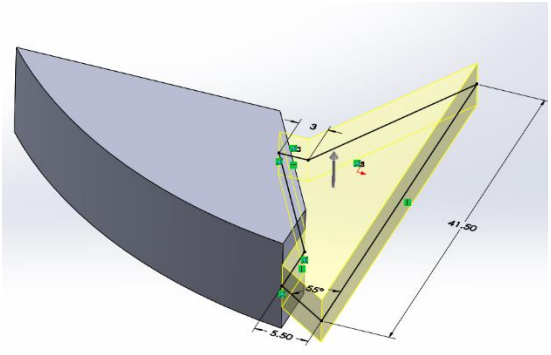


Extrude the sketch **Mid-Plane 10mm**.

Slider Mechanism

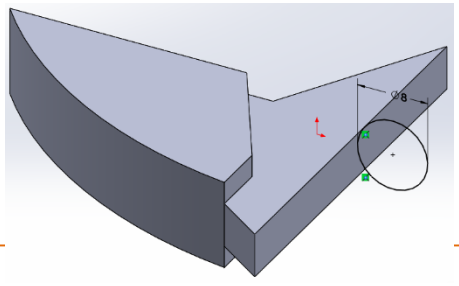
Sketch the outline of the wrench slider mechanism on the **Top Plane**.

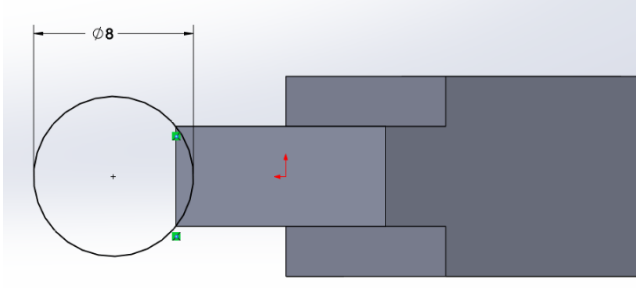




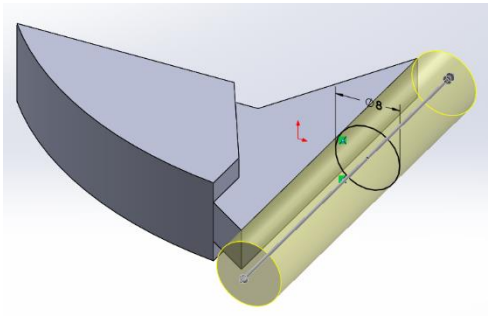
Extrude **5mm** Mid-Plane

Sketch a circle on the **Front Plane**

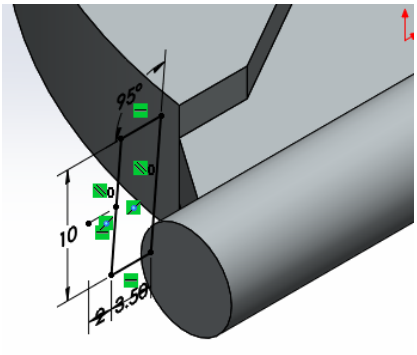


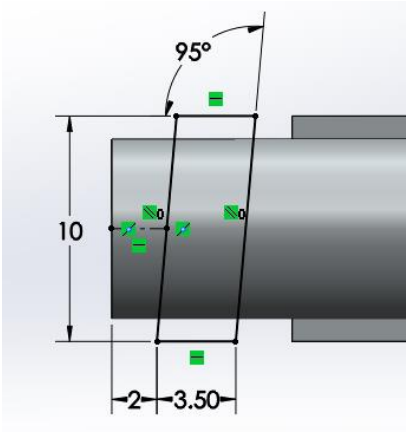


Extrude: Direction 1 - **28mm** and Direction 2 – **18mm**



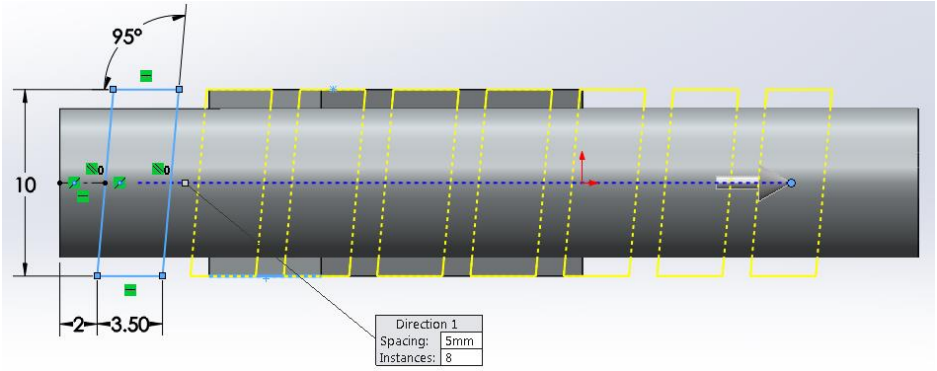
Sketch the thread cut onto the **Right Plane**



