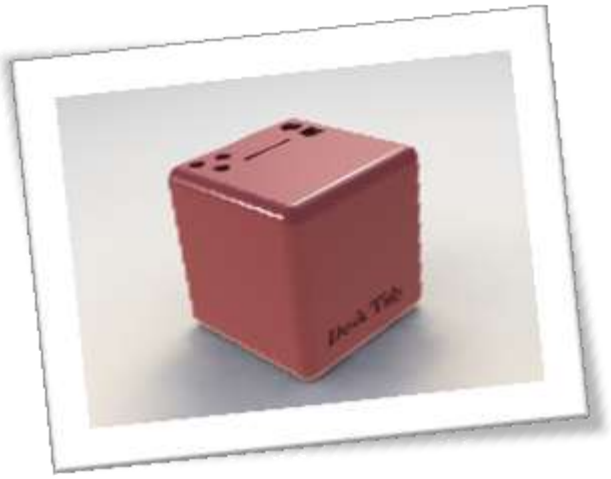
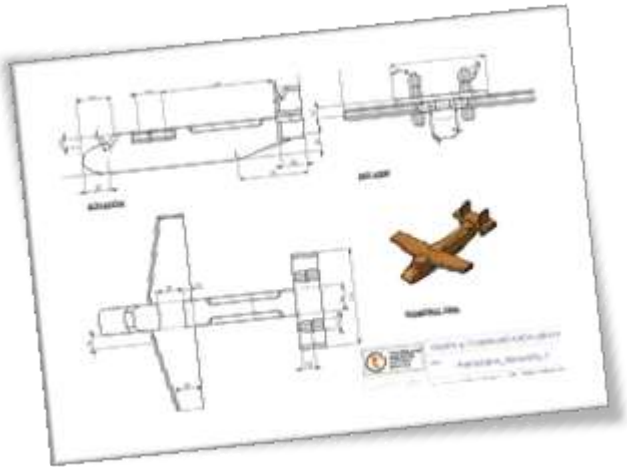
PDST An tSeirbhís um Fhorbairt Ghairmiúil do Mhúinteoirí | Professional Development Service for Teachers

Introduction to SolidWorks 2009





Introduction to SolidWorks 2009

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SolidWorks 2009 – The Interface

SolidWorks User Interface

The first thing that you notice about the SolidWorks® user interface is that it looks like Microsoft® Windows®. That is because it is Windows!

The SolidWorks 2009 (UI) is designed to make maximum use of the Graphics area for your model. Displayed toolbars and commands are kept to a minimum. Commands are accessed in SolidWorks through the drop-down menus, Context sensitive toolbars, Consolidated toolbars, or the CommandManager tabs.

Menu Bar Toolbar

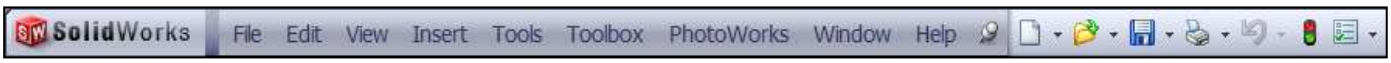
The Menu Bar toolbar contains a set of the most frequently used tool buttons from the Standard toolbar. The available tools are: **New** - Creates a new document, **Open** - Opens an existing document, **Save** - Saves an active document, **Print** - Prints an active document, **Undo** - Reverses the last action, **Rebuild** - Rebuilds the active part, assembly or drawing, **Options** - Changes system options, document properties, and Add-Ins for SolidWorks.

Menu Bar Menu

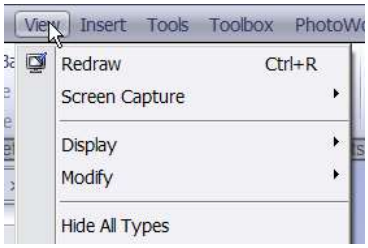
Click the SolidWorks name in the Menu Bar toolbar to display the default Menu Bar menu. SolidWorks provides a context-sensitive menu structure. The menu titles remain the same for all three types of documents: part, assembly, and drawing but the menu items change depending on which type of document is

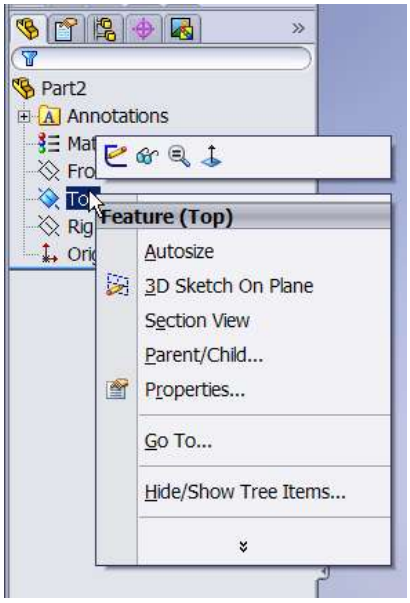
active. The default menu items for an active document are: **File**, **Edit**, **View**, **Insert**, **Tools**, **Window**, **Help**, and **Pin**.

Note: The Pin option displays both the Menu Bar toolbar and the Menu Bar menu.



Drop-down / Pop-up Context Toolbar

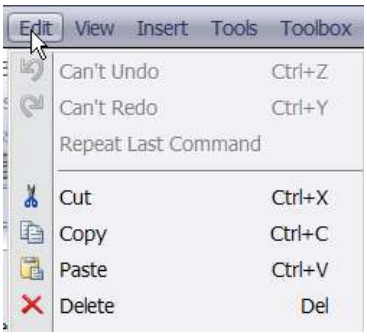




Communicate with SolidWorks either through the Drop-down menu or the Pop-up Context toolbar. The Drop-down Menu from the Menu Bar toolbar or the Menu Bar menu provides access to various commands. When you select, (Click or Right-click) items in the Graphics area or FeatureManager, Context toolbars appear and provide access to frequently performed actions for that context.

Note: Context toolbars are available for most commonly used selections.

Keyboard Shortcuts



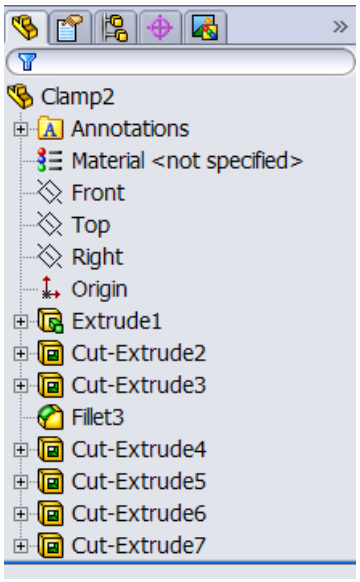
Some menu items indicate a keyboard shortcut. SolidWorks conforms to standard Windows conventions for shortcuts such as **Ctrl+O** for **File, Open**; **Ctrl+S** for **File, Save**; **Ctrl+X** for **Cut**; **Ctrl+C** for **Copy**; and so on. In addition, you can customize SolidWorks by creating your own shortcuts.

CommandManager

The CommandManager is a context-sensitive toolbar that automatically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on your active document type. When you click a tab below the CommandManager, it updates to display that toolbar. Example, of you click the Sketch tab, the Sketch toolbar is displayed. The default tabs for a part document are: **Features**, **Sketch**, **Evaluate**, **DimXpert**, and **Office Products**.

Below are illustrated CommandManagers for a default Part document for both **Sketch** and **Feature** tabs.

FeatureManager



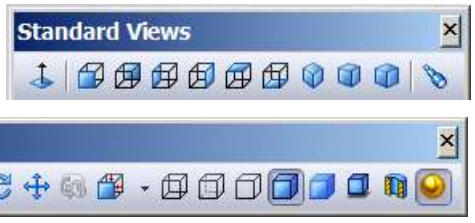
The FeatureManager® design tree is a unique part of the SolidWorks software that employs patented SolidWorks technology to visually display all of the features in a part, assembly, or drawing.

As features are created, they are added to the FeatureManager. As a result, the FeatureManager represents the chronological sequence of modeling operations. The FeatureManager also allows access to editing the features and objects that it contains.

The Part FeatureManager consists of four default tabs:

FeatureManager, **PropertyManager**, **ConfigurationManager**, and **DimXpertManager**.

Heads-up View Toolbar



SolidWorks provides the user with numerous view options from the Standard Views, View, and Heads-up View toolbar.

The Heads-up View toolbar is a transparent toolbar that is displayed in the Graphics area when a document is active

The following views are available:

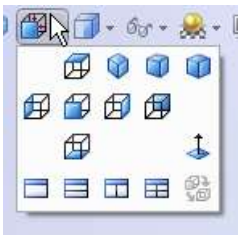
Zoom to Fit Zooms the model to fit the Graphics area.

Zoom to Area Zooms to the areas you select with a bounding box.

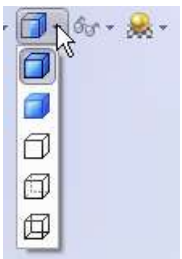
Previous View Displays the previous view.

Section View Displays the cutaway of a part or assembly using one or more cross section planes.

View Orientation Select a view orientation or the number of viewports from the drop-down menu.



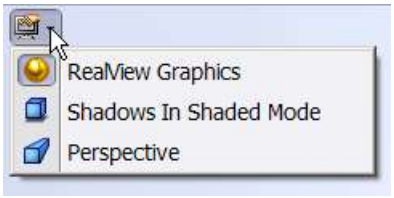
Display Style. Select the style for the active view from the drop-down menu.



Hide/Show Items Select items to hide or show in the Graphics area.

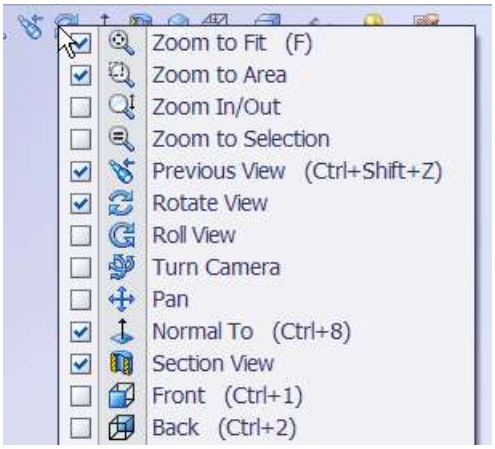
Apply Scene Applies a scene to an active part or assembly document.

View Setting Select the following setting from the drop-down menu: *RealView Graphics*, *Shadows in Shaded Mode*, and *Perspective*.



Rotate view Rotates a drawing view.

The heads up view toolbar may be customised to suit the user.
To add view options; right click on the toolbar. The options which are currently displayed on the toolbar will have a tick in the box next to its description.
Tick the box next to the options you wish to add to the toolbar.



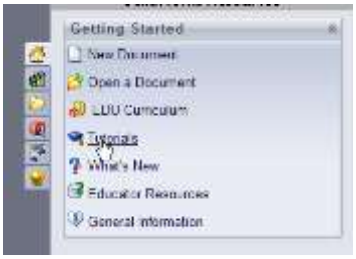
Useful options to add may be **Normal to** and **Isometric**

Task Pane

The Task Pane is displayed when a SolidWorks session starts. The Task Pane contains the following default tabs: **Solid Professor, SolidWorks Resources, Design Library, File Explorer, View Palette, Appearances/Scenes**.

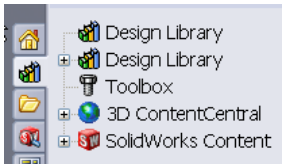
SolidProfessor
SolidWorks Resources
Design Library
File Explorer
View Palette
Appearances/Scenes

The basic SolidWorks Resources tab displays the following default selections: *Getting Started*, *Community*, *Online Resources*, and *Tip of the Day*.
SolidWorks tutorials may be accessed within the "Getting Started tab"



Design Library

The Design Library contains reusable parts, assemblies, and other elements, including library features. The Design Library tab contains four default selections. Each default selection contains additional subcategories. The default selections are: *Design Library*, *Toolbox*, *3D ContentCentral*, and *SolidWorks Content*.



Note: Click **Tools**, **Add-Ins.., SolidWorks Toolbox** and **SolidWorks Toolbox Browser** to activate the SolidWorks Toolbox.

File Explorer

File Explorer duplicates Windows Explorer from your local computer and displays the recent documents opened in SolidWorks.

Search

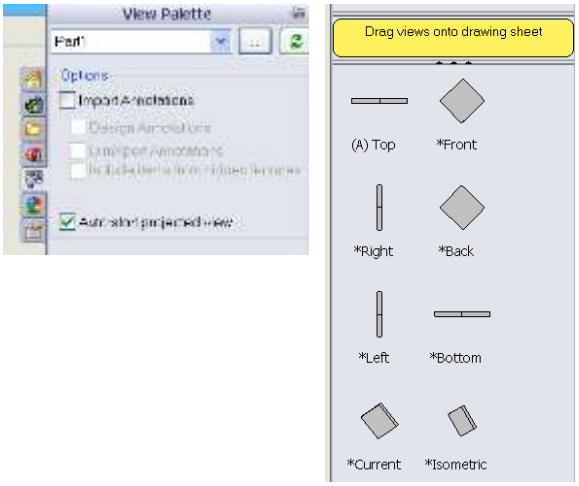
SolidWorks Search is installed with Microsoft Windows Search and indexes the resources once before searching, begins, either after installation, or when you initiate the first searches.

The SolidWorks Search box is displayed in the upper right corner of the SolidWorks Graphics area. Enter the text or key works to search. Click the drop-down arrow to view the last 10 recent searches.

The Search tool in the Task Pane searches the following default locations: *All locations, Local Files, Design Library, SolidWorks Toolbox, and 3D ContentCentral.*

View Palette

The View Palette tab located in the Task Pane provides the ability to insert drawing views of an active document, or click the Browse button to locate the desired document. Click and drag the view from the View Palette into an active drawing sheet to create a drawing view.



Appearances/Scenes

Appearances/Scenes tab provides a simplified way to display models in a photo-realistic setting using a library of appearances and scenes.

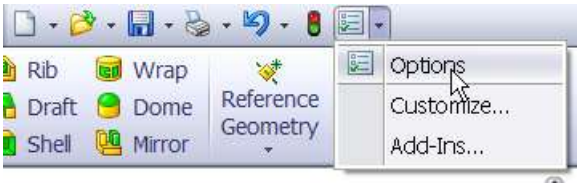
On RealView compatible systems, you can select Appearances and Scenes to display your model in the Graphics area. Drag and drop a selected appearance onto the model or FeatureManager. View the results in the Graphics area.

Note: Appearances/PhotoWorks graphics is only available with supported graphics cards. For the latest information on graphics cards that support Appearances/PhotoWorks display, visit: www.solidworks.com/pages/services/videocardtesting.html.

Changing the background colour to white

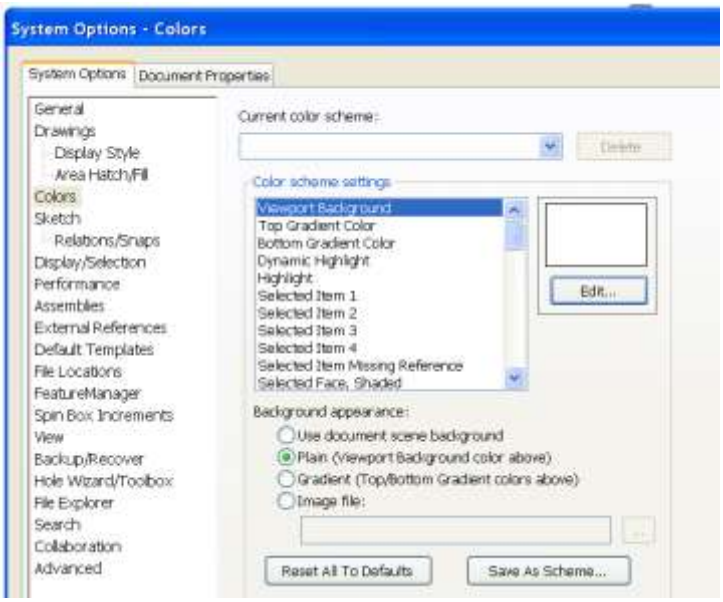
It may be beneficial, for taking screen grabs and saving in a jpeg format for importing into presentations, that the background colour be changed.

In order to do so follow the procedure outlined below.



Choose **Options** from the standard toolbar

Within the **System Options** tab choose **Colors, Viewport Background.**



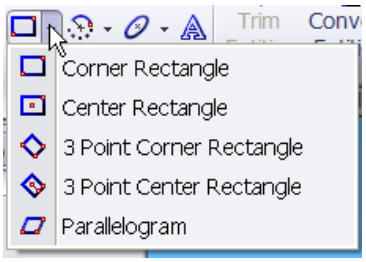
Choose **Edit** and select a colour eg. **White**

Under **Background appearance.** Select **Plain**

Choose **OK**

Consolidated Flyout Tool Buttons

Similar commands are grouped into consolidated flyout buttons on the toolbar and in the CommandManager. For example: Variations of the Rectangle tool are grouped together into a single button with a flyout control.



System Feedback

System feedback is provided by a symbol attached to the cursor arrow indicating what you are selecting or what the system is expecting you to select. As the cursor floats across the model, feedback comes in the form of symbols riding next to the cursor arrow.

Mouse Buttons

The left, middle, and right mouse buttons have specific uses in SolidWorks.

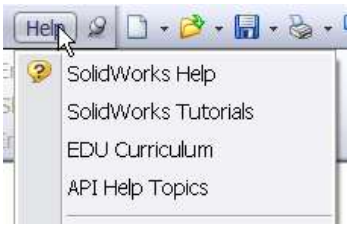
Left — Selects objects such as geometry, menu buttons, and objects in the FeatureManager design tree.

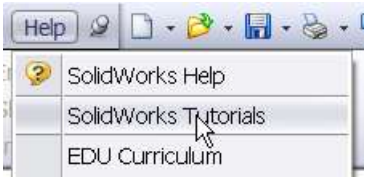
Middle — Holding the middle mouse button as you drag the mouse rotates the view. Holding the **Shift** key down while you use the middle mouse button zooms the view. Using the **Ctrl** key scrolls or pans the view.

Right — Activates context-sensitive pop-up menus. The contents of the menu differ depending on what object the cursor is over. These right-mouse button menus give you shortcuts to frequently used commands.

Getting Help

SolidWorks has a comprehensive help function that is designed to assist you. The SolidWorks help screen is displayed in its own window with three menu tabs. Click **Help**, **SolidWorks Help** from the Menu Bar menu to view the comprehensive SolidWorks help screen.



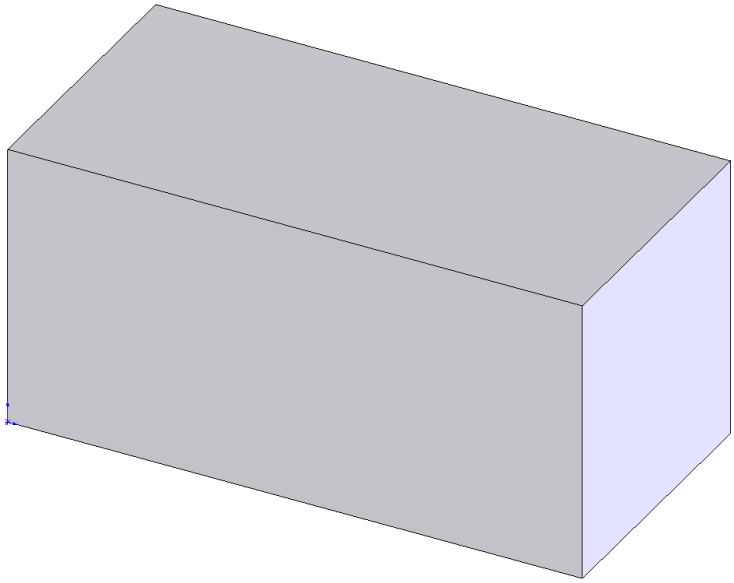


SolidWorks Tutorials

The SolidWorks Tutorials provide step-by-step lessons with sample files covering SolidWorks terminology, concepts, functions, features, and many Add-Ins. Work or view the 30 minute lesson tutorials to learn and strengthen your skills.

Click **Help**, **SolidWorks Tutorials** from the Menu Bar menu, or click the SolidWorks **Resources tab** from the Task Pane and click **Tutorials**.

Block Exercise



Prerequisite Knowledge Navigating the interface, file management, opening and saving of file.

Focus of the Lesson On completion of this exercise you will have used:

- Sketch line
- Automatic relations
- Smart dimension
- Extruded Boss/Base
- Renaming a Feature
- Editing a Sketch

Commands Used ***Line***, ***Extruded Boss/Base*** and ***Edit Sketch***.

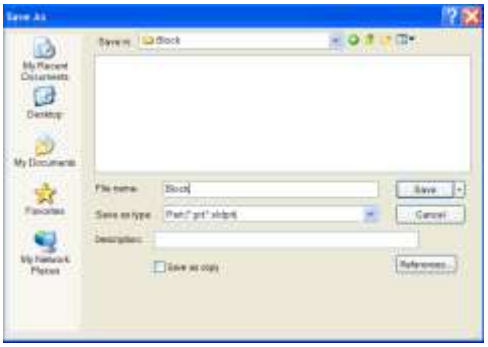
Getting Started

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 *Block.* 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.

Continue to save periodically throughout the exercise



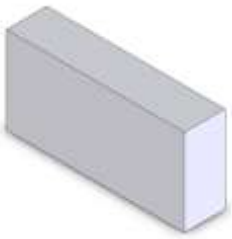
Command Manager

Where to start?

To activate the command manager

The only feature of the part to be created is the Block. This will be an extruded feature based on a sketch.

Sketch to generate the feature



Getting started

Choosing a plane

Select **Top Plane** from the manager tree. When you select the top plane four options will appear over the Top Plane. Select the sketch icon from this list of options.

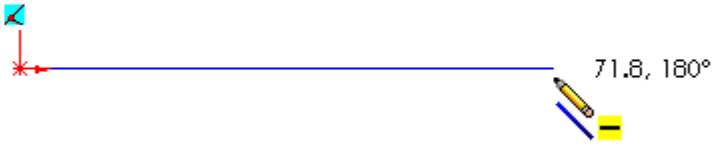
To view the sketch toolbar press **S** on you Keyboard.

Creating a sketch

Click the **Line** command and sketch a horizontal line **coincident** with the origin.

Coincident Relation

The origin is an endpoint to the line and therefore there is **coincident relation** between the endpoint and origin.

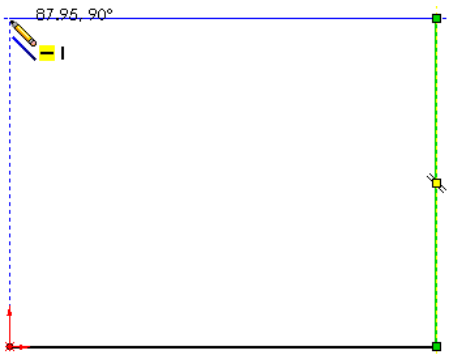


The "=" symbol appears at the cursor, indicating that a **Horizontal** relation is automatically added to the line. Click again to select the endpoint of the line

Draw a line ensuring that a **vertical relation** is automatically added.

