

Changes from SolidWorks 2003 to SolidWorks 2004

The changes from **SolidWorks 2003** to **SolidWorks 2004** are primarily cosmetic. Consequently, it is quite easy to use the current edition of *Learning SolidWorks* (copyright 2004 edition) with the latest version of **SolidWorks 2004**.

The most prominent change in the look of SolidWorks is that many of the pop-up dialog boxes in **SolidWorks 2003** have been replaced with the PropertyManager in **SolidWorks 2004**. Fortunately, the options within the PropertyManager are very similar to those in the dialog boxes. The look is just a bit different.

This guide is helpful in using *Learning SolidWorks* with **SolidWorks 2004**. All significant changes in SolidWorks that affect the tutorial in the book are listed below. In addition, there are many very minor changes, such as slight differences in toolbar buttons and menu item names. Since these changes are barely noticeable and do not affect the tutorial, we do not include them here.

We recommend that you print out this document and mark the pages in the book that are noted below. When you proceed with the tutorial in *Learning SolidWorks*, you can simply refer to this document when you get to any of the marked pages.

To find out about the latest features in **SolidWorks 2004**, click **Help** ⇒ ⇒ **What's New Manual** or **Help** ⇒ **SolidWorks Help Topics** ⇒ **What's New in SolidWorks 2004**. The document that appears contains information on changes, new features, and working examples that are helpful in transitioning to the new version.

<p><i>SOFTWARE NOTICE</i> – Ensure the latest software update of SolidWorks is installed on your machine for the features in the tutorial to work properly. At a minimum, Service Pack 4 (SP4) should be used.</p>
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IN ADDITION – The expanded **Sketch** toolbar (which now includes **Sketch Relations** and **Sketch Tools**), the **Tools** ⇒ **Options** menu item instead of the **Grid** button, and the updated **Units** settings are used throughout the tutorial, including Paragraphs 2.2.2.4, 2.2.2.5, 2.2.3.1, 2.3.1.2, 2.3.4.1, 2.4.2.1, 3.2.5.5, 4.2.1.2, 6.2.1.4, and 7.1.1.1.

3. When creating a new sketch, the default sketching plane is no longer the **Front** plane. When you click the **Sketch** toolbar button, a message will appear on the left side of the screen requesting a plane on which to create a sketch. Select the **Front** plane in the Graphics Window or in the Feature Manager Design Tree to begin sketching on the **Front** plane.

IN ADDITION – When creating the first feature of a part throughout this tutorial, select the **Front** plane as described above. This includes paragraphs 2.3.1.4 and 2.4.1.1.

2.2.6 Extruding the Cross Section

Page 25

3. Several of the viewing commands and toolbar buttons have been updated. Instead of using **View** ⇒ **Display** ⇒ **HLR Edges In Shaded Mode** to remove the lines on the edges of parts, simply click **Shaded** in the **View** toolbar.

2.2.7 Viewing the Guard

Page 25-26

Several of the viewing commands and toolbar buttons have been updated. Use these features to match the views in this tutorial:

- To show the edges in shaded mode, click **Shaded With Edges**.
- The **View Orientation** toolbar button is now located in the **Standard Views** toolbar.
- Two new views show the model in different angled orientations, **Trimetric** and **Dimetric**, which are located in the **Standard Views** toolbar.
- The **View** toolbar includes a new feature, **Previous View**, which can be useful for returning quickly to the view before the current one.

For convenience, the new **Standard Views** and **View** toolbars are shown below.



Figures 2.24 and 2.25. **Standard Views** and **View** toolbars.

2.3.8 Adding a Hole in the Arm

Page 40

9. In order to save the Part and not the Template, be sure to click **File** ⇒ **Save As** instead of **Save**. Change **Save as type** to **Part**. The **arm** part can then be saved in the same location as the **guard**. Be sure to save the **arm** in the same folder as the **guard**.

2.4.1 Sketching the Blade

Page 42

1. When creating a new part from a template, it is necessary to click **Advanced** in the **New SolidWorks Document** dialog box. After clicking **Advanced**, the **tutorial part** template you created in the previous section should be available.

2.4.5 Changing the Definition of the Extrusion

Page 46

- 1-3. Use **Edit Feature** instead of **Edit Definition** in order to update the definition of an existing feature.