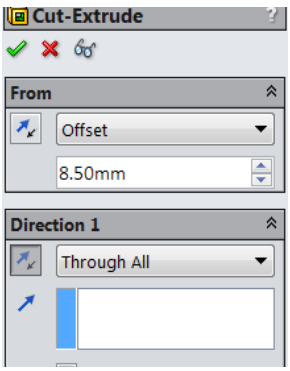


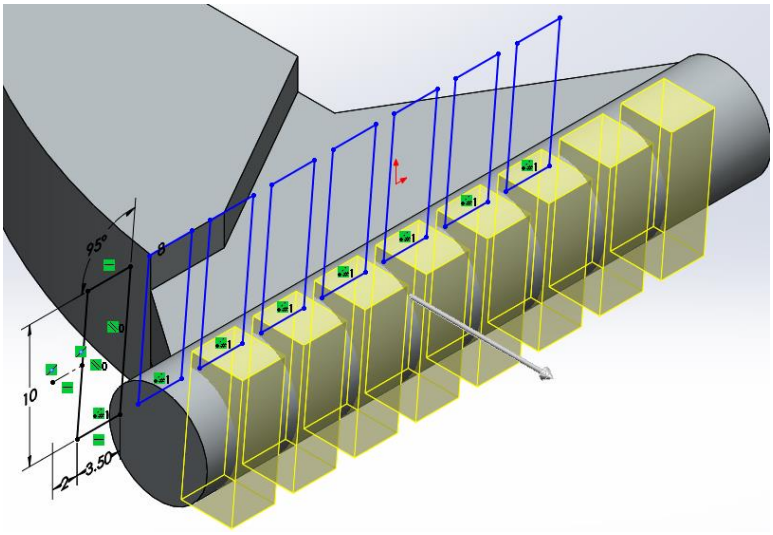
Linear Sketch Pattern to complete the thread cut out.

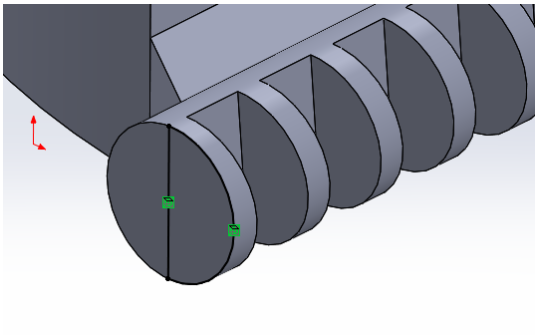
Spacing 5mm



Extrude Cut using the below parameters.



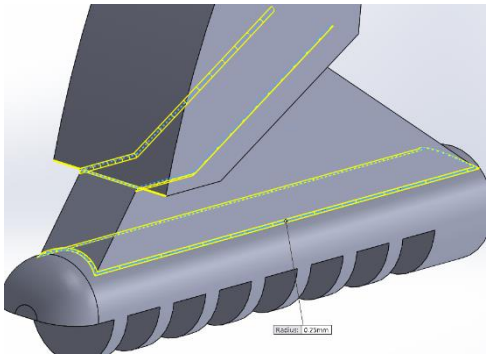




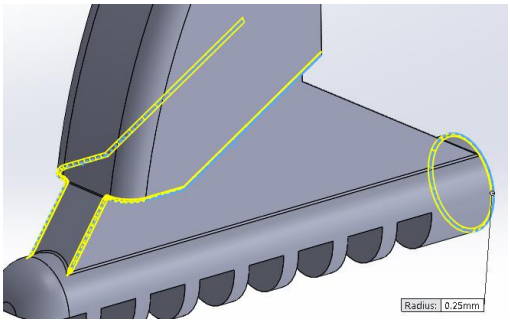
Remove the front thread.