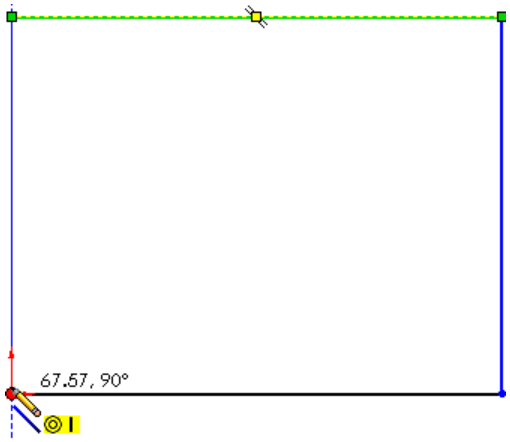
Inferencing

Drag a Horizontal line and using **Inferencing** choose its endpoint directly over the origin.



Inferencing lines are dotted lines that appear as you sketch. When your pointer approaches highlighted cues such as midpoints, the inferencing lines guide you relative to existing sketch entities.

Finish the shape by drawing a vertical line and select the origin as the end point as shown.



Turning off tools

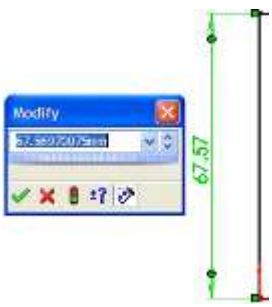
Turn off the active tool using:
Press the **Esc** key on the keyboard

To view the sketch toolbar again press **S** on the keyboard.

Dimensioning the sketch

Select **Smart Dimension** from the sketch toolbar to dimension the sketch.

Using select the line shown.
Click a second time to place the text of the dimension to the left of the line.
The dimension appears with a **Modify** tool displaying the current length of the line.



Set the value

Change the value to **100** and click **Save** ✔ option. The dimension forces the length of the line to be 100 mm.

Pressing Enter has the same effect as clicking the Save ✔ button

Add additional linear dimensions to the sketch as shown

Zoom to Fit

When dimensioning the sketch may become too large in the work area. To overcome this press **F** on the keyboard and the sketch will automatically fit in the work area

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

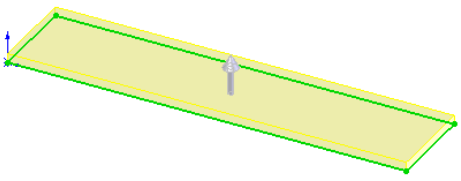
Exiting the Sketch

To exit the sketch, select the sketch tool on the confirmation corner, sketch will be saved. Selecting X will discard changes made.

Creating the feature

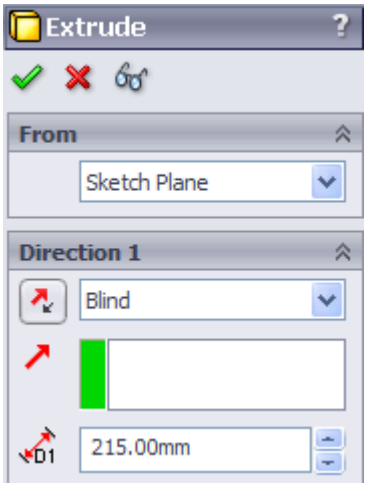
Having exited the sketch, press **S** on the keyboard to view the features toolbar.
select Extrude Boss/ base from the toolbar.

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



Extrude Feature Settings

End Condition = **Blind**



Blind End Condition

Extends the feature from the sketch plane for a ***specified*** distance.

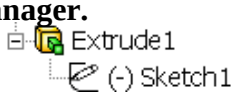
Depth = **215mm**

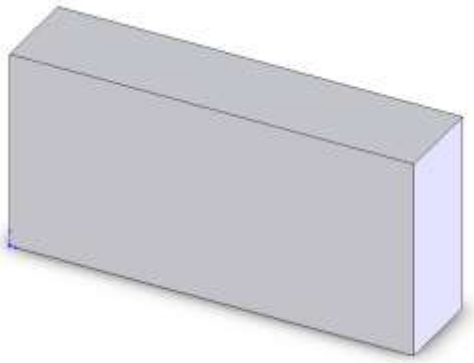
Click **OK** button to create the feature.

Alternatively select the from the confirmation corner

Completed feature

This is the first completed feature of the part. The sketch has been absorbed into the *EXTRUDE 1* feature in the **Feature Manager.**





Renaming a feature

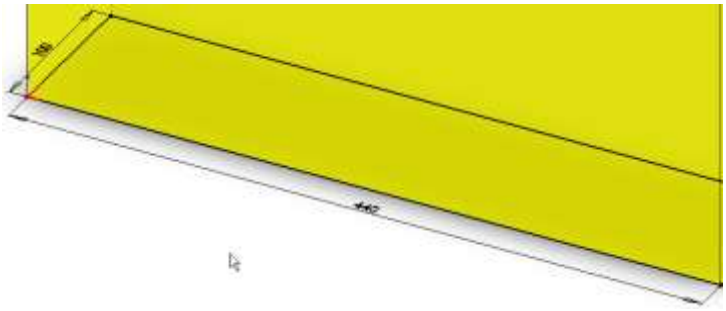
Select the feature in the **Feature Manager Tree.** Press F2.

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

Editing Sketch

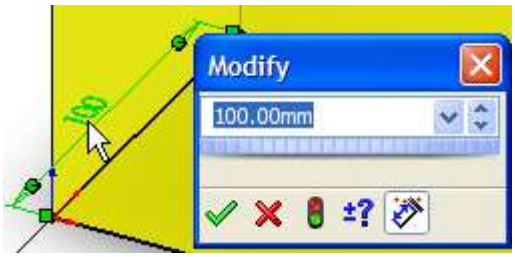
On the **FeatureManager design tree** select the plus beside the Block feature so that Sketch1 appears.

Right click on ***Sketch1*** and select the ***Edit Sketch*** icon. The original sketch used to create the feature appears, with the feature shown as an orange preview.



Editing Dimensions

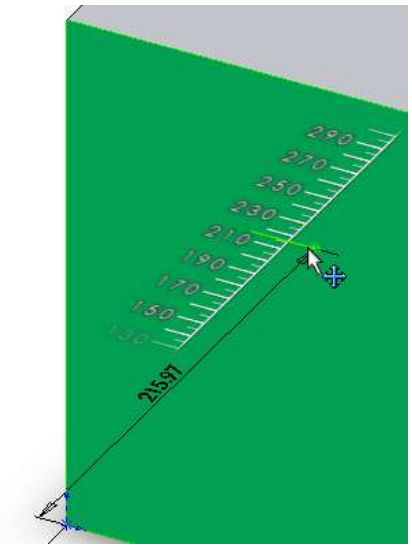
Double click on the 100mm dimension. The **Modify Tool** appears displaying the current length of the line.



Change the value to **215** and click **Save** option. The dimension forces the length of the line to be 215 mm.

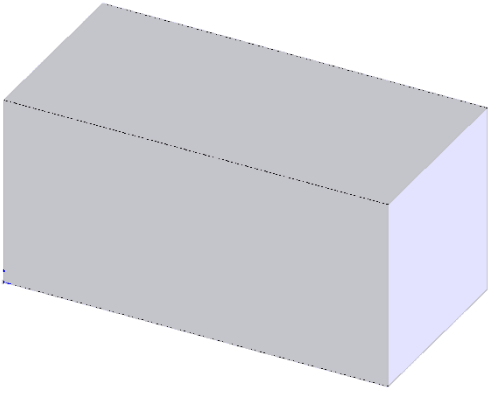
Note you may also change any of the dimensions of the Block by selecting any face of the block and dragging the blue dot at the end of the chosen dimension.

This activates the ruler, you can set the desired Dimension by scrolling along this ruler. If you Scroll the cursor on the ruler it will move in 5mm Increments, but if you scroll with the cursor to the Side of the ruler it will move in .1mm.



Exiting the Sketch

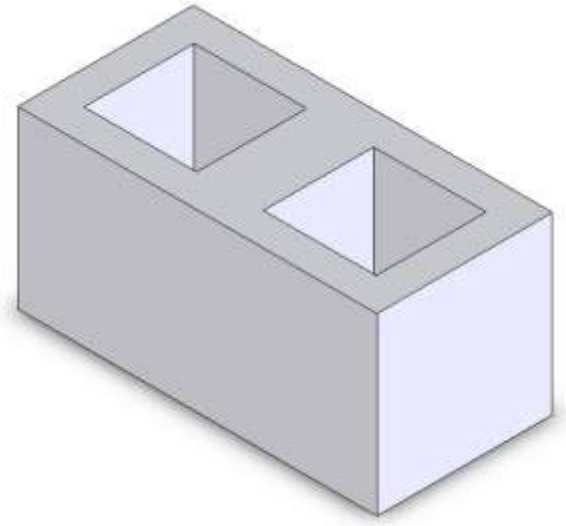
To exit the sketch, select the sketch tool on the confirmation corner.



Save and close

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

Cavity Block Exercise



Prerequisite Knowledge Creating a Part, Saving a Part, Creating a Sketch, Drawing Lines, Automatic Relations – Vertical, Horizontal, Coincident. Smart Dimensioning, Extruded Boss/Base (Blind End Condition).

Focus of the Lesson On completion of this exercise you will have used:

- Sketch rectangle
- Centre line
- Equal relations
- Sketch mirror
- Extruded Boss/Base
- Extruded Cut

Commands Used ***Rectangle, Centreline, Add Relation, Sketch Mirror, Extruded Boss/Base and Extruded Cut***.

Getting Started

New Part Create a new part. Save the part as ***Cavity Block*** in your chosen location.

Getting started

Where to start?

The first feature is the main part of the block and we will then remove the two cavities.

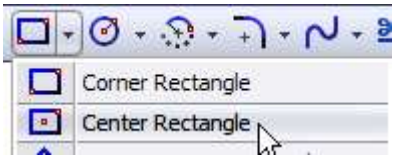
Choosing a plane

Select **Top Plane** from the manager tree. When you select the top plane four options will appear over the Top Plane. Select the sketch icon from this list of options.

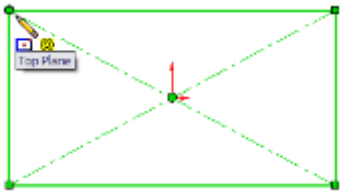
To view the sketch toolbar press **S** on you keyboard.

Creating a sketch

Click the drop down arrow beside the **Rectangle** command and, now select the **Centre Rectangle** command.



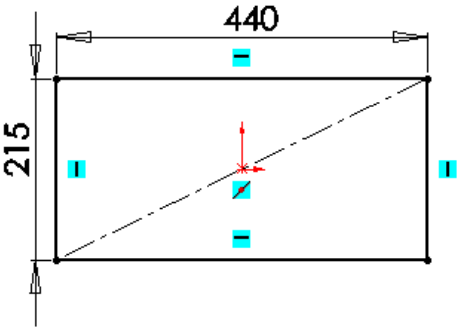
Create a sketch of the rectangle using the origin as centre.



Dimension Sketch

Select **Smart Dimension** from the sketch toolbar to dimension the sketch

Smart dimension the sketch to The values shown.



Zoom to Fit

Press **F** to resize the view to fill the screen.

Exit the sketch.

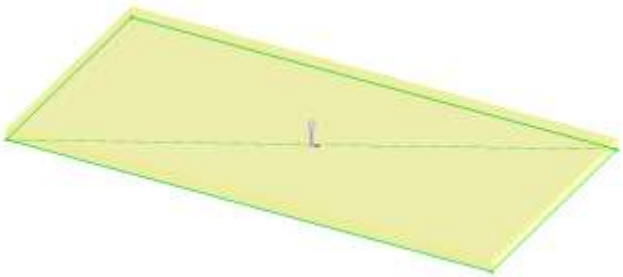
Creating the feature

Press **S** on the keyboard and then choose **Extruded Boss/Base** from the **Features Toolbar.**

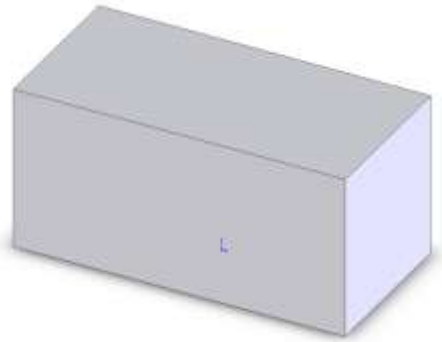
Extrude Feature Settings

End Condition = **Blind**

Depth = **215mm**



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



Renaming a feature

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

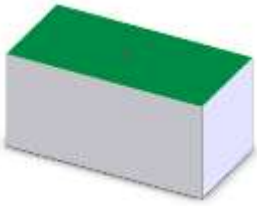
Type the new name *Block* to replace it.

Creating the Cavities

Sketching on a Face

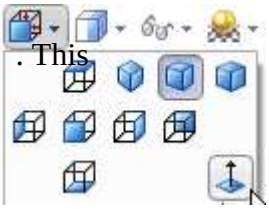
Any face of a model may be used to generate a plane to contain a sketch.

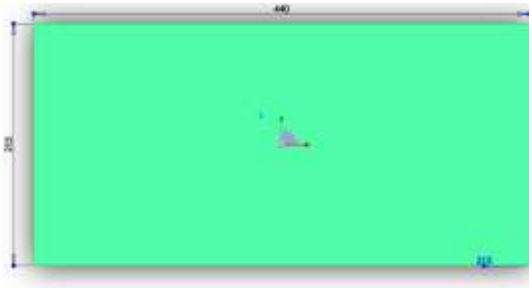
Select the top face of the block. Then select the sketch icon from the List that appears.



Change View

From the options on the top of the graphics area select the View orientation icon and select **Normal To**. This allows us to look in on the surface at a 90 degree angle.



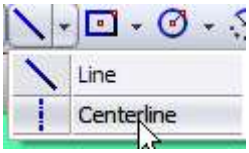


Create a new Sketch

To create a new sketch select the sketch icon from the List on the chosen surface.

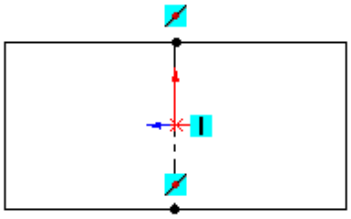
Then Press **S** to view the Sketch toolbar.

From the sketch toolbar Select centreline from the line command.

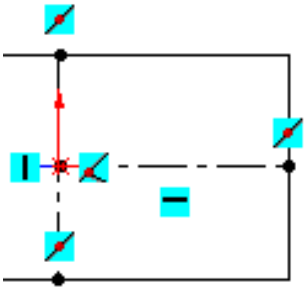


Draw a line from the midpoint of the horizontal lines.

The midpoint is automatically located when the cursor hovers over the required line.

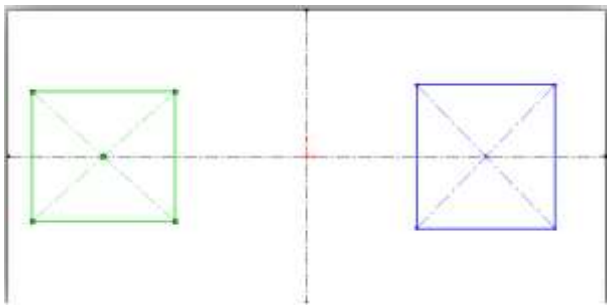


Draw a second centreline from the origin to the midpoint of one of the vertical lines.



Rectangle Command

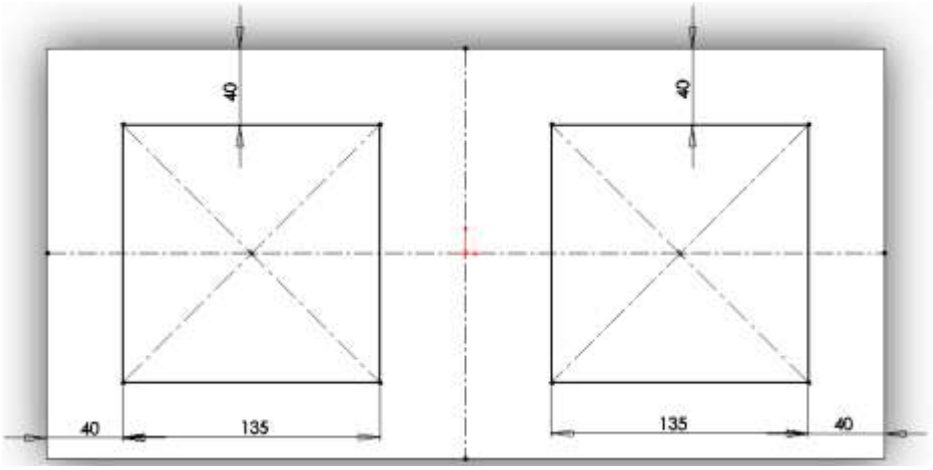
Press **S** and select the **Centre Rectangle** Sketch command and draw 2 rectangles on the centre line.



Press **ESC** on your keyboard

Dimension sketch

Dimension the sketch using the dimensions shown.



Exit Sketch.

Isometric View

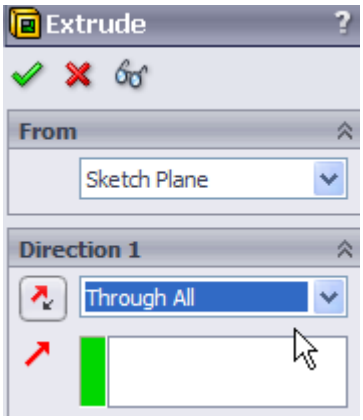
Select **Isometric** from the **View Orientation** toolbar.

Extruded Cut

Cuts a solid model by extruding a sketched profile in one or two directions.

Press **S** on the keyboard and select **Extrude Cut** from the Features toolbar.

Extrude Cut Feature Settings

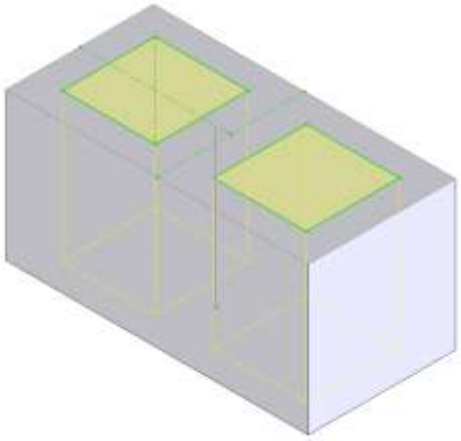


End Condition = **Through All**

Through All
End Condition

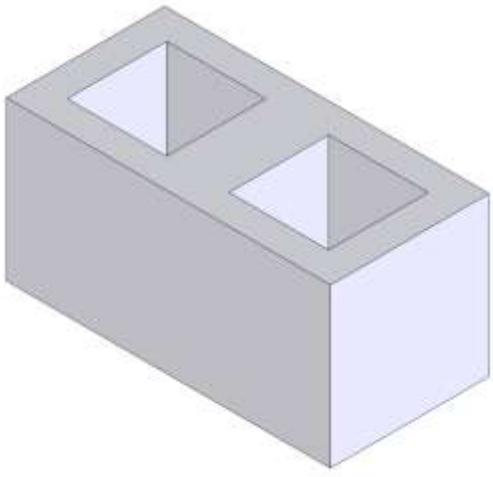
This type of end condition always cuts through the entire model no matter how far.

