




# **SolidWorks<sup>®</sup> 96**


## **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](http://www.solidworks.com)) under the Technical Support area.

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## Assembly Toolbar

The **Assembly** toolbar controls the movement and mating of components.



**Rotate Component**



**Move Component**



**Mate**

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## Toolbars

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

### To display or hide a toolbar:

- 1 Click **View**, **Toolbars**.
- 2 Select each toolbar you want to display.

### 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** menu.

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## Dependency Editing Toolbar











The **Dependency Editing** toolbar controls feature dependencies.

-  **Rollback**
-  **Suppress**
-  **Unsuppress**
-  **Unsuppress with Dependents**
-  **Rebuild**

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## Drawing Toolbar

The **Drawing** toolbar controls the creation of drawing views, notes, and symbols.

-  **Note**
-  **Geometric Tolerance**
-  **Balloon**
-  **Center Mark**
-  **Detail**
-  **Section**
-  **Projection**
-  **Standard 3 View**
-  **Auxiliary**
-  **Named View**



**Relative to Model**

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## Macro Toolbar

The **Macro** toolbar controls macro recording, playback, and editing.



**Run Macro**



**Stop Macro**



**Record Macro**



**Edit Macro**

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## Toolbars

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## Sketch Relations

The **Sketch Relations** toolbar controls dimensions and other geometric relations.



**Dimension**



**Add Relation**



**Display/Delete Relations**



**Scan Equal**

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## Sketch Toolbar

The **Sketch** toolbar controls selection, sketch creation, sketch modification, and the sketch grid.



**Insert/Edit Sketch**



**Grid**



**Select**



**Modify** (This toolbar button is not visible by default. See [Customize Toolbars](#) for information about how to make this button visible.)

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## Sketch Tools Toolbar













The **Sketch Tools** toolbar creates sketch elements. This same toolbar provides **Drafting Tools** for creating draft elements when you are in a drawing. (Note that the **Offset Edges** tool is not available in drawings.)



**Line**



**Centerpoint Arc**

-  **Tangent Arc**
-  **3 Point Arc**
-  **Circle**
-  **Spline**
-  **Rectangle**
-  **Point**
-  **Centerline**
-  **Convert Edge**
-  **Mirror**
-  **Fillet**
-  **Offset Edges**
-  **Trim/Extend**

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## Standard Toolbar

The **Standard** toolbar controls file management and model regeneration.

-  **New**
-  **Open**
-  **Save**
-  **Undo**
-  **Rebuild**
-  **Redraw**

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## View Toolbar

The **View** toolbar controls your view of the model.

-  **Zoom to Fit**





	<b>Zoom to Area</b>
	<b>Zoom In/Out</b>
	<b>Rotate View</b>
	<b>Move View</b>
	<b>Wireframe</b>
	<b>Hidden in Gray</b>
	<b>Hidden Lines Removed</b>
	<b>Shaded</b>

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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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## Dimension

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, Dimension** 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**.

- 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
<b>Length of a line or edge</b>	The line.	
<b>Angle between two lines</b>	Two lines, or a line and a model edge.	If the lines are parallel, this creates a distance dimension.

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

### To change a dimension value:

- 1 Double-click the dimension value.
- 2 Type a new value, or click on the arrows to adjust the value.

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

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To dimension...	Click on...	Note
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<b>Distance between two lines</b>	Two parallel lines or a line and a parallel model edge.	
<b>Perpendicular distance from a point to a line</b>	The point and the line or model edge.	
<b>Distance between two points</b>	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, or click on the arrows to adjust the value.

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

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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.

**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 will be measured perpendicular to this plane.
- 2 Hold down the **Shift** 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 **Draft Outward**.
- 6 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.

### 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 **End 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 **Two Directions**, then click **Go To Direction 2** and repeat steps 3 through 5 for the other end.
- 7 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 **End 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 **Two Directions**, click **Go To Direction 2**, and repeat steps 2 through 6 for the other end.
- 8 Click **OK**.

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## Loft

Creates a feature that adds or removes material by connecting multiple cross sections.

### To create a loft feature:

- 1 Set up the planes needed for the cross sections. (You can use existing faces and planes, or create new planes. The planes do not have to be parallel.)
- 2 Sketch each section on a different face or plane.
- 3 Hold down the **Shift** key and select the sketches in order, picking a single corresponding point on each cross section. (The loft will connect the points you select.)
- 4 If you are creating the first feature in a new part, click **Insert, Base, Loft**. Otherwise, click **Insert, Boss, Loft** or **Insert, Cut, Loft**.

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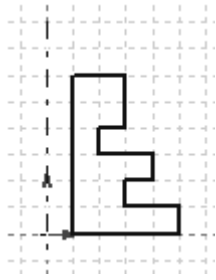
## Revolve

Creates a feature that adds or removes material by revolving a closed profile around a centerline.

### 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 Set the desired rotation angle in the **Angle** box.
- 4 The preview shows the direction of rotation. Select the **Reverse** check box if you want to rotate the feature in the opposite direction.



- 5 Click **OK**.

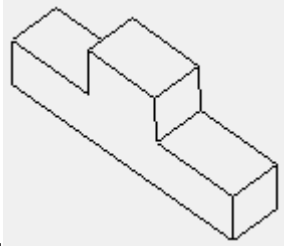
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## Shell

Creates a shell feature, removing selected faces.

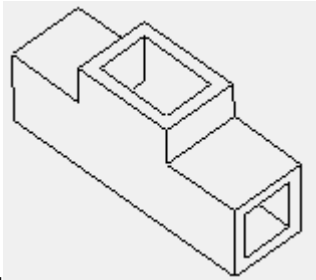
### To create a shell feature:

- 1 Click on the face you want to remove, or hold down the **Shift** 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**.