IN ADDITION – Use **Edit Feature** in Paragraph 6.3.4.7 as described above.

3.2.3 Sketching and Cutting a Single Groove

Page 58

11. Tangent edge visibility is no longer set in the **View⇒Display** menu. In order to show tangent edges in the model, click **Tools⇒Options**. In the **System Options** tab of the dialog box, click **Display/Selection** and choose the appropriate setting for viewing tangent edges under **Part/Assembly tangent edge display**.

IN ADDITION – Tangent edge visibility is set throughout the tutorial, including paragraphs 4.2.6.3, 4.2.6.3, 6.3.2.1, and 6.4.1.1.

3. Creating a *Reference Axis* is now done in the PropertyManager, which is shown below. The **Front** and **Right** planes should appear in the *Reference Entities* list of the PropertyManager and *Two Planes* should be highlighted. Click **OK** to create the axis.

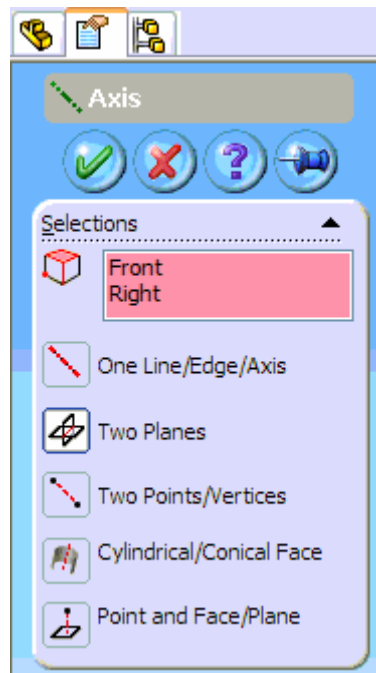


Figure 3.18. *Reference Axis* PropertyManager.

2. Controlling the color of a part, face, or feature is done in the ***Color and Optics*** PropertyManager, which is shown below.
3. In order to change the color of the entire handle (and not a face or feature), it is necessary to select the part by clicking the topmost item in the flyout Feature Manager Design Tree.

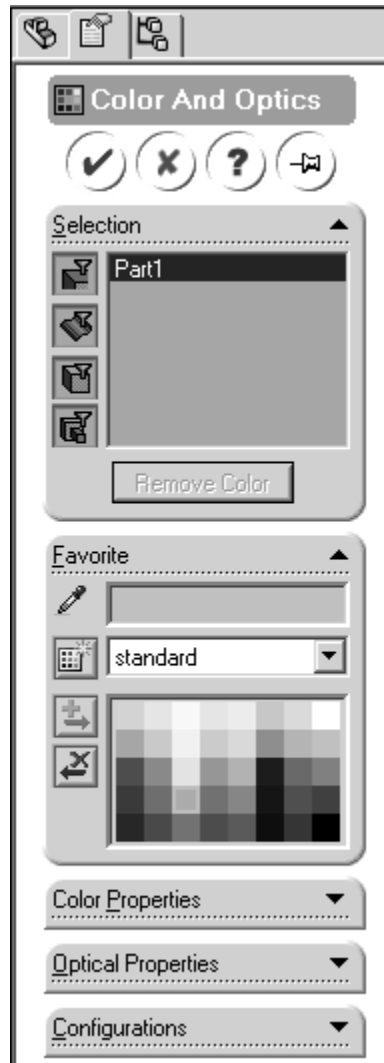


Figure 3.29. *Color and Optics* PropertyManager.

3.3.4 Reopening the Sketch and Adding Dimensions

Page 74

10. Use the Confirmation Corner to exit the sketch. In the top right of the Graphics Window, click the green check mark to accept the changes.

IN ADDITION – Using the Confirmation Corner to exit a sketch is done in paragraph 6.4.1.7.

