SolidWorks® 95
User's Guide

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Preferences

Allows you to customize SolidWorks options and set default values.

To set preferences:

1 Click **Tools**, **Preferences**.
2 Click on the appropriate tab (**General**, **Color**, and so on) for the settings you want to change.
3 Change the settings and click **OK**.

See **General, Color, Edges, IGES, Performance,** and **Grid** preferences.
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Toolbars

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

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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 SKETCH
    MODE 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 Select the sketch you want to edit or a feature built from the sketch, either in the model or in the FeatureManager.

>   **Note** For features built from multiple sketches, (lofts and sweeps), right-click on the sketch name in the Feature Manager.

2 Click ![edit sketch icon] or **Edit, Sketch**.

3 When you are finished editing, click ![edit sketch icon] or **Edit, Sketch**, or right-click anywhere in the sketch and click **Exit Sketch.**

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Dimension

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

1 With a sketch active, click ![dimension icon] or **Tools**, **Sketch Relations**, **Dimension**.

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:

4 Double-click the dimension value.

5 Type a new value, or click on the arrows to adjust the value.

See also Horizontal Dimension, Vertical Dimension, Reference Dimension, Baseline Dimension.

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Top

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 **Reverse Draft Direction**, if necessary.

6 To extrude the feature in both directions from the sketch plane, select **Two Directions**, then click **Go To End 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 **Reverse Draft Direction**, if desired.

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

8 Click **OK**.

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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.)

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

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

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

4 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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Sweep

Creates a feature that adds or removes material by projecting a profile along a path.

To create a sweep:

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

2 Click ![icon] to close the sketch.

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

The path can be an open or closed curve, but the start point of the curve must lie on the plane of the profile. If more than one curve is used in the path, all segments of the path must be tangent to one another and the path must be perpendicular to the plane of the profile at its start point.

4 Click ![icon] 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**.

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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.

Note Each section must have the same number of curves.

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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Hole

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

Hole

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

Fillet

Creates a tangent arc at the intersection of two sketch elements, trimming away the corner.

To create a fillet in a sketch:

1 Hold down the **Shift** key and select two sketch elements that form a corner.

2 Click ⬚ or **Tools**, **Sketch Tools**, **Fillet**.

3 Enter a fillet radius value.

4 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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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 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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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**.

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Axis

Creates an axis in a part or assembly.

To create an axis:

1   Click **Insert**, **Axis**.
2   Click on a model edge, or hold down the **Shift** key and select two vertices.

   Note   Axis display must be turned on (see Axes) to see the new axis.
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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, then click **Edit, Definition**.
   Shortcut   Right-click the feature or its name, then click **Edit Definition.**
2   Change the parameters as necessary and click **OK**.

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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 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.

3 Add new features or edit existing features while the model is in the rolled-back state.

4 If the changes you made did not cause the model to rebuild, click or **Edit**, **Rebuild**.

Design Table

Allows you to build families 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.

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 will appear in the part document, and the Excel menus and toolbars will 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 display the other variations, click on the box that appears above the FeatureManager and select a variation from the drop-down list.

To edit a design table:

1 Click **Edit, Design Table.** The table will appear, and the Excel menus and toolbars will replace the SolidWorks menus and toolbars.

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 will not be reflected in the original file.

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Line

Creates a line.

To create a line:

1 Click or **Tools**, **Sketch 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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Circular Pattern

Creates multiple instances of a feature spaced evenly around a center point.

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

To create a circular pattern:

1 Right-click on the feature that you want to replicate and click **Edit Sketch**.

Note   You need to specify

- a point to serve as the center of rotation for the pattern, and

- an angular dimension that rotates the feature around the center point.

Steps 2 through 8 below describe one way to do this, but you can use any method that produces a center point and a driving dimension.

2 Create a center point for the pattern. You can select an existing point or create a point with the **Point** tool, then either

- dimension the point to fix its location, or

- use the **Add Relation** command to apply a **Fix** relation to it.

3 Sketch two centerlines with the **Centerline** tool. One of the centerlines must be horizontal or vertical, and the other must *not* be horizontal or vertical so it is free to rotate.

