PDST

Professional Development Service for Teachers | An tSeirbhís um Fhorbairt Ghairmiúil do Mhúinteoirí

SolidWorks CPD 2015
Phase 2

Design & Communication Graphics
Technology
Engineering
Construction Studies

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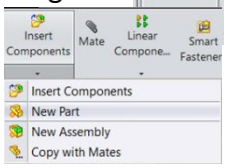
Introduction & Learning Intentions

Introduction: This lesson looks at how SolidWorks 2015 allows us to complete models more efficiently using new features and methods. Completing the PDST Pen will looks at parts, assemblies, drawings and PhotoView 360.

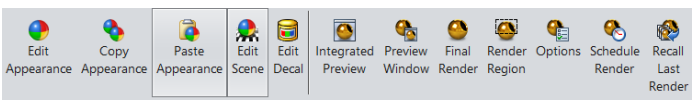
Learning Intentions: At the end of this workshop you should be able to:

- Use the **Combine** command in part modelling



- Make a part using **In-context assembly** modelling`
- Using the **View palette and Auto-arrange** dimensions in drawings

- Use **PhotoView 360** features to create photorealistic images



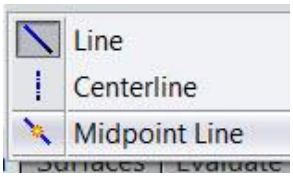
Creating Pen Lid Part

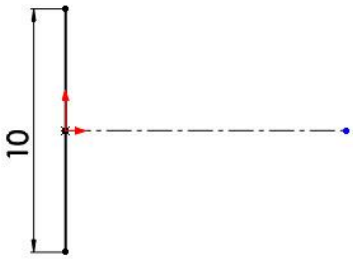
New Part

Start by creating a **New Part** and saving this part as "**Pen Lid**"

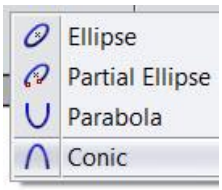
On the **Front plane** draw a construction line.

Select the **Midpoint Line** and draw a vertical line and add a **10mm** dimension.

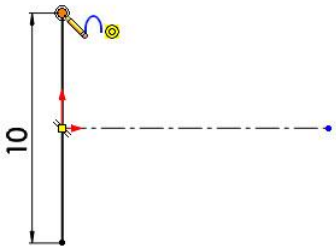


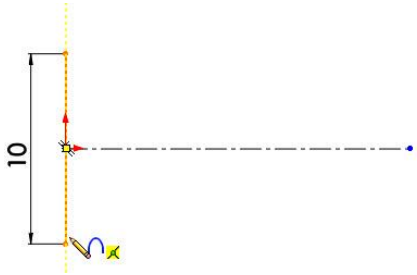


Select the **Conic** command

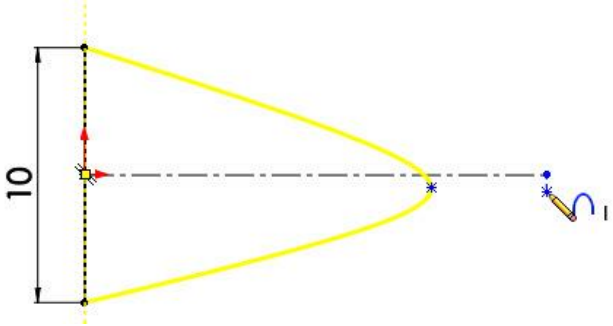


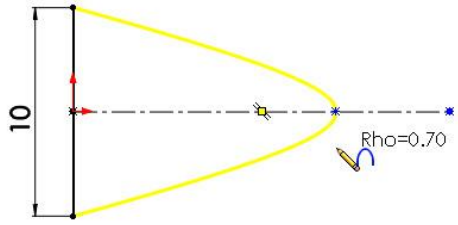
Select the top of the line then select the bottom of the line as shown.





Drag the curser along the construction line and select the end of the construction line to identify the sharp corner.

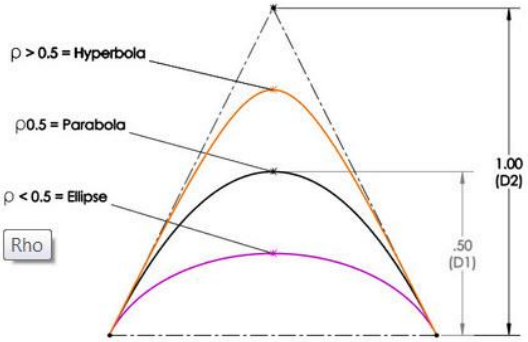




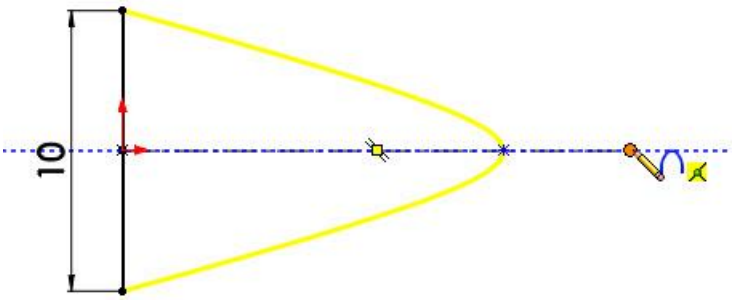
As the curser is moved along the construction line the Rho value is seen to change.

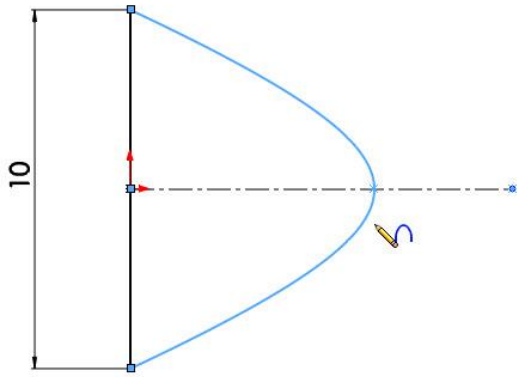
Note

Rho is the ratio of the distance of the peak of the curve (D1) to the apex of the sharp corner (D2) shown.



Select a point on the construction line to create the end of the conic curve.



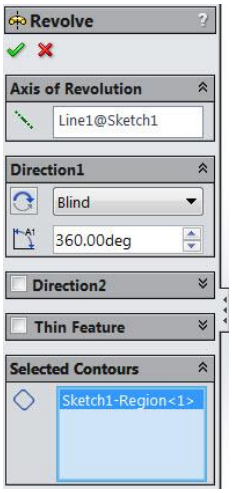


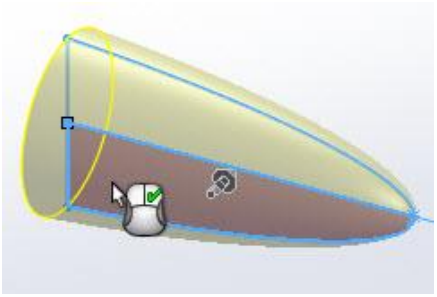
Add the following dimensions.

Accept the sketch.

Select **Revolve Boss/Base**.

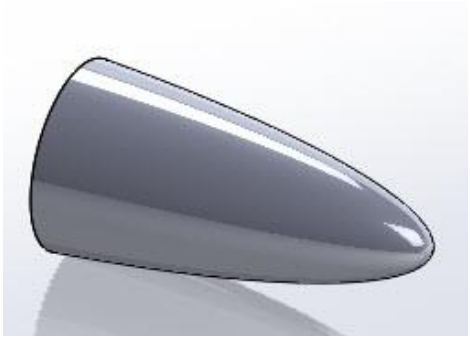
Select the region to revolve.



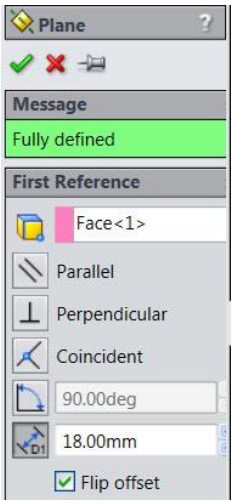


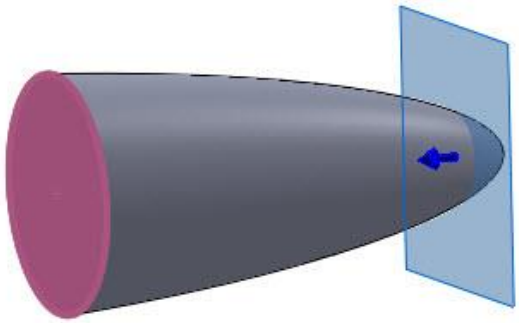
Note: The conic cannot be trimmed and revolved like a semicircle about the centreline as it will no longer be a conic shape and will not be fully defined.

Accept.

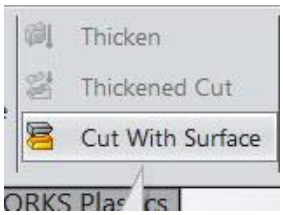


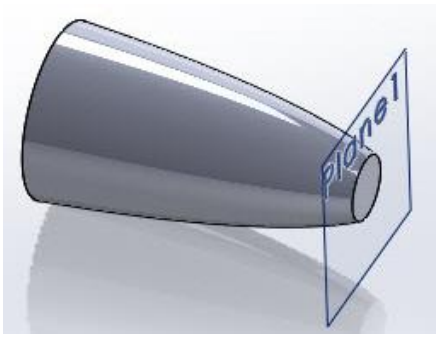
Create a **New Plane 18mm** from the face shown.





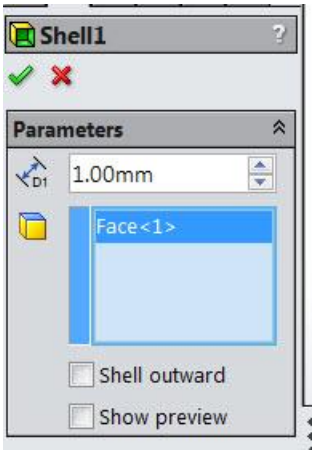
In the **Surfaces** commands select **Cut With Surface** and select the plane shown.

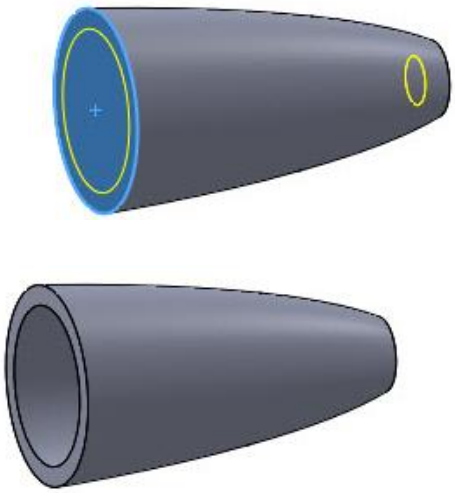




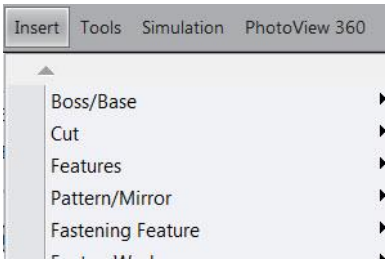
Accept.

In the features commands select **Shell** and select the face and shell by **1mm.**





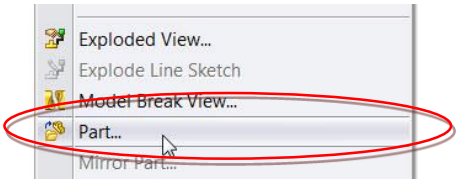
Adding the threads to the Pen Lid



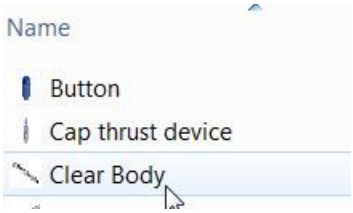
Under **Insert** select **Part**.