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## Hole


See [Hole Wizard](#) and [Simple Hole](#) .

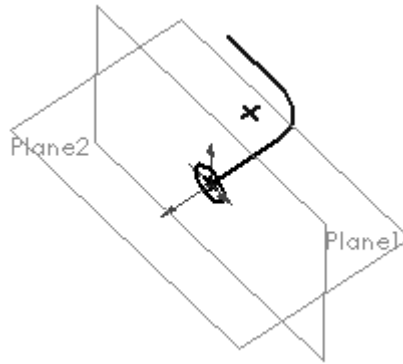
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## Sweep

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



- 4 Click  to close the sketch.
- 5 Hold down the **Shift** 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.**

**Note:** Do not create a sweep that intersects itself.

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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** , **Point-Plane Plane** , **Perpendicular Line-Point Plane** , **Line-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 Axis**, and then select the axis.
  - **Two Planes.** Select **View, Planes**, and then select two construction planes (while holding the **Shift** 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.

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## 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.

**Note** This 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.

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## 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 parameter as necessary.
- 3 Click **OK**.

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## 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**.


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## Rollback

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


### 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.)

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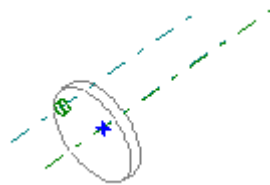
## Circular Pattern

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 shift-select the feature(s) to pattern.
- 4 Click **Insert, Pattern, Circular**.
- 5 Specify the **Spacing** and the **Number of Instances** of the feature.



- 6 Click **OK**.


See also [Linear Pattern](#) .

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## 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.

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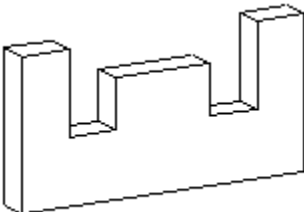
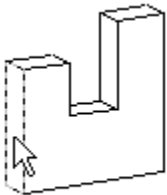
## Mirror

Creates a symmetrical part by mirroring an existing half around a planar face.

### To mirror a part around a planar face:

- 1 Select a planar face on a part.
- 2 Click **Insert, Boss, Mirror**.

A mirror image of the original part is joined to it at the selected face to make a whole part.



See also [Mirror Part](#) , [Mirror Feature](#) , [Mirror Sketch Elements](#)

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## Component

Adds a component to an assembly.

### To add a component:

- 1 Open an assembly window and the part or assembly that you want to add.
- 2 Move the assembly window so you can view both the assembly window and the part window.
- 3 Select **Insert, Component** and click in the part window.
- 4 Click in the assembly window where you want to locate the component.

See also [Move Component](#) and [Rotate Component](#) .

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## Add Mating Relationship

You can establish a mating relationship between the faces, edges, vertices, or reference planes of components. The valid mating relationships are:

- **Coincident:** positions selected faces, edges, and planes (in combination with each other or combined with a single vertex) so they share the same infinite line. Positions two vertices so they touch.
- **Perpendicular:** places the selected items at a 90 degree angle to each other
- **Tangent:** places the selected items so they meet and touch, but do not overlap
- **Concentric:** places the selected cylinders so that they share the same center point
- **Parallel:** places the selected items so they lie in the same direction and remain a constant distance apart from each other
- **Distance:** places the selected items with the specified distance between them

You can specify an alignment condition for the mating relationships. The conditions are:

- **Aligned:** places the selected items so the solids are on the same side of the plane
- **Anti-Aligned (On):** places the selected items so the solids are on opposite sides of the plane
- **Closest:** makes the items aligned or anti-aligned, based on which position they are closest to at the beginning.

See also [Mate](#) and [Edit Mating Relationships](#) .

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## 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.



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## 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 **Shift** 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](#) .

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## Bill of Materials


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

**Note** You must have Microsoft Excel installed on your computer in order to insert a bill of materials in a drawing.

### To insert a bill of materials:

- 1 With a drawing view selected, click **Insert, Bill of Materials**.  
A bill of materials is displayed that lists the parts in your assembly.
- 2 Insert any additional information or columns in the spreadsheet using the appropriate Excel commands and icons.
- 3 Click a corner of the spreadsheet to change its size.
- 4 Click anywhere outside the spreadsheet to close it.
- 5 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**.
- 3 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](#) .

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## Dimension

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, Dimension** 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**.

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

### To change a dimension value:

- 1 Double-click the dimension value.
- 2 Type a new value, or click on the arrows to adjust the value.

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


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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, 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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– or –

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

- 2 Click on the items to dimension, as shown in the table below.
- 3 Click again to place the dimension.

<b>To dimension...</b>	<b>Click on...</b>	<b>Note</b>
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<b>Distance between two lines</b>	Two parallel lines or a line and a parallel model edge.	
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### To change a dimension value:

- 1 Double-click the dimension value.
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See also **Horizontal Dimension, Vertical Dimension, Parallel Dimension, Baseline Dimension, Ordinate Dimension, Horizontal Ordinate Dimension, Vertical Ordinate Dimension.**

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<b>Distance between two lines</b>	Two parallel lines or a line and a parallel	

	model edge.	
<b>Perpendicular distance from a point to a line</b>	The point and the line or model edge.	
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### To change a dimension value:

- 1 Double-click the dimension value.
- 2 Type a new value, or click on the arrows to adjust the value.


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

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## 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**.

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## 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.

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


See also [Detail Circle](#) .

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## 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.

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## 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.

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## 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](#).

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## 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.

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