

Creating Drawings

Drawings

SolidWorks enables drawings to be created from parts or assemblies. These drawings are **fully associative** with the parts and assemblies they reference. If the model is changed the drawing will update and vice-versa. A drawing is created using a pre-prepared **template**.

Drawing Templates

These templates are used when creating presentation drawings. Parameters include sheet size, orientation etc.. The template may include a border, title block projection symbol, and text. When a presentation drawing is to be created using a part model, the template is the starting point.

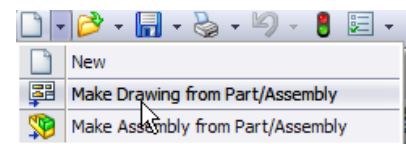
(See folder 7 – *SW templates for SW*, or Website www.t4.ie for **Technology Templates**)

Make Drawing from Part/Assembly

Make Drawing from Part/Assembly takes the current part and steps through the creation of a drawing file and initial drawing views using this part.

Where to find it?

Select **Make Drawing from Part/Assembly** on the Standard Toolbar *or* choose **File, Make Drawing from Part/Assembly**.

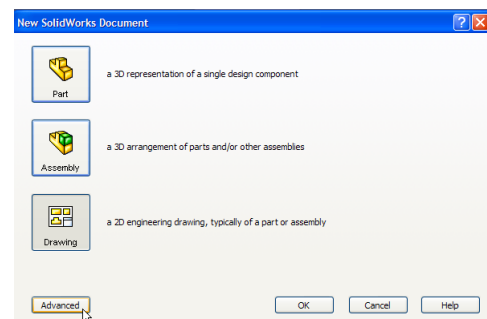


Getting Started

Open the model from which you wish to create the drawing.
For the purpose of this exercise we will use the **Tipper Truck** assembly.

Select **Make Drawing from Part/Assembly**.

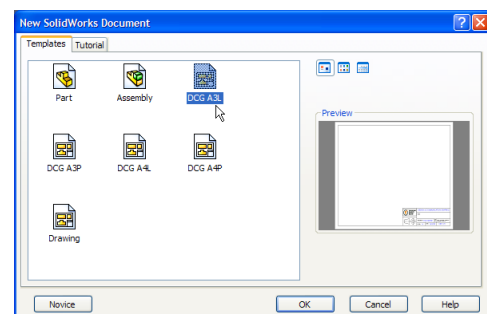
Select **Drawing** and then click **Advanced**



Choosing a Drawing Template

Choose the drawing template you wish to use from the list displayed – **DCGA3L**

Choose **OK**



The DCG Templates will only be displayed here if they have been saved following the instructions given in;

Drawing Templates – Storing & Editing folder 7 or www.t4.ie

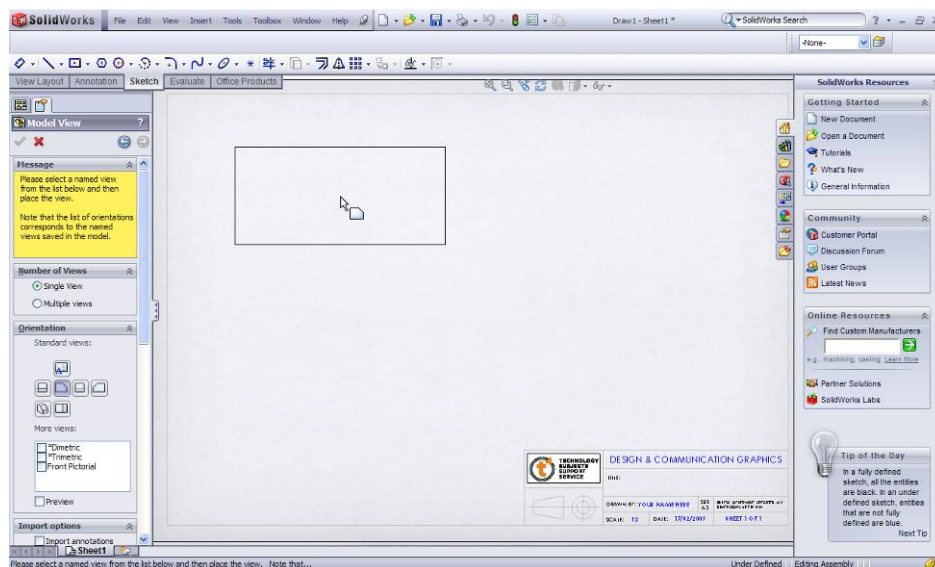
Drawing Template

This template creates an A3 landscape drawing. The sheet format includes a title block, projection symbol, T4 Logo and text.

An outline preview of the first view is displayed as shown below.