4 Create a **Coincident** relation between the center point and each of the centerlines.

5 Create a **Coincident** relation between the *diagonal* centerline and a point on the profile of the feature. For instance, if the profile is a circle, make the center of the circle coincident with the centerline.

6 Create a linear dimension between the center point and the same point on the profile of the feature.

7 Create an angular dimension between the two centerlines, then try incrementing the value of the angular dimension to verify that it rotates the profile around the center point. If it does not, check to make sure that the profile has no horizontal, vertical, or fixed relations.

8 Click ![traffic light icon] or **Edit**, **Rebuild**.

9 Double-click the feature's name in the FeatureManager to display its dimensions.

10 Click **Insert**, **Pattern**, **Circular Pattern**.

11 Click on the angular dimension you created. An **x** will appear in the **Dimension Selected** box.

12 Specify the spacing between instances (**Spacing**) and the total number of instances including the original (**Total Instances**).

13 Click **OK**.

Mirror

Creates copies of sketch elements which are mirrored around a centerline.

When you create mirrored elements, SolidWorks applies a symmetric relation between each corresponding pair of sketch points (the ends of mirrored lines, the centers of arcs, and so on). If you change a mirrored element, its mirror image will also change.

To mirror existing items:

1 In a sketch, click ![centerline icon] and draw a centerline.

2 Hold down the **Shift** key and select the centerline and the items you want to mirror.

3 Click ![mirror icon] or **Tools**, **Sketch Tools**, **Mirror**.

To mirror items as you sketch them:

1 In a sketch, click ![centerline icon] and draw a centerline.

2 Select the centerline.

3 Click ⏃ or **Tools**, **Sketch Tools**, **Mirror**. M's appear at both ends of the centerline to indicate that automatic mirroring is active.

4 Sketch curves. The elements you draw will be mirrored automatically as you draw them.

5 To turn mirroring off, click ⏃ again.

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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.

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Mate Align

Mates two components in an assembly with the selected faces, planes, or edges aligned.

To align two components:

1 In an assembly, select the two faces or planes, or the two edges you want to mate.

2 Click **Insert**, **Mate Align**.

If you selected *planar faces*, the components are oriented so both faces lie on the same plane, with both components on the same side of the plane. (The components do *not* face each other.)

If you selected *cylindrical faces*, the axes of the cylindrical features are aligned.

If you selected *edges*, the components are oriented so the selected edges are colinear.

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Mate Offset

Mates two components in an assembly with the selected faces parallel and a specified distance apart.

To align two components with an offset:

1 In an assembly, select the two planar faces or planes you want to mate.

2 Click **Insert**, **Mate Offset**.

3 Specify the **Offset Distance**.

4 Click **OK**.

5 To adjust the orientation of the components, right-click the mating relation (paper clip icon) in the FeatureManager and click **Edit Definition**. Then select **Flip Offset** or **Flip Component** and click **OK**.

Mate On

Mates two components in an assembly so the selected faces lie on the same plane and the components are on opposite sides of the plane.

To align two components facing each other:

1 In an assembly, select the two planar faces or planes you want to mate.

2 Click **Insert**, **Mate On**.

Dimension

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

1 With a sketch active, click or **Tools**, **Sketch Relations**, **Dimension**.

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.
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To change a dimension value:

4 Double-click the dimension value.

5 Type a new value, or click on the arrows to adjust the value.

See also .

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See also Horizontal Dimension, Vertical Dimension, Reference Dimension, Baseline Dimension.

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 **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.

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 **Insert**, **Drawing View**, **Auxiliary**.

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**, **Sketch Tools**, **Spline**.

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

4   Click and select the spline.

5   Click **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, then press **Enter**.

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 **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.

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**, **Sketch Tools**, **Line** and draw the section line or a series of connected lines.

3   Click ⬜ and select the sketched line(s).

4   Click **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**.

Note   You cannot edit section lines by dragging them, but you can dimension them and dimension them to other elements.

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

To change the section line label or the direction of the view, select the line and click **Edit**, **Properties**.