Note:

This feature allows you to insert parts to edit another part within a part file



In the Pen folder select the **Clear Body** part.



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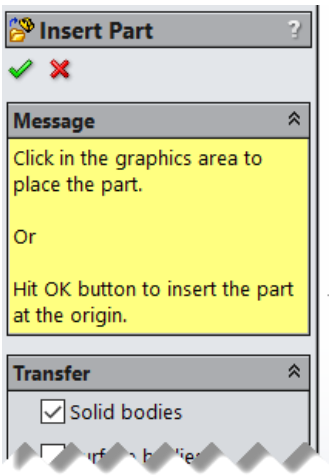
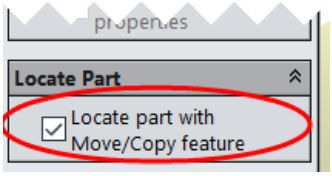
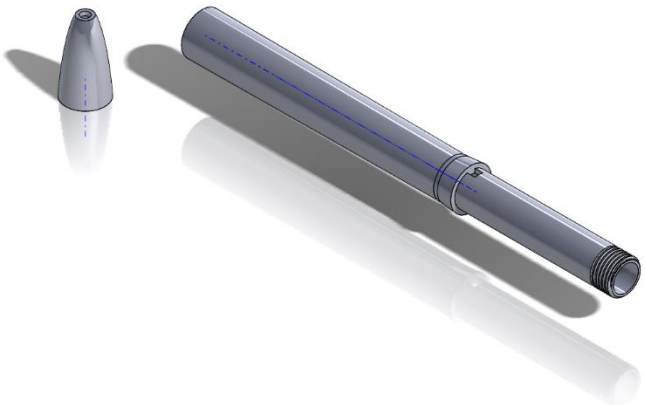


Make sure the "**Locate Part with Move/Copy feature"** button is selected on the **Insert Part** window

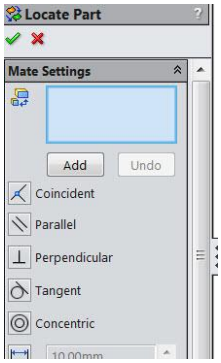


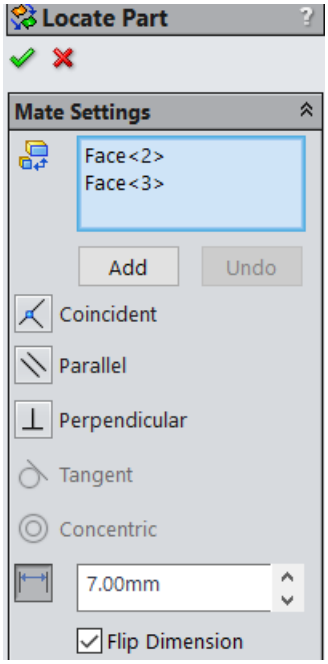
Position the **Clear Body** near but not on top of the existing part.

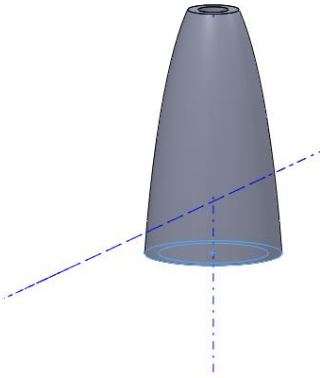


On the left hand side the **Mate Settings** window appears.

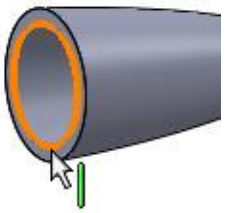
Select the circular face of the **Clear Body** part and the end of the **Pen Lid** part as shown and apply a **Distance** mate of **7mm.** Flip dimension if necessary

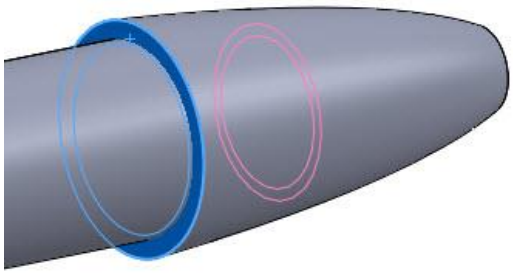


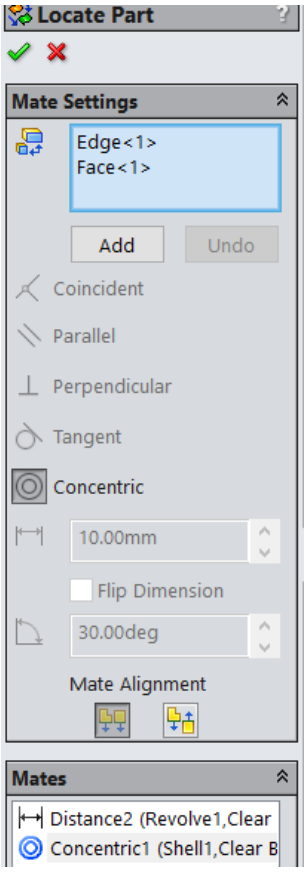




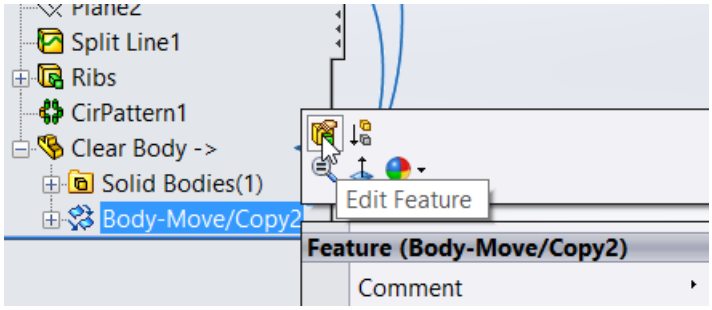
Select the circle **edge** on the face of the **Pen Lid** part and the cylindrical **face** on the **Clear Body** part as shown and select the **Add** button.





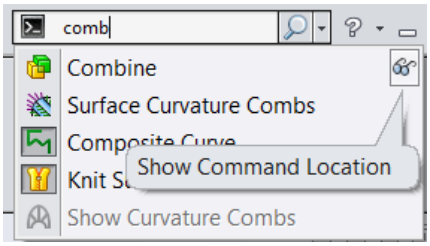


The Move/Copy feature can also be selected for editing after inserting a part in to another part file. Open the Clear Body feature, select the Body-Move feature and edit feature. This will allow you to position the body correctly.



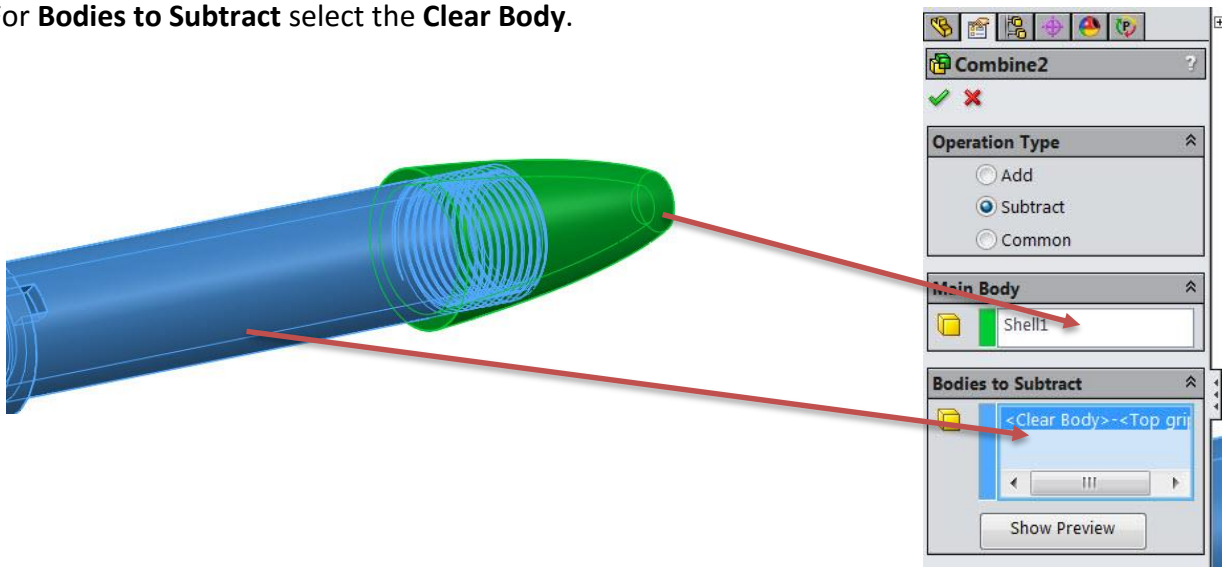
When the Clear Body part is in the correct position select the **Combine** command.

Use the search command to find the feature



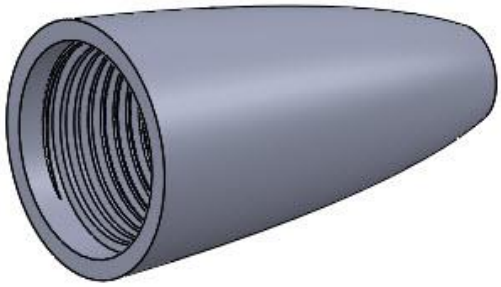
For **Main Body** select the **Pen Lid.**

For **Bodies to Subtract** select the **Clear Body**.



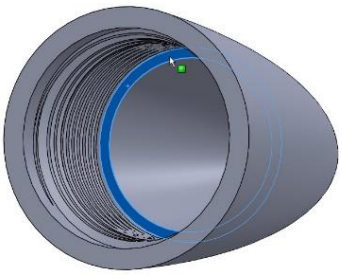
Accept.

The threads are now applied to the Screw Lid part.



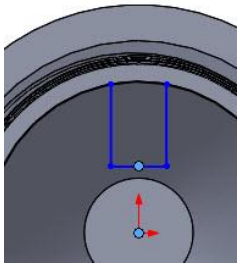
Creating the Ribs & Nib Hole

On the inside face shown draw a new sketch.

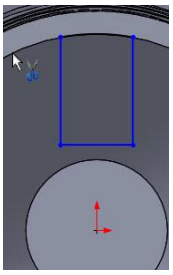


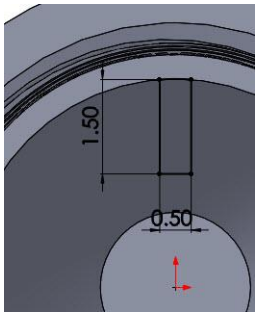
Use **Convert Entities** to draw the circle shown.

Select the **Line** command and complete the sketch, adding a **Vertical Relation** between the origin and the midpoint of the horizontal line as shown



Trim off the remainder of the circle and add the following dimensions.



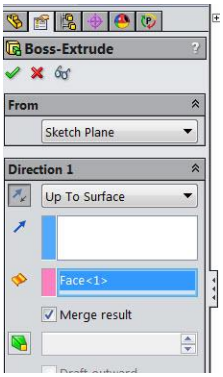


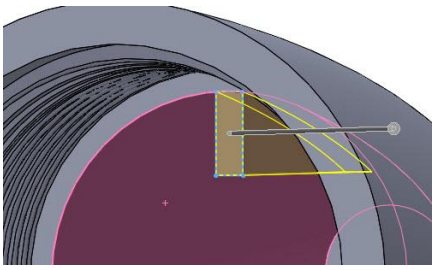
Accept the sketch.

