DesignWave Help

Finding Help

For contents, an index, and keyword searches, press this button.

{button Help Topics,FD()}

Context-Sensitive Help

See *Using Context-Sensitive Help* for how to get context-senstive help on commands, messages, and dialog windows.

This is before the first topic, so it will not get displayed

Notes can go here.

Shows you a tip for more effective use of DesignWave.

Check this box if you want to be offered tips each time you start DesignWave.

Press this button to be shown another tip.

Enter the name for the feature to be created. The name is used in the feature browser.

Enter the name for the feature to be created. The name is used in the feature browser.

Shows the units to be used for parts and assemblies.

Enter the name for the datum that should appear in the label. Typically this is a single letter.

Select the style of datum label you want.

User name.

Company name.

(Static) > The total amount of physical memory, or RAM, that you have on your computer.

(Static) > Amount of free disk space on your computer.

The name of the feature. This is used when the feature is displayed in the feature browser.

This is checked if the feature is suppressed.

Checked if the line is a construction line. Otherwise it is a firm line.

The start point of the line. The coordinates are in terms of the work axes.

The end point of the line. The coordinates are in terms of the work axes.

The start terminator of the line.

The end terminator of the line.

Press this to swap the start and end terminators.

Shows you the progress while updating the design. To interrupt the update, press the Stop button.

Press this button to continue the update after it has been stopped.

Shows the units to be used for distances in the drawing sheet, such as lines created in the drawing.

Select this option if you want the text to be in-line so as to break the dimension line, center line or extension line.

Select this option if you want the text to be above the dimension line, extension line, or extension line. If the line is vertical, the text is placed to the left.

Select this option if you want the text to be below the dimension line, extension line, or extension line. If the line is vertical, the text is placed to the right.

Select this option if you want the text to be shown horizontal, even if the dimension line, extension line, or center line on which it is placed is not horizontal.

Select this options if you want the text orientation to be along the dimension line, extension line, or center line.

Select this option if you want the text to be attached directly to the dimension line.

Check this box if you want the text to be centered between the arrowheads of the dimension line.

Select this option if you want the text to be connected to the dimension line using a shoulder which is drawn to the left of the dimension line. The dimension line must be more than 15 degrees from horizontal for this option to have any affect.

Select this option if you want the text to be connected to the dimension line using a shoulder which is drawn to the right of the dimension line. The dimension line must be more than 15 degrees from horizontal for this option to have any affect, You can enter the length of the shoulder here.

Select a terminator here if you want an additional terminator to be drawn on the dimension line on the side of this callout group, for example, to cope with ISO drafting standards.

This list shows you all the part windows that are currently open. Select one of them to add a view to the drawing which is a snap shot of the view in that part window.

Select the sketch to extrude. This is initially set to the active sketch.

Select this option if you want the extruded profile to be added to the solid material already in your part. If your part is currently empty, this is the only option.

Select this option if you want the extruded profile to be subtracted from the solid material already in your part, for example, to produce a pocket.

Select this option if you want the extruded profile to be intersected with the solid material already in your part.

Enter the extrusion distance. You can use previous values entered and values created with the Measurement command on the drop list.

Select this option to extrude the profile above the active workplane. You are looking at the top of the workplane if the work axes arrowheads are filled.

Select this option to extrude the profile below the active workplane. You are looking at the bottom of the workplane if the work axes arrowheads are hollow.

Select this option if you want the profile to be extruded half the distance above the workplane, and half below the workplane. If you have a taper angle, the workplane will act like a parting line.

If you want a taper angle, enter a value here. A positive angle tapers inwards so that the outer profile shrinks in the extrusion direction, and inner profiles expand.

Enter a name for the new workplane. The name is used when the workplane is shown in the browser.

For an offset workplane, enter the offset distance here. If you are offsetting from a workplane, the offset is above that workplane. If you are offsetting from a planar face, the offset is away from the model.

For an oriented workplane, enter the angle from the active workplane here.

Choose a display color for lines created in the sketch.

Choose a name for the new sketch. The name will be used in the browser.

Check this box if you want a workplane with no sketch. You might do this if you are using the workplane as an intermediate construction for a further workplane.

