

# Changes from SolidWorks 2003 to SolidWorks 2005

The changes from **SolidWorks 2003** to **SolidWorks 2005** 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 2005**.

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 2005**. 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 2005**. 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 2005**, click **Help** ⇒ **SolidWorks Help Topics** ⇒ **What's New**. The document that appears contains information on changes, new features, and working examples that are helpful in transitioning to the new version.

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| <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.</p> |
|---|



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 2.4.1.2.

## 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 use **View** ⇒ **Display** ⇒ **Shaded** or 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 **Tutorial** in the **New SolidWorks Document** dialog box. After clicking **Tutorial**, the **tutorial part** template you created in the previous section should be available.

### 2.4.2 Extruding the Sketch

Page 43

1. Instead of clicking on the **Grid** button, turn off the sketching grid by clicking on the **Sketch** button which is labeled **Exit Sketch** when the cursor is moved over it.
2. After clicking the **Extruded Base/Boss** button, click on either circle.

### 2.4.3 Adding a Chamfer to Form the Edge of the Blade

Page 44

4. **Distance 1** is labeled **D1** and **Distance 2** is labeled **D2**.

### 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.

### 3.2.1 Sketching the Base Feature of the Handle

Page 53

6. To use the **Trim** tool, follow the instructions in the yellow message box of the PropertyManager. Hold down the left mouse button and drag the cursor across the two line segments to be trimmed away.

### 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**. Under **Part/Assembly tangent edge display** choose **As visible** for viewing tangent edges.

**IN ADDITION** – Tangent edge visibility is set throughout the tutorial, including paragraphs 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.

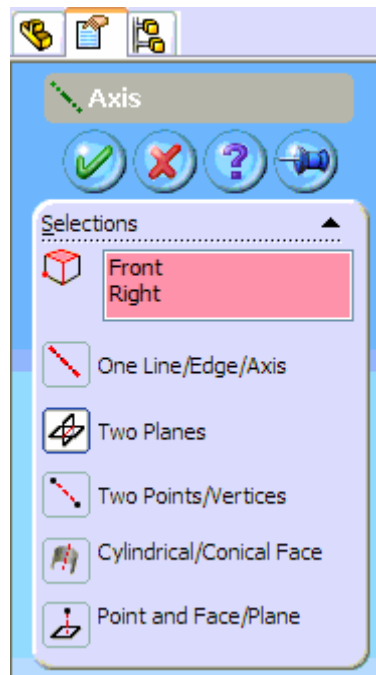


Figure 3.18. *Reference Axis* PropertyManager.

### 3.2.8 Changing the Color of the Handle

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.

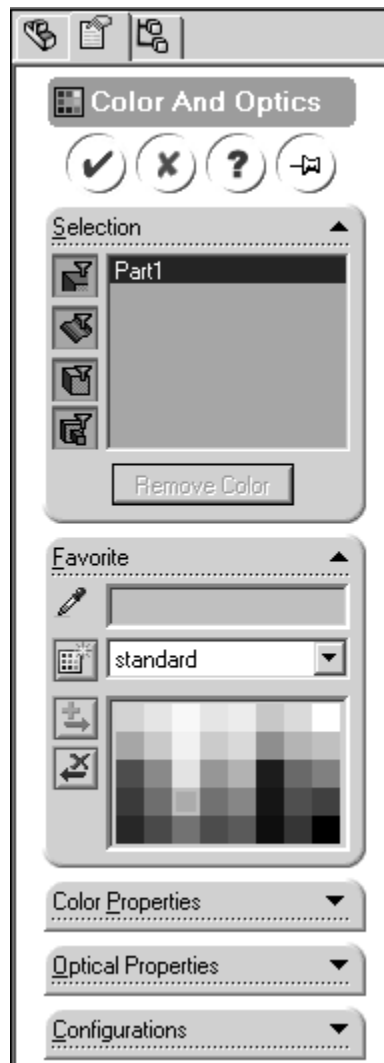


Figure 3.29. *Color and Optics* PropertyManager.

