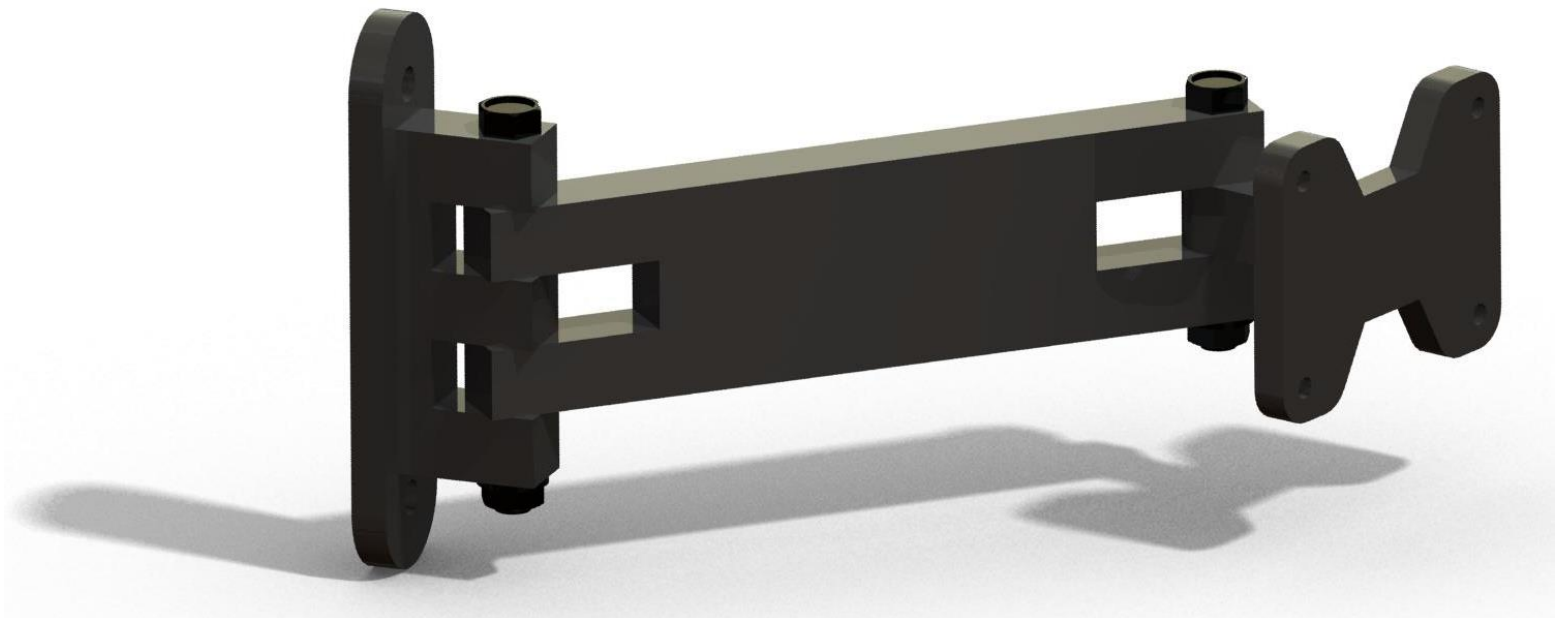
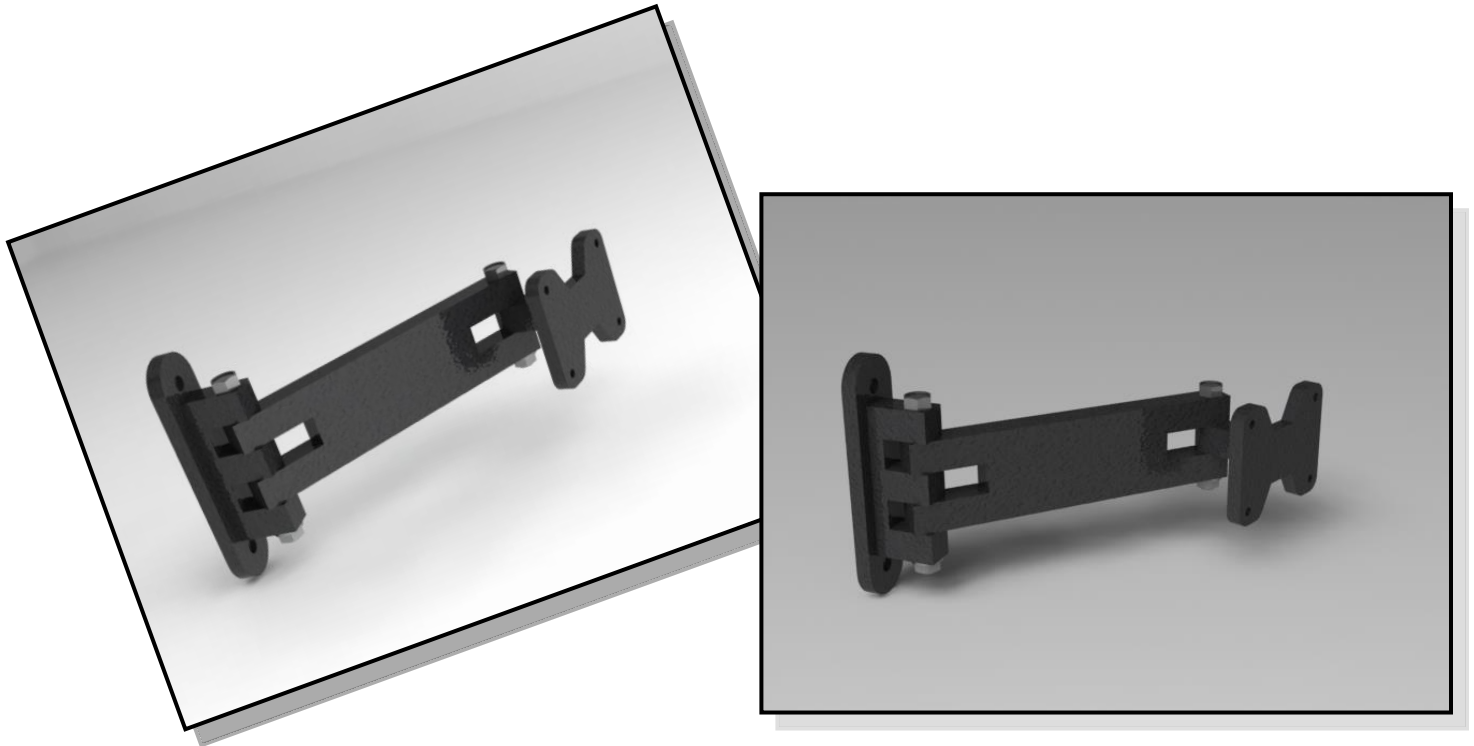


SolidWorks 2009 & DCG Student Assignment



TECHNOLOGY
SUBJECTS
SUPPORT
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SolidWorks 2009 & DCG Student Assignment

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Focus of the lesson

The purpose of this exercise is to introduce SolidWorks 2009 and explore the user interface.

“SolidWorks 2009 – User Interface” should be read in advance of completing this exercise. It may be found on the Round 10 DCG Resource DVD.

We will also address the DCG Student Assignment and its CAD applications. We will focus on the feedback from the DCG Student Assignment 2009 and look at some observations made in relation to it. Whilst some students demonstrated some excellent CAD skills and produced exceptional detailed models in the completion of the assignment there are some areas which need to be addressed. These areas are outlined below.

Output 4

- Greater emphasis needs to be placed on design intent and economy of design
- In some cases there was no attempt made to rename the main features in the part files
- Most students demonstrated excellent file management skills however, in a small but significant number of cases one or more parts were not included on the CD
- In a number of assemblies the parts were not properly mated, just merely moved into position
- In some cases an eDrawing of the CAD model was not presented on the CD

Output 5

- Most students produced excellent layouts containing a variety of appropriately scaled views, details and sections with accompanying notes. The most common error in this output was poor layout due to inappropriate scale of the drawing views and the poor dimensioning of some of the orthographic views.

Output 6

- A significant number of students used PhotoWorks to produce high quality photorealistic images of their models. Some images were taken directly from SolidWorks and were accepted for this first year of the DCG Student Assignment
- In future it is expected that all students will use either PhotoView 360, PhotoWorks or some other image manipulation software to create the required images.

Output 9

- While many students produced excellent drawings, the level of detail and presentation of some of this work was only fair.
- Greater attention should have been given to highlighting and detailing the design modifications and the inclusion of notes and main dimensions where appropriate.
- In some cases it was difficult to determine the difference between Output 5 and Output 9 as students failed to highlight the modification(s) made to the existing artefact.

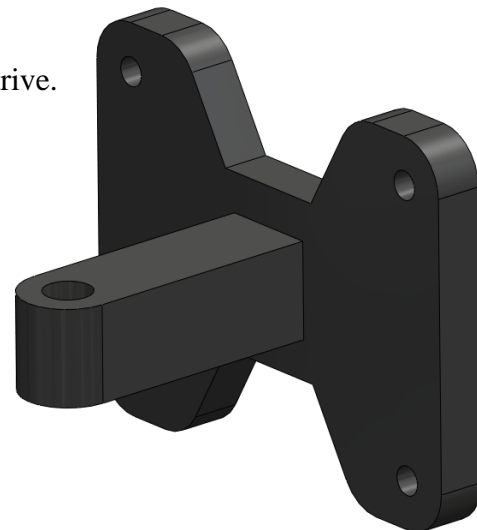
Getting started – Modelling the TV Plate

This exercise is based on the TV Bracket which formed the basis for question C-5 OL 2009 DCG exam.

The individual parts, with the exception of the *TV Plate*, are pre-modelled and saved along with the assembly in D:\SolidWorks\TV Bracket on the Round 10 DCG DVD.

Copy the folder **TV Bracket** from the **Round 10 DCG DVD** to your hard drive.

We will model the TV Plate and save it along with the other parts in the TV Bracket folder.



Starting SolidWorks and Opening a New Part

Start the SolidWorks application.


Click the **Start** menu from the Windows interface.

Click **All Programs, SolidWorks 2009, SolidWorks 2009**.

Tip: You can quickly start a SolidWorks session by double-clicking the left mouse button on the desktop shortcut, if there is a shortcut icon on the system desktop.



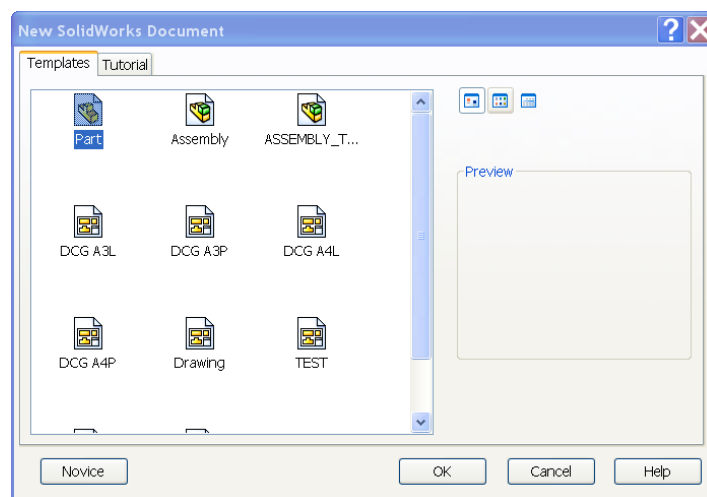
Open a new part.

Click **New**  from the Menu Bar toolbar. The New SolidWorks Document dialog box is displayed.

Note: When SolidWorks 2009 is installed it automatically references the location of the templates from 2006.

Click the **Part** template

Click **OK** from the New SolidWorks Document dialog box. A new Part document window is displayed.



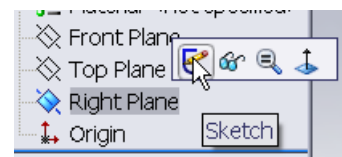
Begin Sketching

Solid models are built from features. Initially, features are based on 2D sketches. The sketch is the basis for a 3D model. Create a 2D sketch that you will later make into a 3D solid.

Sketches are flat or planar. In the same way as we need a sheet of paper on which to sketch you need to choose a plane on which to sketch in SolidWorks. A SolidWorks part contains three default sketch planes. They represent the Front, Top, and Right planes.

Because of the orientation of the part we are going to begin sketching on the right plane.

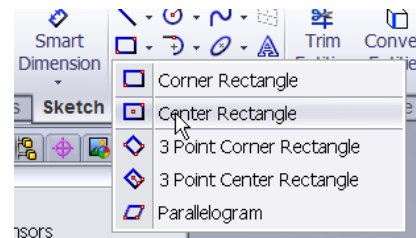
Left click on the **Right Plane** in the featuremanager design tree and choose **Sketch**



We will begin by sketching a rectangle with its centre coincident with the origin.

The importance of the positioning of the origin will become more apparent as we work through the exercise.

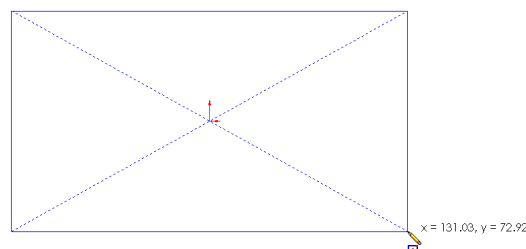
Choose **Center Rectangle** from the sketch toolbar.



Place the centre of the rectangle on the origin and drag the rectangle.

Left click in the graphics area to position the rectangle.

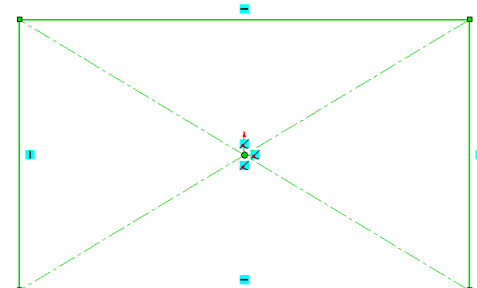
Press **Esc** on the keyboard to exit the command



Zoom to fit.

