Sash Clamp

[1]

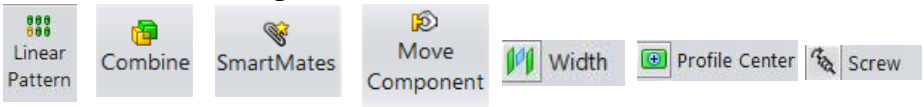
Introduction:

The Sash clamp consists of nine parts. In creating the clamp we will be looking at the improvements made by SolidWorks in linear patterns, adding threads and in assembling the parts.
https://youtu.be/D8n63JswyX8

Learning Intentions:

This lesson will focus on the various ways of using **Linear Pattern,** improvements in **creating threads** by using the **Combine** command, and improvements in assembling the parts e.g. using **Temporary Fix** to give greater control when assembling.

It will also look at improved ways of mating parts e.g. using **ALT** and drag, **Profile Centre, SmartMate** and **Screw Mate.**



Prerequisite knowledge:

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

[1] https://www.flickr.com/photos/toolstop/4860352044/

Part 1 BAR

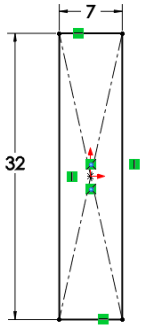
New Part

Start by creating **a New Part** and saving this part as "**Bar**".

> **Note:** It will become apparent later the importance of saving this part at these initial stages.

On the **Right Plane** draw the centre point rectangle to the given dimensions.

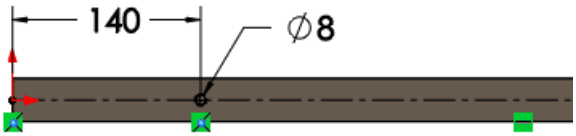
Extrude by 760mm.



On the **Front Plane** draw centreline as shown.

Then draw a circle having a diameter of 8mm and a distance of 140mm in from the end.

Use **Extrude Cut** to drill the hole.

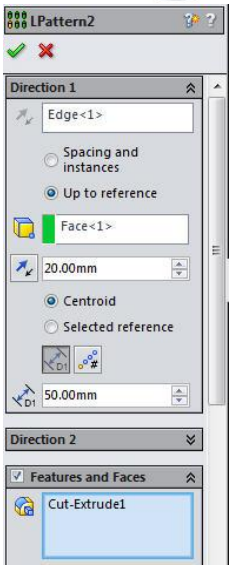


Linear pattern is used to add the additional holes in the bar.

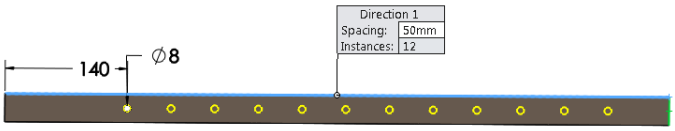
Note: Improvements have been made to **Linear Pattern**.

In adding the additional holes an offset distance can be set from the other end of the bar, so that if the bar is lengthened later this distance is maintained and the number of holes will change to maintain equal spacing between each hole.

Alternatively if the number of holes are important when the bar`s length is changed the spacing between the holes will automatically change to maintain the equal spacing.



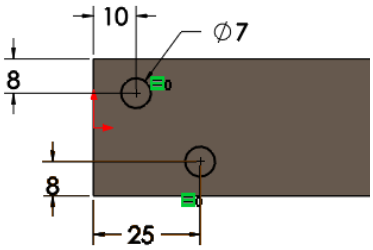
For this exercise the spacing between the holes is important so the distance button is pressed and the spacing is set to **50mm**.



Select the end face of the bar and set the offset distance to **20mm**. Therefore the last hole will always be a minimum distance of 20mm from the end of the bar even if the bar length is changed later.

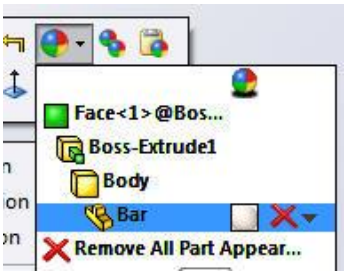
On the front face of the bar sketch two holes on the left hand side to the given dimensions. **Extrude Cut**, **Through All**.

The rivets will be positioned through these holes later.



Appearance

Add a brushed steel appearance to the **Bar**

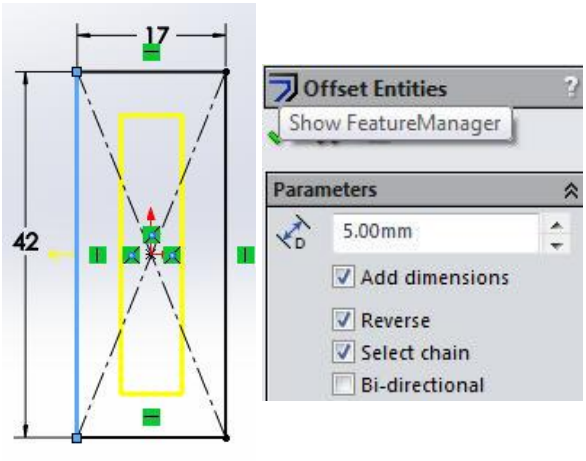


Save

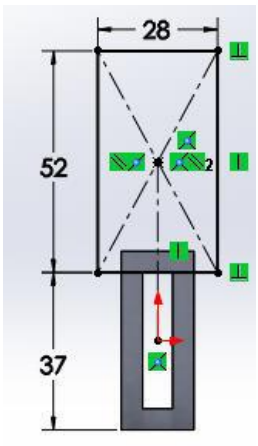
Slider 1

On the right plane, using **Centre Rectangle** draw the rectangle to the dimensions shown and offset by 5mm thickness as shown.

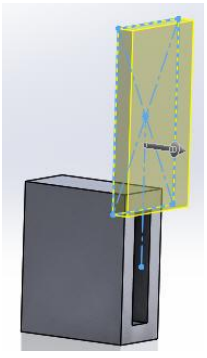
Extrude by 34 mm.



On the face shown draw another rectangle to the following dimensions.

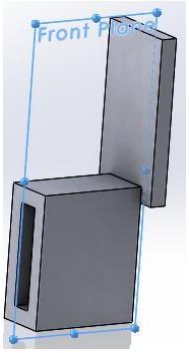


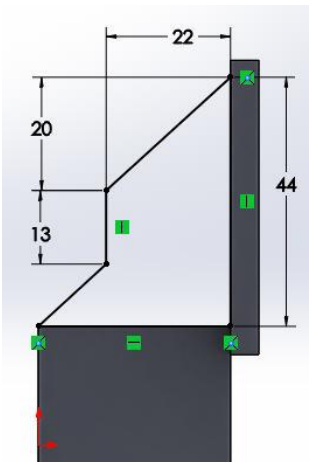
Extrude by 5mm.



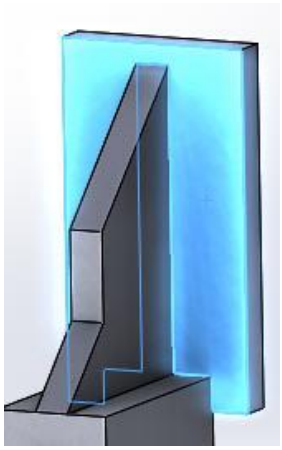
On the **Front Plane** draw the following sketch.

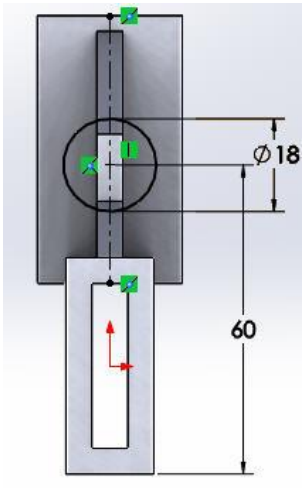
Extrude using **MidPlane** by **5mm.**

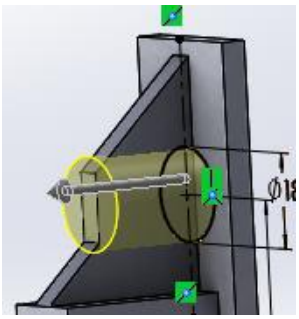




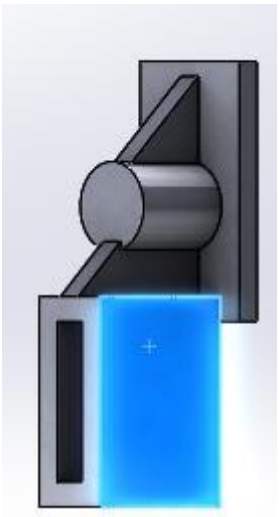
On the back face shown draw a circle diameter **18mm** and extrude it by **22mm**.

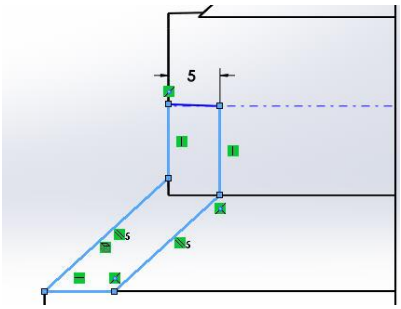




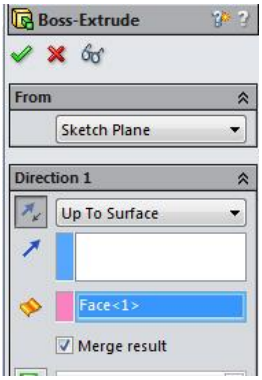


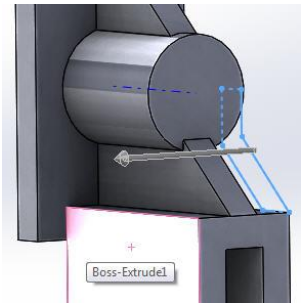
On the face shown draw another sketch. Activate the temporary axis of the cylinder. Then using convert entities and line command draw the shape shown in blue making the sloping lines parallel.

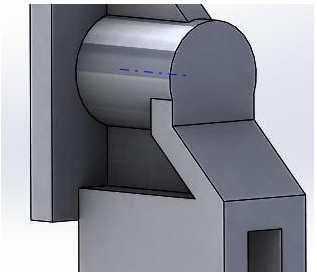




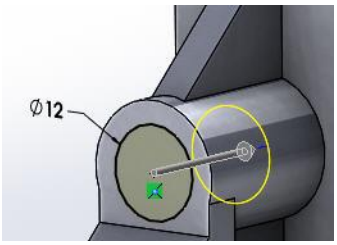
Extrude up to surface as shown.