Apply Fillets

3mm Fillet on front edge



2mm Fillet



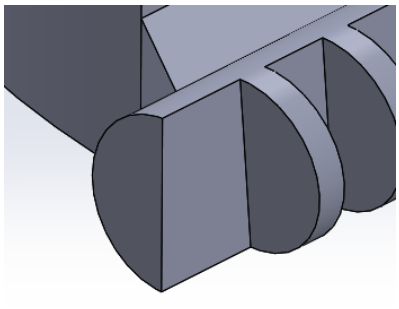
Materials and Appearances

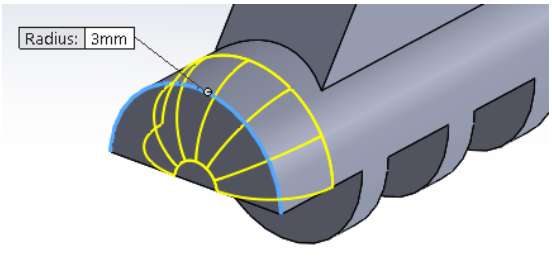
Apply **Tool Steel** as the material

Apply a **Chromium Plate** as

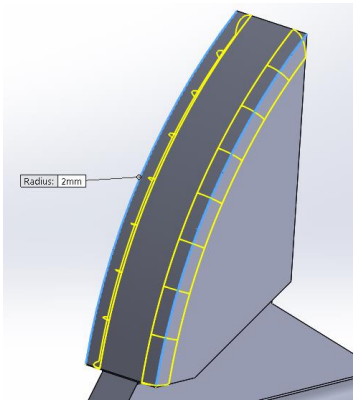
Sketch on the front surface of the cyclinder.

Convert Entities to create the sketch shown.

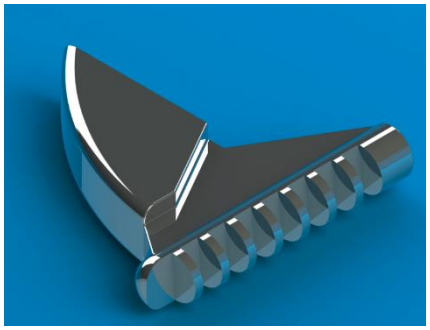




0.25mm Fillet



0.25mm Fillet

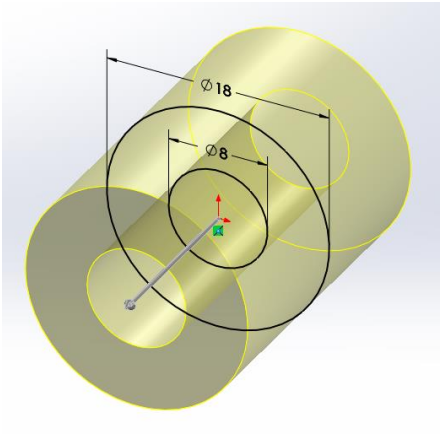


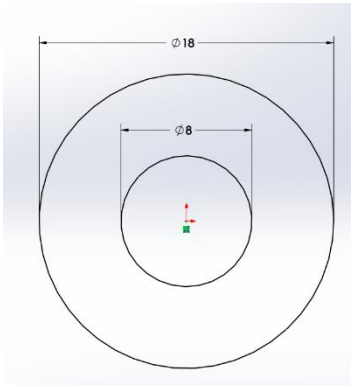
the appearance to the part.

Worm Screw

Worm screw body

Create the sketch over on the **Front Plane**

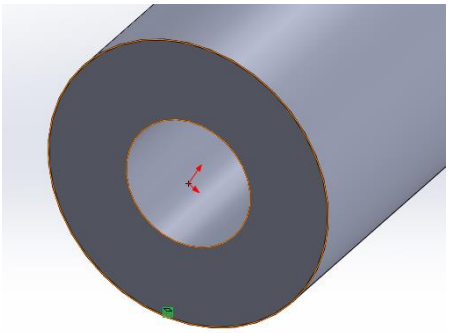


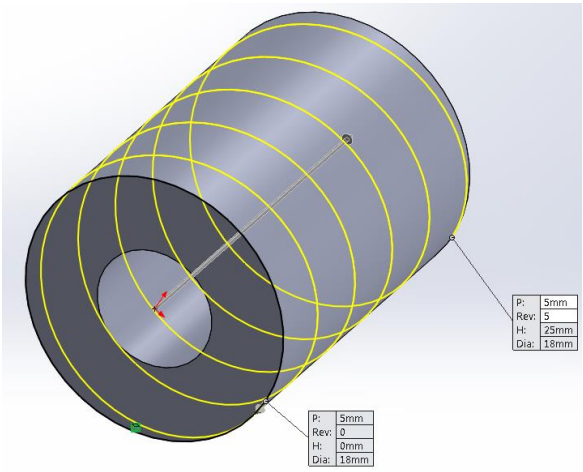


Extrude the sketch **Mid-Plane 25mm**.