Depth is ignored because the **Extruded Cut** is **Through All**



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

Completed feature

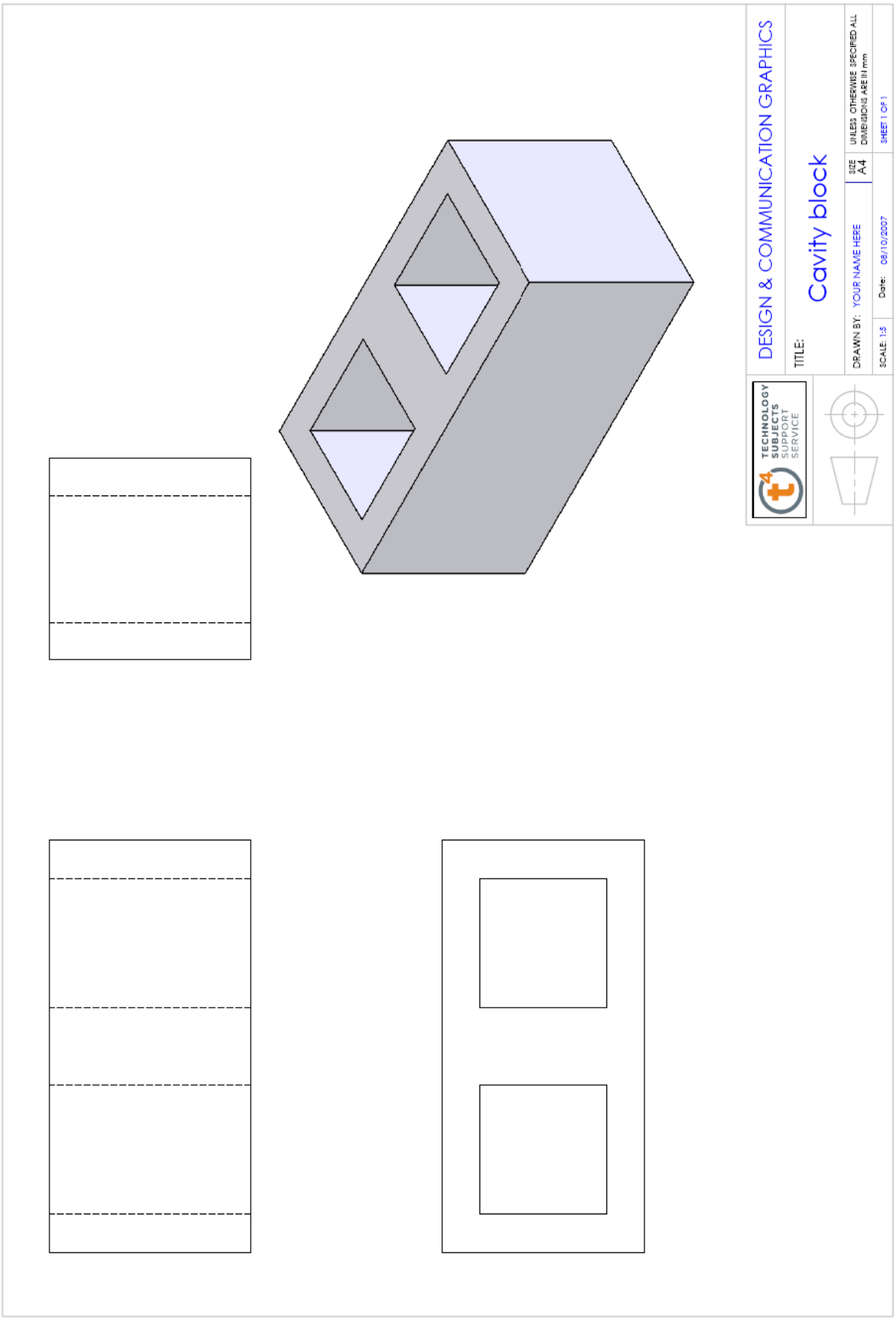


Renaming a feature

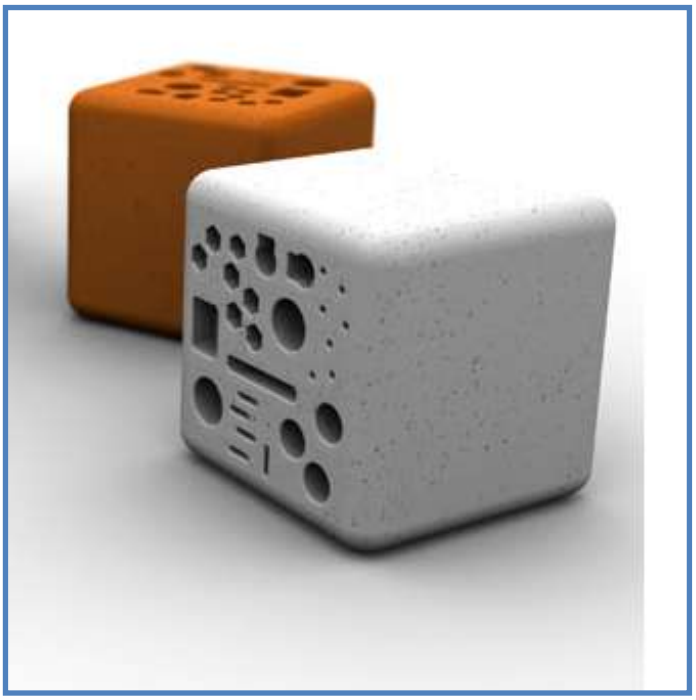
Rename it ***Cavity***

Save and close

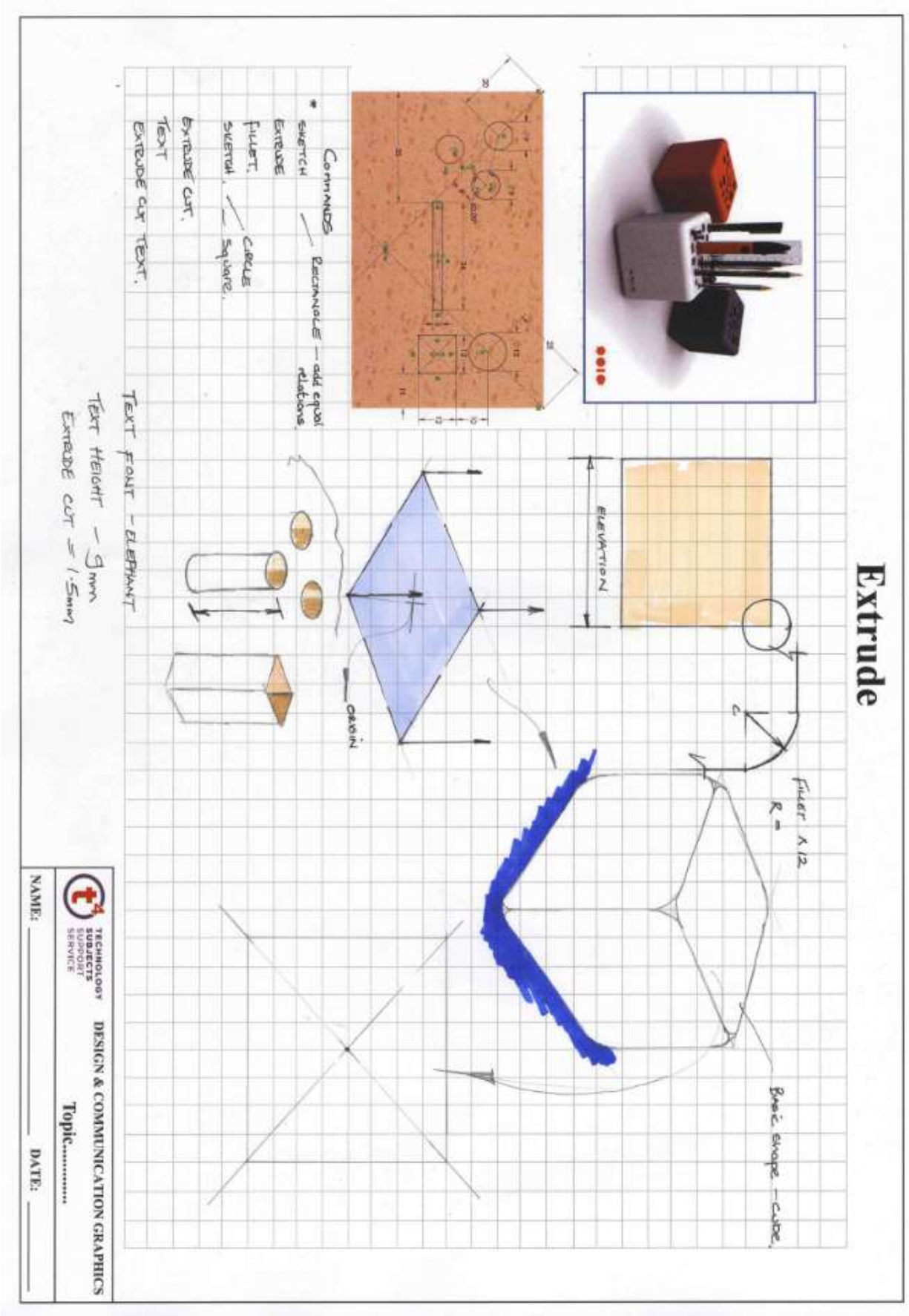
Click **Save** to save your work and click **File, Close** to close the part.


DESIGN & COMMUNICATION GRAPHICS
TITLE:
Cavity block
DRAWN BY: YOUR NAME HERE
SIZE A4
UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm
SCALE: 1:5
Date: 08/10/2007
SHEET 1 OF 1
TECHNOLOGY SUBJECTS SUPPORT SERVICE

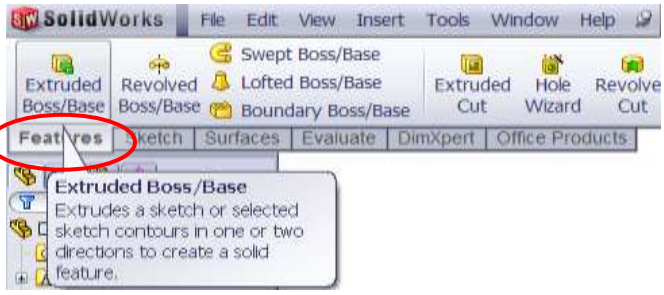
Desk Tidy Exercise



Object Analysis sheet



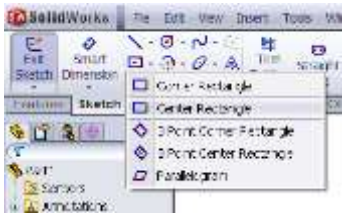
Commands used:

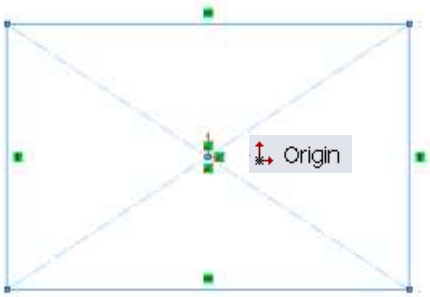


Getting started:

Create Sketch Select the **Top plane** to sketch on.

Create the sketch shown on the **Top plane**.



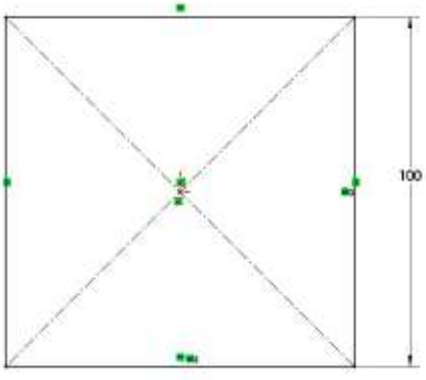


Add Relation

Add an **Equal Relation** to two edges of the rectangle.

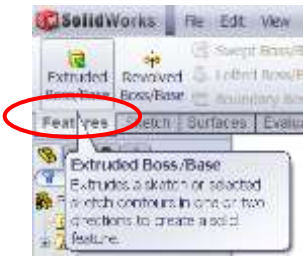
Smart Dimension

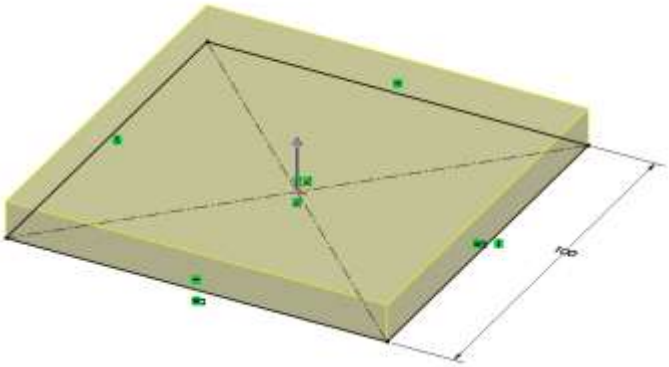
Apply dimensions to the sketch using only the dimension displayed.
Fully define the sketch.



Create Extrude

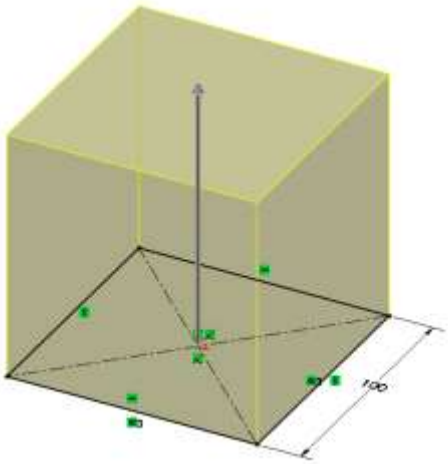
Select Extruded Boss/Base from the Features menu. .





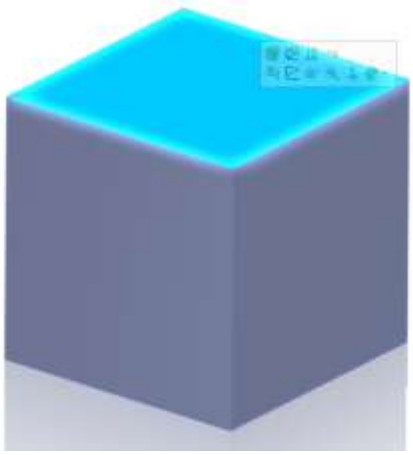
Add Depth and End condition

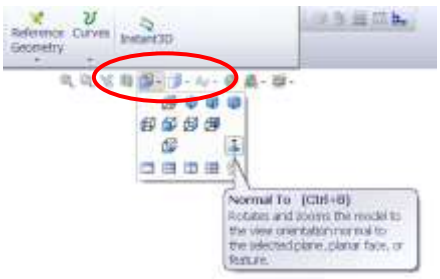
Ensure that the **End condition** is **Blind** and the **Depth** is **100mm**.



Create Sketch

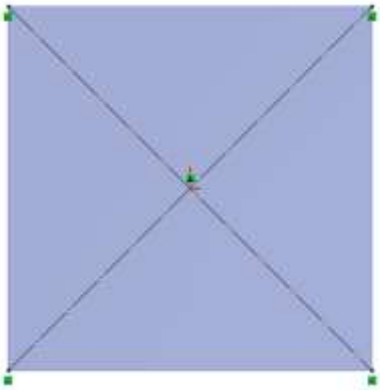
Select the **Top surface of the cube** to sketch on.
Select Normal To





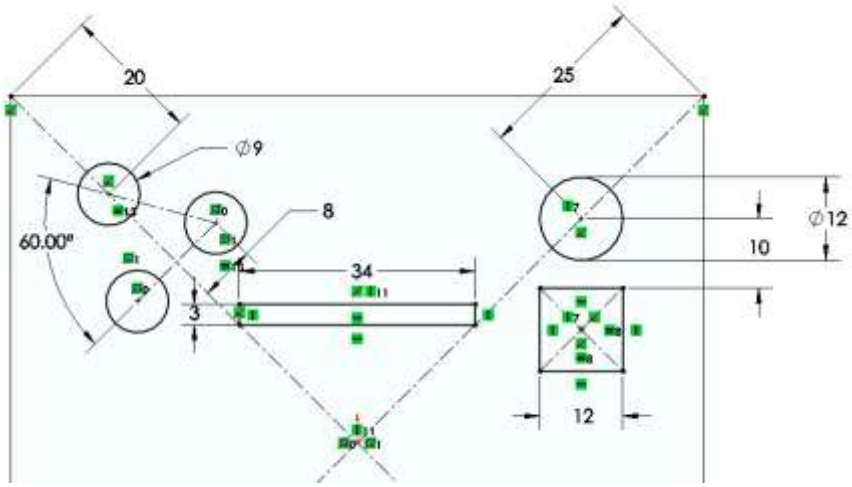
Create Sketch

Sketch the diagonals of the face and as shown, change the line properties to **for construction** as shown.



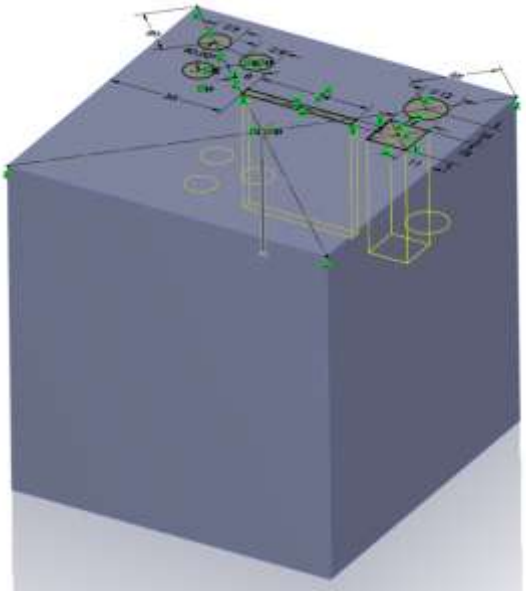
Create Sketch

Sketch the shapes as shown, fully define the sketches



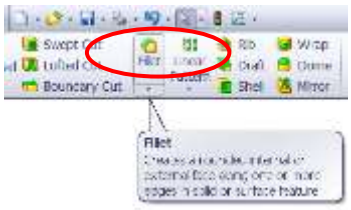
Create Extrude

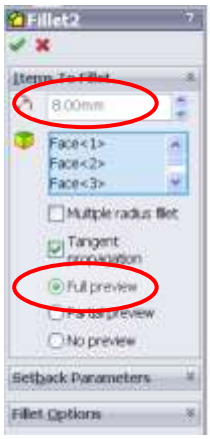
Select Extruded Cut from the Features menu. Ensure that the **End condition** is **Blind** and the **Depth** is **35mm**.

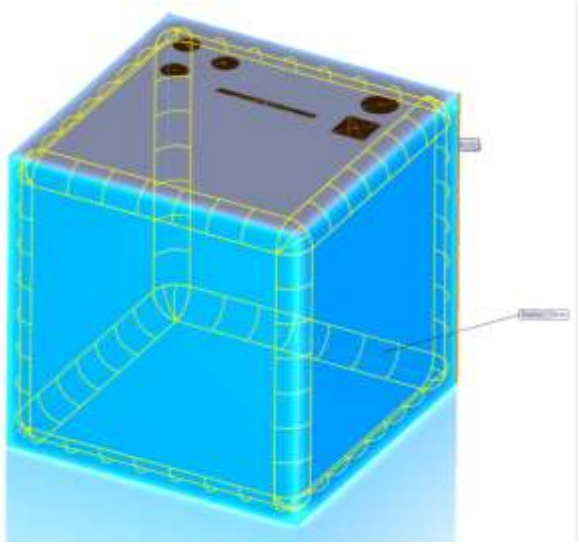


Create Fillet

In the **Features** menu select **Fillet** as shown. Select Fillet Expert.

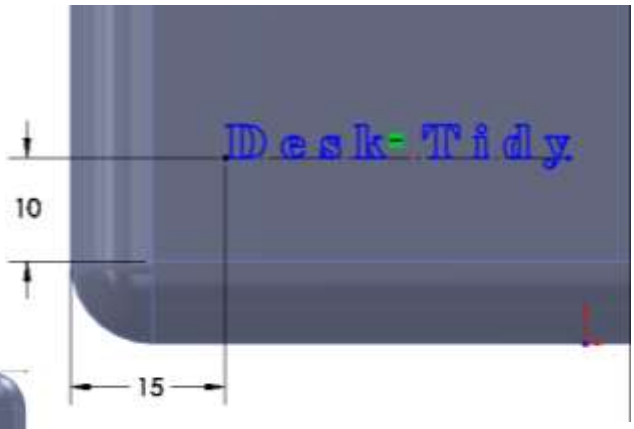


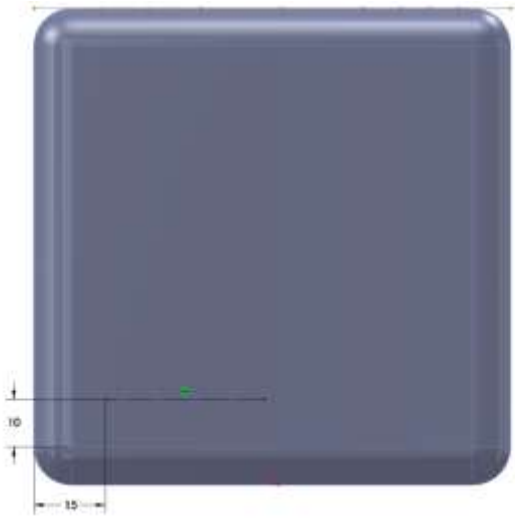




Add Text

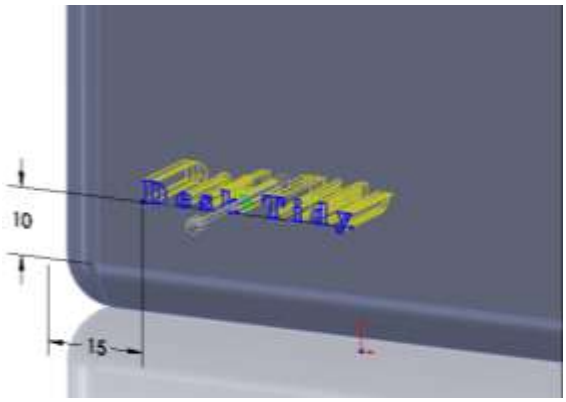
Select the front face of the cube. Sketch a line on the face using the measurements as shown. In the Sketch menu select **Text** as shown and change the Text properties as shown.





Create Extrude Cut of Text

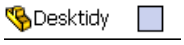
Select Extruded Cut from the Features menu. Ensure that the **End condition** is **Blind** and the **Depth** is **1.5mm**.

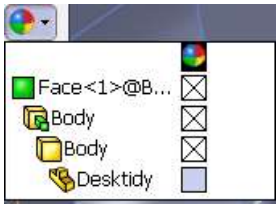


Adding an Appearance to the Desk Tidy

Right click on any face of the model and choose

Appearance Callout

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, Desk Tidy. 



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

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

Within the RealView/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.

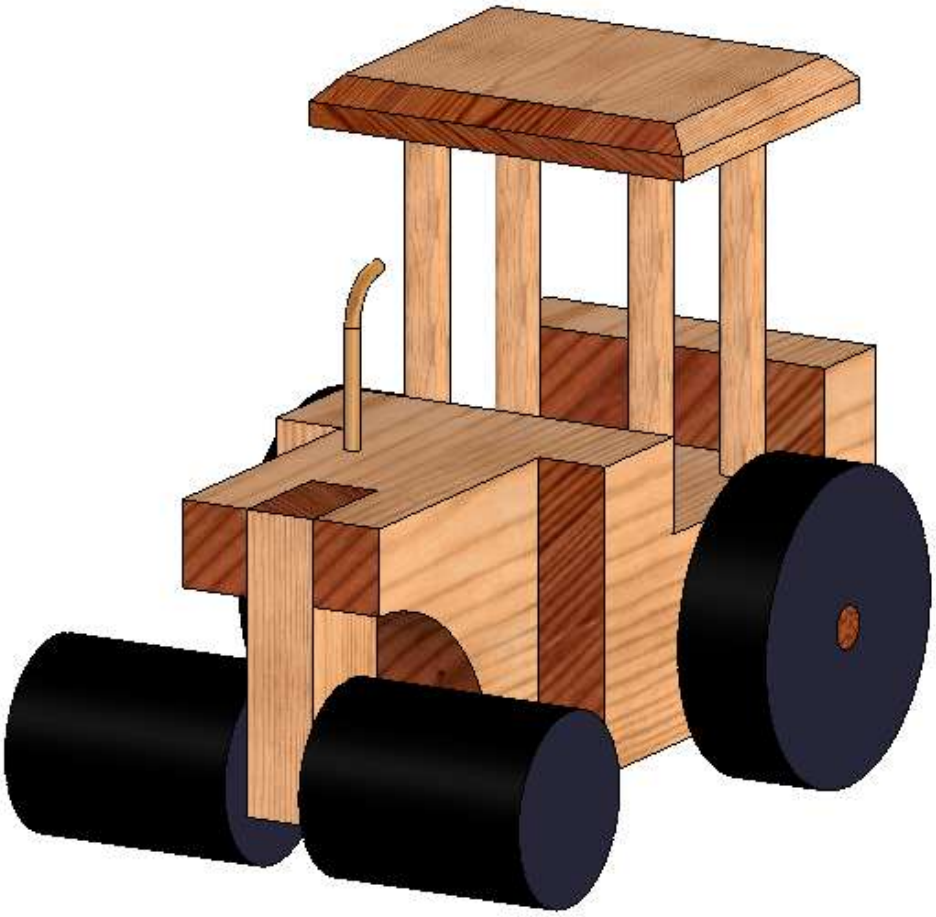
Choose **Organic, Miscellaneous, Sponge.**

The Sponge effect is applied to the entire part.

