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.

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Introduction to Assembly Modeling

In DesignWave, working with an assembly is the same as working with a part. An assembly is simply a part that references other parts that, in turn, may reference other parts. The component and product structure browsers provide a logical breakdown of the structure of your assembly. To enhance visibility, you can hide components or change their color. With mating conditions, you can constrain the position of a component relative to the other components in the assembly. These relative positions are maintained as components move, change size, or change shape.

Designing in Teams

With DesignWave, you can design in teams. That is, you can work on the assembly while others work on the individual piece parts that the assembly references. You can work on a piece part by itself or in the context of an assembly. When a part is opened in context. All of the other parts in the assembly are displayed in a light gray color. You can use their geometry to add new features to the open part. When you change a part (component), those changes are automatically propagated to the assembly. If you have the assembly open, you see your changes immediately. If not, you will see your changes the next time you open the assembly.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Adding Components

When you add a component to an assembly, the contents of the component part file are not copied into the assembly. Instead, the assembly references its contents. When you make changes to that component, those changes are automatically propagated back to the assembly. You cannot add an assembly as a component to itself.

Components are listed in the Component Browser.

Component Names

The name of the component is the name of the part. You cannot change the name of a component. You can add the same component more than once. If you add the same component more than once, a number is appended to the component name. This number distinguishes one occurrence of a component from the others.

Component Placement

When you add a component to an assembly, it is initially placed so that its base workplane coincides with the active workplane. To initially place a component at a specific location, you can create a workplane at that location. Then, you can move that component to a different location by translating or rotating it or by defining mating conditions.

Search for Part Files

If, after you added the component, you decide to move its part file to another directory, the assembly can still reference it. If you move its part file to another directory, the assembly first looks for it in the directory where the assembly resides and then in the directories listed in your search path.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Creating Mating Conditions

A mating condition is a relationship between two components. The order in which you select the faces or edges does not determine which component moves in order to satisfy the mating condition. The younger of the two components always moves. Age is determined by the order in which you added the components to the assembly.

Mating conditions let you precisely constrain the position of components relative to the other components in the assembly. These relative positions are maintained as components move, change size, or change shape. A component is fully constrained if you cannot move or rotate it in any direction without violating a mating condition. Especially with complex assemblies, it can be very difficult to fully constrain a component's location. You can choose to not constrain, partially constrain, or fully constrain a component's location.

Mating conditions are listed in the Component Browser. They are shown as small green clamp icons underneath the icon of the component they constrain. Commands on the popup menu let you delete that mating condition or highlight the geometry used to define it.

Types of Mating Conditions

There are four different types of mating conditions. Mate and Align Planes make the planar faces that belong to two different components coincide with each other. A face has two sides. How the components line up with respect to each other depends on the command chosen. With **Mate Planes**, the components oppose each other. With **Align Planes**, they face in the same direction. Mate and align are opposites. If **Align Planes** does not position a component as you expect, **Mate Planes** likely will. You cannot offset either mated or aligned faces from each other.

Center Axes aligns circular holes or features that belong to two different components. With **Center Axes**, the axes of the circular holes or features will line up with each other. **Orient Axes** lets you rotate the components about their axes. For example, to line up a keyway, the components are rotated so that two selected edges become parallel.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Using the Component Browser

The component browser provides a logical overview of the structure of your assembly.

In the Component browser, components are shown with the small yellow rivet icons. The name of the component is listed after the icon. Except for the first occurrence, a number follows the component name. This number distinguishes one occurrence of a component from the others. The first occurrence never has a number. If the component is hidden, its icon is shown in an outline form.

Mating conditions are shown with the small green clamp icons underneath the icon of the component that they constrain. The type of mating condition (Mate, Align, Center, or Orient) is listed after to each icon. From the popup menu, you can delete that mating condition or highlight the geometry used to define it.

From the popup menu, you can select or hide a component. The **Select** command selects that component in the part window. You may even select a hidden component. The **Hide** command toggles the display of a component on and off in the part window. A check mark appears next to the **Hide** command if the component is hidden.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

 # Moving Components

In addition to creating mating conditions, you can place components in the assembly by simply moving or rotating them. You move or rotate components in the same way you move or rotate any other object in DesignWave; that is, in any direction along the active workplane.

You can move or rotate components that have mating conditions only in directions that do not violate any of those conditions.

See Also

{button ,AL(`assemblyModeling')} *Other topics on assembly modeling*

Opening a Design in Context

As you work on a part, the ability to reference features from other components may facilitate the design of that part. The **Open Part in Context** command on the **Assembly** menu lets you open a part and work on it in the context of an assembly. With a part open in context, you can reference geometry from other components as you build new features.

See Also

{button ,AL(`assemblyModeling')} *Other topics on assembly modeling*

Working on a Design in Context

For a part open in context, all of the components including the open part are in the same locations as in the assembly. The view is the same. All of the unopened components are displayed in a light gray color to distinguish them from the geometry that belongs to the open part. The browsers display the features, workplanes, and components that belong to the open part - not those that belong to the assembly.

The part itself can be modified, while the rest of the assembly is read-only. You can reference the edges and faces that belong to other components just as if they belonged to the open part, but you cannot change them. Nor, can you move any of the components. But, whenever you change the original assembly, those changes are automatically reflected in the open part.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Using Context Geometry

With a part open in context, you can reference the edges and faces that belong to other components as you design new features. You can use them just as if they belonged to the open part, but you cannot change them.

You can use the **Measure** command on the **Tools** menu to measure the dimensions of other components, and then use those measurements in the design of new features. Using a face on another component, you can construct a workplane. As you sketch, attraction points on edges that belong to other components highlight. Any reference to geometry that belongs to another component is not associative.

For example, you are designing a spur gear that fits around the shaft on another component. You may want to create a workplane using the flat face on the end of the shaft. As you sketch the profile for gear, the attraction point for the center of the shaft will highlight and you may select it as the center for the hole in the gear. With the **Measure** command, you may determine the diameter of the shaft and set the diameter of the hole in the gear equal to it.

See Also

{button ,AL(`assemblyModeling')} *Other topics on assembly modeling*

Creating a New Part in Context

You can use the **Open Part in Context** command to open any component and work on it in the context of your assembly. The command works with any component regardless of whether or not it contains geometry. If you create a new, empty part and add it to your assembly, you can open and work on it in the context of your assembly.

