SolidWorks 2009 Interface
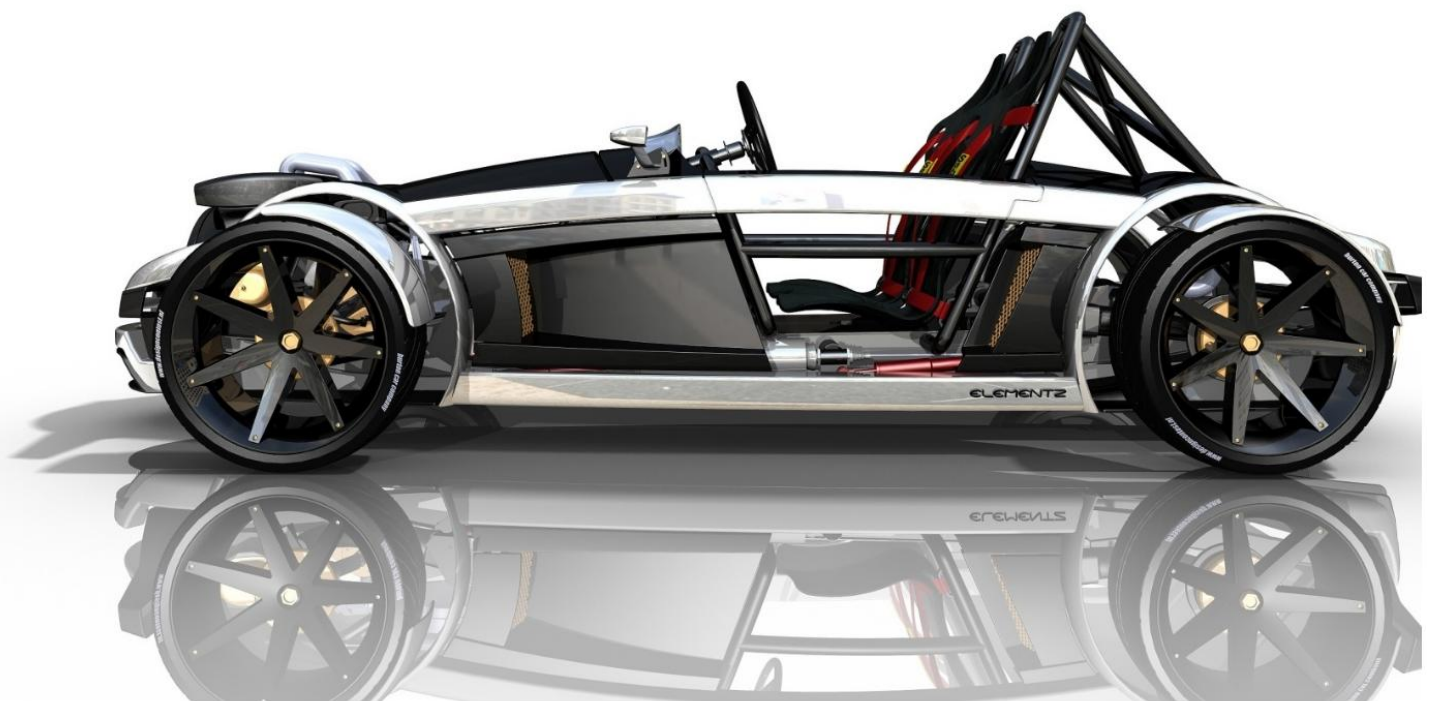
SolidWorks User Interface

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.
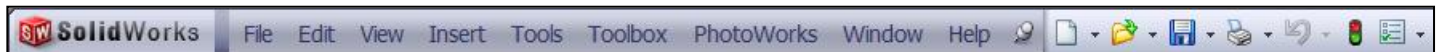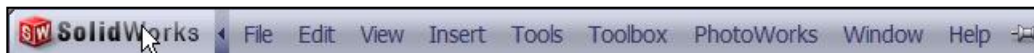
Menu Bar Toolbar
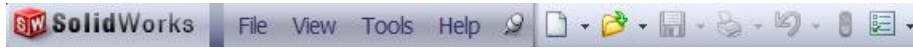
The Menu Bar toolbar contains a set of the most frequently used tool buttons from the Standard toolbar. The available tools are: **New** - Creates a new document, **Open** - Opens an existing document, **Save** - Saves an active document, **Print** - Prints an active document, **Undo** - Reverses the 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.
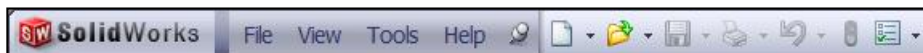
**Drop-down / Pop-up Context Toolbar**

