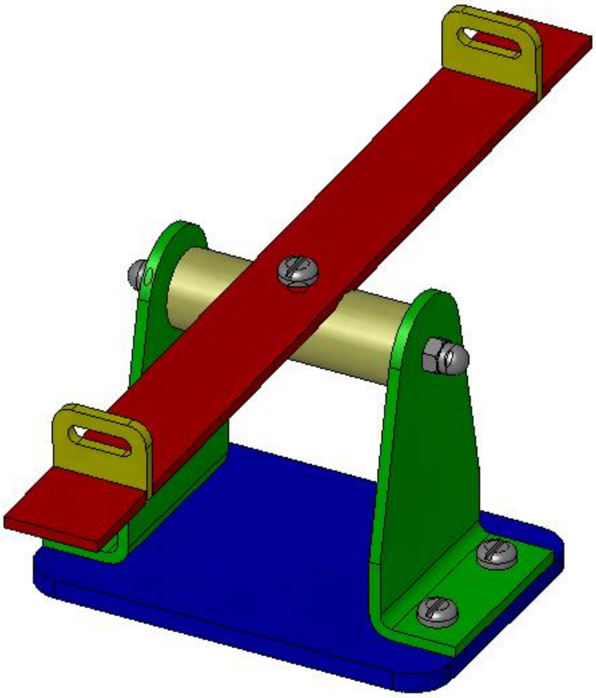
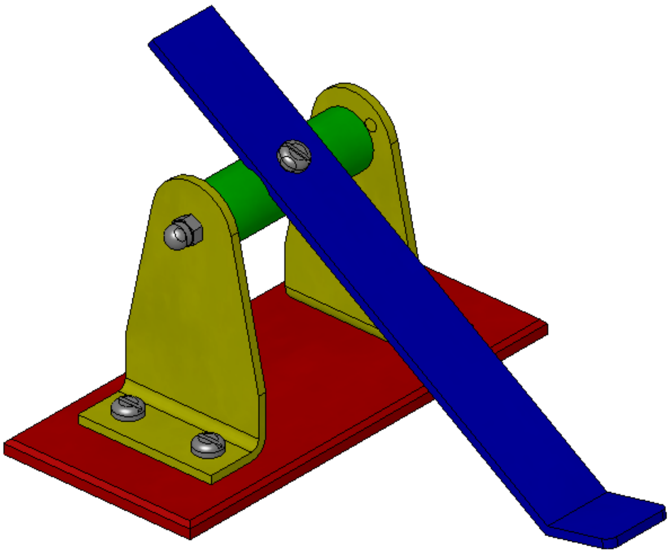
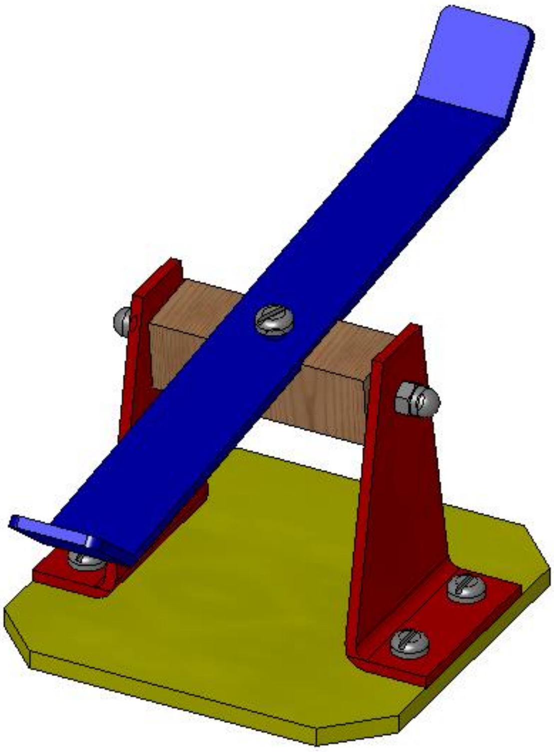
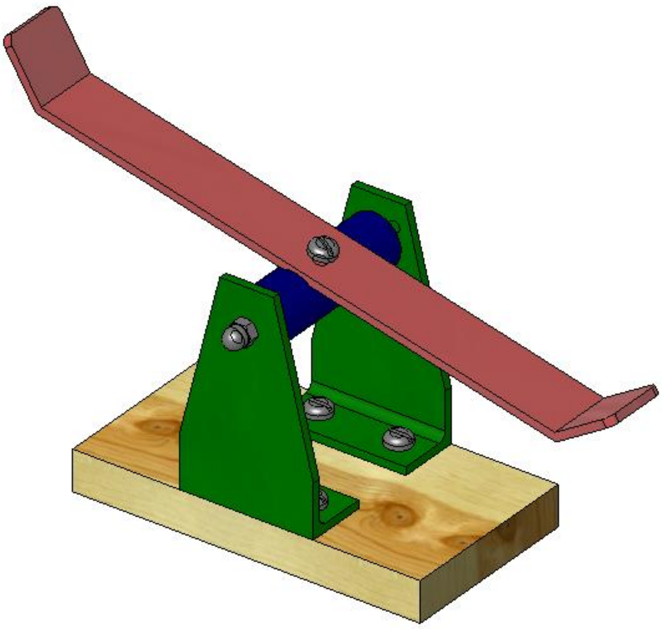
Engineering Technology





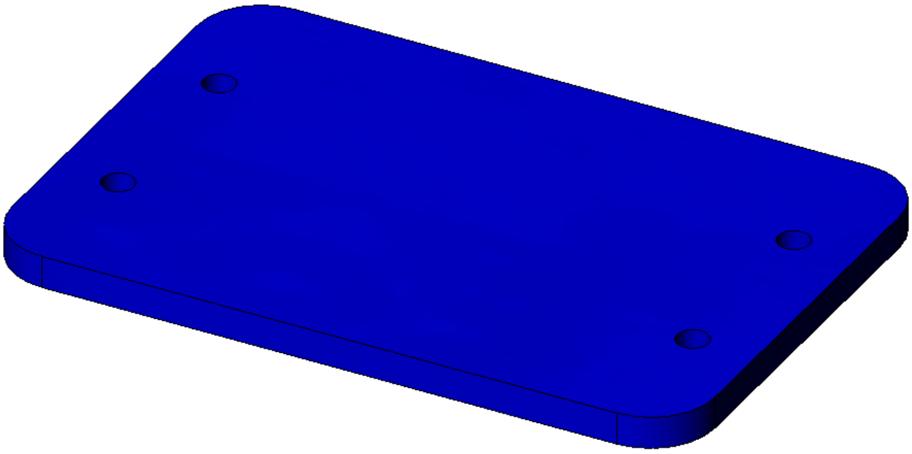
Introduction
to
Parametric Modelling





See Saw Exercise

Part 1 – Base

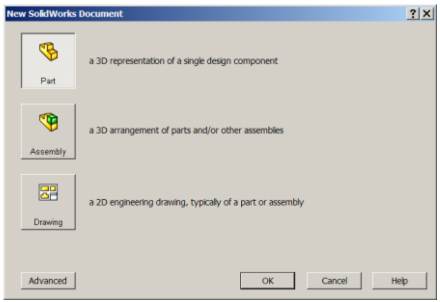


Commands used

This lesson includes **Sketching, *Extruded Boss/Base, Hole Wizard, Cosmetic Thread &Fillet.***

New Part

Click **File, New** on the standard toolbar. Select **Part** from the **New SolidWorks Document** dialog box. Select OK.



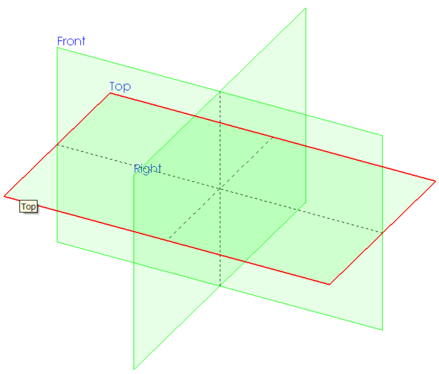
Saving the Part

Select **File, Save as** on the standard toolbar. Save the part in your chosen location as *Base.* A part is identified by its extension **.sldprt* . It is recognised as good practice that a new folder would be used for each project created, and all parts saved to that location.
Continue to save periodically throughout the exercise

Getting started

Choosing a plane

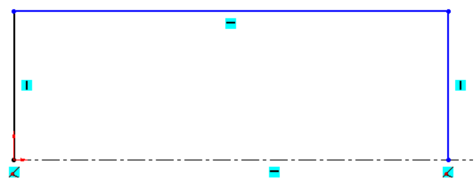
Select **Sketch** on the sketch toolbar. The 3 default sketch planes are displayed. Choose the top (horizontal) plane by moving the cursor over its edge, the periphery of the selected plane will turn red.
The selected plane will rotate to a **normal to** view and the origin will be displayed.



Creating a sketch

Sketch a horizontal **Centre line** through the origin

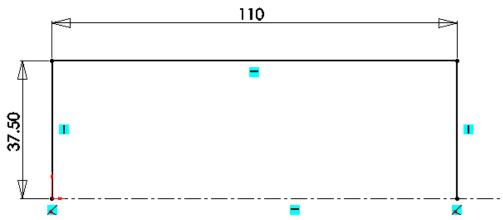
Using the **Line** command,
create a sketch, from the origin as shown,



Note: all lines are either horizontal or vertical indicated by the relations
The **co-incident relation** ensures that the endpoints of the vertical lines will always remain on the origin and the centerline.

Dimensioning the sketch

Select **Smart Dimension** from the sketch toolbar and dimension the sketch as shown.

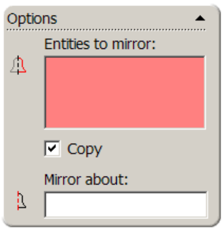


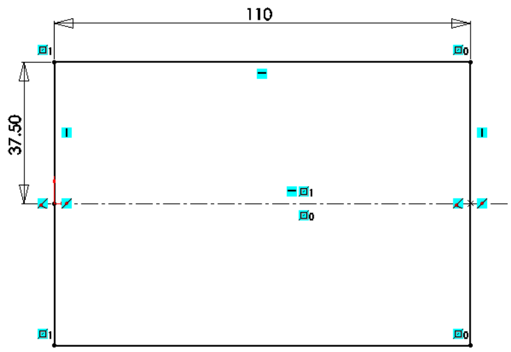
Note – The sketch changes from blue to black when it is fully defined.

Mirror

Choose **Mirror Entities** from the sketch toolbar.

Choose the 3 lines as the **Entities to mirror** and the centre line as the line to **Mirror about.**

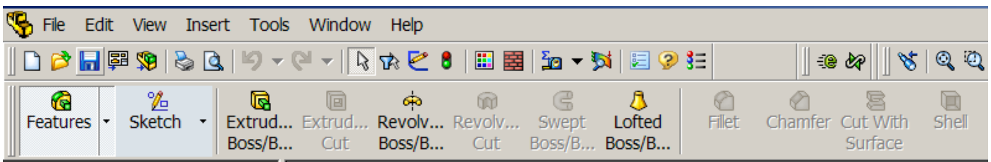




Choose OK

Creating the feature

Select **Features** from the **Command Manager.** The **Features** toolbar has now replaced the **Sketch** toolbar along the top of the screen

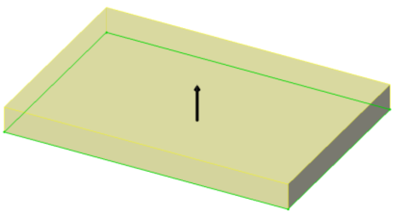


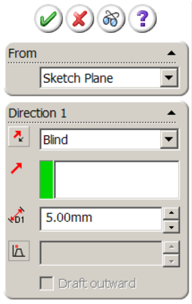
Choose **Extruded Boss/Base,** the sketch rotates to a trimetric view with a preview of the proposed extrude.

Extrude Feature Settings

End Condition = **Blind**

Depth = **5mm**

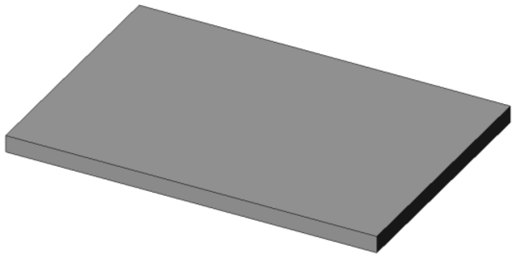




Click **OK** button to create the feature.

Renaming a feature

Select the feature in the **Feature Manager Tree.** Press F2.



The feature name will be highlighted with a flashing cursor on the right hand side. Type the name *base* to replace it.

Creating the holes

The **Hole Wizard** is used to create holes in a solid. It can create both simple and specialised holes using a step by step procedure.

The Hole Wizard requires a face to be pre-selected, not a sketch. A **Normal To** view of this face should be selected in advance.

Where to find it?

Choose the **Hole Wizard** 

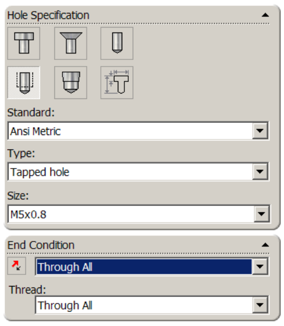
from the **Features Toolbar**
Or from the **Insert Menu** choose **Features, Hole, Wizard…**

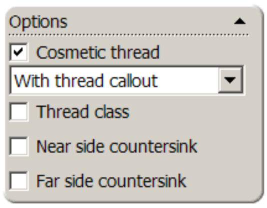
Hole Wizard

Select the Top face of the base feature and click. 

Hole Specification

The **Hole Specification** dialog box appears. Set the properties of the holes as follows:





Type: Tap

Standard: Ansi Metric

Screw Type: Tapped Hole

Size: M5 x 0.8

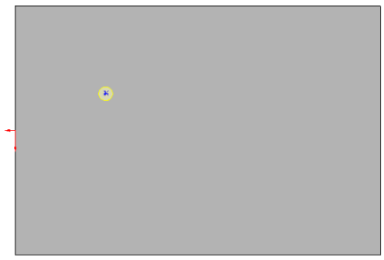
End Condition: Through ALL

Cosmetic thread: Yes
With thread callout

Click the **Positions** tab

Positioning of the holes

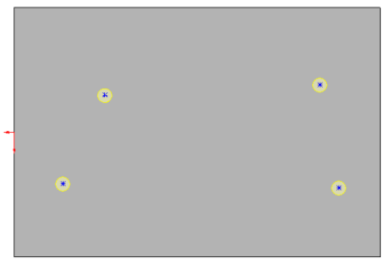
A point and hole preview is placed on the selected face near where you selected it.



Multiple instances of the same hole may be inserted in different positions on the same part by selecting further points on the surface.

Further points

Add in 3 further points, as shown, by clicking on the surface.

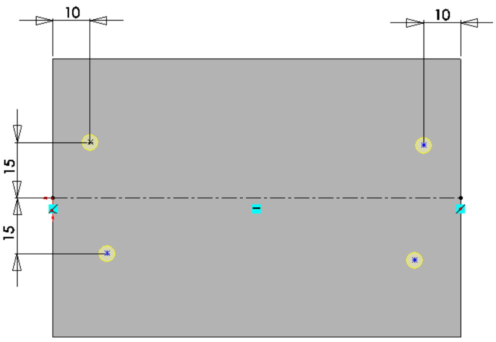


Turn off the **Point** command by pressing **Esc** on the keyboard.

Adding dimensions

Sketch a horizontal centerline from the origin

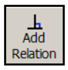
Add dimensions between the model edges, the centreline and the points as shown.



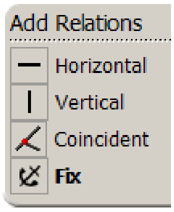
Adding Relations

Add Relations is used to create a geometric relationship between sketch entities, such as collinear, horizontally aligned or concentric.