When you open a new part in the context of an assembly, its base workplane coincides with the workplane that was active when you added that part to your assembly. In a part, before you add a new feature, you decide where to place that feature and you create a workplane at that location. Likewise, before you add a new part to your assembly, decide where to place that part and create a workplane at that location. When you open your new part in context, its base workplane will coincide with that workplane from your assembly.

For example, assume that your new part mates with a face on another component. To facilitate your design, you can make the plane of that face the base workplane for your new part.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Changing the Color of a Part

The **Set Component Color** command on the **Assembly** menu lets you customize the appearance of your assembly. When you add a new component, DesignWave assigns it a color. You can, however, change the color to visually convey information. For example, you can assign colors based on a component's material type or function. Or, you can assign colors to emphasize important components.

All occurrences of a component in an assembly have the same color. When you change the color of one occurrence of a component, the color changes in all occurrences.

For a subassembly, a component comprising other components, you can change the color of the individual base components.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Hiding Components

A complex assembly may contain many components. Some of those components may obscure your view or interfere with your access to other components. You can simplify your assembly by turning off the display of those components. The **Hide** command on the popup menu toggles the display of a component in the part window on and off.

An assembly may also contain complex components. You can sometimes hide these components without affecting what you are working on. By hiding complex components, you will improve system performance and make more efficient use of system resources.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Browsing the Product Structure

The product structure browser provides a detailed logical view of the structure of your assembly. Like the Windows Explorer, it has two windows. The left window displays a hierarchical breakdown of your assembly. The right window displays either a parts-list view or a bill-of-materials view of the component selected in the left window.

In the left window, you can examine any piece of your assembly structure in any degree of detail. You can examine and edit any component, or query its mass properties. This lets you quickly focus on those aspects of the assembly that are important to you.

The right window displays any component in your assembly from one of two different perspectives. The parts-list view lists those components that you added in creating the selected component. The bill-of-materials view lists those components necessary to build the selected component. These are the components that contain geometry not those that are simply composed of other components.

The name, quantity, and file name are displayed for each component. You can sort the list of components by name, quantity, or file name. You can, optionally, exclude hidden components from the list or save the list of components as a comma-separated value file that you can later input into a spreadsheet application.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Analyzing Your Assembly

Measuring Lengths and Distances

With the **Measure** command on the **Tools** menu, you can determine the length, radius, or diameter of an edge on any component, or measure the distance between two faces or edges on two different components. You can also measure the distance from the active workplane or work axes to an edge or a face on any component.

You can use these measurements as you design new features on other components. As you perform a measurement, add it to the drop-down list on the **Measure** command dialog. You can access this measurement from the drop-down lists on other command dialogs, for example, extrusion distance on the **Extrude Profile** command dialog.

Calculating Mass Properties

You can query the mass properties (volume and surface area) for the open part or any component that it contains. To query the mass properties for the open part, choose the **Mass Properties** command on the **Assembly** menu with no components selected. From the product structure browser, you can query the mass properties for any component contained in your assembly.

Understanding How Components Were Assembled

To understand how the components in an assembly fit together, examine their mating conditions. Mating conditions constrain the position of a component relative to the other components in the assembly. They are shown in the Component Browser as small green clamp icons underneath the icon of the component that they constrain. You can choose the **Select Defining Geometry** command to highlight the geometry used to define a mating condition.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

Understanding Update Problems

If Your Mating Conditions Don't Update

Mating conditions may fail to update when you make changes to the components that make up your assembly. In the component browser, a component with mating conditions that do not solve is shown with a small red X across its icon. You can choose the **Explain Problem** command to receive a description of the problem.

A common cause of failure is two mating conditions that conflict with each other. To resolve this conflict, you need to determine which mating conditions to replace or delete. The **Select Defining Geometry** command highlights the geometry used to define each mating condition. With this command, you can examine the mating conditions and determine which ones to delete or replace.

If Your Assembly Can't Find A Part File

An assembly always looks for the parts that it references in the same directory where you last saved the assembly. If you move a part file to another directory, the assembly first looks for it in the directory where the assembly resides and then in the directories listed in your search path. When you move a part file to a new directory, add that directory to your search path.

See Also

{button ,AL(`assemblyModeling')}*Other topics on assembly modeling*

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Notes can go here.

About Derived Models

Component Features

The **Use Component** command on the **Feature** menu allows you to create a feature whose form and position is taken from an assembly component. Such features are called *component features*. When you create a component feature, the component is hidden and equivalent solid material is added to the part in its place. You can then modify the part by adding further features, and this does not affect the original component part. The part is called a *derived model*, since its form is a derivation of the original component.

The component feature mechanism is associative, that is, changes to the original component part are propagated to the derived part. The derived part will show that a design update should be performed. You use the **Update Design** command to update the derived part when you are ready.

Concurrent Design

The original part and the derived part are separate parts that are stored in separate part files. This means that one person can be modifying the original part, while another person works on the derived part. See *Sharing Information* for more on concurrent engineering.

See Also

{button ,AL(`derivedModels')} *Other topics on derived models*

Using Derived Models to Capture a Process

Design Continuation

The simplest application of component features is to create a derived model that adds further features to an existing part model. The existing part might be a conceptual design, and the derived model might be a detailed design. You can achieve this by creating a new part, adding the existing part as a component, then using the **Use Component** command to add that material to the part.

Design Alternatives

You can create two parts derived from the same original. The two parts could represent design alternatives. Perhaps details of the design depend on the manufacturing process to be used, and there are two processes under consideration. Modifications to the conceptual design are propagated to each of the derived models, maintaining and updating their detail features.

In-Process Models

You can add any part as a component. So you can use any model as the original for a derived model, including other derived models. This allows you to chain derivations together as a series. You can use this technique to represent in-process manufacturing models.

Since you can place views of more than one part on a drawing, you can show in-process “before” and “after” models on the same drawing sheet to document a manufacturing setup.

See Also

{button ,AL(`derivedModels')}   *Other topics on derived models*

Using Derived Models for Advanced Design

Repeated Features

You do not have to create a component feature as your first feature in the derived model. If you have a repeated feature - perhaps a mounting turret for a circuit board inside a plastic part – you can model the feature as an isolated part. Then you can create a component feature for each instance of the feature within the main design.

You can position the component in the correct place first using mating condition commands on the **Assembly** menu, or using the **Shift Point-to-Point** command or the **Transform** command on the **Edit** menu. You can also create patterns of features by duplicating the component first, using the **Duplicate** command on the **Edit** menu. Then you can create a component feature for each duplicated component.