4.1.1 Creating a New Assembly Document

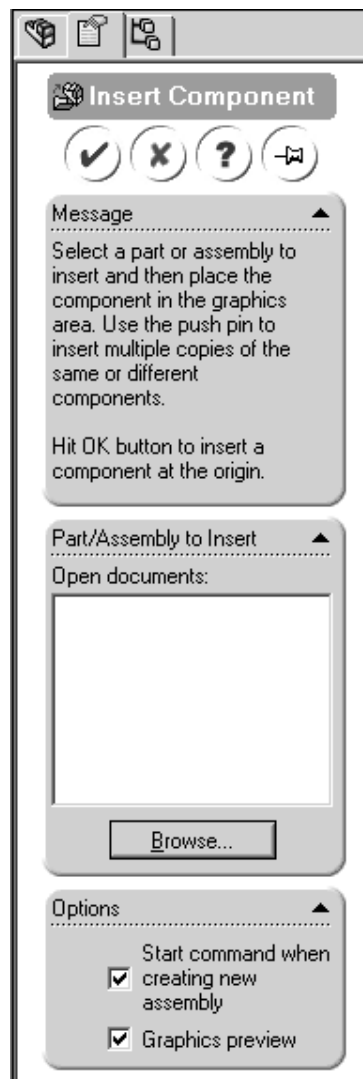
Pages 77

3. The FeatureManager design tree will not appear until section 4.1.2

4.1.2 Bringing the Rivet into the Sub-Assembly

Pages 78-79

1. Parts are brought into assemblies in a slightly different manner. Instead of using **Insert⇒Component⇒From File**, the PropertyManager displays a message as shown below when a new assembly is opened. If this message does not appear automatically, the menu item for assembling components is **Insert⇒Component⇒Existing Part/Assembly**.
2. Click **Browse** to select a saved part or subassembly, in this case *rivet*.
3. Instead of moving the cursor over the origin as described in the tutorial, simply click **OK** (the green check mark) in the PropertyManager to assemble the part so its origin is aligned with the assembly origin.



PropertyManager message.

4.1.3 Bringing the Arm into the Assembly and Orienting It

Page 80

1. The menu item for assembling components is **Insert⇒Component⇒Existing Part/Assembly**. Use **Browse** in the PropertyManager to find the part file.
4. Moving and rotating components can no longer be done using the **Tools⇒Component** menu. Components can still be oriented by using the buttons in the **Assembly** toolbar, as described in the book, or by simply dragging a component with the left mouse button to move it. Drag with the right mouse button to rotate a component.

4.2.1 Creating a New Assembly and Inserting the Handle

Page 86

1. After creating a new assembly, the cutter sub-assembly may automatically appear for assembly. Since the handle should be assembled first, click the red 'X' in the corner to cancel the insertion. Continue with the assembly by inserting the handle part.

4.2.4 Hiding an Object

Page 89

2. Select **Hide** instead of **Hide Components** to follow along with the tutorial.

<p><i>IN ADDITION</i> – The commands Hide and Show are used throughout the tutorial, including paragraphs 4.2.6.1, 4.3.5.2, 6.2.4.4, 7.1.3.4, and 7.1.3.9.</p>
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4.3.1 Checking for Interference Volumes

Pages 93-94

1. The **Interference Detection** control is now located in the PropertyManager, as shown below. Click **OK** to begin the analysis.

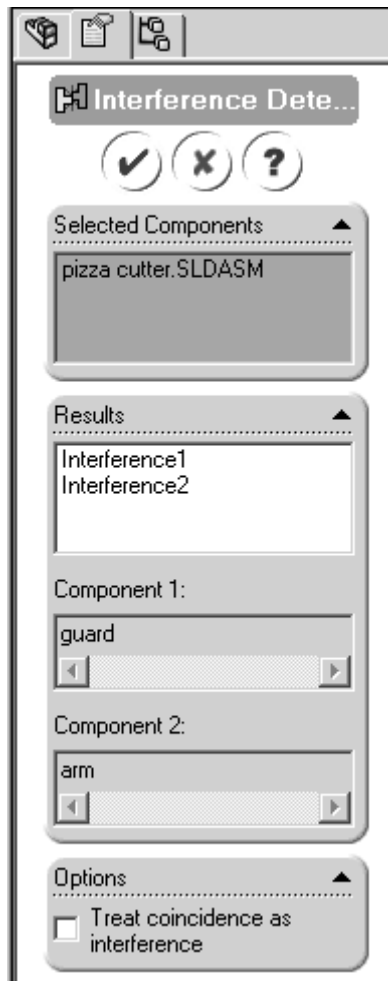


Figure 4.25. **Interference Detection** PropertyManager.