Where to find it

On the sketch toolbar **click Add Relation** 

Or select the sketch entities and select the appropriate relation from the **Add Relations** section of the property manager.

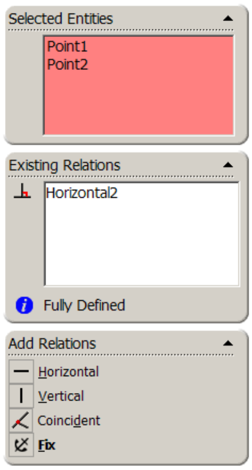


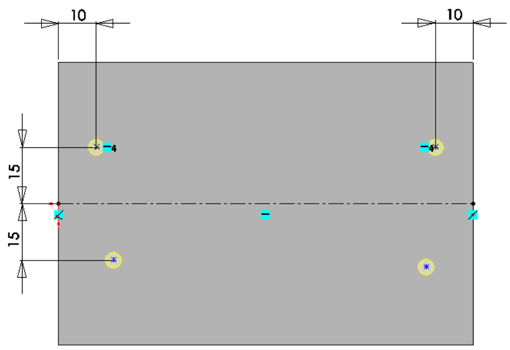
Add a relation

Hold down the **Ctrl** key and select the centre of the two holes as shown below. The property manager displays the relations valid for the selected geometry

Choose **Horizontal.** Choose OK

The two hole centres become horizontally aligned.

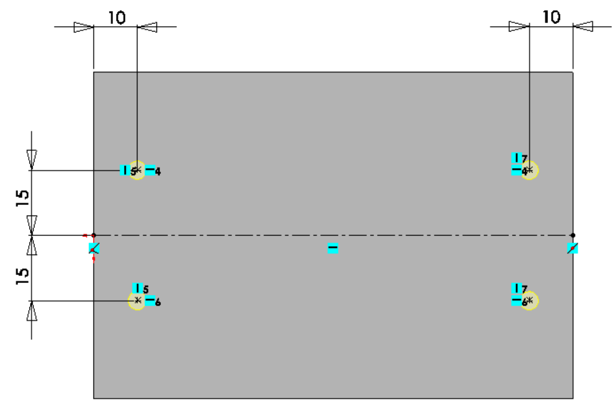




Further Relations

Add further horizontal and vertical relations between the holes to align them as shown opposite.

Choose OK to exit
Hole Wizard



Introducing Fillet

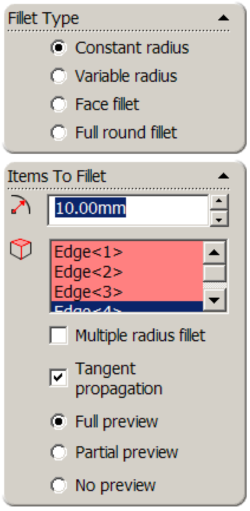
Fillets are generally added to the solid rather than the sketch and are hence referred to as **applied features.**

Where to find it

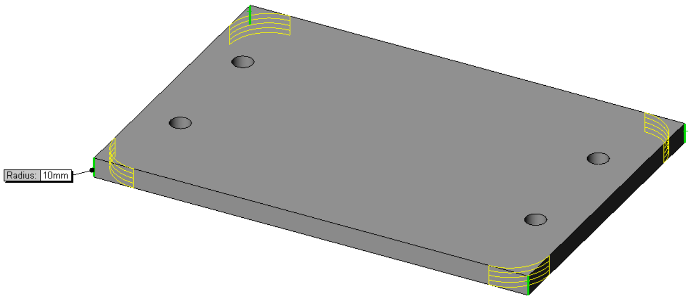
Select the **Fillet tool** from the features toolbar *or* from the **Insert** menu, select **Features/Fillet/Round…**

Insert Fillet

Select the **Fillet** option. The fillet options appear in the property manager.



Set the **Radius** value to **10mm**. Select **Full Preview**

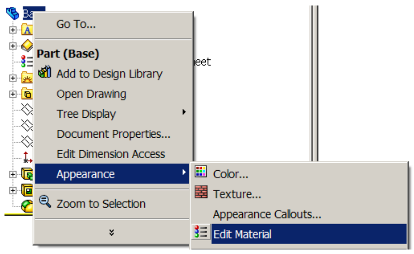


Choose the edges indicated and click OK.

Edit Material

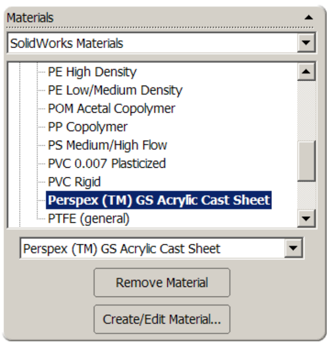
The **Edit Material** is used to add and edit material associated with a part. The material is used for calculations which rely on properties eg mass.

Right click on the part name at the top of the design tree and choose **Appearance, Edit Material…**



Choose **Perspex Acrylic Cast Sheet** under the heading **Plastics.**

Choose OK



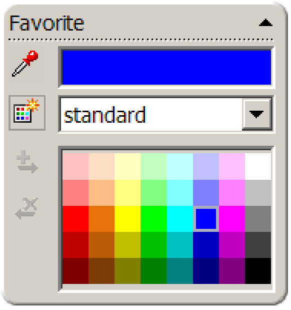
Adding Colour

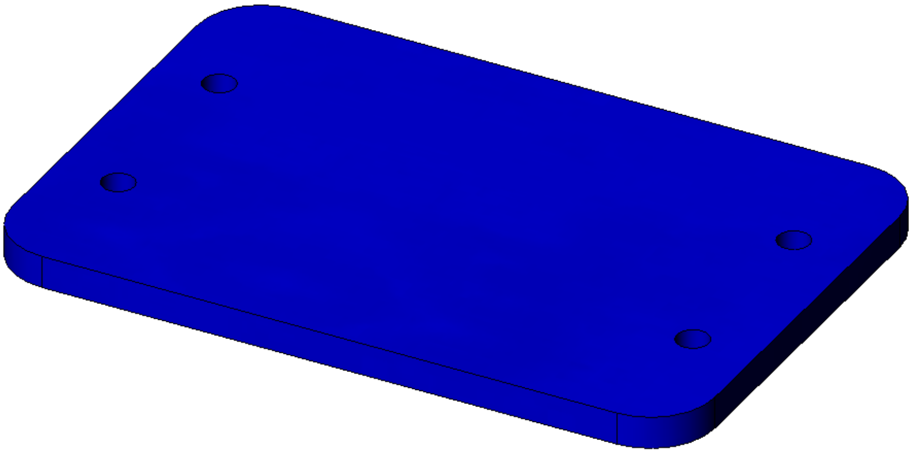
Adding colour to the model can make it look more realistic.

Right click on the part name at the top of the design tree and choose **Appearance, Color…**

Choose a colour from the swatch.

Choose OK.

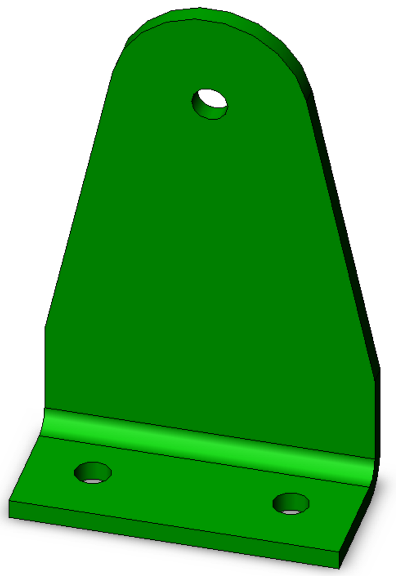




Save and close

Click **Save** to save your work and click **File, Close** to close the part.

Part 2 – Support

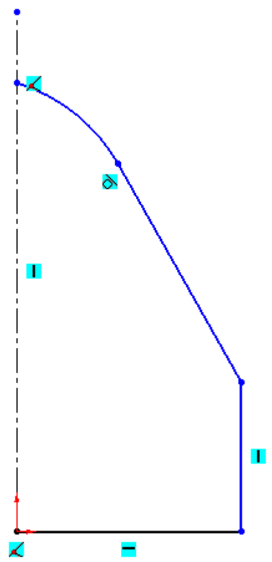


Commands used This lesson includes **Sketching, *Extruded Boss/Base, Extruded Cut* Hole Wizard *& Fillet.***

Saving the part Save the part in the folder created earlier using the name ***Support***

Initial Sketch Create a sketch on the **Right plane.**

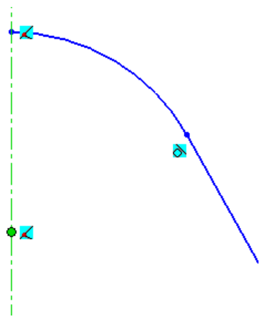
Using **Centre Line, Line** and **Tangent Arc,** create a sketch which resembles that shown opposite.



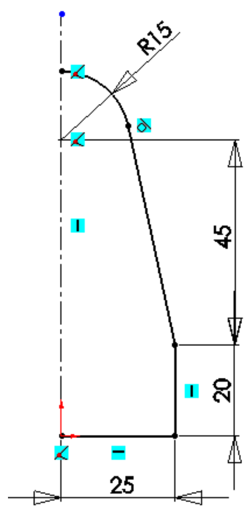
Note the automatic relations ie vertical, co-incident etc.

Add Relations Choose **Add Relations.**

Create a **Co-Incident Relation** between the centre of the arc and centre line

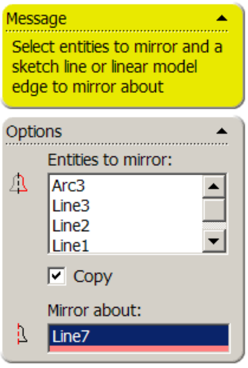


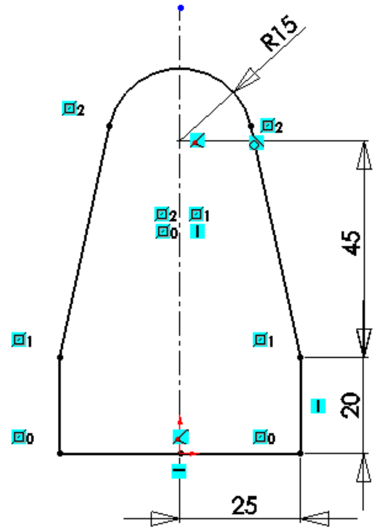
Adding Dimensions **Smart Dimension** the sketch as shown.



Mirror

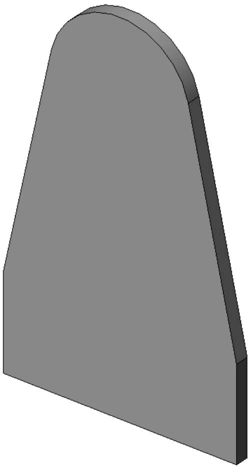
Mirror the fully defined sketch across the centre line using **Mirror** from the **Sketch Toolbar.**





Extruded Boss/Base

Extrude the sketch using the **Blind End Condition** and a **Depth** of **3mm**



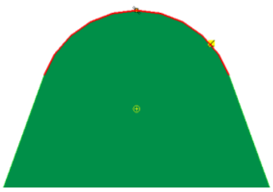
Sketching on a face

Select the front face, choose **Sketch** from the **Sketch Toolbar.**

Choose **Normal To** from the **View** toolbar

Creating the hole

Choose **Circle** from the sketch toolbar and sketch a circle concentric with the tangent arc.



To ensure concentricity, move the cursor to the circumference of the circle as shown, without clicking. The centre becomes highlighted. Choose the centre and drag the circle

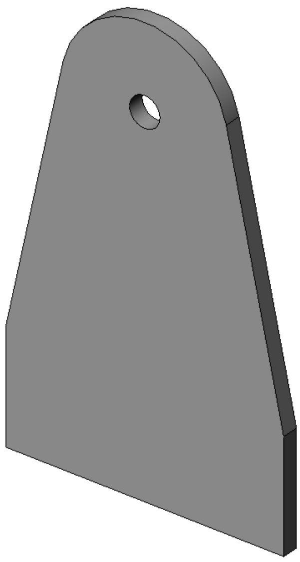
Dimension the circle **Ø5.5.**

Choose **Extruded Cut** from the features toolbar

Select the Ø5.5 circle.

Cut Extrude Feature Settings
End Condition = **Through All**

Choose OK.

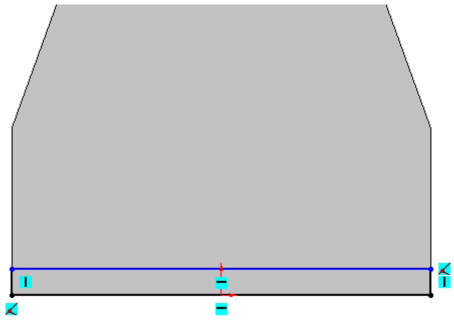


Adding the base feature

Sketch a rectangle on the face ensuring that the coincident relations shown are automatically created.

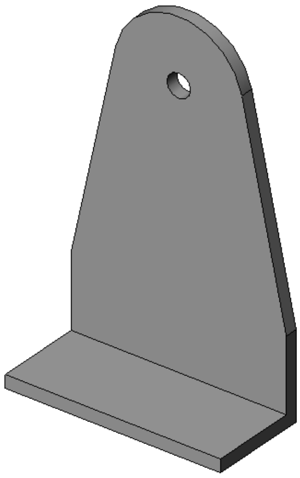
The rectangle must be co-incident with the left hand corner and the right-hand edge.

Smart Dimension the rectangle to a height of **3mm.** It is now **fully defined.**



Extruded Boss/Base

Extrude the sketch using the **Blind End Condition** and a **Depth** of **20mm**



Hole Wizard

Select the top surface of the base and choose **Top** view

Select **Hole Wizard** 

from the features toolbar

The **Hole Specification** dialog box appears. Set the properties of the holes as follows:

Type: Hole

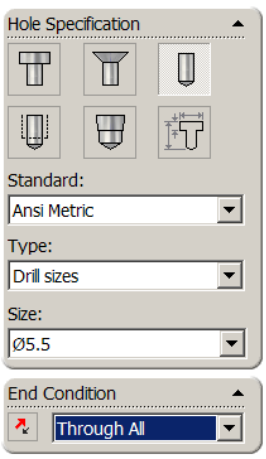
Standard: Ansi Metric

Screw Type: Drill sizes

Size: Ø5.5

End Condition: Through ALL

Click the **Positions** tab

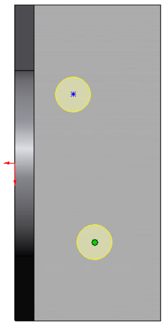


Positioning

One hole position will be displayed, add another by clicking on the face.

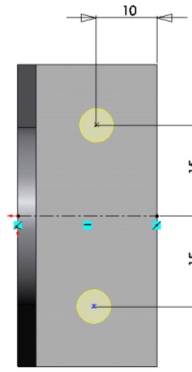
Press ESC to turn off point command.

Add a horizontal centerline from the origin.



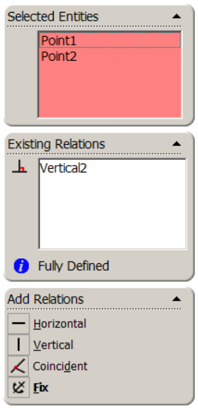
Dimensioning

Add <u>only</u> the dimensions shown opposite



Add Relations

Hold down the **Ctrl** key and select the centre of the two holes. The property manager displays the relations valid for the selected geometry



Add a Vertical Relation, choose OK

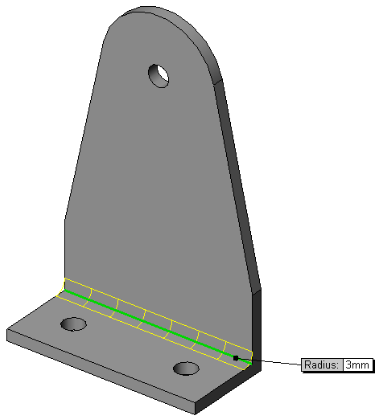
To exit Hole Wizard choose OK

Fillet

Select the **Fillet** option. The fillet options appear in the property manager.