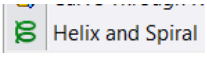
Screw Thread

Convert Entities on the outer front edge of the body.

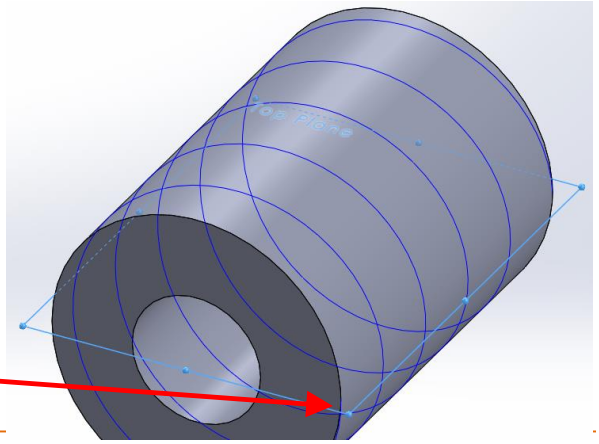




Create a **Helix** about this circle using these parameters.

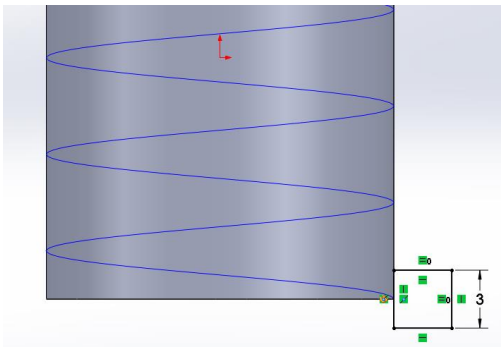


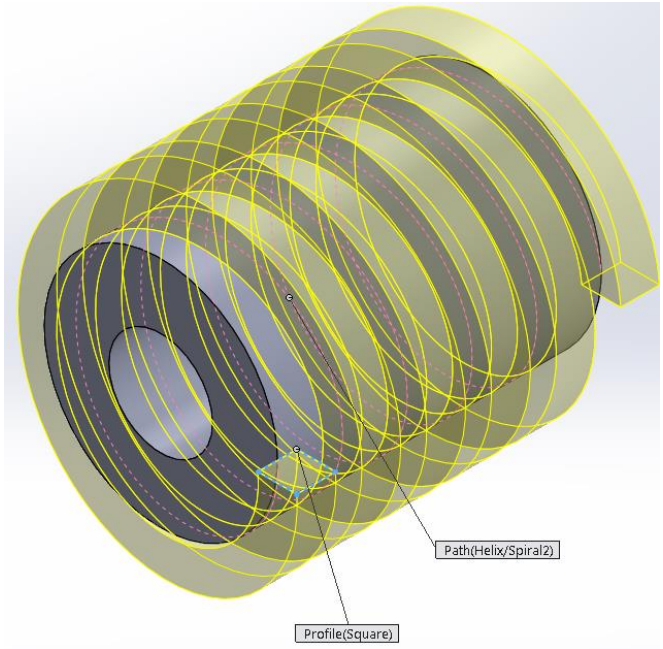
Create a sketch on the **Top Plane** for the thread profile. The helix start position is on the Top Plane.



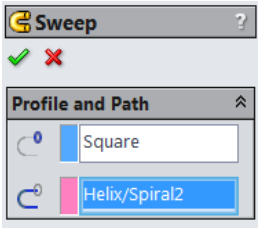
Helix start position

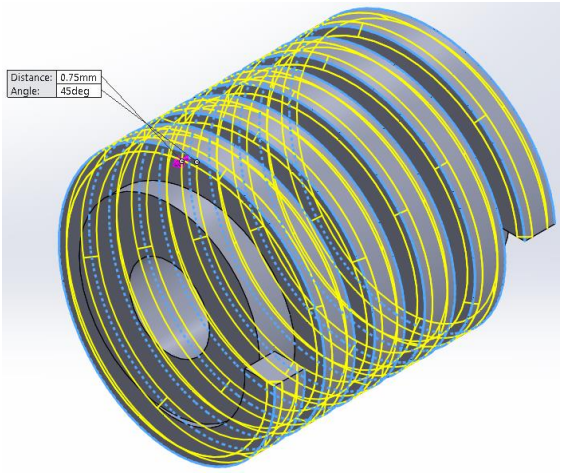
Sketch a square on the Top Plane with a side of **3mm**. Create a coincident property between the Midpoint of one side and the Helix.