Press the **f** key on the keyboard to return to the full Graphics area.

Note: The horizontal and coincident geometrical relations are automatically added.

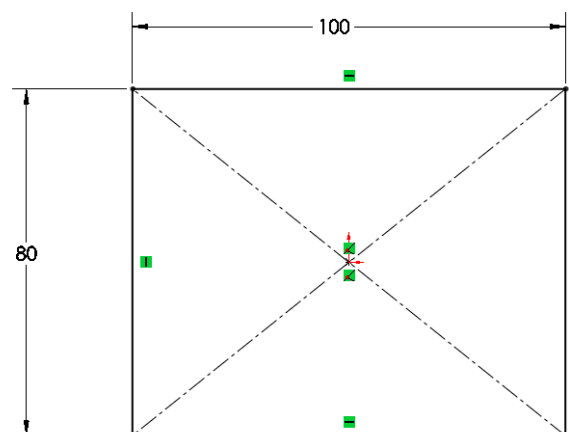


Smart Dimension

Choose **Smart Dimension**  from the sketch toolbar.



Add the dimensions shown across.

The sketch turns black and is fully defined.




Exiting the sketch.

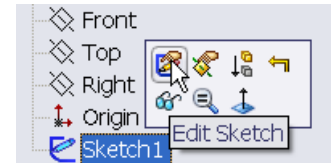
Three options exist to exit the sketch. As in SolidWorks 2006 you may exit the sketch by choosing

Exit Sketch  from the confirmation corner or choose **Exit Sketch**  from the **CommandManager**

The third option, new to SW 2008, is to double left click on a blank space in the graphics area

Accessing a sketch to edit it.

If you wish to edit an existing sketch left click on the sketch in the feature manager and choose **Edit Sketch** 

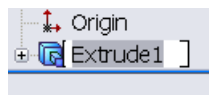


Renaming features on creation

It is good practice to rename features with names that are easily identifiable. Within SolidWorks 2009 a feature exists to rename features on creation.

To enable this feature **Choose Tools, Options, FeatureManager – Name feature on creation**

By enabling this option, when a feature is created SolidWorks will automatically prompt you to name the feature, as shown below.



Extruding the TV Plate.

Click the **Features** tab from the CommandManager. The Features toolbar is displayed. Select the **Extruded Boss/Base** tool. The Extrude PropertyManager is displayed

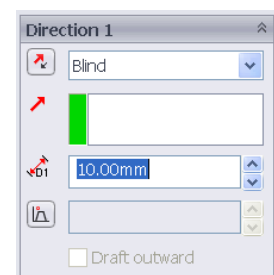


Choose Sketch1 as the sketch to use to create the feature

Set the End Condition and Depth.

Select **Blind** for End Condition in Direction 1. Enter a **Depth** of 10mm.

You can use the up and down arrow buttons next to the Depth box to change the value by 10mms (default in system options) at a time.



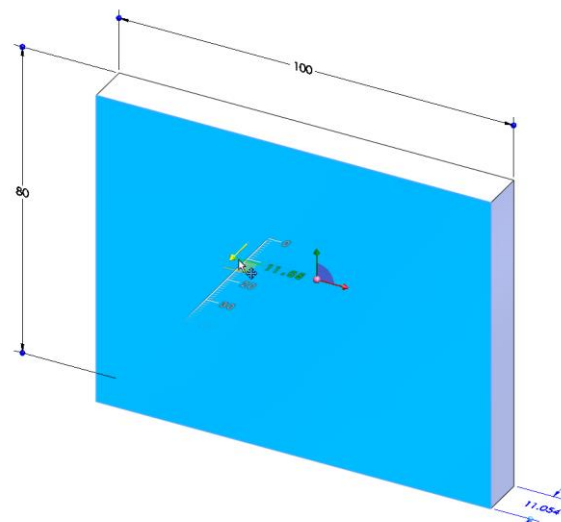
Click **OK** in the Extrude PropertyManager.

The Extrude1 feature is created.


Rename the feature **plate**

Tip: Instant3D provides the ability to click and drag geometry and dimension manipulator points to resize or create features directly from the Graphics area.

Use the on-screen ruler to measure your modifications.

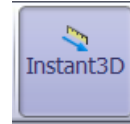


Using Instant3D

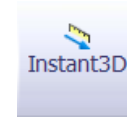
Double click on the top surface of the model. An arrow  will appear pointing upwards.

Click and hold the tip of the arrow with the left hand mouse button. Drag the cursor, a ruler will appear. Move the cursor over the ruler and drag the cursor to change the **Depth** of the extrude in realtime. Press **Esc** to exit Instant3D and record the changes.

Note: Ensure Instant 3D is enabled on the **Features** toolbar.



Enabled



Disabled

Note: Return the Depth of Extrude1 to 10mm to continue with the exercise.

Saving Your Work

Save the part document as **TV Plate.sldprt** in the TV Bracket folder

You can save your files as often as you wish. However, there are really only two situations that require you to save your work:

- After you have done something you want to keep.
- Before you try something that you are not sure will work.

Note: A file saved in a later version of SolidWorks cannot be opened in an earlier version i.e. A SolidWorks model created in SW2009 cannot be opened in SW2006.

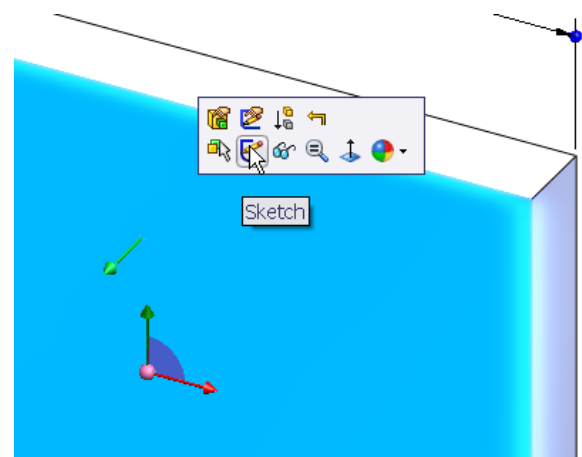
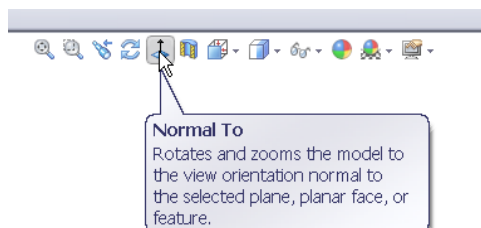
Removing the semi-hexagonal portions

The sketch used to create the Extruded Cut feature will be created on the surface of the *plate* feature

Left Click on the surface and choose **Sketch** from the pop-up toolbar

A new sketch will be created on the surface.

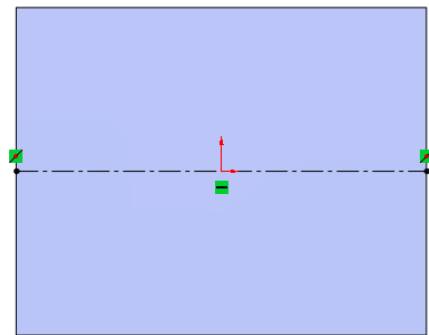
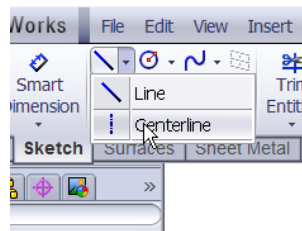
Choose **Normal To** from the heads up toolbar.



The view rotates to display the surface as a true shape.

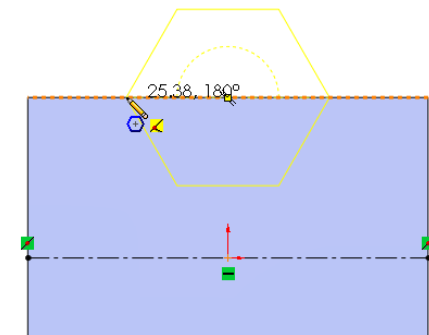
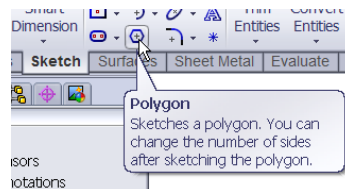
Centreline

Sketch a centreline with endpoints coincident with the midpoints of the vertical lines.



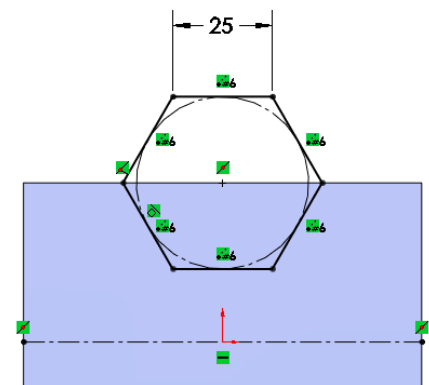
Polygon

Sketch a hexagon with centre coincident with the midpoint of the horizontal edge and one vertex coincident with the horizontal edge.



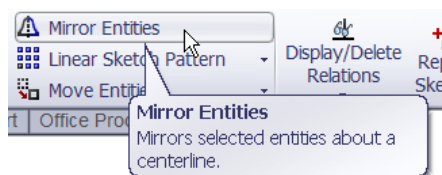
Smart Dimension as shown across.

The sketch turns black and is fully defined.



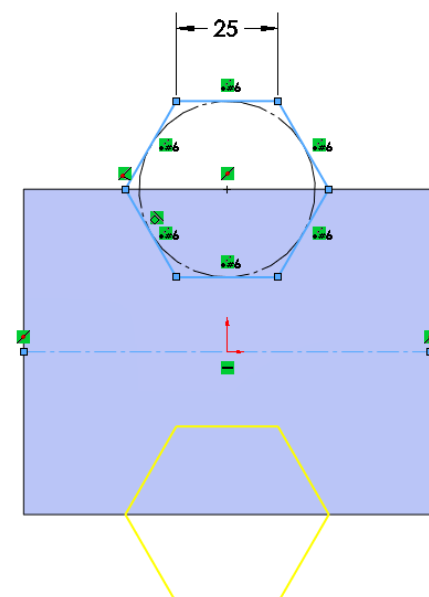
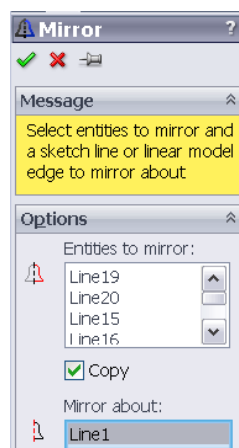
Mirror

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



Choose the 6 lines of the hexagon as **Entities** to mirror and the centreline as the line to **Mirror about**.

Choose **OK**.



Extruded Cut

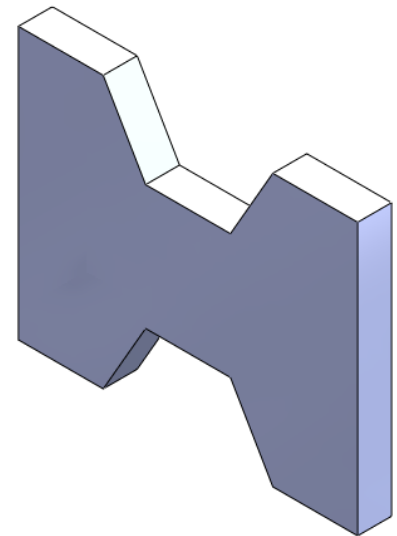
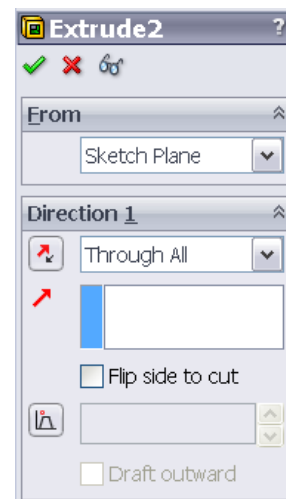
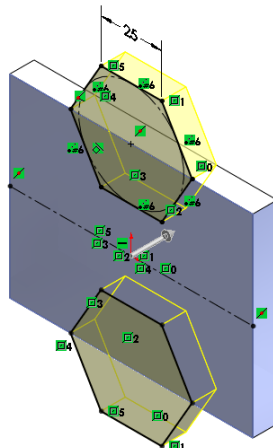
Choose **Extruded Cut** from the **Features** tab.



Select **Through All** End Condition.

Note: This ensures should the width change the hexagonal cuts will still cut all the way through.

Choose **OK**.



Rename the feature **hexagonal cuts**

Adding the holes

To add the holes we will create the sketch shown on the face of the plate. The added dimensions will fully define the sketch.

The other holes are symmetrical located on the face of the plate.

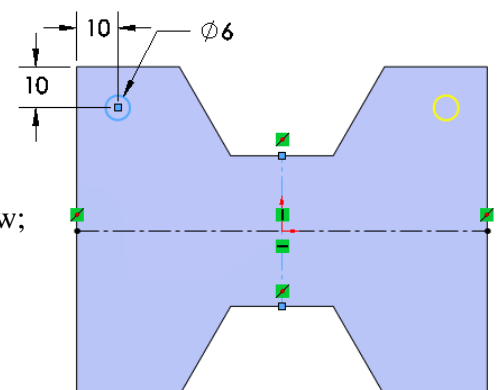
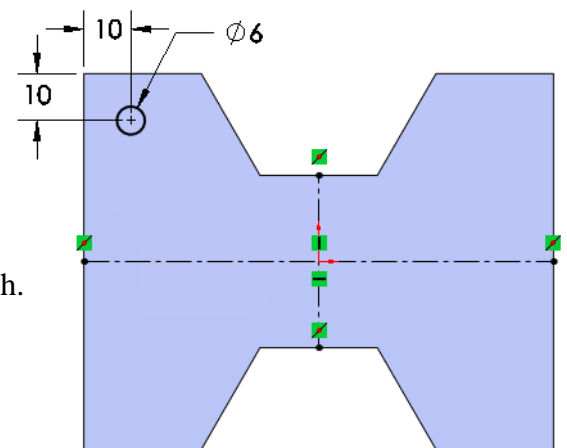
Rather than redrawing 3 more holes we will mirror this hole across both centrelines individually.

