SolidWorks® 97
User's Guide

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SolidWorks API

The SolidWorks API is an OLE programming interface to SolidWorks. The API contains hundreds of functions that can be called from Visual Basic, VBA (Excel, Access, and so forth), C, C++, or solidWorks macro files. These functions provide the programmer with direct access to SolidWorks functionality such as creating a line, extruding a boss, or verifying the parameters of a surface.

For a detailed description of the API and the syntax used to call each function, please refer to the API online help file. This help file, API_HELP.HLP is located in the ..\SAMPLES\APPCOMM\ subdirectory of your SolidWorks installation. Also included in the ..\SAMPLES subdirectory are several Visual Basic and C++ example projects. Feel free to use these projects as a reference or as a starting point for your own applications.

You can also find a detailed description of the API functions on the SolidWorks web page (www.solidworks.com) under the Technical Support area.

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Toolbars

Displays or hides toolbars. See **Standard**, **View**, **Sketch**, **Sketch Tools**, **Sketch Relations**, **Macro**, **Dependency Editing**, **Assembly** ,**Drawing** toolbars and **Selection Filter** .

To display or hide a toolbar:

1 Click **View**, **Toolbars**.

2 Click the checkboxes to select each toolbar you want to display.

3 Click **OK** to make the changes and close the dialog; or click **Cancel**.

To move a toolbar:

Toolbars can be either "docked" (attached to one of the edges of the SolidWorks window) or "floating."

1 Point at the space between the buttons on the toolbar and drag the toolbar to the desired location. If you drag it to an edge of the SolidWorks window, the toolbar docks to that edge automatically.
2 To change a toolbar's orientation (from horizontal to vertical), drag the toolbar near a horizontal or vertical edge of the window before placing it in the desired location.

To display large size toolbar buttons:

Click **View, Toolbars** and select **Large Buttons** from the **Toolbars** dialog box.

To show tooltips:

Click **View, Toolbars** and select **Show Tooltips** from the dialog box. When checked, a small note pops up to identify each tool icon that you pause your cursor over.

For information about customizing your toolbars, see **Customize Toolbars** .

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Sketch

Toggles in and out of sketch mode.

To create a new sketch:

1. Select a face or a plane.
2. Click  or **Insert**, **Sketch**. The sketch grid appears (unless you have turned it off) and EDITING SKETCH appears in the status bar at the bottom of the SolidWorks window.
3. Use the tools on the **Sketch Tools** and **Sketch Relations** toolbars to draw and dimension the sketch.
4. When you are finished, click  or **Insert, Sketch** to close the sketch (for use in the creation of a loft or sweep, for instance), or click one of the sketch-based feature commands (e.g., **Insert, Boss, Extrude**) to create a feature from the sketch.

To edit a sketch:

1. Right-mouse click the sketch you want to edit or a feature built from the sketch, either in the model or in the FeatureManager design tree.

   Note For features built from multiple sketches, (lofts and sweeps), right-click on the sketch name in the FeatureManager design tree.
2. Select **Edit, Sketch** from the right mouse menu.

3 When you are finished editing, click or **Edit, Rebuild**, or right-click anywhere in the sketch and select **Exit Sketch** from the right-mouse menu.

See also **Fully Defined Sketches** .

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See also **Fully Defined Sketches** .

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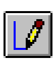
To create a new sketch:

1 Select a face or a plane.

2 Click  or **Insert**, **Sketch**. The sketch grid appears (unless you have turned it off) and EDITING SKETCH appears in the status bar at the bottom of the SolidWorks window.

3 Use the tools on the **Sketch Tools** and **Sketch Relations** toolbars to draw and dimension the sketch.

4 When you are finished, click  or **Insert, Sketch** to close the sketch (for use in the creation of a loft or sweep, for instance), or click one of the sketch-based feature commands (e.g., **Insert, Boss, Extrude**) to create a feature from the sketch.

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See also **Fully Defined Sketches** .

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Dimensions

Creates dimensions. The type of dimension (point-to-point, or angular) is determined by the items on which you click.

1 With a sketch active, click or **Tools**, **Dimensions** and select from **Parallel, Vertical, Horizontal**.

- or -

With a drawing active, click or **Tools, Dimensions** and select from **Parallel, Vertical, Horizontal, Baseline, Ordinate, Horizontal Ordinate**, or **Vertical Ordinate**.

- or -

Press the right mouse button and select **Dimension** from the menu. Right mouse click on a dimension and select the dimension type (**Horizontal**, **Vertical**, etc.) from the menu.

2 Click on the items to dimension, as shown in the table below.

3 Click again to place the dimension.

To Dimension...	Click on...	Note
Length of a line or edge	The line.	
Angle between two lines	Two lines, or a line and a model edge.	If the lines are parallel, this creates a distance dimension.
Distance between two lines	Two parallel lines or a line and a parallel model edge.	
Perpendicular distance from a point to a line	The point and the line or model edge.	
Distance between two points	Two points.	One of the points can be a model vertex.
Radius of an arc	The arc.	
Diameter of a circle	The circumference.	

To change a dimension value:

1 Double-click the dimension value.

2 Type a new value, click on the arrows to set a new value, or enter a calculation in the box.

- Click to accept the current value and exit.
- Click to restore the original value and exit.

- Click  to reverse the value of those parameters that are reversible, such as offset.
- Click  to regenerate the model with the current value.(Not available in a sketch.)
- Click  to reset the spin box increment value.

See also **Horizontal Dimension**, **Vertical Dimension**, **Parallel Dimension** , **Baseline Dimension**, **Ordinate Dimension**, **Horizontal Ordinate Dimension**, **Vertical Ordinate Dimension**.

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Dimension Preferences

Sets the default properties for new dimensions.

To set default dimension properties:

1 Click **Tools, User Preferences**.

2 Click the **Detailing** tab.

3 Change the settings. See **Dimension Properties** and **Dimension Tolerance** properties for details.

4 Click **OK**.

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Hole

See **Hole Wizard** and **Simple Hole** .

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Draft

Creates a feature that tapers selected model faces by a specified angle, relative to a neutral plane.

You can also use the draft feature in combination with a split line to create draft angles to a parting line on parts to be molded.

Note You can also apply a draft angle as part of an extruded base, boss, or cut. (See **Extrude**.)