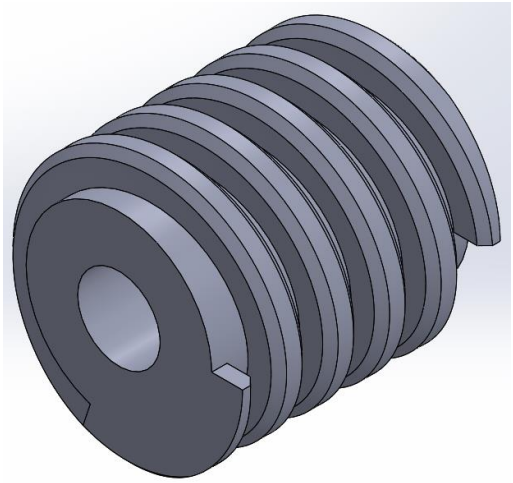
Select the **Swept Boss/Base** command. Using the square as **Profile** and the helix as the **Path**.





Add a **0.75mm** chamfer to the outer edges of the screw thread.

Trim off the excess thread on either end of the worm screw.



Materials and Appearances

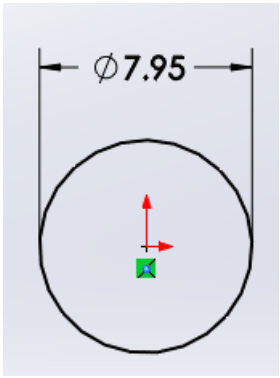
Apply **Tool Steel** as the material

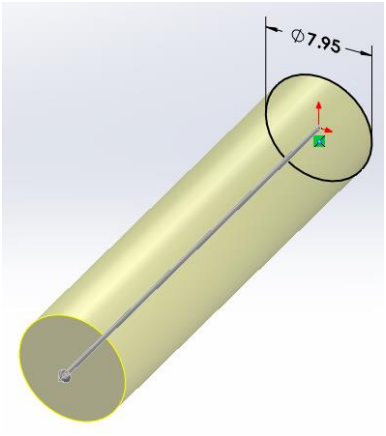
Apply a **Chromium Plate** as the appearance to the part.

Dowel

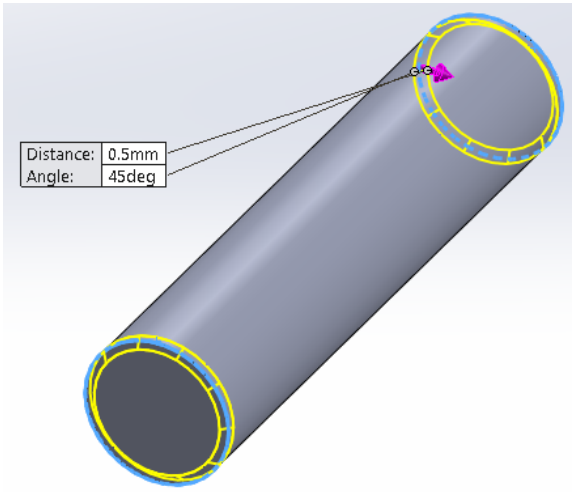
Extrude Boss/Base

Create the sketch over on the **Front Plane**





Extrude the circle **35mm**.



Add a **0.5mm** chamfer to either end of the dowel.

Materials and Appearances

Apply **Tool Steel** as the material

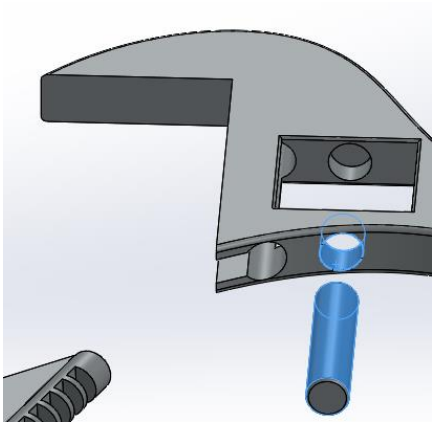
Apply a **Chromium Plate** as the appearance to the part.