Set the **Radius** value to **3mm**.

Choose the internal edge and select OK.



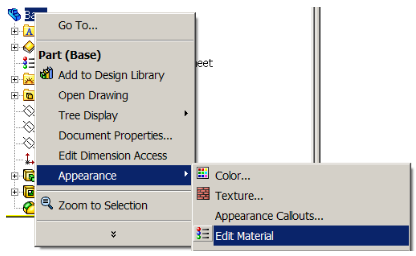
Repeat the procedure to add a **6mm** fillet to the external edge.

Edit Material

Right click on the part name at the top of the design tree and choose **Appearance, Edit Material…**

Choose **Perspex Acrylic Cast Sheet** under the heading **Plastics**

Select **OK.**

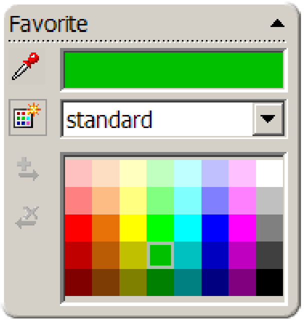


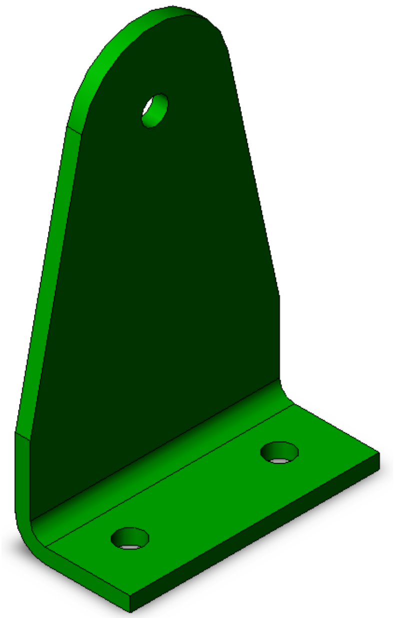
Adding Colour

Right click on the part name at the top of the design tree and choose **Appearance, Color…**

Choose a colour from the swatch.

Choose OK.

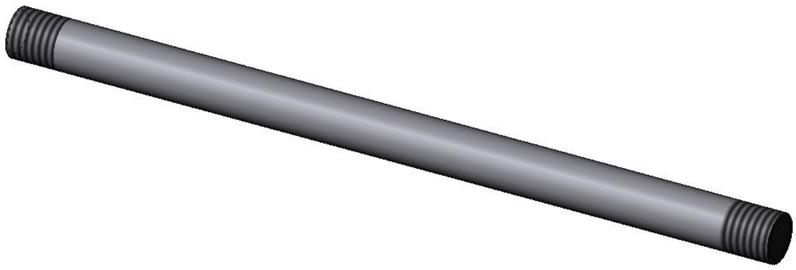




Save and close

Click **Save** to save your work and click **File, Close** to close the part.

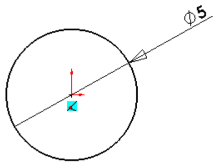
Part 3 – Axle



Commands used This lesson includes **Sketching, *Extruded Boss/Base, Cosmetic Thread.***

Saving the part Save the part in the folder created earlier using the name ***Axle***

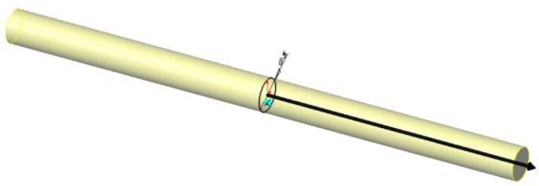
Initial Sketch Create a sketch on the **Right plane.**



Sketch a circle **co-incident** with the **origin**.

Smart Dimension as shown.

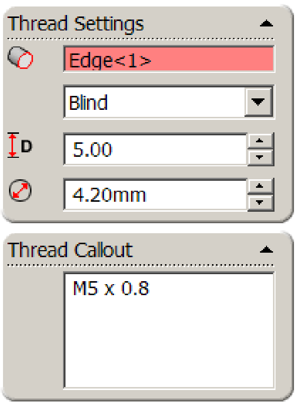
Extrude Boss/Base **Extrude** the sketch using the **Mid Plane End Condition** and a **Depth** of **80mm**



Cosmetic Threads A **Cosmetic Thread** represents the inner diameter of a thread on a boss or the outer diameter of a thread on a hole and can include a **thread callout** on a drawing.

Where to find it? Choose **Insert, Annotations, Cosmetic Thread…** from the drop down menu.

Cosmetic Thread Settings The **Cosmetic Thread Settings Box** appears.



Choose the two ends of the bar as **Circular Edges**

Add **5** as the **thread depth**

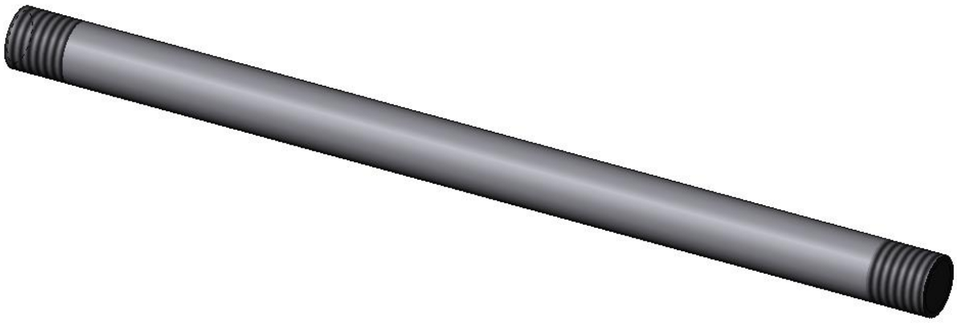
Insert **4.2mm** as the **minor diameter**

Type **M5 x 0.8** as the thread callout

Choose OK. 

Thread callouts will only appear in drawing documents.

Cosmetic Thread



Edit Material

Right click on the part name at the top of the design tree and choose **Appearance, Edit Material…**

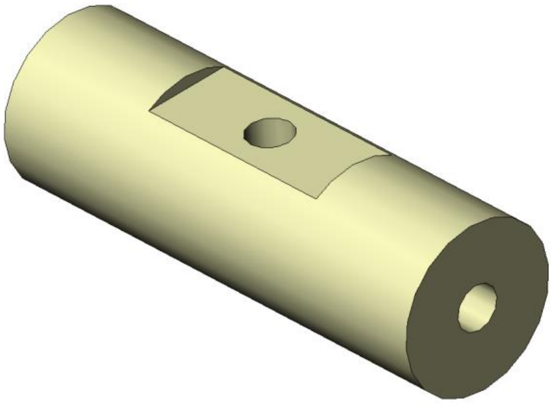
Choose **Alloy Steel** under the heading **Steel**

Select **OK.**

Save and close

Click **Save** to save your work and click **File, Close** to close the part.

Part 4 – Swivel



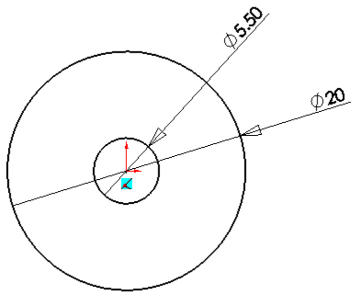
Commands used This lesson includes **Sketching, *Extruded Boss/Base, Extruded Cut & Hole Wizard.***

Saving the part Save the part in the folder created earlier using the name ***Swivel.***

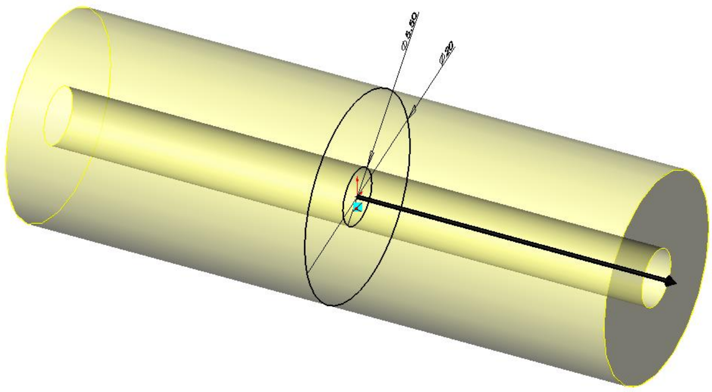
Initial Sketch Create a sketch on the **Right plane.**

Sketch 2 circles **co-incident** with the **origin**.

Smart Dimension as shown.



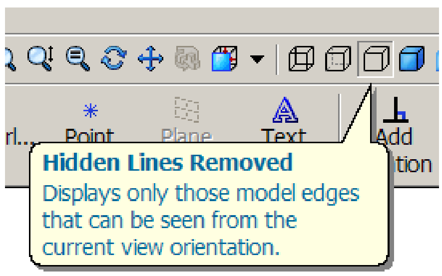
Extrude Boss/Base **Extrude** the sketch using the **Mid Plane End Condition** and a **Depth** of **60mm**



Select **OK.** 

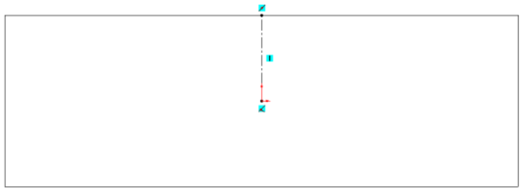
Removing the flat Create a sketch on the **front plane**
Choose **Normal To View** from the View Toolbar

Choose **Hidden Lines Removed.**

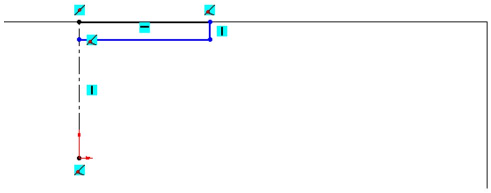


Adding the sketch entities.

Add in a centre line **coincident** with the **origin** and **the edge of the part** with an **automatic vertical relation**

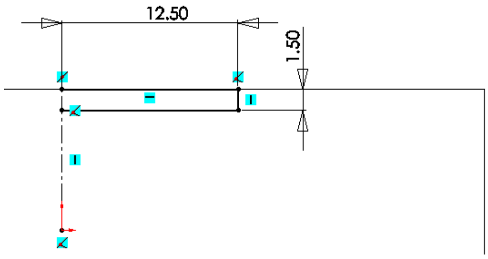


Sketch the lines on the plane ensuring that the **Co-Incident, Horizontal and Vertical Relations** shown are automatically created.



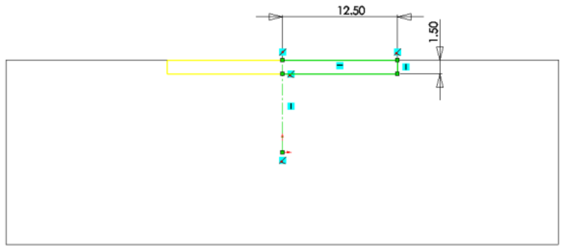
Smart Dimension

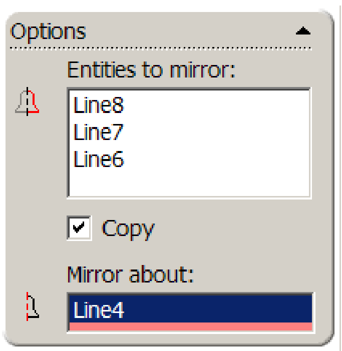
Smart Dimension using the given dimensions. It is now **fully defined.**



Mirror

Mirror the fully defined sketch across the centre line using **Mirror Entities** from the **Sketch Toolbar.**

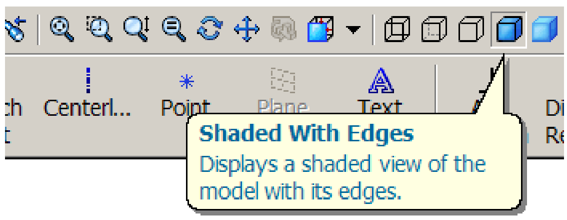


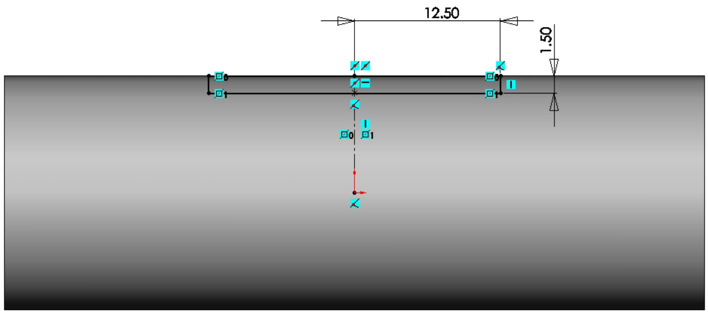


Select **OK.** 

Shaded With Edges

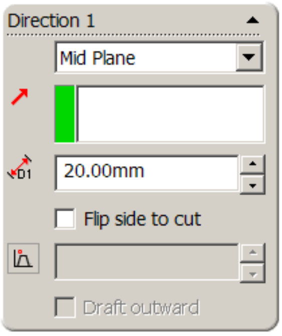
Choose **Shaded With Edges** from the **View Toolbar**



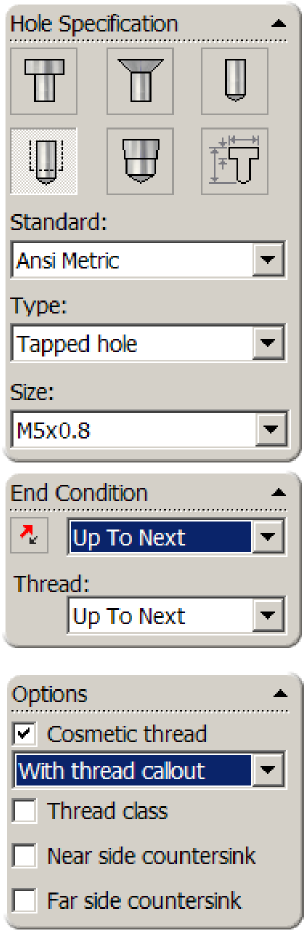


Choose **Trimetric View** from the **View Toolbar**

Extruded Cut — Use **Extruded Cut Feature** to remove the cut using **Mid Plane** End Condition and a **Depth** of **20mm**



Select **OK.** 



Adding the M5 Hole — Select the flat surface on top.

Hole Wizard — Select **Hole Wizard** from the features toolbar

Set the properties of the hole as follows:

Type: Tap

Standard: Ansi Metric

Screw Type: Tapped Hole

Size: M5 x 0.8

End Condition: Up to Next

Cosmetic Thread: Yes
With thread callout

Click the **Positions** tab

Positioning the hole — Press **ESC** to avoid adding further holes

Adding Relations — Holding down the ctrl key select the origin and hole centre.
Add a co-incident relation between them.

Select **OK** 

Choose OK to exit the hole wizard.