Mirror

Choose Mirror from the sketch toolbar. Make the selections outlined below;

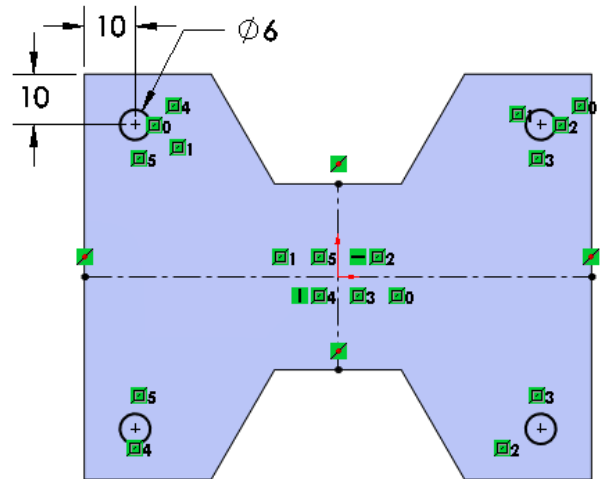
- Entities to mirror - **Circle**
- Mirror about - **Vertical Centreline**

Choose **OK**



Repeat this procedure to sketch the bottom holes.

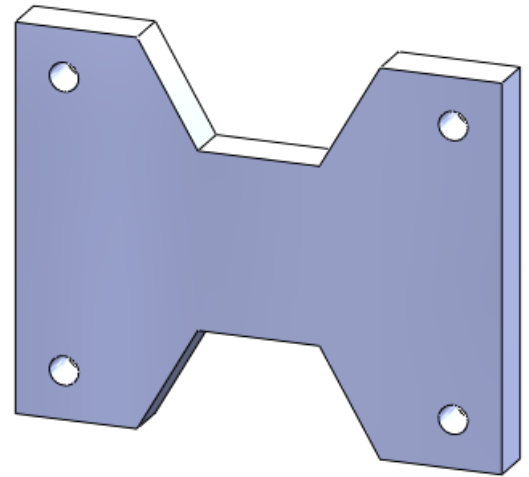
Note: Should the hole size or location need to be changed, it may be done by changing one dimension. This is known as **design intent**.



Extruded Cut

Extrude Cut the holes using a **Through All** end condition.

Rename the feature **6mm holes**



Tessellation

The display of geometry on your system may appear somewhat different from the illustrations. The lines may appear rougher. This is called tessellation. Tessellation or line display is related to the performance of the computer. Higher quality graphics or higher system settings will improve model appearance.

Choose **Tools, Options, Document Properties. Image Quality...**

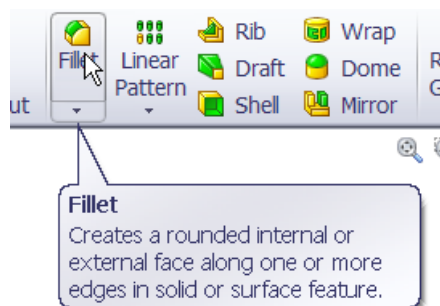
Inserting a Fillet Feature

The **Fillet** feature rounds sharp edges and faces. We will use the **Fillet** feature to round the sharp edges of the TV Plate. The Fillet feature requires an edge or face with a specified radius.

In general, it is best to follow these few rules when inserting a fillet:

1. Add larger fillets before smaller ones. When several fillets converge at a vertex, create the larger fillets first.
2. Add drafts before fillets. If you are creating a moulded or cast part with many filleted edges and drafted surfaces, in most cases you should add the draft features before the fillets.
3. Save cosmetic fillets for last. Try to add cosmetic fillets after most other geometry is in place. If you add them earlier, it will take longer to rebuild the part.

Choose **Fillet** from the features toolbar



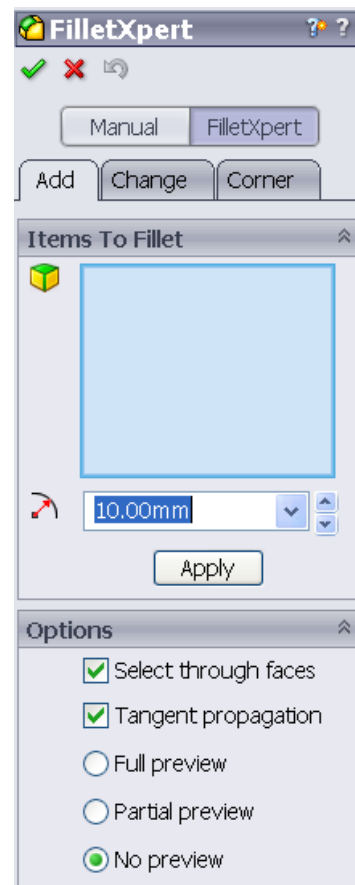
Two PropertyManager tabs are available:

1. **Manual**. Use this tab to maintain control at the feature level similar to the use of fillet command in SW2006.
2. **FilletXpert**. Use this tab when you want the SolidWorks software to manage the structure of the underlying features for a constant radius fillet.

Fillet / FilletXpert PropertyManager

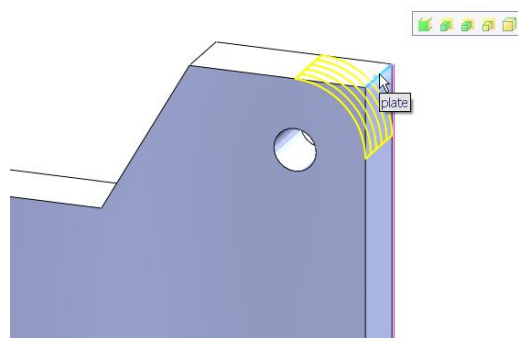
The FilletXpert PropertyManager is displayed when you click the FilletXpert tab in the Fillet PropertyManager. The FilletXpert manages, organises, and reorders constant radius fillets. Use the **Add** tab to create new constant radius fillets.

Use the **Change** tab to modify existing fillets. Use the **Corner** tab to create and manage fillet corner features where exactly three filleted edges meet at a single vertex.



Choose the **FilletXpert** tab. Choose the **Add** tab.

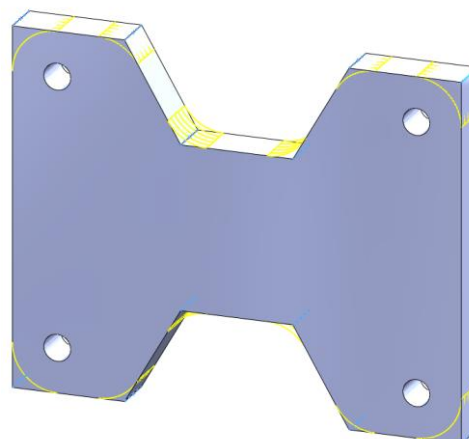
Choose the edge of the TV Plate as shown. A context sensitive toolbar appears.



Moving the cursor over the buttons will select various other edges to add the fillet to. Choose the first one. This will include the remaining 11 edges in the selection.



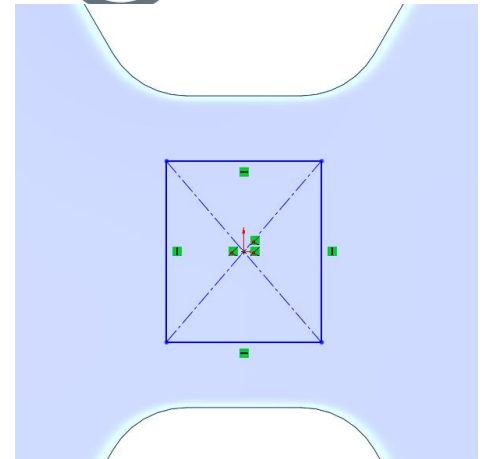
Choose **OK**



Adding the swivel bar to the back

Using Centre Rectangle, create the sketch shown on the back face of the plate.

Note: Prudent positioning of the origin at the beginning makes it easy to locate the sketch in the required position.



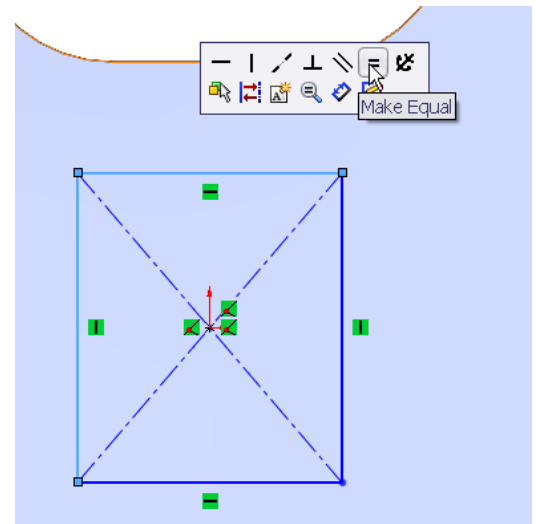
Equals Relation

The swivel bar has a square cross section. We will add an equals relation between the two sides of the rectangle.

*Note: To make multiple selections in solid works; Hold down the **Ctrl** key whilst making the selections*

Add Relation - Choose the horizontal side of the rectangle, hold **Ctrl** and choose the vertical side.

Release **Ctrl** and choose the appropriate relation from the pop-up menu. Choose **OK**.



Smart Dimension

Smart dimension the side of the square **20mm**. The sketch is now fully defined.

Extruded Boss/Base

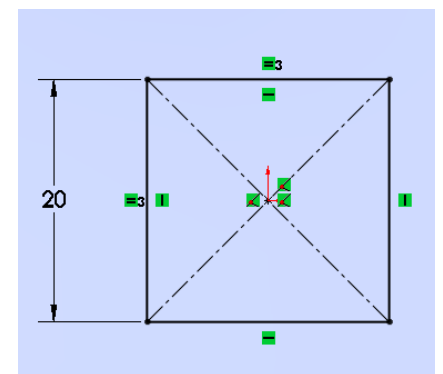
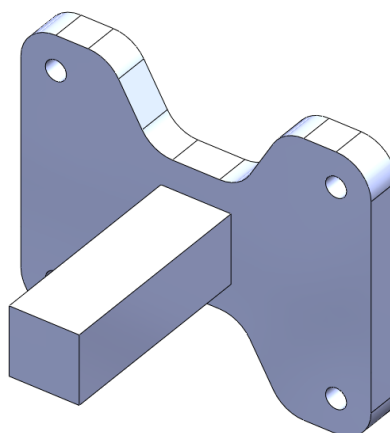
Extrude the sketch using the following settings;

Depth - **60mm**

End Condition - **Blind**

Choose **OK**

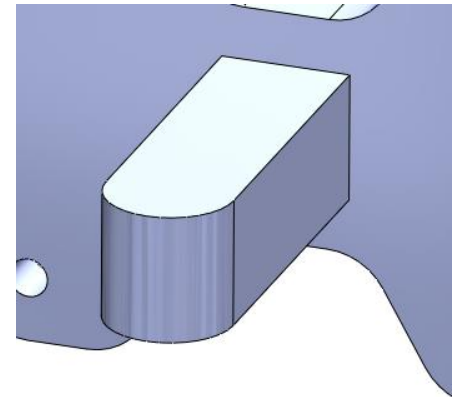
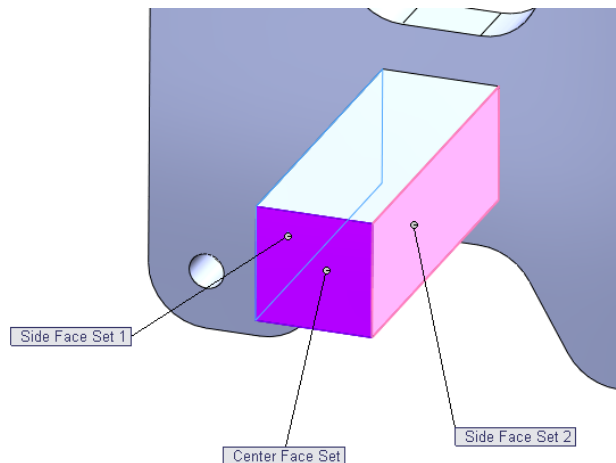
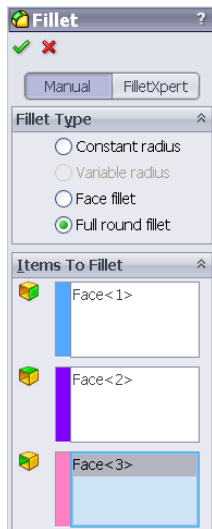
Rename the feature **Swivel**



Full round fillet will create a complete filleted edge between adjacent faces.

Choose **Fillet** from the feature toolbar. Choose the **Manual** tab and select **Full round fillet**

Make the selections shown below. Choose **OK**



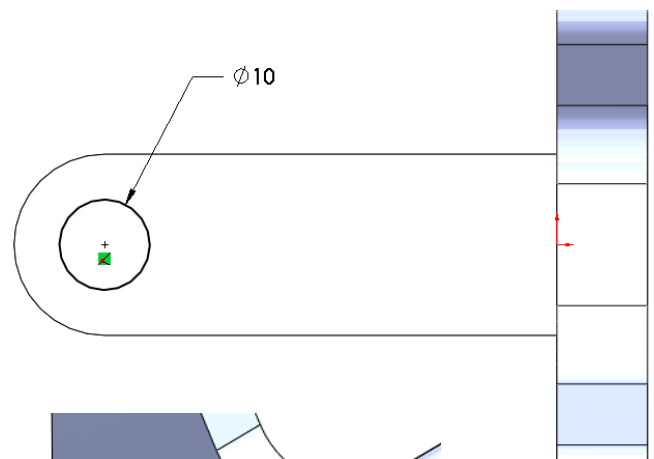
Note: The front of the swivel is completely rounded. Should the size of the cross section change the fillet size will update and remain fully rounded. The centre of the fillet may now be used to locate the centre of the $\varnothing 10\text{mm}$ hole.