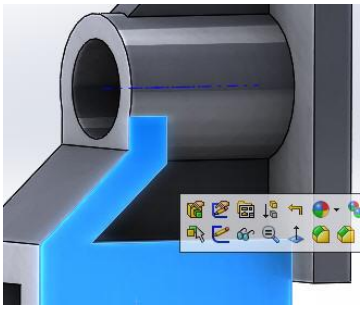


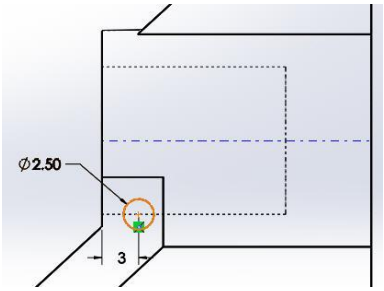


Next draw a circle on the face of the cylinder and **Extrude Cut** by **13mm.**

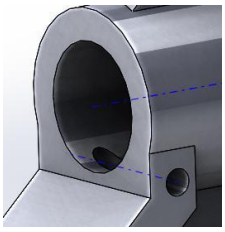


To accommodate the pin, sketch on the face shown. Change the display style to **hidden lines visible** mode. Draw a circle with centre point on the dotted line to the shown dimensions.

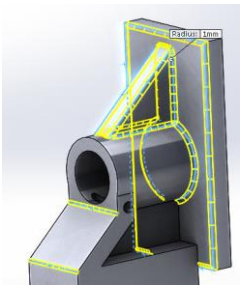


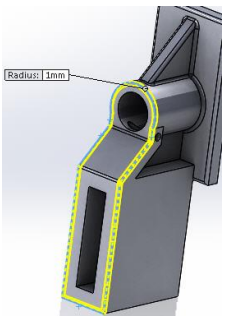


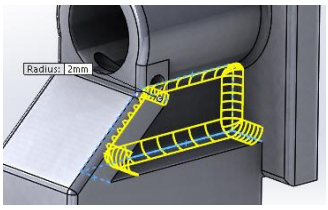
Extrude Cut, Through All.



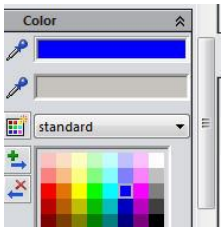
To complete the slider add a few fillets of **1mm** and **2mm** as shown.

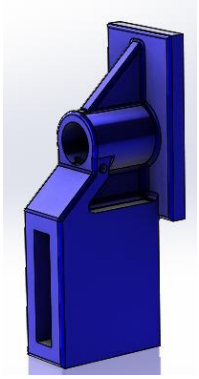






Appearance is **Brushed steel, Blue**



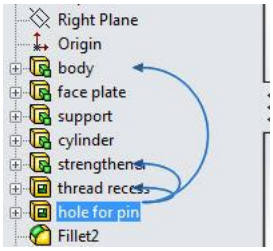


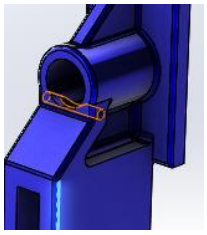
Save

Slider 2

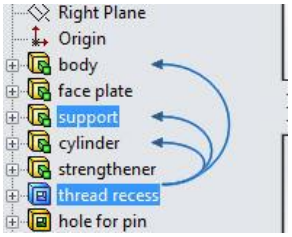
Open **Slider 1** and make the following modifications.

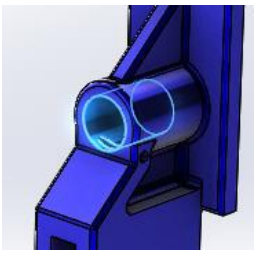
a) Delete hole for pin on the design tree and associated sketch.



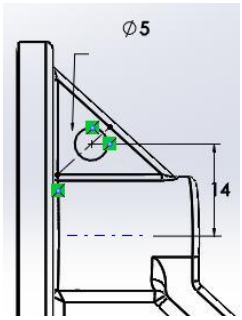


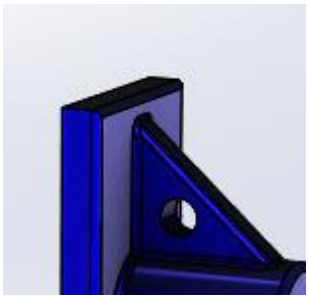
b) Delete thread recess and associated sketch.



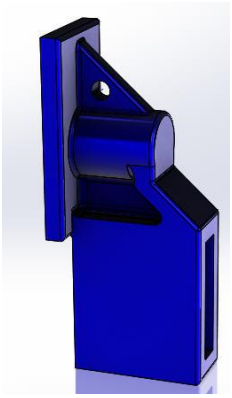


On the face shown draw the following sketch.



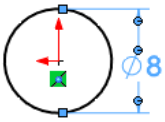


Save as **Slider 2.**

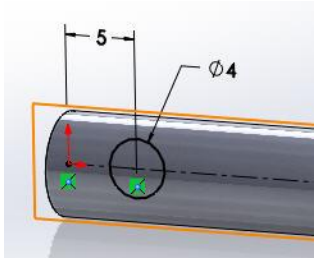


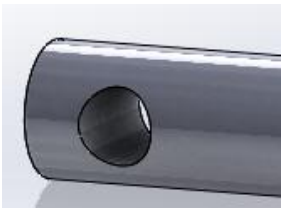
Peg 

Draw a circle on the right plane and extrude by **50mm**.

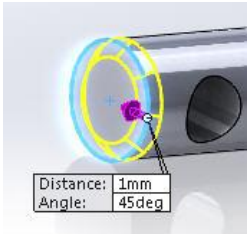


On the **Front plane** draw a circle on the centreline to the dimensions shown and **Extrude Cut** in both directions.

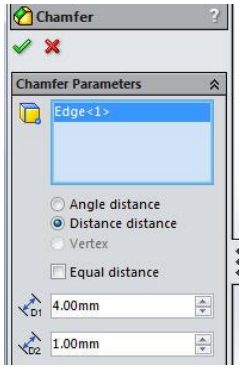


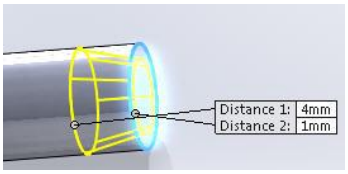


Add a **1mm** chamfer to the edge shown



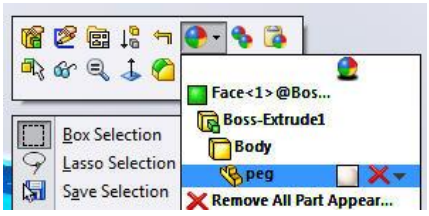
Using **Distance Distance Chamfer** add a chamfer to the other end as shown.

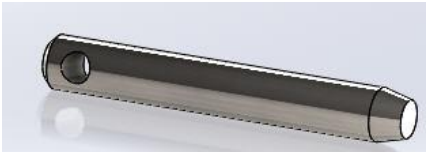




Appearance

Brushed Steel.



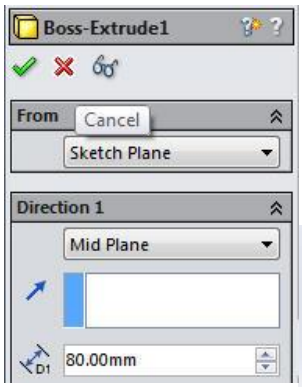


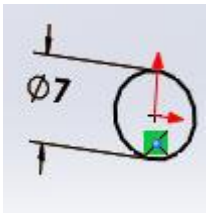
Save as Peg

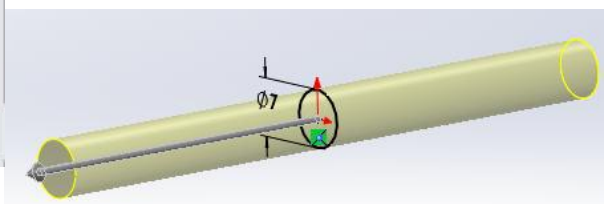
Tommy Bar

Draw a circle on the **Front Plane** having a diameter of **7 mm**.

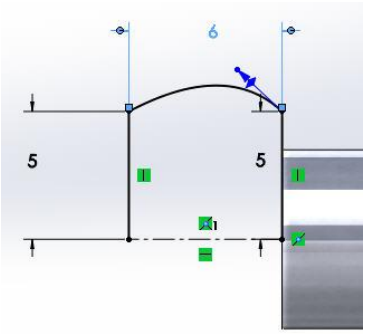
Extrude mid plane by **80mm**.

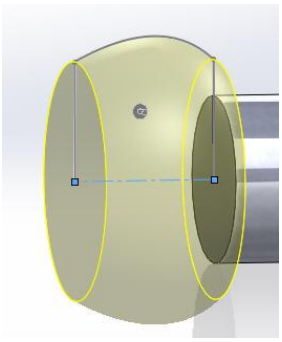




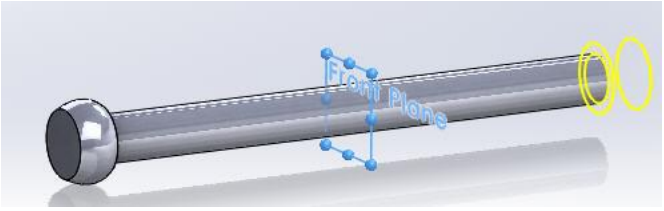


On the end of the bar the following sketch (spline to your own spec.) is drawn on the **Right Plane,** and revolved about the centreline.



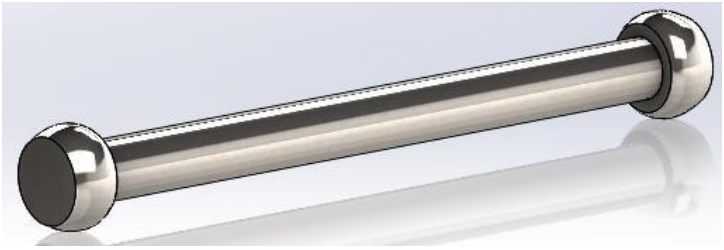


The feature is mirrored about the Front plane to finish the Tommy Bar.