5.2.2 Checking the Options Settings

Page 107

1. To match the settings used in the tutorial, select the **Display Style** item on the left side of the **System Options** tab. Under **Display style for new views**, select **Hidden Lines Visible**. Under **Tangent Edges in new views**, select **Visible**.
3. The options for Notes and Balloon can now be set separately. Set the both **Note** and **Balloon** list items in the **Document Properties** tab to match what is shown in the tutorial.

5.2.5 Modifying the Lines of the Sheet Format

Page 112

1. There is no longer an **Edit⇒Sheet Format** command. To edit (or complete editing) a sheet format, *right-click* on an open section of the drawing and select **Edit Sheet Format** (or **Edit Sheet** to complete the editing). To see whether or not you are editing the format, the Status Bar at the lower right hand of the screen displays the current mode.

IN ADDITION – Editing the format or sheet through the right mouse button is also used in paragraph 5.3.1.2.

5. The *Symbol* dialog box no longer requires clicking the *Use Symbol* button for placing symbols.

There is no *Custom Sheet Format* button when saving the format. SolidWorks will automatically save it as a custom format.

IN ADDITION – Selecting a custom sheet format is similar to loading it, which is used in paragraph 5.4.1.2.

5.3.2 Adding a Named View

Page 116

1. The **Named View** toolbar button has been removed. To insert a view (that was created in the model), click **Model View** in the **Drawing Toolbar**. The properties of the view can be updated in the PropertyManager as described in the tutorial.

5.4.2 Adding Orthographic and Isometric Views to the Drawing

Page 127

3. If the view import in a 1:2 scale instead of the desired 1:4 as shown in the Status Bar, select each view and set the appropriate scale in the PropertyManager.

2. A Bill of Materials can be inserted by clicking **Insert**⇒**Tables**⇒**Bill of Materials**. The **Bill of Materials** PropertyManager appears, as shown below. Set the appropriate properties to match the tutorial.

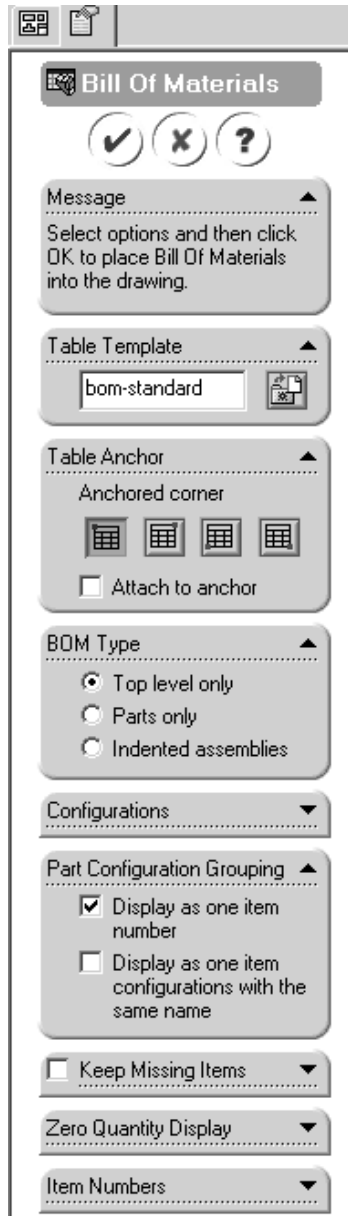


Figure 5.31. *Bill of Materials* PropertyManager.

- 5-7. A Bill of Materials is now edited directly in SolidWorks, without the use of Excel. The commands and functionality are the same.

6.5.3 Creating a Coincident Mate Using SmartMates

Page 171-174

SmartMates, a simple and quick way to mate components in assemblies, has been integrated into the general method of creating mates with the use of keyboard shortcuts. In the tutorial, instead of explicitly using the SmartMates feature, do the following in order to mate components quickly:

- Hold down the ALT key.
- Click and drag the feature that you would like to mate on one component.
- Drag the component to the feature of the mating component.
- Let go of the ALT key.
- Click **OK** in the flyout dialog box.