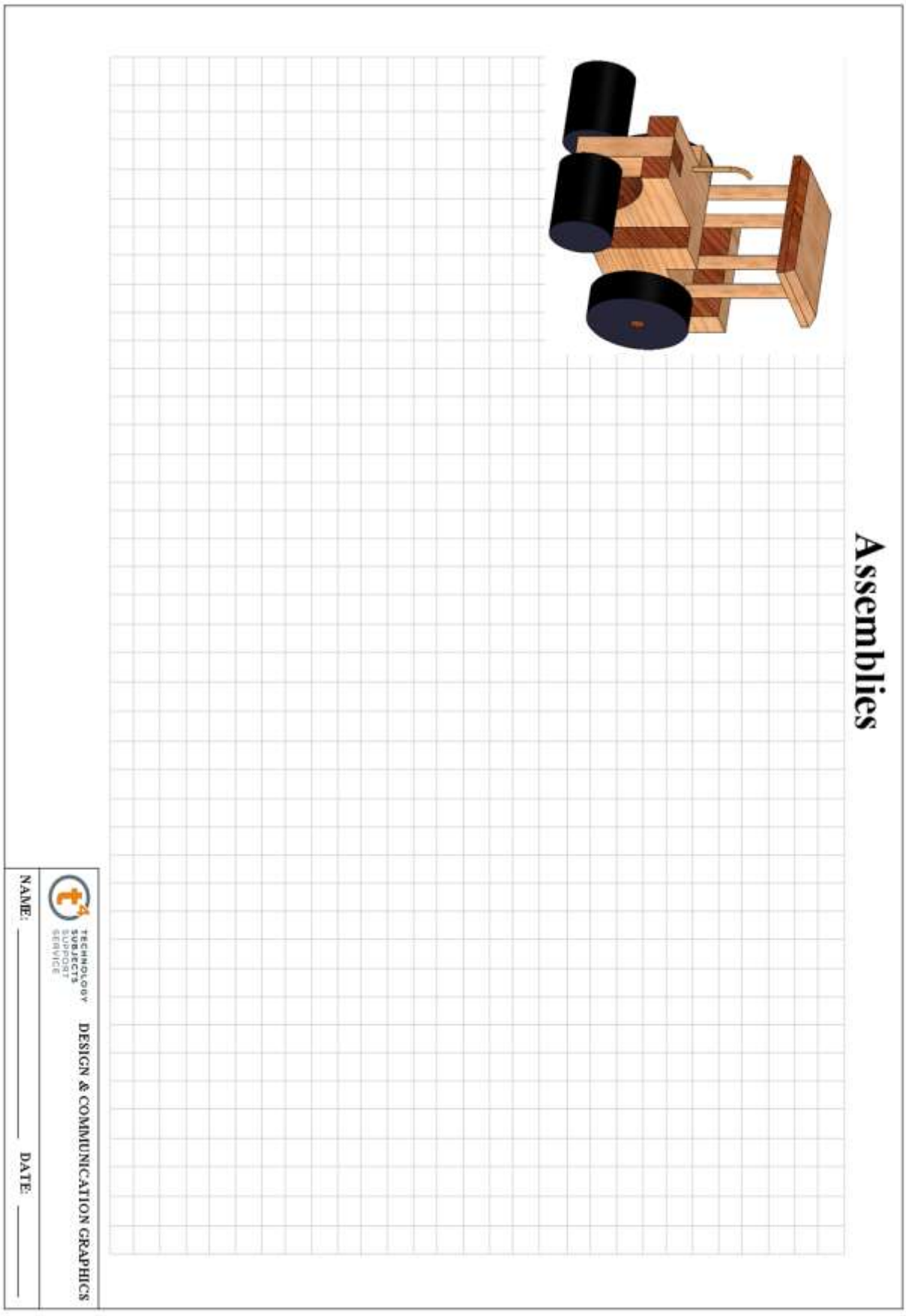
Choose **OK**

Other Possible Extrude Options

Steam Roller



Object Analysis sheet



Prerequisite Knowledge

Previous knowledge of the following commands is required to complete this lesson. **Sketching (**Line, Rectangle, Arc, Add Relations, Dimensioning), **Extrude, Assemblies.**

Commands Used

This lesson includes **Sketching** (line, circle, arc, Smart Dimension), **Cut Extrude** with a line, **Add relations, Sweep Boss/Base, Appearance, Assemblies,** and **Advanced Mates.**

Getting started

Create a **New Folder** in your chosen location called '**Steam roller'**.

The nine parts that make up the project will be saved here.

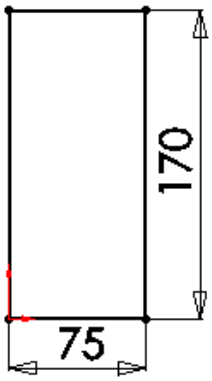
Part One - Steam Roller Body

New Part

Open **New Part** from the **SolidWorks Document** dialog box.
Select **File.** Click **Save as** on the standard toolbar. Save as **'**Steam roller body' in the **'**Steam roller**'** folder.
Continue to save periodically throughout the exercise.

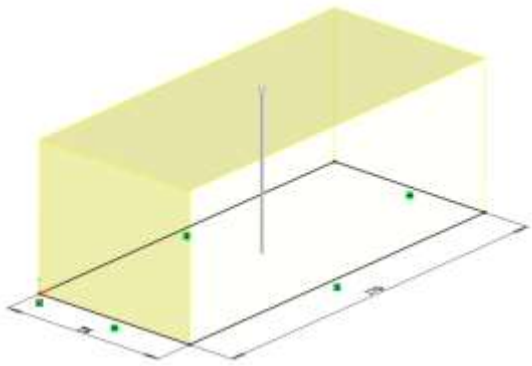
Create sketch

Create a sketch on the **Top Plane** using the dimensions shown.



Confirm the sketch.

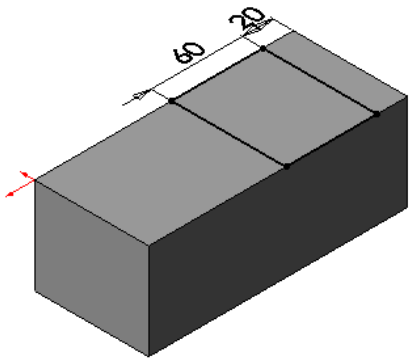
Extrude the sketch to a **depth** of **63mm.** Use a **Blind** end condition



Rename the feature as 'Main Body'

Sketching recess for Cab

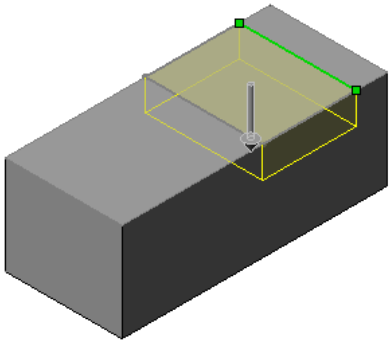
In the sketch menu select **Rectangle** command and draw the rectangle shown. Add the dimensions shown to fully define the sketch.



Exit the sketch.

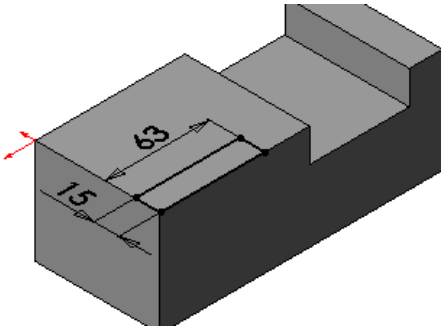
Select **Extruded Cut** to a depth of **20mm**.

Rename the feature cut as ‘Cab’

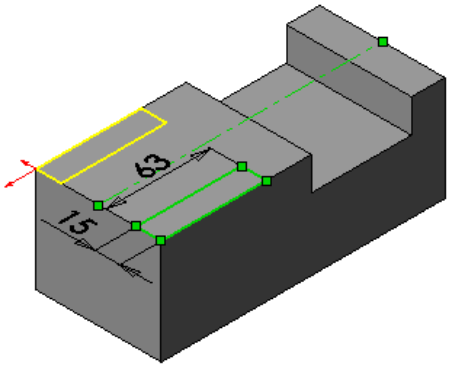


Sketch for Engine

On the top surface draw the **rectangle** shown to the following dimensions**.**



Sketch a centerline on the top surface so that the rectangle can be mirrored**.**

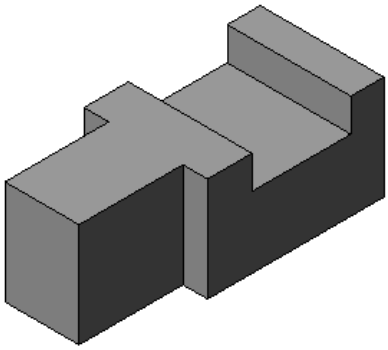


Click the ok button to accept.

Confirm the sketch.

Select **Extrude Cut** and **Through all**.

Rename feature as ‘Engine’.



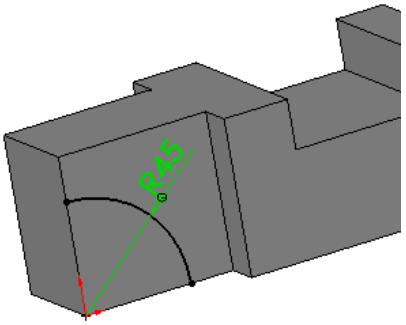
Creating the Nose Section

Select sketch from the command manager.

Choose the front face shown

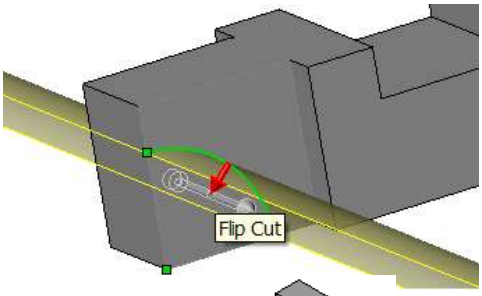
Using **Centrepoint Arc**

draw the arc to the dimension shown



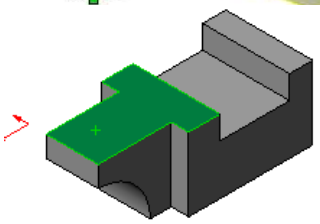
Exit the sketch and select **Extrude Cut** and **Through all**.
Flip the side to cut if necessary.

Rename the feature as 'Front curve'.



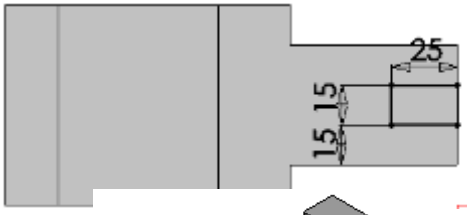
Creating Recess for Upright

Select **sketch** from the **Command Manager** and select the top face. Select **Normal to** from the custom tool bar.



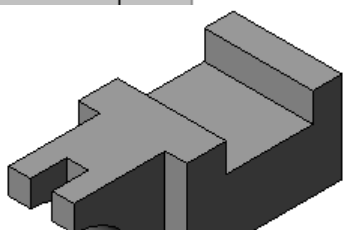
Creating the sketch

Sketch the rectangle shown to the given dimensions.



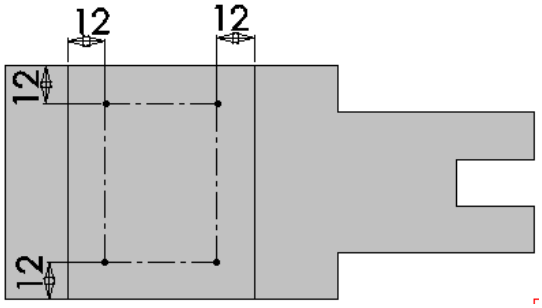
Exit the sketch and select **Extruded cut** and **Through all** as the end condition.

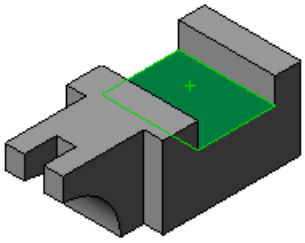
Rename the feature as 'Cut for upright'.



Creating the holes for dowels

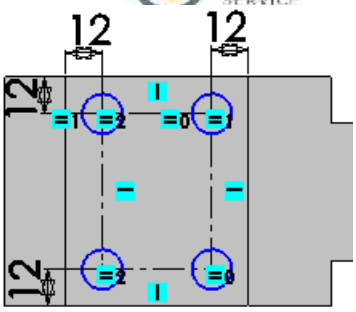
On the surface shown draw a rectangle using the **centerline** command.
Add the dimensions shown





Add Relations

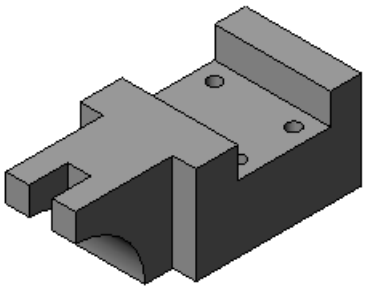
Draw a circle at each corner.
Use **Add relations** to make the circles **equal**.



Add the diameter of **9mm** to the circles.
Exit the sketch.

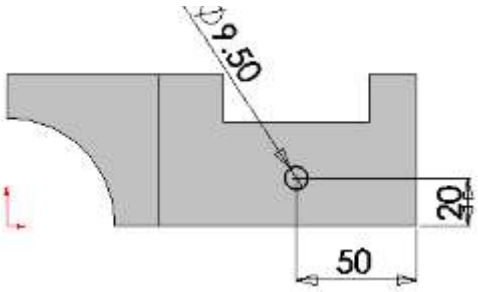
Select **extrude cut** to a depth of **15mm**

Rename feature as 'Holes for dowel'



Hole for Back Axle

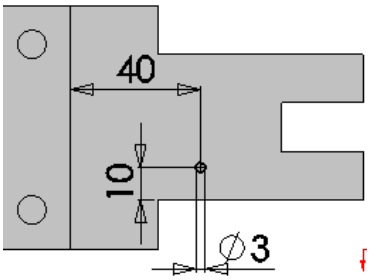
Using the sketch commands draw a circle on the front face to the given dimensions.



Exit the sketch and use **Extruded cut through all.**
Rename feature as 'Hole for back axle'.

Hole for Exhaust Pipe

On the top surface sketch a circle to the dimensions given.

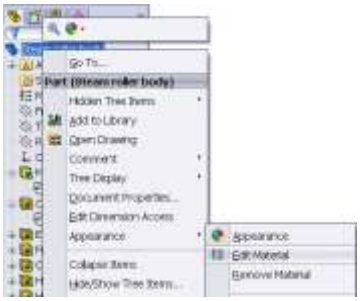


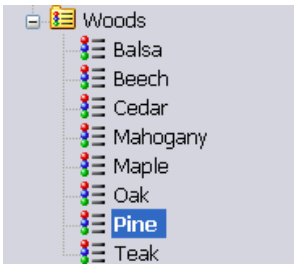
Select **Extruded cut** to a distance of **10mm**

Rename the feature as 'Hole for exhaust Pipe'.

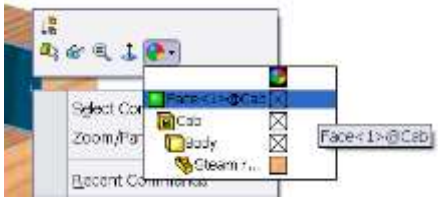
Edit material

In the design tree right click on **Steam roller body/ Appearance/ Edit material.**
Select '**Pine**' from the '**Woods**' folder and adjust the scale and angle of the grain.





Apply an **'Unfinished Pine Endgrain"** texture to the faces that contain end grain as shown. Select **all** of the faces together and right click on Face<1>@Cab and select **Unfinished Pine Endgrain** from the Pine folder. Adjust the colour, scale and angle of the grain



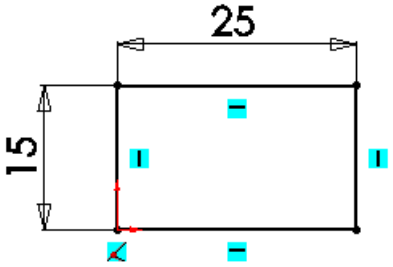
Part two - Front Upright

New Part

Save part as **'**Front upright' in the 'Steam roller|' folder.

Sketch

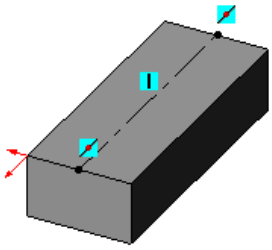
Sketch a rectangle on the **Front Plane.**



Add dimensions.

Accept the sketch and extrude by **63mm**.

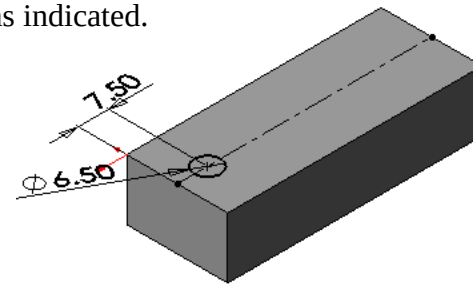
Rename the feature as 'Front upright'.



Drilling Hole

On the top surface draw a centerline between the midpoints of the opposite short edges.

Draw a circle on this centerline and dimensions as indicated.



Accept the sketch.

Select **Extruded Cut** and **Through all**.
Rename the feature as 'Hole for front axle'.

Edit Material

Apply a Pine material to the part and add an **'Unfinished Pine Endgrain"** Material to the faces containing end grain.

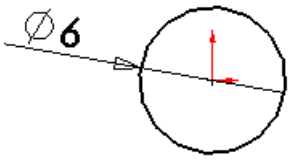
Part Three – Front Axle

New Part

Save the part as 'Front Axle' in the 'Steam Roller' folder

Sketch

Sketch the following circle on the **Front Plane.**

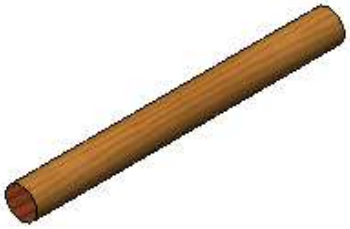


Accept the sketch and extrude by **60mm**
Rename the feature as **'**Axle'.

Edit Material

Apply a Pine material to the part and add an **'Unfinished Pine Endgrain"** Material to the faces containing end grain

Save the part



Part Four – Front Wheel

New Part

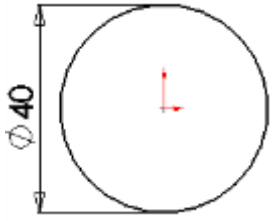
Save the part as 'Front Wheel' in the 'Steam Roller' folder.

Sketch

Sketch the circle shown on the **Front Plane.**

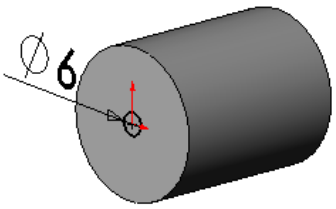
Accept the sketch.

Extrude the sketch by **50mm.**
Rename the feature as 'Wheel'.



Hole for Axle

Sketch a circle on the face of the wheel. This circle must be concentric with the outer circle. Add dimension.

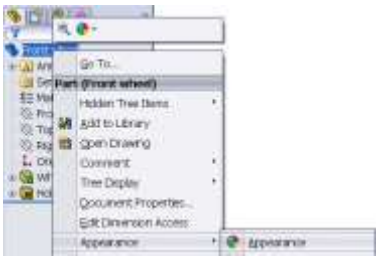


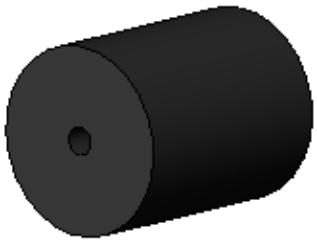
Accept the sketch.

Select **Extruded Cut** and apply a distance of **20mm**.

Edit Appearance

Right click on the part in the design tree
Select **Appearance** and choose black as the colour for the wheel.





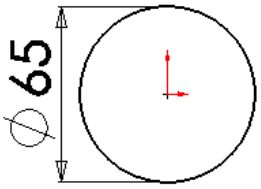
Save the part

Part Five -Back Wheel

New Part

Save the part as 'Back Wheel' in the 'Steam Roller' folder.

Sketch the circle shown on the **Front Plane**.

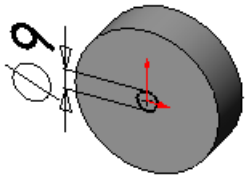


Accept the sketch and **extrude** by **20mm**.

Rename the feature as 'Back wheel'..

Hole for Axle

Sketch another circle with diameter **9mm**



on the face of the wheel as shown.

Accept the sketch and select **Extruded cut**
With **Through all** as the end condition.

Edit Appearance

Apply black as the colour for the wheel as before.

Save the part

Part Six – Back Axle

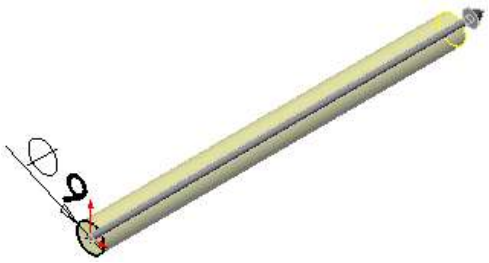
New Part

Save the part as 'Back axle' in the 'Steam Roller' folder.

Sketch

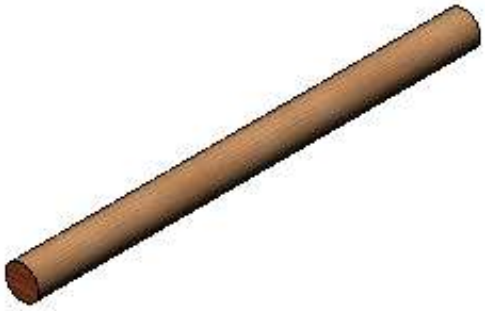
Sketch a circle on the **Front Plane** with a diameter of **9mm.**
Accept the sketch.
Extrude the sketch by **120mm.**

Rename the feature as 'Back Axle'.



Edit Material

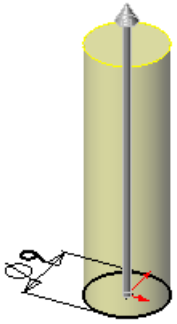
Apply a Pine material to the part and add an **'Unfinished Pine Endgrain"** Material to the faces containing end grain



Part Seven – Cab Support

New Part

Save the part as ‘Cab Support’ in the ‘Steam Roller’ folder.



Sketch

Sketch a circle of diameter **9mm** on the **Top Plane.**
Accept sketch.
Extrude the sketch by **95mm.**
Rename extrusion as ‘Cab support**’.**

Edit Material

Apply a Pine material to the part and add an **‘Unfinished Pine Endgrain”** Material to the faces containing end grain

Save the part

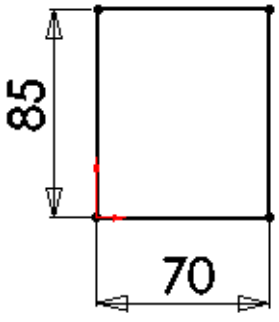
Part Eight – Cab Cover

New Part

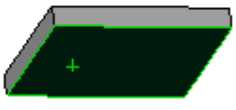
Save the part as ‘Cab Cover’ in the ‘Steam Roller’ folder.

Sketch

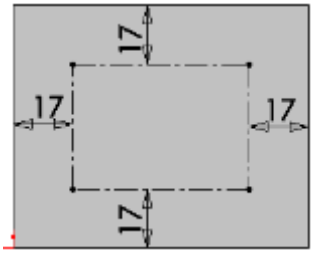
Sketch a rectangle on the **Top Plane.**
Dimensions as shown.



Accept the sketch and extrude by **10mm.**
Rename the feature as ‘Cab Cover’.



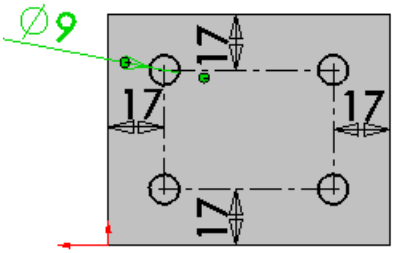
Select the underside of the cab.
Select **Normal to** and draw the centerlines shown. Sketch a circle at each corner.



Add Relations

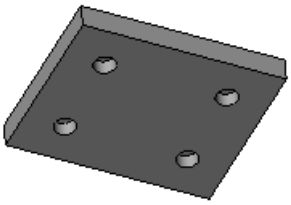
Use **Add relations** to make all the circles equal.

Dimension the diameter of the circles **9mm.**



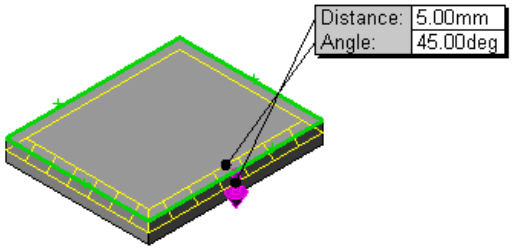
Accept the sketch.

Select **Extruded cut** and apply a depth of **6mm.**
Rename the cut feature as 'holes'**.**



Chamfer

Select **chamfer** in the features toolbar.
Apply a chamfer of **5mm** to the top edges.



Edit Material

Apply a Pine material to the part and add an **'Unfinished Pine Endgrain"** Material to the faces containing end grain

Save the part.