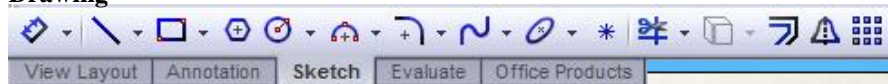
*Note: For detailed instructions on editing the sheet format refer to; **Drawing Templates – Storing & Editing** folder 7 or www.t4.ie*



Drawing Environment

Toolbars specific to the process of creating drawings and detailing are;

Drawing



Annotation



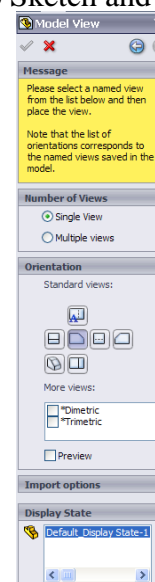
These will be accessed through the **Command Manager**, similar to Sketch and feature within the Part Environment.

Model View

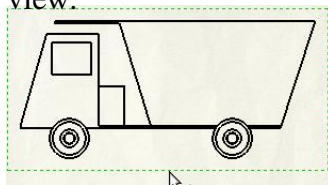
Model View is displayed within the **View Layout** option on the standard toolbar. Model view refers to any view that may be derived from the model.

The outline preview, shown above, is driven by **Single/Multiple View selection** under **Number of Views** along with the selected view under **Orientation, Standard views**.

In this case **Single View** is chosen along with **Front View**.

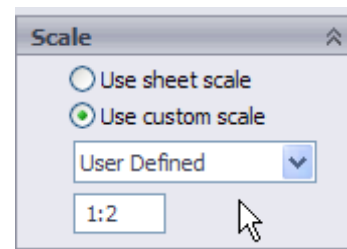


By checking the box next to **preview**, the outline view will be replaced with a detailed view.



Scale

By default, the **sheet scale** – **1:5** will be used. This scale may be changed by checking **Use custom scale**. Select the down arrow at **User Defined** and choose **1:2**.



Auto-start projected view

The preview will appear larger in the drawing area.

Ensure that this option is checked. This will allow the projection of further views once the first is positioned. Position the front view on the sheet by left clicking.

Drawing Views

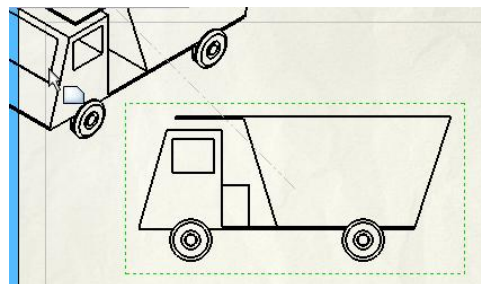
Projected Views

Drag the cursor to the right to project an end view, left click to position the view. Repeat the procedure to project a plan view.

Projecting an Isometric View

A number of isometric views may be projected from this view also.

To project an isometric view, drag the cursor to any of the 4 corners of the front view. A different isometric view of the model will be displayed in each position.



Positioning the view

When the cursor is dragged to position the isometric view on the sheet the projected view changes.

This may be overcome by holding down the **ctrl** key while dragging, thus allowing the current view to be positioned correctly.

Drag the view to a position overhead the title block, left click to position.

Choose **OK** in the Property Manager.

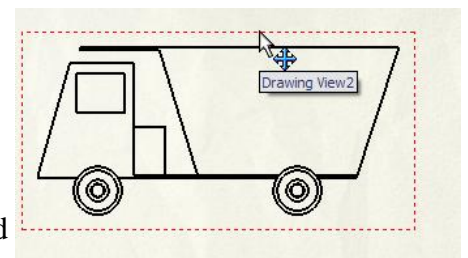


Repositioning Views

The orthographic views may be moved around the sheet but will maintain alignment.

To reposition a view, move the cursor over the view. A red dotted line will surround the view.

Position the cursor on the red dotted line, hold down the left mouse button and drag.



Display Style

Individual views may be displayed in a number of ways;



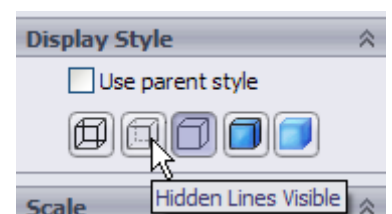
- **Wireframe** – Displays all edges.
- **Hidden lines visible** – Displays all edges, hidden lines are displayed as dashed
- **Hidden lines removed** – Displays edges that are visible at the chosen angle; obscured lines are removed.
- **Shaded with edges** – Displays items in shaded mode with hidden lines removed.
- **Shaded** – Displays items in shaded mode.

Edit display style

Highlight the **end view** by left clicking on the view.

All of the properties of that view appear in the PropertyManager.

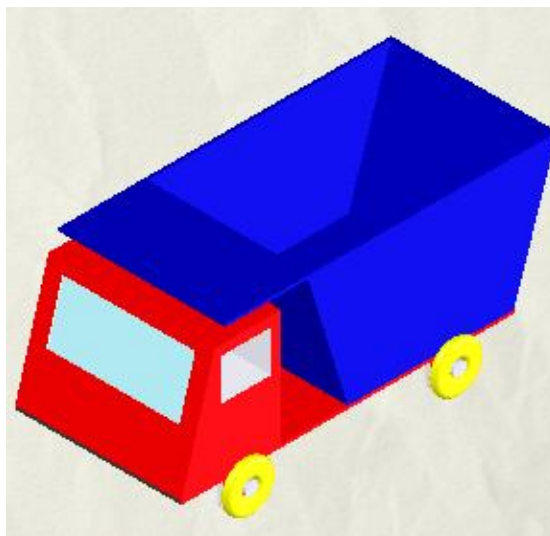
Choose **Hidden Lines Visible** from **Display Style**
All hidden features will be displayed as dashed lines.