### 3.3.4 Reopening the Sketch and Adding Dimensions

Pages 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

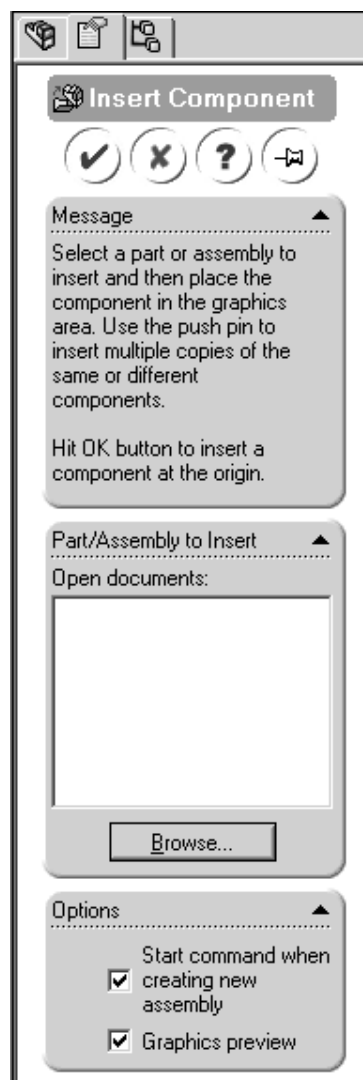
Page 77

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.

### 4.1.4 Adding a Concentric Mate

Page 82

5. SolidWorks automatically adjusts the parts to the position indicated in the PropertyManager without using **Preview**. In addition, the **Anti-Aligned** button is used to flip the arm, if necessary. After clicking **OK**, close the PropertyManager.

### 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.
2. If **Insert Component** appears in the PropertyManager, cancel it by clicking the red 'X.' Rather than using the **Grid** button, use **Tools ⇒ Options ⇒ Document Properties** instead. Click **Units** on the left side of the dialog box. Then set the **Unit system** to **Custom**, and set the **Length units** to **inches**

### 4.2.4 Hiding an Object

Page 89

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

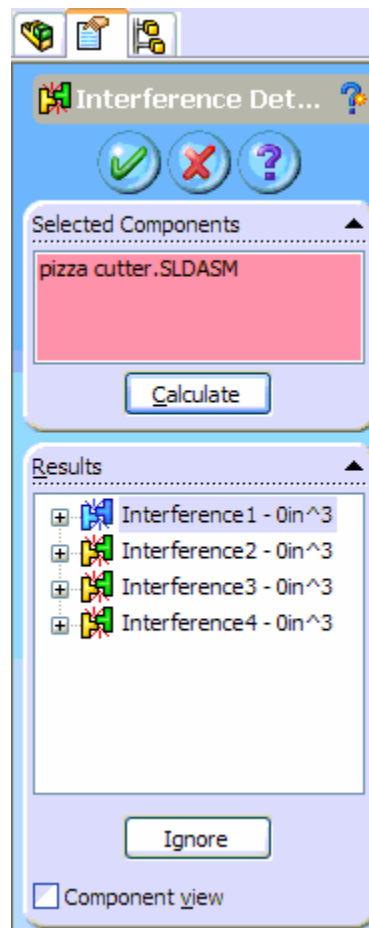
|  |
|--|
| <p><i>IN ADDITION</i> – The commands <b>Hide</b> and <b>Show</b> are used throughout the tutorial, including paragraphs 4.2.6.1, 4.3.5.2, 6.2.4.4, 7.1.3.4, and 7.1.3.9.</p> |
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### 4.3.1 Checking for Interference Volumes

Pages 93-94

1. The **Interference Detection** control is now located in the PropertyManager, as shown below.
3. Click **Calculate** to begin the analysis. Four interference results will appear. The interfering components are listed in the PropertyManager by clicking on the + sign next to each interference. Click **OK** to close the **Interference Detection** PropertyManager.