Changes made to the master feature can then be automatically propagated to the main design, updating each instance of the feature. You should use the **Update Design** command when you are ready to see the effect of such changes.

You can also create repeated features that are subtracted from the main design, such as a repeated pocket. The **Use Component** command offers you this option.

Mold Cavities

The cavity of a mold is the negative of the product it will produce. You model a mold cavity by bringing in the product design as a component and then using the component as a feature to be subtracted. This gives the mold has the desired negative form. Any changes to the product design are propagated to the mold design when you use the **Update Design** command.

Welded Parts

You can use component features to unite components to form a single solid. If you have two plates to be welded, you can model each plate as a component part, and then create a component feature from each plate. When the second component feature is created, the solid material is combined with the solid material from the first feature, so that the two plates are fused as one. Then you might add a blend feature to represent a fillet weld.

See Also

{button ,AL(`derivedModels')} *Other topics on derived models*

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Notes can go here.

Introduction to Feature-Based Drafting

DesignWave has a feature-based drafting system. Unlike traditional 2D drafting, where dimensions are created between 2D lines that are part of views on the drawing, DesignWave dimensions and other annotation objects reference 3D model features directly. For example, the length of a rectangular block is dimensioned as the separation of two planar faces in the model.

Intelligent Drafting

Since feature-based drafting is based on 3D model features, DesignWave is able to validate features for suitability. When dimensioning in the drawing, suitable drafting features prehighlight as you move the cursor over them. If a feature is unsuitable, a message to that effect appears in the status bar. When you create a geometric tolerance, the datum reference frame and the orientation of the feature relative to the datums are validated for you, so that correct geometric tolerances are produced. See *Feature-Based Tolerancing* for more information.

Invalid Objects

When you make changes to the parts that are shown on your drawing, the annotation updates automatically. In some cases, you could make changes that invalidate the information shown on the drawing. The linear dimension showing the length of the block is invalidated if you change the part so that the two end faces are no longer parallel. A geometric tolerance of perpendicularity could be invalidated if you change the orientation of the feature or its datum. In such cases, the affected annotation in the drawing turns bright green to indicate that there is a problem. See *About Invalid Annotations* for more information. You can double-click on an invalid object to get an explanation as to why it is invalid.

See Also

`{button ,AL(`dimensioning')}` *Other topics on dimensioning*

About Callouts and Callout Groups

When you create a linear dimension in the drawing, a dimension line and a dimension callout are created for you. The dimension line annotates the separation being dimensioned, and the dimension callout indicates the dimension value.

Callouts

The dimension callout is an object that displays text showing the value of a dimension. You can use the **Properties** command and select the **Number** page to see some of the properties of the dimension callout itself. You can change these settings – such as the units or the number of decimal places - but they only affect the way the dimension is displayed. By using **Copy** and **Paste** commands, you can create two dimension callouts for the same dimension. Each dimension callout can display the dimension value in a different way. For example, one callout could show the dimension in inches, and another could show the same dimension in centimeters.

Each type of information on the drawing has a type of callout used to display it. There are dimension callouts, datum callouts, tolerance callouts, and note callouts. You can use **Copy** and **Paste** to create two note callouts for the same note. One might be in a large sized font and the other might be in a small sized font and have a box around it. You can also use the **Properties** command to change the note itself. Then both callouts update to show the new text. See *Using the Clipboard with Callouts* for more information.

Callout Groups

Sometimes, as in our example above, the dimension line contains a callout group. A callout group is a set of callouts, formatted into rows. When you create a linear dimension, a callout group containing a single dimension callout is created. Afterwards you can insert more callouts into this group. For example, using the **Insert Callout** command allows you to insert a note callout before the selected callout. This produces a callout group containing two callouts: a dimension callout and a note callout. The **Move Callout** commands allow you to re-format the callouts within the callout group. See *Re-Organizing Callouts in a Group* for more information.

When you create a leader-directed note by dragging from a feature, you see a callout group containing a single note callout. You can insert more callouts into this callout group too.

Callout groups can be presented in three ways:

- A callout group can be attached to a dimension line, like the linear dimension example above. A dimension line can have zero or more callout groups attached to it. To create a new group attached to a dimension line, create a note, starting your drag on a dimension line, and holding the SHIFT key down. To remove a callout group from a dimension line, select all of the callouts in the callout group, and use **Cut** and **Paste** to move those callouts to somewhere else.
- A callout group can be attached directly to a feature. When creating notes, datum labels, or geometric tolerances, you can drag from a feature with the SHIFT key held down to produce a callout group attached to that feature.
- A callout group can be free standing in the drawing. If you **Paste** some callouts when you do not have another callout selected to identify a target callout group, a new callout group is created in the center of the drawing ready for you to drag it to where you want it. Free standing notes can be created by dragging from a free-space area of the drawing, with the SHIFT key held down.

In all three cases, a callout group can have one or more leaders drawn from it. In the example of a leader-directed note above, the callout group has a single leader drawn from it. To add a leader to an existing callout group, add a note to the drawing, but move the cursor over an existing callout before releasing the mouse button to finish the drag. The note callout is inserted before the existing callout. See *Using Leaders* for more information.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Moving Annotation Objects

Repositioning Annotation

You can reposition annotation object, such as leaders, dimension lines, and callout groups, by dragging them. You have to select the object before you can drag it. Use the **Annotations** command on the **Select** menu to enter the mode for selecting annotation objects. The selected object is highlighted in red.

You can reposition dimension lines and callout groups by dragging them. A callout group attached to either a dimension line or feature moves along the object to which it is attached. It the callout group is attached using a shoulder, the shoulder stretches, if necessary, as you drag the callout group, unless you hold the SHIFT key down to maintain the length of the shoulder. You can add a shoulder to a callout group using either the **Shoulder** commands on the **Dimension** menu or the **Properties** command and selecting the **Placement** page.

Moving Callouts

Callouts within a callout group cannot be dragged. You can move the callout within the callout group using the **Move Callout** commands on the **Dimension** menu. You can also use **Cut** and **Paste** to move one or more callouts to another callout group. See *Using the Clipboard with Callouts* for details.

Dragging Leaders

Any callout group, even those attached to dimension lines or features, can have leaders connected. If you drag the callout group, any leaders connected to it move too. So if you have a leader-directed note, you drag the note to reposition it, rather than the leader. Dragging the leader has special behavior. See *Using Leaders* for details.