Repeat the procedure for the **Front View**

Shaded with Edges

Edit the **Display Style** of the **Isometric View** to reflect **Shaded with Edges**



Edit View Scale

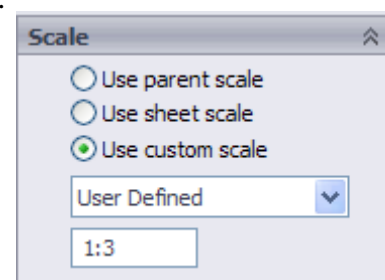
The scale in a drawing applies to the sheet and views. However, the scale may be changed for individual views.

To edit the scale of a view select the view. The properties of that view, including the scale, are displayed in the PropertyManager.

Edit Scale of Isometric View

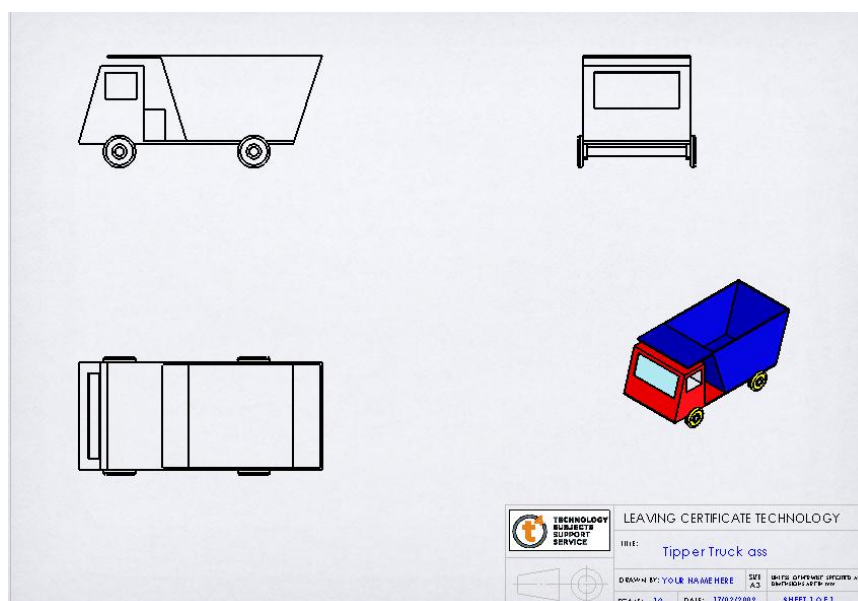
Choose the Isometric View.
Within the **Scale Property Manager** edit the settings as shown.

The isometric view will be reduced in size to reflect the updated scale value.



Drawing Views

Drawing layout showing hidden lines visible in the front and end view, along with the reduced scale in the isometric view, is shown below.



Adding Centre Marks

Centre Marks may be placed on circles or arcs in drawings. These may then be used as a reference for dimensioning.


Centre Marks must be added to the two circles in the end view.

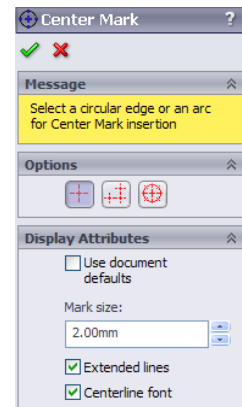
Choose **Center Mark**  from the **Annotation Toolbar**.

Deselect **Use document defaults**

Input a **Mark Size** of **2mm**

Select the two circles representing the wheels in the front view.

A centre mark will be added as shown 



Dimensioning

Dimensions in a SolidWorks drawing are associated with the model, and changes to the model are reflected in the drawing.