To draft an existing model face:

1. Select a face or reference plane to serve as the neutral plane. The draft angle is measured perpendicular to this plane.
2. Hold down the **Ctrl** key and select one or more faces to draft.
3. Click **Insert**, **Draft**.
4. Set the **Draft Angle** and verify that you have a **Neutral Plane** and **Faces to Draft** selected.
5. If you want the draft to slant in the opposite direction, click **Reverse Direction**.
6. Choose an item from the **Face Propagation** list that describes how you want the draft to propagate across additional faces:
   - **None -** Draft only the selected face. **Example**
   - **Along Tangent -** Extend the draft to all faces that are tangent to the selected face. (The faces meet with filleted corners.) **Example**
   - **All Faces** - Draft all faces next to the neutral plane and extruded from the neutral plane. **Example**
   - **Inner Faces** - Draft all faces extruded from the neutral plane. **Example**
   - **Outer Faces** - Draft all faces next to the neutral plane. **Example**
7. Click **OK**.

To draft to a parting line:

1. With a part open that already has a split line, click **Insert, Draft**.
2. In the **Type of Draft** box, scroll to select **Parting Line**.
3. In the **Draft Angle** box specify the angle that you want.
4. Click in the **Direction of Pull** box and click the face toward which you want the draft angle to go. A directional arrow displays; click **Reverse Direction** if the arrow is not pointing in the correct direction.
5. Click in the **Parting Lines** box.
6. Hold the **Ctrl** key and select the parting line on each face of the part. Use the **Select Other** right-mouse option as necessary. The list of selected line segments displays in the **Parting Lines** box.

7 Choose an item from the list that describes how you want the draft to propagate across additional faces:

- **None** - Draft only the selected face.
- **Along Tangent** - Extend the draft to all faces that are tangent to the selected face.

8 Click **OK** to complete the draft of the part.

See also **Split Line** .

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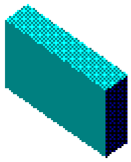
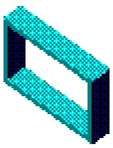
Extrude

Creates a feature that adds or removes material by extruding a profile for a specified distance.

You can extrude in one or two directions and add a draft while extruding.

An extrusion extends the sketched profile as either a solid feature or a thin feature. 



To create an extruded boss:

1 Create a sketch and draw the profile you want to extrude, or select an existing sketch that contains a profile. The profile must be closed and cannot intersect itself.

2 Click **Insert, Boss, Extrude** (or **Insert, Base, Extrude**, if this is the first feature in the part.)

3 Select the **Type**, and specify the **Depth** if necessary.

4 Click **Reverse Direction**, if necessary.

5 To add a **Draft**, select the **Draft While Extruding** box. Enter a draft **Angle** and check **Draft Outward**, if necessary.

6 To extrude the feature in both directions from the sketch plane, select **Both Directions**, then click **Go To Direction 2** and repeat steps 3 through 5 for the other end.

7 Set the **Extrude As:** box to **Solid Feature**.

8 Click **OK**.

To create a thin feature extrusion:

1 Sketch a profile for the extrusion. This does not have to be a closed profile.

2 Click **Insert, Base, Extrude**.

3 Select the **Type**, and specify the **Depth**.

4 To add a **Draft**, select the **Draft While Extruding** box, and enter a draft **Angle**.

5 To extrude the feature in both directions from the sketch plane, select **Both Directions**, then click

Go To Direction 2 and repeat the steps above.

6 If you are extruding a closed profile sketch, you must set the **Extrude As:** box to **Thin Feature**.

7 Click the **Thin Feature** tab at the top of the dialog box.

8 Select the extrusion **Type**, and specify the **Wall Thickness**.

9 If you created a closed profile you can select the following options:

- **Cap Ends.** Specifies that the ends of the thin feature are capped. A capped thin feature must be extruded from a closed profile sketch. When the ends are capped, all walls of the feature are closed; the center is hollow.

- **Cap Thickness**. Lets you specify the thickness of the cap. (Available only if **Cap Ends** is selected.)

- or -

If you created an open profile you can select the following options:

- **Auto Round.** Automatically creates a round at the each edge where lines meet at an angle.
- **Round Radius.** Specifies the inside radius of the round.

10 Click **OK**.

To create an extruded cut:

1 Create a sketch and draw the profile you want to extrude, or select an existing sketch that contains a profile. The profile does not have to be closed.

2 Click **Insert, Cut, Extrude**.

3 Select the **Type**, and specify the **Depth** if necessary.

4 Click **Reverse Direction**, if necessary.

5 Select **Flip Side to Cut** if you want to remove the material outside of the profile.

6 To add a **Draft**, check the **Draft While Extruding** box. Enter a draft **Angle** and check **Draft Outward**, if desired.

7 To extrude the feature in both directions from the sketch plane, select **Both Directions**, click **Go To Direction 2**, and repeat steps 3 through 6 for the other end.

8 Click **OK**.

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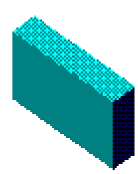
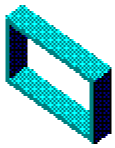
Extrude

Creates a feature that adds or removes material by extruding a profile for a specified distance.

You can extrude in one or two directions and add a draft while extruding.



An extrusion extends the sketched profile as either a solid feature or a thin feature.



To create an extruded boss:

1 Create a sketch and draw the profile you want to extrude, or select an existing sketch that contains a profile. The profile must be closed and cannot intersect itself.

2 Click **Insert, Boss, Extrude** (or **Insert, Base, Extrude**, if this is the first feature in the part.)

3 Select the **Type**, and specify the **Depth** if necessary.

4 Click **Reverse Direction**, if necessary.

5 To add a **Draft**, select the **Draft While Extruding** box. Enter a draft **Angle** and check **Draft Outward**, if necessary.

6 To extrude the feature in both directions from the sketch plane, select **Both Directions**, then click **Go To Direction 2** and repeat steps 3 through 5 for the other end.

7 Set the **Extrude As:** box to **Solid Feature**.

8 Click **OK**.

To create a thin feature extrusion:

1 Sketch a profile for the extrusion. This does not have to be a closed profile.

2 Click **Insert, Base, Extrude**.

3 Select the **Type**, and specify the **Depth**.

4 To add a **Draft**, select the **Draft While Extruding** box, and enter a draft **Angle**.

5 To extrude the feature in both directions from the sketch plane, select **Both Directions**, then click **Go To Direction 2** and repeat the steps above.