Creating the hole in the swivel bar.

Create a sketch on the top face of the swivel.

Sketch a circle with centre coincident with the fillet centre.

Smart dimension as shown.

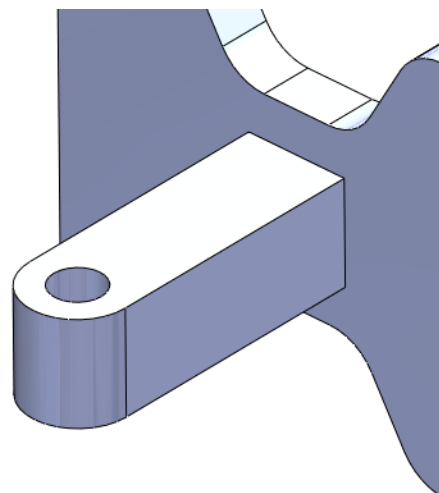


Extruded Cut

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

Choose OK

Rename feature **10mm hole**



RealView Graphics

With RealView enabled appearance and scenes may be added to create photorealistic views of a model.

RealView Graphics uses the graphics card to give more advanced shading in real time, when the model is rotated the rendering is retained.

Once a physical material has been applied RealView Graphics may be used to give the model a realistic appearance i.e. a steel physical material could be applied to a model, to facilitate physical properties, but it could then be enhanced with the appearance of glossy paint.

To enable **RealView** choose **View, Display, RealView Graphics...**

Appearance

Appearances are the likeness of how a material would look without adding the physical properties.


A hierarchy exists when adding appearance to a model;

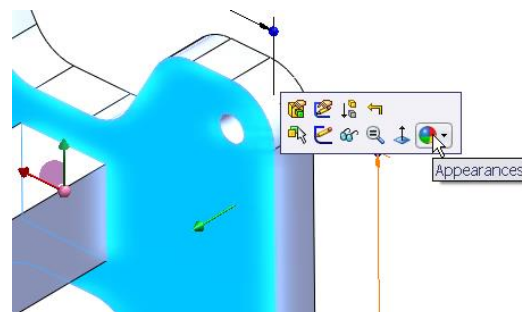
- Faces override all other instances in a model
- Features override bodies but not faces
- Bodies override documents



By default the model will appear in default plastic

Adding an appearance to the TV Plate.

Right click on any face of the model and choose **Appearances** 

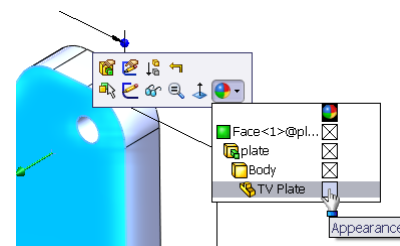


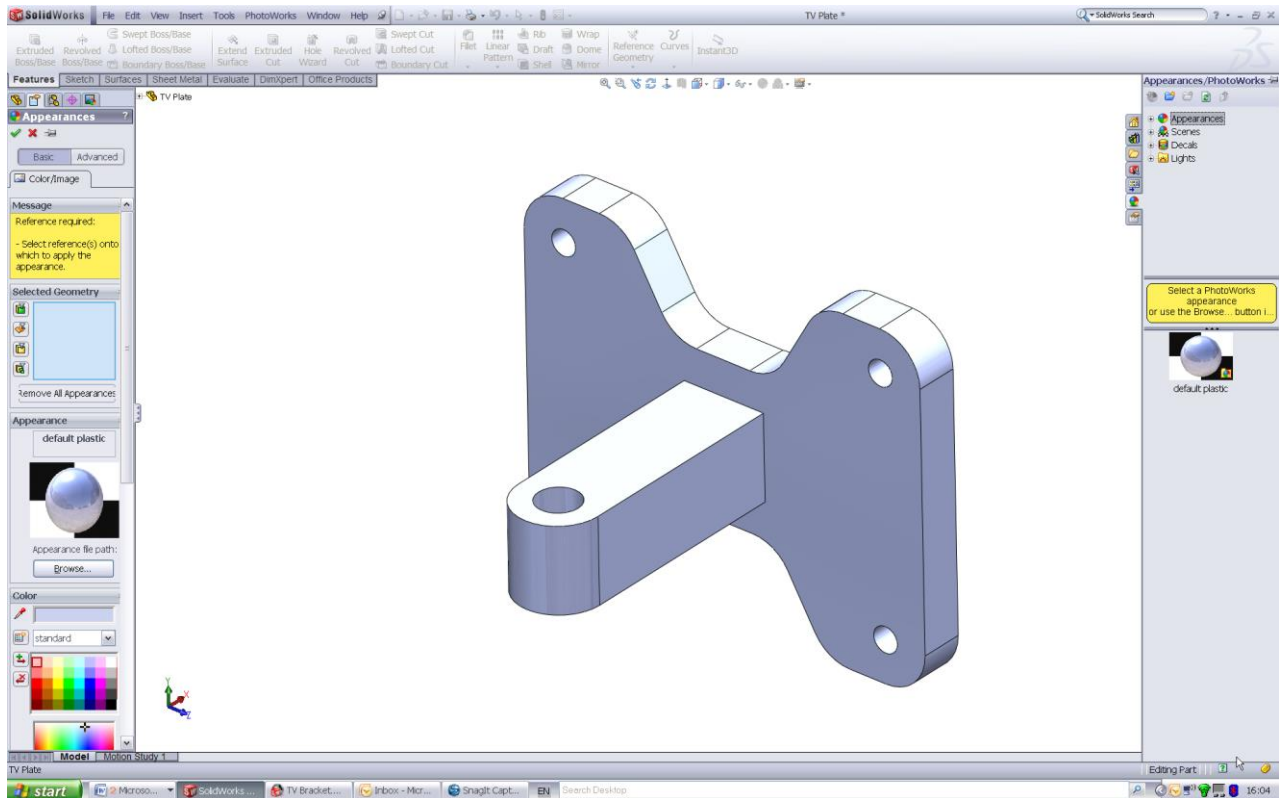
A drop-down menu appears with the various aspects of the model in hierarchical order.

Click on the colour swatch next to the part name, TV Plate.

The **RealView/PhotoWorks Tab** appears on the right hand side of the screen and the **Colors Appearancemanager** on the left as shown overleaf.

A **preview window** appears to preview the resulting render.





Within the **Appearances/PhotoWorks** tab we can access appearances, scenes, decals and lights folders. Choose **Appearances**. The categories are listed on top with the various options within each category displayed below.

Navigate to **Painted, Car, Metallic Cool Grey**.

Drag the appearance from the task pane and drop it onto the part in the graphics window.

The appearance is applied to the entire part.

The colour may be changed by choosing from the colour swatch in the Appearance PropertyManager.

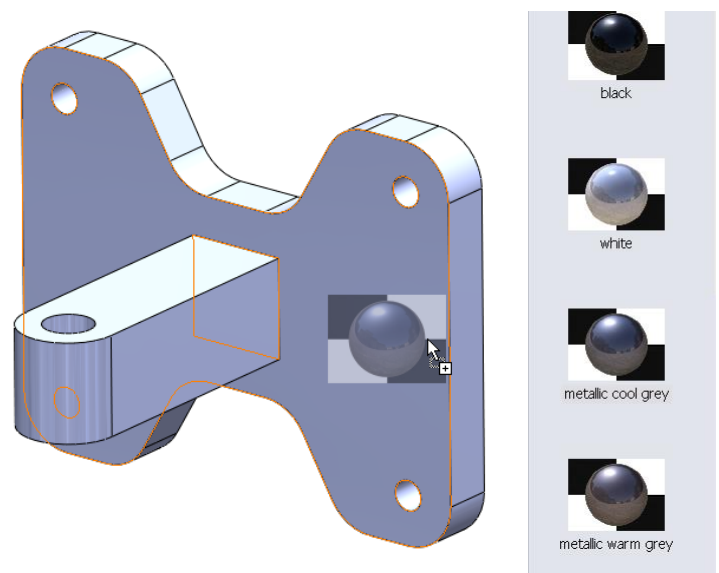
Choose **OK** 

How would I apply a different colour to a face?

Right click on the face and choose **Appearances**.

In the dropdown menu choose the **face** option.

Follow the same procedure as before for applying the colour.



Adding a scene

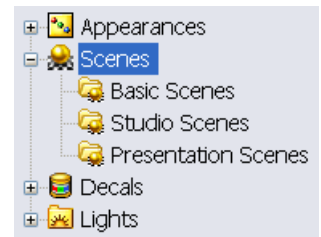
Document scene backgrounds are used to enhance appearances. Scenes are made up of a background and the addition of lighting. Within the task pane scenes are placed in three categories; Basic, Studio and Presentation.

To view the various scenes choose  from the task pane. Choose **Scenes**.

Double click on any of the folder categories to display a preview of the various scenes within that category

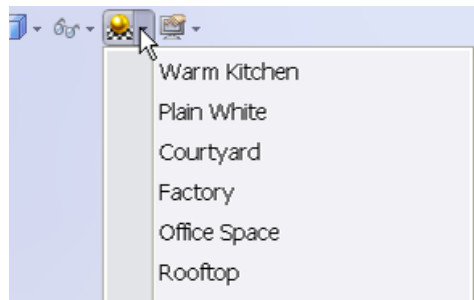
Drag and drop the chosen scene from the task pane into the graphics area.

The background updates immediately to reflect the changes.



Investigate the various scenes available within the various categories by dragging and dropping them into the graphics area.

These scenes may also be accessed, without preview, under **Apply scene** on the heads up toolbar



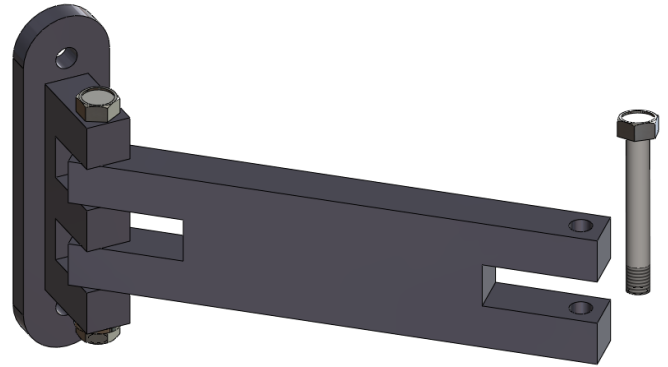
Note: A plain white background is available in the basic scenes folder, to facilitate screen grabs, saving as a jpeg etc.



Assemblies

Open the assembly file, **TV Bracket**, from within the TV Bracket folder. Some of the assembly has been completed with parts mated appropriately.

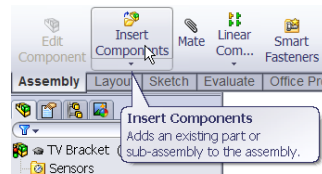
The bolt M10 X 70 bolt has also been inserted but not mated to the assembly.



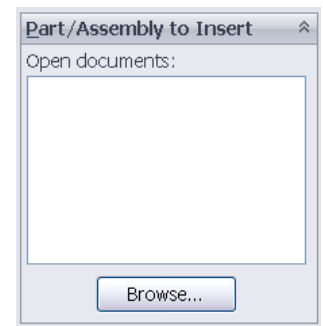
We will insert the TV Plate, which we have modelled, and mate it to the existing parts. We will then mate the bolt and add the washer and the nut.

Insert Component

Choose **Insert Component** from the **Assembly** tab.

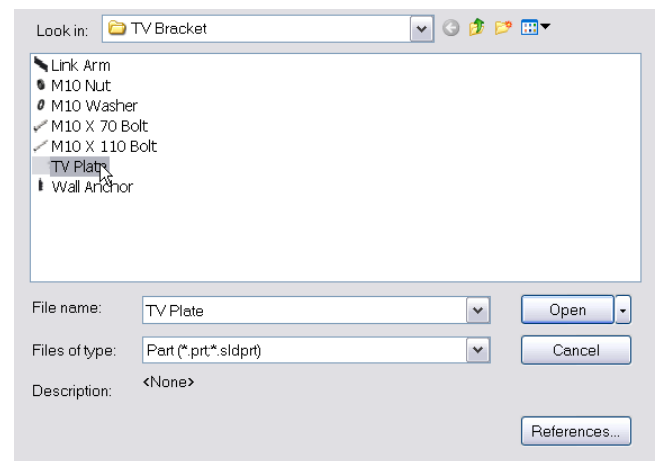
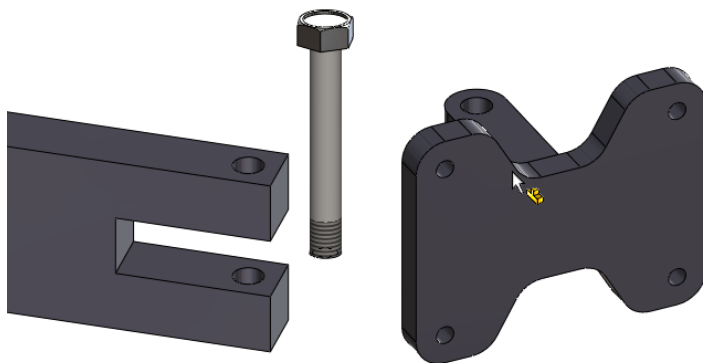


If TV Plate is opened it will appear in the **Open documents** window, otherwise choose **Browse...**



Navigate to the **TV Bracket** folder in which you saved **TV Plate** and select the part file. Choose **Open**

Left click in the graphics area to drop the part in the position shown below.