Appearance

Add a brushed steel appearance to the tommy bar.

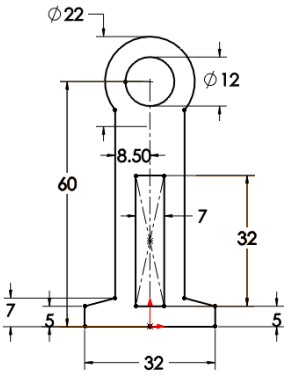


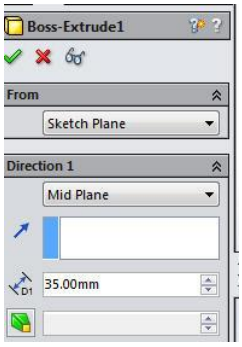
Save as Tommy bar

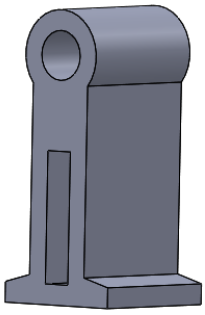
Head

On the **Right Plane** draw the sketch shown to the given dimensions.

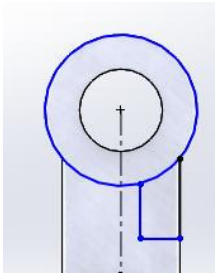
Extrude by **35mm** using midplane.





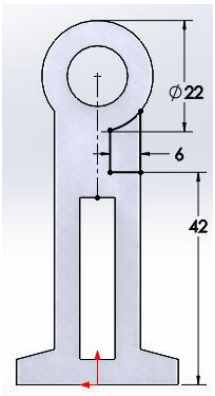


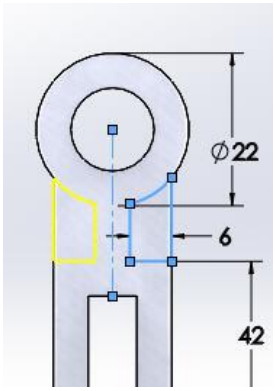
To create the recess a new sketch is drawn on the **Right Plane** as shown.



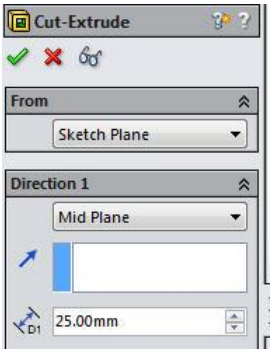
Draw the centreline and circle of **22mm** as shown. Complete the remainder of the sketch.

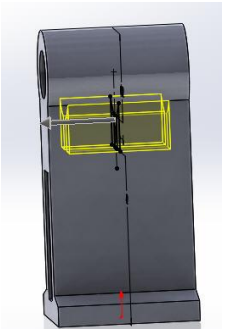
Use the trim command to get the portion of the circle that is required.



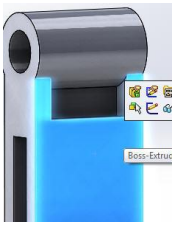


Mirror about the centreline and **Extrude Cut** using **Mid Plane** by **25mm**.

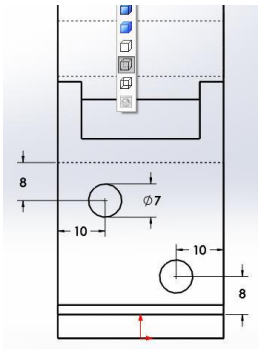




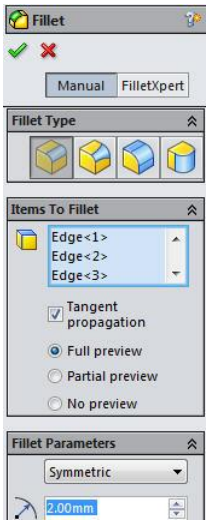
To add the holes for the rivets select the face shown for the sketch.

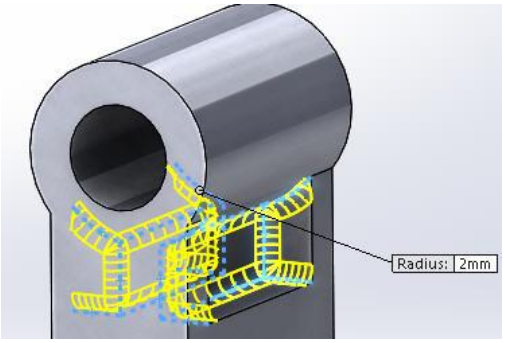


Draw the two circles in the given position and **Extrude Cut** through all.

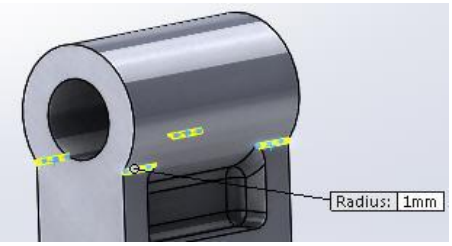


Add a **2mm** fillet to the edges of the recess as shown.



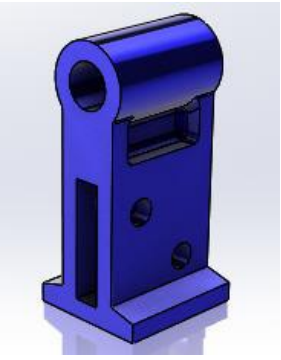


Add a **1mm** fillet to the edges shown.



Appearance

Give the part a Blue **Brushed Steel** finish.



Save as Head

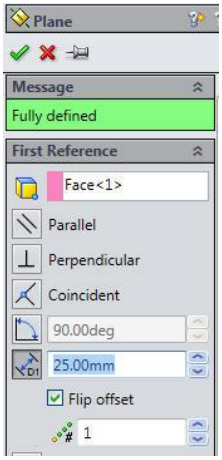
Thread

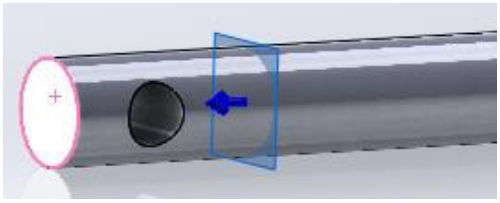
Draw a circle diameter **12mm** on the **Right Plane** and extrude by **175mm**.

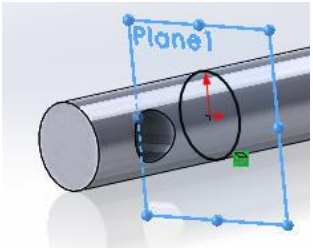
On the front plane, 10mm from the end of the cylinder, draw a circle of diameter 7mm. Add a **Horizontal Relation** between the centre of the circle and the origin and **Extrude Cut** in both directions.

To create the threads.

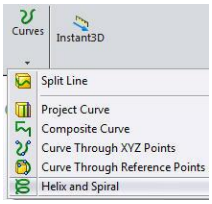
Create a plane parallel to the end of the bar and offset by **25mm**.
Use **Convert Entities** to draw the circle on this plane.





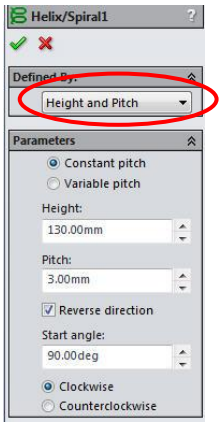


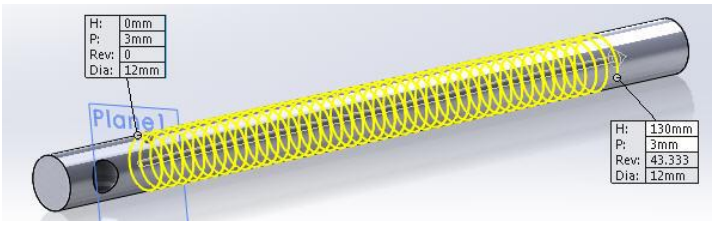
Select **Helix/Spiral** as shown.



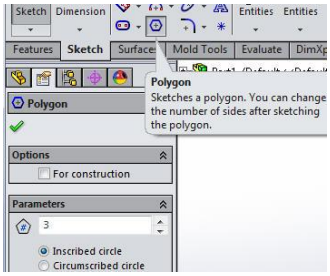
Draw the spiral as shown using **Height and Pitch.**
Select **Constant pitch**.
Height 130mm.
Pitch 3mm.
Start angle 90°

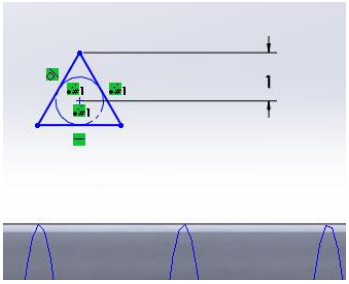
Clockwise.



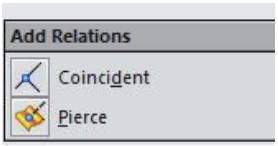


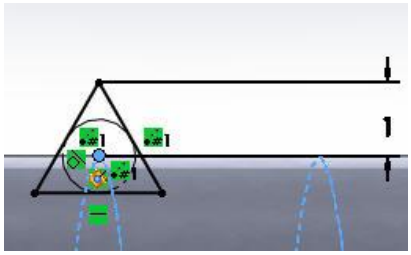
On the **Front Plane** sketch the three sided polygon (equilateral triangle) shown.
Draw it close to the spiral for convenience.
Add a **Horizontal relation** to the bottom of the triangle.
Give a dimension of **1mm** between the centre of circle and apex of triangle.



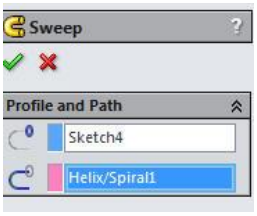


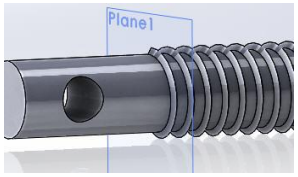
Add a **Pierce relation** between the centre point of triangle and the spiral.