Model Dimensions

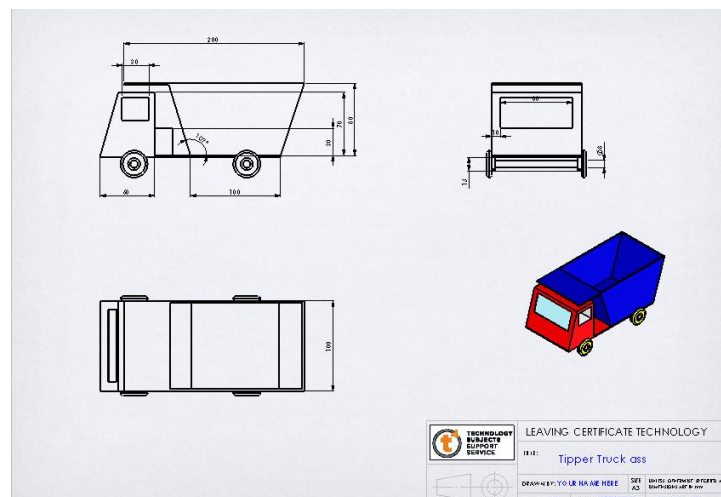
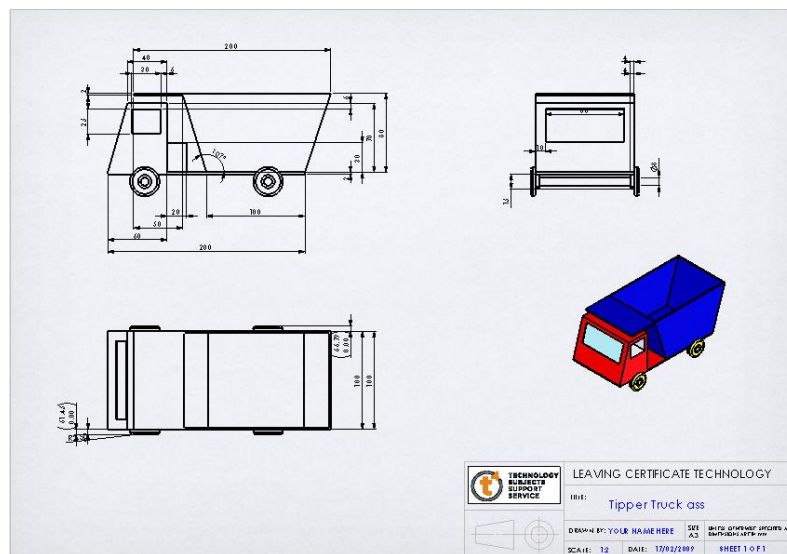
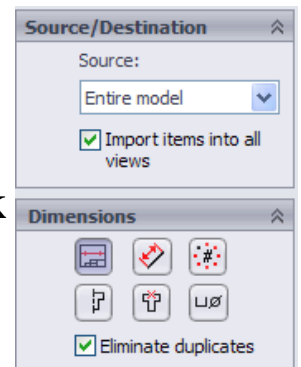
Typically, dimensions are created as each part feature is created. These dimensions are then inserted into the various drawing views. Changing a dimension in the model updates the drawing. Changing an inserted dimension in a drawing, changes the model. These dimensions are known as **driving dimensions**

Reference Dimensions

Dimensions may also be added in the drawing document. These are known as **Reference Dimensions** and are **driven dimensions**; you cannot edit the value of reference dimensions to change the model. However, the values of reference dimensions change when the model dimensions change.

Model Items

Dimensions, annotations, and reference geometry may be inserted from a part or assembly into a drawing. Items may be inserted into a selected feature, an assembly component, an assembly feature, a drawing view, or all views. When inserting items into all drawing views, dimensions and annotations appear in the most appropriate view. Features that appear in partial views, such as detail or section views, are dimensioned in those views first.



Move or copy dimensions to other views

The dimensions may not always appear in the required view or it may be necessary to dimension a feature in two views. It is possible to move and copy dimensions between views.



To move a dimension, hold down **shift** and drag the dimension to another view. To **copy** a dimension, hold down **ctrl**, drag it into the required view and drop it.

Manual Dimensioning

All of the necessary dimensions may not be present on the model. When this occurs the dimension must be inserted manually using **Smart Dimension**. This type of dimension is called a **driven dimension** because its value is driven by the model. *Driven dimensions are shown in a different colour.*

Smart Dimension

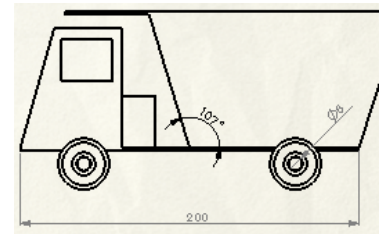
Select **Smart Dimension**  from the **Annotation** toolbar.

Using the same technique as in sketching, select the two edges defining the distance to be dimensioned.

Drag and place the dimension.



Ensure that the zoom features are used to make placement and alignment of dimensions easier.

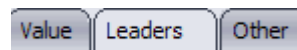


Use smart dimension to insert the driven dimensions shown.

Reversing arrow position

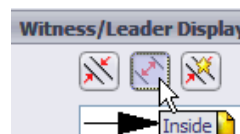
Highlight the dimension. The dimension properties are displayed in the Property Manager.

Scroll the options at the top of the Property Manager and select **Leaders....**

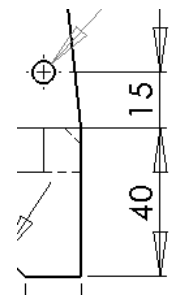


Within the **Leaders Properties** dialog box, select **Inside**.

Select **Apply, OK**



The arrows will now be displayed as shown .



Repeat this procedure for the 28mm dimension in the end view.

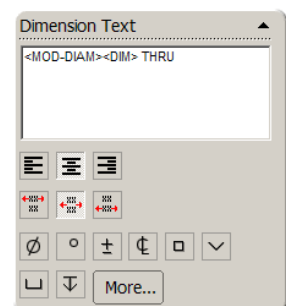
Dimension Text

Dimension text may be edited or added to within the PropertyManager

Select the dimension **Ø6 THRU**.