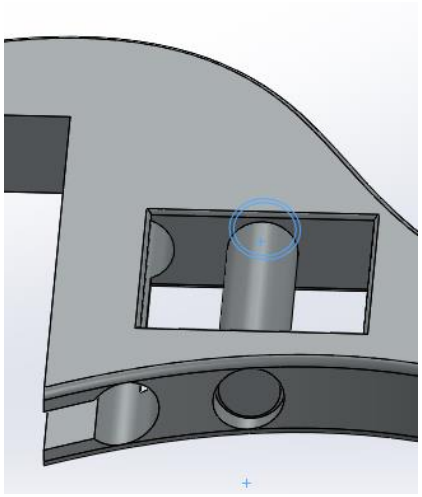
Assembly the Adjustable Wrench

Insert the adjustable wrench parts into a new **Assembly**.

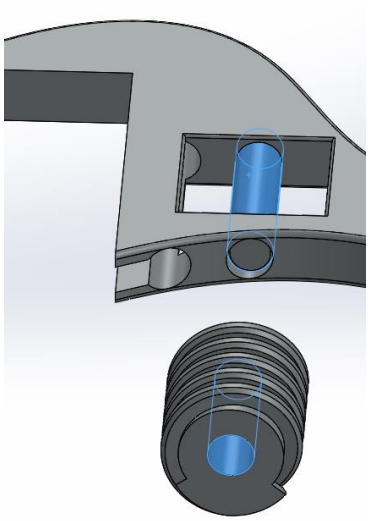
Mate the Dowel

Add a **concentric mate** between the dowel and the hole.



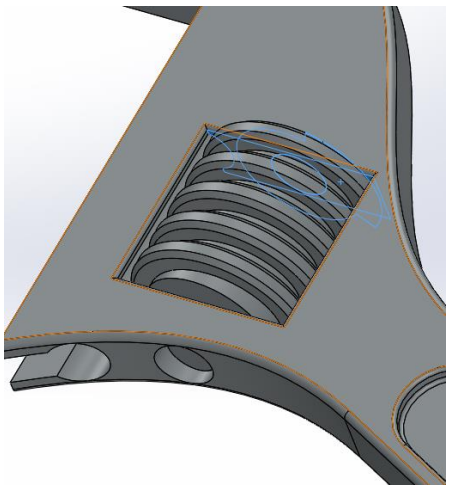


Coincident Mate between end of dowel and top of the hole.



Mate the Worm Screw

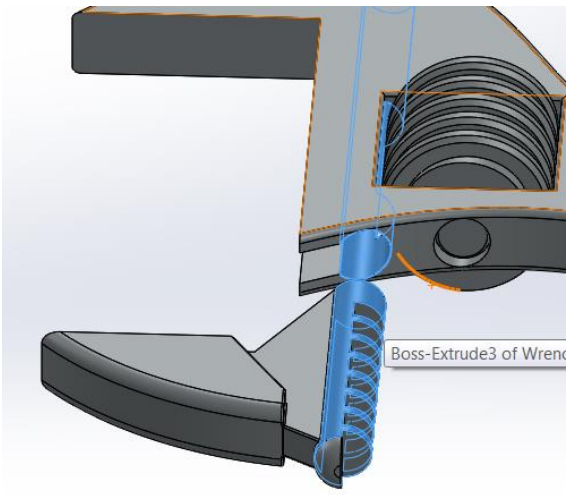
Concentric mate between dowel and worm screw.

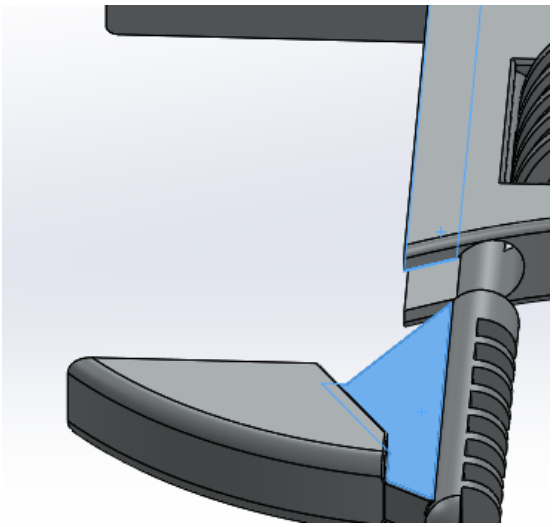


Coincident mate to place worm screw in position.