6 If you are extruding a closed profile sketch, you must set the **Extrude As:** box to **Thin Feature**.

7 Click the **Thin Feature** tab at the top of the dialog box.

8 Select the extrusion **Type**, and specify the **Wall Thickness**.

9 If you created a closed profile you can select the following options:

- **Cap Ends.** Specifies that the ends of the thin feature are capped. A capped thin feature must be extruded from a closed profile sketch. When the ends are capped, all walls of the feature are closed; the center is hollow.

- **Cap Thickness**. Lets you specify the thickness of the cap. (Available only if **Cap Ends** is selected.)

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If you created an open profile you can select the following options:

- **Auto Round.** Automatically creates a round at the each edge where lines meet at an angle.
- **Round Radius.** Specifies the inside radius of the round.

10 Click **OK**.

To create an extruded cut:

1 Create a sketch and draw the profile you want to extrude, or select an existing sketch that contains a profile. The profile does not have to be closed.

2 Click **Insert, Cut, Extrude**.

3 Select the **Type**, and specify the **Depth** if necessary.

4 Click **Reverse Direction**, if necessary.

5 Select **Flip Side to Cut** if you want to remove the material outside of the profile.

6 To add a **Draft**, check the **Draft While Extruding** box. Enter a draft **Angle** and check **Draft Outward**, if desired.

7 To extrude the feature in both directions from the sketch plane, select **Both Directions**, click **Go To Direction 2**, and repeat steps 3 through 6 for the other end.

8 Click **OK**.

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Loft

1 In the **Loft** dialog box, make certain that the profile sketches are listed in the correct order in the **Profiles** box.

2 If you are making a guide curve loft, make certain that the guide curve(s) list is correct in the **Guide Curves** box.

3 If necessary, rearrange the list items by clicking the **Move Up** or **Move Down** buttons.

4 To make the loft continue to create a closed body with the first and last sketches connected, select the **Close Along Loft Direction** checkbox.

5 Click **OK** to create the loft, or click **Cancel**.

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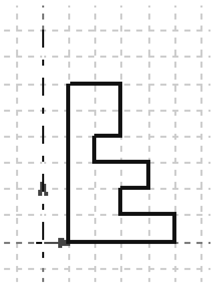
Revolve

Creates a feature that adds or removes material by revolving a closed profile around a centerline. The feature may be either solid or thin walled.

To create a revolved feature:

1 Create a sketch and draw a profile and a centerline. (The profile must be closed, and it cannot cross the centerline.)

If the sketch contains more than one centerline, select the centerline you want to use as the axis of the revolution.



2 If you are creating the first feature in a part, click **Insert**, **Base**, **Revolve**.

Otherwise click **Insert**, **Boss**, **Revolve** or **Insert**, **Cut**, **Revolve**.

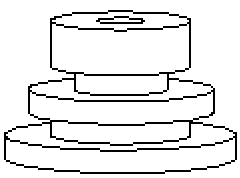
3 From the **Revolve As:** box, select either **Solid Feature** or **Thin Feature**.

4 If you select **Thin Feature**, click the **Thin Feature** tab, choose a direction from the **Type** list box, and specify a **Wall Thickness**.

- or -

If you select **Solid Feature**, choose a direction from the **Type** list box and set the desired rotation angle in the **Angle** box.

5 The preview shows the direction of rotation. Select the **Reverse** check box if you want to rotate the feature in the opposite direction.



6 Click **OK**.

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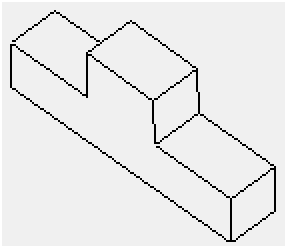
Shell

Creates a shell feature, removing selected faces.

To create a shell feature:

1 Click on the face you want to remove, or hold down the **Ctrl** key and select multiple faces.

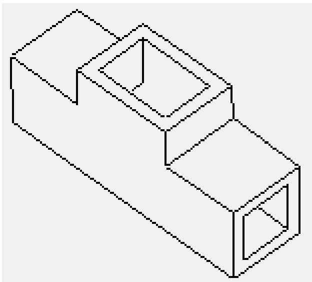


2 Click **Insert, Shell**.

3 Set the wall thickness in the **Thickness** box.

4 Select **Shell Outward** if you want the shell feature to increase the outside dimensions of the part.

5 Click **OK**.



To shell faces to different thicknesses:

1 After you select a face to shell and click **Insert, Shell**, click in the **Multi Thickness Faces** box.

2 Hold the **Ctrl** key and click on the faces that will remain as walls of varying thicknesses.

3 As you click the faces, a list forms in the **Multi Thickness Faces** box.

4 Click each face name on the list and enter a **Thickness** value for each one.

5 Choose whether to shell the part inward or outward.

6 Click **OK**.

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Hole

See **Hole Wizard** and **Simple Hole** .

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Sweep

Creates a feature that adds or removes material by projecting a profile along an open or closed sketched path or a model edge.

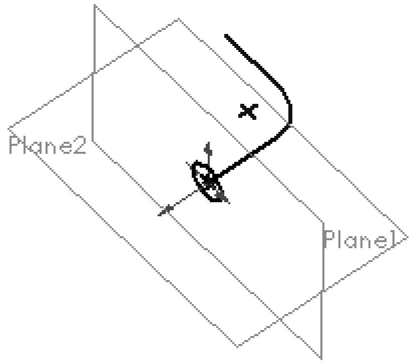
To create a sweep:

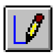
1 Sketch the profile on a face or a plane. The profile must be closed.

2 Click  to close the sketch.

3 On another face or plane, sketch the sweep path the profile will follow.

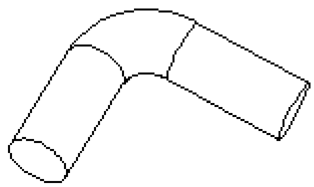
The start point of the path curve must lie on the plane of the profile. The path curve must not intersect itself.



4 Click  to close the sketch.

5 Hold down the **Ctrl** key and select both sketches.

6 If you are creating the first feature in a new part, click **Insert, Base, Sweep**. Otherwise, click **Insert,**



Boss, Sweep or **Insert, Cut, Sweep**.