See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Using Center Lines and Center Points

Centerline Features

A centerline represents a feature. Centerlines are the feature-based drafting equivalent of construction lines. For example, you can use the **Mid-Plane** command on the **Center Line** menu to construct a mid-plane. A centerline is displayed that represents a planar feature. You can use this centerline anywhere you want to use a planar feature. So you can dimension to the centerline, or even create a further mid-plane using it as one of the defining planes.

Centerlines are not only produced when you use commands on the **Center Line** menu. They are also produced when dimensioning to axial features. See *Dimensioning in a Specific Direction* for more information. Overlapping centerlines are automatically merged. So if you dimension to an axial feature that has also been used to construct a row of holes centerline using the **Common Plane** command, the resulting centerline still phases correctly through the holes.

Center Point Features

DesignWave draws the short and long dashes of centerlines so as to phase correctly as they pass through the centers of features such as holes. Where the centerline passes through such a center, a center point is displayed. Center points also represent features. In the case of a center point at the center of a hole, the center point represents end-on straight line that is the axis of the hole. You can use the center point anywhere you would want to use such a feature. So you can dimension to the center point as an alternative to dimensioning to the hole itself.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Creating Single, Datum and Chained Dimensions

Datum Dimensions

To create linear dimensions, choose the **Linear** command from the **Dimension** menu. This enters the mode for creating linear dimensions. You create dimensions by dragging in the drawing. First you must click on a feature to select it as the datum feature. Drafting features are prehighlighted as you move the cursor over them. The selected datum feature is highlighted in red. Now you can drag from the second feature to where you want to position the dimension line, then release the mouse button.

The datum feature remains selected allowing you to reuse it as you drag out further dimensions from other second features. This allows you to quickly create a set of linear dimensions all measured from the same datum feature.

Spacing out Dimensions

As you position the dimension lines, they snap to grid points, so you can easily space out dimension lines equally by eye. Note that the grid size depends on how far in you are zoomed – see *Selecting Grid Points* for more information. So it is easier to space out dimension lines if you do not zoom in when you are part way through.

Single and Chained Dimensions

To select a new first feature, you must select another feature, without dragging. To create single or chained dimensions, select the first feature, then drag from the second feature each time.

Selecting Features

Remember that you can use centerlines and center points as features. See *Using Center Lines and Center Points* for more information.

See Also

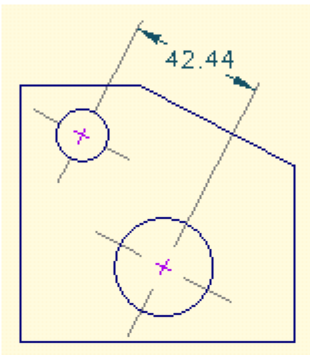
{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Dimensioning in a Specific Direction

Linear Dimension Centerlines

When you create a linear dimension between a planar feature and an axial feature, such as a hole, a perpendicular pair of centerlines is produced passing through the hole center. These centerlines take their orientation from the planar feature at the other end of the dimension. Each centerline represents a planar feature passing through the axis of the hole. One centerline is parallel to the planar feature at the other end of the dimension, and the other is perpendicular to it.

Construction Dimensions



Whenever you want to select a feature, you can select a centerline. The geometry represented by that centerline is used. So you can select either of these the two hole centerlines as a planar feature in the creation of another linear dimension. Having done that, you are free to delete the first dimension line. The second dimension captures the definition of the centerline selected, but does not depend on the first dimension line still being around.

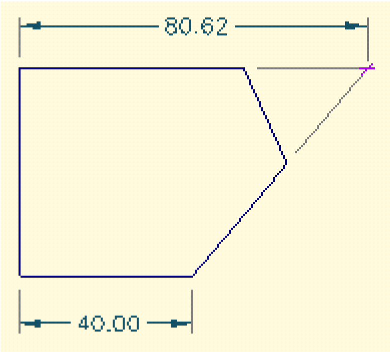
This means that you can create throwaway linear dimensions as a means of constructing centerlines through axial features in a specific orientation. Having done that, you can use that orientation to orient subsequent linear dimensions. So you can create the dimension between two holes in a direction parallel to some other feature, by first creating a dimension from that other feature to one of the holes.

See Also

`{button ,AL(`dimensioning')}` *Other topics on dimensioning*

Dimensioning to Corners

Using Existing Corners



When creating dimensions, suitable features are prehighlighted as you move the cursor over them. Usually linear dimensions are created between surface features, such as planar faces. For example, the linear dimensioning showing the length of a block is defined as the separation of the two planar faces at each end. If the part is tapered, edges are prehighlighted instead, since the planar faces there are not side-on and are therefore unsuitable for a dimension in that view.

Sometimes you want to dimension to end-on straight edges at corners of a profile. Such edges are only one pixel in size on the screen, so they are prehighlighted using small black squares. This ability is only available when creating linear dimensions. To select end-on straight edges, you have to be quite close to them.

If you select an end-on straight edge at a corner as the first feature of a linear dimension, you can continue to reuse this feature as you drag from further second features, but the small black square will not be visible all the time.

Constructing Corners

If you need to dimension to a corner that must first be re-constructed, such as the corner where there is a chamfer, you must use the **Phantom Intersection** command on the **Center Line** menu to produce the side faces so that they intersect. The feature that is created is the end-on straight line that represents the intersection of the two planes.

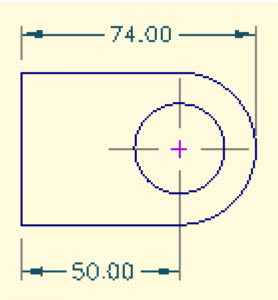
See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Choosing the Measurement

To keep dimension creation as simple as possible, set many of the options that you might want to specify for the dimension by first creating the dimension, and then using the **Properties** command to edit what you created.

Inside, Outside, or Between Centers



One common example is when creating a linear dimension involving axial features that have a radius. When you create a linear dimension between two holes, the distance between the holes is shown. But if you have a slotted hole, a rectangular cutout with semi-circular ends, and you dimension between the two end faces, you probably want the dimension to the outsides of the faces, not the axes. To do this, you first create the dimension between the axes, and then choose the **Properties** command, select the **Measurement** page, and change the dimension to use the outsides. You can select either the dimension line or the dimension callout to do this.

See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Creating Angular Dimensions

Angular dimensions are created in the same manner as linear dimensions. See *Creating Single, Datum and Chained Dimensions* for details. You first click on a feature to select it, and then you drag from the second feature out to where you want to position the dimension line. You can reuse the first feature as you drag out from further second features in a datum-dimensioning scheme. To select a new first feature, click on another feature without dragging.