Mate the Jaw

Concentric mate between jaw and handle.

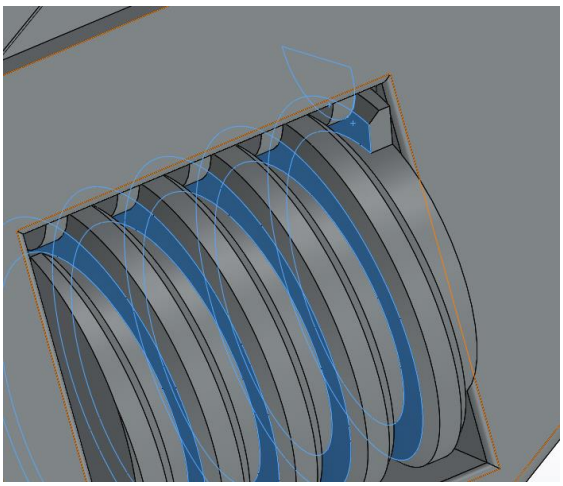




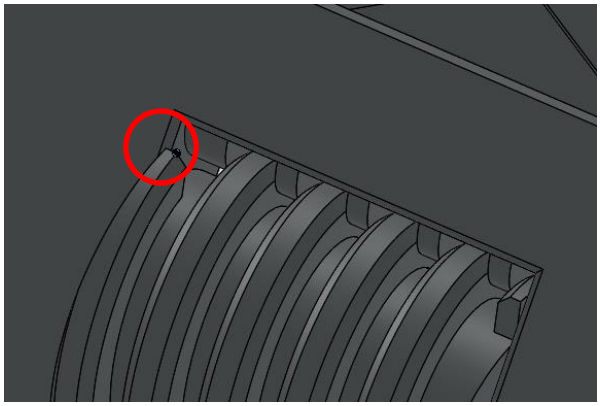
Coincident mate between the surfaces

Mate the screw thread

Add a **tangent mate** between the inner surface of the screw thread helix and the side of the rack (choose the inner surface on the top slot of the rack).

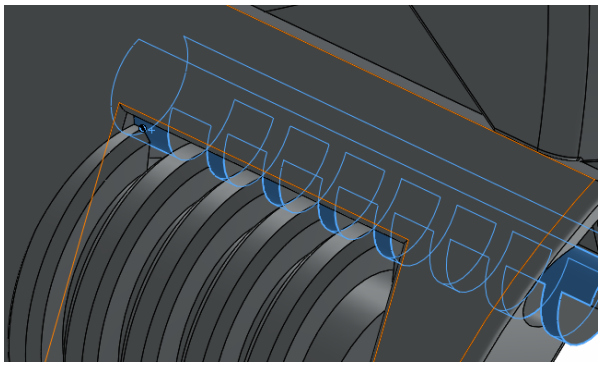


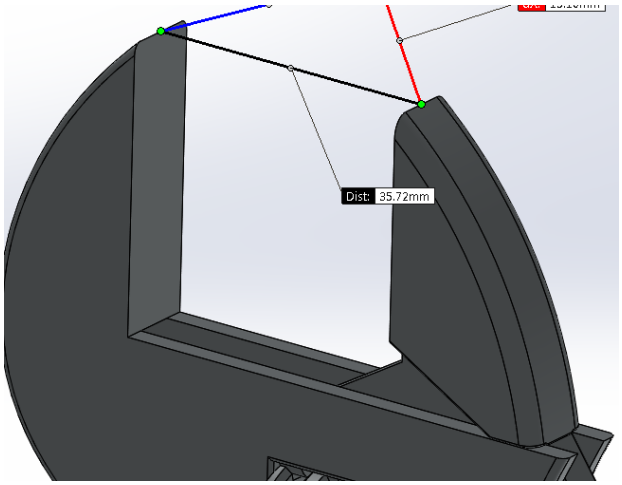
Note: The rack and pinion mechanism can now function in the solidworks assembly but the fully closed and fully opened positions must be established.



Create the fully open position for the jaw. Mate the end vertex of the screw thread.

Mate this vertex to the outer surface of the cylindrical rack. Thereby, the screw thread will stop once it meets the outer cylindrical surface.