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Plane

Creates a construction plane in a part or assembly.

To create a construction plane:

1 Click **Insert**, **Plane**.

2 Select the type of plane you want to create and click **Next**.

3 Select the appropriate number of vertices, faces, or edges for the type of plane you want to create.

4 Enter the required distance or angle, if any, and click **Reverse Direction,** if necessary.

5 Click **Finish**.

See also **Plane at Angle** , **Offset Plane** , **On Surface Plane** , **Parallel Plane at Point** , **Perpendicular to Curve at Point Plane** , **Line and Point Plane** , **Three Point Plane** .

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Axis

Creates an axis in a part or assembly.

To create an axis:

1 Click **Insert**, **Axis**.

2 Select from the **Options to make an axis** box.

- **One Axis.** Select **View,** **Temporary Axes** , and then select the axis.
- **Two Planes.** Select **View, Planes**, and then select two construction planes (while holding the **Ctrl** key).
- **Two Points.** Select two sketched points or two vertices.
- **One Line.** Select one line.
- **One Surface.** Select one cylindrical surface.

3 Verify that the items listed in the **Valid Items for an Axis** box correspond to your selection(s).

4 Click **OK**.

Note Axis display must be turned on (see **Axes**) to see the new axis.

Top

Design Table

Allows you to build multiple configurations of parts by driving dimension values from cells in an embedded Microsoft Excel spreadsheet.

- **Insert, Design Table** creates a new design table from a spreadsheet.
- **Edit, Design Table** allows you to change the values in a design table.
- **Edit, Delete Design Table** allows you to delete a design table.

Note You must have Microsoft Excel installed on your computer to use design tables.

To create a design table:

1 Open Microsoft Excel and create a spreadsheet.

NoteThis procedure describes entering the data in Excel before inserting the table. You can also insert an empty or partially empty spreadsheet and edit it in SolidWorks, as described below.

2 In the first *row*, enter the names of the dimensions or features that you want to control.

Dimension names are in the form ***Dimension name @ Feature*** or ***Sketch name***, or just the ***Feature name***. For example, the default name for the depth of the first extrusion in a part is D1@Base-Extrude1.

To see a dimension’s name, right-click on the dimension, then click **Properties**. You can also use the **Properties** command to assign more meaningful names to dimensions. Dimension names are case-sensitive, so the name in the spreadsheet must match the name in the part exactly.

3 In the first *column*, enter the names of the design variations you want to create.

4 Fill in the dimension values in each column. For example:

	D1@Base-Extrude	D2@Cut1-Extrude
Housing, rev 1	**45**	**88**
Housing, rev 2	**48**	**94**

Note You can unsuppress a feature by putting the feature name in the first row and **yes** in the appropriate cell. You can suppress a feature by leaving the related cell *blank*. For example, the following will suppress the feature **Cut1** in the version named **rev 2**.

	D1@Base-Extrude	Cut1-Extrude
Housing, rev 1	**45**	**yes**
Housing, rev 2	**48**	

5 Save the spreadsheet.

6 Open the part where you want to use the design table.

7 Click **Insert, Design Table**.

8 Locate the spreadsheet file and click **OK**.

The spreadsheet appears in the part document, and the Excel menus and toolbars also appear.

9 Edit the table, if necessary. When you are finished editing, click on the part, anywhere outside the table. The SolidWorks menus and toolbars will reappear, and the part will rebuild, using the first row of dimension values in the table.

To select another configuration, click **Tools, Configuration**, and select a configuration from the list displayed by the **SolidWorks Configurations Manager** dialog.

Note The various design table instances can be associated to drawing views.

Different design table instances can be displayed simultaneously in different drawing views.

To edit a design table:

1 Click **Edit, Design Table.** The table appears, and the Excel menus and toolbars also appear.

2 Edit the table. You can change the dimension values in the cells, or add new columns to add design variations or new rows to control additional dimensions.

3 Click anywhere outside of the design table to close it.

Note The design table is saved in the part file and is not linked back to the original Excel file. Changes you make in the part are not reflected in the original file.

Top

Definition

Allows you to change the original definition of a feature.

To change a feature's definition:

1 Click on the feature or on its name in the FeatureManager design tree, then click **Edit, Definition**.

Shortcut Right-click the feature or its name, then click **Edit Definition.**

2 Change the **Depth**, **Distance**, **Radius**, or other parameters as necessary.

3 Click **OK**.

Top

Equations

Creates mathematical relations between model dimensions, using dimension names as variables. When using equations in an assembly, you can set equations between parts, between a part and a subassembly, with mating dimensions, and so forth.

Note Dimensions driven by equations cannot be changed by editing the dimension value in the model.

To add an equation:

1 Click Tools, Equations.

2 Click **Add**.

3 In the model or the FeatureManager design tree, double-click on the feature that contains the first dimension you want to use in the equation.

4 Click on the dimension to paste its name into the equation. (Dimension names are in the form ***dimension name*** @ ***feature*** or ***sketch name***.)

5 Complete the equation by typing or clicking on the calculator buttons, or by clicking on other dimensions to paste their names.

Equations are solved left to right (i.e., the dimension on the left is driven by the value on the right), in the order in which they appear in the equation list.

6 Click **OK**. The equation will appear in the **Equations** window.

7 Click **OK**, then click  or **Edit, Rebuild** to update the model. (All equations are solved before the geometry is regenerated.)

To edit equations:

1 Click **Tools, Equations**.

2 Click **Edit All.**

3 Edit the equations. Each equation must be on a separate line, and dimension names must be enclosed in quotes.

4 Click **OK** to close the **Edit Equations** window.

5 Click **OK** to close the **Equations** window.

6 Click  or **Edit, Rebuild**.

Top

Copy

Copies the selected item(s) to the Clipboard.

To copy:

1 Select the item(s) to copy.

2 Click , or **Edit**, **Copy**, or press **Ctrl-C**.

Top

Suppress

Suppresses a feature so you can work on the model with the selected feature temporarily omitted from the model. Features that are children of the selected feature are also suppressed.

To suppress a feature:

1 Select a feature in the FeatureManager design tree.

2 Click  or **Edit**, **Suppress**.

The feature disappears from the model view and its icon is grayed in the FeatureManager design tree.

Use **Unsuppress** or **Unsuppress with Dependents** to restore suppressed features.