Edit Material

Right click on the part name at the top of the design tree and choose **Appearance, Edit Material…**

Choose **Nylon 6/10** under the heading **Plastics**

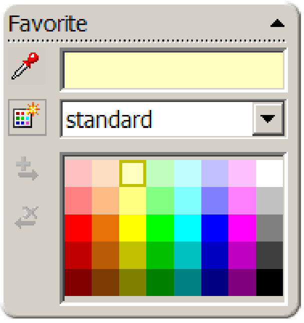
Select **OK.**

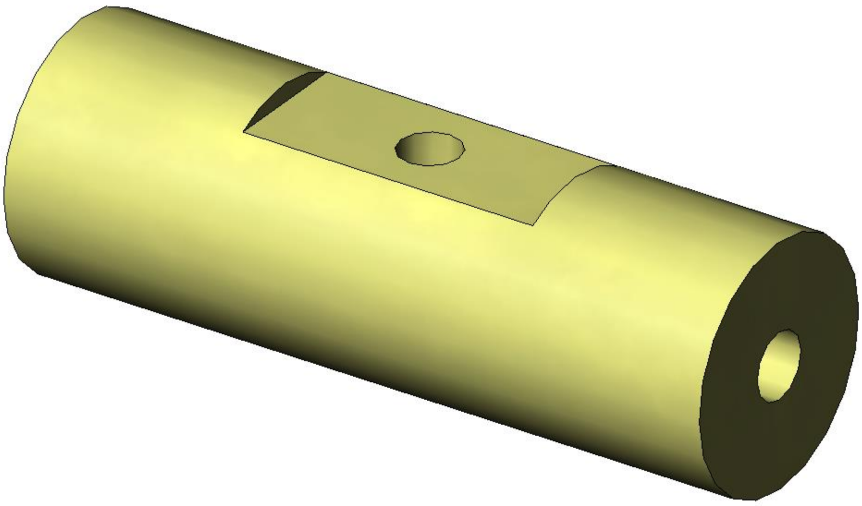
Adding Colour

Right click on the part name at the top of the design tree and choose **Appearance, Color…**

Choose a colour from the swatch.

Choose OK.

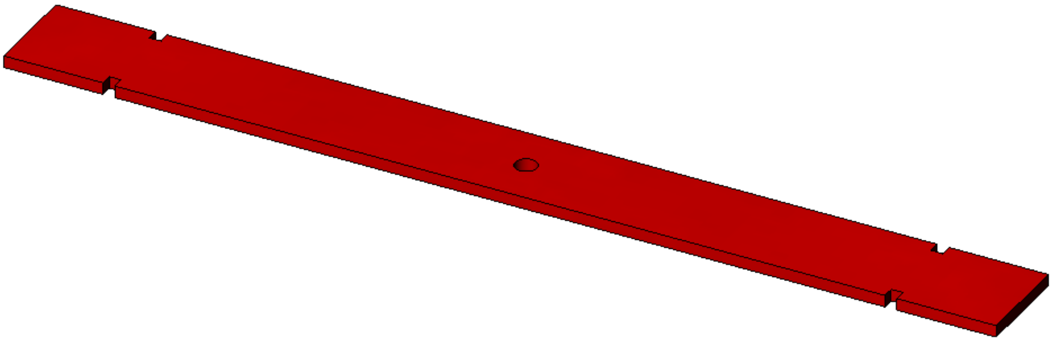




Save and close

Click **Save** to save your work and click **File, Close** to close the part.

Part 5 – Board



Commands used This lesson includes **Sketching, *Extruded Boss/Base.***

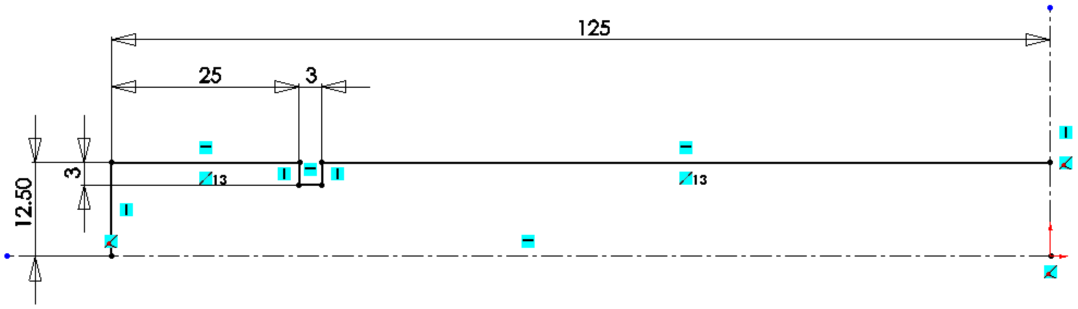
Saving the part Save the part in the folder created earlier using the name ***Board***

Initial Sketch Create a sketch on the **top plane**

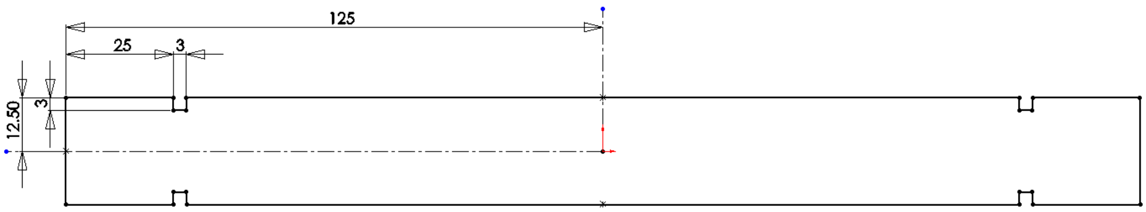
Add a vertical and horizontal centre line through the origin.

Sketch Relations Create the sketch as shown, using only the dimensions given and the sketch relations indicated.

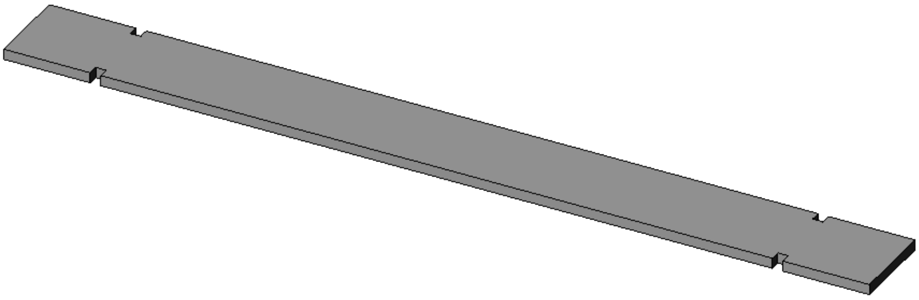
A co-linear sketch relation must be added between the two horizontal lines on top.



Mirror Entities **Mirror** the sketch, initially across the horizontal centre line, then across the vertical centre line.

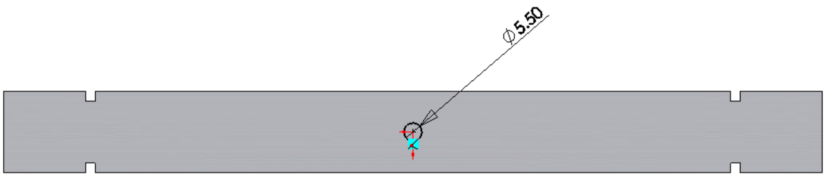


Extruded Boss/Base — **Extrude** the sketch using the **Blind End Condition** and a **Depth** of **3mm.**



Adding the hole — Create a sketch on the top face.
Choose **Top View** from the **View Toolbar**

Sketch a **Ø5.5 Circle coincident** with the **origin.**



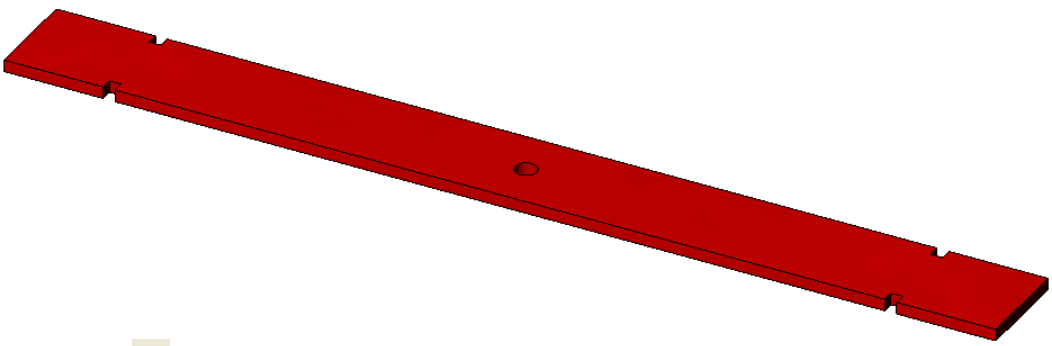
Extruded Cut — Extrude Cut the circle with a **Through All End Condition.**

Edit Material — Right click on the part name at the top of the design tree and choose **Appearance, Edit Material…**

Choose **Perspex Acrylic Cast Sheet** under the heading **Plastics**

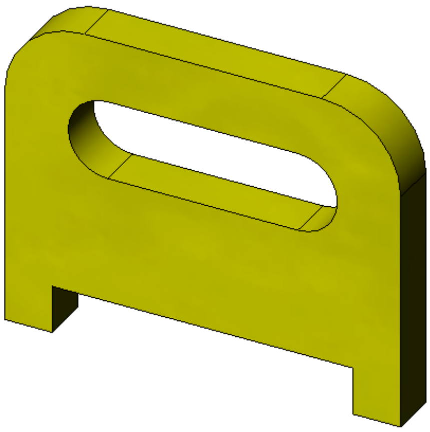
Select **OK.**

Adding Colour — Choose a colour for the part as before.



Save and close — Click **Save** to save your work and click **File, Close** to close the part.

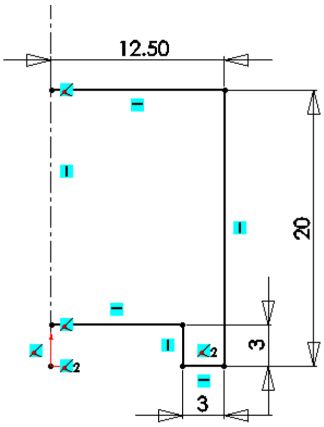
Part 6 – Handle



Commands used This lesson includes **Sketching, *Extruded Boss/Base* & *Fillet.***

Saving the part Save the part in the folder created earlier using the name ***Handle***

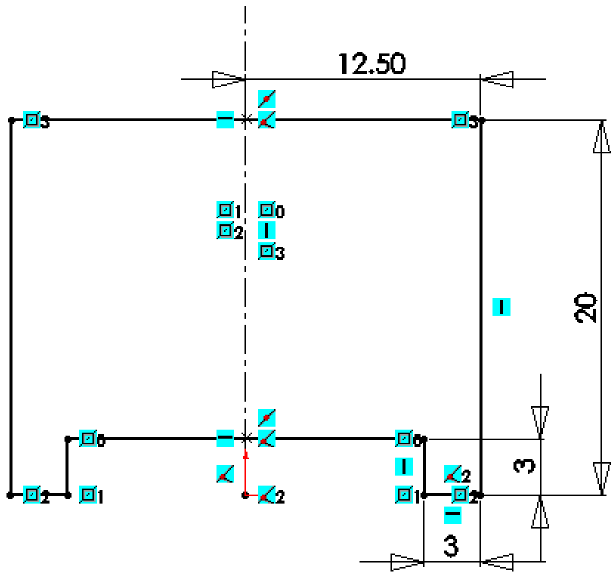
Initial Sketch Create the sketch shown, on the **front plane**



Relations All lines are either vertical or horizontal.

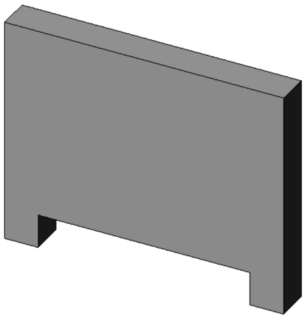
All relations are **Automatic Relations**

Mirror Entities **Mirror** the entities across the centre line.



Extruded Boss/Base

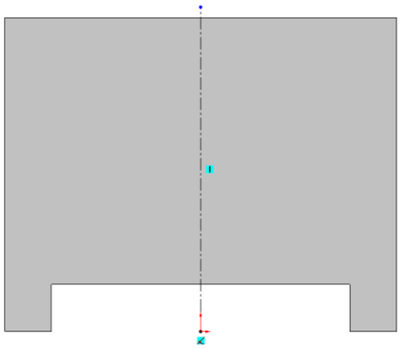
Extrude the sketch using a **Blind End Condition** to a **Depth** of **3mm**



Produce cutout.

Create a sketch on the front face**.**

Add in a centre line through the origin as shown

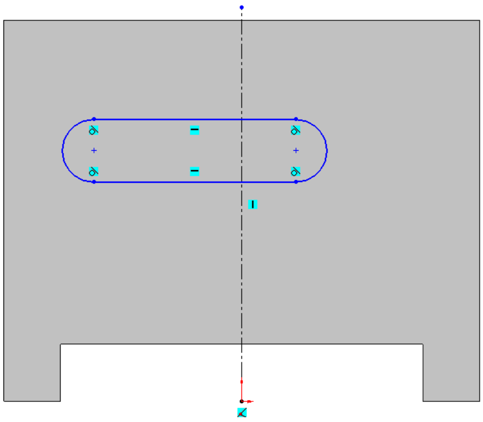


Sketch detail

Using **Line** and **Tangent Arc** create the sketch shown opposite.

Ensure that the Automatic Relations shown are included.

Add a centre line between the centres of the two tangent arcs.

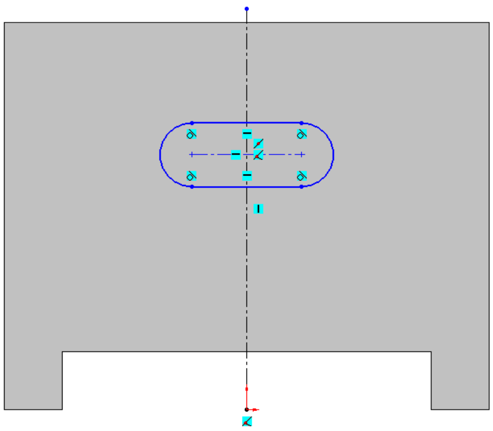


Positioning the sketch

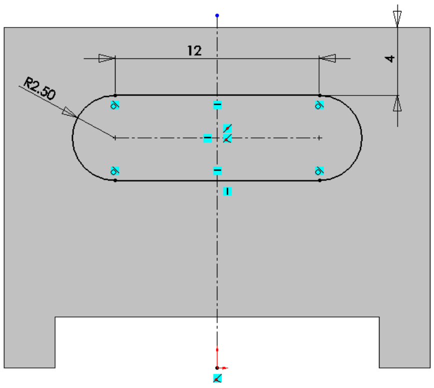
Right Click on the horizontal centre line and choose **Select Midpoint.**

Hold down the **Ctrl** key (to create multiple selections) and select the vertical centre line.

Add a **Coincident Relation** between the centre line and the midpoint.