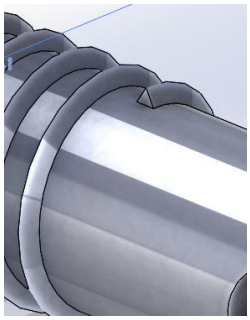


Use the **Sweep** command to draw the threads.

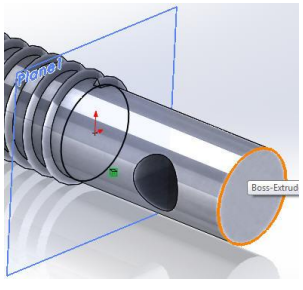




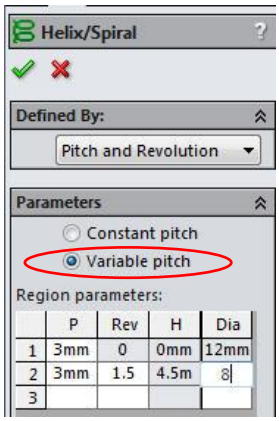
Looking closely at the thread we see that the thread ends abruptly. This would not be the case in reality.

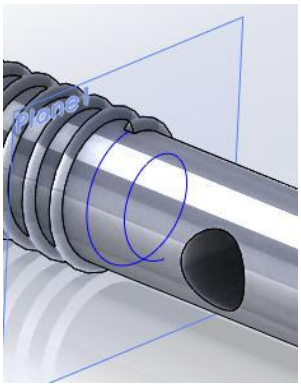


To rectify this on **Plane 1.** Use **Convert entities** draw the circle shown.



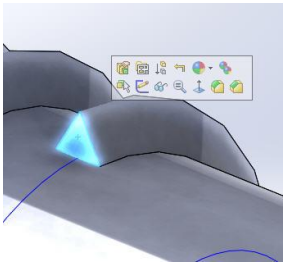
Accept the sketch and draw a new **Helix/Spiral.**

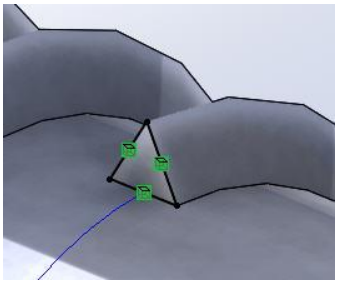


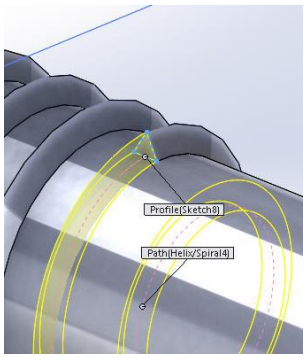


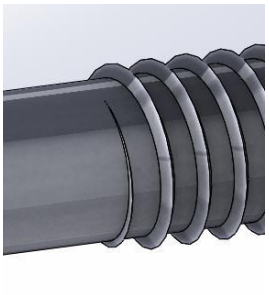
a. Select **Pitch and Revolution.**
b. Select **Variable pitch.**
c. Change the revolutions to **1.5mm**.
d. Change direction to counter clockwise.
e. Change the diameter to **8mm**.

On the face of the triangle draw a new sketch. Use **Convert entities** to transfer the triangle onto this new plane.
Use **swept boss/base** to complete the thread.



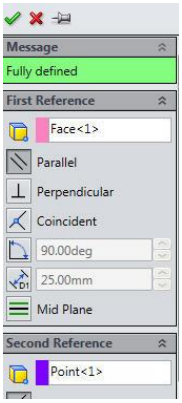


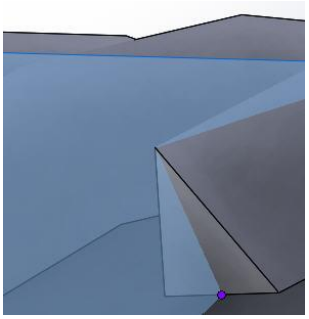


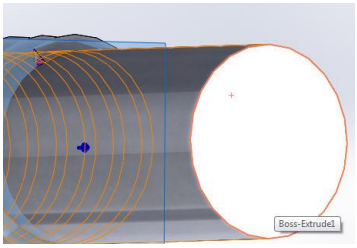


The thread on the other end is completed in the same way.

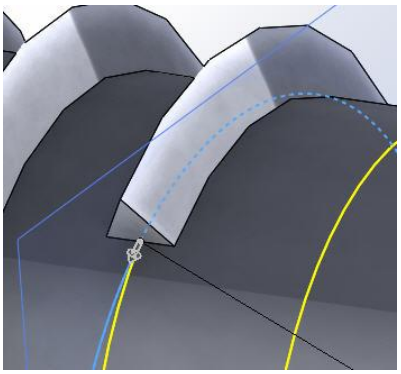
First create a plane which is parallel to the end of the cylinder and passes through the midpoint of the base line of the triangle.

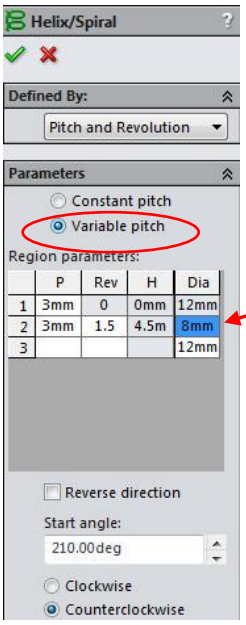






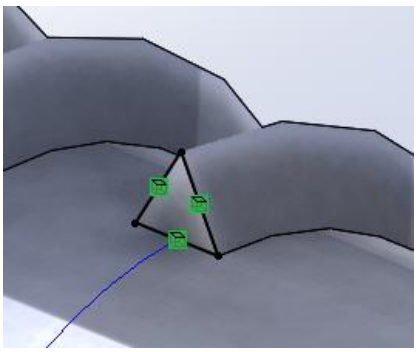
On this plane use **Convert Entities** to draw the circle. Select **Constant pitch** first to align the start of the helix with the midpoint of the triangle by altering the start angle.

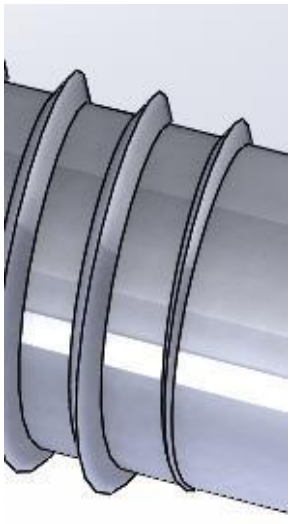




Then without exiting the command, select **Variable pitch** and as in the other side change the diameter to **8mm** keeping the revolution at **1.5.**

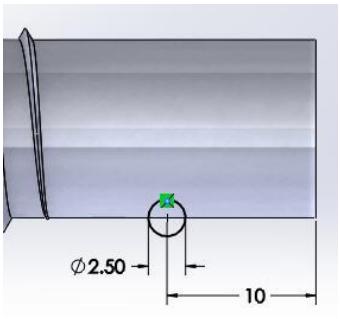
Select the face of the triangle as a new sketch plane. Select sketch and use **Convert entities** to produce the triangle on to this new plane. Select sweep to complete the thread.

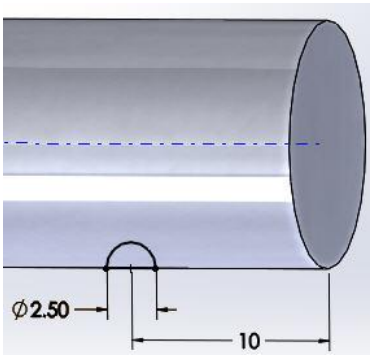




Recess for pin

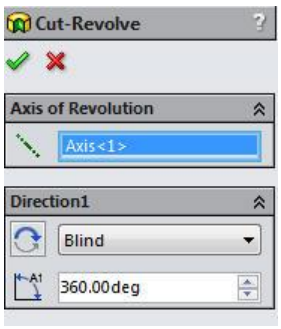
Draw a circle on the **Front Plane** as shown, having a diameter of **2.5mm.**

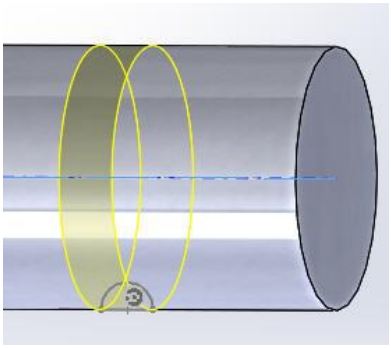




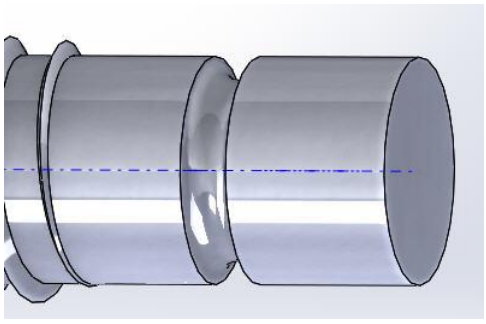
Draw diameter and trim bottom half of circle.

Show temporary axis and revolve cut to achieve result shown.





Add a **1mm Fillet** to each end**.**



Appearance

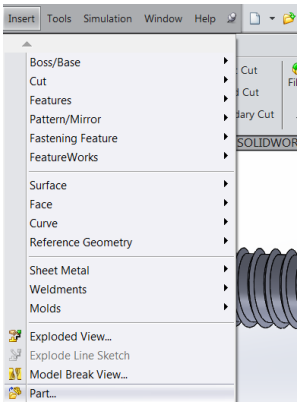
Give the part a **Brushed Steel** finish.

Save as Thread

Adding Thread onto the Head.

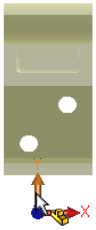
Open **Thread** part.

Select **Par**t under the **Insert** menu.

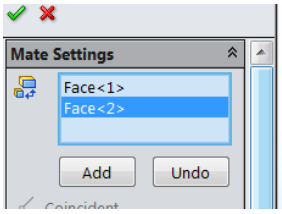


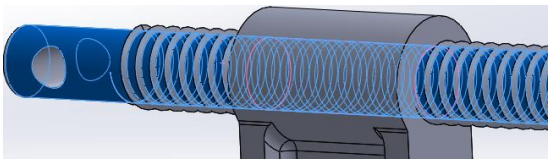
Select **HEAD** part.