Part Nine – Exhaust Pipe

New Part.

Save the part as 'Exhaust Pipe' in the 'Steam Roller' folder.

To create the Exhaust pipe **Sweep boss/base**

will be used.

For this two sketches need to be drawn

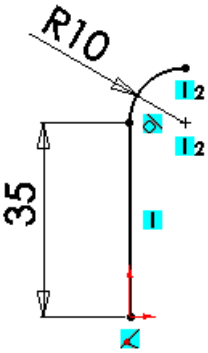
1. The sweep path.
2. The sweep section.

Sketch 1.

Use the **Line** command on the **Front Plane** to sketch the profile shown.
One end of the line segment is placed at the origin.

Add Relations

Use **Add Relations** to make the end of the arc **vertical** with its center point
Add the dimensions.
Confirm the sketch**.**

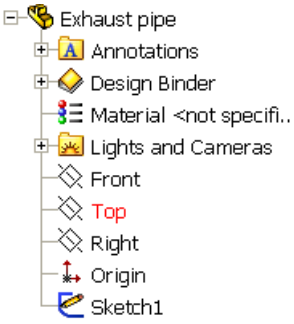


Sketch 2.

Select sketch command again and select the **Top Plane** from the design tree.

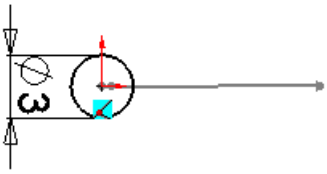
Select **Normal to** to look perpendicular to the plane.

Draw the circle with its center point at the origin as shown.



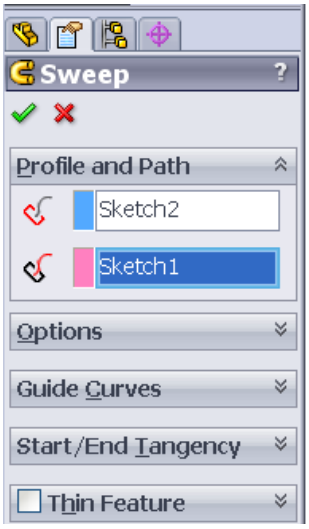
Dimension the diameter **3mm**.

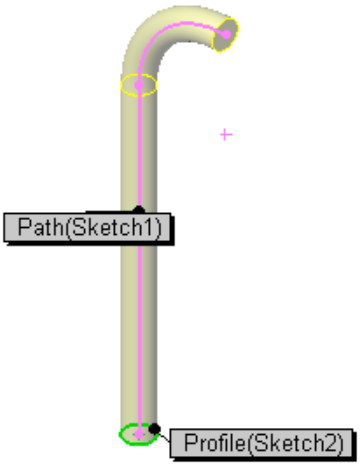
Confirm the sketch**.**



Sweep

Choose **Sweep Boss/Base** from the features toolbar.
Select the circle as the profile and the line as the path.

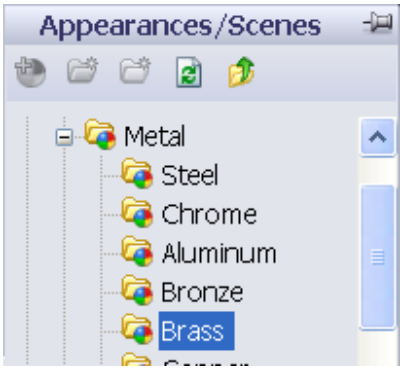




Confirm the sketch**.**
Rename the feature as 'Exhaust pipe'.

Edit Material

Select the material **brushed brass** from the brass folder as shown



Save the part.

Steam Roller Assembly

The part files for this assembly are saved in the folder titled 'Steam Roller'.

Open an existing part

Open the part called 'Steam roller body'**.**

Click **Make Assembly from Part/Assembly.**

Insert component dialog box appears with **'**Steam roller body' displayed.

Click on ı the property manager.

The part origin will snap to the origin of the assembly.

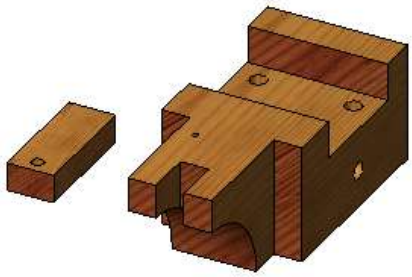
Save

Select **File, Save as** on the standard toolbar.
Save the assembly as 'Steam Roller' in the same folder as its parts.

Adding Component

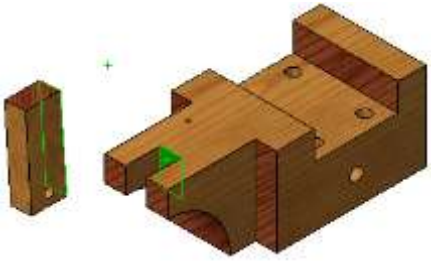
Select **Insert component**

from the **Assembly** toolbar**.**
Choose **Browse** from the **Insert Component** dialog box.
Click on 'Front upright', open and click in the graphics area to place the part as shown.



Use the **Rotate** components

to rotate the pai to the required orientation.



Insert Mates

Select **Mate**

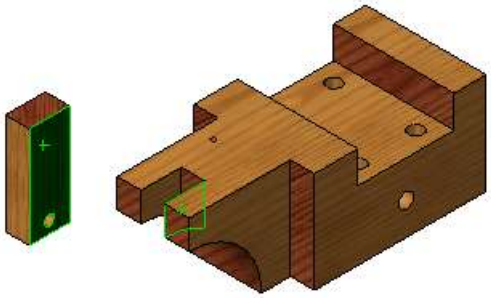
from the assembly toolbar.

Mate the back edge of the upright with the back of the trench shown.

A **Coincident Mate** will be selected by default.

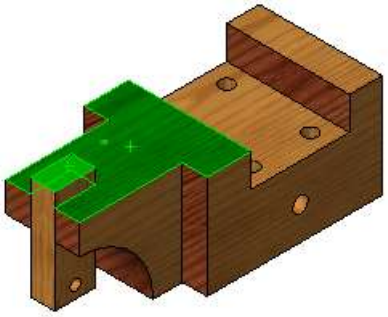
Accept

Mate the side of the upright with the side of the trench.



Accept .

Finally mate the top of the main body with the top edge of the upright.



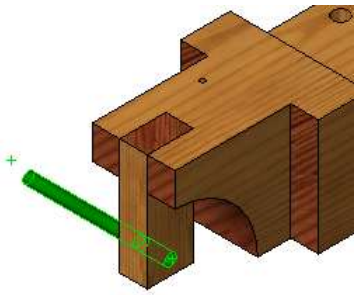
Accept

Select OK again to exit the property manager.

The upright is now fully defined i.e. it has no degree of movement

Inserting Further Components

Choose **Insert Component** from the assembly toolbar. Browse, select the 'Front axle' and insert into the graphics area as before.

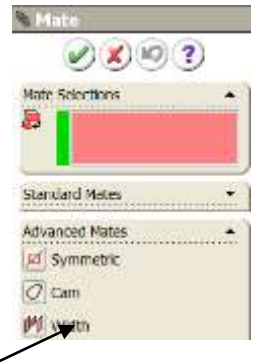


Insert Mates

Mate the surface of the dowel with inside surface of the hole

A **Concentric Mate** will be chosen by default.

Accept this mate.



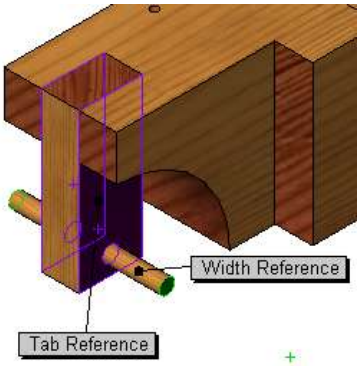
Additional Mates

The axle has to be centered in relation to the upright

In the mate property manager expand **Advanced Mates** and select **Width Mate.**

Select the two end surfaces of the axle for the **Width Selection.** Choose the two sides of the upright for the **Tab Selections.**

Should any dimensions change the axle will always remain centered on the upright.



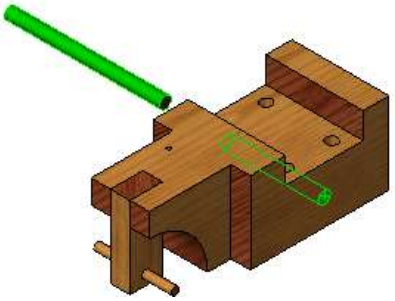
Accept the mate.

Select **OK** again to exit the property manager.

The axle is under defined i.e. it is free to rotate as it would in reality.

Adding Back Axle

Select **Insert Component** from the assembly toolbar
Choose the "**Back axle**" and drop it into position.



Insert Mates

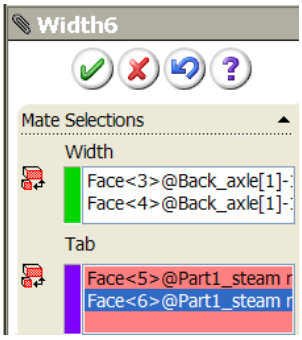
Follow the same process as for the front axle.
Mate the surface of the axle with the inside surface of the hole.

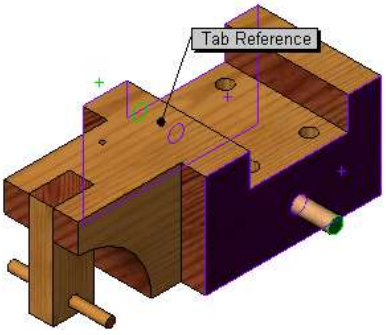
Accept .

Select **Advanced Mates** to centre the axle in Relation to the body of the steam roller.

Select **Width** in the **Advanced Mates** dialog box.

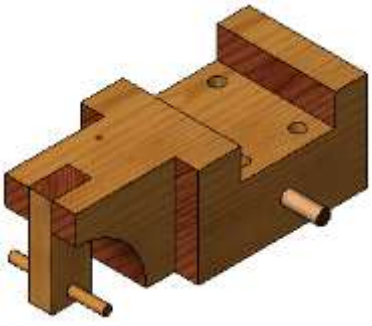
Select the two end surfaces of the axle as the **Width Selections**
Select the two sides of the body as the **Tab Selections**





Accept .

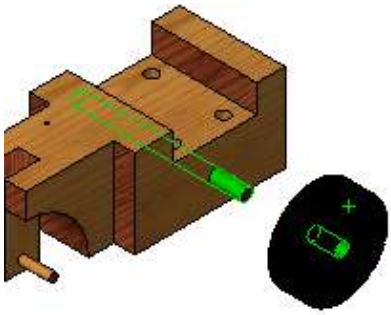
Select **OK** again to exit the property manager.



The axle is under defined i.e. it is free to rotate as it would in reality.

Adding the Back Wheels

Select **Insert Component** from the assembly toolbar. Choose the "**Back Wheel**" and drop it into position.

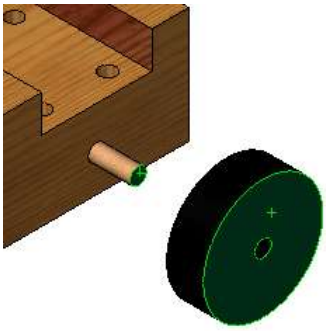


Insert Mates

Select **Mate** and select the surface of the axle and the inside of the hole.

A Concentric Mate is selected by default.

Accept

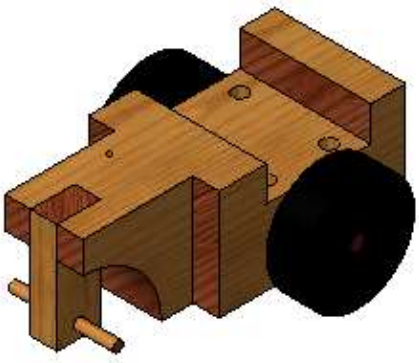


Mate the outside surface of the wheel with the end surface of the axle.

Accept

Select **OK** again to exit the property manager.

Follow this process again to assemble the other back wheel.

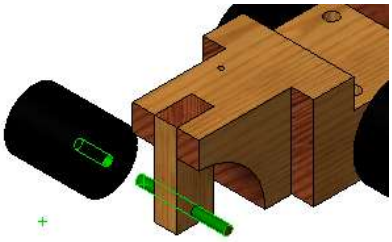


Inserting the Front Wheel

Select **Insert Component** from the assembly toolbar. Choose the "**Front Wheel**" and drop it into position.

Insert Mates

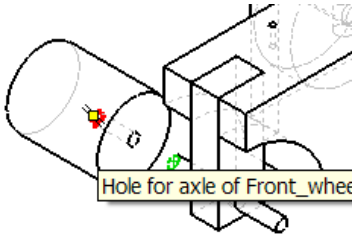
Select **Mate** and mate the surface of the front axle with the inside surface of the hole.



Accept

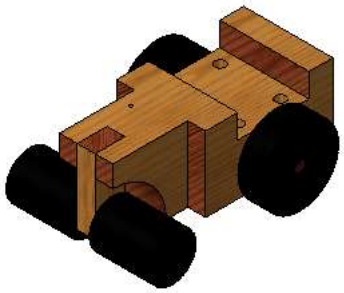
Mate the end of the axle with the bottom surface of the hole

To select the bottom of the hole **Hidden lines visible** may need to be selected.



Accept

Select **OK** again to exit the property manager.

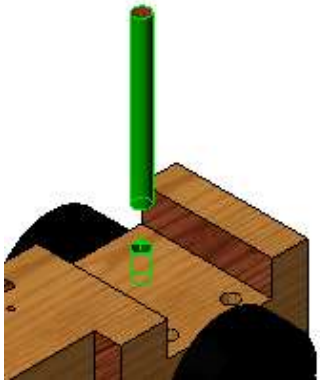


Carry out the same procedure to assemble the other back wheel.

The wheels are under defined i.e. they are free to rotate as they would in reality.

Inserting the Cab Supports

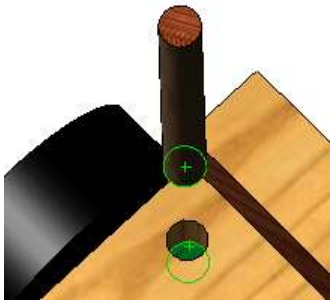
Select **Insert Component** from the assembly toolbar. Choose the 'Cab Support' and drag it into position.



Insert Mates

Choose **Mate,** select the surface of the support and the surface of the hole as shown.

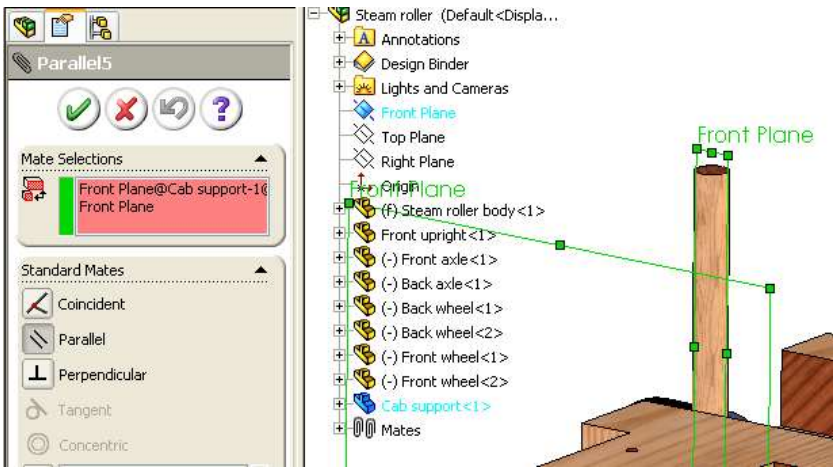
Accept the **Concentric Mate.**



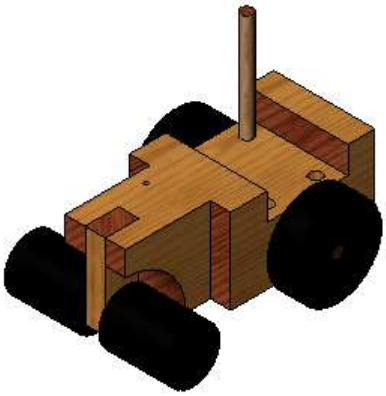
Mate the bottom of the hole with the end surface of the support.

Accept the **Coincident Mate.**

Mate the **Front Plane** of the assembly with the **Front Plane** of the 'Cab Support'. Apply a **Parrallel** mate.



The component is now fully defined.

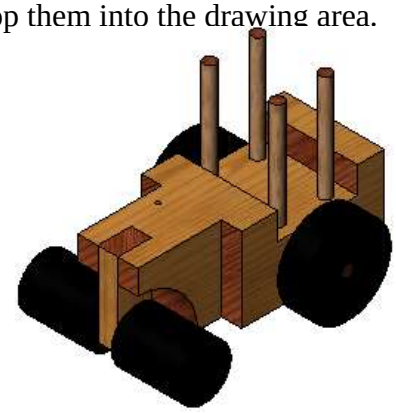


Select **OK** again to exit the property manager.

Adding the other Supports

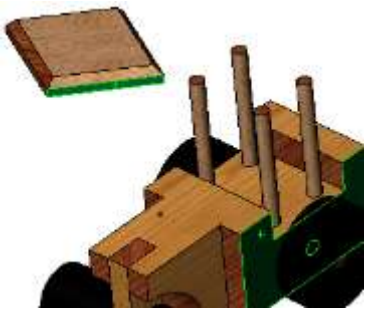
To add in the other **Supports** hold down the **Ctrl key**, select and drag the supports from the **Feature Manager Tree.** Drop them into the drawing area.

Mate the supports as before



Inserting Cab Cover

Select **Insert Component** from the assembly toolbar. Choose the "**Cab Cover**" and drop it into position.

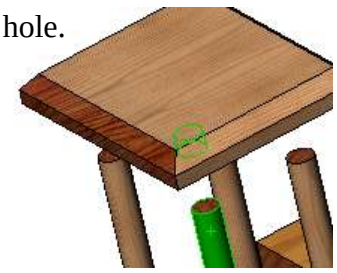


Insert Mates

Select **Mate** and select the side of the cab cover and the side of the body. Apply a **Parallel Mate.**

Accept

Mate the surface of the support shown with the appropriate hole.



Accept

Mate the end of the support with the bottom of the hole.

Accept

The component is now fully defined.

Select **OK** again to exit the property manager.

Inserting the Exhaust Pipe

Select **Insert Component** from the assembly toolbar.
Choose the "**Exhaust Pipe**" and drag it into position.

Insert Mates

Mate the surface of the pipe with the surface of the hole.

A **Concentric Mate** is chosen by default.

Accept

Mate the bottom of the hole with the end of the pipe.

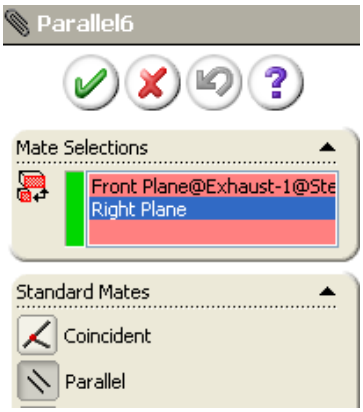
A **Coincident Mate** is chosen by default.

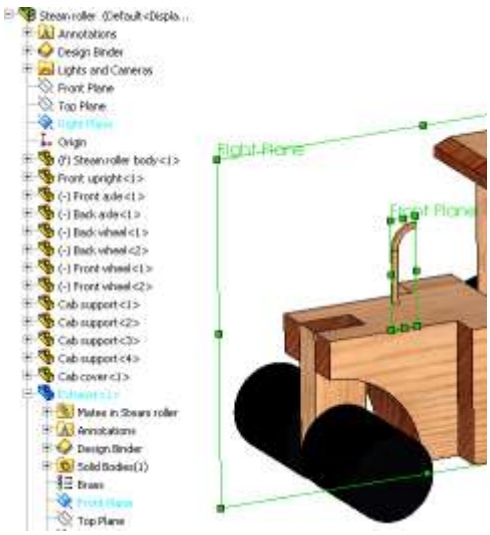
Accept

Preventing the Pipe From rotating

To fully define the exhaust pipe, a parallel mate must be added between the **Right Plane** of the assembly and the **Front Plane** of the component.

Select **Mate.** Expand the "Exhaust Pipe" part in the graphics area.
Select the **Front Plane** as shown.
Choose the **Right plane** of the assembly



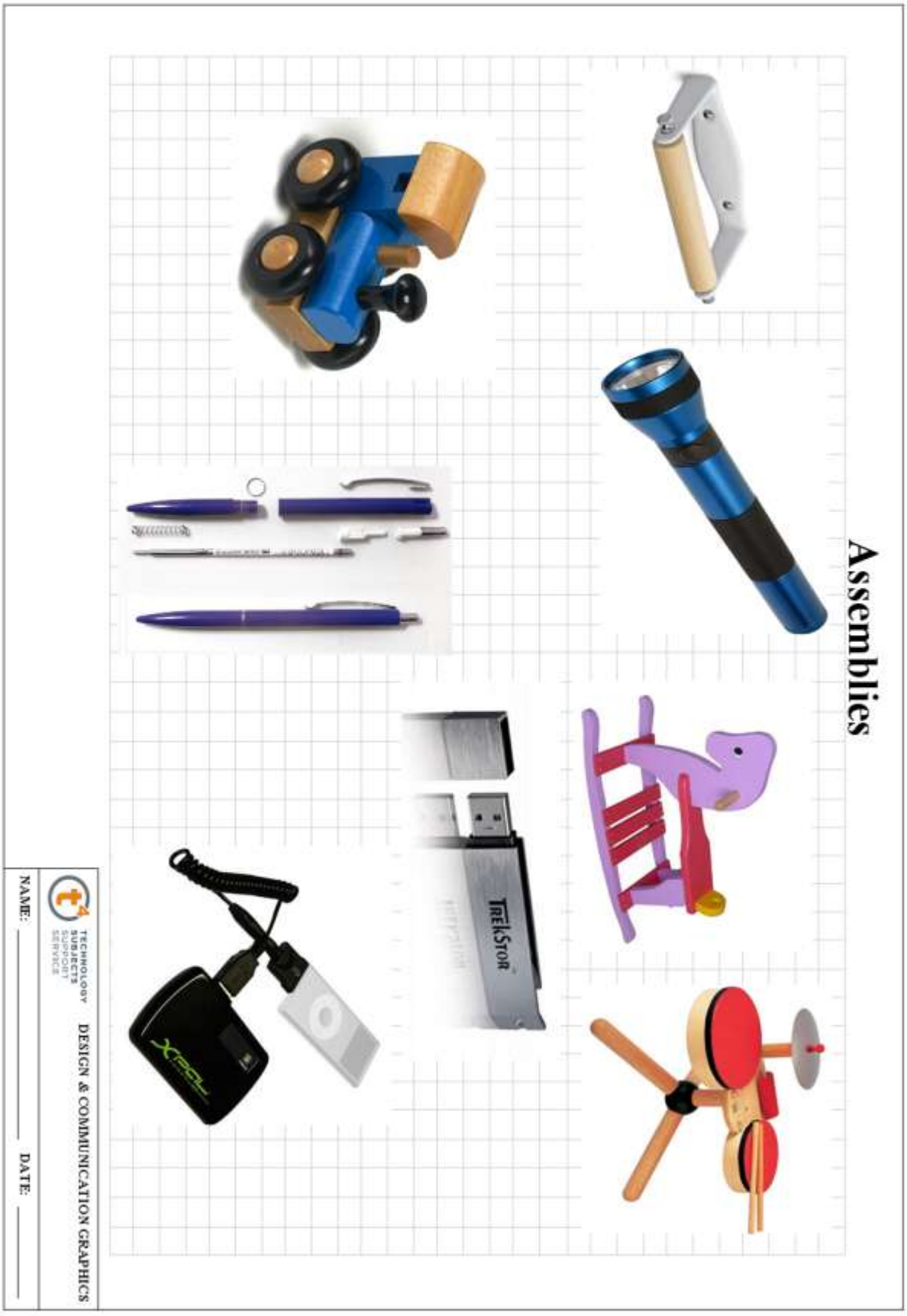


Apply a **Parallel** mate**.**

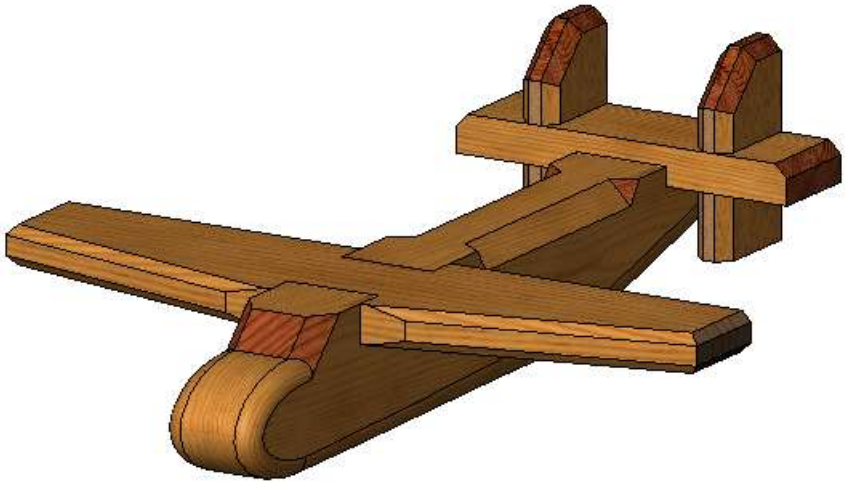
Accept

Select **OK** again to exit the property manager.