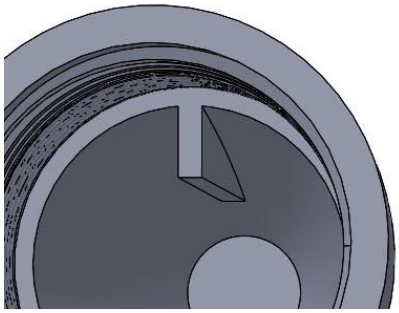
Select **Extrude Boss/Base.**

Select **Up To Surface.**

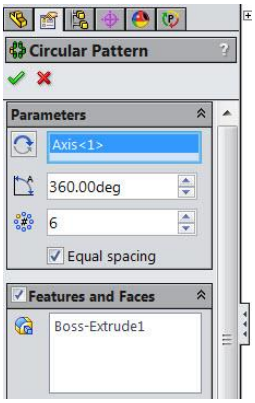
Select the inside as shown and accept.

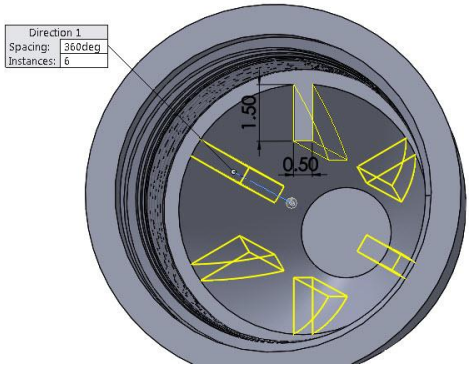


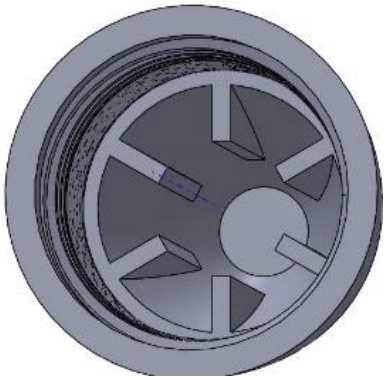




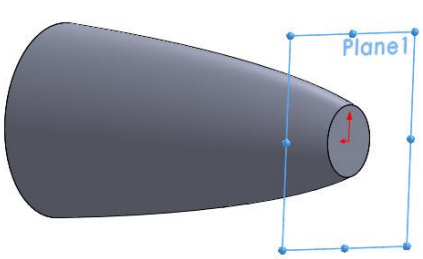
Using **Circular Pattern** complete the feature about the temporary axis as shown

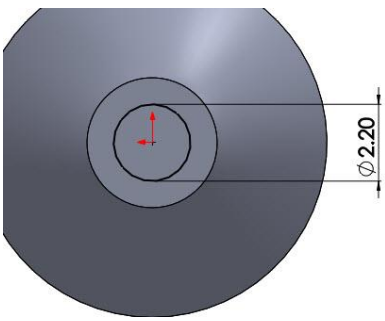




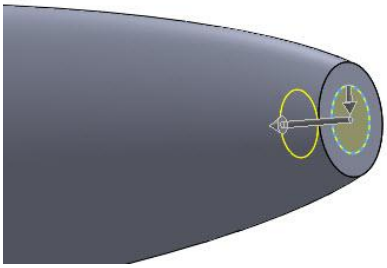


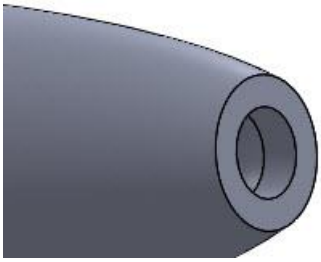
On **Plane 1** draw the circle shown to the given dimensions.





Extrude Cut by 2mm



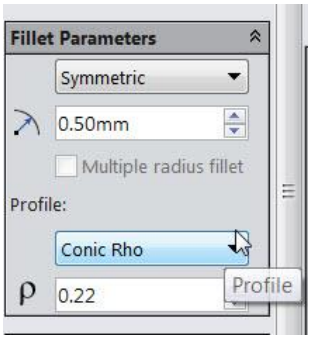


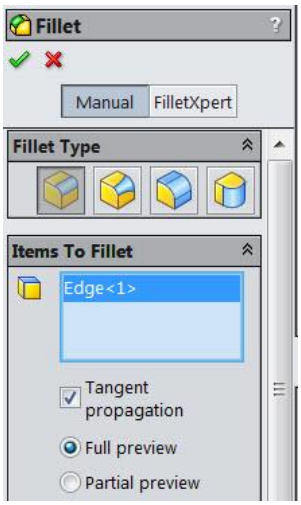
Adding the fillet to the edge

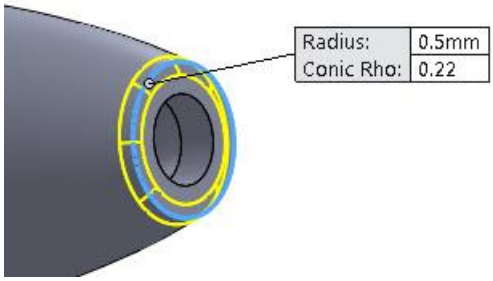
Select **Fillet.**

Select **Conic Rho**

Give the fillet a radius of **0.5mm** as shown and a Rho value of 0.22mm

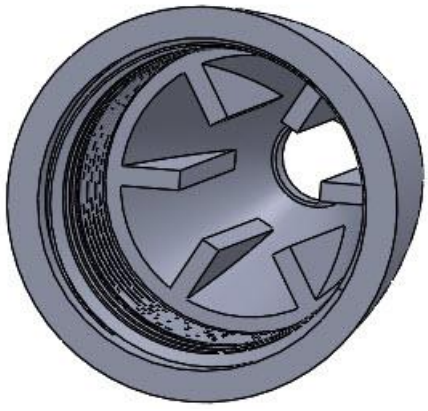






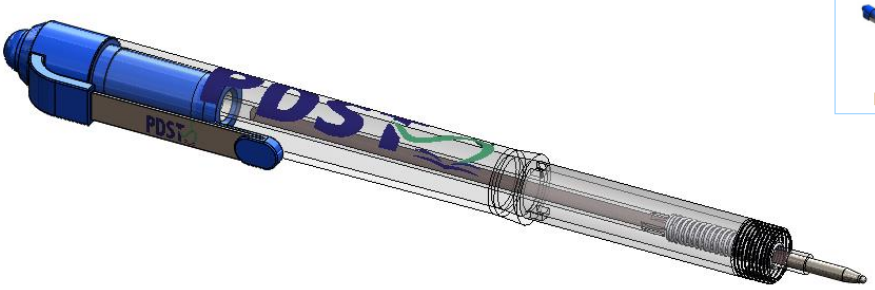
Accept

Save as **Pen Lid**



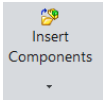
Assembling the Lid

Open the **PDST Pen** Assembly file.

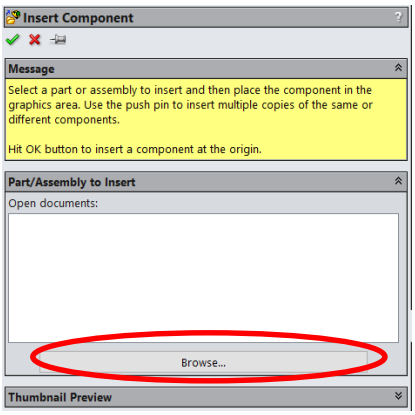


Note: The pen lid and rubber grip must be added to this incomplete assembly.

Select **Insert Components**

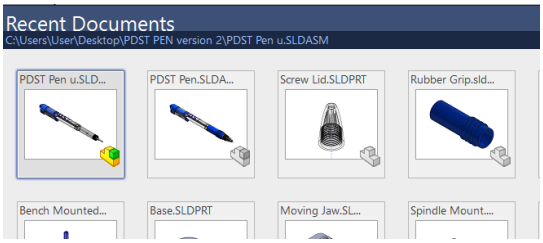


Browse to select the **Pen Lid**

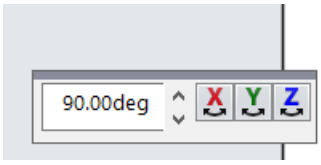


Note: The part can be dragged into the assembly by clicking the **R** key. This will display all the recent documents opened in SolidWorks.

Select the Lid and drag into the assembly.

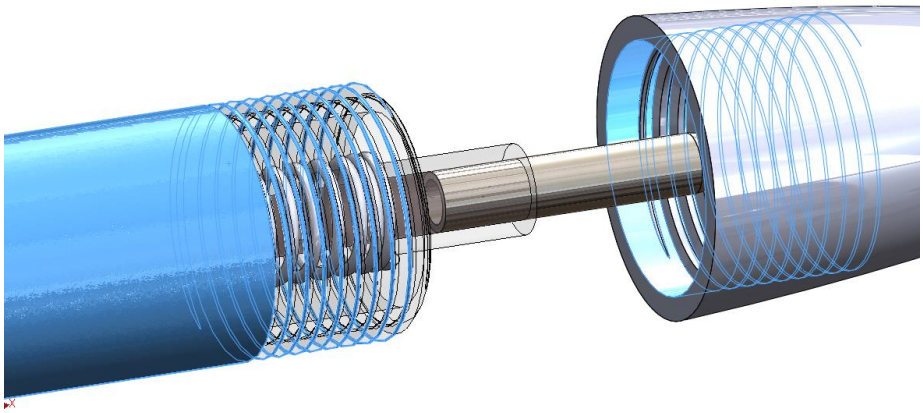


Orientate the Pen Lid using the **X**, **Y** and **Z** orientation tool.

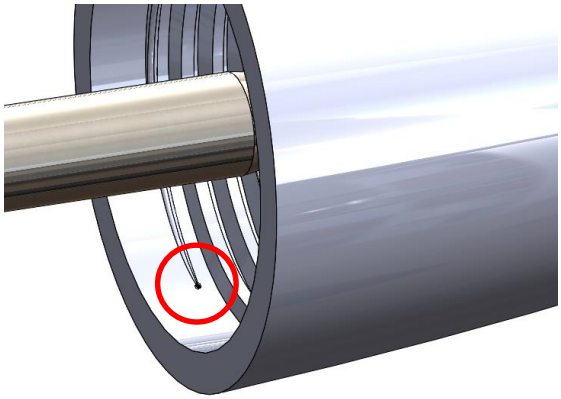


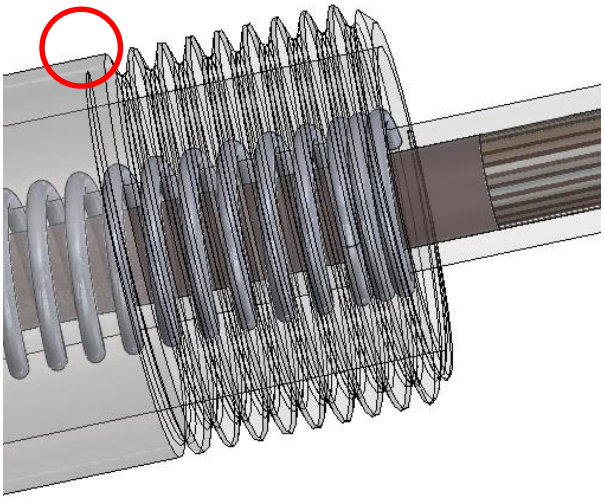
Place the lid into the assembly once correctly orientated.

Add a **concentric mate** between the circumference of the pen body and the internal circular surface of the lid.