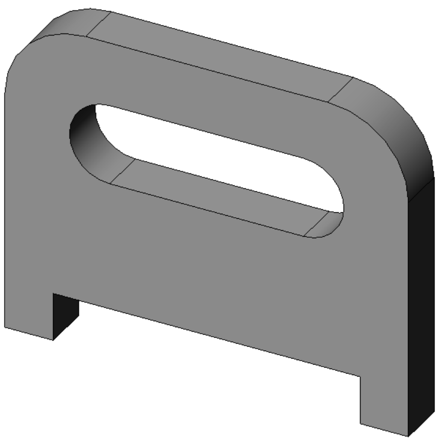


Smart Dimension Add <u>only</u> the dimensions shown opposite



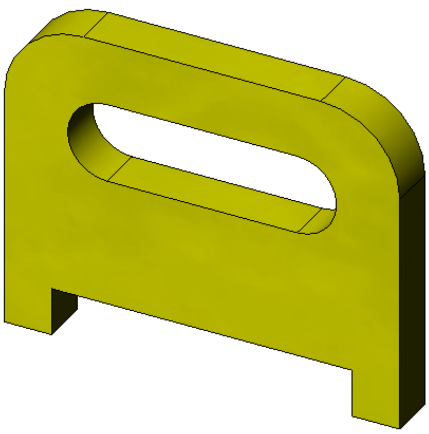
Extruded Cut **Extrude Cut** the sketch with a **Through All** End Condition

Fillet Add a **5mm Fillet** to the corners as shown



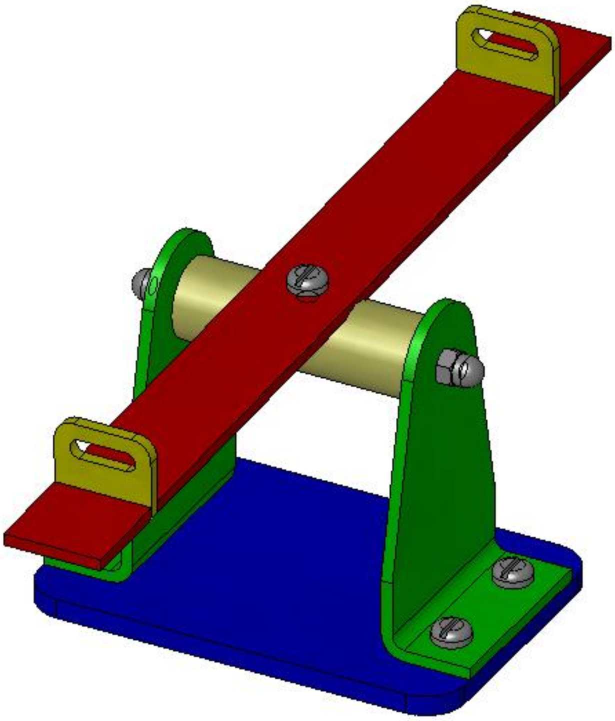
Edit Material Choose **Perspex Acrylic Cast Sheet** as the part material.

Adding Colour Choose a colour for the part from the colour swatch.



Save and close Click **Save** to save your work and click **File, Close** to close the part.

Creating the Assembly



Bottom-Up Assembly

Bottom–Up Assemblies are created by adding and orientating existing parts in an assembly. Parts added to the assembly appear as **Component Parts.** Component parts are orientated and positioned in the assembly using **Mates.** Mates relate faces and edges of component parts to planes and other faces/edges.

Stages in the process

Creating a new assembly
New assemblies are created using a similar method as new parts

Adding the first component
Components may be dragged and dropped from an open window or selected from a standard browser.

Position of the first component
The initial component added to an assembly is automatically fixed as it is added. Other components may be repositioned after they are added.

Feature Manager Design Tree and Symbols
The Feature Manager includes many symbols which contain information about the assembly and the components in it.

Mating components to each other
Mates are used to position and orientate components with reference to each other. Mates remove degrees of freedom from the components

Make assembly from Part/Assembly

Use the **Make Assembly from Part/Asssembly** option to generate a new assembly from an open part. The part is used as the first component in the new assembly and is fixed in space.

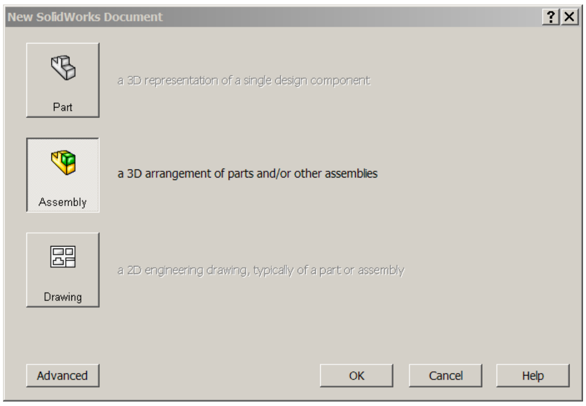
Where to find it.

Click **Make Assembly from Part/Assembly**  on the standard toolbar
Or, Select **File, Make Assembly from Part**

Open an existing part

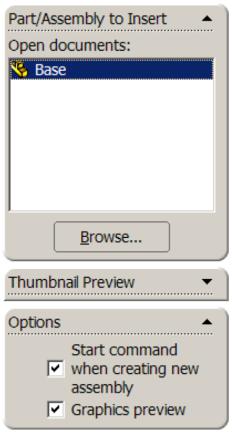
Open the part **base.** A new assembly will be created using this part

Click **Make Assembly from Part/Assembly** 



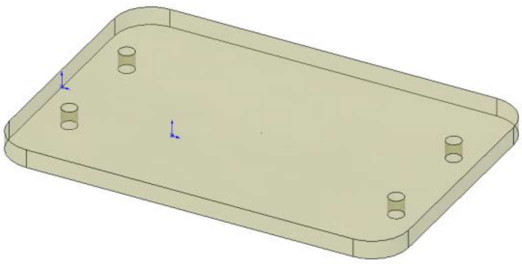
Choose the default assembly template. Click **OK**

Inserting the first part

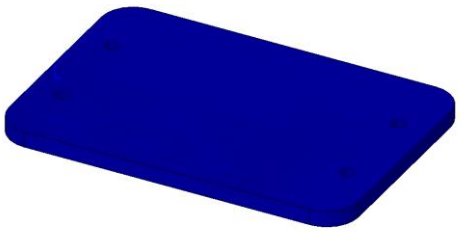


The **Insert Component** Dialog box appears with **base** displayed. Ensure **graphics preview** is selected

Move the cursor into the drawing area. A preview of the part along with the origins are displayed



Move the cursor to the origin and select, the part origin will snap to it as displayed.

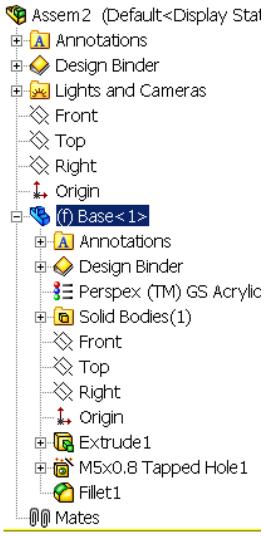


Saving the Assembly

Select **File, Save as** on the standard toolbar. Save the assembly into the folder containing the parts used to create it. An assembly is identified by its extension **.sldasm*.

Components

Parts that are inserted into the assembly appear in the Feature Manager Design Tree and may be expanded to show the individual features of that part



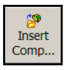
State of the Component The part may be fully, over or under defined. A (+) or (-) sign will precede the part name if it is **Over** or **Under Defined.** Parts that are under defined have some degrees of freedom available. **Fully defined** have none.

Mates

Mate Group: All **Mates** in an assembly are placed in a folder, identified by a double paper clip icon in the feature manager tree.

Mates may be used to fully define a component that does not move, or under define a component that is intended to move.

Adding Components

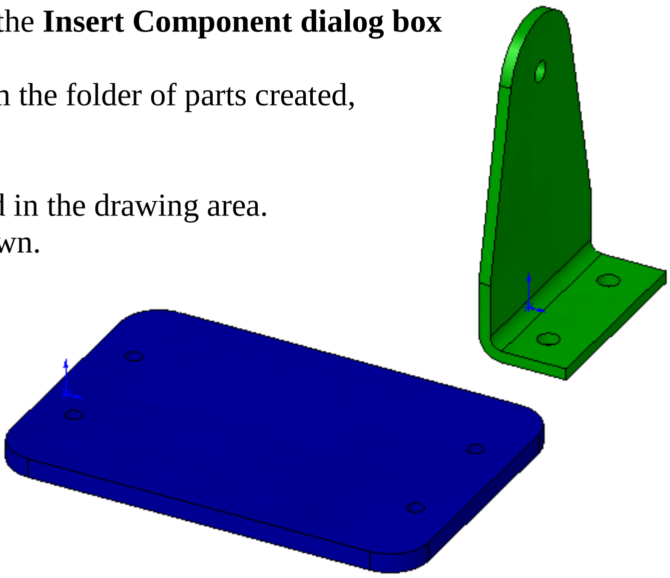
Select **Insert Component** 

from the **Assembly Toolbar**

Choose **Browse** from the **Insert Component dialog box**

Choose **Supports** from the folder of parts created, Choose **Open**

A preview is displayed in the drawing area. Click to drop it as shown.



Moving Components

Holding down the left hand mouse button on the component will allow you to drag it around

Rotating Components

Select **Rotate Components** 

from the **Assembly Toolbar.**

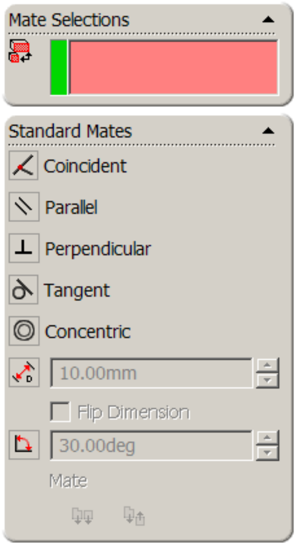
Place the rotate symbol over the component, hold down the left hand mouse button and drag. The component will rotate through its available degrees of freedom.

This is not to be confused with Rotate View from the View toolbar.

In order to create mates it is essential that we are proficient at rotating views of parts, in order to select faces/edges.

Insert Mate

Insert Mates creates relationships between component parts or between parts and an assembly.



Where to find it.

Choose **Insert, Mate…**
Or Select **Mate** from the **Assembly Toolbar**

Adding the Mates

Select **Mate,** the **Mate Property Manager** will appear.

Selecting Faces

Select the top face of the base. Rotate the view and choose the underneath face of the support.

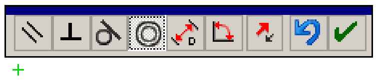
The parts will move so that the selected faces are contained on a single plane.

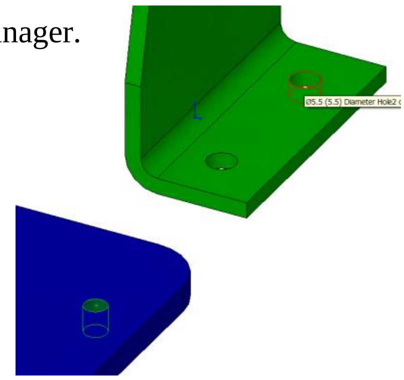
Choose OK

A Coincident Mate has been applied.

Mate Pop-up Toolbar

The **Mate Pop-up Toolbar** is used to make selections easier by displaying the available mate types on the screen.
These mirror those that appear in the property manager.





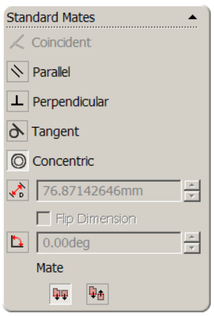
Further Mates

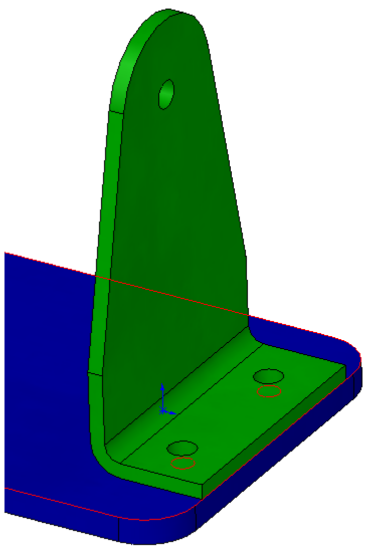
The Mate Property Manager remains open.
Select the inner wall of the hole on both parts as shown.

The support will move across so that the two holes become concentric.

Note: Concentric is selected as the default mate, as shown below.

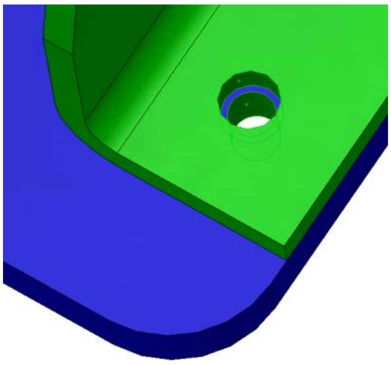
Choose OK





Fully Define the Component

If you try to move the support you will find that it rotates around the concentrically mated holes. This is because it is the only degree of freedom which remains.



To eliminate this degree of freedom a further concentric mate must be added between the two remaining holes.

Select the inner faces of the two holes.

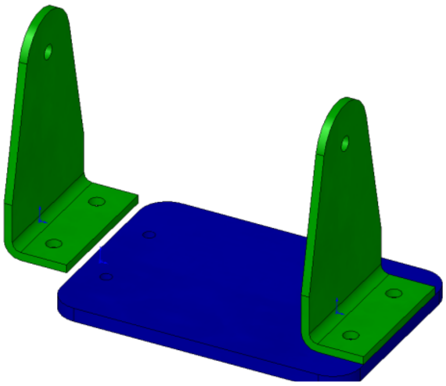
Concentric Mate will be displayed by default in the mate property manager.

Choose OK  to apply the mate.

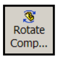
Choose OK  to exit the property manager.

Adding further parts

Insert the part **support** as before

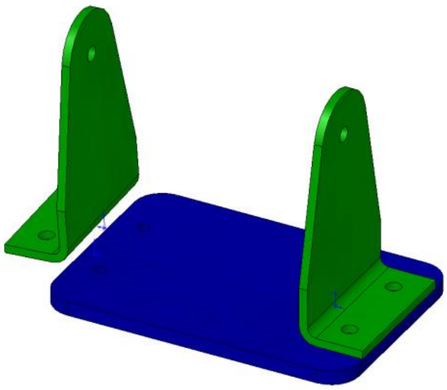


Rotate Component

Choose **Rotate Component** 

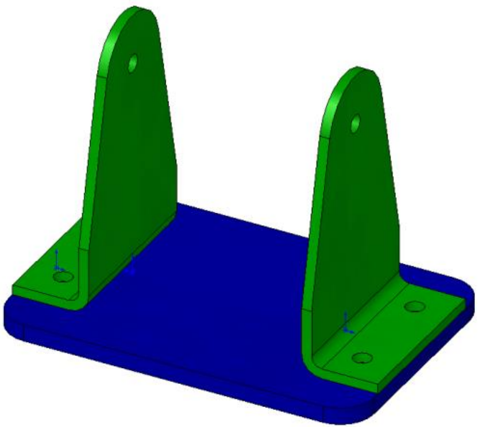
from the assembly toolbar.

Rotate the component to the orientation shown.



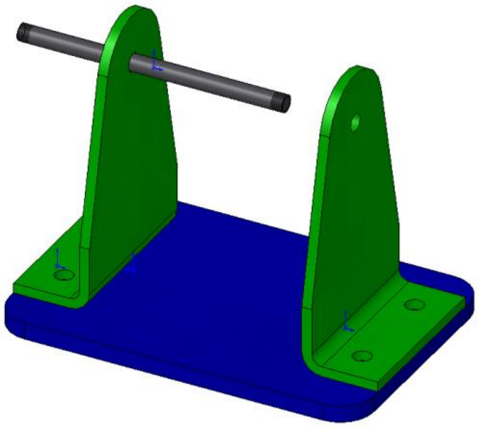
Adding Mates

Add a Coincident Mate between the top face of the base and the underside of the support.
Add concentric mates between the holes as before to locate the part.



Inserting the Axle

Insert **axle** into the assembly



Adding a Concentric Mate

Add a concentric mate between the axle and the inner face of the hole.

The axle is still free to move along its axis.