Within the Dimension PropertyManager the text reads;

<MOD-DIAM><DIM>THRU

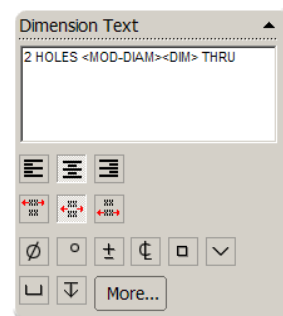
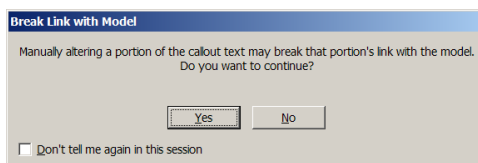


Edit Text

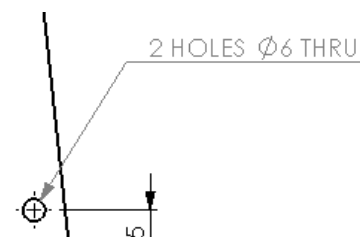
Insert the text “**2 HOLES_**” prior to this string, as shown opposite. Include a space after HOLES



The warning below may appear on commencement of typing. Changes made here will not break the link with the model



The dimension text will now appear as shown.

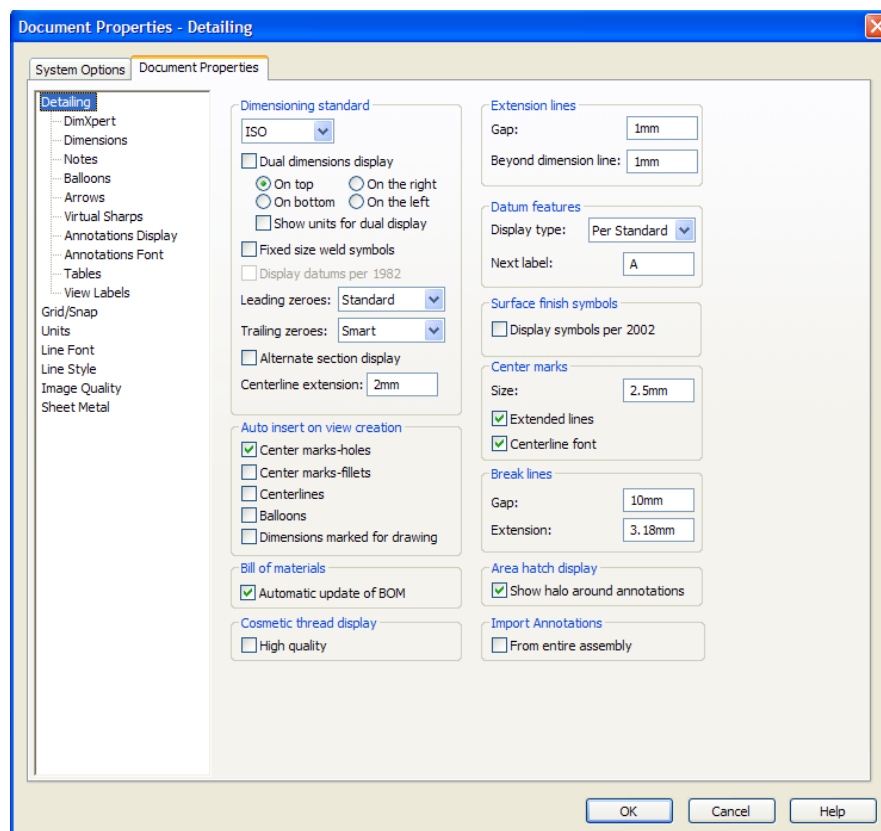


Edit text and arrows

Text and arrow settings, along with a whole range of other settings, may be altered within **Document Properties**.

Where to find it?

To access the **Document Properties** choose **Tools, Options...** and select the **Document Properties** tab.

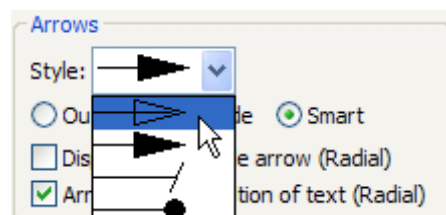


Editing arrow type

Double Click on **Dimensions**.

Select the arrow type as shown.
Choose **OK** to apply the changes.

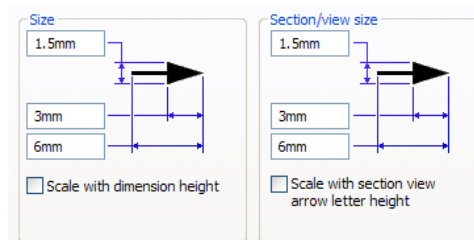
All dimensions will be updated



Editing arrow size

Within the **Document Properties** tab double click on **Arrows**

Edit the arrow size to reflect
the settings shown.