*IN ADDITION* – Use **Calculate** in paragraph 4.3.5.3.



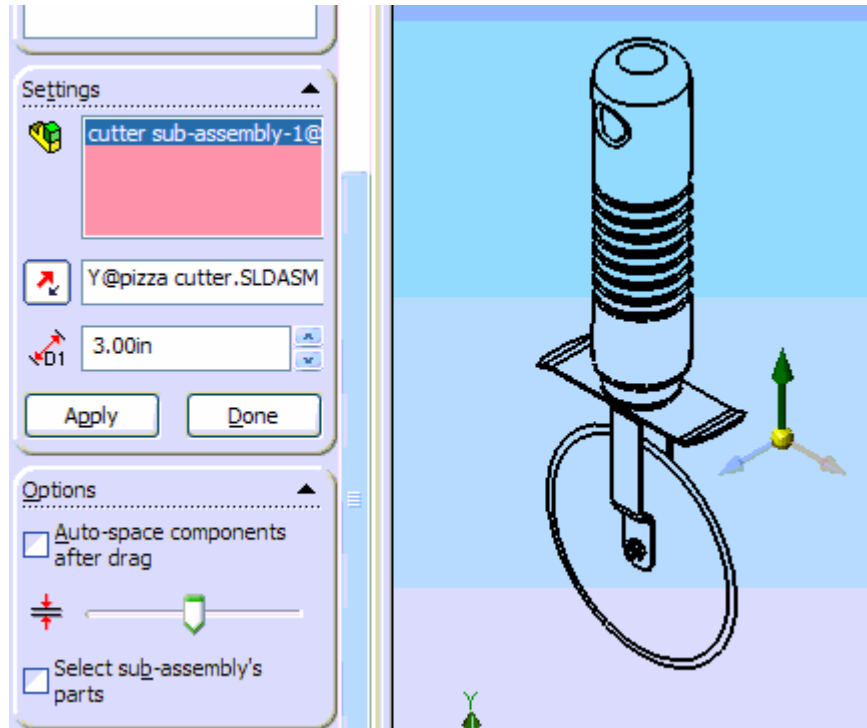
**Figure 4.25.** *Interference Detection* PropertyManager.

### 4.3.2 Opening a Sketch Within the Guard

Page 94

1. Select **Open Part** from the menu that appears after the *right click*.

2. The **Assembly Exploder** is now located in the PropertyManager. There is no need to click on the **New** button to see the full PropertyManager. Use the steps below instead of steps 3-5.
3. Scroll down to **Options** in the PropertyManager, if it is not visible. Unselect **Select sub-assembly's parts** so that the entire cutter sub-assembly will be selected, rather than individual parts of the sub-assembly. Use the **Isometric** view of the assembly. Click the blade of the cutter sub-assembly. A set of arrows appears in the Graphics Window and **cutter sub-assembly-1** appears in the **Settings** field, which is highlighted in pink as shown below.



**Figure 4.33.** Cutter explode direction.

4. Click on the head of the vertical arrow to indicate the direction for the explode.
5. In the **Explode distance** field, enter **3**. Click **Apply** under **Settings**. If the cutter sub-assembly explodes in the wrong direction, click the **Reverse direction** button under **Settings**. Click **Done** under **Settings**. **Explode Step 1** and **cutter sub-assembly-1** appear under **Explode Steps** in the upper part of the PropertyManager indicating that this is the first step of the exploded view.

## 4.4.2 Exploding the Arm

Page 101

Use steps 1-4 below instead of those in the book.

1. Under **Options** near the bottom of the PropertyManager, click on **Select sub-assembly's parts** so the checkbox is checked. This allows individual parts of the cutter sub-assembly to be exploded. Be sure the **Isometric** view is used.
2. To have the arms explode outward away from the blade, click on the left arm in the Graphics Window. The three arrows should appear.
3. Select the blue arrow (in the z-direction), so that the left arm will explode along the axis of the rivet.
4. Set the **Distance** under **Settings** to **2** and click **Apply** in the **Settings** portion of the PropertyManager. If the arm moves to the direction opposite that shown in Fig. 4.36, click the **Reverse direction** button under **Settings**. Click **Done** to finish this step of the explode.