Enter the name for the feature. The name is used when the feature is shown in the feature browser.

Enter the blend radius. The blend is created as if you were rolling a ball along the intersection of the two faces that meet at the edge. The value is the radius of such a ball.

Check this box if you want a chamfer, rather than a blend. The chamfer is defined the same way as a blend, but the surface cuts across chordally, rather than being round. Note that for faces not meeting at 90 degrees, this means that the value entered is not the same as the set-back distance.

Check this box to have other edges that are smooth continuations of the selected edges added to the edges to be blended.

This list shows all your sketches. The check mark shows whether the sketch is visible. Select a sketch from the list and press the Activate button to make it the active sketch.

Press this button to hide all sketches apart from the one selected in the list.

Press this button to create a new sketch.

Press this button to examine or modify the properties of the sketch selected in the list.

If the sketch is not “in use”, you can press this button to delete the sketch and any lines in it. If the sketch is “in use” you can press the Properties button to see how many times it is used.

Check this box to list all sketches. Clear the box to list only sketches that have some lines in them.

Choose a display color to be used for lines created in the sketch.

Enter a name for the new sketch. The name is used wherever the sketch is shown in the browser, and wherever lists of sketches are presented in dialog windows.

Select a workplane for the sketch. Initially this is set to the active workplane.

Press this button to bring up the Windows calculator so that you can perform some calculation. You can use Copy and Paste to bring the resulting value back to this dialog window.

Enter the radius to be used for fillets you are about to create.

Select the sketch that defines the section cut. Initially this shows the active sketch.

Select this option if you want a sectional view. You will see all of the model that lies behind the section.

Select this options if you want a cross section. You will only see where the model is cut, and not anything behind the section cut.

The tree shows the product structure. Double-click on an icon to expand or collapse that node in the tree. The asterisk and minus keys on the numeric keypad can be used to fully expand or collapse a complete branch of the tree.

Select this option to see the components of the part selected in the browser.

Select this option to see a complete list of parts used by the part selected in the browser.

Check this box to include components that are currently hidden.

Press this button to save the current report to a file. The information is saved as a Comma Separated Value (CSV) file, which is suitable for reading into spreadsheet applications.

This shows the file name of the part selected in the browser.

Press this button to examine the mass properties of the selected part.

The measurement is shown here, and previous values are shown in the list. Press the Add button to add this value to the list of recent values. This list is available as a drop list in many dialog windows, such as feature creation dialogs.

Select the type of measurement from the enabled options presented.

Press this button to display the Windows calculator. The current measurement is automatically pasted into the calculator so you can perform a calculation. Use Copy and Paste to bring the result back from the calculator and into the value field, before pressing the Add button.

Enter a name for the hollow feature. The name is used wherever the feature is shown in the browser.

Enter the wall thickness here. The hollow is performed inwards, so that the existing faces become the outer walls.

Enter the offset distance here.

Select this option to offset inwards. This option is only available if the line chain is a closed loop.

Select this option to offset outwards. This option is only available if the line chain is a closed loop.

Select this options to offset on both sides of the line chain. If the line chain is not a loop, this is the only option, and you may have to delete the side you did not want.

Press this button to bring up the Windows calculator so that you can perform a calculation. Use Copy and Paste to return the result to this dialog window.

Enter the number of objects in the X direction. This number includes the original. The X direction is defined by the work axes.

Enter the number of objects in the Y direction. This number includes the original. The Y direction is defined by the work axes.

Enter the spacing distance in the X direction.

Enter the spacing distance in the Y direction.

Choose the new workplane for the sketch. Only workplanes that are not positioned in terms of geometry generated from the sketch are presented.

The list shows the search directories in the order in which they will be searched.

Press this button to add another search directory to the list.

Press this button to remove the selected directory.

Press this button to move the selected directory up one place in the list.

Press this button to move the selected directory down one place in the list.

Select this option if you want to enter a single line of text.

Select this option if you want to enter a multi-line note.

Check this box if you want your text to be shown with a box around it.

Enter a single line of text here.

Enter a multi-line note here.