If you hold the SHIFT key down, the major angle is presented instead of the minor angle. You can also choose the **Properties** command and select the **Measurement** page to change between the major and minor angle after you have created the dimension.

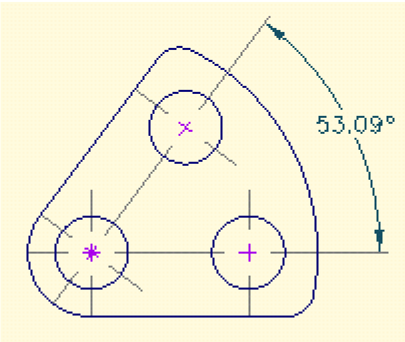
See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Dimensioning the Angle Between 3 Holes

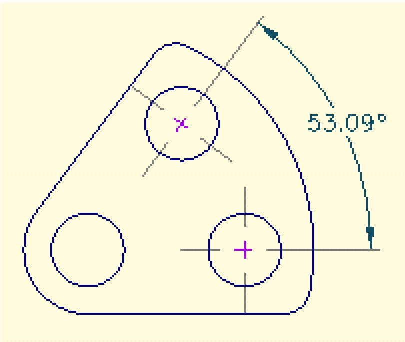
Unlike linear dimensions, which can take many combinations of feature geometry, angular dimensions are always created between two planar features that are side-on in the drawing view. Since planar features have a natural axis of intersection, this allows angular dimensions to be created without having to specify an axis.

Using Centerlines to Construct Planes



In situations where you do not have planar features, you can construct them using commands on the **Center Line** menu. A common case is the angle between two holes, measured about a third axis hole. You should use the **Common Plane** command twice to construct the two planes that pass through the axis of measurement. Then you can create the angular dimension between the two centerlines you have produced.

Using Linear Dimensions to Construct Planes



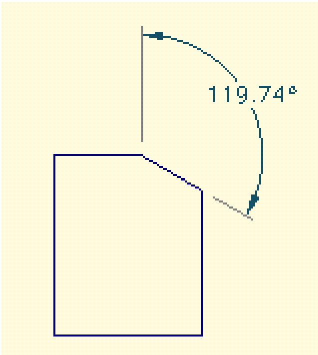
The topic *Dimensioning in a Specific Direction* describes another way of achieving the angle between three holes. You first create two throwaway linear dimensions, between the axis hole and each of the other two holes. This produces centerlines at the other two holes oriented in the direction of the axis hole. These centerlines represent planar features, so you can select these to create the angular dimension. Then you can delete the two linear dimensions.

The result you get with these two approaches is slightly different. Use the approach that gives you the result you want.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Creating the Angle from Vertical



You can use the same technique as described in *Dimensioning in a Specific Direction* to dimension the angle from the vertical, or from any other phantom direction. If you create a linear dimension from a vertical feature, such as the end of the part, to a corner where there is no vertical feature, an extension line is created from the corner to the dimension line.

The corner is actually an end-on straight edge in the model. Like centerlines, extension lines are features that you can use. In this case, the extension line represents a planar feature passing through the straight edge at the corner. You can use this planar feature to create an angular dimension. Then you can delete the linear dimension line, which was only used to construct the feature to use.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Dimensioning the Angle of a Cone

When creating angular dimensions, you are required to use two planar features that are side-on in the drawing view. Dimensioning the angle of a conical face does not fit well into method of creation, since there is only one feature involved, and it is not planar.

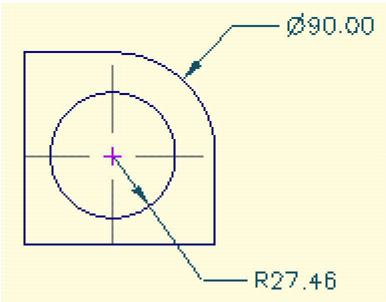
Nevertheless, you can still dimension the angle of a conical face by another approach. You can use the **Insert Callout** command to insert an angular dimension callout before the selected callout. The selected callout must be one that identifies a conical feature. To do this, you must create a leader-directed datum label or feature control symbol that references the conical feature. Creating a leader-directed note for the feature is not satisfactory, since a note does not store a reference to the feature you selected. Having selected the callout, you can use the **Insert Callout** command to insert the angular dimension callout into the callout group. Then you can delete the first callout if you want.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Controlling Styles and Measurements

Converting Between Radial and Diametric



When you use the **Radial** command on the **Dimension** menu to create a radial dimension, it is directed using a leader. When you use the **Diametric** command, a dimension line is produced instead. Sometimes you will want to show a diametric dimension using a leader, or a radial dimension using a dimension line.

To do this, you can use the **Properties** command afterwards and select the **Measurement** page. You can toggle the dimension between radial and diametric. This changes the callout to show twice or half the measurement. So, if you want a diametric dimension using a leader, you should create a radial dimension, then change it to diametric. And if you want a radial dimension using a dimension line, you should create a diametric dimension, then change it to radial.

Radial Dimension Lines

For a dimension line, changing the measurement from diametric to radial also changes the style of the dimension line so that it only has one arrowhead, instead of two. The arrowhead that is kept is on the side where the cursor was when you positioned the dimension line. So, you can drag the dimension line to change the side.

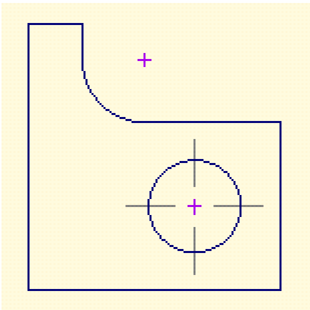
See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Suppressing Center Lines

Centerlines are produced when you use commands on the **Center Line** menu or when you create dimensions using axial features. You can control the style of centerlines choosing the **Properties** command and selecting the **Center Lines** page.

Displaying Hole Centers



When you use the **Center Points** command on the **Center Line** menu, two perpendicular centerlines are drawn crossing at the axis of the selected hole feature. At the axis, a center point is drawn.

This is most likely what you want for a hole, but if you just want to mark the center of a radius, you probably do not want centerlines drawn, just the center point. To do this, you double-click on one of the centerlines produced to display the **Properties** dialog, and you clear the box labeled *Draw centerlines to boundary* on the **Center Lines** page. This removes the centerlines, leaving only the center point.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Feature-Based Tolerancing

DesignWave drafting is feature-based. See *Introduction to Feature-Based Drafting* for more information. When you add datum labels and feature control symbols to the drawing, geometric tolerance information is captured in terms of 3D features in your part model. DesignWave has built-in intelligence for geometric tolerances to help you apply valid tolerances to the drawing.