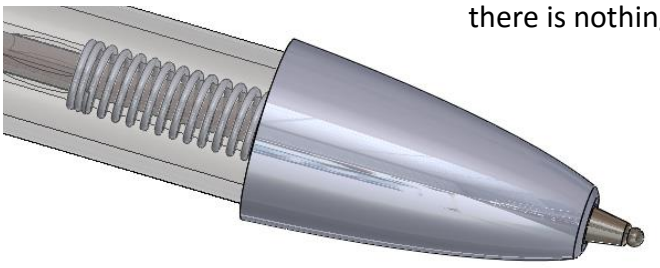


Add a **coincident mat**e between the end-points of the threads.





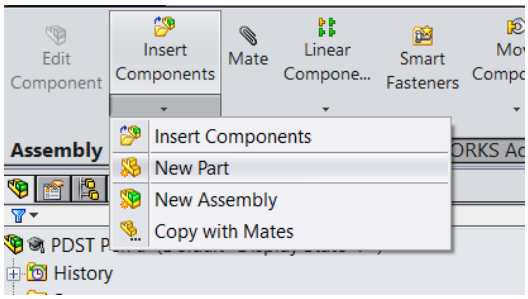
The endpoint of the thread on the body will be difficult to select because of the transparent appearance. The pen body must be rotated into a position where the end of the thread is at the edge and there is nothing else behind the point.

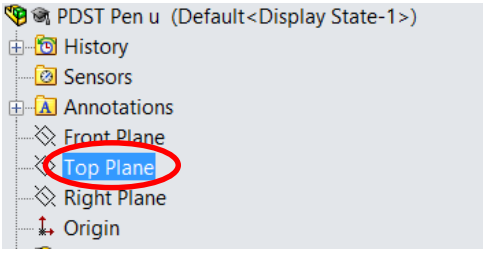


Modelling the Rubber Grip

NOTE: You can create a **new part** in the context of an assembly. That way you can use the geometry of other assembly components while designing the part. This can be saved as a separate part

To create a new part within the assembly, select **Insert Component** and **New Part** from the assembly tab.

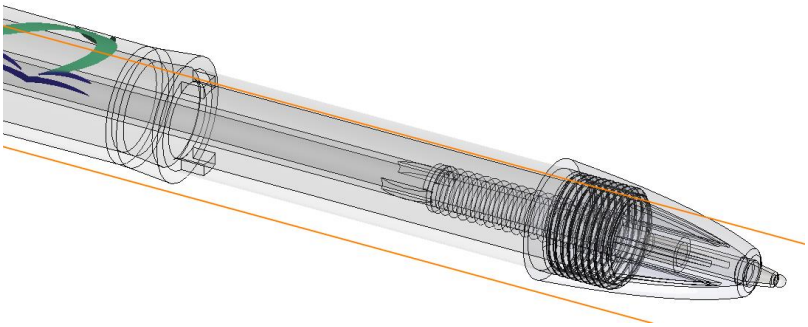


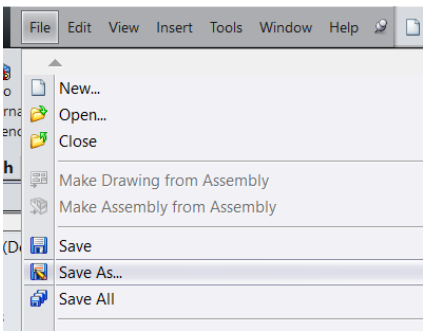


Select the plane or face the initial sketch in the new part is to be created. Select the **Top Plane**.

Note: A surface or face within the assembly could also be selected as the initial sketch plane.

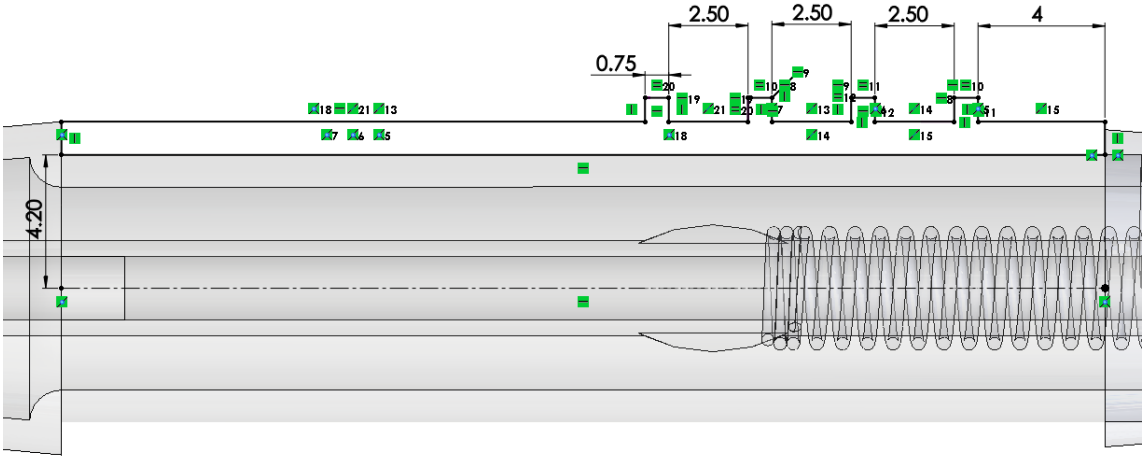
The assembly will turn **translucent** once the sketch plane / surface is selected.



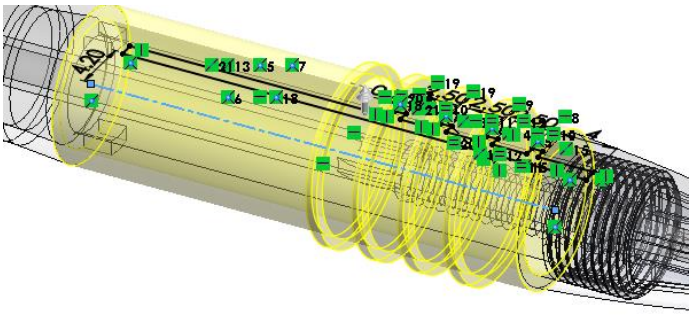


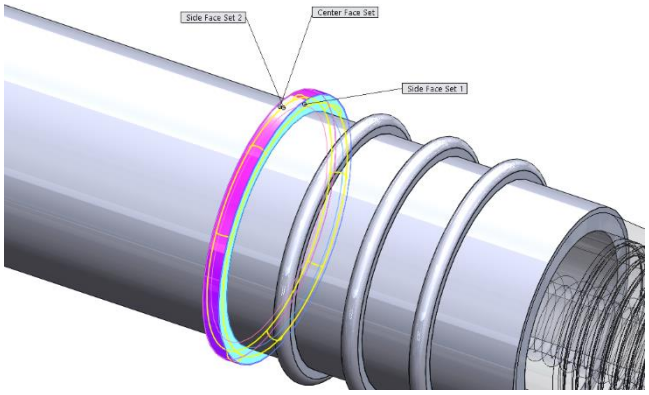
Save the new part as **Rubber Grip**.

Create the below sketch in the position for the grip.

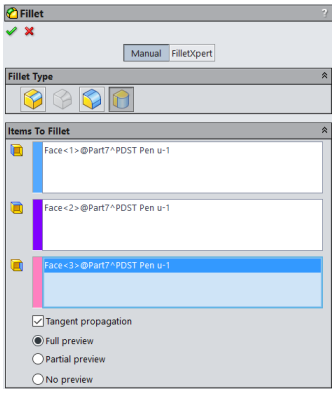


Revolve Boss/Base the sketch about the centreline to create the grip.



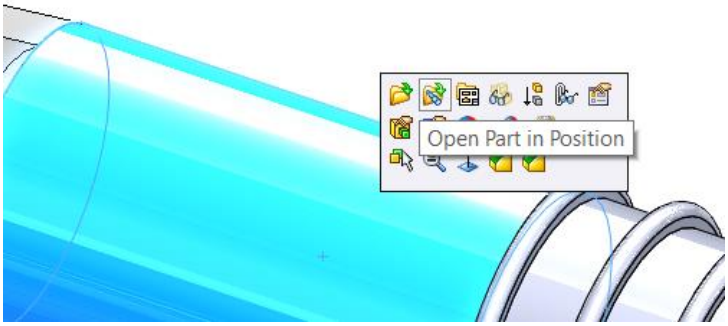


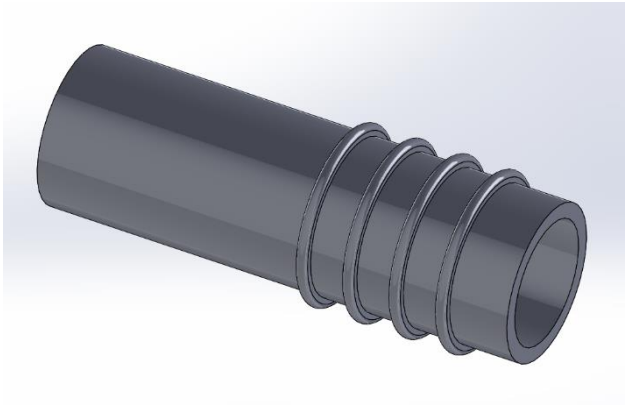
Add **full round fillets** to the ribs on the rubber grip.



Exit the Part by selecting the confirmation corner

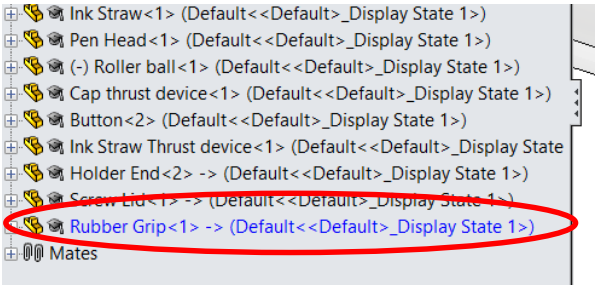
This part can also be opened separately by clicking on the part and selecting **Open Part in Position**.

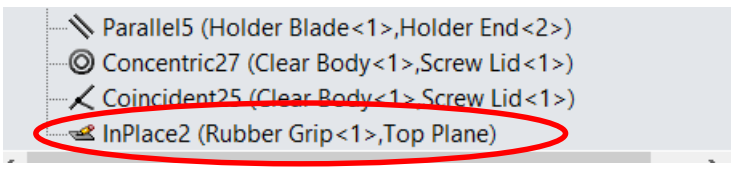




The part will open and further edits could be made.

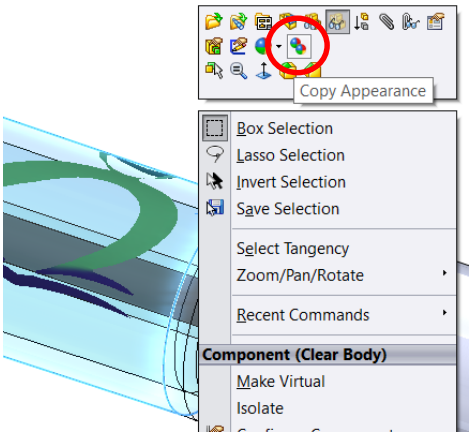
The new part can also be seen in the **Feature Manager** and mates have been added in the **Mates** folder.