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.
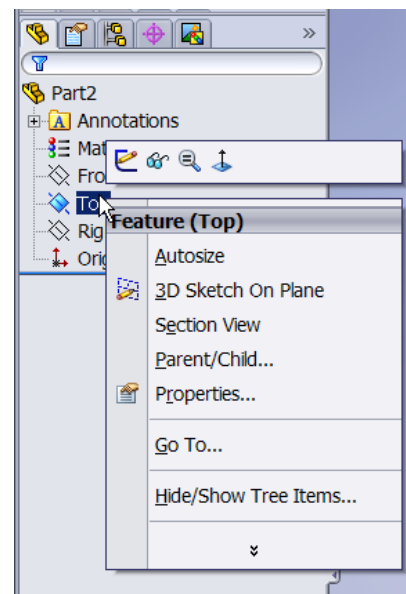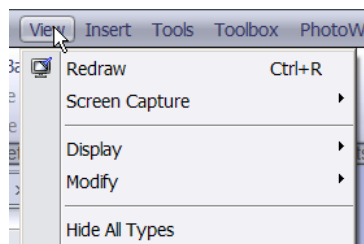
Keyboard Shortcuts

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.
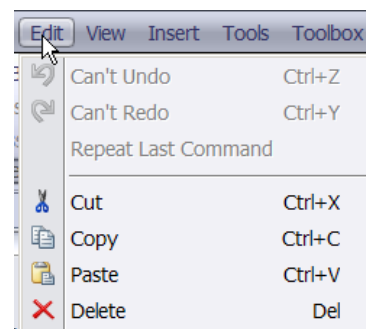
CommandManager

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** ⊕.
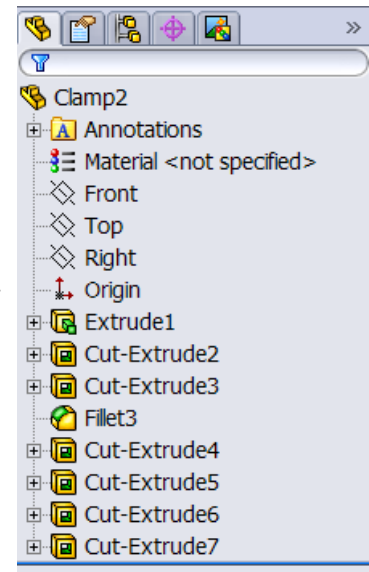
Heads-up View Toolbar

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*.
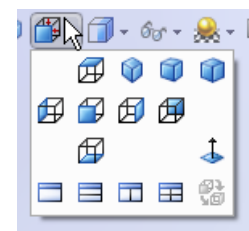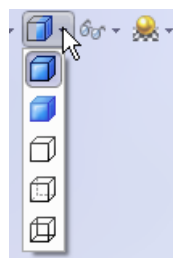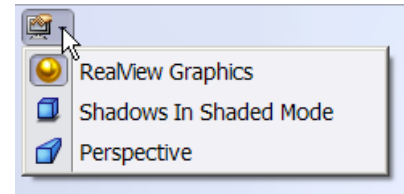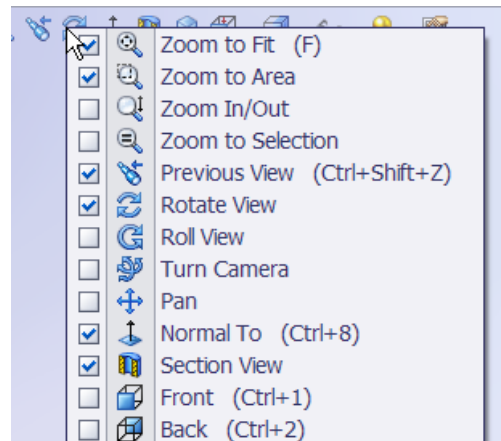
Rotate view 🔁 Rotates a drawing view.

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"
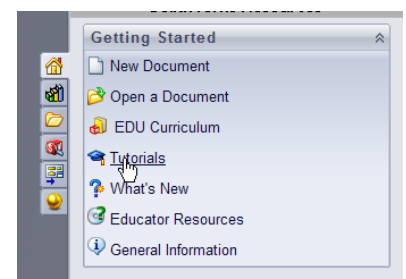
Design Library

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*.
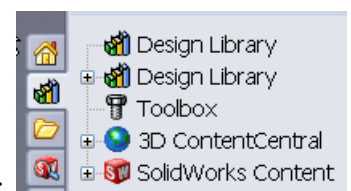
Note: Click **Tools**, **Add-Ins.., SolidWorks Toolbox** and **SolidWorks Toolbox**

Browser to activate the SolidWorks Toolbox.
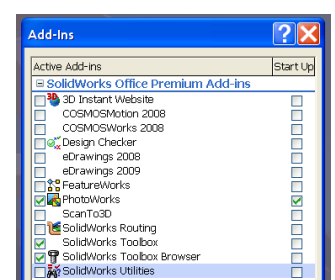
File Explorer