- You are only invited to choose datum features if datums are valid for the selected tolerances type. For example, a flatness tolerance does not have a datum feature.
- You are only invited to supply a material condition if the feature is a feature of size.
- The datum reference frame you supply is validated. For example, you cannot provide two datum features that are parallel.
- The orientation of the feature being controlled is validated against the datum feature according to the tolerance type chosen. For example, you can only add a parallelism tolerance if the feature really is parallel to the datum selected.

You might make changes to the part so as to invalidate geometric tolerances you have applied to the drawing. For example, you might change the part so that a feature with a parallelism tolerance is no longer parallel to its datum feature. When you change the part, the drawing is automatically updated and any geometric constraints are validated. If tolerances are invalidated, the tolerance callout turns bright green to indicate that there is a problem. You can double-click on the callout to get an explanation of the problem.

See Also

`{button ,AL(`dimensioning')}` *Other topics on dimensioning*

Adding Datum Labels

When you declare datums for a feature control symbol, you identify datum features that have already been named. So you should label datum features first.

Labeling Features

To create datum labels, choose the **Datum Feature** command on the **Dimension** menu, or press the button on the drafting toolbar. Then you can drag from a feature to create a datum label for it. If you hold the SHIFT key down, the label is attached directly to the feature, otherwise it is directed to the feature using a leader. To set the name of the datum, double-click on the datum callout and enter the name in the **Properties** dialog on the **Datum** page.

Mid-Plane or Axis as a Datum Feature

Where the axis or mid-plane of a feature is used as a datum, it is common to show the datum label attached to a dimension line for that feature. According to your drafting standard, you will either want to add the datum callout into the same callout group as the dimension callout (typical of ANSI), or you will want to attach a new callout group to the dimension line containing the datum callout (typical of ISO).

To add a new datum callout after a dimension callout, select the dimension callout and then use the Insert Callout command to insert a datum feature callout. Inserted callouts appear before the selected callout, so you might want to move the datum callout within the callout group. See *Re-Organizing Callouts in a Group* for details.

To attach a new callout group to the dimension line to contain the new datum callout, choose the **Datum Feature** command on the **Dimension** menu, and then drag from the dimension line. If you hold the SHIFT key down, a new callout group is attached directly to the dimension line, otherwise it is directed to the dimension line using a leader.

Dimension Line Extras

If you apply the datum label as a new callout group (typical of ISO), either directed using a leader, or attached directly to the dimension line, you may also want to add a triangle terminator to the dimension line.

If the callout group is directed using a leader, you can double-click on the leader to bring up the **Properties** dialog and go to the **Leader** page. Or, if the callout group is attached directly to the dimension line, you can double-click on the leader to bring up the **Properties** dialog and go to the **Placement** page. In either case, you can select a terminator where it says *Dimension line extras*. This terminator is added to the dimension line on the side closest to the leader terminator or attached callout group. If you drag the leader terminator or callout group to the other end of the dimension line, the terminator will jump to the other end too.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Adding Feature Control Symbols

Application of Feature Control Symbols

You create feature control symbols, also known as geometric tolerance callouts, in the same manner as datum labels. You can create them attached to features, leader-directed to features, attached to dimension lines, or leader-directed to dimension lines, and you can insert them in existing callout groups - just the same as with datum labels. See *Adding Datum Labels* for details.

Entering the Tolerance Control

The feature control symbol produced does not display a geometric tolerance until you edit its properties. Use the **Properties** command and select the **Tolerance** page. You can set the tolerance type, the tolerance value, the material condition, and the datum features. You are only offered options that are appropriate for the tolerance type you select. See *Feature-Based Tolerancing* for more information.

See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Creating Textual Notes

Adding Notes

To create textual notes, choose the **Note** command on the **Dimension** menu, or press the button on the drafting toolbar. Then you can drag to create notes. Different forms of presentation are produced depending on where you start the drag:

- If you drag from a feature, a note is created that is directed to the feature using a leader. If you hold the SHIFT key down, the note is attached directly to the feature, using an extension line or centerline as appropriate.
- If you drag from a dimension line, a note is created that is directed to the dimension line using a leader. The leader has a null terminator. You might use this style when there is not enough space to show a dimension callout on the dimension line. In that case, you would use **Cut** and **Copy** to move the dimension callout to the leader-directed callout group, then delete the note callout. If you hold the SHIFT key held down while dragging, the note is attached directly to the dimension line as in new callout group.
- If you drag from empty space on the drawing, a leader-directed note pointing to that space is created. A blob terminator is used since this is the style typically used to point to faces. If you hold the SHIFT key held down while dragging, the note is created without a leader, as a free standing textual note. You could also delete the leader to produce the same result.

Prefixing a Note

To insert a note before an existing callout, you can choose the **Insert Callout** command. You might do this for a dimension applied to a pattern of features, in order to quantify the number of features in the pattern.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Using Leaders

Creating Leaders

Many commands can produce leaders: adding notes, datum labels, feature control symbols, or radial dimensions. In these cases, a leader is created for a newly created callout group. You can also add a leader to an existing callout group. You might do this because the callout group does not have a leader, or because you want it to have more than one leader.

Whenever you are creating an annotation object and you are dragging to position the callout group and leader, if you release the mouse over an existing callout, the leader is added to that callout group, and the new callout is inserted before the existing callout. If you just want to add a leader, you might create a note in this way, and then delete the note callout.

Modifying Leaders

Leaders update whenever their callout group, or the object to which they point, changes. To move a leader, you move the callout group and the leader follows.

If you select a leader, you can modify it by dragging from different points on the leader:

- If you drag from the shoulder of the leader, a new jog point is created. To remove all jog points, you can double-click on the leader to bring up the **Properties** dialog, select the **Leader** page, and check the *Remove jog points* option.
- If you drag an existing jog point, the jog point is repositioned.
- If you drag the terminator, the terminator is repositioned. Leaders pointing to features and dimension lines, are initially created pointing perpendicular to that object. You can drag the terminator to override this. To make the termination perpendicular again, you can double-click on the leader to bring up the **Properties** dialog, select the **Leader** page, and check the *Perpendicular to target* option.