Top

Rollback

Reverts the model to the state it was in before the selected feature was created.

Rollback also flattens sheet metal bends. See also **Bends** .

To revert to an earlier state:

1 Select a feature in the FeatureManager design tree or in the graphics display area.

2 Click  or **Edit**, **Rollback**.

The model reverts to the state it was in before the selected feature was created.

You can now add new features or edit existing features while the model is in the rolled-back state.

3 To roll forward again, click  or **Edit**, **Rebuild**. (The model will not rebuild unless you answer **Yes** in the dialog box.)

Top

Circular Pattern

Creates multiple instances of a feature spaced evenly around an axis.

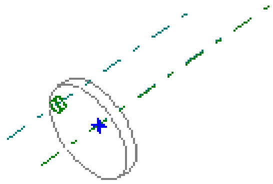
If you modify the original feature, all instances in the pattern are updated to reflect the change.

To create a circular pattern:

1 Create a feature that you want to replicate.

2 Create an axis around which to pattern the feature.

For information about how to create an axis, see **Axis**.

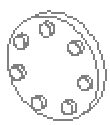


3 Select the axis, and Ctrl-select the feature(s) to pattern.

4 Click **Insert, Pattern/Mirror, Circular Pattern**.

5 Specify the **Spacing** and the **Total Instances** of the feature.

6 Click **OK**.



To delete pattern instances, see **Pattern Deletion** .

See also **Linear Pattern** .

Top

Line

Creates a line.

To create a line:

1 Click  or **Tools**, **Sketch Tools**, **Line**, or, if you are in a drawing, click **Tools**, **Drafting Tools**, **Line**.

2 Point where you want the line to start.

3 Press the mouse button and drag to where you want the line to end.

4 Release the mouse button.

Top

Angle Increment

Enter the number of degrees to add or subtract when you click on a spin box arrow to change an angular dimension value.

Top

Component

Adds a component to an assembly.

To add a component:

1. With an assembly open, click **Insert**, **Component**.

   The **Open** dialog box appears.
2. Browse to the directory that contains the component that you want to insert into the assembly.
3. Click **Open**. The cursor changes to a cross .
4. Click in the assembly window where you want to locate the component.

See also **Move Component** and **Rotate Component** .

Top

Angle Increment

Enter the number of degrees to add or subtract when you click on a spin box arrow to change an angular dimension value.

Top

Explode Assembly

Explodes the view of an assembly so you can look at it with the components slightly separated. At times, it is useful to separate the components of an assembly to visually analyze their relationships.

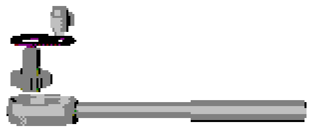
When you explode an assembly, the application displaces all of the offset and coincident mating relationships by an equal amount. The components retain their positions in relation to one another.

To explode an assembly:

1 Right-click the assembly name in the FeatureManager design tree.

2 Select **Explode** from the right mouse menu.

An exploded display of the assembly appears.



To return the display of the assembly to a non-exploded view, repeat this procedure and select **Collapse** from the right mouse menu.

See also **Edit Explode** .

Top

Interference Detection

Checks to see whether portions of any two components in an assembly occupy the same space. The volume of interference is graphically displayed as highlighted edges showing the over-lapping portions of the components. (One volume of interference is displayed at a time.)

To check for interference in an assembly:

1 Hold the **Ctrl** key and select two components in the assembly.

2 Click **Tools**, **Interference Detection**.

The **Interference Volumes** dialog opens and the **Selected Components** list displays the names of the selected components.

If there is interference,

- The names of the components are reported in the Component 1 and Component 2 boxes
- The interference name (Interference1, Interference 2, and so on) is reported in the **Interference List**. When you click an interference name, the related interference volume is highlighted in the graphics display area.
- The volume of the interference is reported in the form of length, width, and height of the bounding box around the area of interference. These numbers are displayed on the graphic display of the component.

3 With the dialog box still active, you can reselect other components to check for interference.

4 Click **Recheck** to see if the newly selected components interfere.

5 Click **OK** to dismiss the dialog box.

When the dialog box is dismissed, the volumes are dismissed also.

See also **Interference Volumes** .

Top

Component

Adds a component to an assembly.

To add a component:

1 With an assembly open, click **Insert**, **Component**.

The **Open** dialog box appears.

2 Browse to the directory that contains the component that you want to insert into the assembly.

3 Click **Open**. The cursor changes to a cross .

4 Click in the assembly window where you want to locate the component.

See also **Move Component** and **Rotate Component** .

Top

Dimensions

Creates dimensions. The type of dimension (point-to-point, or angular) is determined by the items on which you click.

1 With a sketch active, click or **Tools**, **Dimensions** and select from **Parallel, Vertical, Horizontal**.

- or -

With a drawing active, click or **Tools, Dimensions** and select from **Parallel, Vertical, Horizontal, Baseline, Ordinate, Horizontal Ordinate**, or **Vertical Ordinate**.

- or -

Press the right mouse button and select **Dimension** from the menu. Right mouse click on a dimension and select the dimension type (**Horizontal**, **Vertical**, etc.) from the menu.

2 Click on the items to dimension, as shown in the table below.

3 Click again to place the dimension.

To Dimension...	Click on...	Note
Length of a line or edge	The line.	
Angle between two lines	Two lines, or a line and a model edge.	If the lines are parallel, this creates a distance dimension.
Distance between two	Two parallel lines	

lines	or a line and a parallel model edge.	
Perpendicular distance from a point to a line	The point and the line or model edge.	
Distance between two points	Two points.	One of the points can be a model vertex.
Radius of an arc	The arc.	
Diameter of a circle	The circumference.	

To change a dimension value:

1 Double-click the dimension value.

2 Type a new value, click on the arrows to set a new value, or enter a calculation in the box.

- Click  to accept the current value and exit.
- Click  to restore the original value and exit.
- Click  to reverse the value of those parameters that are reversible, such as offset.
- Click  to regenerate the model with the current value.(Not available in a sketch.)
- Click  to reset the spin box increment value.

See also **Horizontal Dimension**, **Vertical Dimension**, **Parallel Dimension** , **Baseline Dimension**, **Ordinate Dimension**, **Horizontal Ordinate Dimension**, **Vertical Ordinate Dimension**.