Select the type of geometric tolerance you want to be applied to the feature. According to what you select, the material condition and datums will be enabled.

Enter the required tolerance here.

Select this option if you want the tolerance to apply at maximum material condition.

Select this option if you want the tolerance to apply at least material condition.

Select this option if you want the tolerance to apply regardless of feature size.

Select the primary datum. The list offers you features that have been labeled. You have to have added a datum label for the feature before you can use it as a datum.

Select the secondary datum. The list offers you features that have been labeled. You have to have added a datum label for the feature before you can use it as a datum.

Select the tertiary datum. The list offers you features that have been labeled. You have to have added a datum label for the feature before you can use it as a datum.

Select the material condition for the primary datum. This options are disabled if the datum feature is not a feature of size.

Select the material condition for the secondary datum. This options are disabled if the datum feature is not a feature of size.

Select the material condition for the tertiary datum. This options are disabled if the datum feature is not a feature of size.

Check this box if you want the primary and secondary datum features to be treated as a common datum. This is typically done for runout tolerances where the axis of rotation is set up using two co-axial features on the part.

Select the tolerance display style for the dimension. The dimension carries a tolerance even if it is not currently displayed. The inline plus-minus tolerance style displayed the positive tolerance.

Enter the permitted positive dimensional tolerance. The positive tolerance controls the maximum permitted dimension.

Enter the permitted negative dimensional tolerance. The negative tolerance controls the minimum permitted dimension.

Check this box if you want a basic dimension. The tolerance values are ignored and the dimension text is shown with a box around it.

Check this box if you want a reference dimension. The tolerance values are ignored and the dimension text is shown in parentheses.

Choose the units in which you want the number to be displayed on the drawing.

Select the number of decimal places to be used when displaying the number.

Check this box if you want do not want the leading zero to be displayed for numeric values less than one.

Check this box if you want any trailing zeros to be suppressed. If this box is checked, the decimal places is a maximum number. If this box is cleared, you will always see the specified number of decimal places, even if they are zeros.

Check this box if you are displaying a dimensional tolerance and your drafting standard says you should not show the plus or minus sign for a zero tolerance.

Check this box if you want to use a comma for the decimal separator, instead of a point.

Check the boxes for the styles or groups of styles that you want to pick up. These headings come from the labels used on the Placement page of the Properties dialog window.

Check the boxes for the styles or groups of styles that you want to pick up. These headings come from the labels used on the Number page of the Properties dialog window.

Check the boxes for the styles or groups of styles that you want to pick up. These headings come from the labels used on the Geometry page of the Properties dialog window.

Select this option if you want the leader to be drawn from the left side of the callout group.

Select this option if you want the leader to be drawn from the right side of the callout group.

Enter the length of the shoulder of the leader. The shoulder is the short horizontal line drawn from the callout group.

Check this box to remove all jog points from the leader. To add a jog point, drag the leader from its shoulder point.

Choose the terminator you want for the leader.

Select this option when the leader points to a feature that has an axis or center, and you want the leader to be drawn to the surface, rather than the axis or center.

Select this option when the leader points to a feature that has an axis or center, and you want the leader to be drawn to the axis or center, rather than the surface of the feature.

Check this box if you want the leader to be drawn so as to terminate on the feature perpendicular to it.

If the leader points to a dimension line, you can select an additional terminator to be drawn on the dimension line on the side nearest the termination of the leader. This is common for geometric tolerances and datum labels according to the ISO drafting standard.

Check the boxes for the styles or groups of styles that you want to pick up. These headings come from the labels used on the Leader page of the Properties dialog window.

Check the boxes for the styles or groups of styles that you want to pick up. These headings come from the labels used on the Dimension page of the Properties dialog window.

Check the boxes for the styles or groups of styles that you want to pick up. These headings come from the labels used on the Tolerance page of the Properties dialog window.

Enter a name for the revolved profile feature. The name is used when the feature is shown in the browser.

Select the sketch that defines the profile to be revolved. Initially, this shows the active sketch.

Select the sketch that defines the axis of revolution for the profile. The sketch must be another sketch in the same workplane as the profile sketch, and must contain a single straight line.