Other Possible Assembly Options



AEROPLANE



Prerequisite Knowledge

Previous knowledge of the following commands is required to complete this lesson.
Sketching (Line, Rectangle, Arc, Add Relations, Dimensioning),
Extrude, Assemblies and Creating a drawing from Part/Assembly.

Focus of the Lesson

On completion of this lesson you will have used:

- **Cut Extrude** with a line.
- Edit **Appearance**.
- **Variable Fillet.**
- Created an **Assembly**.
- **Mate** with planes**.**
- **Exploded view** of an assembley.
- **Animation** of exploded view

Commands Used

This lesson includes **Sketching** (line, circle, arc, ellipse Smart Dimension), **Cut Extrude** with a line, **Add relations, Appearance, Variable Fillet, Assemblies, Mates, Exploded View, and Animation.**

Getting started

Create a **New Folder** in your chosen location called '**Aeroplane'**.

The four parts that make up the project will be saved here.

Part One – Aeroplane Body

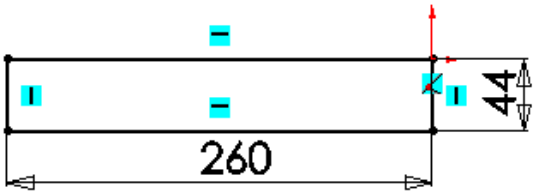
Open **New Part** from the **SolidWorks Document** dialog box.
Select **File.** Click **Save as** on the standard toolbar. Save as **"aeroplane body**" in the **"Aeroplane|"** folder.

Continue to save periodically throughout the exercise.

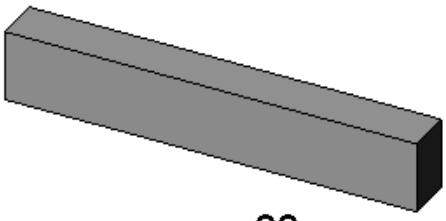
Create sketch

Create a sketch on the **Front Plane** using the dimensions shown.

Confirm the sketch.



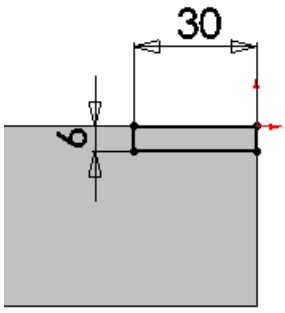
Extrude the sketch to a **depth** of **30mm.** Use **Mid Plane** End Condition.



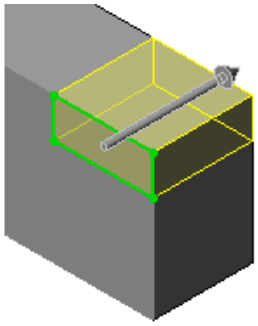
Rename the extrusion as "**Main Body**".

Sketching recess for Back Wing

On the **front face** draw the **Rectangle** shown. Add the dimensions shown to fully define the sketch.

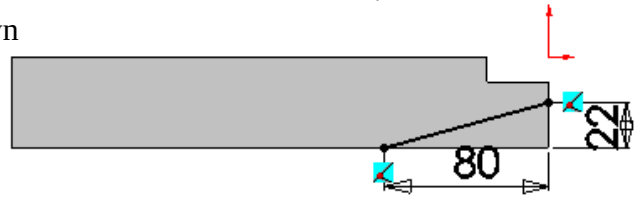


Exit the sketch.
Select **Extrude Cut**, **Through All**.



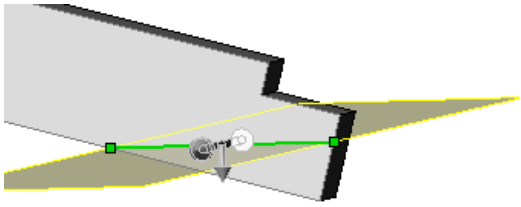
Sketching the Sloped Tail Section

On the **front face** draw the **line** shown to the following dimensions**.**



Confirm the sketch.

Select **Extrude Cut** and **Through All**

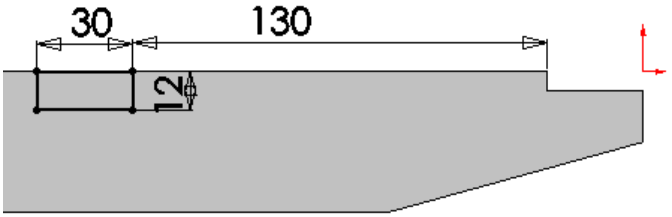


Flip the side to cut if necessary.

Rename Extrusion as "**Underside**".

Sketching Recess for Front Wing

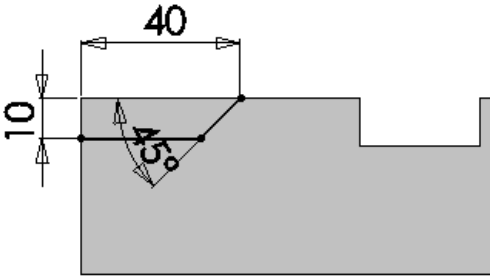
Draw the **Rectangle** to the dimensions shown on the **front face**.



Extrude cut, Through all as before.

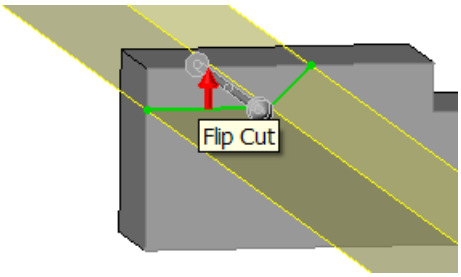
Sketching Front Screen.

On the **front face** sketch the profile to the dimensions shown.



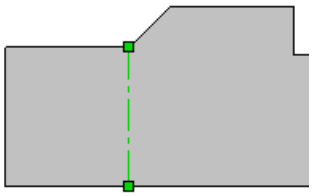
In the feature manager select **Extrude Cut, Through all.**

Flip the side to cut if necessary.

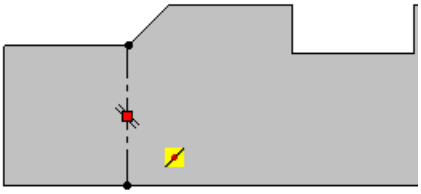


Sketching the Nose Section

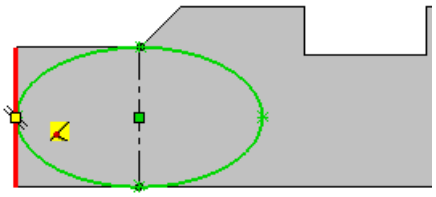
Using the **Centerline** command draw the line as shown.



Select the **Ellipse** command and select the **Midpoint** of the centerline as the centre of the ellipse.

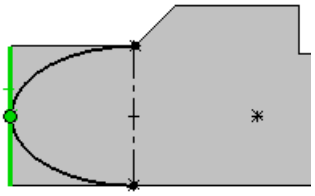


Select the minor and major axis as shown.



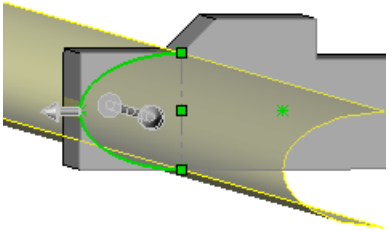
Use the **Trim** command to remove the unwanted portion of the ellipse.

Using **Add Relations** make the line and end of the major axis coincident, to fully define the sketch.



As before select **Extrude Cut** and **Through all**.

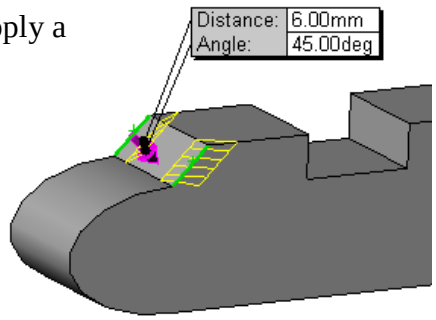
Flip the side to cut if necessary.



Rename the extrusion as **Nose Section**

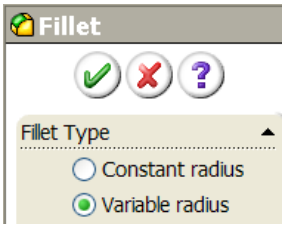
Shaping front section

In the feature manager select **Chamfer** and apply a **6mm** chamfer to the edges shown.

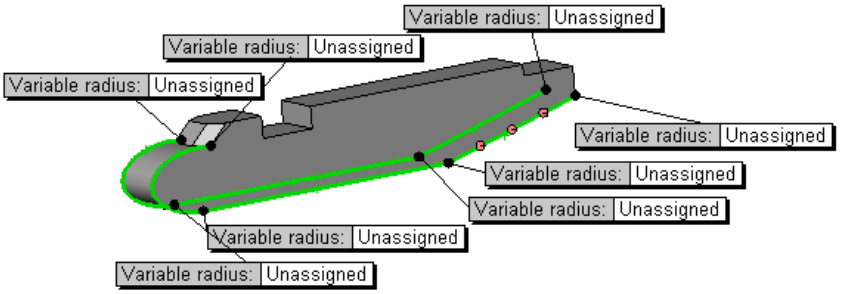


Applying Fillets

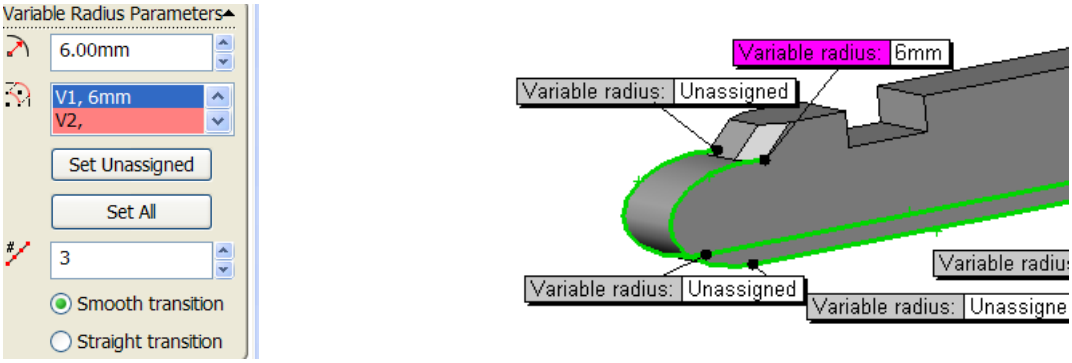
Select **Fillet** and select **Variable radius** as shown.



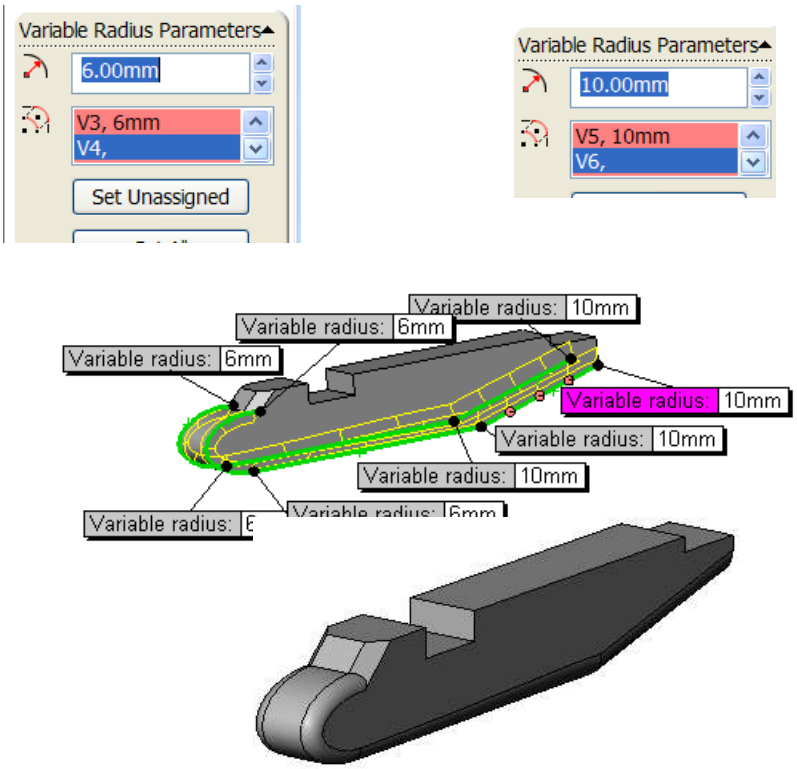
Select the edges to fillet as shown.



In the variable windows parameter box select Variable radius1 **(V1)** and type a radius of **6mm** as shown.



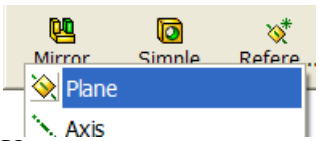
Select the next chain of edges and give them the following radii.



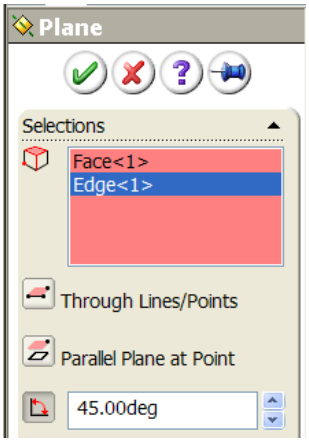
Applying the
Stopped chamfers

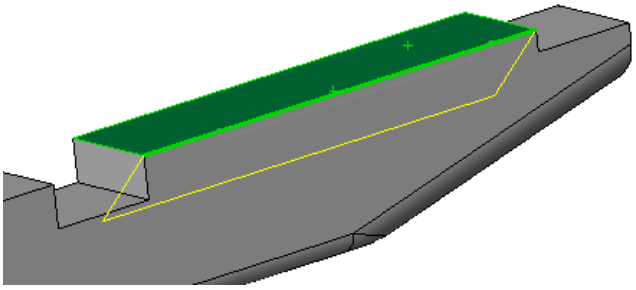
Step 1 Set up a plane perpendicular to the chamfer

Select **plane** under **Reference Geometry**

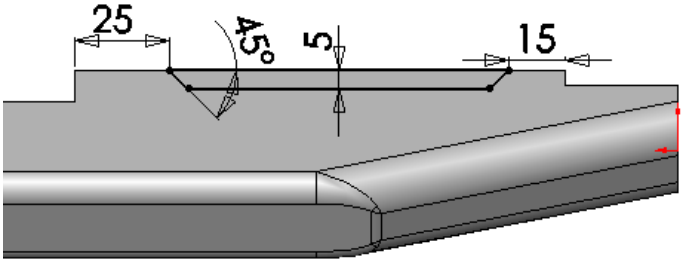


To draw the plane at the required angle, select the top face and the edge to be chamfered. Change the angle to **45 degrees** as shown.

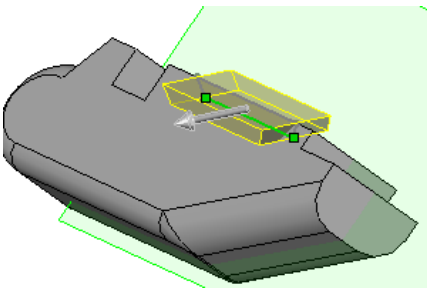




Step 2 Draw the profile on this plane to the dimensions given.



Step 3 Select **Extrude Cut** as shown

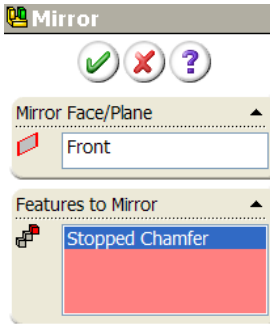


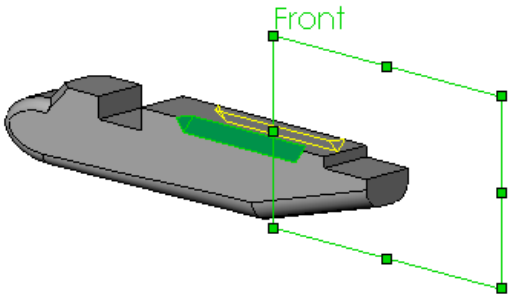
Rename the extrusion as "**Stopped Chamfer**"

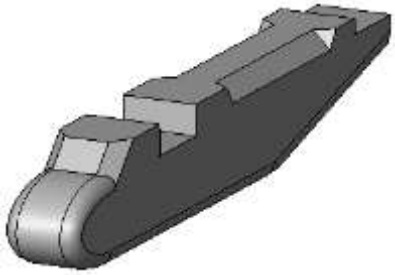
Mirroring the Stopped Chamfer

Select **Mirror Components**
Select the **Front Plane** from the design tree, as the mirror plane.

Select Stopped chamfer from the design tree as the feature to mirror**.**

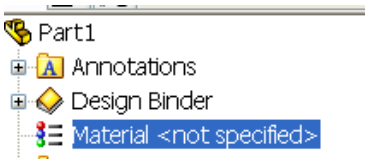




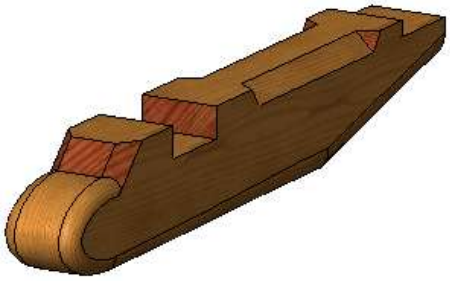


Edit Material

In the design tree right click on **Edit material** shown. Apply a **pine** texture to the object from the **wood** menu.



Apply a **grain 2** texture to the faces that contain end grain.



Save

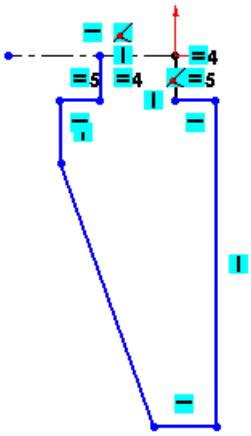
Part two - Front Wing

New Part

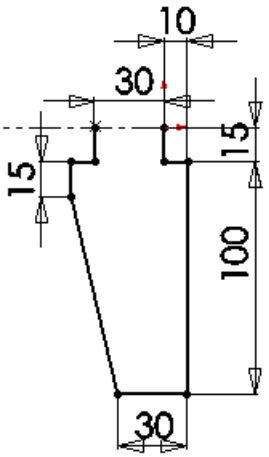
Save part as **"Front wing**" in the **"Aeroplane|"** folder

Sketch

Sketch on the **Top Plane** the shape shown.
Draw the **Centerline** and add the following relations.



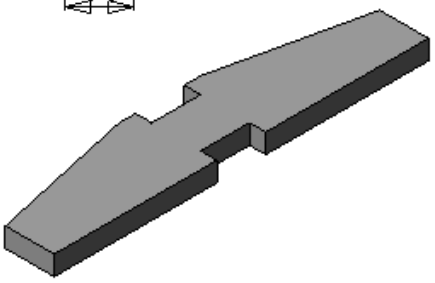
Add the following dimensions.



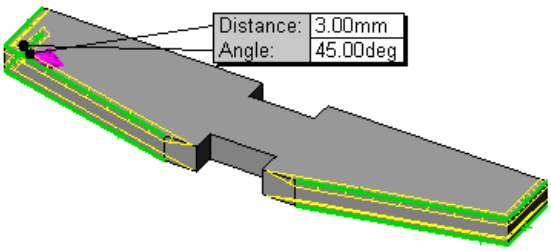
Select **Mirror** and mirror about the centerlin

Accept the sketch and **Extrude** by **12mm**.

Rename the extrusion as "**Front wing**".

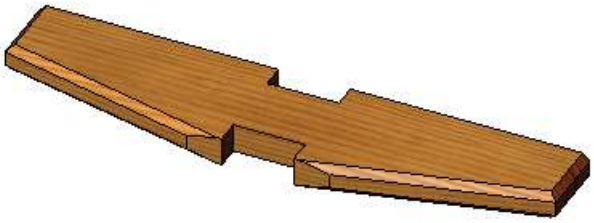


Select **chamfer** and apply a **3mm** chamfer to the four edges shown.



Edit Material

Apply a **pine** texture to the wing.
Apply a **grain 2** texture to the end grain.



Save

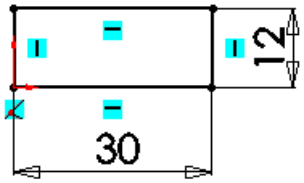
Part three - Horizontal Tail Wing

New Part

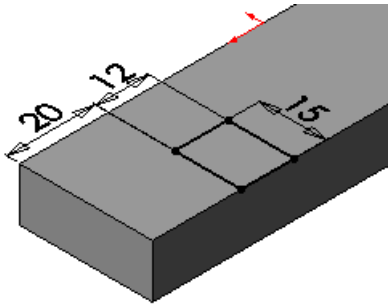
Save part as **"Horizontal tail wing**" in the **"Aeroplane|"** folder.

Sketch

Sketch a **Rectangle** on the **Front Plane** as shown.
Extrude using **Mid Plane** a distance of **110mm**.



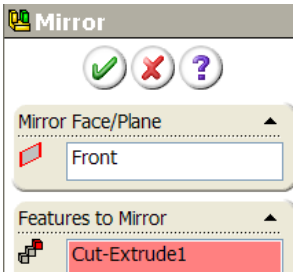
On the top surface draw the rectangle to the measurements given.