Note: In a number of cases in the DCG Student Assignment 2009 the assembled parts were not properly mated, just merely moved into position. It is essential that the parts are mated appropriately and when dragged will behave as expected.

Applying Mates

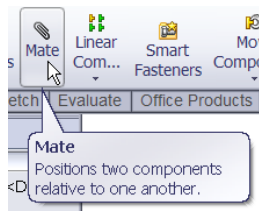
Before we begin to apply mates we will look at a useful skill which we will use throughout the process.

It is often necessary to rotate the assembly when selecting elements of the parts for mating. The assembly may be rotated by holding down the middle mouse button and dragging the mouse. Give it a try!

Mates may be done in any order. The hole of the TV Plate must be located concentrically with the hole in the link arm. We will create this mate first.

Choose Mate from the Assembly toolbar.

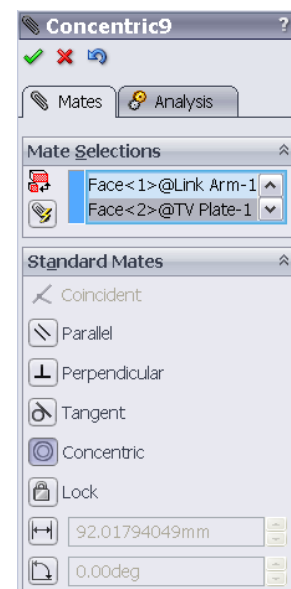
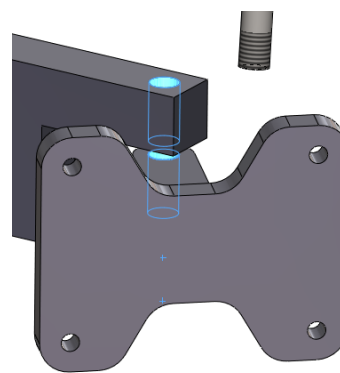
Choose the inside of the hole on both the Link Arm and the TV Plate.



SolidWorks automatically chooses a mate type appropriate to the geometry selected.

In this case a **Concentric Mate**.

The parts move into position to reflect the chosen mate. This mate may be accepted by selecting OK or another mate type may be chosen. Choose **OK**.



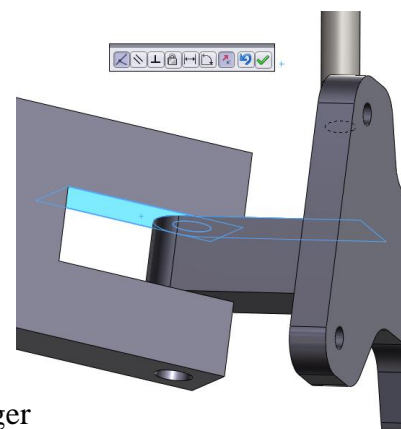
The part is now confined to move such that the holes remain concentric but is not yet sufficiently defined. The mate property manager remains displayed awaiting more selections to further define the position of the part.

What geometry of the parts/mate type would you choose next?

The surfaces of both parts will be in contact with each other during rotation of the parts hence we will mate these surfaces using a **Coincident Mate**.

Choose the top surface of the TV Plate, rotate the assembly as described Earlier (using the middle mouse button) and choose the underneath surface of the 'u slot' in the link arm.

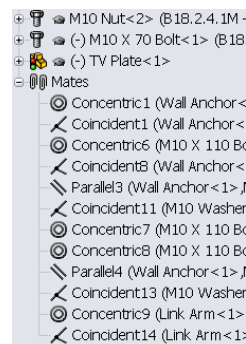
SolidWorks automatically applies a coincident mate type to suit the geometry selected. Choose **OK**. Choose **OK** to exit the Mate Property Manager



Note: The mate type may be also be chosen from the pop-up toolbar which appears when the surfaces are selected.

Click and hold any edge of the TV Plate and drag the mouse. The TV Plate along with the rest of the assembled parts moves and behaves as you would expect due to the mates applied.

Note: All mates in the assembly may be viewed in the mates folder at the base of the featuremanager design tree.



Positioning of the M10 X 70 Bolt.

Add the mates outlined below between the assembly and the bolt to position it correctly.

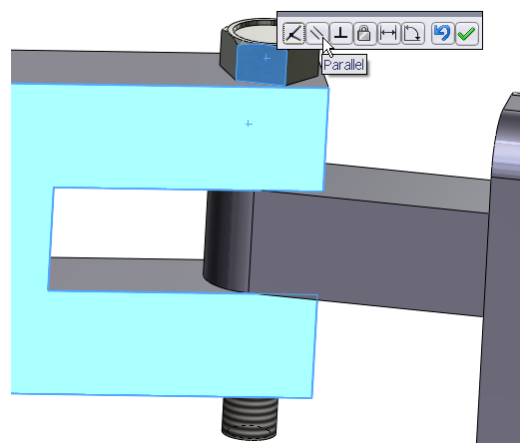
Underneath surface of the head of the bolt + top surface of the link arm – Coincident Mate

Cylindrical surface of the bolt + Cylindrical surface of the hole – Concentric Mate

How do you prevent the bolt from rotating?

The bolt may be prevented from rotating by applying a parallel mate between the surface of the head of the bolt and the surface of the link arm. Complete this mate.

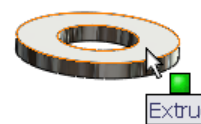
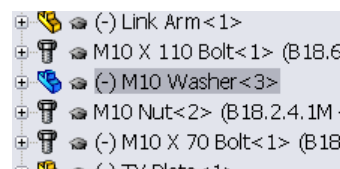
Select and drag the edge of the bolt. It will no longer rotate.



Inserting the nut and washer.

The nut and washer may be inserted in the same way as the TV Plate. However, these parts already exist in the assembly. When parts already exist within an assembly a second copy may be inserted very quickly.

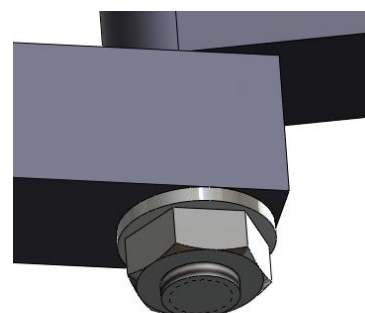
Hold Ctrl, left click and drag M10 Washer, from the featuremanager tree into the graphics area. Release the left hand mouse button to drop it in position



Repeat this procedure to insert the nut.

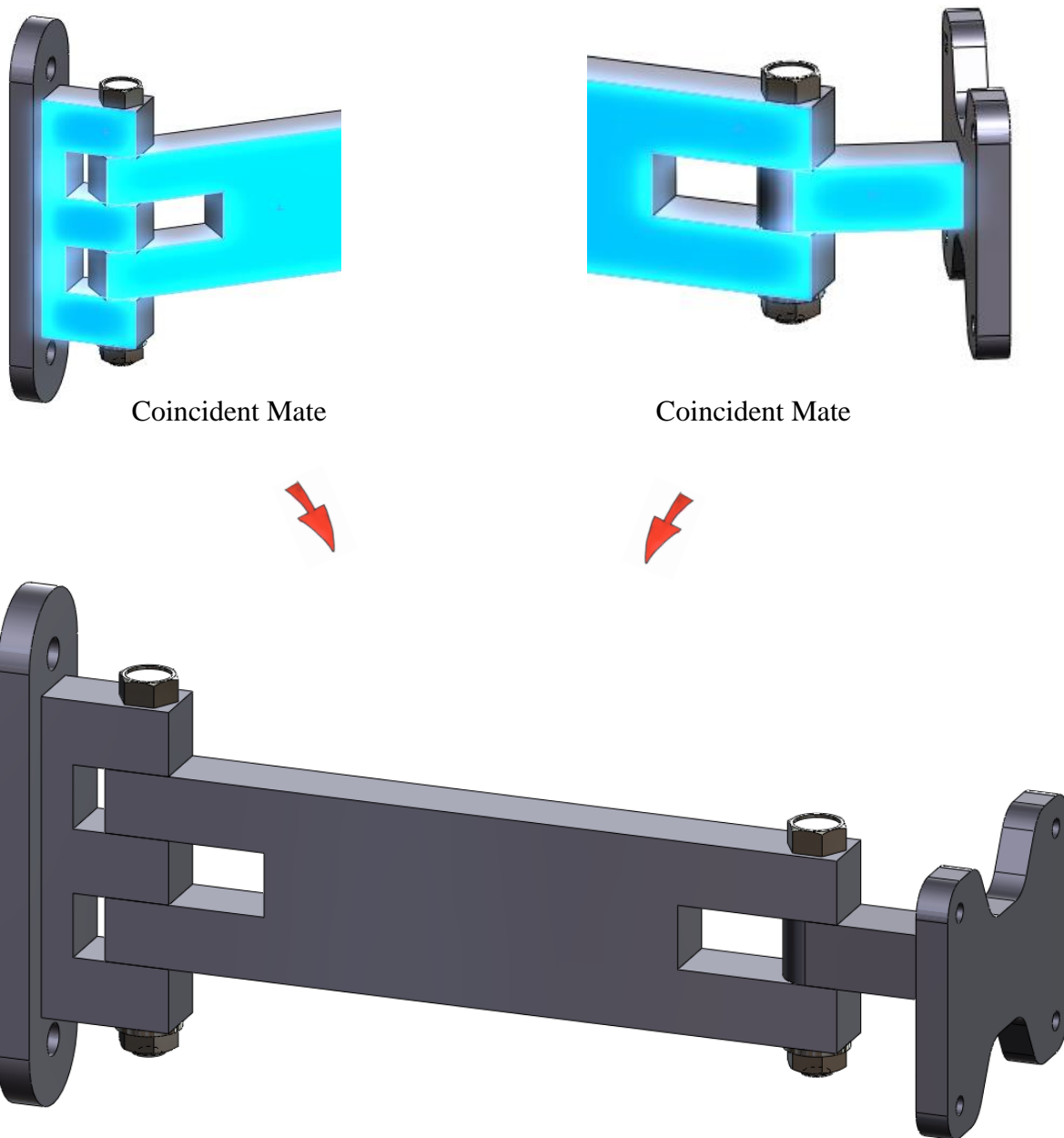
Apply appropriate mates to position the washer and nut in relation to the assembly.

How do we stop the nut from rotating?



We want to create a drawing of the assembly in the fully extended position. For that reason the assembly must be positioned in a fully extended position. Because SolidWorks parts, assemblies and drawings are associative should the assembly be repositioned the drawing would update and no longer display our TV Bracket fully extended.

For this reason we will apply two mates to constrain our TV Bracket from moving. We will apply two separate coincident mates between the faces shown below.



Save the assembly.

Creating a drawing

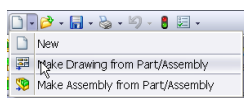
SolidWorks allows you to create drawings from parts or assemblies. These drawings are fully associative with the parts and assemblies they reference. If dimension is changed on the finished drawing, that change propagates back to the model. Likewise, if the model is changed, the drawing updates automatically.

Drawings communicate three things about the objects they represent:

- **Shape** – Views communicate the shape of an object.
- **Size** – Dimensions communicate the size of an object.
- **Other information** – Notes communicate non-graphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treatment, remove burrs, and so forth.

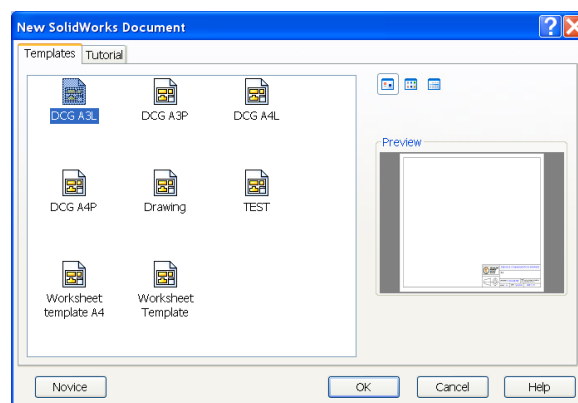
Note: When creating drawings it is essential that the time is taken to plan the sheet layout to ensure that the views chosen best present the information required. It is important that an appropriate scale is chosen and dimensions are edited and presented to reflect appropriate standards. Should you wish to display particular detail a section or detail view may be chosen.

Select **New, Make Drawing from Part/Assembly**, from the standard toolbar.



Choose the DCG A3L template from the available list.

The Drawing document is opened with the DCG A3L template displayed.



Note: To change the drawing paper to plain white choose;

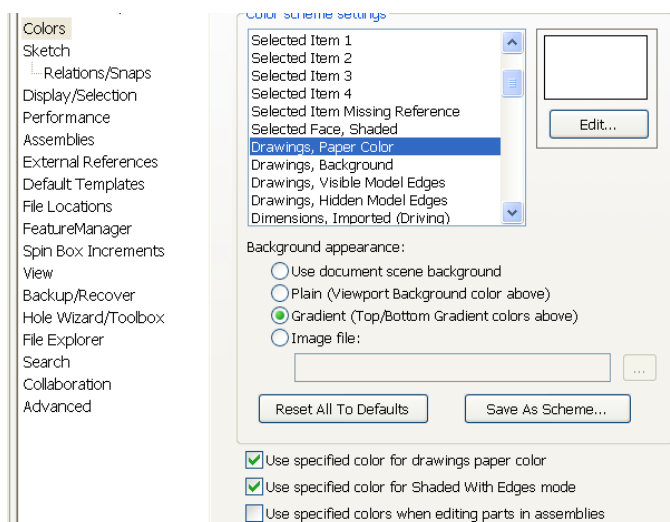
Tools, Options, Colors, Drawings Paper Color

Select **Edit** and choose **White**. Choose **OK**.


Select **Use specified color for drawings paper color**

Choose **OK**

The drawing sheet will now be plain white.