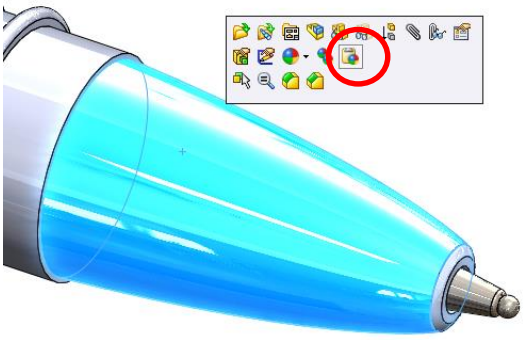


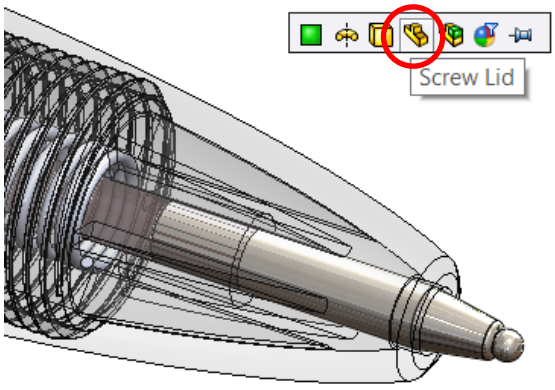
Adding Appearances

Copy the **Translucent** appearance added to the **Pen body** by clicking on the body and selecting **Copy Appearance**.



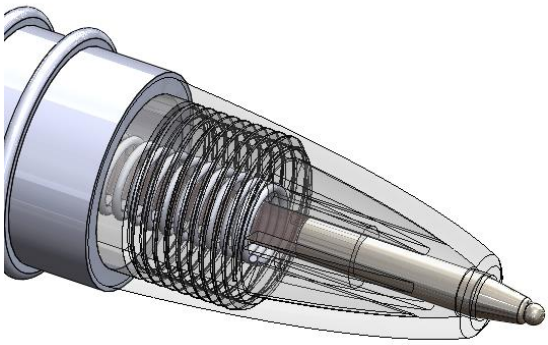
Click on the Pen Lid and select **Paste Appearance**.



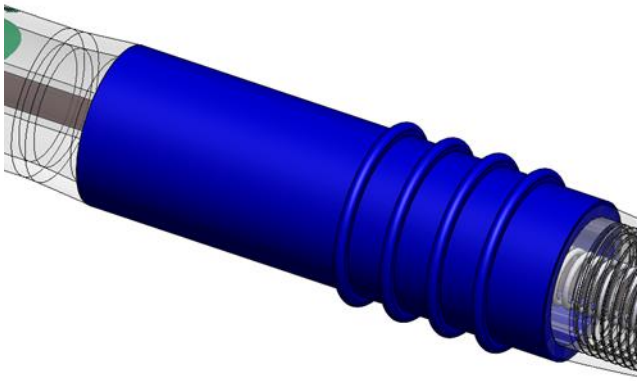


A dialogue box will offer a number of choices how to add the copied appearance to the part – choose **Part**.

A translucent appearance is now added to the entire part.



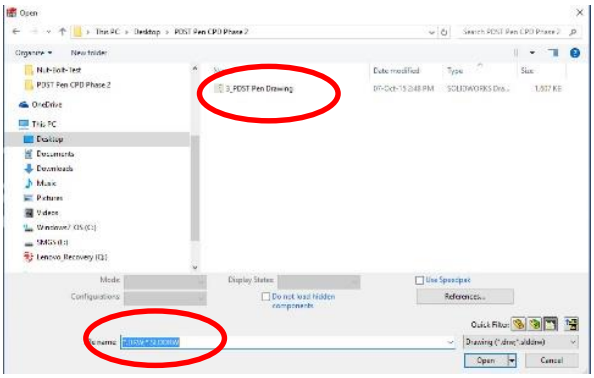
Add a Blue **Matt Rubber** appearance to the grip.



Save the completed Assembly.

Creating a Drawing

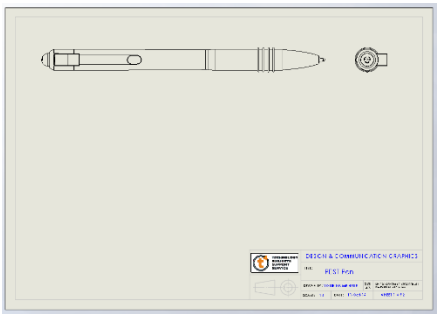
Open the Drawing in the folder

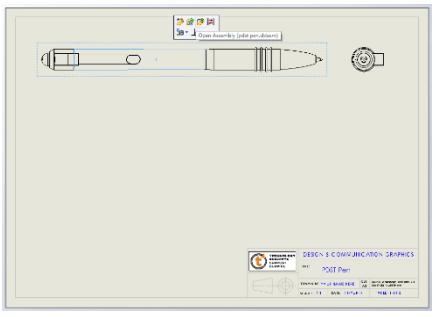


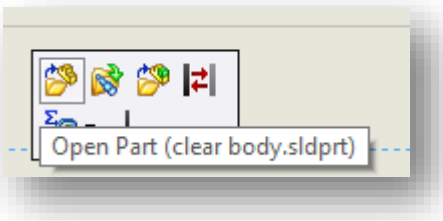
Opening a Part/Assembly from a Drawing

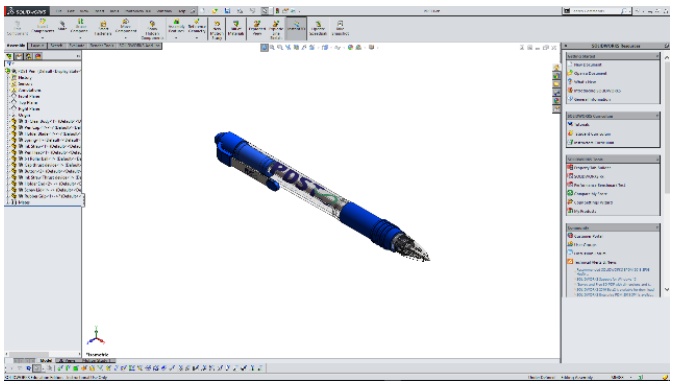
Click on a Part or Assembly a Popup menu will appear

Select the option required

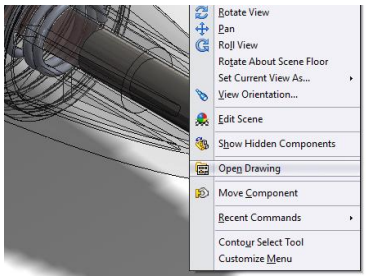








Toggle between the drawing and part/assembly

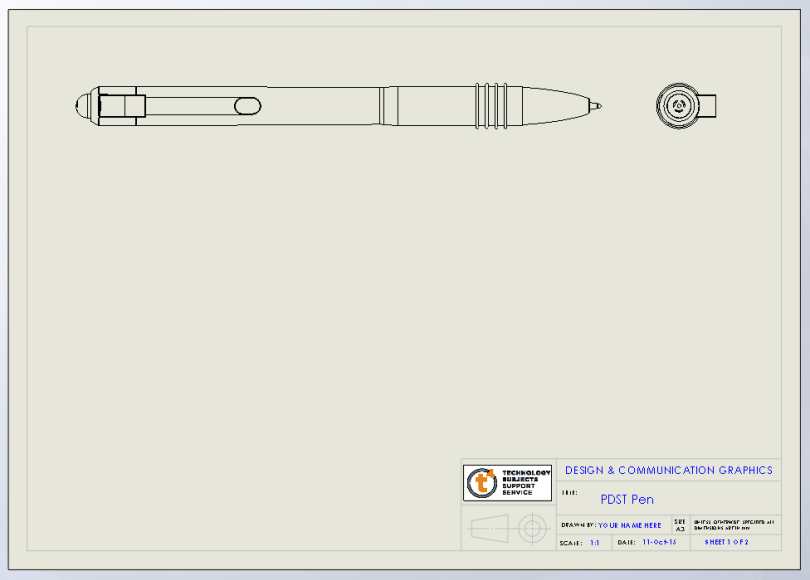


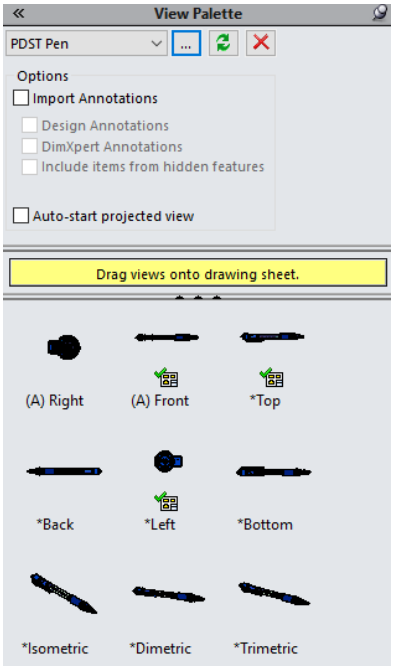
- Use the **Ctrl Key** and **Tab Key** to toggle between the drawing and assembly

- Select **Open Drawing** to return to the drawing

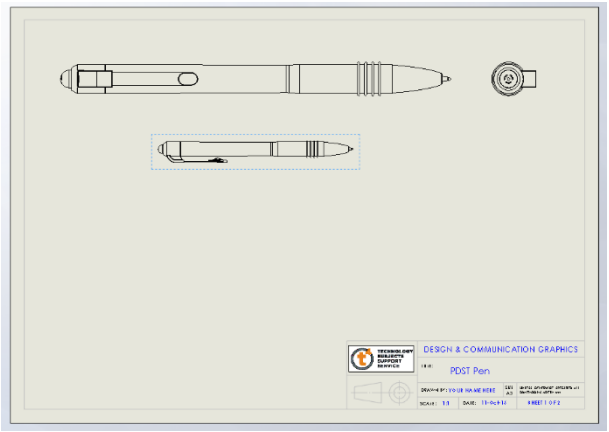
View Palette (Drag & Drop)

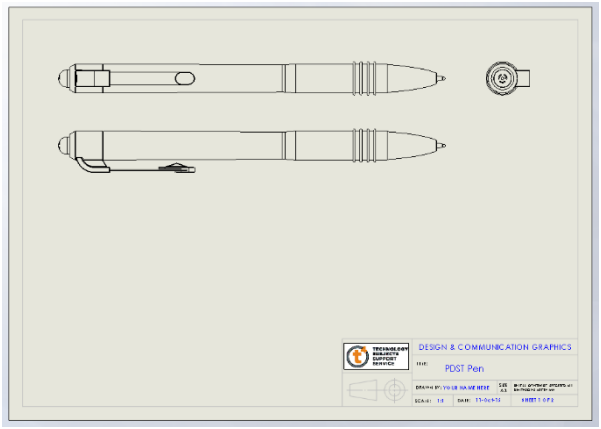
The Model to be used can be selected on the feature manage the on the right hand side





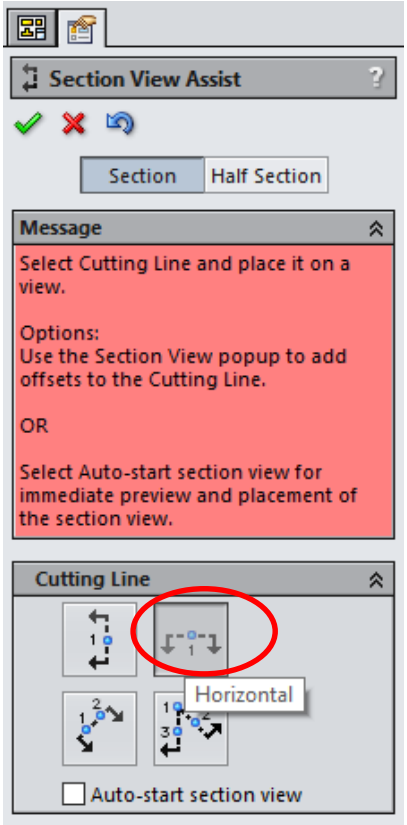
Drag the required view (Top) from the new **View Palette** onto the drawing sheet and edit the view