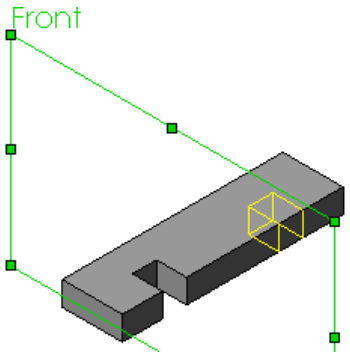


Accept the sketch.

Extrude cut the rectangles **through all**.

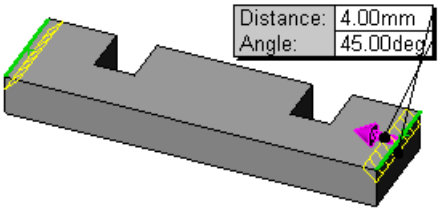
Mirror the feature about the Front Plane as shown.





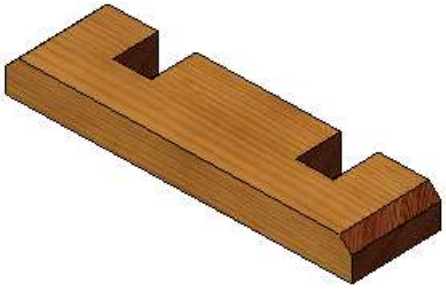
Accept.

Apply a **4mm** chamfer to the following edges.



Edit Material

Apply a **pine** texture to the part
Apply a **grain 2** texture to the end grain.



Save

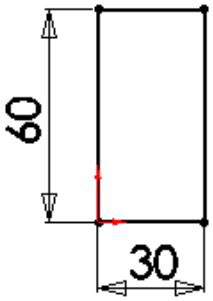
Part Four - Vertical Tail Wing

New Part

Save part as **"Vertical tail wing**" in the **"Aeroplane|"** folder.

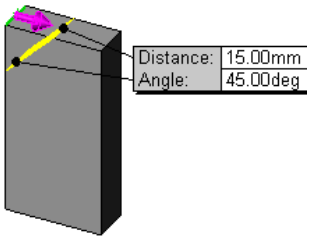
Sketch

On the **Front Plane** sketch the **Rectangle** to the following dimensions.

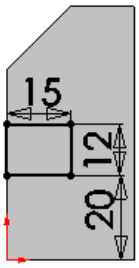


Extrude by **12mm.**

Apply a **Chamfer** of **15mm** to the top edge.

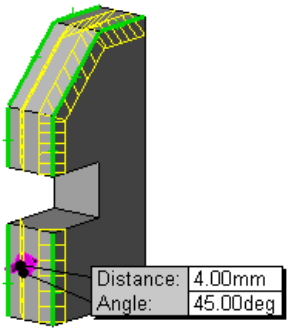


Sketch another **rectangle** on the front face.



Extrude cut, Through all.

Apply **4mm chamfers** to the following edges.



Edit Material

Apply a **pine** texture to the part
Apply a **grain 2** texture to the end grain.

Save the part

Aeroplane Assembly

The part files for this assembly are saved in the folder titled "**Aeroplane**".

Open an existing part

Open the part called "**Aeroplane body".**

Click **Make Assembly from Part/Assembly.**

Insert component dialog box appears with **"Aeroplane body**" displayed.

Click on in the property manager.

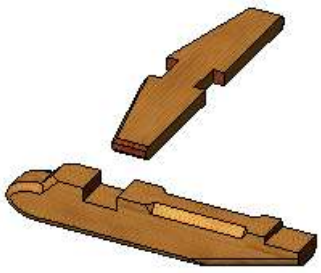
The part origin will snap to the origin of the assembly.

Save

Select **File, Save as** on the standard toolbar.
Save the assembly as "**Aeroplane Assembly**" in the same folder as its parts.

Adding Component

Select **Insert component** from the **Assembly** toolbar**.**
Choose **Browse** from the **Insert Component** dialog box.
Choose "**Front Wing**" and click in the graphics area to place it in as shown.



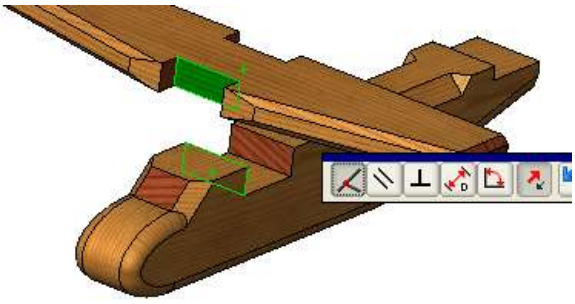
Insert Mates

Select the mate toolbar

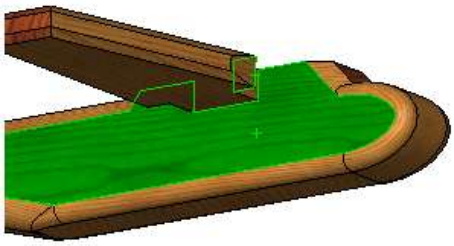
Mate the front face of the trench on the aeroplane body with the bottom of the trench on the wing.

A **Coincident Mate** will be selected by default.

Accept.



Mate the bottom of the trench on the body with the underside of the wing.



Accept

Finally mate the side of the body with the side of the trench on the wing

Accept

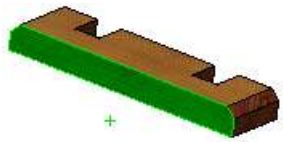
Select OK again to exit the property manager.

Adding Further Parts

Select **Insert Component** from the assembly toolbar.
Choose the "**horizontal tail wing**".

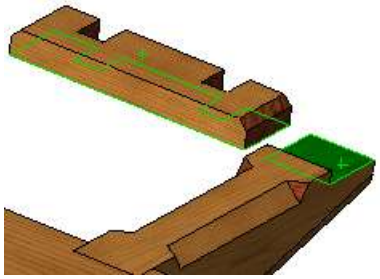
Insert Mates

Mate the front of the horizontal tail with the shoulder shown



A **Concentric Mate** will be chosen by default.

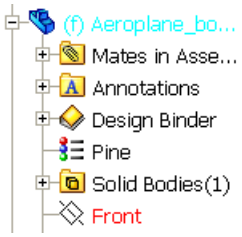
Accept this mate.

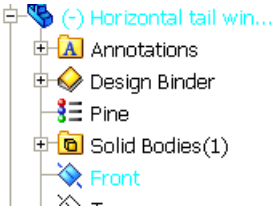


Mate the underside of the tail wing with the recess

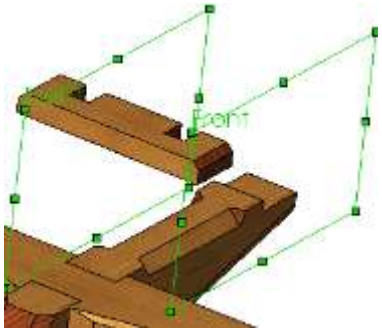
Additional Mates

The horizontal tail wing has to be centered on the plane body
One way to do this is as follows.
Expand the design trees of the two parts and mate the **Front planes** of each as shown..





A coincident mate is selected by default



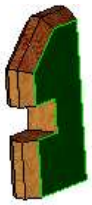
Accept the mate.
Select **OK** again to exit the property manager.

Adding Vertical Tail Wing

Select **Insert Component** from the assembly toolbar
Choose the "**Vertical tail wing**" and drag it into position.

Insert Mates

Mate the bottom of each trench.

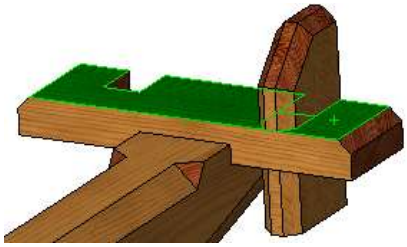


Accept.

Mate the side of the vertical tail with the side of the trench on the horizontal wing

Accept

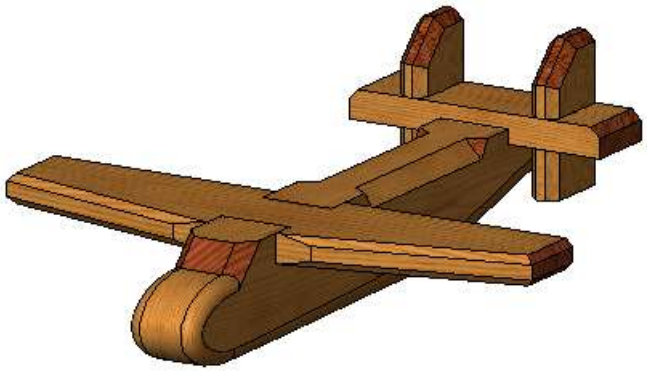
Mate the side of the trench on the vertical tail with the side of the horizontal wing



Accept.

Select **OK** again to exit the property manager.

Follow the process again to insert the other vertical tail wing.



Exploded View

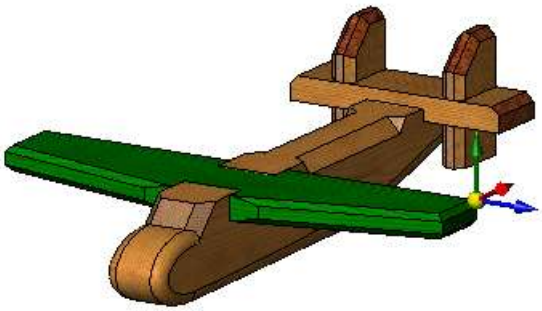
Click the **Exploded View** button

on the Assembly toolbar.

The Exploded view dialog box appears.

Exploding Front wing

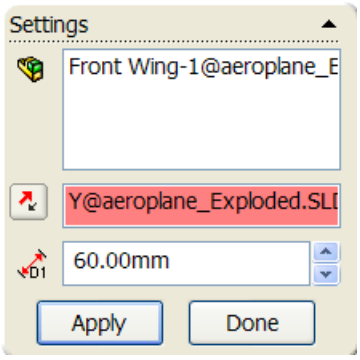
To move the front wing, select it as shown

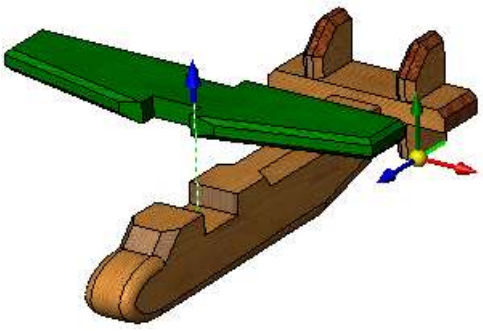


Explode View
Explode by one of two methods:

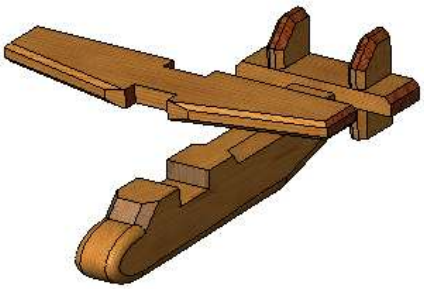
1) Dragging the relevant arrow (in this instance the green one) to the required distance.

2) By scrolling down the explode property manager and selecting the part to explode as shown.
Select the direction **(x, y or z)** by clicking the relevant arrow, and insert the distance
Select **Apply** to preview the selection.
Select **Done** to accept.

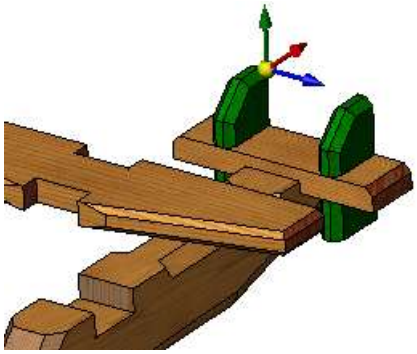




Select **ok** to exit the property manager.



Exploding the Vertical Tail Wings

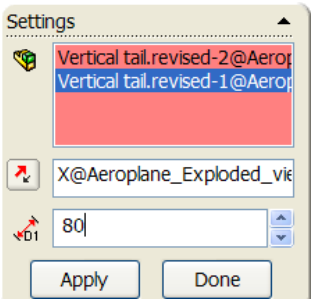


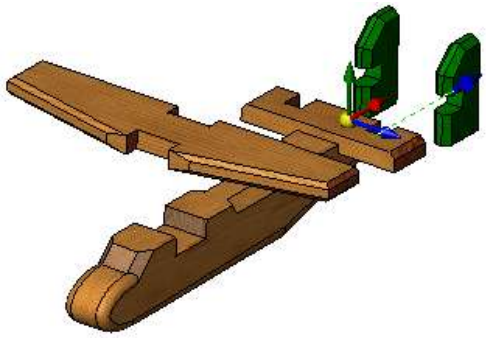
Select the **Explode View** button again.

Select the two vertical tail wings.

The move manipulator arrows appear.

Select the **X direction arrow** and drag outwards or insert a distance of **80mm** as shown.

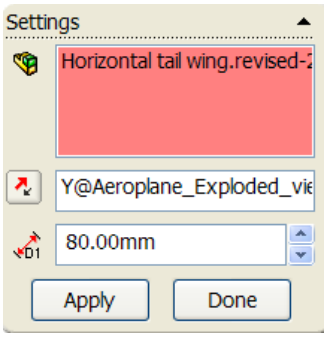


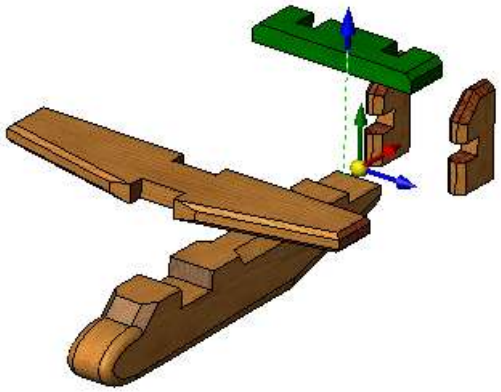


Accept

Explode the Horizontal Tail Wing

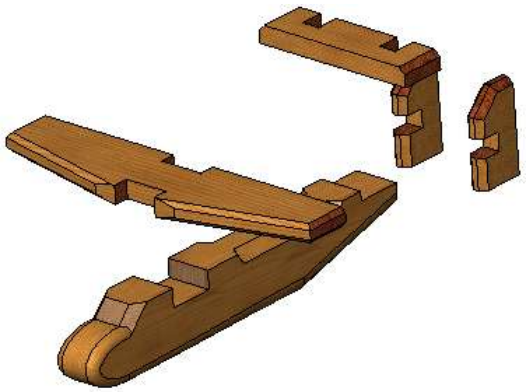
Select **Exploded view** as before.
Select the horizontal tail wing**.**
Drag the **green arrow** away from the assembly.
Insert the distance of 80mm.





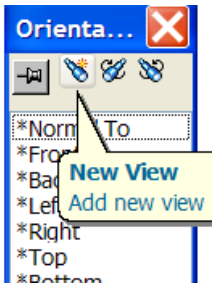
Click Apply and Done.

Accept



Saving the desired Exploded view

Press the space bar to show the **Orientation dialog box.**



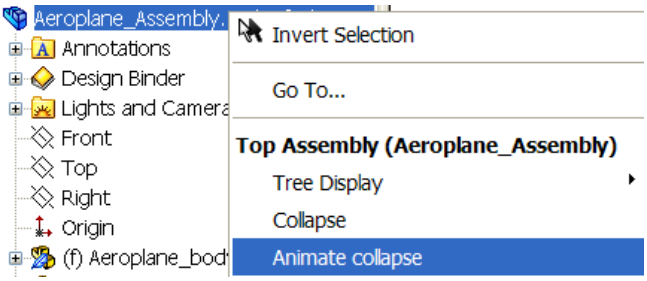
Select New View as shown.

Name it as **"**Front Pictorial"

This view is now added to the list of views.

Animating the Exploded View

The Animation Controller can be used to animate the **explode** or **collapse** motion. Right click on **Aeroplane Assembly** as shown and select **Animation collapse.**

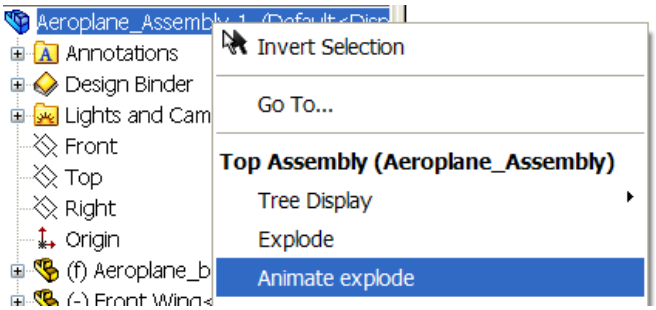


The animation controller display appears.

Select **play** or **loop.**

The Collapse motion will be shown.

To show the **Animation Explode motion,**
Select the animation explode in the same way.



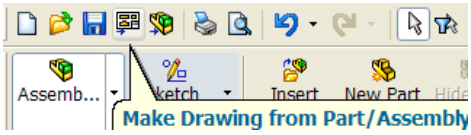
Save

Save **"AEROPLANE ASSEMBLY"** in the **Collapsed View**

Creating the Drawing

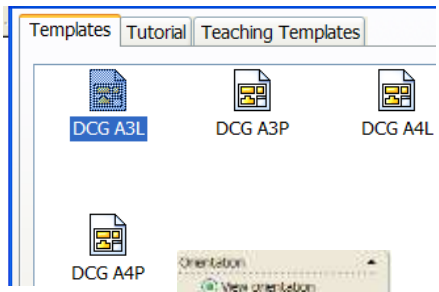
To make drawing from Part/Assembly see the notes on "Creating Drawings" R6 in-service.

Click on **Make Drawing** in the Standard Icon Toolbar as shown.



Choosing a Drawing Template

Select the **A3L** template



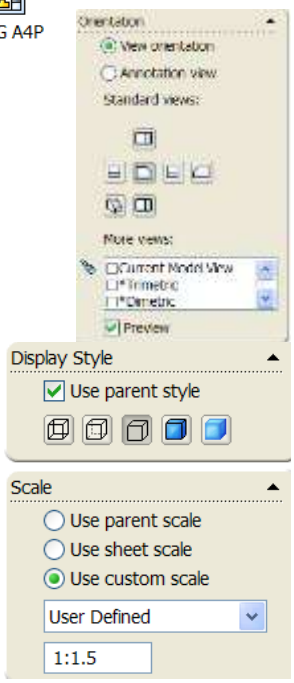
Select OK

Model View

The **Model View** property manager is displayed

In the property manager make sure the front view is selected.

Tick the preview button.



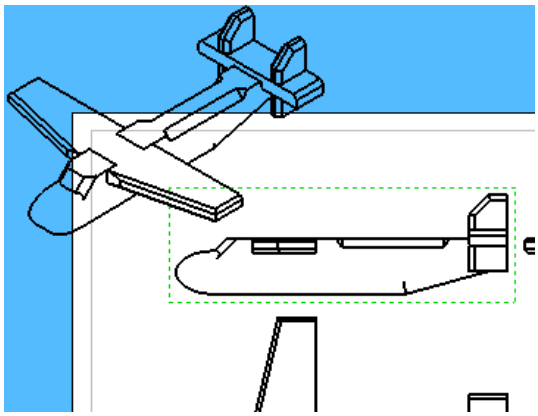
Scale

Set the scale to **1:1.5**.

Ensure that start projected view is ticked so that further views are projected once the first one is positioned.

Positioning the Views

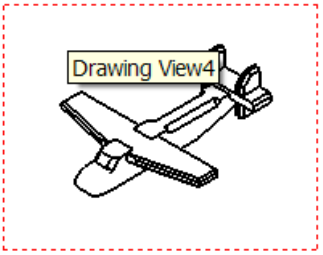
Position the front view on the sheet.
Drag the courser to the right to project an end view.
Drag down to project a plan.

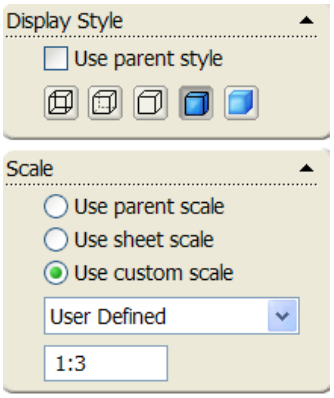


Project an isometric view by dragging the courser to the top corner of the front view shown.

To position this isometric view in the required location hold down the **ctrl** key while dragging.

Change the scale of this view by selecting the box shown and changing the scale to **1:3**

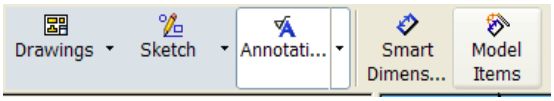


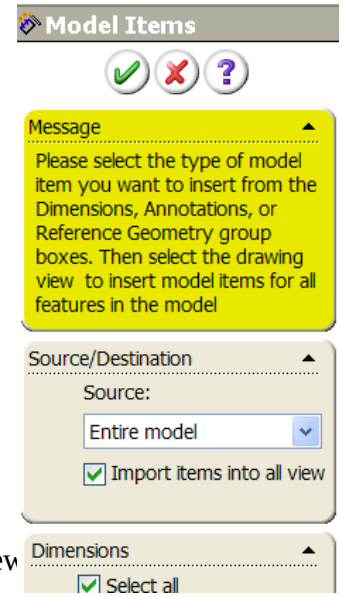


Change its display style to **Shaded with Edges**.

Inserting Dimensions

Select **Model items** from the **Annotations** toolbar.





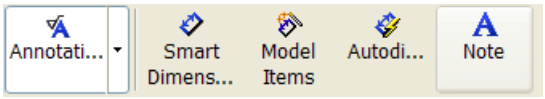
Choose the **entire model** as the source.

Tick **select all** dimensions.

Select **ok** to accept.

Delete some of the dimensions that are not needed.

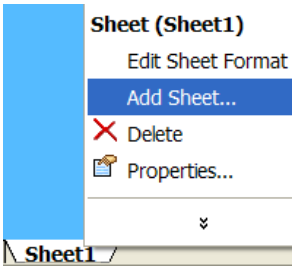
In the **Annotation** toolbar select **Note** and name the view

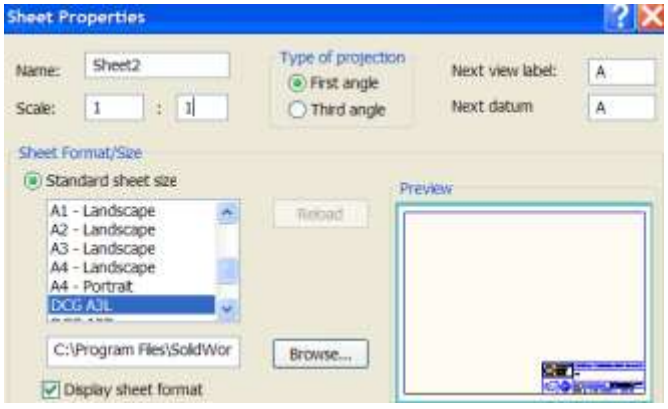


Drawing Exploded View

Adding Sheet 2

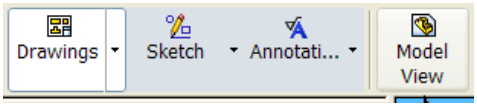
Right click on **Sheet 1 tab** (located at bottom left of graphics area) and select **Add Sheet**





Change the scale to **1:1.**
Accept.
Sheet2 now becomes the current sheet

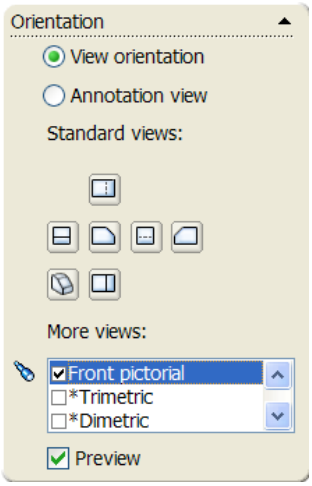
Select the **Drawings** toolbar and **Model views**



In the **Model view** command manager select browse and open **"Aeroplane Assembly".**

The front view is selected by default.