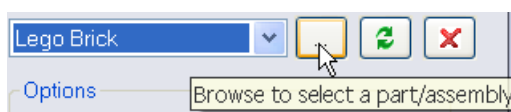
The view palette, located on the task pane may be used to insert drawings views.

Note: The **View Palette** tab  provides the ability to click and drag drawing views into an active sheet.

The View Palette contains images of standard views, annotation views, section views.

TV Plate is chosen as the model from which to create the drawings as it was accessed when we selected **Make Drawing from Part/Assembly**.

A different model may be chosen using the browse feature.



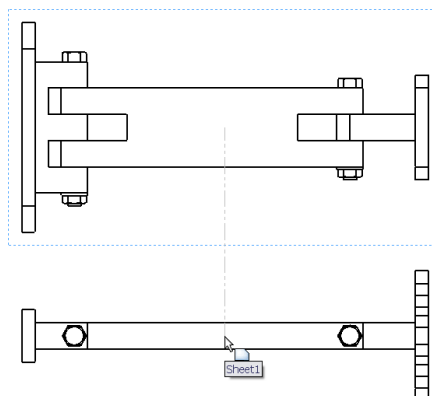
Importing drawing views.

Note: Ensure **Auto-start projected view** is ticked. This will allow us to project other views from the parent view.

Click and drag the **front view** from the view palette onto the drawing sheet.

By moving the cursor down a **plan** is projected from the parent view. Click to add the view.

Project a plan view from the parent front view also. Press Esc to stop **Auto-start projected view**



Drag the **Isometric View** from the view palette.

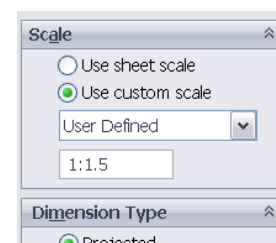
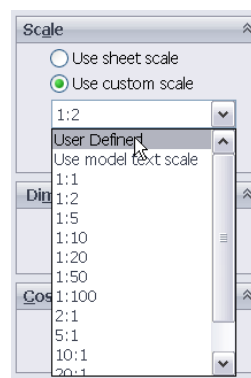
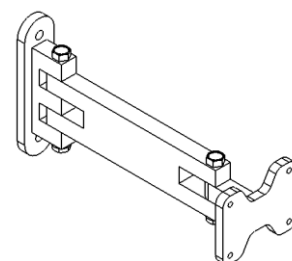
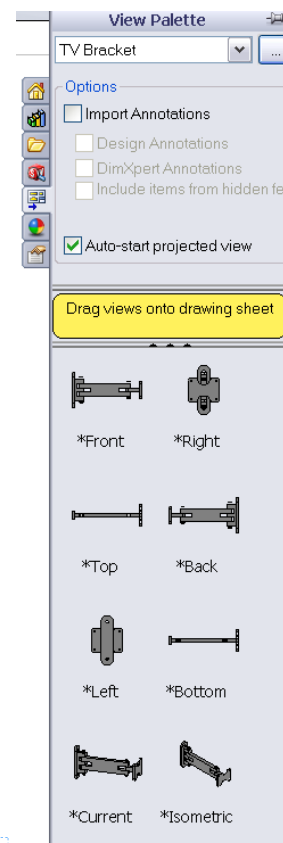
Choose the front view (parent view) and change the **scale** to **1:1.5**.

*Note: Select the down arrow and choose **user defined**. Input **1:1.5** in the text box underneath.*

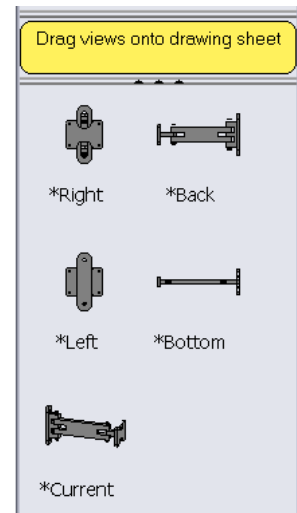
Note the end view and plan update to reflect the same scale but the isometric view doesn't. *Why?*

Because the end view and plan were projected from the front view they adopt the same scale. The isometric view was dragged from the view palette independently and hence keeps its original scale.

Reposition the views by dragging, to position them appropriately on the sheet.



Note: When the views are dragged from the view palette they disappear from it and are no longer available.



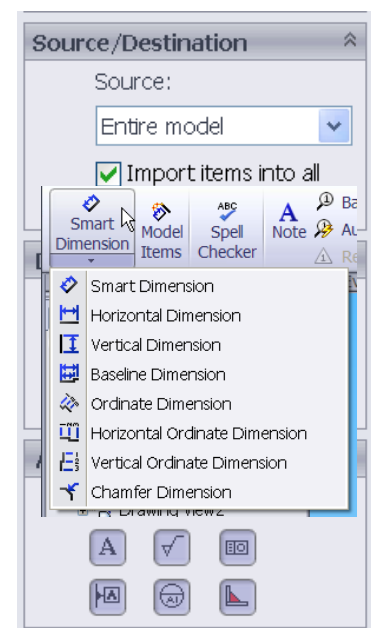
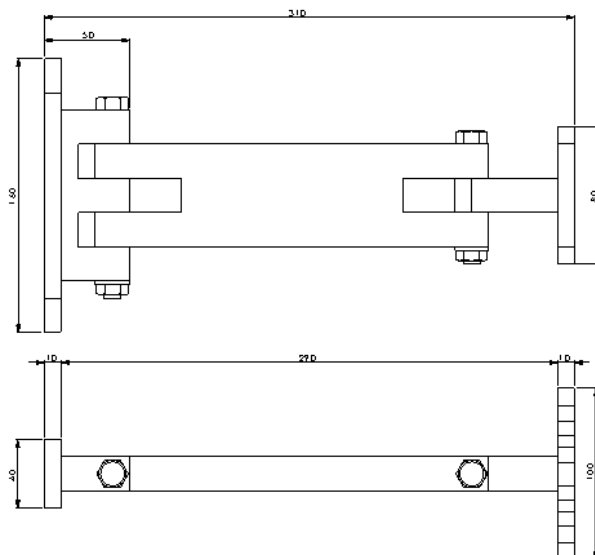
Adding dimensions

Choose **Model Items** from the annotations tab of the **commandmanager**



Choose the settings shown across. Choose **OK**

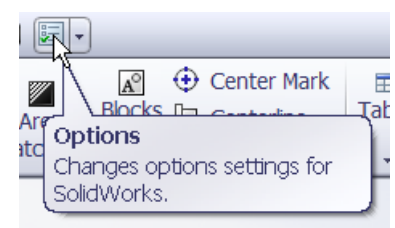
Drag the dimensions into the desired positions. Use Smart Dimension from the Annotation tab to add further dimensions if required.



Dimension settings, including arrow and text size may be edited by choosing **Options** from the standard toolbar.

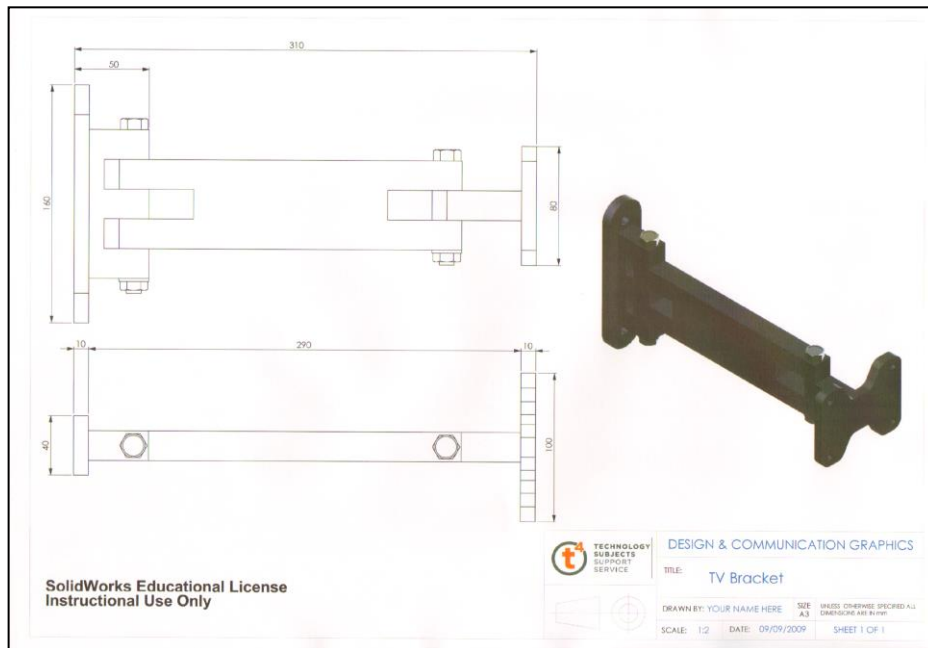
Choose the **Document Properties** tab and navigate to **Dimensions**

Save the drawing as **TV Bracket** in the **TV Bracket** folder.

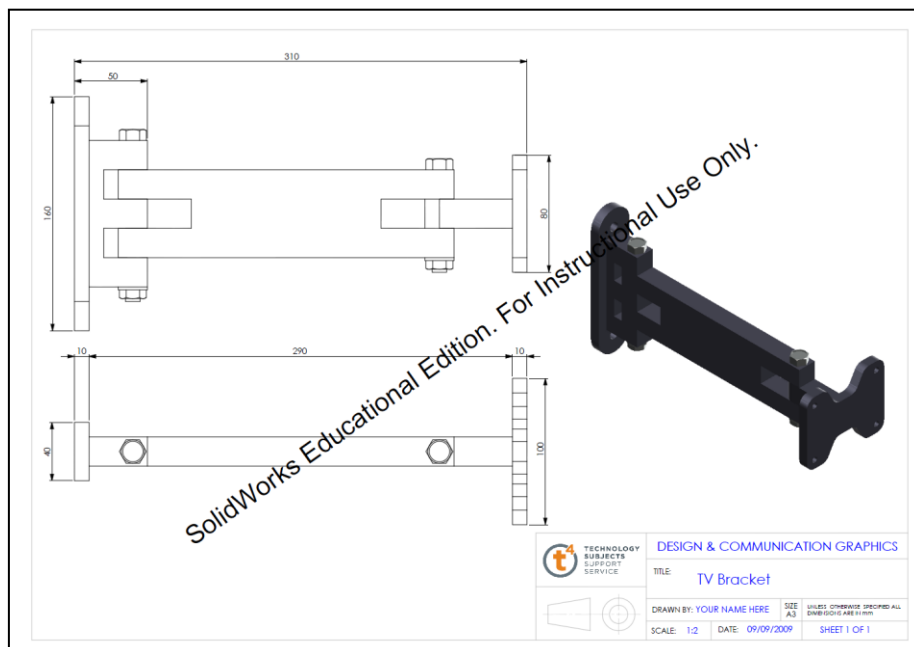


Printing a drawing

A SolidWorks drawing may be printed directly from SolidWorks. When printed, a watermark will appear in the bottom left hand corner, as shown below.



In SW2006, saving in a pdf format would eliminate the watermark. Saving as a pdf in SW2009 yields the output shown below.



It is advised to print from SolidWorks and avoid placing any drawing views or text in the position where the watermark will appear.

A free to download pdf creator (eg primo pdf, cute pdf) could be used to create the pdf and hence eliminate the need to scan the printed SolidWorks drawing. However, the watermark will still appear on the pdf.

Saving an e-drawing

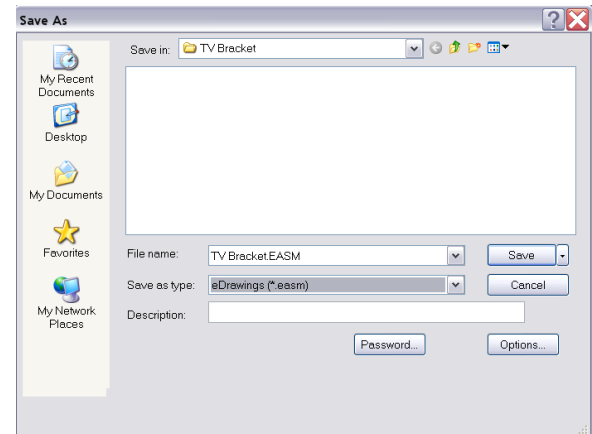
An eDrawing is a file format which allows a SolidWorks document to be viewed in an eDrawing viewer or **eDrawings 2009** without having the SolidWorks software.

Saving the assembly as an eDrawing is a requirement of output 4 of the DCG Student Assignment. In a number of cases this year the eDrawing was omitted.

Saving a SolidWorks document as an eDrawing

With a SolidWorks document open:

1. Click **File, Save As**.
2. Input the **File name**.
3. Select **eDrawings (*.easm)** from the **save as type** options.
4. Click **Save**.



The file will appear as shown in the TV Bracket folder .