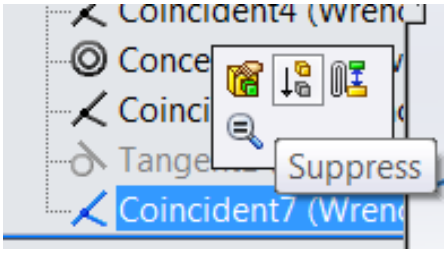




Use the **Measure Tool** to measure the distance between the wrench jaws in the fully open position.

Distance = 35.72mm

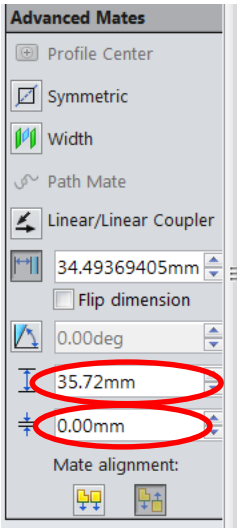
Suppress the 2 previous mates in the **Mates** folder.

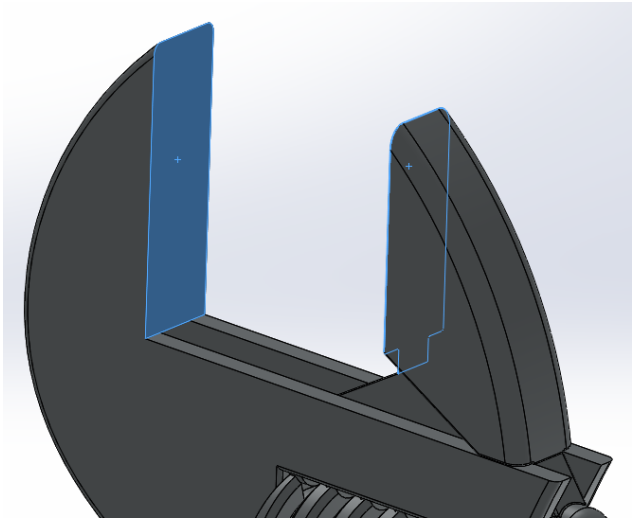


Note: The previous mates were used to establish the open position.

Distance Mate

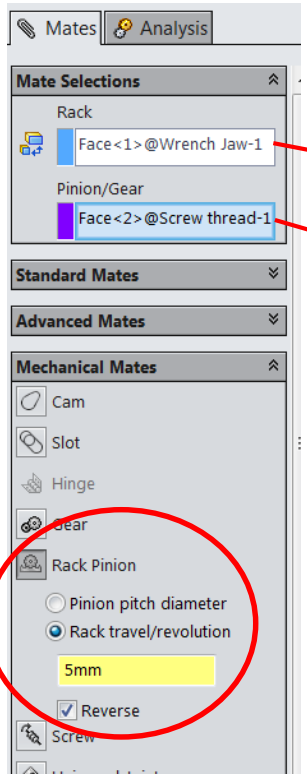
Add a **distance** mate between the wrench jaws using the below parameters.

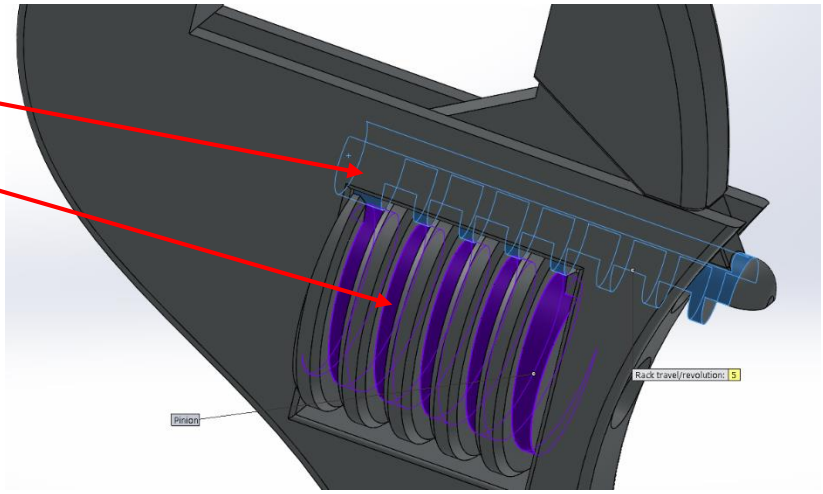




Rack Pinion Mate

Create a rack pinion mate between the screw and rack using the below parameters.





Note: The screw thread can now be turned to adjust the wrench jaws.

Complete