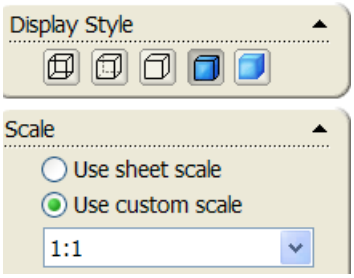
To select the desired view tick "Front Pictorial" as shown.



Tick the preview box also.

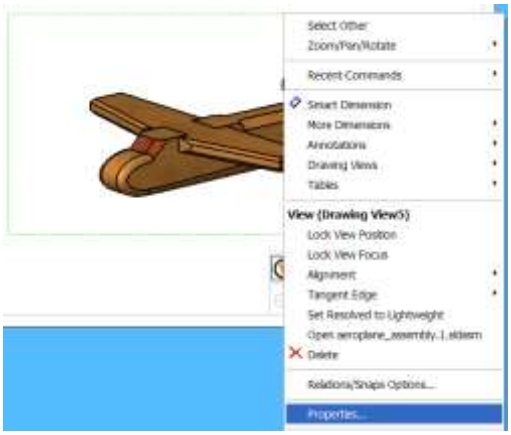
Scroll down the property manager and change the display to shaded with edges.

Change the scale to **1:1**



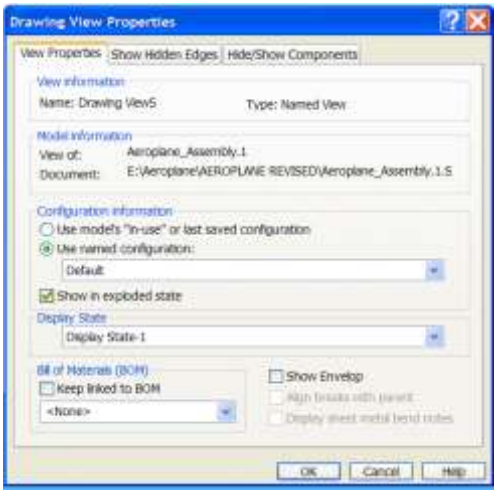
Place the drawing on the sheet.

To show the Exploded view right click on the box and select properties as shown.

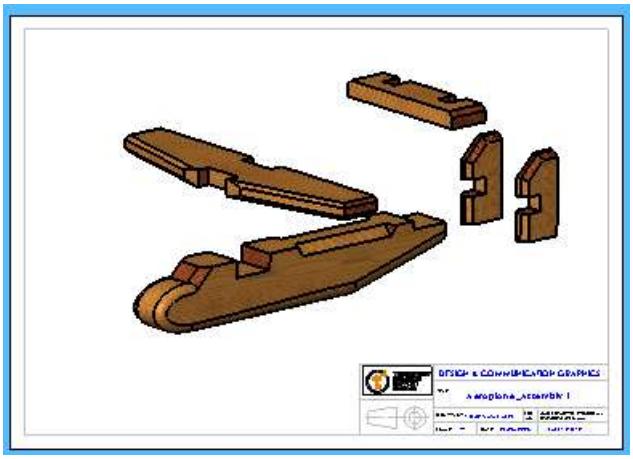


In the Properties dialog box tick the **"show in exploded state box".**

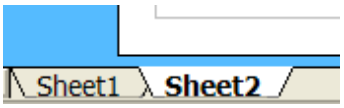
Select ok.

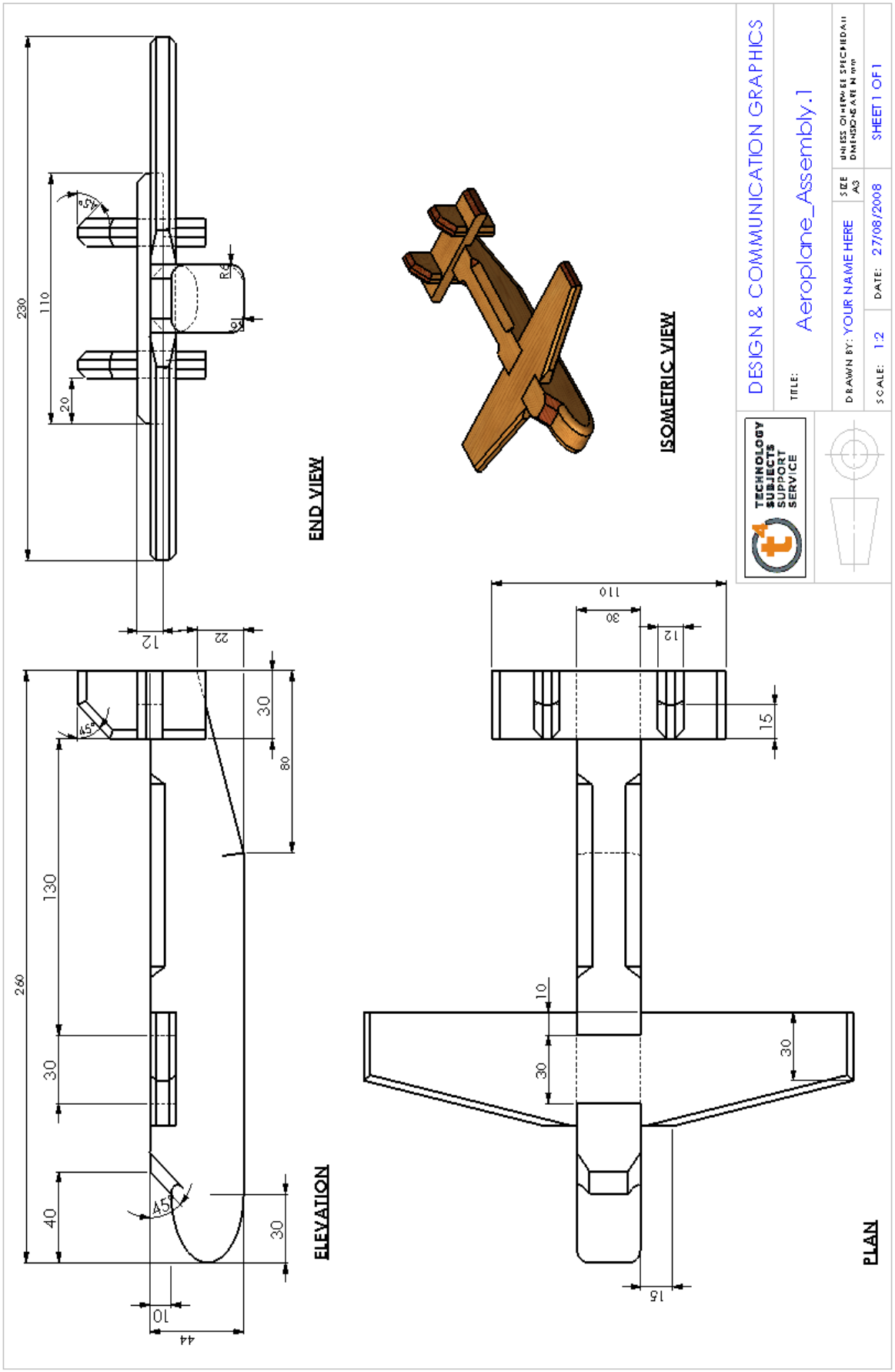


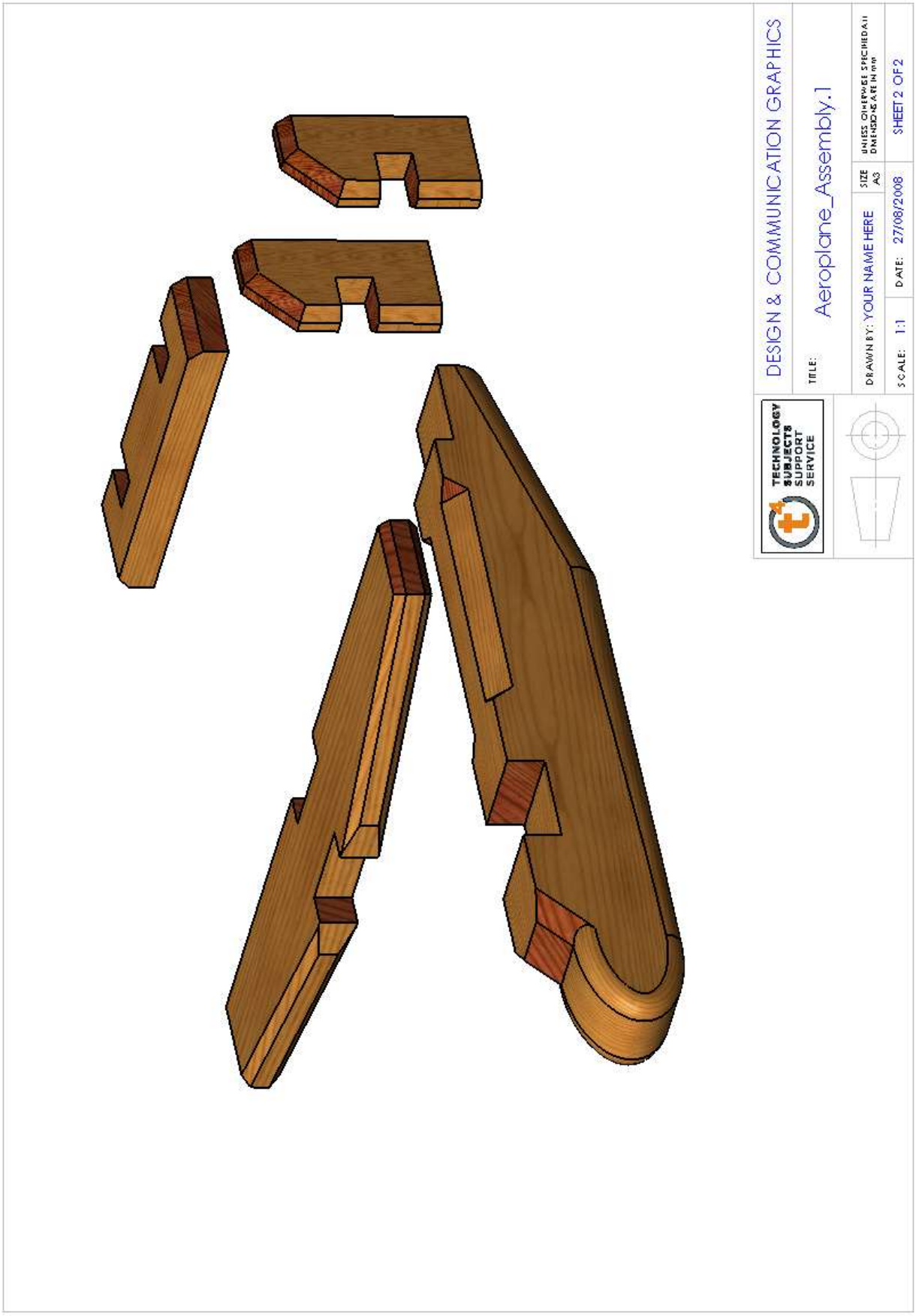
Save .



To switch between sheet 1 and sheet 2 just select sheet1 or sheet2 at the bottom of the drawing area.




DESIGN & COMMUNICATION GRAPHICS
TITLE:
Aeroplane_Assembly.1
DRAWN BY: YOUR NAME HERE
SIZE A3
DATE: 27/08/2008
SCALE: 1:2
SHEET 1 OF 1
TECHNOLOGY SUBJECTS SUPPORT SERVICE
ISOMETRIC VIEW
END VIEW
ELEVATION
PLAN


DESIGN & COMMUNICATION GRAPHICS
TITLE:
Aeroplane_Assembly.1
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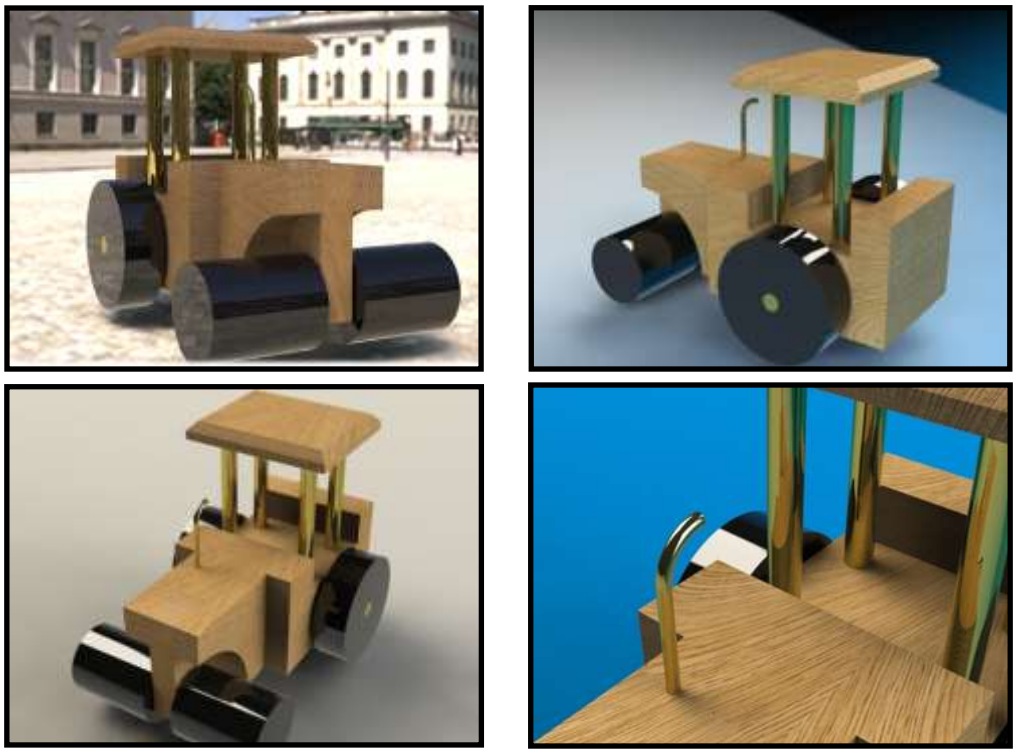
SIZE A3
UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm
SCALE: 1:1
DATE: 27/08/2008
SHEET 2 OF 2
TECHNOLOGY SUBJECTS SUPPORT SERVICE

Photorealistic Images - Photoview 360



Prerequisite knowledge

A basic knowledge of SolidWorks 2009 is required.

Focus of Lesson

This lesson will focus on using PhotoView 360 to create photorealistic images from SolidWorks models.

PhotoView 360

PhotoView 360 is a software package, separate to SolidWorks, which generate digital images (photorealistic images) from SolidWorks files.

PhotoView360 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.**

Getting Started

To launch the program; double click on the PhotoView 360 icon located on the desktop.

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.

1. Open file
(Import SolidWorks File)

2. Apply appearances

3. Apply Environment

4. Edit settings

5. 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 the **Photoview 360** window in the same way as S**olidWorks.**

The **select** icon must be highlighted in order to apply appearances to any aspect of the SolidWorks model.

Open File

Open the assembly named **Steam Roller** located in the folder Steam Roller. The parts and assembly were created in SolidWorks using the default material appearances.

When you open a SolidWorks file in PhotoView 360, any appearances applied in SolidWorks will be displayed. However, there are enhanced appearances available in PhotoView 360 and it is recommended that these are applied to achieve better results.

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.

Applying Appearances

We are going to apply appearances to each individual part of the model

Appearances

A wooden finish is to be applied to the cab of the Steam Roller.
Highlight the **part** icon in the selection toolbar.

Ensure that **select** is highlighted in the navigation toolbar.

Adding Appearance

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

The **Presets** window will appear. Click on the triangle next to the category name to expand or collapse the selection tree.

Navigate to **Organic, wood, oak, polished oak end grain.**

Drag and drop **polished oak endgrain** onto the cab part in the graphics area.

Note: Because **Part** was preselected, the appearance is applied to the entire part not just the face onto which it was dropped.

Apply **polished oak endgrain** to the Steam Roller body and front upright.

Wheels

Navigate to **plastic, high gloss, black high gloss plastic,** and apply the appearance to the front and back wheels by dragging and dropping as before – making sure **part** is still preselected.

Cab Supports Rear Axle

Employing the same procedure apply **Exhaust polished brass** to the cab supports exhaust and rear axle.

Face Appearance

All appearances applied thus far have been applied to entire parts. We can also constrain an appearance to be applied to a particular face of a part. Just as in SolidWorks, a hierarchy exits when applying appearances within PhotoView 360.

A face appearance will override an appearance which has been applied to an entire part.

End Grain

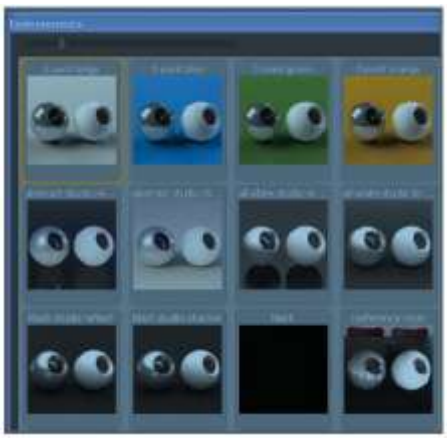
We will apply a different appearance to where the end grain would be visible on the body, cab and front upright of the steam roller.

Pre-select **face** on the selection toolbar.

Navigate to **Organic, wood, oak, finished oak.** Drag and drop **finished oak** onto the faces indicated on the model in the graphics area.

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.



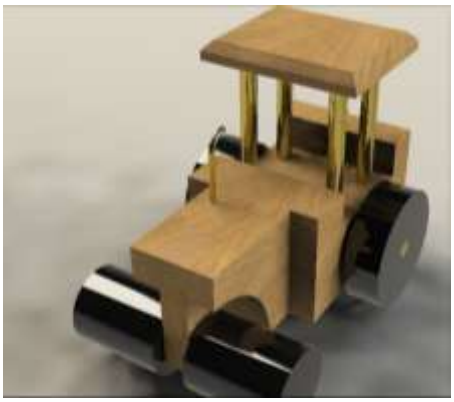
Select **Environments** in the main menu.

The **Environments** selection box appears.

Drag and drop the chosen environment into the graphics area.

Alternatively **double click** on the desired environment.

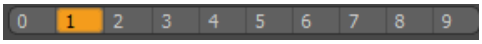
Dragging and dropping an alternative environment will override the previous selection.



Final Render

When you are happy with the selected appearances and environments, we are now ready to start saving jpeg images of the steamroller.

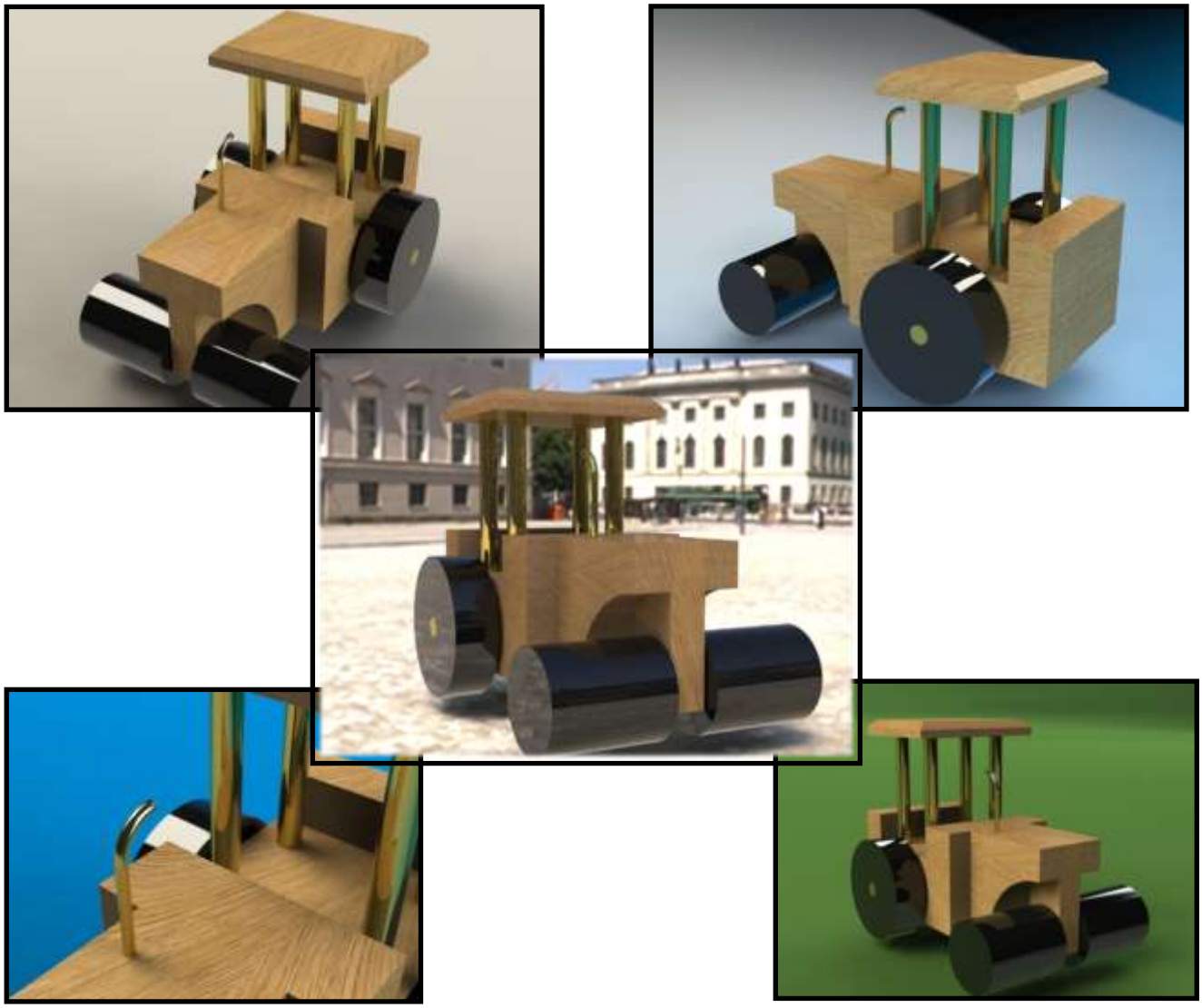
Click on **Final Render** in the main toolbar.



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

Save the image as **Steamroller 1** in the Steam Roller folder as a **JPEG.**

Once rendered, the image will be allocated a number 1 – 9 as indicated above. Selecting that number will allow you to retrieve the image at a later stage.

Add different environments and manipulate the positioning of the model to capture various images of the model.

Zoom and Pan to specific areas of the model to take photorealistic images of different components