## 4.4.3 Exploding the Other Components

Pages 101-102

It is not necessary to use the **New** button to start an explosion step. Just click on the part. Use the **Undo** button at the top of the **Explode** PropertyManager if you make a mistake. Continue with four additional explode steps using the procedure above.

1. Explode the other arm away from the blade using a **Distance** of **2** inches. Be sure that **Select sub-assembly's parts** is checked, so that individual parts are selected.
2. Explode the rivet **4** inches to the side of the blade toward the head of the rivet. It may be necessary to **Zoom to Area** followed by **Zoom to Fit** to see the rivet head.
5. Each step can be modified by right-clicking on the step and choosing **Edit Step**. Changes are made under **Settings** in the PropertyManager. Click **Done** when you are satisfied with the results for each step. When you are satisfied with the entire exploded view, click **OK** (the green check) at the top of the PropertyManager.
6. Click the ConfigurationManager tab at the top of the PropertyManager window and follow the directions in the book. To change the parameters of the exploded view use **Edit Feature**.

## 5.2.1 Creating a New Drawing Document

Page 106

3. The drawing opens as a standard A size sheet format. If a different drawing size is necessary, the size can be changed by *right clicking* **Sheet Format1** in the PropertyManager and selecting **Properties**. The sheet size is changed in the **Sheet Properties** dialog box.
5. In the **View ⇒ Toolbars** menu be sure that **Annotation, Drawing, Line Format, Sketch, Standard,** and **View** are checked.

## 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.

## 5.2.4 Modifying the Format's Text

Page 111

10. Change the font size to **18** using the **Formatting** dialog box that appears before typing in **Arm**.

## 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. To continue, the mode should be Editing Sheet Format.

**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.
7. There is no longer a Revisions block, so it need not be deleted.

There is no **Custom Sheet Format** button when saving the format. SolidWorks will automatically save it as a custom format. Use the **Save in:** box to locate the folder in which you want to save the format.

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

### 5.3.1 Placing Orthographic Views

Page 114-115

2. To edit the sheet, *right click* in the Graphics Window and select **Edit Sheet**.
- 3-4. The PropertyManager now displays parts that can be used for the drawing. Since the **arm** window is open, **arm** will appear in the PropertyManager. Click **OK** to insert the **arm**.
5. After clicking on the arm, select **View⇒Display** to be sure that **Tangent Edges Visible** is selected.

**IN ADDITION** – Changing the tangent edges using **View⇒Display** is also necessary in paragraph 5.3.3.4.

6. The views in the drawing should be like Fig. 5.13 in size. If the scale is different, click on the arm, if you have not done so already, to display **Drawing View** in the PropertyManager. Under **Scale**, click **Use custom scale** and select **1:1**. It may be necessary to do this for all of the views. Click **OK** to close the PropertyManager.