The **Properties** dialog also allows you to change the terminator.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Re-Organizing Callouts in a Group

Callouts are formatted in rows within a callout group. See *About Callouts and Callout Groups* for more information. You can move the selected callout around within its callout group using the **Move Callout** commands on the **Dimension** menu. The arrow keys and the ENTER key are provided as keyboard shortcuts to these commands.

You can also move callouts to a different callout group, or to become a new callout group. See *Using the Clipboard with Callouts* for more information.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Using the Clipboard with Callouts

You can use the clipboard commands – **Copy**, **Cut** and **Paste** – with callouts. Using **Cut** and **Paste** allows you to move the selected callouts to another callout group. See *Dimensioning Hole Features* for an example of this. You can **Cut** several callouts at once, and they do not have to be in the same callout group. Using **Copy** and **Paste** allows you to duplicate callouts. See *Creating Tabular Dimensions* for an example of this.

Dual Units Dimensioning

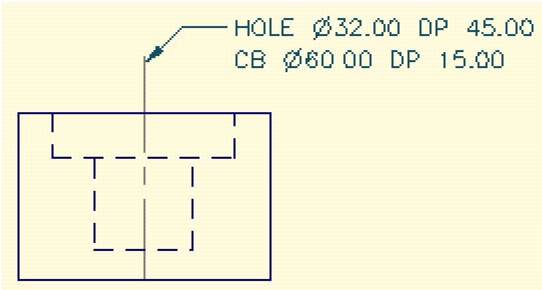
You can use **Copy** and **Paste** to duplicate a callout within the same callout group. You might do this to produce a dual units dimension. Having created the dimension, you can select the dimension callout, then choose **Copy** immediately followed by **Paste** to duplicate the callout. Then you might press the DOWN arrow key to move the new callout to the next line within the callout group to produce a presentation style typical of dual units dimensioning. Now you can choose the **Properties** command, go to the **Number** page, and change the units used to display the value.

See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Dimensioning Hole Features

You can use **Cut**, **Copy** and **Paste** to move and duplicate callouts. See *Using the Clipboard with Callouts* for details. This allows you to remove dimension callouts from dimension lines and present them in some other manner.



You can use this approach to produce hole dimensions. See *Adding Holes* for more on hole features. Typically for holes, you might show all the dimensions for the hole in a single leader-directed callout group.

In the example shown, the hole was first dimensioned using two diametric dimensions and two linear dimensions. Then a note was added pointing to the hole feature. To point to the axis of the feature, rather than the surface, you can double-click on the leader to bring up the **Properties** dialog, go to the **Leader** page, and change the termination to be *To center or axis*.

All the dimension callouts were selected, and then **Cut** and **Paste** were used to move them to the leader's callout group. The four dimension lines were then deleted. Finally some further textual notes were inserted and the callouts were arranged in their intended rows.

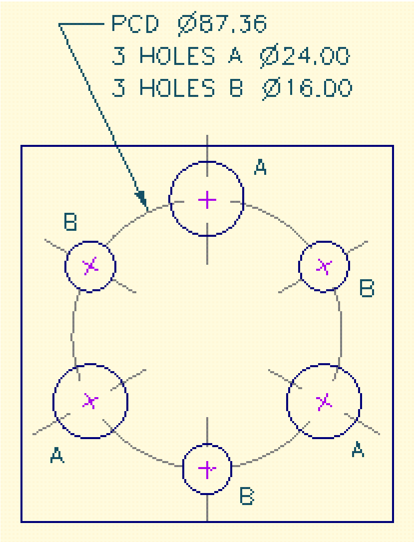
See *Creating Textual Notes* and *Re-Organizing Callouts in a Group* for details.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Dimensioning Pitch Circles

You can use **Cut**, **Copy** and **Paste** to move and duplicate callouts. See *Using the Clipboard with Callouts* for details. This allows you to remove dimension callouts from dimension lines and present them in some other manner.



You can use this approach to dimension pitch circles. In this example, the **Pitch Circle** command on the **Center Line** menu was used to produce the centerlines for the pitch circle.

The pitch circle was dimensioned using a radial dimension, so as to produce a leader rather than a dimension line, then the measurement was changed to diametric. See *Controlling Styles and Measurements* for details. One of each size of hole was dimensioned using a diametric dimension. Then these two dimension callouts were moved to the leader's callout group using the **Cut** and **Paste** commands. The redundant dimension lines were then deleted.

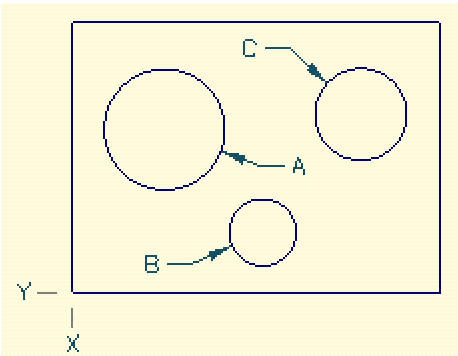
Finally some further textual notes were inserted and the callouts were arranged in their intended rows.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Creating Tabular Dimensions

You can use **Cut**, **Copy** and **Paste** to move and duplicate callouts. See *Using the Clipboard with Callouts* for details. This allows you to remove dimension callouts from dimension lines and present them in some other manner, such as in a table. The callouts still update automatically when you make changes to the part.



HOLE	X	Y	DIA
A	28.03	18.00	Ø36.88
B	58.00	54.00	Ø20.40
C	88.00	49.41	Ø28.00

In the example here, the dimension values in the table are dimension callouts in free standing callout groups. The callout groups were individually positioned in rows and columns.

The view was first dimensioned using regular linear and radial dimensions. Radial dimensions were used since this produces leader lines pointing to the holes, but each dimension callout was then changed to a diametric measurement. See *Controlling Styles and Measurements* for details.

A prefix note was inserted before each of the three diametric dimensions. See *Creating Textual Notes* for details. Each note was duplicated in the first column of the table using the **Copy** and **Paste** commands. Each dimension callout was moved to the table using the **Cut** and **Paste** commands. The six linear dimension lines – without callout groups attached – were then deleted.