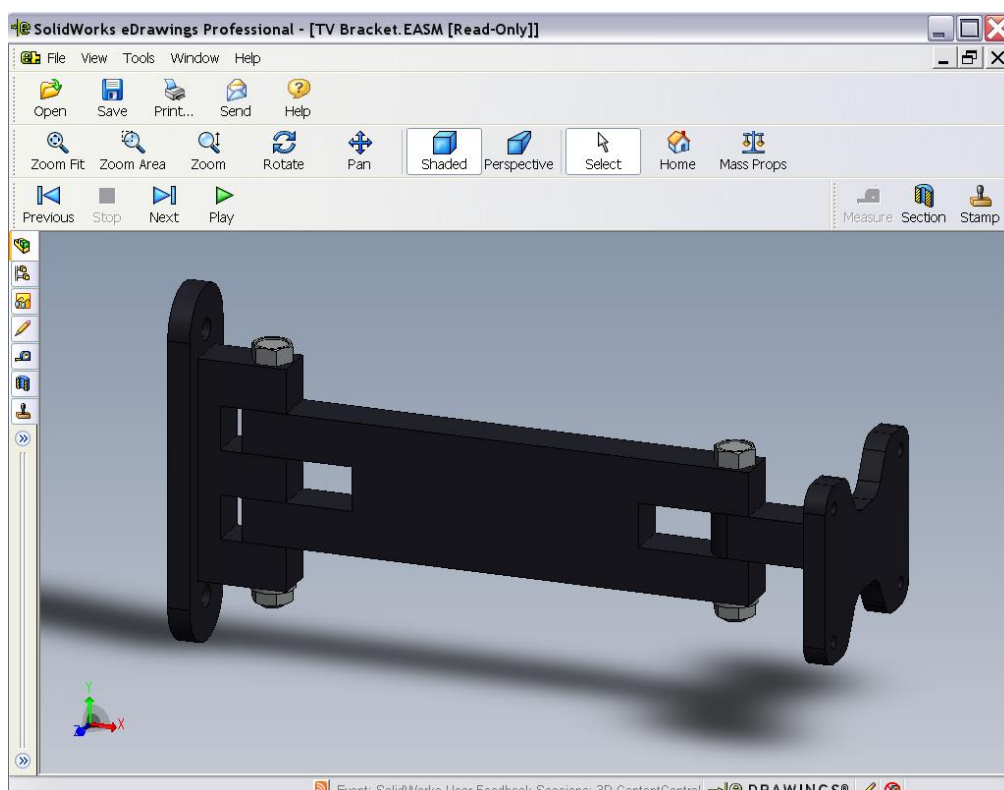
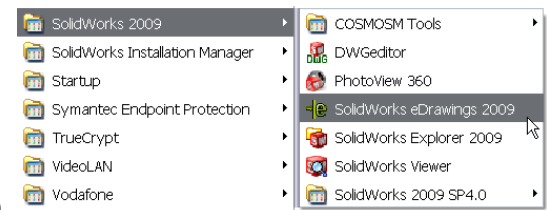
The file may be opened in eDrawings 2009.

This may be accessed through;

Start, All Programs, SolidWorks 2009, SolidWorks eDrawings 2009

Note: Double clicking on the eDrawing file may not launch the program.

The interface appears as shown below.



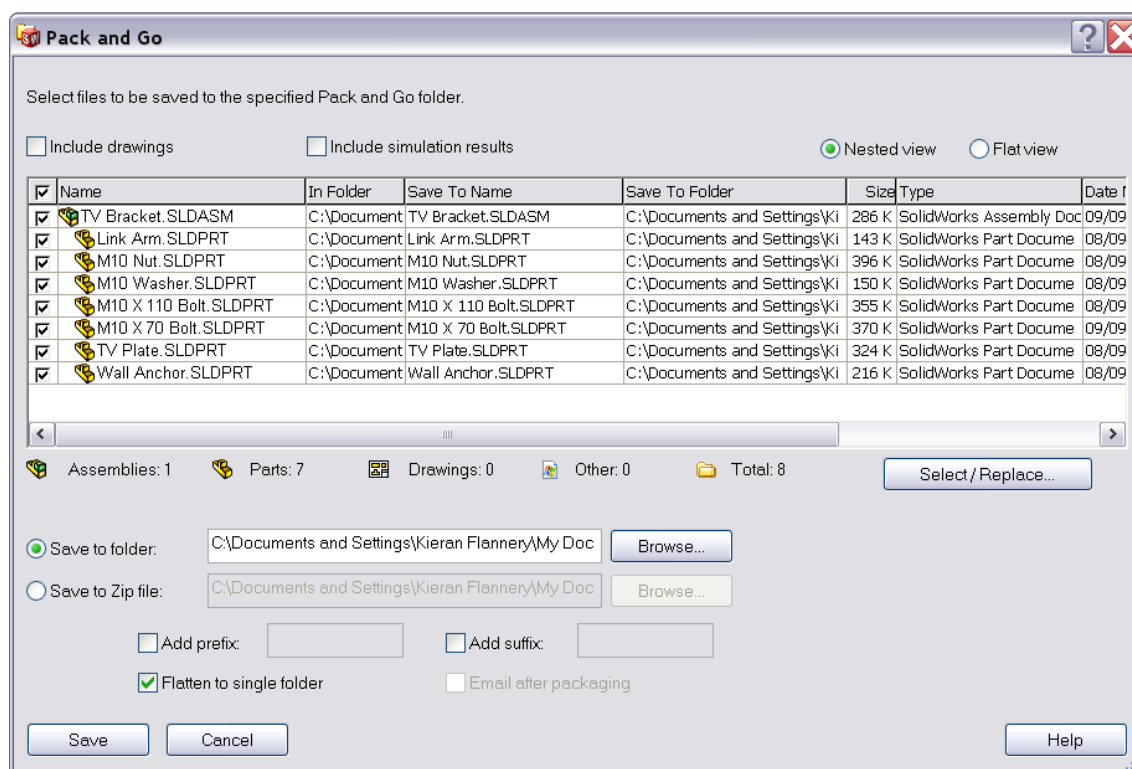
Pack and Go

Most students demonstrated excellent file management skills in the Student Assignment 2009 however, in a small but significant number of cases one or more parts were not included on the CD. Whilst there is no substitute for good file management, as we have practiced throughout this exercise, SolidWorks 2009 has a feature which may help to reduce this problem.

Pack & Go gathers all related files for a model design (parts, assemblies, and drawings) into a folder or zip file. We will use the TV Bracket project to demonstrate this feature.

We will first create the folder into which the files will be saved. Create a folder in an appropriate location and call it **TV Bracket Complete**.

With the TV Bracket assembly opened choose **File, Pack and Go...** The window shown below will appear.



The details of the files which are associated with the assembly are displayed in the display window along with the number of assemblies, parts etc underneath.

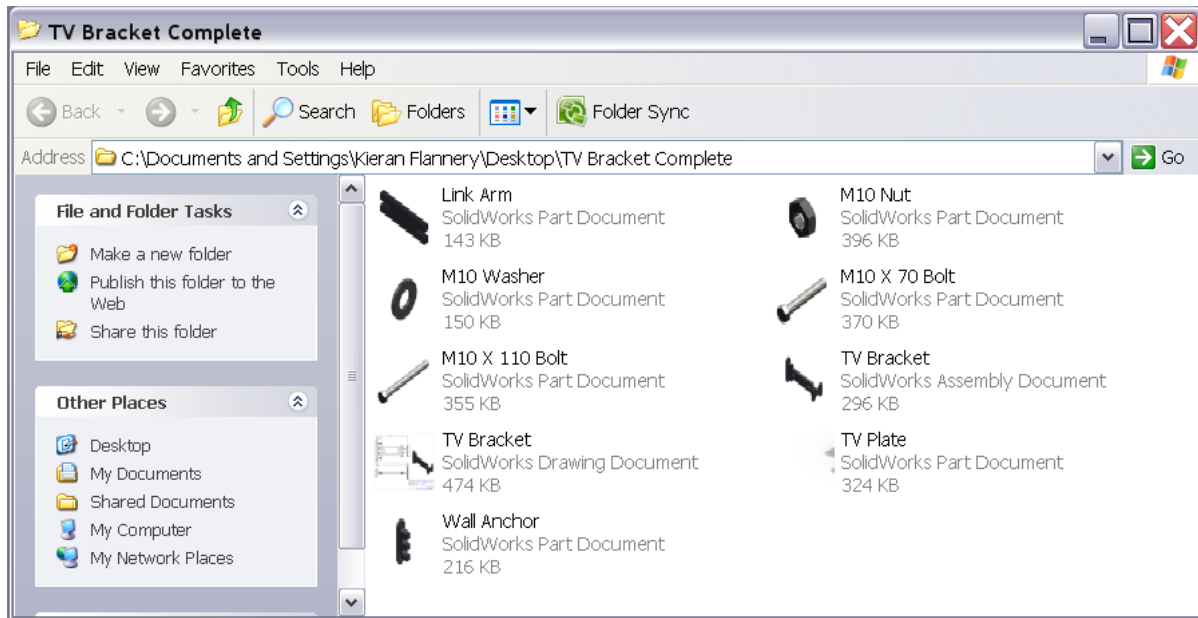


Check **Include drawings** ☒ Include drawings to ensure all drawing files are included in the new destination folder

Note: Should a file be stored in a different location SolidWorks will find it and include it in the list.

Choose **Browse** and locate the folder you have just created to contain the parts. Choose **OK**. Choose **Save**

Navigate to the folder into which the parts were saved. Open the folder. All of the Assembly, Part and Drawing files are present and displayed as shown below.



Pack and Go cuts all ties with the original files. Therefore, any changes made to these files will not be reflected in the originals. This can prove to be very useful.

Photorealistic Imaging

A significant number of students used PhotoWorks to produce high quality photorealistic images of their models. Some images were taken directly from SolidWorks and were accepted for this first year of the DCG Student Assignment

In future it is expected that all students will use either PhotoView 360, PhotoWorks or some other image manipulation software to create the required images.

We will take a look at producing a photorealistic image using both PhotoWorks and PhotoView360.

PhotoWorks

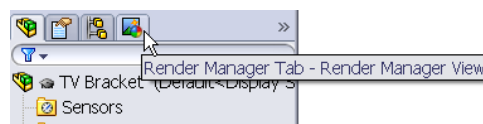
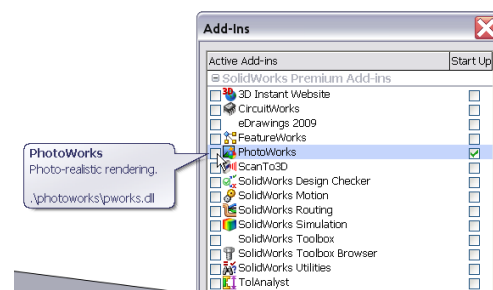
PhotoWorks is an Add-In to SolidWorks and hence must be 'added in' to make it available.

Choose **Tools, Add-Ins** from the drop down menu.

Tick the PhotoWorks Checkbox.

Choose **OK**

The PhotoWorks dropdown menu will appear along with the render manager tab.



PhotoWorks is integrated into SolidWorks 2009 much more so than 2006. Appearances and scenes are applied in the SolidWorks graphics area and are used in PhotoWorks to generate the photorealistic image.

Open the TV Bracket Assembly.

Appearance - Earlier in this exercise (*page 14 & 15*) we applied a **Painted, Car, Metallic Cool Grey** appearance to the TV Plate at part level. A similar approach was taken with the various other parts. Therefore all parts appear with an appearance applied to them.

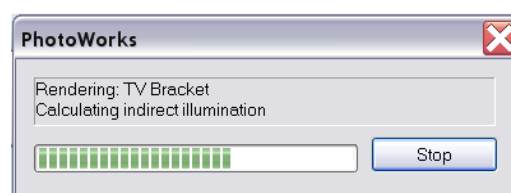
Scene – Change the scene by dragging the scenes from the Appearances/Scenes tab in the task pane, as discussed on page 16.

Render

Note: Once the scene has been selected it must be rendered to see the image. Before rendering orientate the model and zoom in or out to position the required view. Whatever is displayed on screen is what will be captured in the photorealistic image.

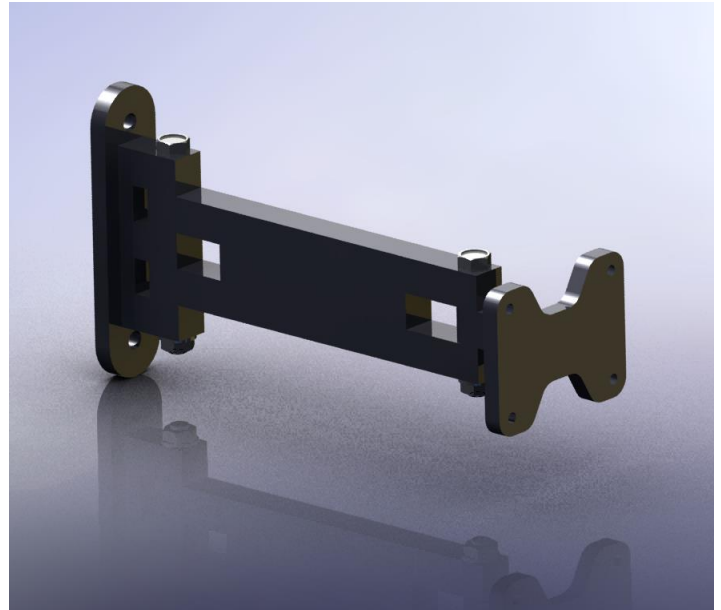
Choose **PhotoWorks, Render**

The dialog box shown across displays the progression of the render.



The rendered image is displayed on screen. Just like a photograph this is a capture of a moment in time. Should you rotate the model the image will be lost.

Choosing **PhotoWorks, Render** again will reproduce the image on screen.