Top

Bill of Materials

Lets you insert a bill of materials into the drawing of an assembly.

Note To insert a bill of materials in a drawing, you must have Microsoft Excel installed on your computer.

To insert a bill of materials:

1 With a drawing view selected, click **Insert, Bill of Materials**.

2 Specify the information to include in the bill of materials by clicking the radio button in front of either **Show parts only** or **Show top level subassemblies and parts only**.

3 Select from the following options:

- **Use Summary Info title as part number.** If you assigned a part identifier number in the title box of the **Summary Info** for the part, you can use that identifier in the bill of materials.

- **Add new item by extending top border of table.** New components are added at the top of the bill of materials instead of at the bottom.

  A bill of materials is displayed that lists the parts in your assembly.

4 Insert any additional information or columns in the spreadsheet using the appropriate Excel commands and icons.

5 Click a corner of the spreadsheet to change its size.

6 Click anywhere outside the spreadsheet to close it.

7 Double-click on the spreadsheet to open it again for editing.

To move the bill of materials:

1 Click on the spreadsheet. Your cursor should change to its *Move* shape .

2 Hold the left mouse button and move the cursor to where you want to place the bill of materials.

3 Release the mouse button.

To update the bill of materials:

1 Click in the drawing view originally selected to create the bill of materials.

2 Click **Insert**, **Bill of Materials**.

The existing bill of materials updates to include any new components that you added to the assembly; components that you deleted are removed from the bill of materials.

See also **Balloon** .

Top

Dimensions

Creates dimensions. The type of dimension (point-to-point, or angular) is determined by the items on which you click.

1 With a sketch active, click or **Tools**, **Dimensions** and select from **Parallel, Vertical, Horizontal**.

- or -

With a drawing active, click or **Tools, Dimensions** and select from **Parallel, Vertical, Horizontal, Baseline, Ordinate, Horizontal Ordinate**, or **Vertical Ordinate**.

- or -

Press the right mouse button and select **Dimension** from the menu. Right mouse click on a dimension and select the dimension type (**Horizontal**, **Vertical**, etc.) from the menu.

2 Click on the items to dimension, as shown in the table below.

3 Click again to place the dimension.

To Dimension...	Click on...	Note
Length of a line or edge	The line.	

Angle between two lines	Two lines, or a line and a model edge.	If the lines are parallel, this creates a distance dimension.
Distance between two lines	Two parallel lines or a line and a parallel model edge.	
Perpendicular distance from a point to a line	The point and the line or model edge.	
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1 Double-click the dimension value.

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- Click to accept the current value and exit.
- Click to restore the original value and exit.
- Click to reverse the value of those parameters that are reversible, such as offset.
- Click to regenerate the model with the current value.(Not available in a sketch.)
- Click to reset the spin box increment value.

See also **Horizontal Dimension**, **Vertical Dimension**, **Parallel Dimension** , **Baseline Dimension**, **Ordinate Dimension**, **Horizontal Ordinate Dimension**, **Vertical Ordinate Dimension**.

Top

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Creates dimensions. The type of dimension (point-to-point, or angular) is determined by the items on which you click.

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Press the right mouse button and select **Dimension** from the menu. Right mouse click on a dimension and select the dimension type (**Horizontal**, **Vertical**, etc.) from the menu.

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3 Click again to place the dimension.

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Distance between two lines	Two parallel lines or a line and a parallel model edge.	
Perpendicular distance from a point to a line	The point and the line or model edge.	
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- Click to restore the original value and exit.
- Click to reverse the value of those parameters that are reversible, such as offset.
- Click to regenerate the model with the current value.(Not available in a sketch.)
- Click to reset the spin box increment value.

See also **Horizontal Dimension**, **Vertical Dimension**, **Parallel Dimension** , **Baseline Dimension**, **Ordinate Dimension**, **Horizontal Ordinate Dimension**, **Vertical Ordinate Dimension**.

Top

Sheet

Adds a new sheet to the active drawing.

To add a new drawing sheet:

1 With a drawing active, click **Insert**, **Sheet**.

2 Specify the drawing name, sheet size, scale and other drawing requirements in the **Sheet Setup** dialog box.

3 Click **OK**.

Tip To move between sheets in a drawing, click the sheet tabs at the bottom of the drawing window, or click **View**, **Next Sheet** or **View**, **Previous Sheet**.

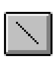
Top

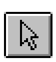
Aligned Section

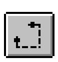
Creates a section view through a model, or portion of a model, that is aligned with the selected section line.

To create an aligned section view:

1 Double-click the drawing view to activate it.

2 Click  or **Tools, Drafting Tools, Line** and draw the section line. The section line should be two connected lines at an angle to each other.

3 Click  and select the line to which you want to align the view.

4 Click  or **Insert, Drawing View, Aligned Section**. This converts the lines into a section line, with arrows indicating the direction of the view.

5 If necessary, double-click on the section line to reverse the direction of the view.

You can edit section lines by dragging them to make them longer or shorter, and then clicking  or **Edit, Rebuild**.

To change the section line label or the direction of the view, right-mouse click the line and select **Properties**. Make your changes, then click **Rebuild**.

See **Section Line Properties** for more information.

See also **Crosshatch Preferences** .

Top

Auxiliary

Creates an auxiliary view, unfolded from a drawing view along a selected edge.

To create an auxiliary view:

1 In a drawing view, select the edge that you want to use as the axis for unfolding the auxiliary view. (To unfold horizontal or vertical projections, use the **Projection** command.)

2 Click  or **Insert**, **Drawing View**, **Auxiliary**.

Top

Angle Increment

Enter the number of degrees to add or subtract when you click on a spin box arrow to change an angular dimension value.

Top

Detail

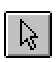
Creates a detail view in a drawing.

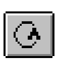
To create a detail view:

1 Activate a drawing view. (Double-click the view, or right-click on the view and click **Activate View**.)

2 Click  or **Tools**, **Drafting Tools**, **Spline,** or click  or **Tools, Drafting Tools, Circle**.

3 Sketch a closed spline or circle around the area of the drawing that you want in the detail view.

4 Click  and select the spline or circle.

5 Click  or **Insert**, **Drawing View**, **Detail**.

A detail circle appears in the original view and a detail view appears.

6 Click on the detail view and drag it to the desired location on the sheet.

NoteTo change the detail label on the original view, double-click the label and type the new text. Click **OK**.