If the head part comes in in the wrong orientation click on the **X, Y or Z axis** until it is the right way up.



On the left hand side the following window appears. Mate the cylinder of the **Thread** part with the inside hole on the **Head** part. Press **add** and accept.





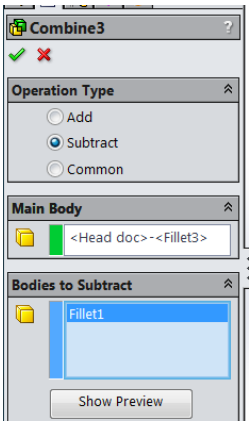
Select **Combine** 

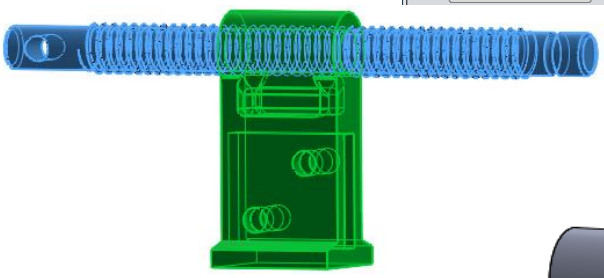
under the Features commands or under search.

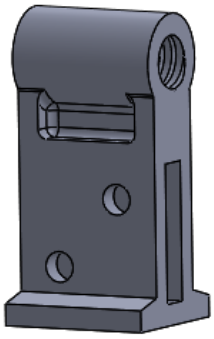
Under **Operation Type** press **Subtract**.

For **Main Body** select the **Head** part.

For **Bodies to Subtract** select the **Thread** part.







Now the threads are on the required part.

Appearance

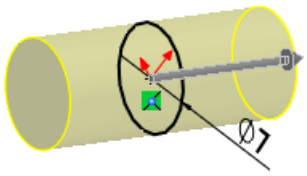
Give this part a **Blue, Brushed Steel** finish.

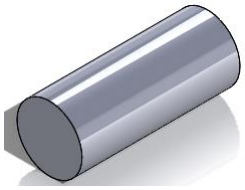
Save this new part as Thread 2.

Rivet 

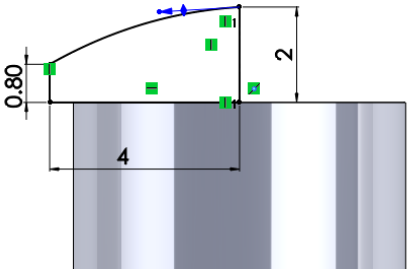
The rivet is drawn to the following dimensions

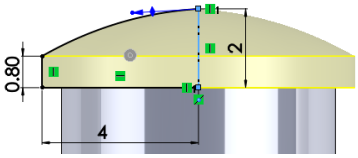
Circle diameter **7mm** and extruded by **17mm.**



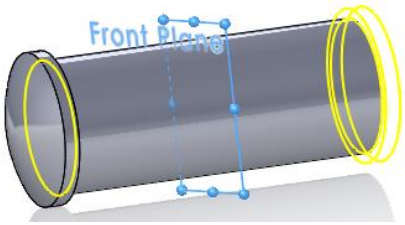


On the **Top Plane** the following shape is drawn on the end of the cylinder, and revolved to produce the head of the rivet.





This shape is mirrored about the **Front plane** as shown.



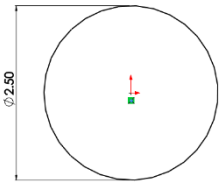
Appearance

Add a **Brushed Steel** appearance to the part.

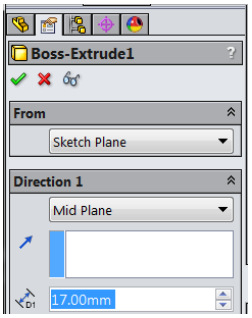
Save as Rivet

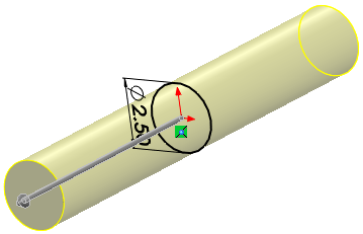
Pin 

Draw the circle of diameter **2.5mm** on the **Front Plane.**



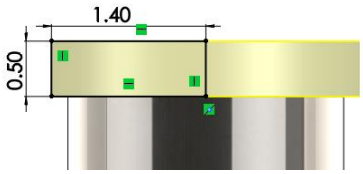
Extrude by **17mm** using mid plane as shown.



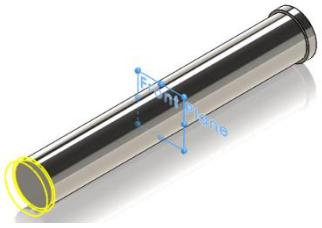


On the **Top Plane** draw rectangle as shown onto the end of the cylinder.

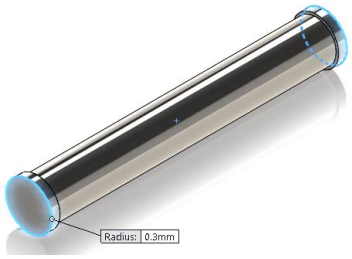
Revolve.



Mirror the end about the front plane as shown.



Add **0.3 mm** fillets.



Appearance To complete give a **Polished Steel** finish to the pin.

Save as Pin

Assembly

MATES

In assembling the sash clamp we see there has been improvements to mates which will increase the speed in which the assembly is built.

ALT and drag

This method was used in SolidWorks 2012. This has been improved. Now you can change the sensitivity of when this mate takes effect. Slowing the speed allows "hovering" over the target, thus giving more control over the operation,(see instructions on page 22).

SMART MATES 

Select smart mates from the tool bar. Then double click a reference. Then click the corresponding reference and SolidWorks presents the mates toolbar.

The advantage of this method is that you can rotate the model while selecting the mate reference (cannot do this in Alt and drag method).

QUICK MATES

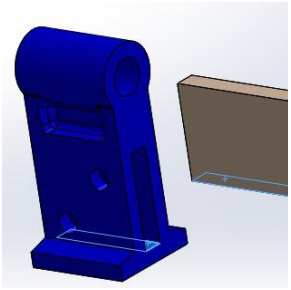
Just select the faces you want to mate and a quick mate toolbar appears and make your selection.

Mating Head to Bar

Open **NEW Assembly.** Click OK.

Bring in the Bar.

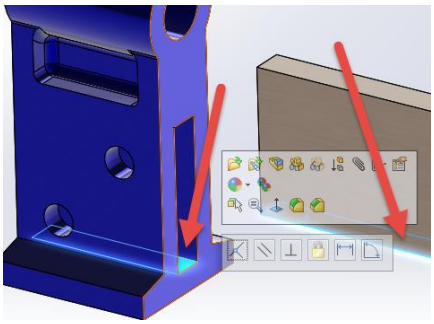
Select **Insert Component** and bring in the **Head**.

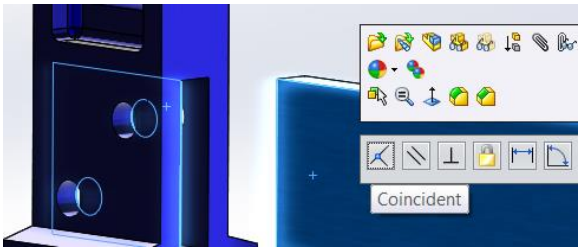


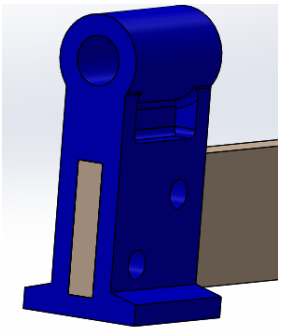
Using quick mates select the two faces to be mated (hold down the shift key to select the second one).

The quick mate toolbar appears.

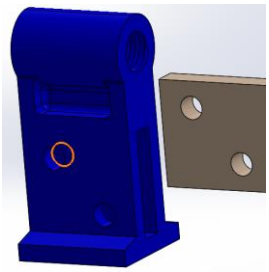
Select the coincident mate and repeat so that Head is in the proper location.







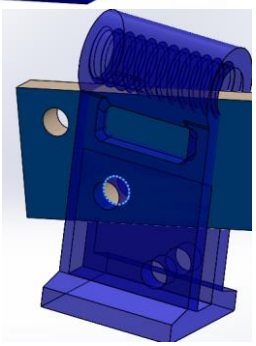
Alternatively



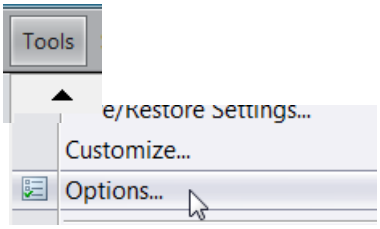
Mating the **Head** to the **Bar**.

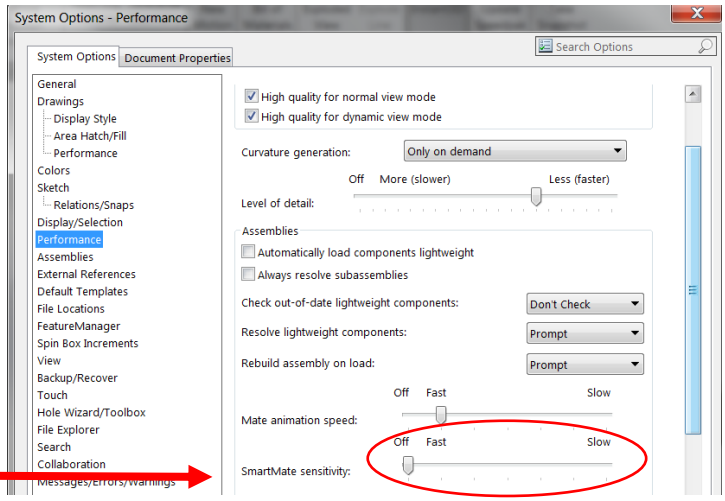
This can also be achieved by using **ALT and drag** method.

When using this method to mate the two holes as shown, the mate tends to jump to the nearest hole which can be a nuisance if there are a lot of holes in the vicinity.