### 5.3.2 Adding a Named View

Page 116

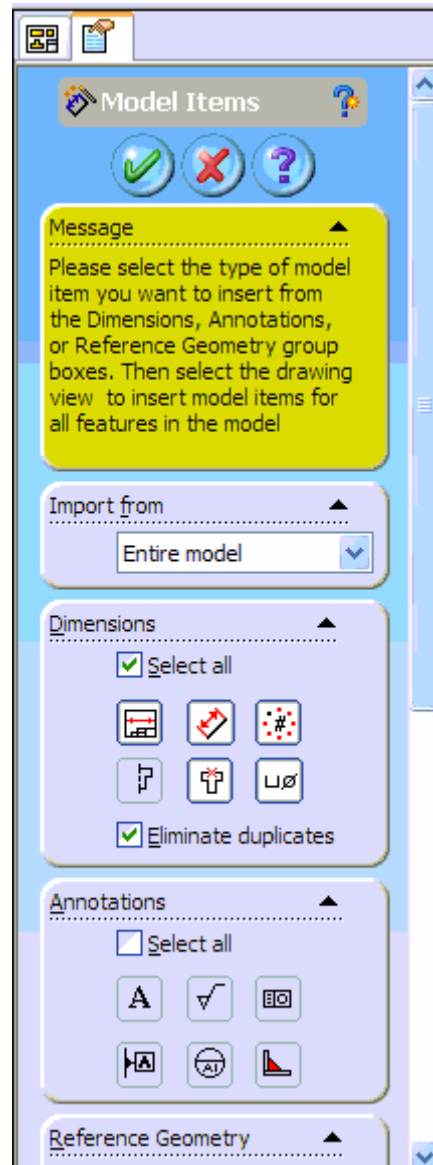
1. The **Named View** toolbar button has been removed. To insert a view (that was created in the model), click **Model View** in the **Drawing Toolbar**. The properties of the view can be updated in the PropertyManager as described in the tutorial.
2. Rather than clicking on view of the arm in the Graphics Window, use the PropertyManager, which should display **arm**, since the arm is open in another window. Click the **Next** button (a right arrow) and be sure that **\*Isometric** is highlighted.
3. Click **OK** after placing the view.

### 5.3.3 Adjusting the Views on the Sheet

Page 117

5. In the **Drawing View** PropertyManager that opens upon selecting the isometric view, activate **Use custom scale** under **Scale**. In the pull-down menu, select **User Defined** and type in **3:2** just below.

1. The **Model Items** button may not be available so select **Insert⇒Model Items**. The *Insert Model Items* dialog box has been replaced by the *Model Items* PropertyManager, shown below. Under *Import from*, select *Entire model*. Under *Dimensions* be sure *Select all* and *Eliminate duplicates* are checked. Then click *OK*.



**Figure 5.17** *Model Items* PropertyManager.

8. Instead of *right clicking* on the **.048** dimension, just select the dimension. To add parentheses, check *Parentheses* under *Display Options* in the PropertyManager followed by *OK*.

### 5.3.6 Specifying Tolerances

Page 123

1. In the **Dimension** PropertyManager, change the number of digits after the decimal by selecting **.12** in the **Primary Unit Precision** pull-down menu under **Tolerance/Precision**.

### 5.4.1 Setting Up the Pizza Cutter Drawing

Page 125

2. Instead of right clicking on the drawing, *right click **Sheet Format1*** in the FeatureManager and then select **Properties**. Be sure **Third angle** is selected. Use **Browse** to navigate to *tutorial format* that you saved when doing the arm drawing and **Open** the format.
8. Use **Display Style** instead of **Default Display Type** to set **Display style for new views** to **Hidden lines removed**.
10. **Sketch Relations** and **Sketch Tools** are no longer in the **View⇒Toolbars** menu and need not be checked.

### 5.4.2 Adding Orthographic and Isometric Views to the Drawing

Page 127

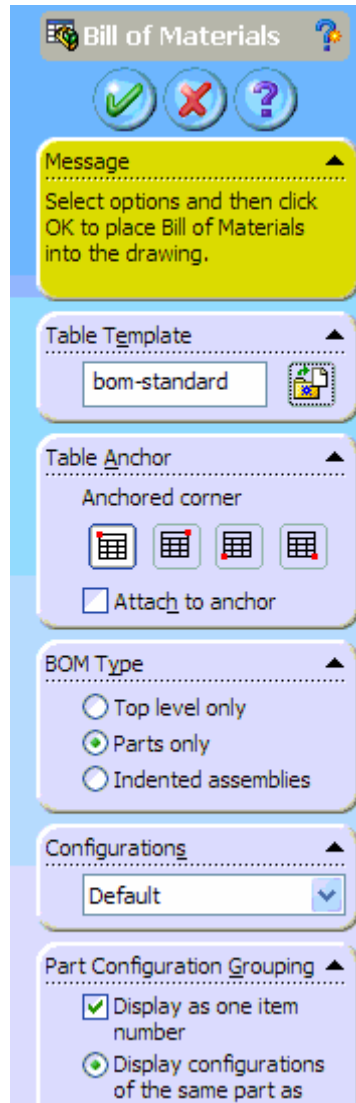
1. If the pizza cutter assembly is exploded after opening the file, *right click* on pizza cutter and select **Collapse**.
3. If the view imports in a different scale than the desired 1:4, *right click **Sheet Format1*** in the FeatureManager and select **Properties**. Set the **Scale** to **1:4**.
4. Use the **Model View** button instead of **Named View** to insert the **Isometric** view.