Choose OK 

Width Mate

Width Mate is an **Advanced Mate**
Selections include a pair of **Width Selections** and a pair of **Tab Selections.**

Width References

Width Selections form the ‘outer faces’ used to contain the other component

Tab References

Tab Selections form the ‘inner faces’ used to locate the other component

Adding a Width Mate

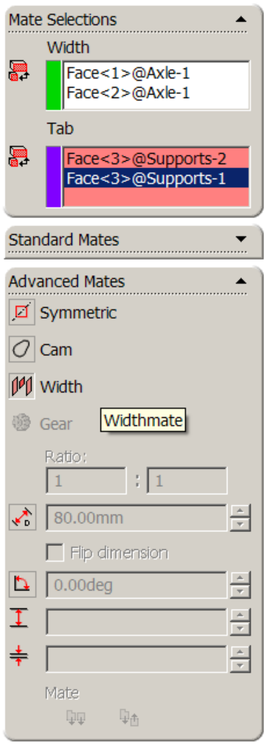
Choose **Mate,** and select the **Advanced Mates Tab**

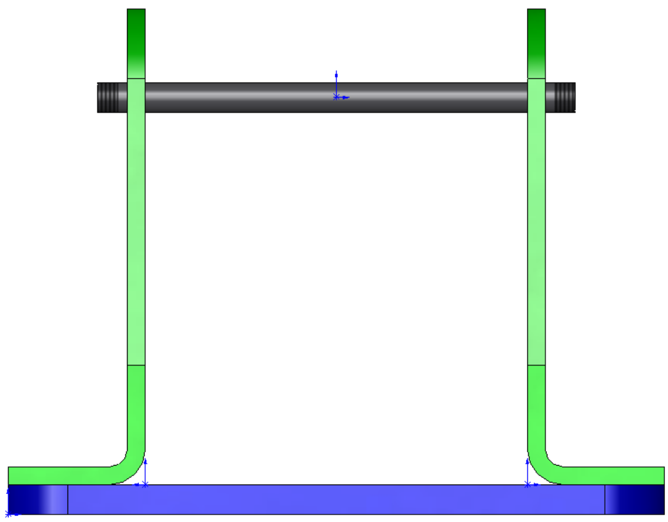
Select **Width Mate.**

Expand the **Mates Selections Tab**

Select the faces of the axle as **Width Selections** and the inside faces of the supports as **Tab Selections.**

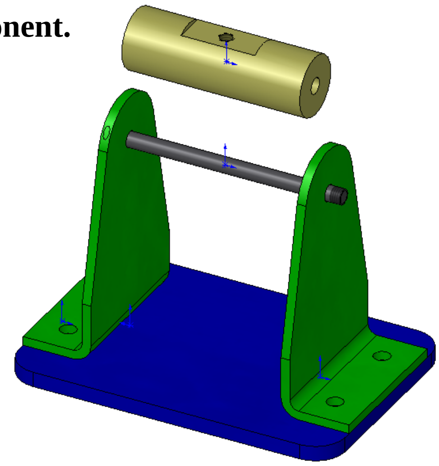
The Axle moves so that it is centred, with equal protrusion on either side**.**





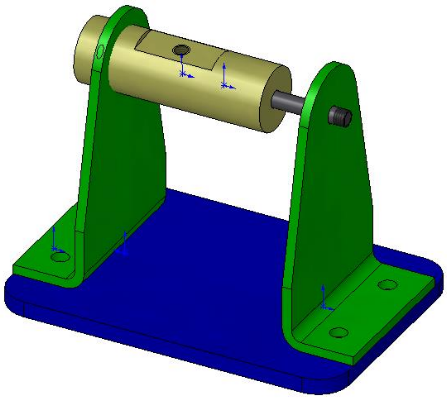
Inserting the Swivel

Insert the swivel part using **Insert Component.**



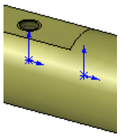
Add Concentric Mate

Add a **Concentric Mate** between the surface of the bar and the inside of the axial hole



If the origins of the parts are not displayed as shown, choose **View, Origins**

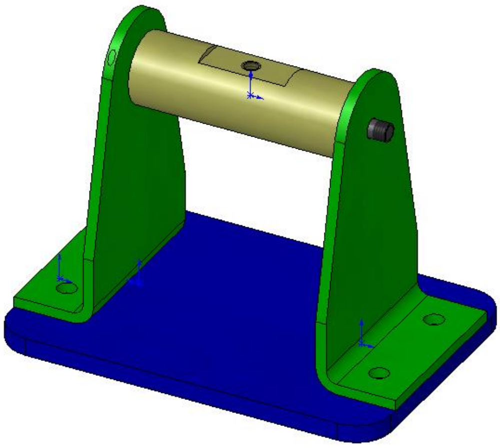
Add Coincident Mate between the Part Origins

Select the **part origins** 

Coincident Mate is created by default

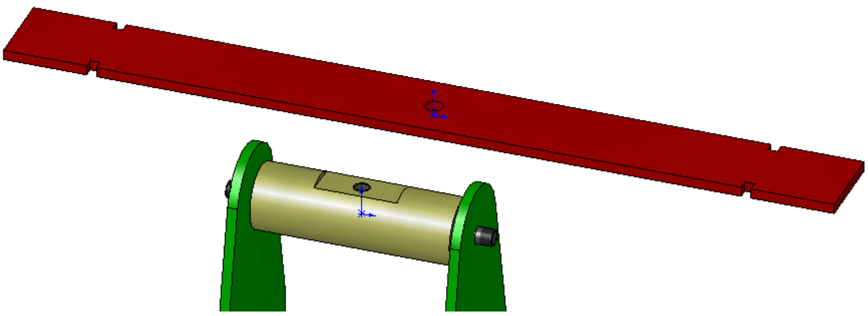
Choose OK to apply the mate.

Choose OK to exit the property manager.



Inserting the Board

Insert the part **board** using **Insert Component**

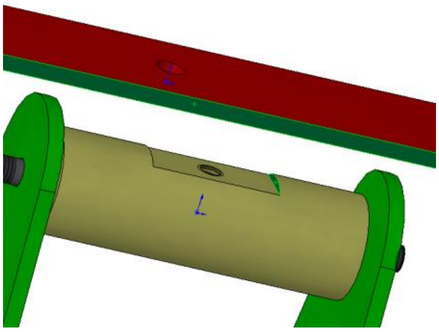


TECHNOLOGY SUBJECTS SUPPORT SERVICE

Adding Mates

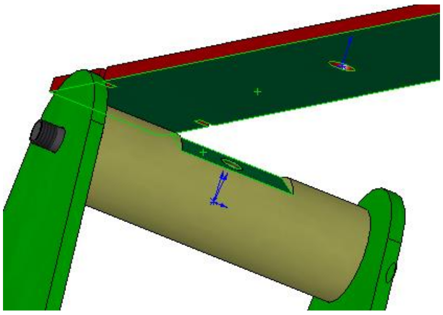
Coincident Mate

Add a **Coincident Mate** between the selected face of the board and the right hand face of the cut out, as shown.



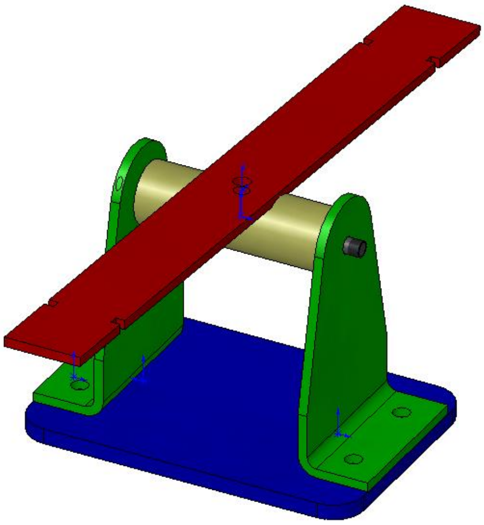
Coincident Mate

Add a **Coincident Mate** between the underside of the board and the top face of the cut out.



Concentric Mate

Add a **Concentric Mate** between the two holes

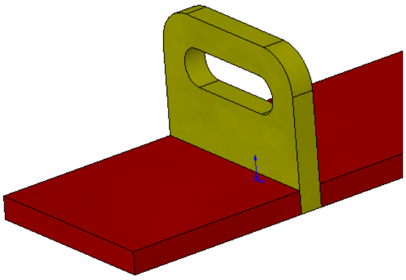


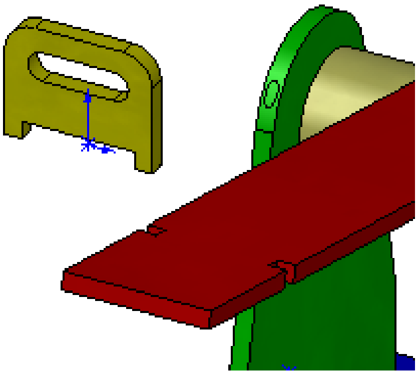
Inserting the Handles

Insert the part **handle**

Adding Mates

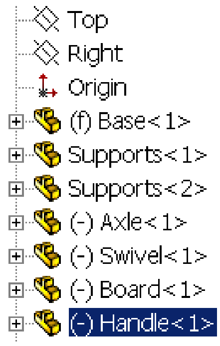
Mate the necessary surfaces using **Coincident Mate** to position the handle as shown below.





Adding another handle

To add in another handle, hold down the **Ctrl** key, select and drag the handle from the **Feature Manager Tree**, drop it into the drawing area.

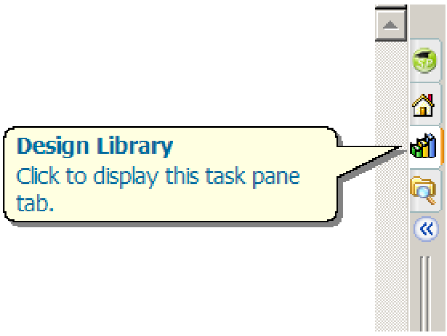


The inserted handle will have the same orientation as the existing handle in the assembly.

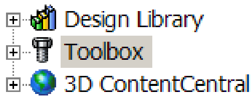
Position the handle as before using **Coincident Mates** between the necessary faces.

Adding the Fasteners Design Library

The **Design Library Tab** in the **Task Pane** provides a central location for reusable elements.



The design library contains the following folders;



If Toolbox is not displayed choose **Tools, Add-In's…** from the drop down menu and select **SolidWorks Toolbox** and **SolidWorks Toolbox Browser**

Accessing the toolbox

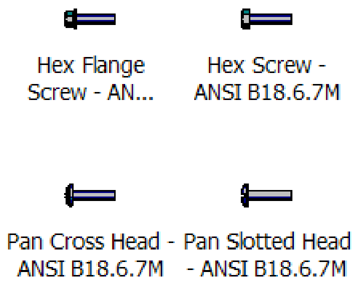
Double Click on **Toolbox** to open.
All folders are displayed in the task pane underneath

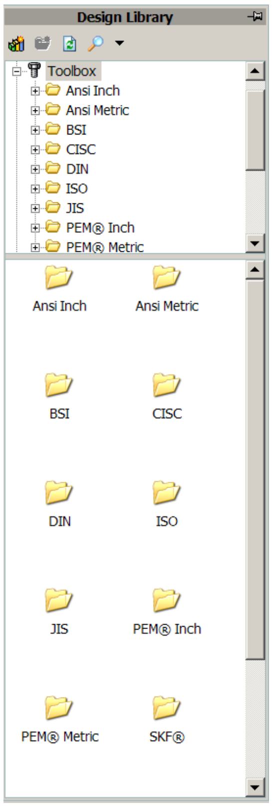
Choose **Ansi Metric**

Choose **Bolts & Screws**

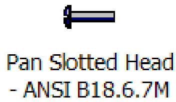
Choose **Machine Screws**

The screw types below are displayed.

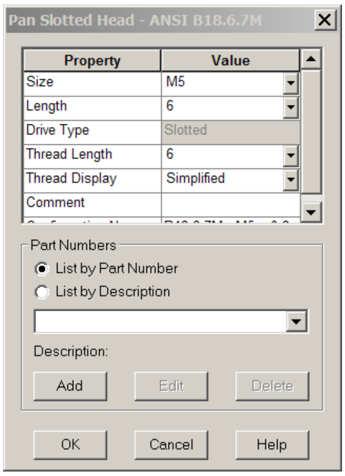




Inserting a screw

Hold the left hand mouse button over 

Drag and drop the part into the drawing area.



A preview of the inserted component appears along with the property dialog box shown opposite.

Change the following values;

Size: M5

Length: 6

Thread Length: 6

Select **OK.**

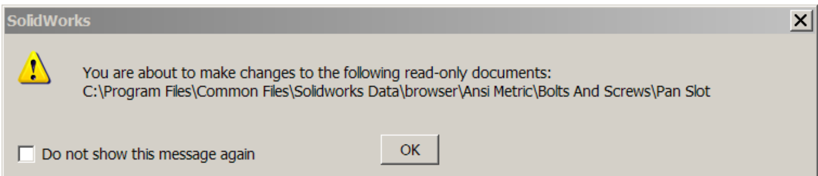
A preview of a second component will be introduced. Select **X** in the **Insert Components** property manager to avoid any further instances being created at this time.

Saving the part

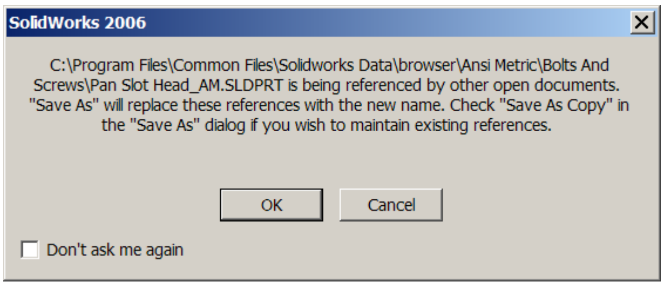
The screw must be saved into the folder along with the other parts which make up the assembly.

Right Click on the screw feature in the feature manager tree and select **Open Part**

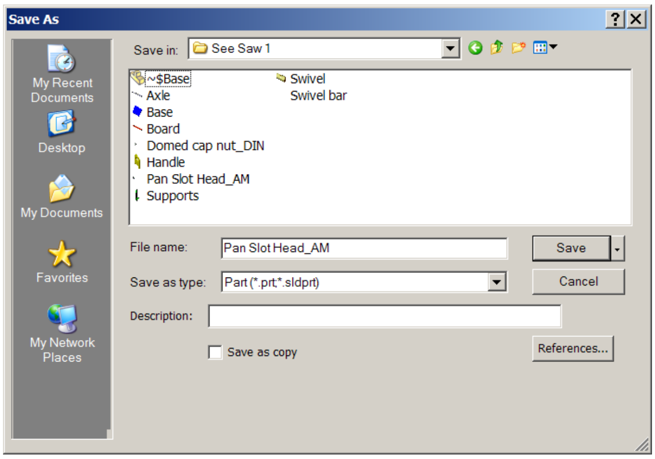
The following dialog box will appear. Choose **OK**



Choose **File, Save As…**
The following dialog box will appear. Choose **OK**

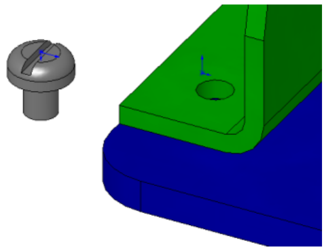


Navigate to the folder containing the other assembly parts. **Save** the screw in this folder using the default name – Pan Slot Head_AM

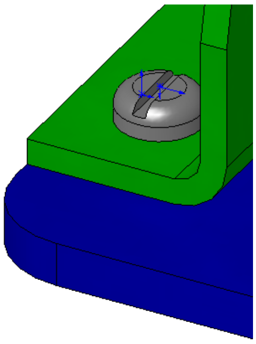


Positioning the screw.

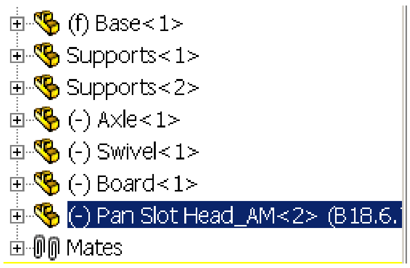
Adding Mates Create a **Coincident Mate** between the under-face of the head of the screw and the top face of the support as shown.