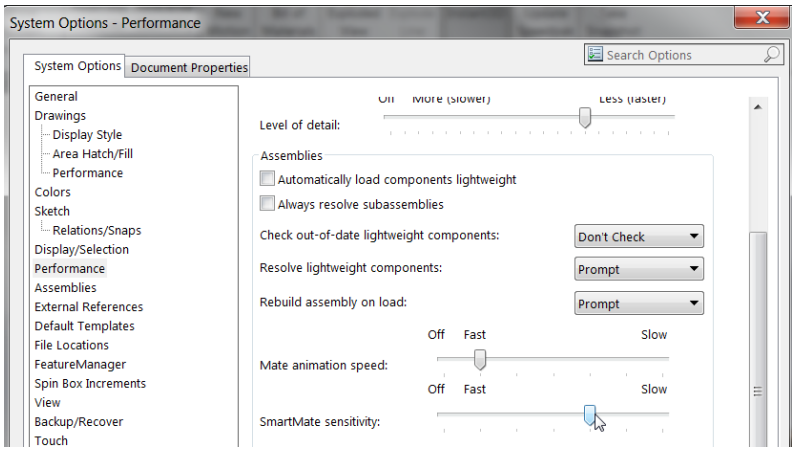


In SolidWorks 2015 you can slow down the smart mate sensitivity by selecting **Tools** on the toolbar and select **Options**, **System Options**, **Performance** and move the smart mate sensitivity to slow.

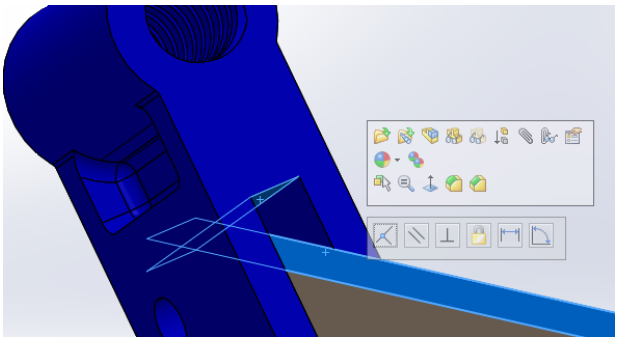


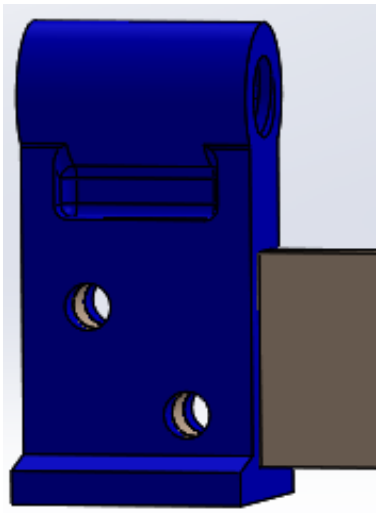


When the mate is left over the correct position for a while it will accept it.



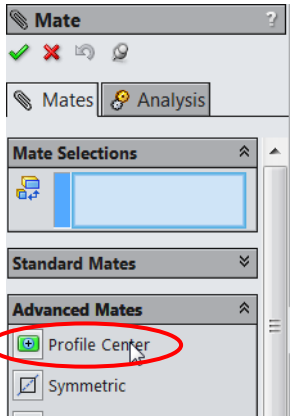
To complete, select the two faces to be mated and select the coincident mate.





Inserting Rivets

To insert rivets the quickest way is as follows -

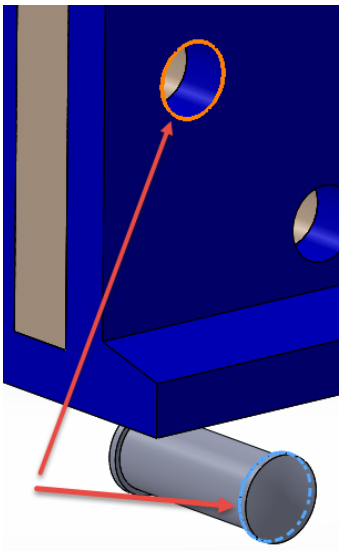


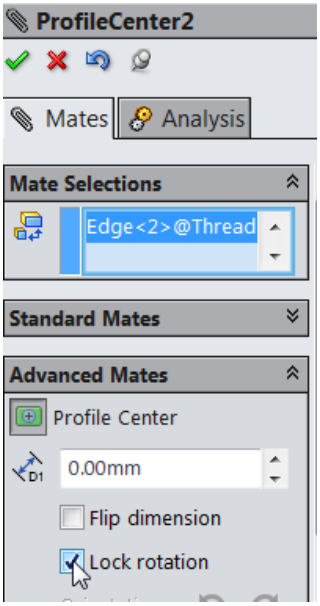
Select **Mates.**

Under **Advanced Mates** select **Profile Centre.**

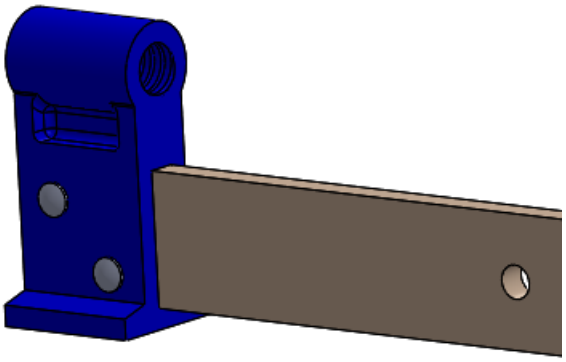
Left click on the two circles shown to mate.

They move into the correct position immediately.





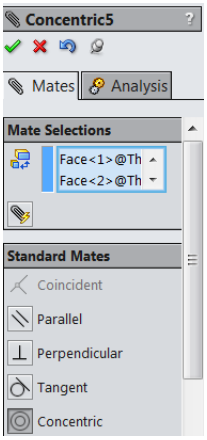
If the **Lock rotation** box is ticked as shown the rivet is locked and will not rotate.

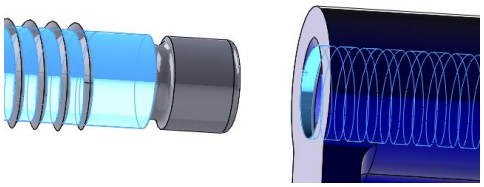


Assembling the Thread to the Head

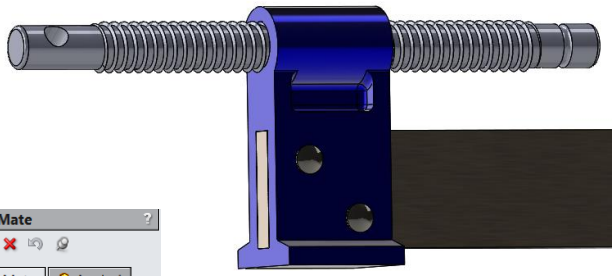
Insert the Thread.

Select the two objects and accept the concentric mate.

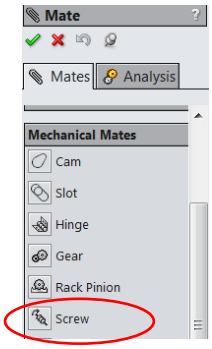


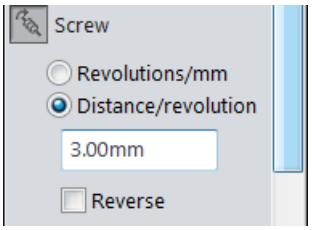


Move the thread further into the Head.



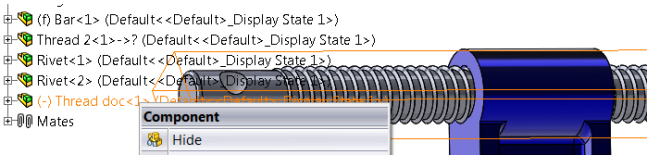
In **Mechanical Mates** select **Screw**.

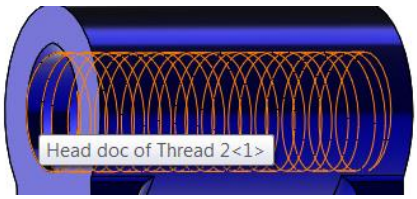




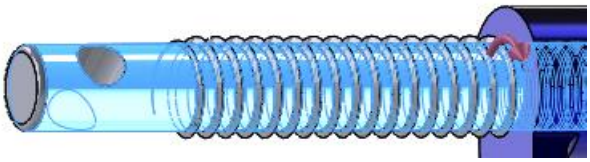
Select **Distance/revolution** and set at **3mm**

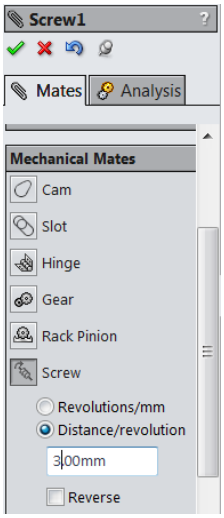
Hide the thread bar and select the inside of the Head.





Then unhide the Thread bar and select the Thread bar





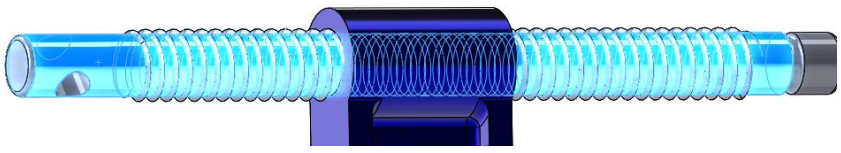
The distance/revolution automatically goes back to default of 1mm.

Change this to **3mm** also.

An arrow appears on the display for rotation. If the rotation is wrong tick the reverse box.

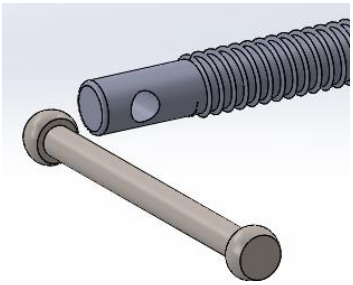
Accept.