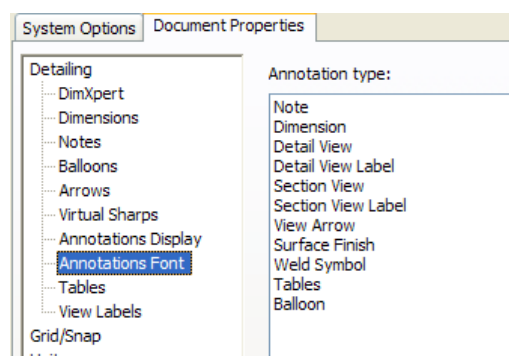
Section/View size refers to arrows used to represent a section line, used to create a section view.

Editing Text



The text within a drawing document may be edited by selecting **Annotations Font** within the **Document Properties** tab.

The font type is grouped according to application. This allows a different font type and size to be used for each of the 9 categories.



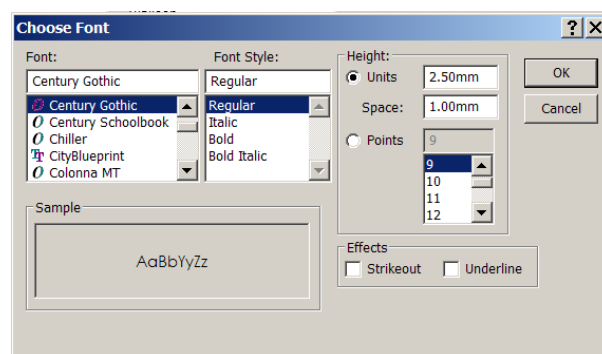
Editing Dimension Text

Double click on **Dimension**.

The **Dimension Font Property** dialog box will appear.

Edit the settings to reflect
those shown.
Choose **OK**

All dimensions will be updated



Further Text Editing

Apply the changes outlined below to the text categories listed. These will be used later in the exercise.

- **Section** Century Gothic, Regular, Height - 3mm
- **Note** Century Gothic, Regular, Height - 3mm



Take time to reposition/centre any text which may have been misaligned by the changes made.

Section View

A **Section View** is created in a drawing by cutting the parent view, the view it is to be projected from, with a line.

Where to find it

Select **Section View**  from the **View Layout** toolbar.
or choose **Insert, Drawing View, Section**.

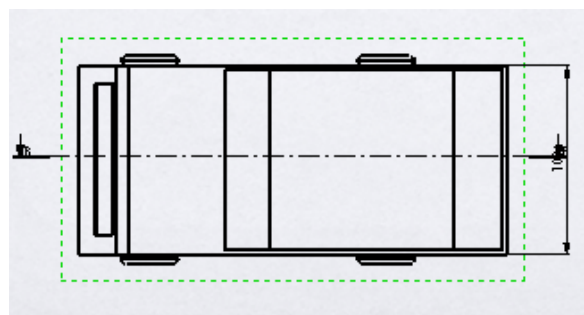
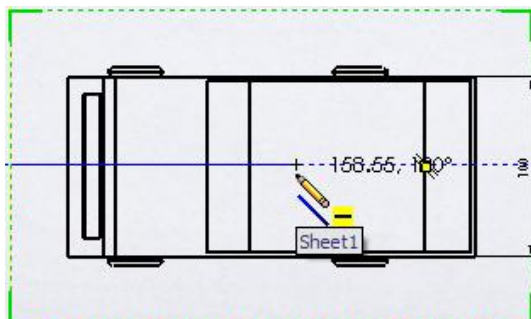
Create a Section View

Click **Section View**, the following prompt will appear.

Message
Please sketch a line on a drawing view to create a Section View.

Adding the Cutting line

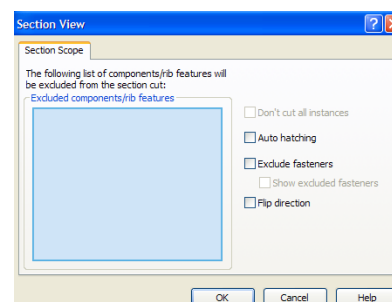
Using inferencing, align the cutting line with the midpoint of one of the horizontal lines as shown. Ensure that the sketched line extends beyond the view on both sides



Excluding Features

In some cases, following convention, you may wish to avoid sectioning some features. These may be chosen by selecting from the drawing area. These will then appear in the selection box.

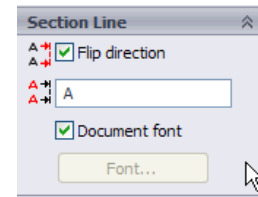
This does not apply on this occasion



Reverse Section

A section view projected to the right is required.

The arrows on the section line indicate that the opposite section has been generated by default.



To reverse the direction of projection select **Flip Direction** under **Section Line** within the PropertyManager.

Left Click in the drawing area to position the view.

Edit display style Hatch Edit

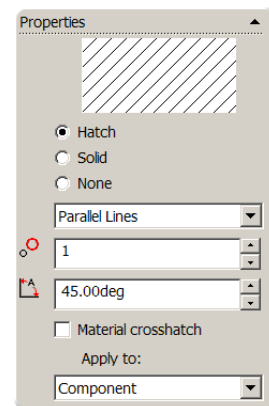
Change the display style to **Hidden Lines Removed**
Select the hatch pattern on the cut surface to open the **Hatch Property** dialog box.