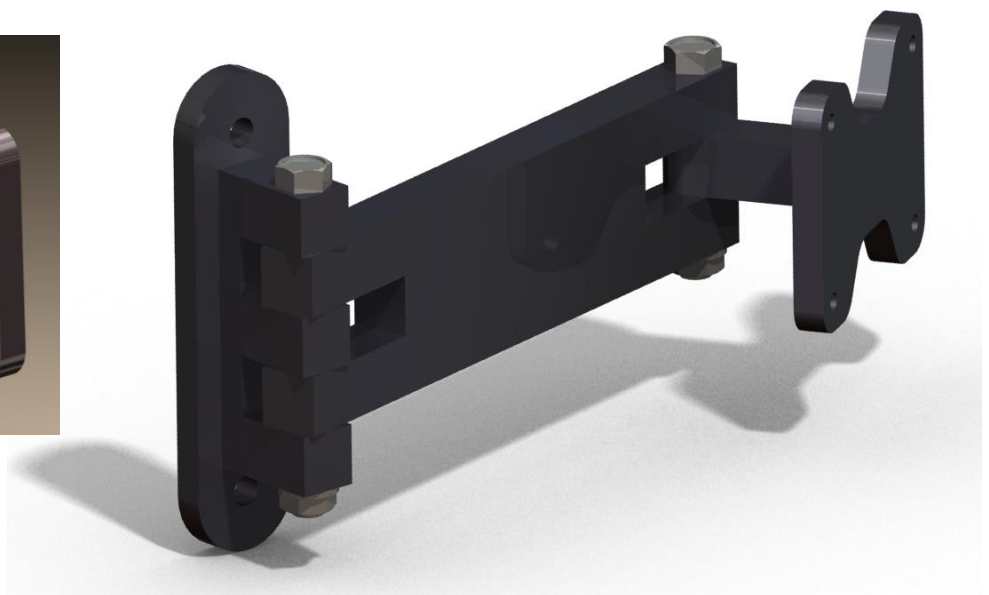
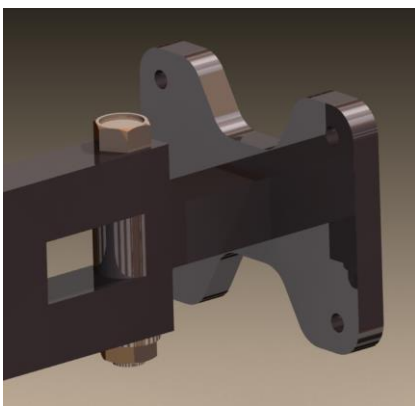
Save the file

To save the file choose **File, Save as**. Save the model in the TV Bracket folder in an appropriate file format, **JPEG** being the most common.

File name:	TV Bracket.JPG	<input type="button" value="Save"/>
Save as type:	JPEG (*.jpg)	<input type="button" value="Cancel"/>
Description:		

Render to File


Alternatively you may choose **PhotoWorks, Render to File**. This will create the jpeg image file without producing a rendered image on screen.



Note: A more comprehensive set of notes dealing with PhotoWorks, PhotoWorks Photorealistic Imaging, is available on the Round 10 DCG DVD

PhotoView 360

PhotoView 360 is a software package, separate to SolidWorks, which generates digital lifelike images (photorealistic images) from SolidWorks files. PhotoView 360 is an easy and quick way to produce high quality photorealistic images. There are a wider range of appearances in **PhotoView 360** than in **SolidWorks**.

PhotoView 360 is a separate program which loads with SolidWorks 2009. To launch the program; double click on the PhotoView 360  icon located on the desktop.

PhotoView 360

PhotoView 360 allows you to import a SolidWorks part or assembly and apply particular appearances and scenes to that file.

The Interface When using PhotoView 360 we work from left to right on the main menu.

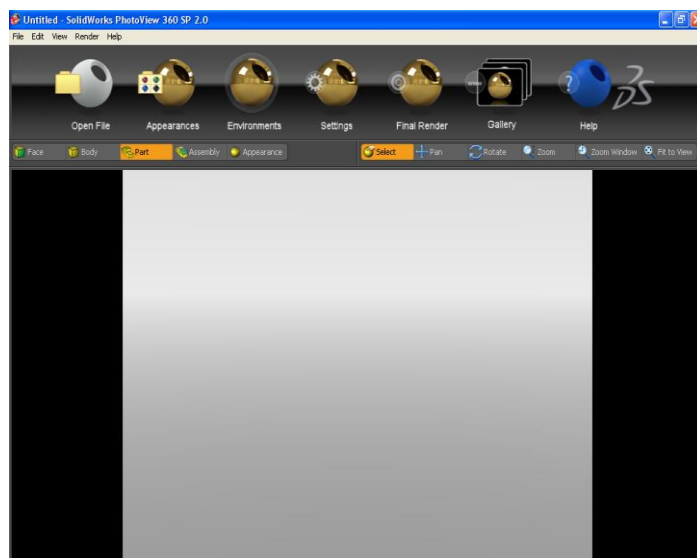
Open file - Import SolidWorks File

Apply Appearances

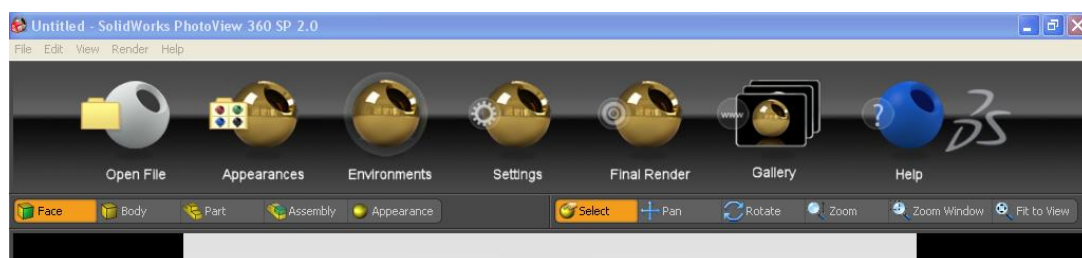
Apply Environment

Edit settings

Final render, save file



The main menu is shown below;



Toolbars

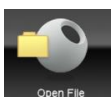


There are four options when applying an appearance – you may choose to apply an appearance to an **entire assembly**, **single part**, **body of a part** or an individual **face**. The appearance will be applied based on whichever option is pre-selected.

There is a second toolbar under the main toolbar. Its function is to move and rotate the model to enable the user to choose particular faces, features or parts as well as positioning the model to capture the photorealistic image.



The **roller ball** of the mouse can be used to manoeuvre a solid in **Photoview 360** window in the same way as **SolidWorks**. The **select** icon must be highlighted in order to apply appearances to any aspect of the SolidWorks model.

Use the **open file** button  and open the **TV Bracket** assembly

Rotate and Pan

Practice moving and rotating the model using the various tools in the navigation toolbar.



Highlight the command, move to the graphics area, and manipulate the positioning of the model.

Adding Appearance

Click on **appearances** button  in the main toolbar

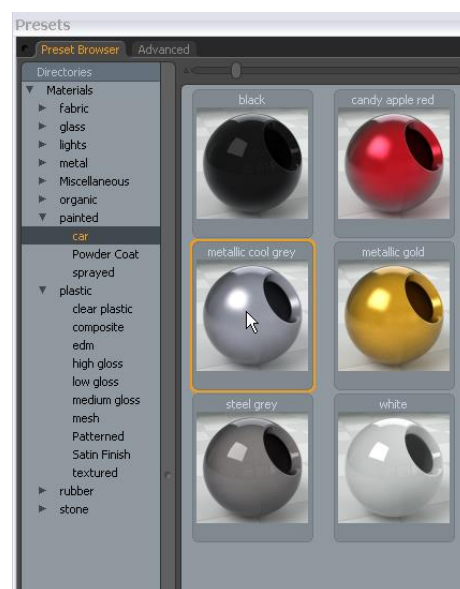
Choose **Part** on the selection toolbar



The **Preset Browser** window will appear.

Click on the triangle next to the category name to expand or collapse the selection tree.

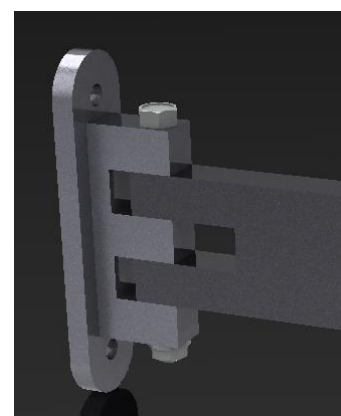
Navigate to **Painted, Car, metallic cool grey**



Drag and drop **metallic cool grey** onto the desired part in the graphics area

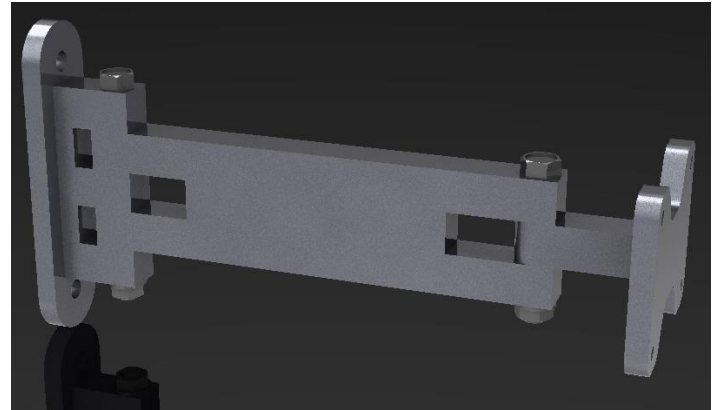
The appearance is only applied to this part because **Part** was preselected.

Repeat this procedure to add **metallic cool grey** to the **Link Arm** and the **TV Plate**.



With **Part** preselected apply a **Metal, Steel, Carbon Steel** appearance to the Bolts, Nuts and Washers.

You may need to utilise some of the view commands in order to apply the appearance to the smaller parts.



*Note: If you wished to apply a different material to the faces of one of the parts **Face** would be selected on the selection toolbar and the appearance would then be dropped onto it in the same way.*

Environments

Environments can be looked upon as **backdrops** and can be applied in a similar manner as the appearances are applied to faces or parts

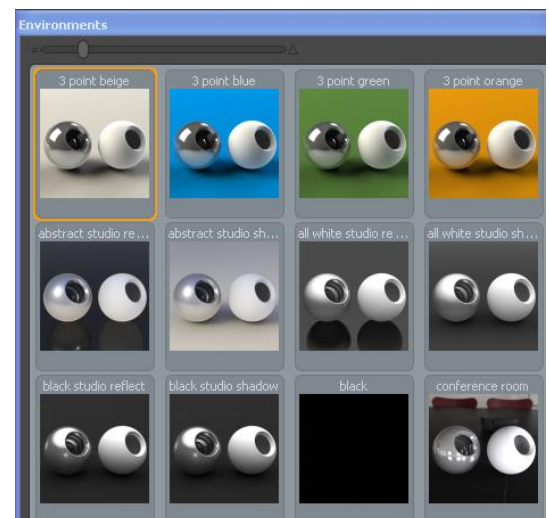
Select **environments** in the main menu.

The **Environments** selection box appears.

Drag and drop the chosen environment into the graphics area.

Dragging and dropping an alternative environment will override the previous selection.

Apply a **3point beige** environment.

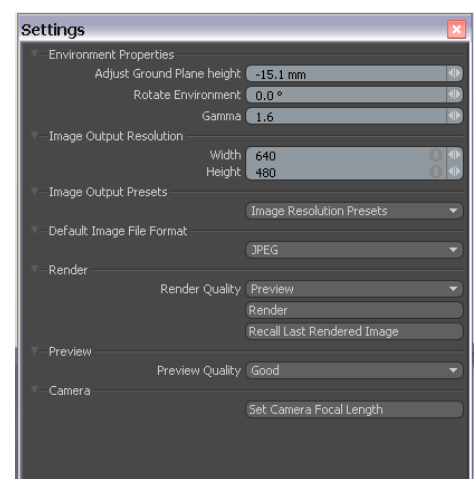
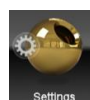


Positioning Capturing a good photorealistic image can be compared to setting up a camera for taking a photo. Use the navigation tools to orientate the model so that it fills the majority of the display window and shows good detail of the solid.

Settings for rendering

Settings allows us to set the format in which the image will be saved along with the quality of the image.

Open **settings** option from **main menu**



Environment properties

Adjust Ground Plane height: Sets the floor height of the environment in relation to the model. Decreasing this number moves the floor down, increasing the number moves the floor up, closer to the model.

Rotate Environment: Rotates the environment in relation to the model. This will affect lighting, shadow and shade etc.

Gamma: corrects the output to compensate for the output device i.e. a printer or monitor.

Image Output Resolution: changes the number of pixels in the final rendering. Increased number of pixels, increases files size and rendering time.

Image output Presets: height and width may be chosen or choose a preset value.

640 x 480 is suitable or an A3 size output.

Default Image File Format: JPEG, BMP, etc

Render: good, better, best, max

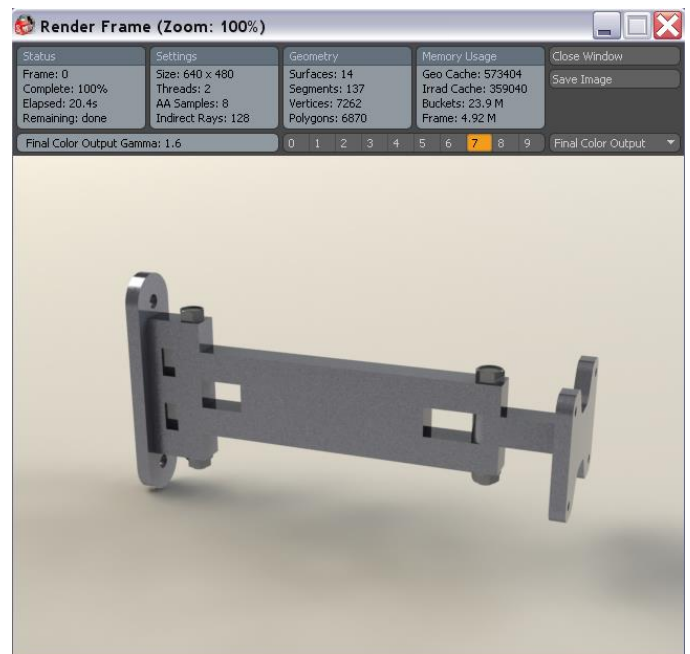
The higher the quality of the render, the longer the time it takes to complete the render

A better quality image is sufficient to complete this exercise but sample the other quality images also.

Final Render The render is now complete and a JPEG file can be created from this window.

Choose **Save Image**

Save the image in the **TV Bracket** folder



Note: A more comprehensive set of notes dealing with PhotoView 360, PhotoView 360 - Photorealistic Imaging, is available on the Round 10 DCG DVD