See also **Detail Circle** .

Top

Angle Increment

Enter the number of degrees to add or subtract when you click on a spin box arrow to change an angular dimension value.

Top

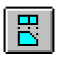
Projection

Creates a new orthographic view that is unfolded from an existing view.

To create a projection view:

1 Open the drawing that has the view you want to project.

2 Click on the view you want to project.

3 Click  or **Insert**, **Drawing View**, **Projection**.

4 Click on the drawing sheet to insert the projection. If you click to the right of the selected view, the right face is inserted; if you click above, the top face is inserted, and so on.

Top

Relative to Model

Creates a drawing view by selecting two orthogonal faces to define an orientation.

To create the view:

1 Open a part, and open the drawing where you want to insert a view of the part.

2 From the drawing, select  or **Insert**, **Drawing View**, **Relative to Model**. A small part symbol appears on the cursor to indicate that you need to select a part.

3 Activate the part window by clicking on it or by choosing it from the **Window** menu.

4 Click on a face on the part.

5 Select the direction you want the selected face to be oriented towards, then click **OK**.

6 Click on a second face, then select its orientation and click **OK**.

7 Activate the drawing window, and click where you want to place the view.

Top

Section

Creates a cross section drawing view of the model.

To create a section view:

1. Double click the drawing view where you want to draw the section line. (This activates the view so you can draw on it.)
2. Click or **Tools**, **Drafting Tools**, **Line** and draw the section line or a series of connected lines.
3. Click and select the sketched line(s).
4. Click or **Insert**, **Make Section Line**. This converts the line(s) into a section line, with arrows indicating the direction of the view.
5. If necessary, double-click on the section line to reverse the direction of the view.
6. Select the section line.
7. Click **Insert**, **Drawing View**, **Section**.

You can edit section lines by dragging them and then clicking **Rebuild**.

To change the section line label or the direction of the view, right-mouse click the line and select **Properties** . Make your changes, then click **Rebuild**.

The solid interior of the section view is crosshatched. To change the pattern, see **Crosshatch**.

Top

Standard 3 View

Creates three standard orthographic views on a drawing sheet.

To create a drawing:

1. Open a part or an assembly and a drawing.
2. With the drawing window active, click or **Insert**, **Drawing View**, **Standard 3 View**.
3. From the **Window** menu, select the part or assembly to make it the active window.
4. Click anywhere in the part or assembly window.

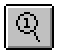
   The three views will appear on the drawing sheet. The type of projection (first or third angle) depends on the setting specified by the **Sheet Setup** command.
5. If necessary, drag the drawing views to position them on the sheet. The top and side views retain their alignment with the front view as you drag them; dragging the front view moves all three views.

Top

Balloon

Inserts balloons in a drawing of an assembly. If you previously inserted a bill of materials in the drawing, the balloons label the parts in the assembly and relate them to items on the bill of materials. The balloon leader attaches to the part in the drawing view where you click.

To insert a balloon:

1 Click  or **Insert**, **Annotations**, **Balloon**.

2 Click on a part in a drawing view.

A balloon that contains a number corresponding to the part number on the bill of material attaches to the part that you clicked.

3 To move the balloon or leader arrows, drag them with the **Select** tool, .

To change the balloon text, font style, or leader configuration:

1 Double-click on the balloon.

2 Make the changes in the **Note** dialog and click **OK**.The balloon leader attaches to the point on a drawing view where you click. Use balloons to label drawing features and relate them to items on a bill of materials.

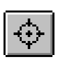
See also **Bill of Materials** .

Top

Center Mark

Creates axis lines for showing center marks on circles that can be used as a reference points for dimensioning.

To create a center mark:

Click  or **Insert, Annotations, Center Mark**, and click the circle.

To change the display attributes of individual center marks:

1 Right-mouse click the center mark and select **Properties**.

2 Click to uncheck **Use document's defaults**.

You can set the default size of center marks and choose whether or not to display extended axis lines.

3 Make the changes that you want and click **OK** to accept the changes; click **Cancel** to close the dialog box without saving any changes.

To set your default preferences for center marks, click **Tools, User Preferences** and select the **Detailing** tab.

Top

Cosmetic Threads

Represents threads on a part, assembly, or drawing.

To insert cosmetic threads on a part or drawing:

1. On a cylindrical part, click a circular edge where the thread should begin.
2. Click **Insert, Cosmetic Thread**.

   - or -

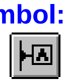
   in a drawing view, click a circular edge and click **Insert, Annotations, Cosmetic Thread**.
3. In the **Cosmetic Thread** dialog box, select the thread to apply:
   - Select **Blind** to run the thread for a specified distance, and enter the distance in the value box.
   - Select **Up to Next** to run the thread to the next face.
4. Enter a value in the **Minor Diameter/Major Diameter** box.
5. In the **Thread Callout** box, enter thread callout text, if desired.
6. Click **OK** to create the thread; click **Cancel** to exit.

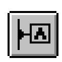
Top

Datum Feature Symbol

Attaches a datum feature symbol to a selected place on a drawing.

To insert a datum feature symbol:

1. With a drawing active, click  or **Insert, Annotations, Datum Feature Symbol**.
2. Click the surface on which you want to place a datum feature symbol.

   You can place several datum feature symbols; the datum letters are assigned alphabetically (omitting I, O, and Q to comply with ANSI standards).
3. To stop placing datum feature symbols, click  or **Insert, Annotations, Datum Feature Symbol** again.

To edit the datum feature symbol, right-click the symbol and select **Properties** from the menu.

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Datum Target Point

Locates target points on the model surface in a drawing.

To place target points on a drawing view:

1 Activate the drawing view.

2 Click or **Insert, Annotations, Datum Target Point**.

3 Point and click the mouse where you want to place a target point.

Notice that an inferencing line is available to assist you in placing the target points in relation to each other.

You can also apply constraints (**Add Relations**) to the datum target point, as you can with a sketch point.

4 Click again to end making target points.

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Datum Target Symbol

Creates and places datum target symbols on a drawing view.

To create a datum target symbol:

1 Click a model edge or a datum target point in the drawing view.

2 Click or **Insert, Annotations, Datum Target Symbol**.

The **Datum Target Symbol** dialog appears.