Deselect **Material crosshatch**.

Input the selections shown opposite.

Select **OK**

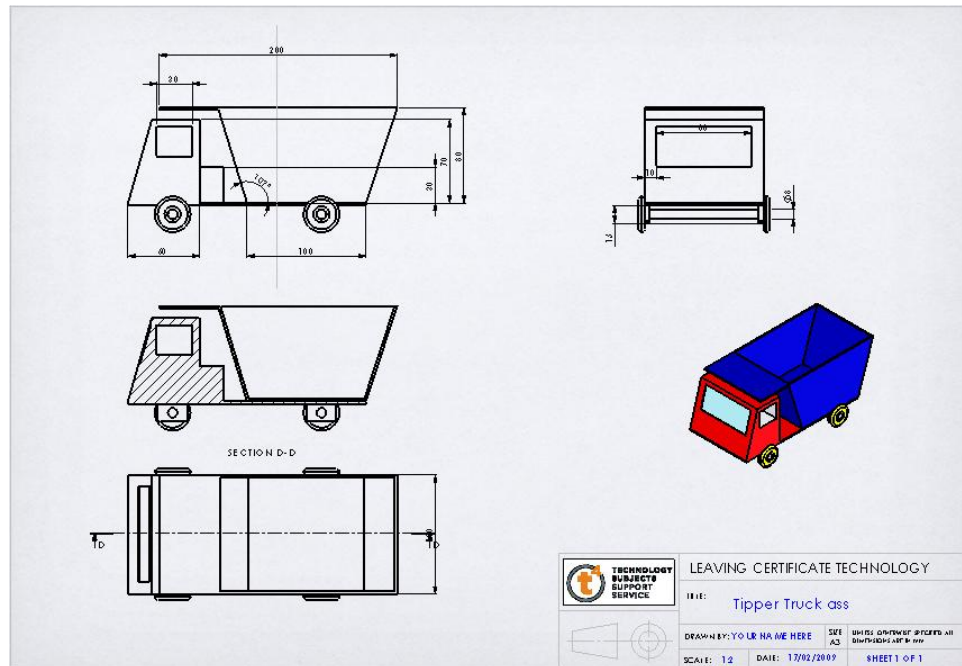
The changes are reflected only on the selected region.



Further hatch editing

By varying the **scale** and **angle**, recreate the hatch pattern shown on the remaining cut surfaces.

View Layout



Detail View

A **Detail View** may be created in a drawing to show a portion of a view, usually at an enlarged scale. This detail may be taken from an Orthographic View, an Isometric View, a Section View or another Detail View.

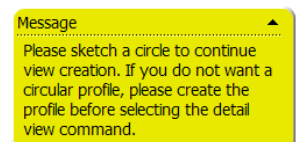
The detail view is determined by the contents of a closed sketch. The default used is a circle

Where to find it

Select **Detail View**  from the **View Layout** toolbar.
or choose **Insert, Drawing View, Detail**.

Create a Detail View

Click **Detail View**, the following prompt will appear.



Move the cursor to the approximate position of the centre of the viewing circle. Click and drag the radius.



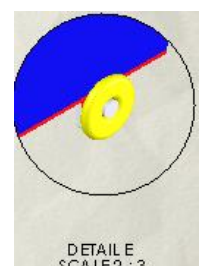
Note – The position and radius of the viewing circle may be altered afterwards.



Positioning the view

When the viewing circle has been defined the detail view will appear.

Drag the view to position, click to drop it.



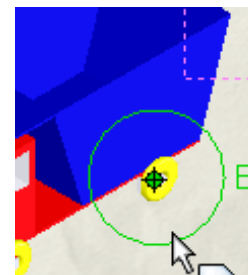
Editing the view

The view may be edited by changing the centre position and radius.

Editing the centre position

To edit the centre position, click on the circle.
A green dot with crosshairs appears at the centre.

Hold and drag this to reposition the centre.



Editing the radius

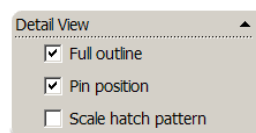
Hold and drag the circumference to increase the radius.



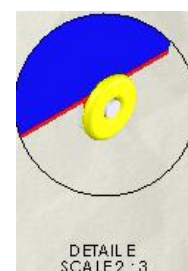
Using these two techniques, whilst displaying the detail view, will establish the detail view to show the required information.

Full Outline

Selecting **Full Outline** will display the viewing circle in the detail view.



Select **OK**.



Saving a model view

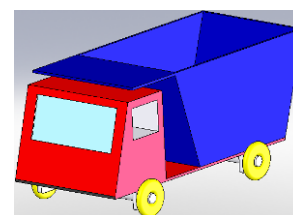
Within the **Part/Assembly** environment, any view of the model may be saved and named.
This non standard view may then be imported into the drawing document.

Saving a view of the model

The view we wish to create is a view showing the housing detail at the back of the spice rack.