7.1.1 Setting Up the Document and Customizing the Interface

Page 180

3. The **Polygon** toolbar button is no longer located in the **Sketch** toolbar. If you would like to practice adding and removing buttons, the **Move or Copy Entities** button can be removed and replaced using the steps described in the tutorial. Even though there is no **Polygon** button, be sure to add the **Ellipse** button.

7.1.7 Creating the Loft Feature

Pages 189-191

<p><i>SOFTWARE NOTICE</i> – Ensure the latest software update of SolidWorks is installed on your machine for the loft feature to work properly. At a minimum, Service Pack 4 (SP4) should be used.</p>
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2. The **Dome** feature is now located in the PropertyManager, as shown below. Follow the instructions in the tutorial to complete the dome feature.

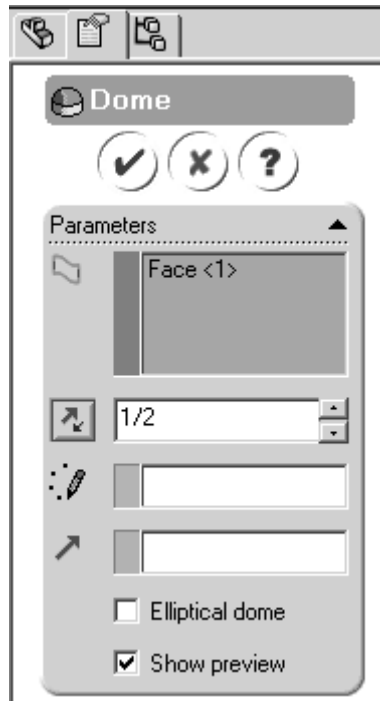


Figure 7.23. *Dome* PropertyManager.

SOFTWARE NOTICE – Ensure the latest software update of SolidWorks is installed on your machine for the radiate surface feature to work properly. At a minimum, Service Pack 4 (SP4) should be used.

5. **Edge<1>** through **Edge<10>** (instead of **Edge<6>**) may appear in the **Edges to Radiate** field. Continue to create the radiate feature as described in the tutorial.

7. Instead of locating the text point **X Coordinate** to **7/8**, use **3/4** to better match the images shown in the tutorial.

- 2-3. There is no **Apply** button in the new **Color and Optics** PropertyManager. Click the green check mark to accept the color. Then redo the process for changing the color of the handle part.

7.4.1 Setting up the Assembly Document

Page 208

- 1-3. The lofted handle part will automatically appear for placement in the assembly. To locate the handle so that it is aligned with the origins of the assembly, simply click **OK** (the green check mark). There is no need to show or hide the origins.

7.4.2 Mating the Cutter Sub-Assembly with the Handle

Page 210

6. If tangent edges are not visible, they can be activated using **Tools⇒Options**. Instead, a quicker method is to activate **Shaded With Edges** in the **View** toolbar to show tangent edges while the part is shaded.

7.4.3 Modeling the Rectangular Hole Using the Cavity Feature

Page 211-212

2. To edit a part within an assembly, click the **Edit Component** in the **Assembly** toolbar while the part is selected or *Right-click* the part and select **Edit Part**.
3. To begin the **Cavity** feature, click **Insert**⇒**Mold**⇒**Cavity**.
4. The **Cavity** feature is now located in the PropertyManager, as shown below.

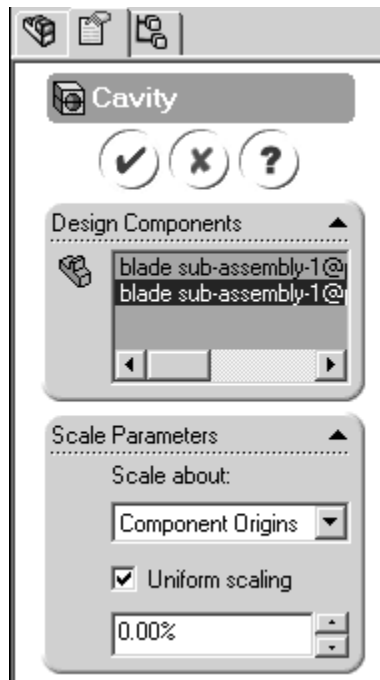


Figure 7.46. *Cavity* PropertyManager.