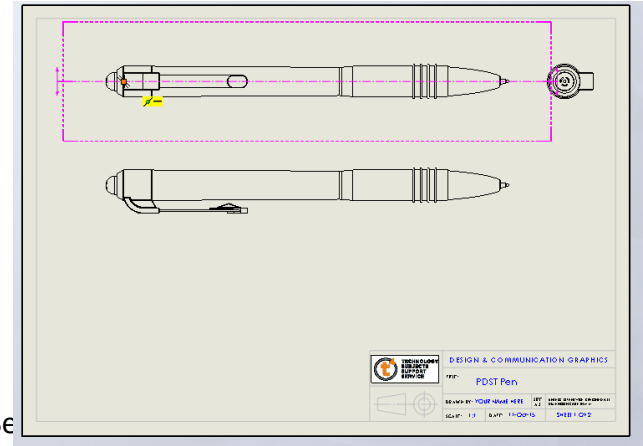
Section View

Select the **Section View** from the View Layout tab

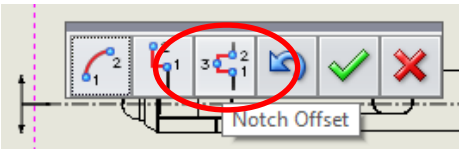


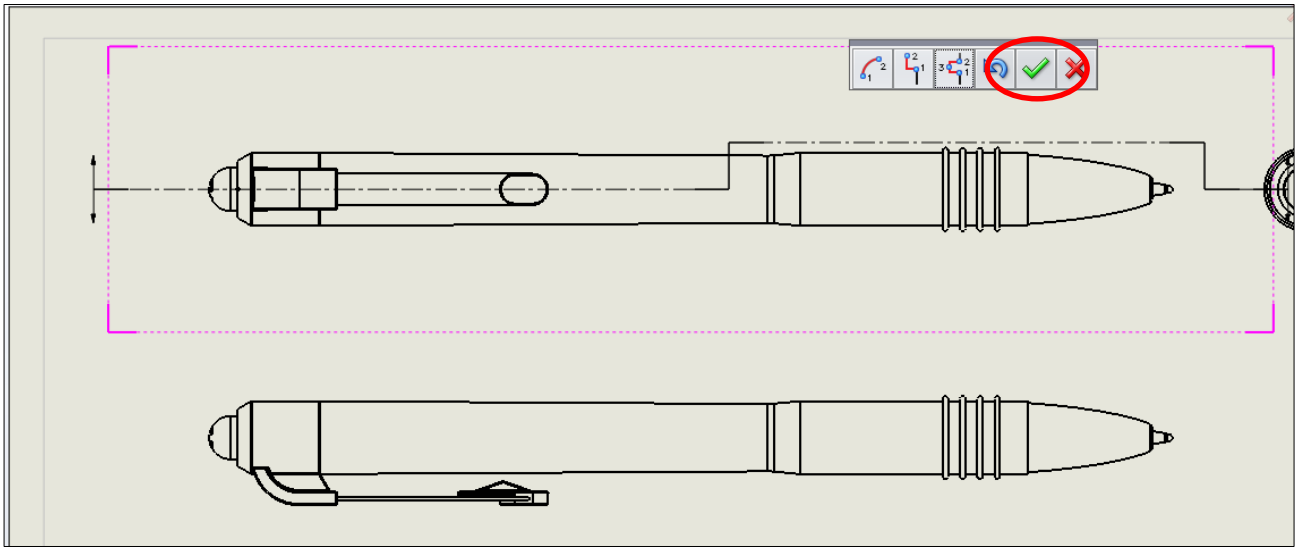
A number of options for the cutting line are given in the Feature manager. Select the Horizontal Option

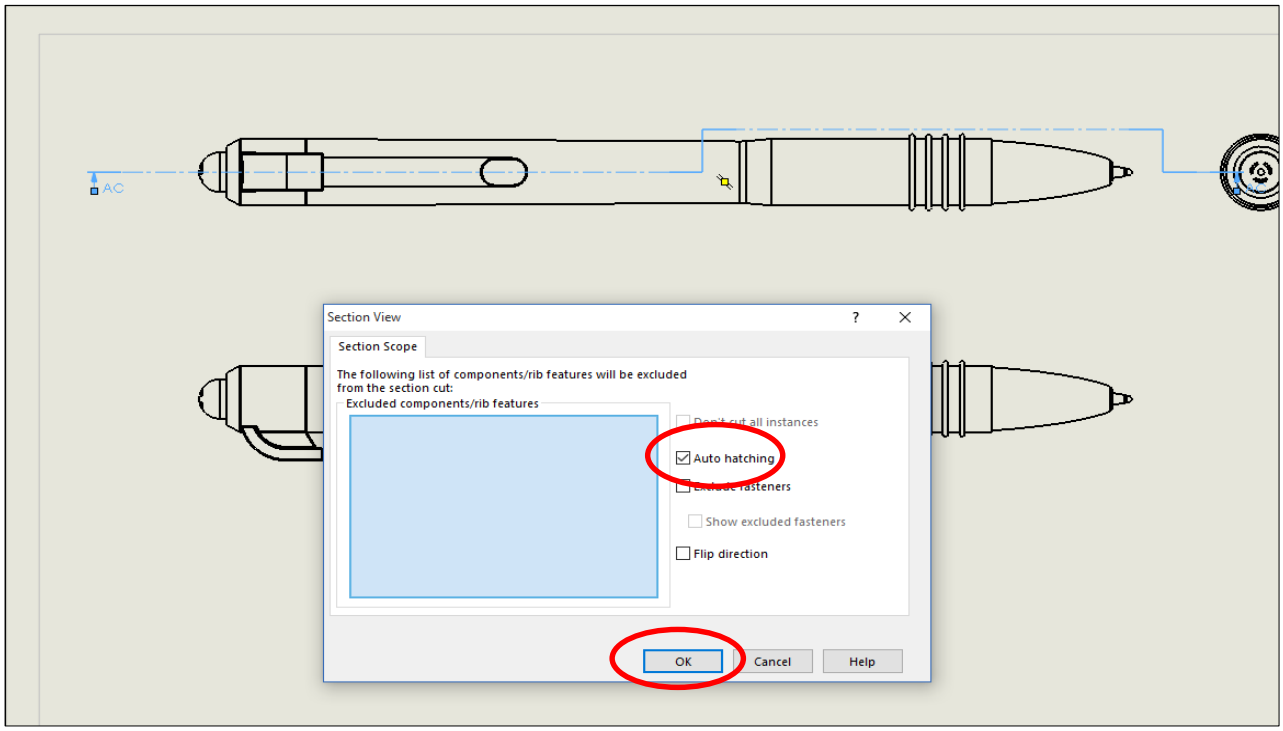
Align the Horizontal cutting line to the centre of the pen

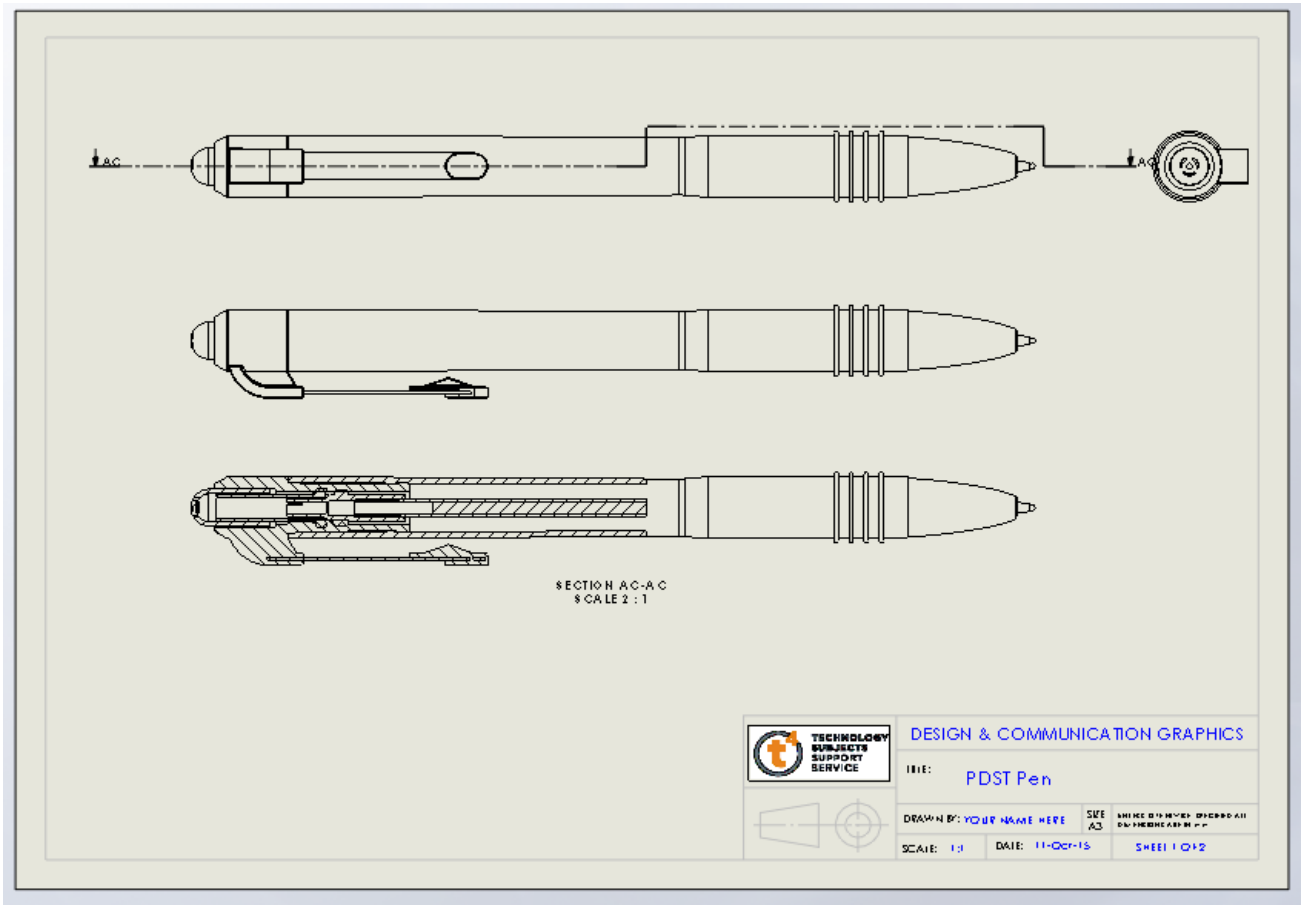


Se



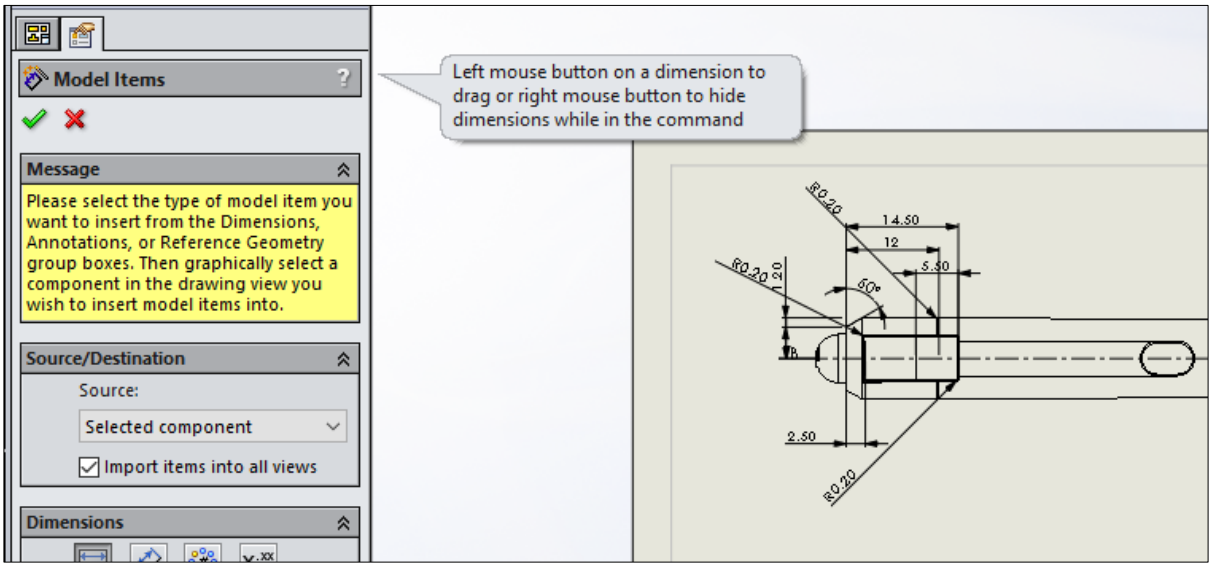



A
A
Section View
Section Scope
The following list of components/rib features will be excluded from the section cut:
Excluded components/rib features
Auto hatching
Show excluded fasteners
Flip direction
OK
Cancel
Help