Add a Concentric Mate between the hole and the screw to complete positioning.



Feature Manager Tree The **feature Manager Tree** displays all of the inserted components along with the number of instances of each.

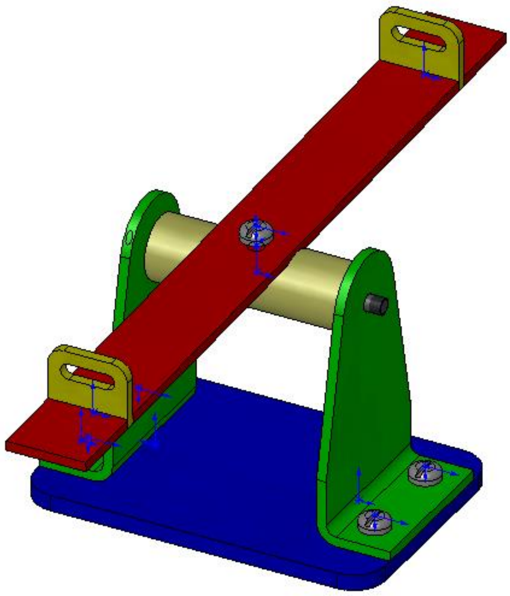


Adding further Screws To add in another M5 screw hold down the **Ctrl** key, select and drag the screw from the **Feature Manager Tree**, drop it into the drawing area.

The inserted screw will have the same orientation as the existing screw in the assembly.

Position the screw as before using **Coincident** and **Concentric Mates**.

Repeat the procedure for the remaining screws in the base and the screw to secure the board to the swivel



Adding Cap Nuts Cap nuts are contained within the **Toolbox Library Folders**

Where to find them Within the **Toolbox** folders in the **Task Pane** choose;

DIN, Nuts, Hex Nuts - Cap

Drag and drop 

into the drawing area.

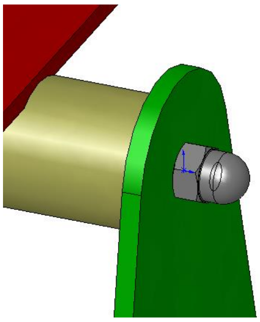
Edit Properties Change the **Size** to **M5**

Property	Value
Size	M5
Thread Display	Simplified
Thread Undercut	No undercut
Comment	
Configuration Name	DIN 1587 - M5 --NNU

Select **X** in the **Insert Components** property manager to avoid any further instances being created at this time.

Saving the part Save the domed cap nut, to the folder containing the other assembly parts, in the same manner as the *pan slotted head screw* earlier.

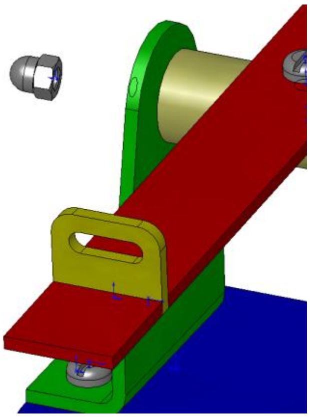
Positioning the Cap nut



Apply Mates

Add a **Concentric Mate** between the hole in the cap nut and the outer diameter of the axle.

Add a **Coincident Mate** between the face of the cap nut and the face of the support.



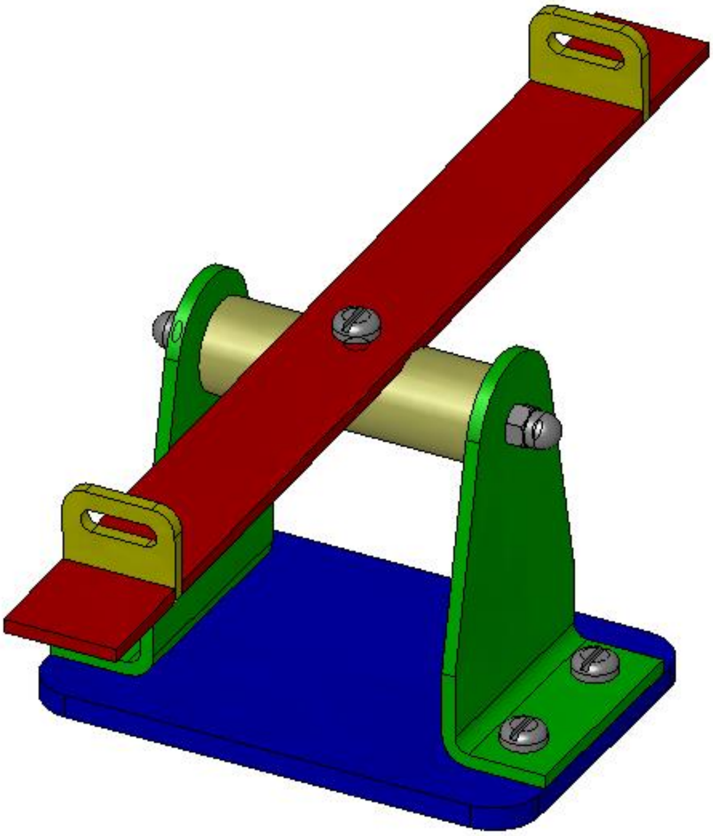
Adding another Cap Nut

Holding down Ctrl drag another cap nut from the feature manager design tree.

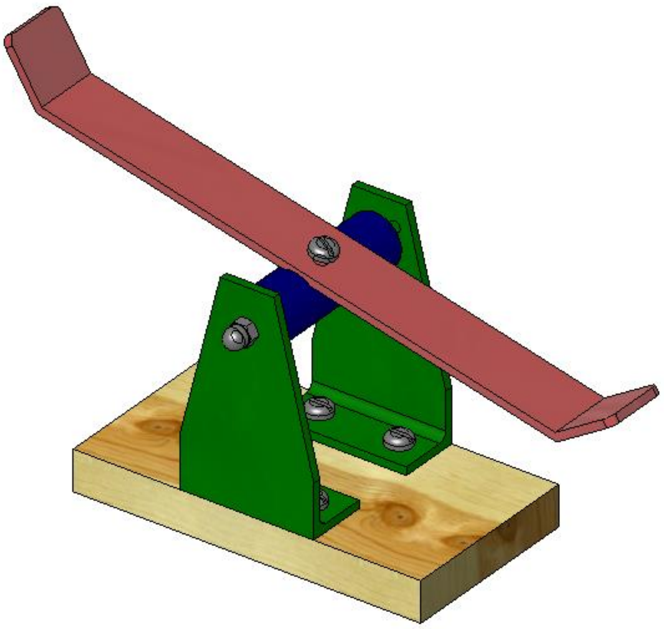
Rotate the cap nut to the orientation shown opposite

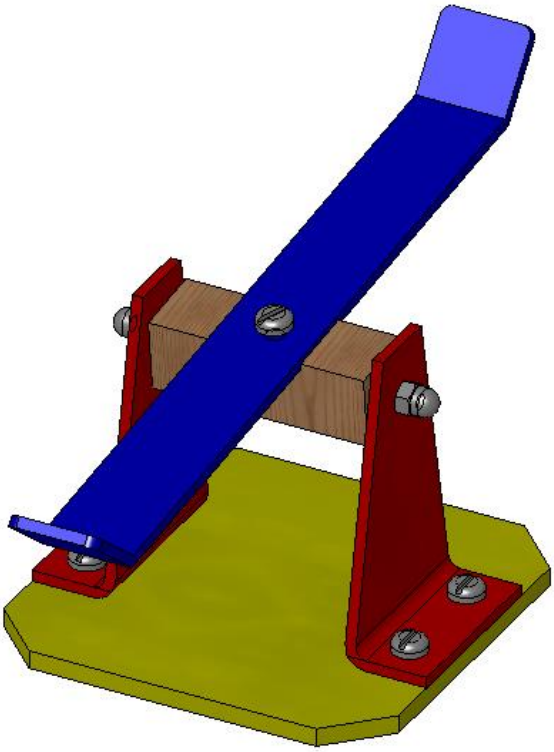
Applying Mates

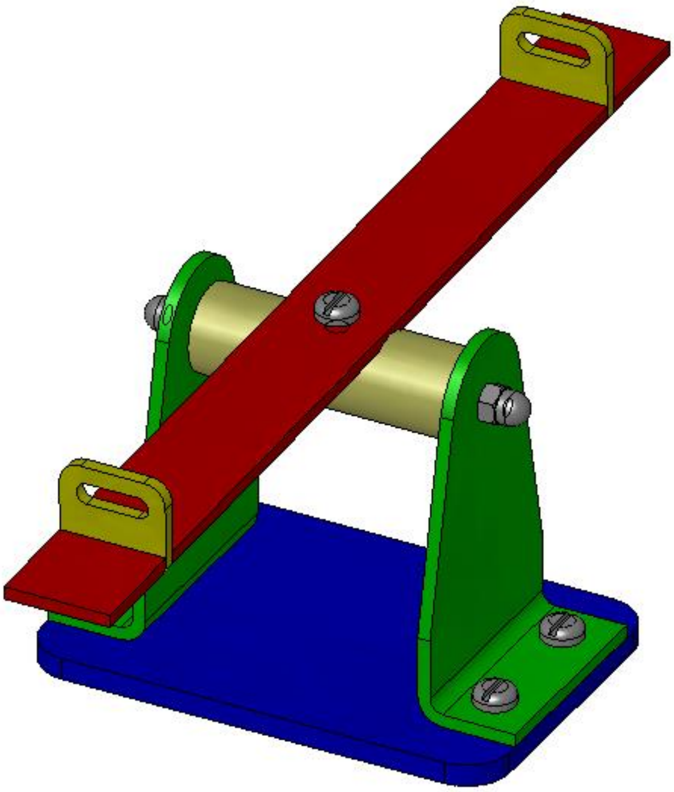
Add mates as before to position the cap nut.

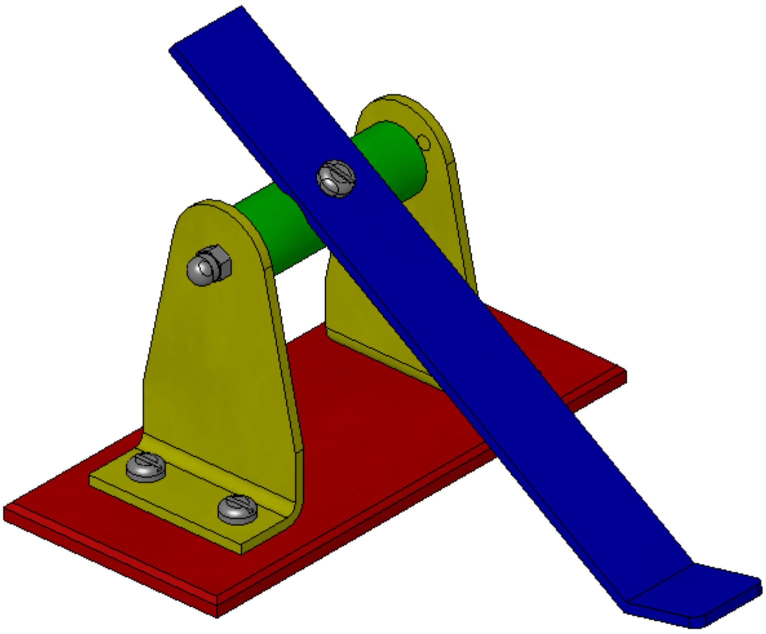


Variations of the design









Moving the Assembly

Drag a component in the graphics area. The component will move within its degrees of freedom.

Select the left hand side of the board, hold down the left hand mouse button and drag. The board and swivel will rotate around the axle through 360º, through the base.

Move Component

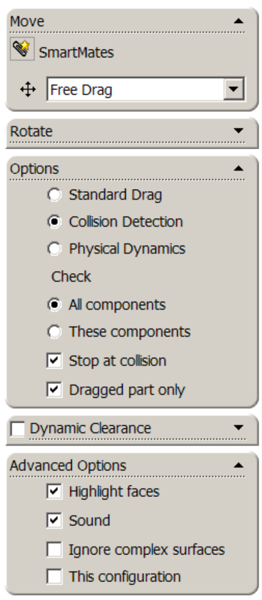
Select **Move Component** 

from the **Assembly Toolbar.**

The **Move Component** properties dialog box appears.

Click the box next to **Collision Detection, Stop at collision,** and **Dragged part only.**

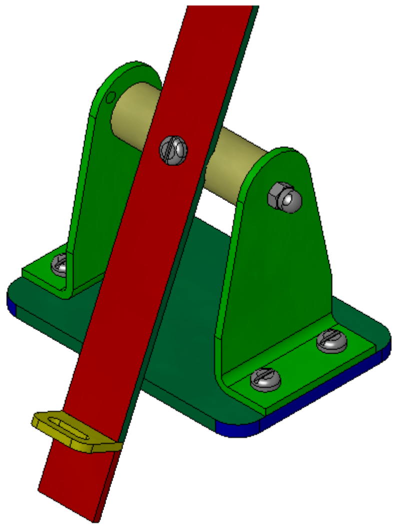
Ensure that **Highlight faces** and **Sound** are selected.



Moving the Components

Select the left hand side of the board, hold down the left hand mouse button and drag.

The board and swivel will rotate around the axle.



Collision Detection

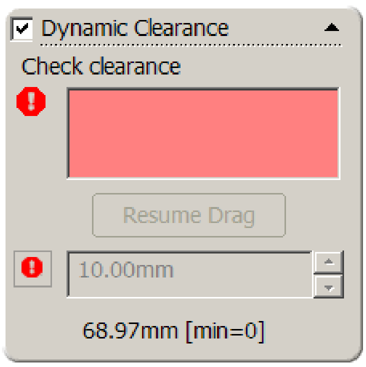
When the board reaches the base it will stop**,** the two colliding faces will be highlighted and a sound will be heard.

Dynamic Clearance

You can dynamically detect the clearance between components when moving or rotating a component. As you move or rotate a component, a dimension appears indicating the minimum distance between the selected components.

Where to find it

Click in the box next to **Dynamic Clearance** in the **Move Component** dialog box.



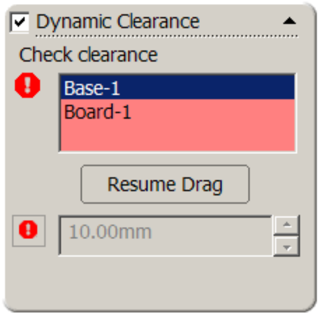
Dynamic Clearance

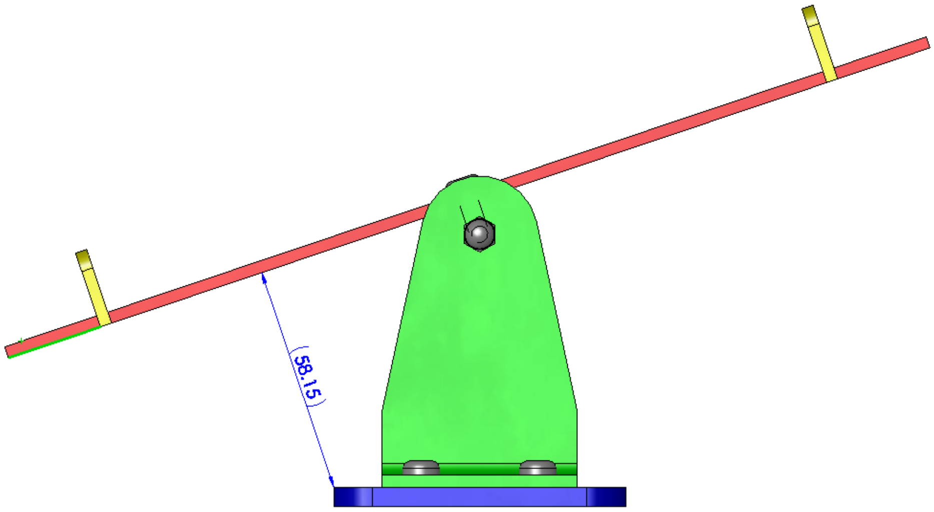
Choose the board and base as the **Components for Collision Check**

Select **Resume Drag**

Drag the board in the drawing area.