### 5.4.3 Adding a Section View

Page 128

1. If the pizza cutter assembly is exploded after opening the file, *right click* on pizza cutter and select **Collapse**.
2. The light bulb no longer appears, but it should be obvious when the cursor is aligned with the centerline.
3. If the **Tangent Edge Display** dialog box appears after placing the section view, select **Removed** and then **OK**.

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 in the PropertyManager to match the figure below.



**Figure 5.31.** *Bill of Materials* PropertyManager.

5. A Bill of Materials is now edited directly in SolidWorks, without the use of Excel. To remove the empty DESCRIPTION column in the table, *right click* on the word DESCRIPTION in the Bill of Materials and select **Delete** and then **Column**.
6. To change the title of the PART NUMBER column, move the cursor over Bill of Materials just above the word PART NUMBER. When a downward arrow appears, click so that the **Column** PropertyManager appears. Change the **Title** from **PART NUMBER** to **PART** and click **OK**. The width of the column can be changed by moving the cursor over the vertical line forming

the side of the column until a pair of horizontal arrows appear and then dragging the vertical line left or right.

## 6.2.2 Cutting the Solid Handle in Half

Page 142

4. When using the **Trim** tool, select **Trim to closest** under **Options** in the **Trim** PropertyManager so only the upper part of the circle is trimmed.

## 6.3.2 Creating the Sketch of the Lower Ribs

Page 157

1. To remove the tangent edges, use **Tools⇒Options** and select **Display/Selection** under **System Options**. Under **Part/Assembly tangent edge display**, use **Removed** and click **OK**.

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| <i>IN ADDITION</i> – Changing the tangent edges using <b>Tools⇒Options</b> is also necessary in paragraph 6.4.1.1. |
|--|

## 6.5.1 Setting up the Assembly Document

Page 169

1. The **Grid** button is no longer available and it is not possible to set up the **Units** until there is a part added to the assembly, so skip this part of the step. **Cancel** the **Insert Component** PropertyManager to close it.
3. After the shelled handle is added to the assembly, use **Tools⇒Options** and select **Units** under **Document Properties** to set the **Length units** to **inches**. Click **OK**.

## 6.5.3 Creating a Coincident Mate Using SmartMates

Page 171-172

2. 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. However, the first mate is a bit difficult to do using the keyboard shortcut, so it is done in the usual way, ignoring the remaining instructions in the book. Click the **Mate** button in the **Assembly** toolbar. The **Mate** PropertyManager appears.
3. Click on the face indicated in Figure 6.45 to highlight it.
4. Click on the same face on the other shelled handle. Both faces should appear under **Mate Selections** in the PropertyManager and one of the shelled handles will shift position. In addition, a toolbar should appear in the Graphics Window that can be used instead of the PropertyManager. The **Coincident** button should already be highlighted to indicate a coincident mate. Click on the **Flip Mate Alignment** button in the popup toolbar so the selected surfaces face one another.
5. Click **Add/Finish Mate** in the popup toolbar to finish the mate and **Close** the **Mate** PropertyManager.