3 Enter a value for the **Target area size** box; click if the value describes a diameter.

4 Click the **Display target area size outside** check box if you want to place the value outside the target symbol circle.

5 Enter up to three **Datum Reference** labels in the boxes.

6 Specify the leader style that you want:

- Check **Bent Leader**, if appropriate.
- Select an **Arrowhead Style** from the pull-down menu.
- Select a **Line Style** from the scroll-down list. Typically, a **Solid** line indicates the target is on the near side; a **Dashed** line indicates the target is on the far side.

7 Click **OK** to accept the datum target symbol that you constructed; click **Cancel** if you do not want to accept the symbol.

To change the location of the datum target symbol, click and drag the circle.

To edit the datum target symbol, right-mouse click on the circle and select **Properties** from the menu. The **Datum Target Symbol** dialog appears.

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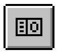
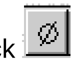
Geometric Tolerance

Inserts ANSI GD&T (Geometric Distancing and Tolerance) symbols in a drawing.

To insert geometric tolerances:

1. In a drawing, click  or **Insert**, **Annotations**, **Geometric Tolerance**.
2. In the first **Feature Control Frame** area, click **GCS**, then select a Geometric Characteristic Symbol and click **OK**.

   Notice that as you make each selection, the information appears in the preview display box.
3. Enter a tolerance value in the **Tolerance 1** box, and click  if you want to include a **Diameter** symbol.
4. Click **MC**, to choose the material condition symbol for **Tolerance 1** and click **OK**.
5. Repeat steps 3 and 4 for **Tolerance 2**.
6. Enter tolerance values and material condition symbols for the **Primary**, **Secondary**, and **Tertiary** datums.
7. Repeat the process for the second **Feature Control Frame**,

   - or -

   Click the **Composite Frame** check box and complete entering values and material condition symbols.
8. Click **Between Two Points** if the tolerance value applies to a measurement between two points or entities.
9. If you want to enter a **Projected Tolerance Zone** height, click the **Show PTZ** check box and enter a value in the **Height** box. The value is displayed in the first frame's tolerance box.
10. To specify styles for leaders, arrows and/or fonts, click the **Options** button.
11. When you are finished, click **OK** and drag the geometric tolerance frame to the desired position on your drawing.

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Hole

See **Hole Wizard** and **Simple Hole** .

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Note

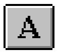
Inserts a note into the drawing. You may create a free-floating note or a note with a leader line attached to a specific point on the drawing.

Also used for **Customizing Drawing Templates** .

To create a note:

1 On a drawing view, click where you want to place the note.

Note If you choose to attach a leader line, the leader's arrow will point where you clicked.

2 Click  or **Insert**, **Annotations**, **Note**.

3 Type the text into the **Note Text** box.

4 Select **Use Document's Font**, or click the **Font** button to choose from various font styles and sizes.

5 Select **Display with Leader** if you want a leader line.

6 To add a symbol to the note text, place the cursor in the text box where you want the symbol to appear and click **Add Symbol**.

7 Select a **Symbol Library** from **Geometric Tolerancing**, **Hole Symbols**, and **Modifying Symbols**.

8 Select a symbol name from the list, and click **OK**.
(The symbol's name displays in the text box, but the actual symbol appears in the drawing note.)

9 Click **OK**.

Note Selections:

- **Display with Leader.** Specifies that note is attached to a relevant place on the drawing with leader line.
- **Use Document's Font.** Specifies that the font you selected in **User Preferences** is used for note text.
- **Arrow Style.** Specifies style (**Open, Filled,** or **Simple** arrowheads, **Open** or **Filled** dots, or **Slash**) of dimension arrows.
- **Display Bent Leader.** Specifies the display of notes with a bent leader.
- **Display with Balloon.** Specifies the display of balloons.
- **Leader Anchor.** Specifies the default position of leaders in drawings: **Closest, Left**, or **Right**.

See also **Geometric Tolerance** .

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Symbol

Inserts a symbol.

To add a symbol to text:

1 From **Symbol Libraries**, select **Geometric Tolerancing**, **Hole Symbols**, or **Modifying Symbols**.

2 Select a symbol name from the list, and click **OK.**

The selected symbol appears in the preview box.

Note The symbol is represented by its name in the **Note Text** preview box, but the actual symbol appears in the drawing note.

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Symbol

Inserts a symbol.

To add a symbol to text:

1 From **Symbol Libraries**, select **Geometric Tolerancing**, **Hole Symbols**, or **Modifying Symbols**.

2 Select a symbol name from the list, and click **OK**.

The selected symbol appears in the preview box.

Note The symbol is represented by its name in the **Note Text** preview box, but the actual symbol appears in the drawing note.

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Angle Increment

Enter the number of degrees to add or subtract when you click on a spin box arrow to change an angular dimension value.

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Microsoft Office Compatible

SolidWorks is a Microsoft Office Compatible product. This means that the SolidWorks toolbars, menus, and accelerator keys are similar to the ones used by the Microsoft Office suite of applications, including Microsoft Word, Microsoft Excel, Microsoft Access, and PowerPoint.

If you already know how to use Microsoft Office, you will find that you already know how to do many of the similar tasks in SolidWorks.

SolidWorks and Microsoft hope that these many similarities will make it easier for you to use our products together and with other Microsoft Compatible products. Look for the Microsoft Office Compatible logo when you select new software to purchase.

For more information about the Microsoft Office Compatible program, and for a complete list of Microsoft Office Compatible products,

in the United States, call Microsoft Customer Service at 1-800-426-9400

\- or -

outside the United States, contact your local Microsoft office.

See also:

{button ,JI(`SLDWORKS.HLP',`MS_Office_Features')} SolidWorks and Microsoft Office Compatible Features

{button ,JI(`SLDWORKS.HLP',`MS_Office_with_SolidWorks')} SolidWorks Interaction with Microsoft Office

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SolidWorks Interaction with Microsoft Office

SolidWorks interacts with Microsoft Office tools:

- Using Microsoft’s OLE, you can link or embed SolidWorks part, assembly, or drawing documents in Microsoft products such as Word, Microsoft Excel, and PowerPoint.
- You can use Microsoft products such as Microsoft Excel to insert data into SolidWorks documents.
- You can send SolidWorks part, assembly, or drawing documents over the network using Microsoft’s mail application, as well as other Microsoft Office compatible mail products.

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