Select this options if you want the revolved profile to be added to the material already in your model. If you currently have an empty part, this is the only option.

Select this options if you want the revolved profile to be subtracted from the material already in your model.

Select this options if you want the revolved profile to be intersected with the material already in your model.

Select the sketch that contains the profile to be projected. Initially this shows the active sketch.

Select this option if you want the profile to be projected above the workplane. You are looking at the top of the workplane if the arrowheads of the work axes are shown filled.

Select this option if you want the profile to be projected below the workplane. You are looking at the bottom of the workplane if the arrowheads of the work axes are shown hollow.

Select this option if you want the projected profile to be added to the material already in your model. For this option your profile should not be positioned inside the part.

Select this option if you want the projected profile to be subtracted from the material already in your model. For this option your profile should not be positioned outside the part.

This field shows the extrusion distance. You can change this value to edit the feature.

This field shows the taper angle applied in the extrusion direction. A positive angle tapers inwards so that the outer profile shrinks in the extrusion direction, and inner profiles expand. You can change this value to edit the feature.

Enter a name for the feature to be created. The name is used when the feature is shown in the browser.

Select the sketch containing the profile that you want to sweep. Initially this shows the active sketch.

Select the sketch containing the path along which you want to sweep the profile. Only sketches in workplanes that are perpendicular to the profile are offered.

Select this option if you want the swepth profile to be added to the material already in your model.

Select this option if you want the swept profile to be subtracted from the material already in your model.

Select this options if you want the swepts profile to be intersected with the material already in your model.

This is the current blend radius. You can change this value to edit the feature.

This box tells you whether the feature is a chamfer, rather than a blend. You can change this setting to edit the feature.

This box tells you whether other edges that are smooth continuations of the originally selected edges are also required to be blended. You can change this setting to edit the feature.

This is the wall thickness for the hollow feature. You can change this value to edit the feature.

The view scale is used for all views added to the drawing that are not scaled. The scale factor is paper divided by model. For example, a Paper:Model ratio of 1:2 would give you half scale views.

Select this option to use an existing format. A format is just another drawing, usually containing just lines and text. The size of the chosen format is copied, but you can then change it for this drawing, if you want to.

Enter the full path name of the format file. The format file is an existing drawing, usually containing just lines and text.

Press this button to bring up a file browser so you can locate the format file you want.

Select this option if you do not want to use an existing format file, so you can specify the size of the drawing.

You can choose a standard paper size from this list, or you can just enter the width and height in the fields below.

This shows the width of the drawing paper.

This shows the height of the drawing paper.

This shows the angle of the section hatching lines from the horizontal. You can change this value.

This shows the separation of the section hatching lines. You can change this value. Warning: if you enter a large number such that no hatch lines are displayed within the area to be hatched, you will no longer be able to select the hatch lines.

You can adjust the phase of the section hatching lines so as to avoid adjacent hatched areas appearing as continuations. This is recommended by many drafting standards.

Shows the color used to display lines that belong to this sketch. You can change this color so that lines in the sketch stand out from lines in other sketches. The color has no other significance.

This is the name of the sketch, which is used when the sketch is shown in the browser. You can change this name.

This tells you whether the sketch is currently visible. You can change this setting to hide or show the lines in the sketch.

The number here tells you how many times the sketch has been used. If a sketch has been used to define a feature, or has been used to define the cuts for a section view on a drawing, these count as a usage. A sketch that is “in use” cannot be deleted.

This shows the workplane to which this sketch belongs. You might want to move the sketch to another workplane in order to move a feature to a different face.

Press this button to move the sketch to a different workplane.

Select this option to activate the set of controls in the section of the dialog window to the right of this button.

This shows the start point of the line. If you prefer to think of the line the other way round, press the Reverse button.

This shows the end point of the line. If you prefer to think of the line the other way round, press the Reverse button.

This shows the bend angle of the line. A straight line has an angle of zero. A semi-circle has an angle of 180 degrees.

This shows the distance between the start and end points. For an arc, this is the chordal length.

Select this option if you want to enter a new length for the line. The start point will stay fixed and the end point will move to achieve the length.