When you rotate the Thread bar clockwise with the mouse it moves further in the Head

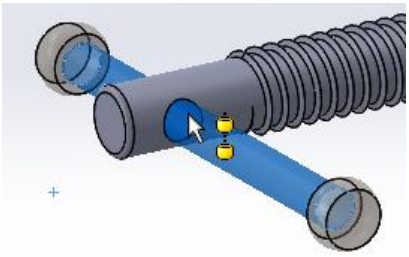


Assembling the Tommy bar to the Thread

Mate using **ALT** and drag.

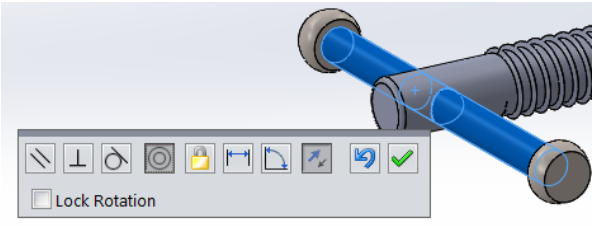


Here we can press the **ALT** button on the keyboard and drag the Tommy Bar to the correct position.



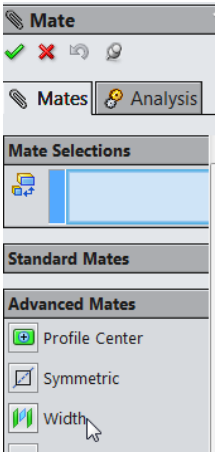
Accept the concentric mate.

The tommy bar must be free to rotate so do not tick the lock rotation box.

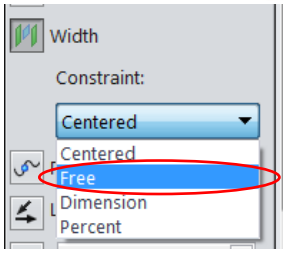


In reality the tommy bar can move until either end touches the threaded bar. To show this limited movement select **Mate**.

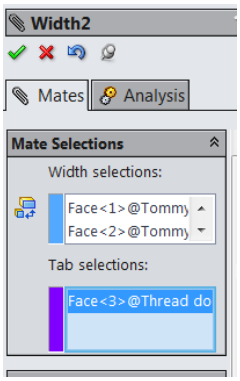
In **Advanced Mate** select **Width** mate.



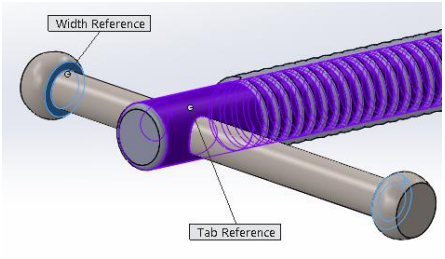
In the down arrow under width mate select **Free.**



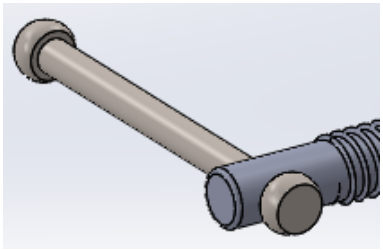
Select the ends shown on the **Tommy bar** as **Width selection**.



Select the **Thread** bar as **Tab selection**.



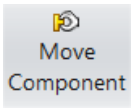
Now the tommy bar is free to move until it touches the treaded bar at either end.



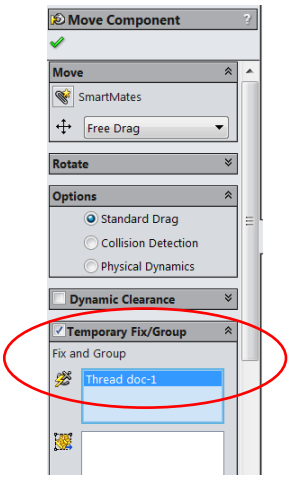
This is quite difficult to see as when we try and move the tommy bar to its end limits the thread bar rotates instead.

We want to isolate the movement. To do this we can use another tool (which is new to SolidWorks 2015) called **Temporary Fix**

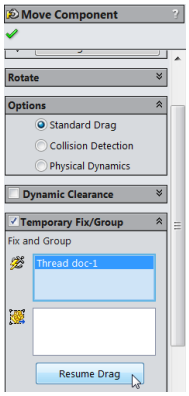
Select Move



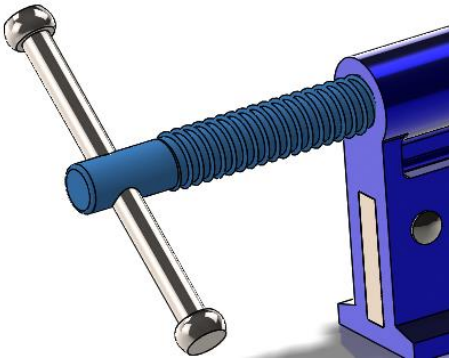
Tick the **Temporary Fix** button and select the Thread bar to fix.



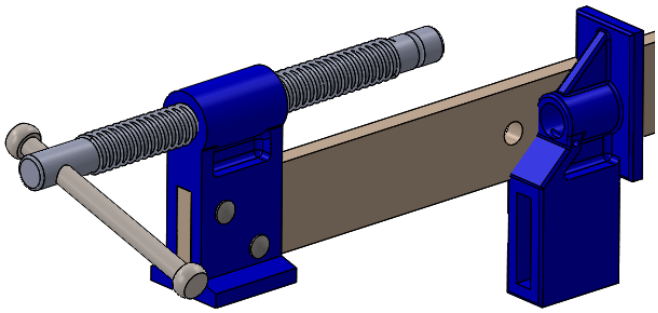
Press **Resume Drag** button.

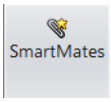


Now the thread bar is fixed temporarily and the limited range of movement of the tommy can be examined.



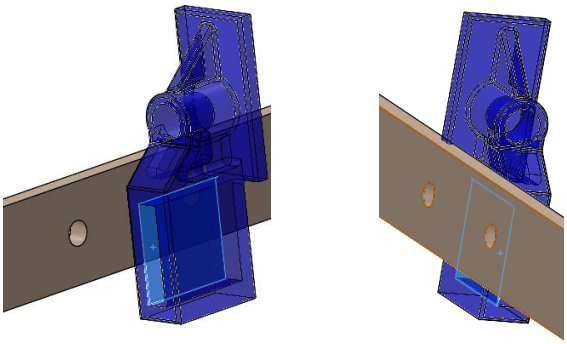
Inserting the Slider 1



Here we will use smart mates 

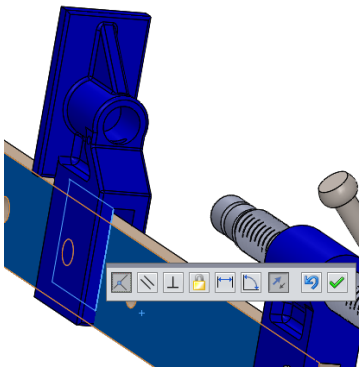
Select **Smart Mates**

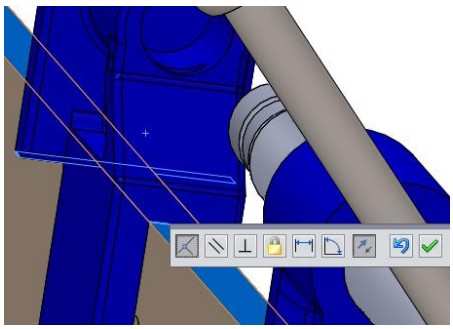
Double click the reference and then click the corresponding reference.



Note:

The advantage of this method is that you can rotate the object to select the second reference.

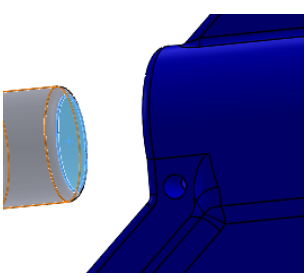


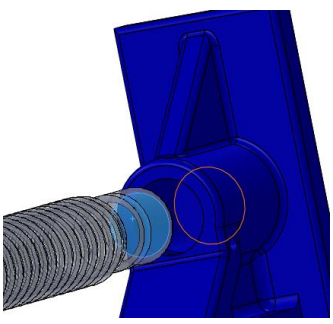


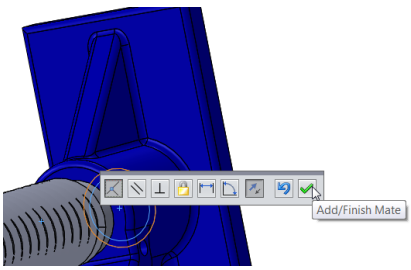
Select **SmartMate** 

again and mate the end of the thread bar to the inside of the hole on slider 1

Accept

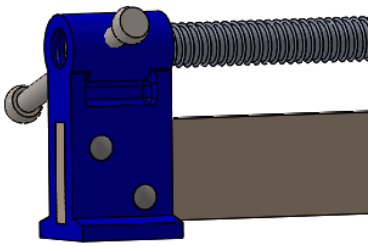




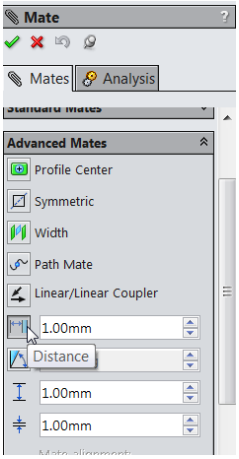


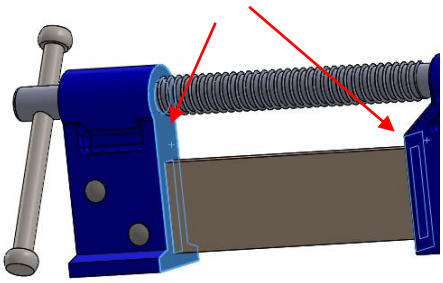
Finally in reality the slider 1 will have limited movement along the bar.

To do this select Mates.



Under Advanced mates select Width and activate distance.

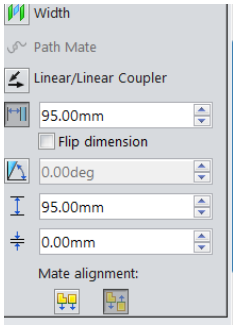




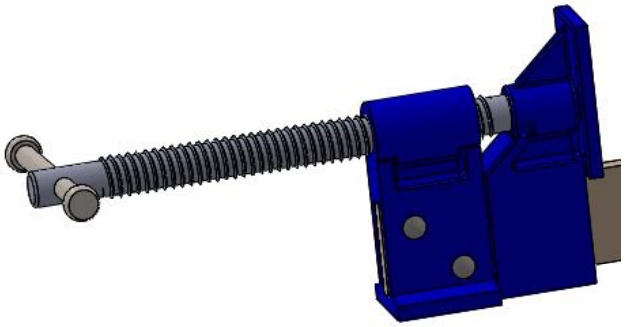
Select the two faces that the distance limits will refer to.