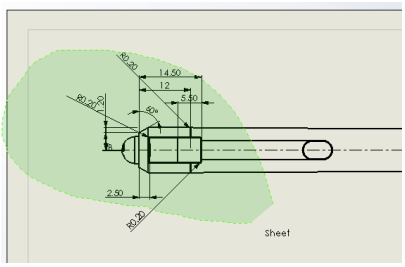

A
A
SECTION A-A
SCALE 2 : 1
TECHNOLOGY SUBJECTS SUPPORT SERVICE
DESIGN & COMMUNICATION GRAPHICS
TITLE:
PDST Pen
DRAWN BY: YOUR NAME HERE
SIZE A3
SCALE: 1:1
DATE: 11-Oct-15
SHEET 1 OF 2

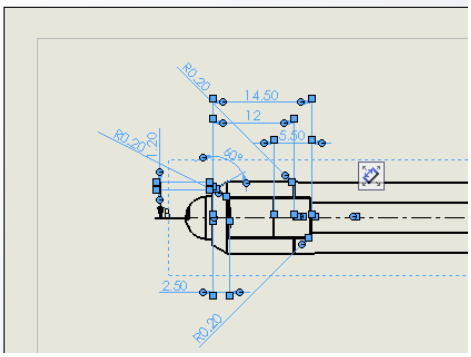
Dimensioning

Select Model Items from the annotation tab. The source will be a component. Select the **Pen Cap** as the component to import the dimensions onto all views

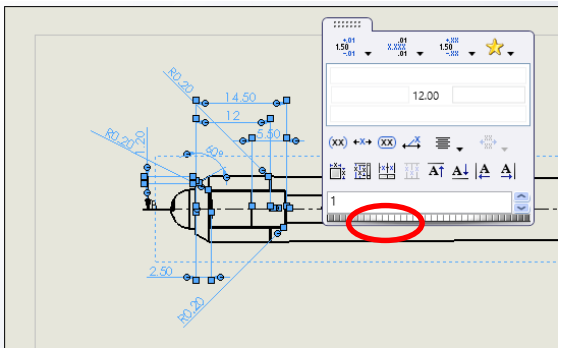


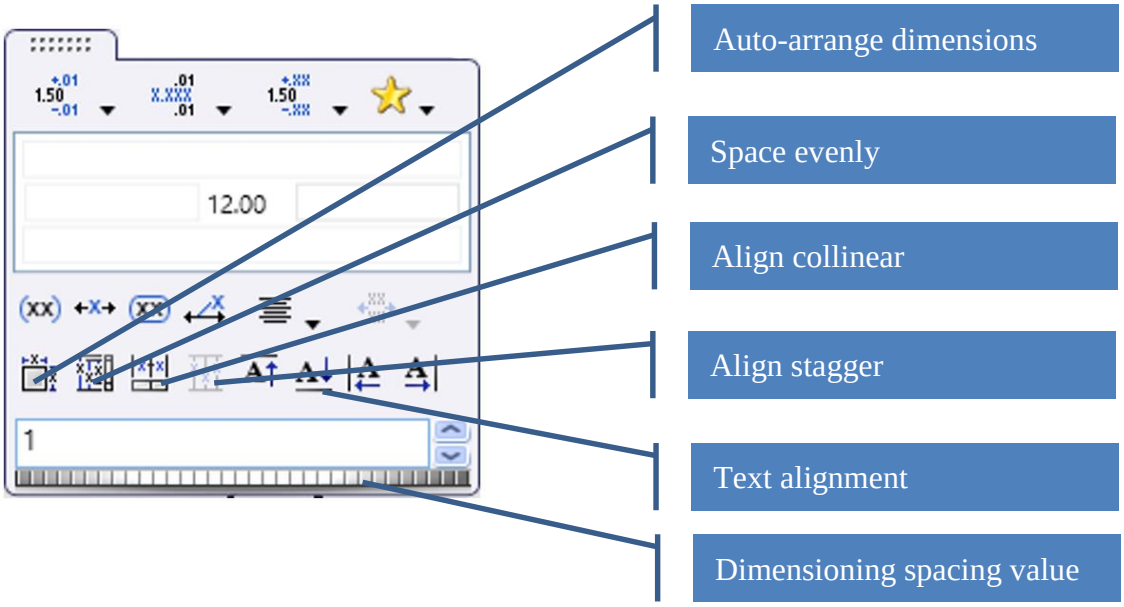
Select the Dimensions using the Lasso Right to Left and click the dimension palette rollover button. The dimension palette offers a number of adjustments that can be made to the dimension format.



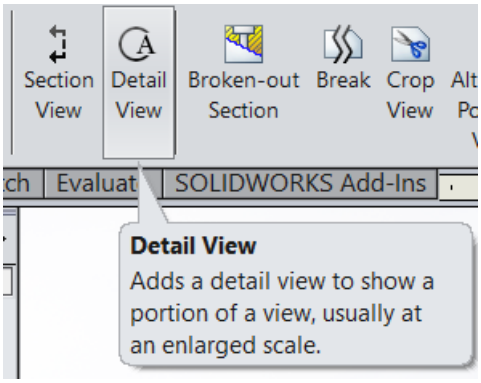


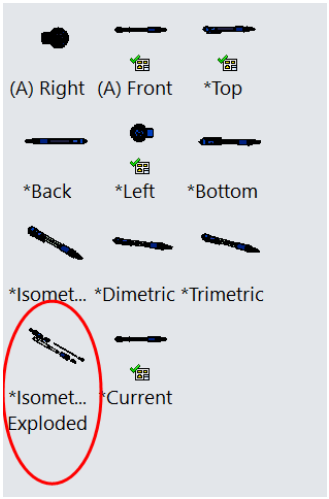
Select the auto-arrange dimension. Other options are available on the Palette