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.
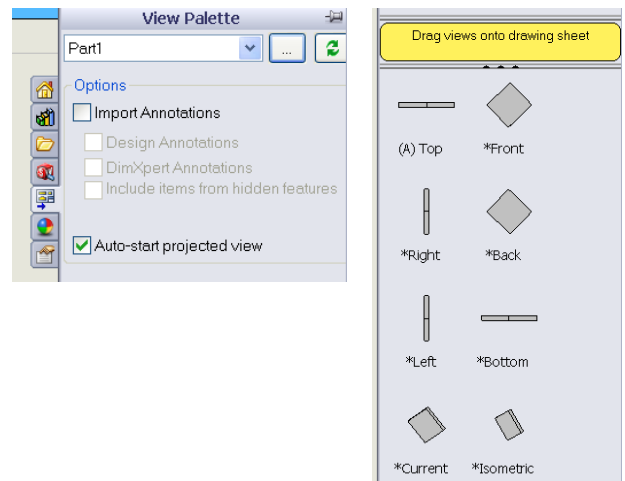
Appearances/Scenes

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.
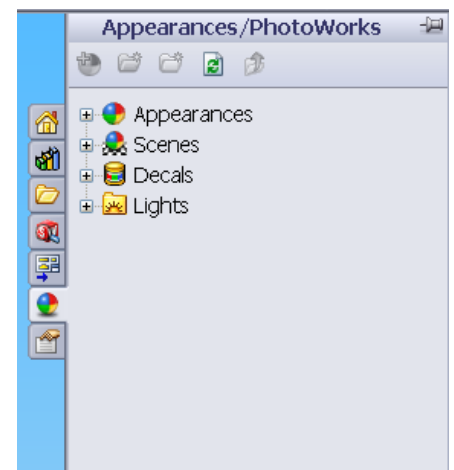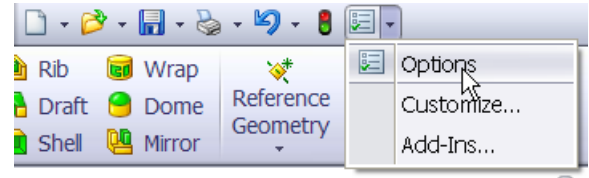
Changing the background colour to white

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.
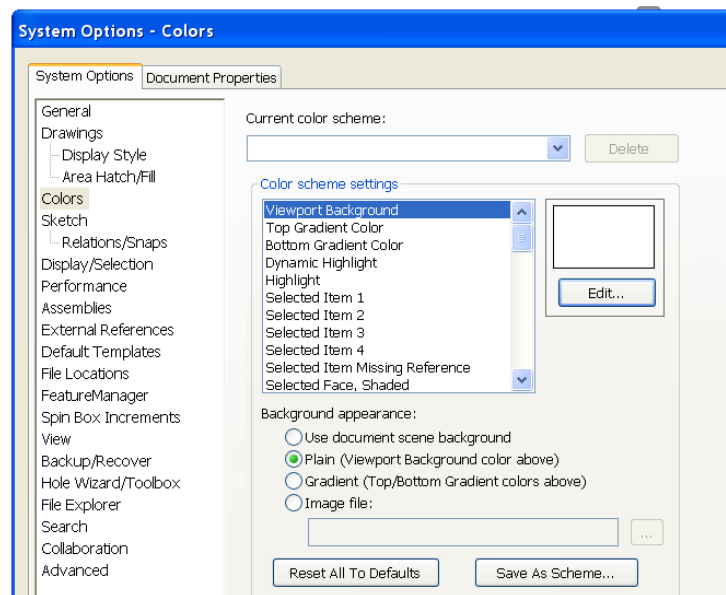
Choose **Options** from the standard toolbar

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

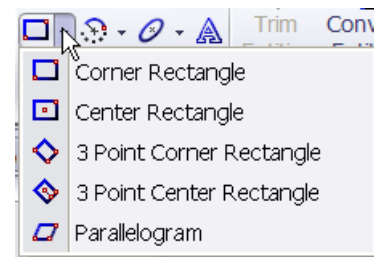Choose **Edit** and select a colour eg. **White**

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.

Face   Edge   Dimension   Vertex
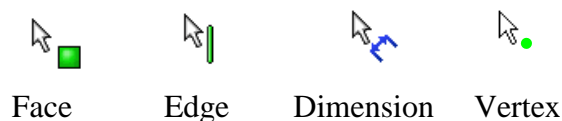
Mouse Buttons

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.
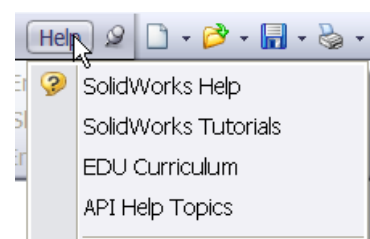
SolidWorks Tutorials

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**.