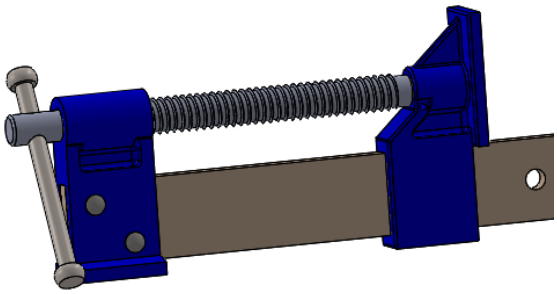
Set the **max value** to **95mm**

Set the **min value** to **0mm**



Accept.

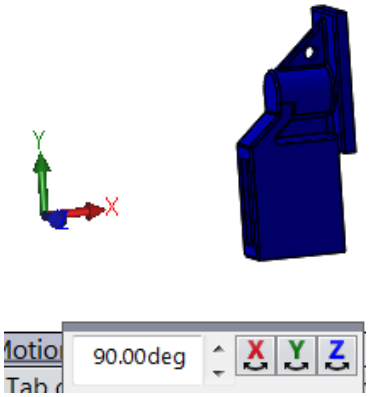


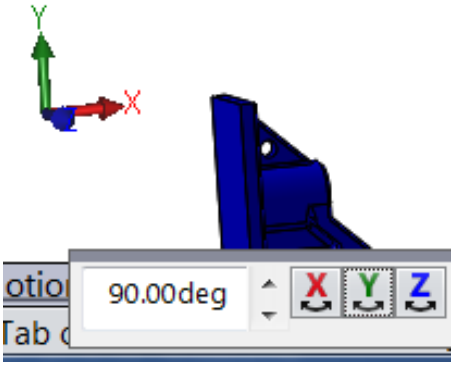


Save

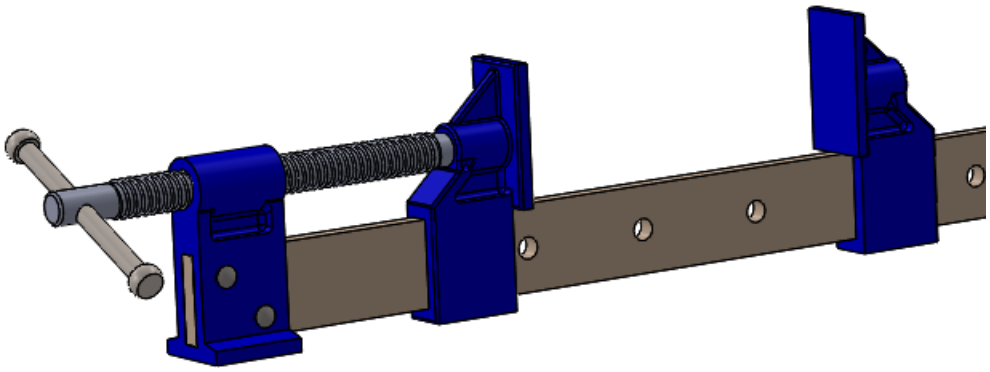
Inserting Slider 2

When slider 2 is brought in its orientation is wrong. A temporary pop up toolbar appears at the bottom of the screen. Use the Y- axis orientation tabs to rotate the part into the correct direction.





Then use **SmartMates** to position the slider correctly on the bar.

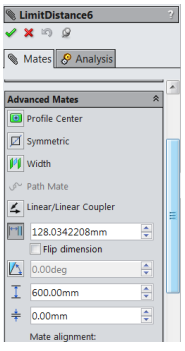


To enable the slider 2 to react as it would in reality additional mates can be added.

<u>For example</u>

In advanced mates select width mate and set the minimum distance to 0mm.

The max distance can be left at say 600mm for now.

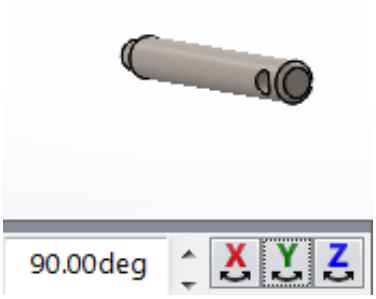


This will prevent slider 2 from passing though slider 1

Save

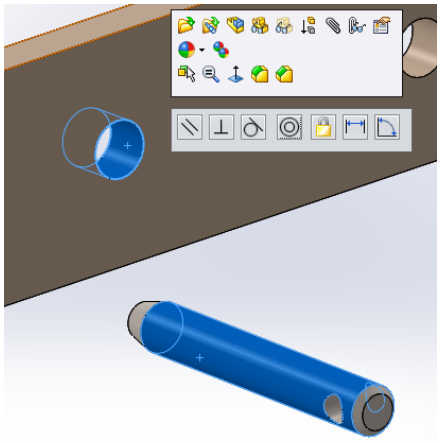
Insert Peg

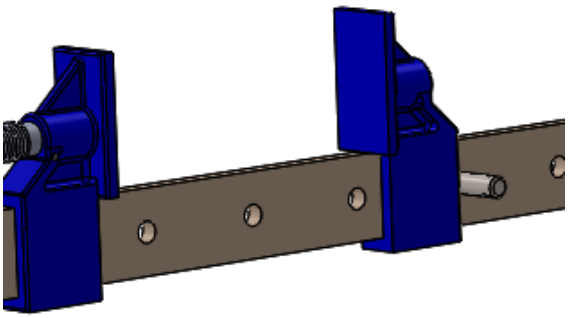
When the peg is brought into the screen rotate it about the Y-axis to align it up properly with the holes.



In mating the peg with one of the holes we can use **quick mates**.

Select the two references and select the concentric button.



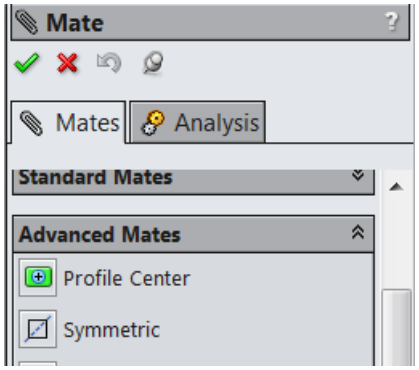


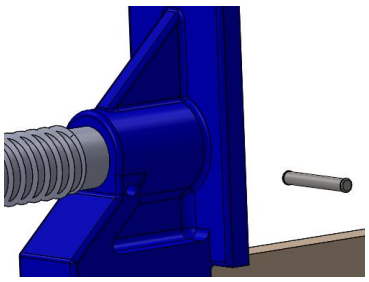
Save

Inserting the Pin

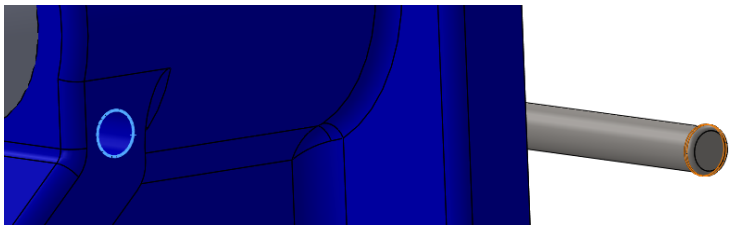
To insert the pin:

Under **Advanced Mates** select **Profile Center.**



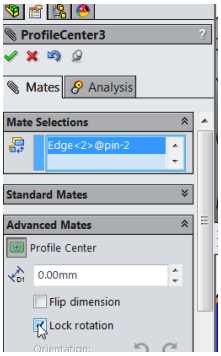


Select the two circles as shown.

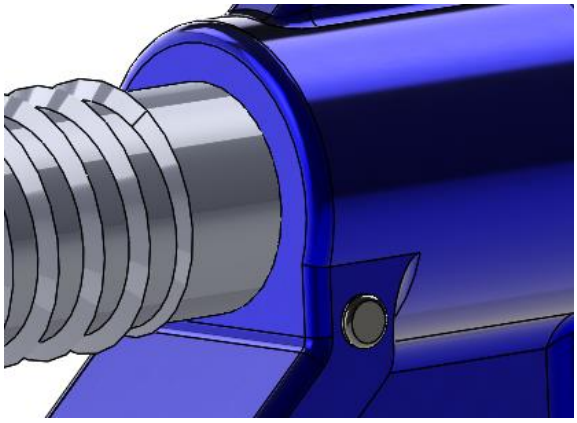


The mate is created immediately.

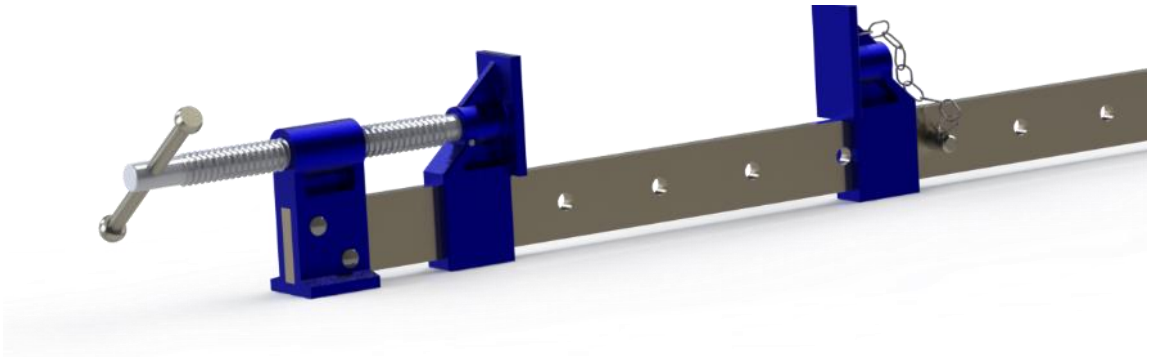
Tick the lock rotation box if required.



Accept the mates.



Save



The Chain added