## 6.5.4 Adding a Concentric Mates

Page 172-173

1. Ignore steps 1-4 in the book and do the following. **Zoom to Fit** if necessary. Hold down the ALT key and click on left hole surface shown in Figure 6.47.
2. While holding down the ALT key, drag the surface over the mating hole on the right piece until the paper clip icon near the cursor turns to two boxes, one above the other. It may be necessary to slowly move the cursor around a bit until it touches the second surface shown in Fig. 6.47 before the two-box icon appears at the cursor.
3. The mating surfaces should be highlighted on both parts and the **Concentric** button should be highlighted in the popup toolbar to indicate a concentric mate. Click **Add/Finish Mate** to finish the mate.
4. Repeat steps 1-3 above using the large hole at the tops of the handles to create a second concentric mate. Again it may be necessary to slowly move the cursor around a bit until it touches the surface of the hole in the second shelled handle before the two-box icon appears at the cursor. Click **Add/Finish Mate** to finish the mate.

## 6.5.5 Finishing the Handle Assembly

Page 175

3. Instead of switching to the **Bottom** view at the end of this step switch to **Isometric**.
4. Follow the steps to open ***Section View***, which is now in the PropertyManager. ***Top*** should already appear in the ***Reference Section Plane/Face*** field. If not, select it from the three buttons above the field. Type ***1.125*** in the ***Offset Distance*** box and click enter. If the preview shows the bottom portion of the handle, click on the ***Reverse Section Direction*** button in the ***Section View*** PropertyManager so the top half appears. Click the **Bottom** toolbar button to show the section as shown in Fig. 6.49 and click ***OK***.

## 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.

## 7.1.2 Creating the First Profile Sketch

Page 181

2. The **Mirror** tool has been replaced by two tools: the **Mirror** tool (labeled **Mirror Entities** button), which is used to mirror a sketch after it has been drawn, and the **Dynamic Mirror** tool, which mirrors as you sketch. You will use the **Dynamic Mirror** tool. First select the centerline so that it is highlighted, followed by **Tools⇒Sketch Tools⇒Dynamic Mirror** to activate the **Dynamic Mirror** tool.

6. It is only necessary to add the 1/2" vertical dimension and the 1/4" horizontal dimension to the sketch. The 1/2" horizontal dimension is driven by the other horizontal dimension, so it should not be included.
7. Use **Tools**⇒**Sketch Tools**⇒**Dynamic Mirror** to deactivate the **Dynamic Mirror** tool.

### 7.1.3 Creating the Second Profile Sketch

Page 183

3. *Right click* on **Plane1** in the FeatureManager design tree and click on **Edit Feature** to open the **Plane1** PropertyManager. Change the **Distance** to **5** and click **OK**.

### 7.1.7 Creating the Loft Feature

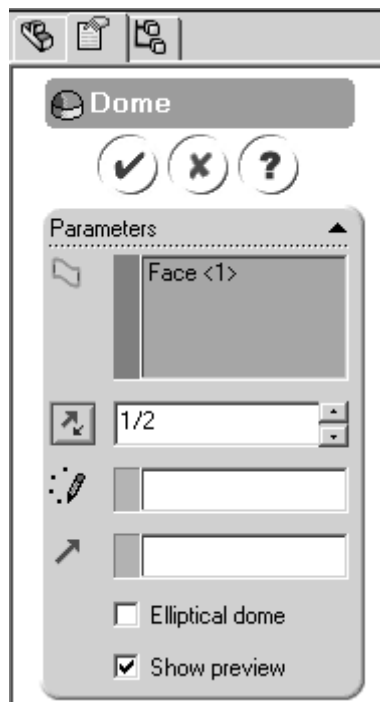
Pages 189-191

**SOFTWARE NOTICE** – Ensure the latest software update of SolidWorks is installed on your machine for the loft feature to work properly.

### 7.1.8 Modeling the Round End Using a Dome Feature

Page 191

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.

## 7.2.1 Creating a Radiated Surface

Pages 193-194

**SOFTWARE NOTICE** – Ensure the latest software update of SolidWorks is installed on your machine for the radiate surface feature to work properly.