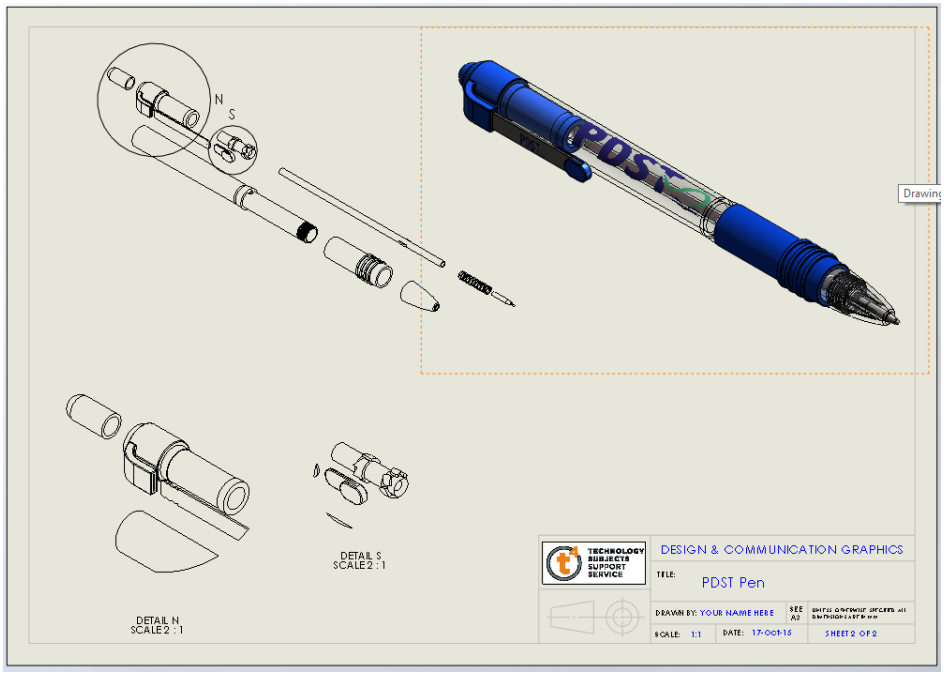


Open sheet 2 and drag and drop the **Exploded** view from the pallette





Create 2 **Detail Views** of some of the components as shown, **scale 2:1**

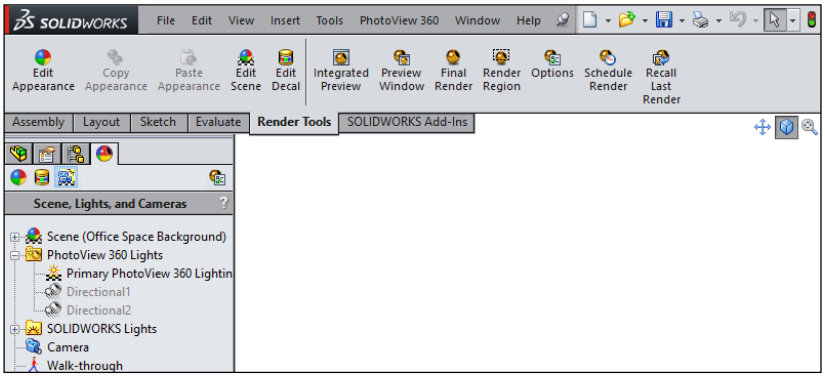


PhotoView 360

PhotoView 360 is now fully integrated into SolidWorks

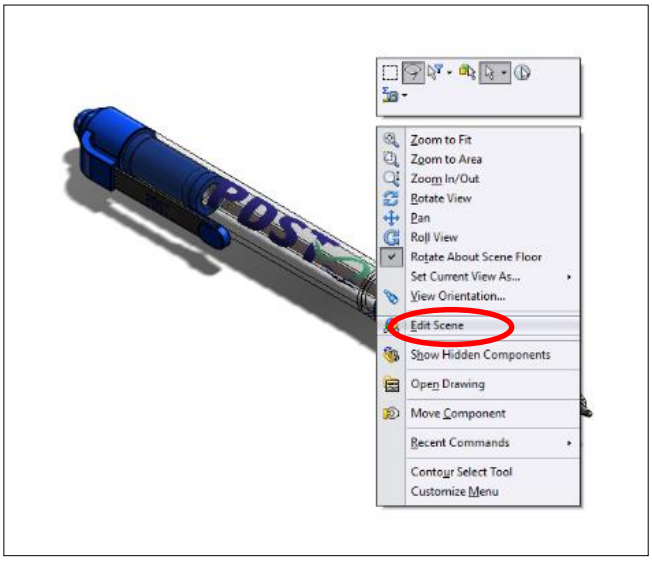
To add in render tool see

http://t4.ie/sw/Photoview360.html

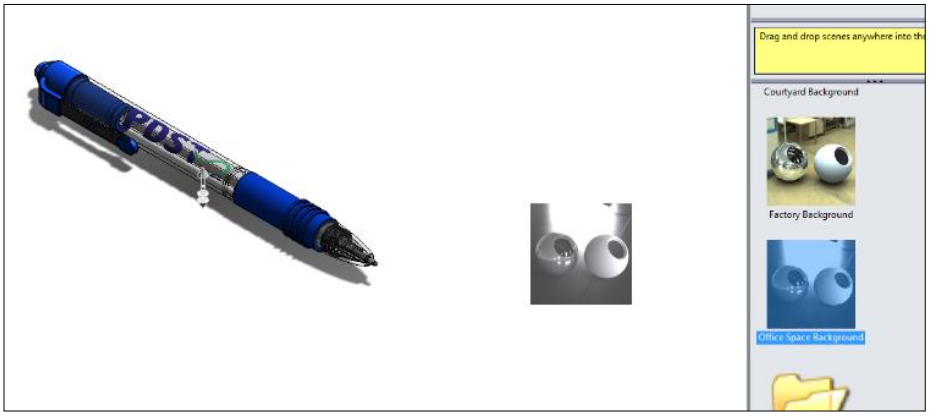


Create an image using an existing background

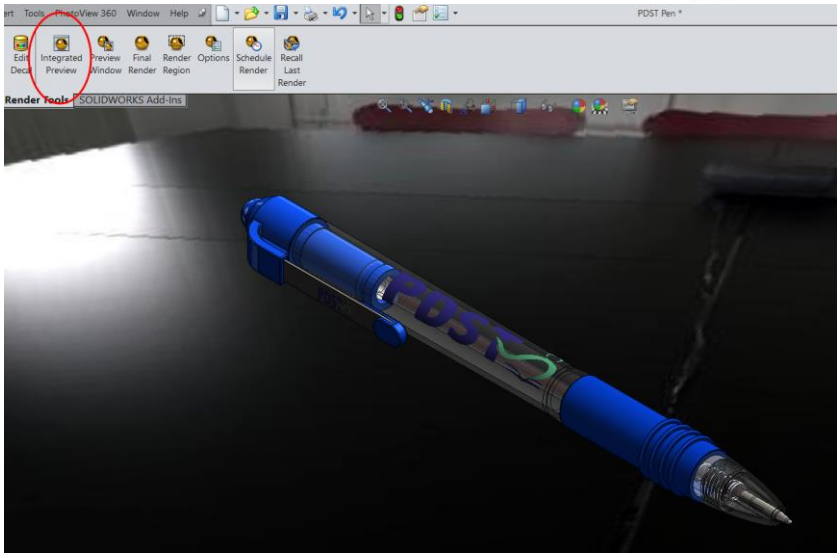
Open the Assembled Pen and select Edit Scene



Drag and Drop the Office Background



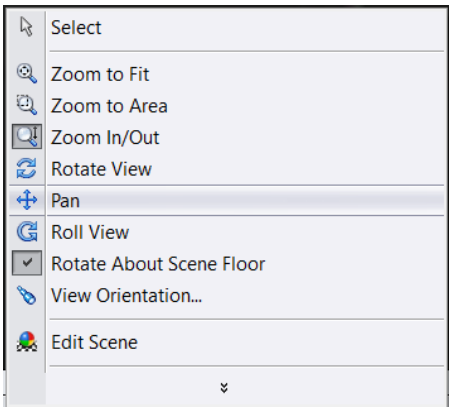
Open the **Integrated Preview Window** and edit the scene



NOTE: Integrated preview slows the responses of your computer, move the model slowly and carefully to edit scene.

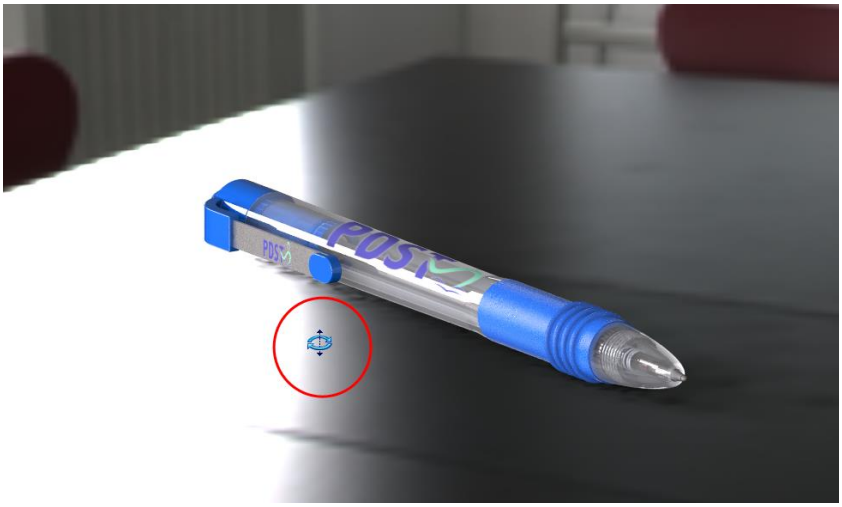
Right click to select the**,** and **Pan, Rotate View** and **Zoom In/Out** commands

The middle mouse button can also be used to rotate, pan and zoom



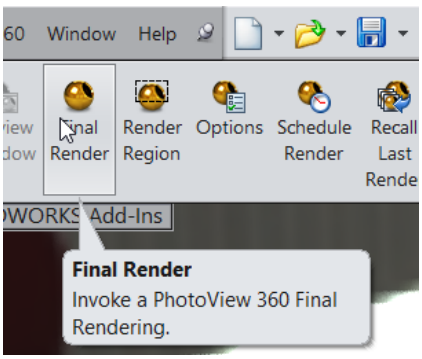
Use **Pan** to move model up, down etc. in scene

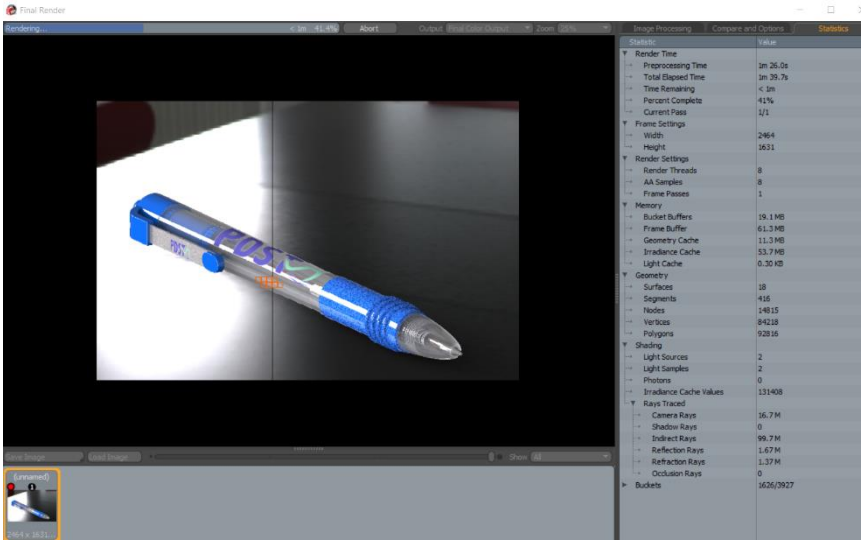
Use **Rotate** to rotate model around the scene



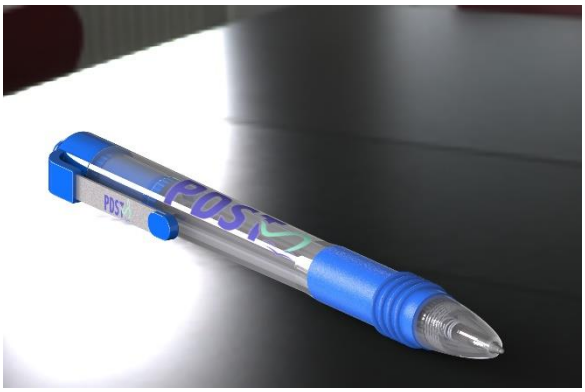
Use the **Zoom In/Out** command to zoom in and out of model

Choose **Final Render** when edits are completed



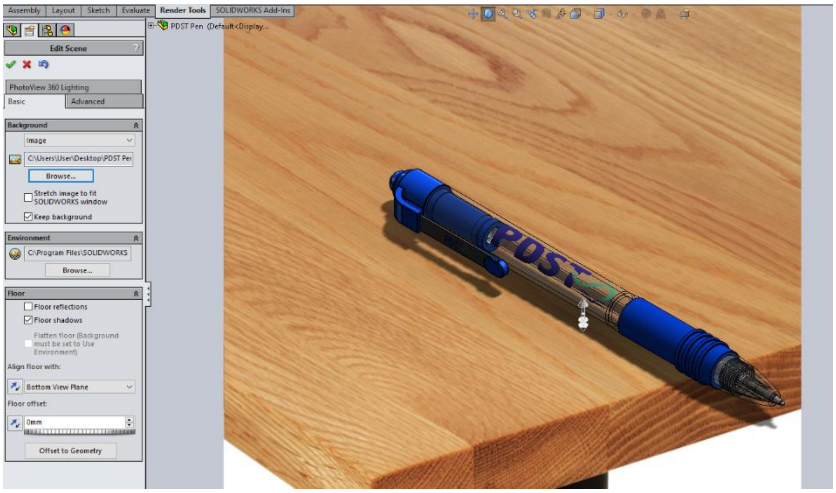


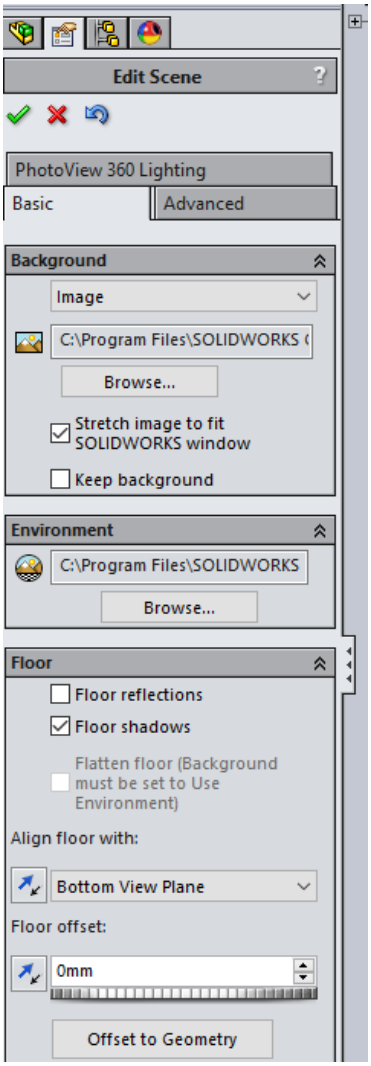
Save the image onto your Memory Key when the rendering is complete. Save as **PDST Pen.**



Importing an image background

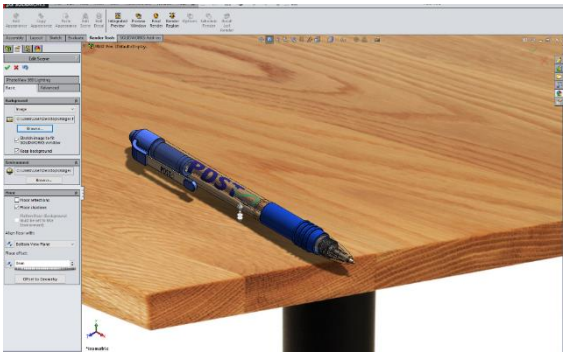
Open the PDST Pen Select **Edit Scene** select background image and browse to the file location, select **Stretch image to fit SOLIDWORKS Window**





Edit the scene and select the required views and do a **Final Render** and save the image

Manipulate the pen assembly and parts to create a series of rendered images displaying details of the pen




PDST


PDST