Select this option if you want to enter a new length for the line. The end point will stay fixed and the start point will move to achieve the length.

Select this option if you want to enter a new length for the line. The mid-point will stay fixed and the end points will move to achieve the length.

For a line that is not straight, this shows the center of the line. You can enter a new center to re-position the line.

For a line that is not straight, this shows the radius of the line. You can enter a new radius to scale the line.

For a line that is not straight, this show the start angle of the line. The angle is measured anti-clockwise from the X direction of the work axes.

For a line that is not straight, this show the end angle of the line. The angle is measured anti-clockwise from the X direction of the work axes.

Enter the number of copies you want here. The number includes the original.

Enter the angle here. This is either the total angle, or the angle between copies, according to the option selected. The angle is measured about the origin of the work axes.

Select this option if the angle entered represents the total angle between the original and the last copy. If the angle is 360 degrees, choose this option to the original and the copies spaces equally around a complete circle.

Select this option if the angle entered represents the angle between each copy.

Enter the translation vector in the workplane here. The vector is defined in terms of the orientation of the work axes.

Enter the rotation angle here. The rotation is anti-clockwise about the origin of the work axes.

Enter the scale factor here. The scaling takes place about the origin of the work axes.

This shows the name of the workplane. The name is shown in the browser. You can edit this name.

For an offset workplane, this shows the offset distance. You can edit this value to change the offset.

For an oriented workplane, this shows the angle of orientation. The angle is about the edge used to define the workplane, and is measured from the plane of what was the active workplane at the time of construction.

Enter a name for the feature being created. The name is used when the feature is shown in the browser.

Select this option if you want the component to be added to your model as a solid feature.

Select this option if you want the comonent to be subtracted from your model.

Select this option if you want the component to be intersected with your model.

The feature will be created in your model, and the component used to define it will be hidde. Clear this box if you do not want the component to be hidden.

Check this box if you want to see hidden edges shown dashed. Clear this box if you do not want to see hidden edges at all.

The view scale is taken from the drawing sheet properties. Check this box if you want to override the view scale, for example, to create an enlargement.

Enter the zoom factor here. A value greater than one will enlarge the view. A value less than one will shrink the view.

For a section view, select this option if you want a sectional view. You will see all of the model that lies behind the section.

For a section view, select this option if you want a cross section. You will only see where the model is cut, and not anything behind the section cut.

Select this option to create a partial view. Then select the sketch that contains the clipping polygon from the list offered.

Select the sketch that defines the clipping profile. The profile must be a closed profile consisting only of straight lines.

This shows the name of the part.

This shows the volume of the part. For an assembly, this includes the volume of all components.

This shows the surface area of the part. For an assembly, this includes the surface area of all components.

This value tells you the accuracy of the physical property it follows.

These settings affect the measurement of a linear dimension between features that have a radius. You can change the setting to change the measurement.

These settings affect an angular dimension. You can change the setting to change the measurement.

These settings affect a radial or diametric dimension. You can change the setting to change between a radial and diametric measurement.

This is the text height used in the callout group. You can change this value to scale the text.

Select the pen thickness you want to use for the text. The thickness only affects the plotted drawing. It has no affect on the text as you see it on the screen.

Check this box to have centerlines drawn to the boundary of the circular features. Clear this box if you only want the centerpoint.

Clear this box if you do not want to display the centerpoint.

For centerlines passing through features that have a center or axis, check this box to have perpendicular centerlines drawn through that center or axis too. Clear the box if you do not want perpendicular centerlines.

For an extension line from a feature that was split by the application of another feature, check this box to have the extension line drawn between the fragments of the feature to indicate that the fragments as a set are referenced.

For a linear dimension, check this box to show a centerline for the mid-plane of the separation of the two features.

Press this button if the start and end points are the other way round from the way you think of the line.

Press this button when you have made some changes to the enabled section of the page and you would like to see the affect on the other two sections.

This message explains the problem.

This is the message area. It tells you what is happening and explains update problems to you.

This file is used by the Dialog Box Help Editor. It provides a topic that is used to alias topic IDs that should display no Help.   Include this file in the FILES section of the Help Project (.hpj) file.