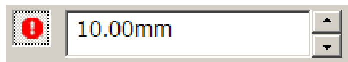
The dynamic clearance is displayed between the two parts.





Stop at Specified Clearance

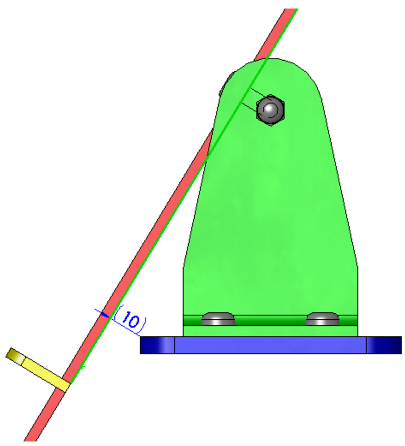
Select **Stop at Specified Clearance** by selecting



Select **Resume Drag.**

Drag the board in the drawing area.

The board will stop when it reaches a distance of 10mm from the base.



Creating a drawing Select **Make Drawing from Part/Assembly** 

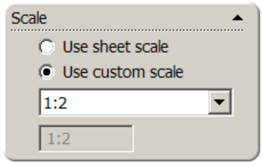
Choose a **Drawing Template** to create the drawing**.**

The **DCG A4L** Template will be used to create this drawing.
(This template is available on Resource CD Round 4, along with instructions detailing how to load them onto your computer)

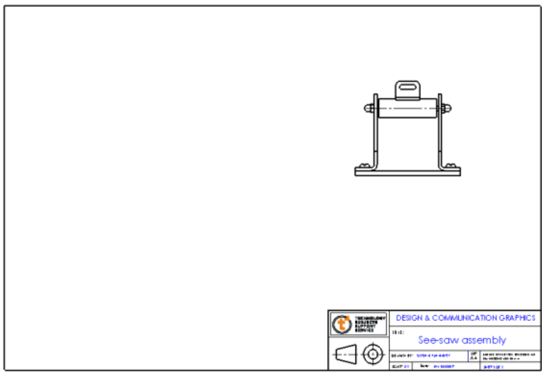
Model View A box will appear on screen which outlines the size of the view.

Scale Scroll down to **Scale** in the **Model View** dialog box.

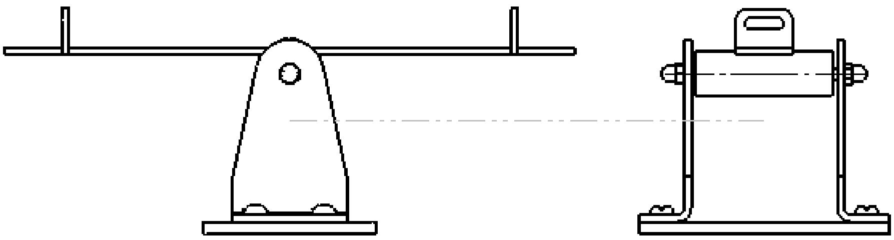
Choose **Use custom scale** and select **1:2**



Placing the first view Move the cursor back into the drawing area and click to drop the view in the position shown.



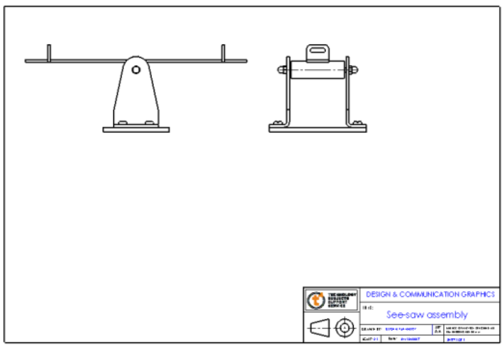
Projected View Drag the mouse to the left to view the projected view. Left click to position the view.



Press **Esc** to complete the insertion of views.

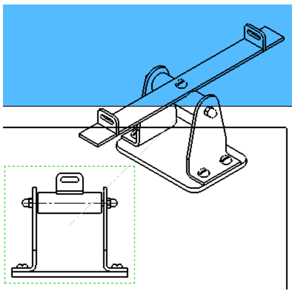
Repositioning Views Move the cursor close to the view.
A red dotted line will surround the view.
Select and drag the dotted line.
The view will be repositioned.

Position the views as shown.



Adding a 3D View Choose **Projected View** from the **Drawings Toolbar.**

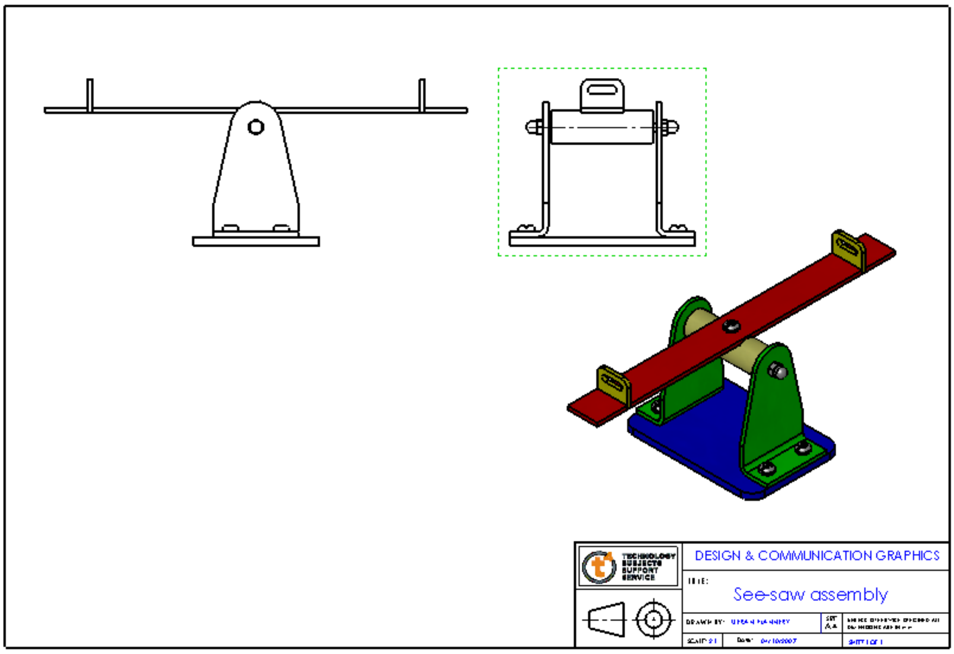
Select the end view and drag upwards as shown.



Positioning the view. Hold down the **Ctrl** key and drag to a position overhead the title block.

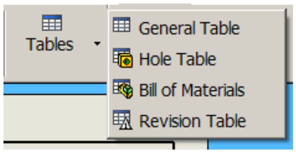
Click to position the view and press **Esc**

View Appearance Select this view and choose **Shaded with Edges** from the view toolbar.



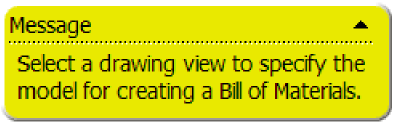
Bill of Materials A **Bill of Materials** may be inserted into the drawing of an assembly. If you add or delete components in the assembly, the **Bill of Materials** automatically updates.

Where to find it From the **Annotations** toolbar choose **Tables, Bill of Materials**



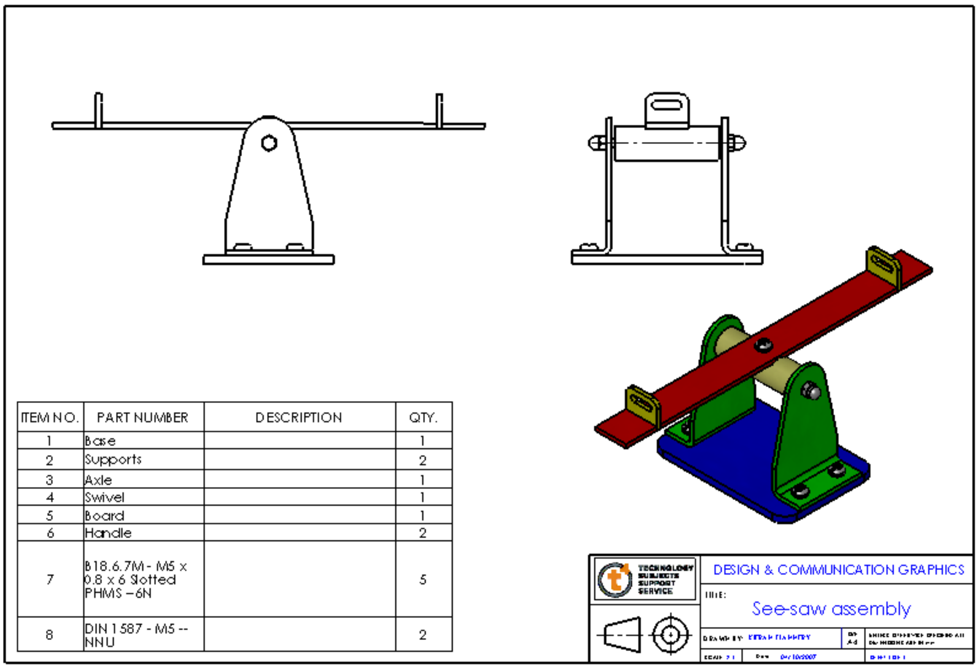
When prompted, choose the front view.

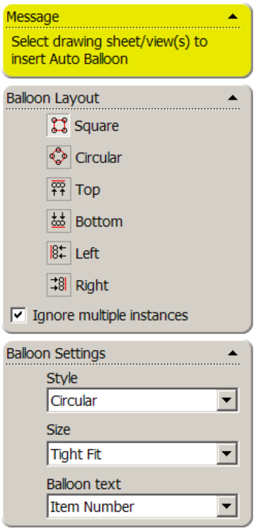
Choose **OK**



Positioning of the Table Position the table in the lower left hand corner. Select OK

Resize the table by dragging the top right hand corner.





Balloon Referencing

Balloons may be created in a drawing document. The **Balloons** label the parts in the assembly and relate them to item numbers on the **Bill of Materials (BOM).**

Where to find it

Choose **Auto Balloon** 

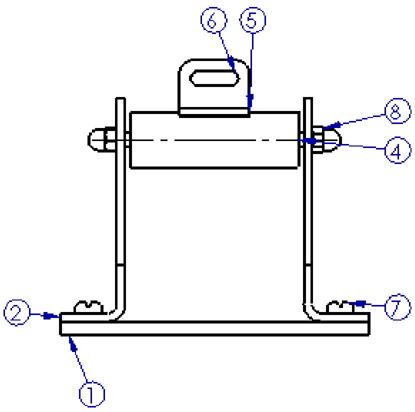
on the **Annotations Toolbar**

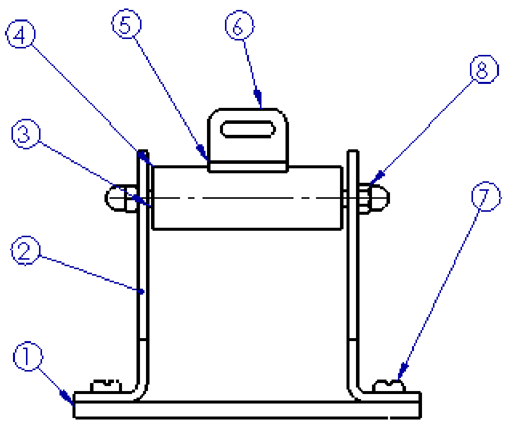
Edit the properties in the Property Manager as needed

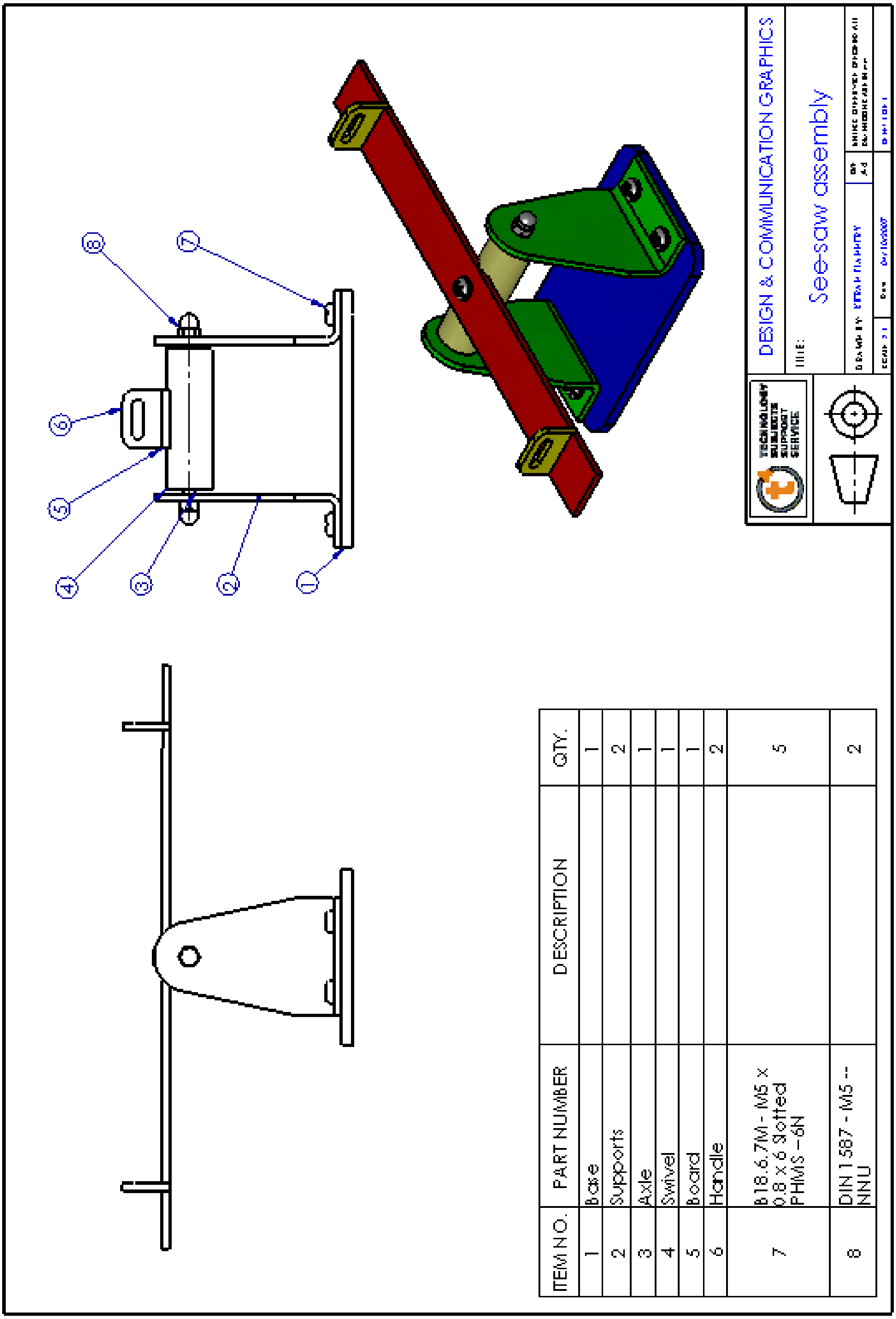
Select the Right Hand View

Repositioning balloons

The balloons are displayed as shown below, on the left.
Drag the balloons and arrows to display as shown on the right.







ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Base		1
2	Supports		2
3	Axle		1
4	Swivel		1
5	Board		1
6	Handle		2
7	B18.6.7M - M5 x 0.8 x 6 Slotted PHMS --6N		5
8	DIN 1587 - M5 --NNU		2