5. Other edges may appear in the **Edges to Radiate** field than **Edge<1>** through **Edge<6>** and a different number of arrows may appear than shown in Figure 7.27. Continue to create the radiate feature as described in the tutorial.

## 7.2.7 The Mass Properties Dialog Box

Page 200

2. It may be necessary to click **Use custom settings** to be able to set the **Density** to **.284**.

## 7.3.1 Adding a Configuration to the Model of the Handle

Page 201-202

2. **Add Configuration** now appears in the PropertyManager.
4. The **Suppress features** checkbox is under **Advanced Options**.
5. In some situations the **What's Wrong** dialog box will appear with an error along with a dialog box asking to **Rebuild now?** First, click **No** to **Rebuild now?** Then **Close** the **What's Wrong** dialog box. Note that the **Fillet4** between the loft and the flange is no longer present in the model, so it must be added to the **no text** configuration. To do this, first select **no text** in the ConfigurationManager. Then click on the FeatureManager design tree tab. Note the **X** next to the last fillet that was added, indicating an error. **Right click** on the fillet in the FeatureManager design tree and select **Edit Feature**. The **Fillet** PropertyManager should appear as shown in Figure 7.34 with all of the fields filled in except **Edges, Faces, Features, and Loops**. Select the loop shown in Figure 7.34 and click **OK**. This should re-create the fillet properly.

## 7.3.2 Creating the Text Sketch

Page 203

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

## 7.3.4 Changing the Color of the Cut-Extrude Feature

Page 206

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

## 7.3.5 Modifying a Configuration Dimension

Page 207

3. Set the new dimension to **4 5/8** instead of **4 3/4**.

## 7.4.1 Setting up the Assembly Document

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- 2-3. The lofted handle part will automatically appear in the *Open documents* field of the *Insert Components* PropertyManager for placement in the assembly. To locate the handle so that it is aligned with the origins of the assembly, simply select *lofted handle* in the *Open documents* field so it is highlighted and click **OK**.

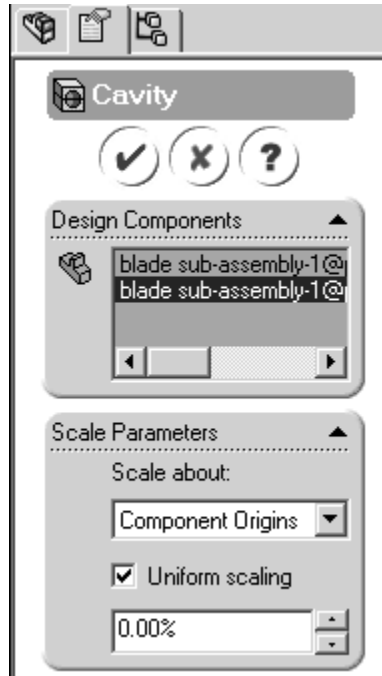
## 7.4.2 Mating the Cutter Sub-Assembly with the Handle

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1. Use **Insert⇒Component⇒Existing Part/Assembly** to insert the *cutter sub-assembly*. Use **Browse** in the *Insert Component* PropertyManager to open the *cutter sub-assembly*.
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. Use *Mate* instead of *SmartMates*, which is no longer available. Click on the edge of the arm shown in Figure 7.45 to select it.
7. Select the face of the handle shown in Figure 7.45 to select it. Create a *Coincident* mate and click **OK**.

### 7.4.3 Modeling the Rectangular Hole Using the Cavity Feature Page 211-212

1. The **Suppress feature** checkbox is under **Advanced Options**.
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**.
4. The **Cavity** feature is now located in the PropertyManager, as shown below.



**Figure 7.46.** *Cavity* PropertyManager.

### 7.4.4 Changing the Handle's Configuration in the Assembly Document Page 213

1. Select **Properties** instead of **Component Properties** from the menu that appears.