Creating the view

Switch to the **Assembly** document of the **Tipper Truck**.
Within the **Assembly** document, rotate the view of the model to the desired position.

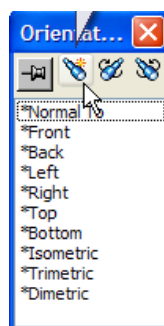


Naming & saving the view

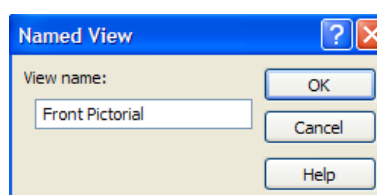
Press the space bar. The **Orientation** dialog box will appear containing all of the standard views.

Select **New View**. Input the name '**Front pictorial**'

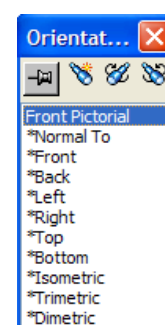
1. New View



2. Name the view



3. View added to list.



Displaying the Rear pictorial

To display this view, double click on **Rear pictorial** within the **Orientation** dialog box.

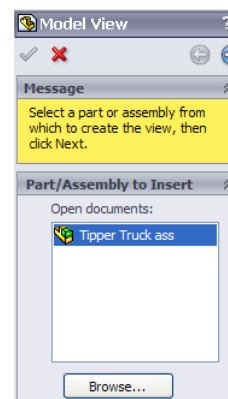
Save the changes to the **Tipper Truck Assembly** document and return to the **Tipper Truck Drawing** document.

Importing the saved view

Within the **Tipper Truck Drawing** document, select **Model View** from the **Drawing** toolbar.

Tipper Truck will be chosen by default under **Open documents**.

Choose **Next** 

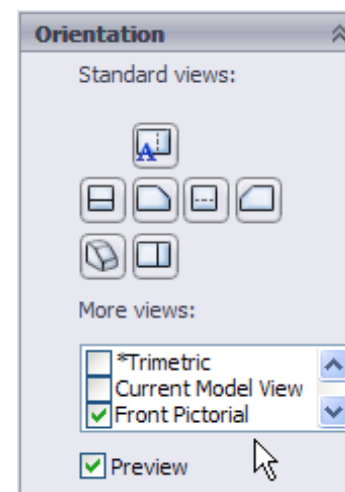


Orientation

Ensure **Single View** is pre-selected.

Rear pictorial will be checked in the window under **Orientation**.

Select **Preview**.



Positioning the view

Move the cursor into the drawing area.

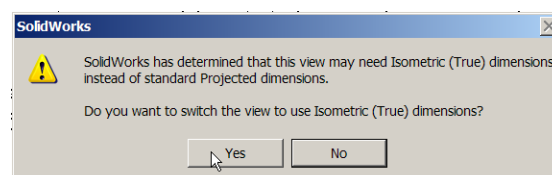
A preview of the 'Rear pictorial' will be displayed.

Position the view next to the title block, as shown.



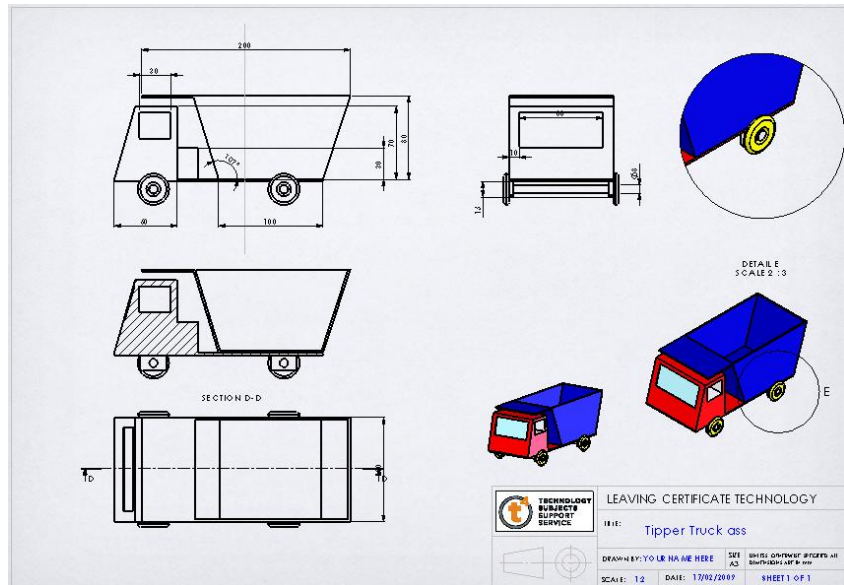
The dialog box shown opposite will appear.

Select **No**, as this view will not be Dimensioned on this occasion.



Display State

Within the PropertyManager change the **Display State** to **Shaded With Edges**.



Drawing Paper Colour

In order to print the drawing document with a white background it is necessary to change the setting - **Drawings, Paper Colour**.

Choose **Tools, Options** and select the **Systems Options** tab.

Select **Drawings, Paper Color** under **Colors**.

Select White as the paper Colour.

Choose **OK**