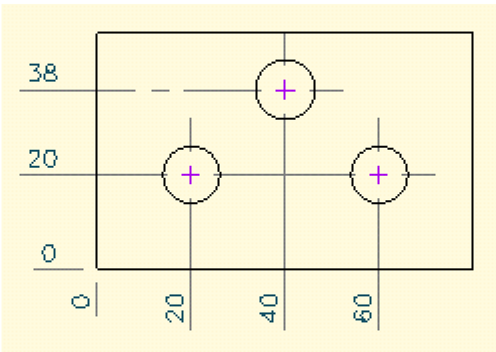
The labels on the axes were added as notes attached to the feature. Then these were duplicated in the table headings using the **Copy** and **Paste** commands.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Creating Ordinate Dimensions

You can generate alternative styles of dimensioning by using the centerlines and dimension callouts from regular dimensions. The centerlines are valid features to which you can attach callouts. You can then use the **Cut** and **Paste** tools to move the dimension callouts from the dimension lines into the new callouts that you have created. See *Using the Clipboard with Callouts* for details on how you can do this.



In the example here, the holes were first dimensioned from the side faces using regular linear dimensions. Callout notes were then attached to the centerlines generated by the linear dimensions. The SHIFT key was held while dragging each callout in order to attach it directly instead of it being leader directed. See *Creating Textual Notes* for more details.

Each dimension callout was moved into its corresponding note callout using the **Cut** and **Paste** tools. The callouts on the side faces were edited to contain '0's. The unused notes and remaining dimension lines - without callout groups attached – were then deleted.

The orientation of one of the callouts was changed to be *Along* and *Above* the line using the **Placement** page of its **Properties**. The **Pick Up Properties** tool was then used to capture its placement and this was copied to the other callouts using **Apply Properties** to give them the same appearance. See *Copying Styles to Other Objects* for how to do this.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

Setting Up Drafting Standards

Setting up Styles

Many of the styles that affect annotation in the drawing are taken from settings you establish using the **Options** command. There are various pages in the **Options** dialog, and many of these affect the style of annotation:

- The **Dimension** page allows you to set up the tolerance style and default tolerance for dimensions created in the future. You can also set up options to create basic dimensions or reference dimensions.
- The **Datum** page allows you to set the style used for datum labels.
- The **Placement** page allows you to say how you want the text to be drawn when attached to dimension lines or directly to features. You might say that you want the text to be horizontal and in-line with the dimension line.
- The **Number** page allows you to set the units and decimal places for dimensions in the drawing.
- The **Leader** page allows you to set the length of the horizontal shoulder that connects the leader to the text.

The settings you choose using the **Options** command only affect annotation created from that point onwards. So you should establish all these settings beforehand, so as to conform to your drafting standards.

Changing Styles

You can change settings for annotation already created using the Properties command. See *Editing Annotation Properties* for more information. You can also apply settings to many or all of your annotation objects. See *Copying Styles to Other Objects* for how to do this.

See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Editing Annotation Properties

When you create annotation objects, many of the styles come from the settings you have chose using the **Options** command. You should use the **Options** command to set up your drafting standard. See *Setting Up Drafting Standards* for more information.

You can change the style of an existing annotation object using the **Properties** command. You can select the object and then choose the **Properties** command, or you can simply double-click on the object to bring up the dialog window. To change the properties of more than one object, you can use the **Pick Up Properties** and **Apply Properties** commands. See *Copying Styles to Other Objects* for more information.

See Also

{button ,AL(`dimensioning')} *Other topics on dimensioning*

Copying Styles to Other Objects

To use the **Properties** command to examine or modify settings, you must have a single object selected. See *Editing Annotation Properties* for more information.

If you want to change the properties of several objects, you should modify one object first so that it has the settings you want, and then use the **Pick Up Properties** and **Apply Properties** commands to apply these settings to the other objects. You can use the **Select** All command to apply properties to all annotation objects in your drawing.

The **Pick Up Properties** command offers you categories that correspond to settings or groups of settings that you see in the pages of the **Properties** dialog. You should select those categories you wish to pick up. Then the **Apply Properties** command applies just those categories of settings that you chose to the currently selected objects.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

About Invalid Annotations

Implied Constraints

When you create dimensions and geometric tolerances, as well as calling out information on the drawing, you are also applying geometric constraints to the model. For example, if you add a linear dimension between two planar faces, you are implying that these two faces are parallel. And if you add a perpendicularity tolerance to a feature, you are implying that it is indeed perpendicular to its datum feature.

When you construct centerlines, you are also implying geometric constraints. For example, if you use the Common Axis command to produce a centerline through two co-axial features, you are implying that they are indeed coaxial.

Validation

You cannot create dimensions, geometric tolerances, or centerlines that are invalid. If you try to create the linear dimension between two planar features that are not parallel, the cursor changes from the creation pencil to the selection arrow to indicate that you cannot create the dimension.

However, you can produce invalid annotation objects by changing your part. If you change the part so that the two planar faces of a linear dimension are no longer parallel, or the two axial features defining the centerline are no longer co-axial, this means that the geometric constraints are not satisfied, and the objects become invalid.

Invalid Objects

An invalid object is displayed in bright green to indicate that it has a problem. You can double-click on the object to get an explanation for the problem. In many cases, pressing the F1 key will give you some additional help along with some suggested courses of action. Note that you cannot see the bright green color if the object is currently selected, since it will be shown in red.

Dormant Features

When you make changes to your part, these changes are reflected immediately in views of that part shown in the drawing. If you suppress a feature and then update your design, the faces and edges for that feature no longer appear in the model. So they no longer appear in views of that part in the drawing. Such faces and edges are called *dormant*, because they are not actually deleted, just taken out of service until further notice. If you unsuppress the feature and update the design, the faces and edges come back to life.

Annotation objects can become invalid because their inherent geometric constraints are not satisfied, or because the features used to define them are currently dormant. For example, if you dimension the position of a hole, and then suppress the hole and update the design, the dimensions to that hole will turn bright green to show that they are invalid. If you double-click on the dimension line or dimension callout, the explanation will tell you that a feature is currently dormant. If you then unsuppress the hole and update the design, the dimensions will spring back to life, updating their values if need be.

See Also

{button ,AL(`dimensioning')}   *Other topics on dimensioning*

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

Notes can go here.

Introduction to Drawings

Drawings convey design information in a 2D representation for parts and assemblies previously modeled. You create a drawing by selecting one of the common paper sizes, defining your own size or by referencing a template drawing from which the new drawing derives its size.

Sketches

You can add your own sketch to drawings, using the same tools you use to create sketches in the modeling environment. Sketches are used as input to some of the view creation tools.

Drawing Views

A drawing can contain views of any number of parts. You add primary views to a drawing by selecting from a list of the open part windows. You can individually scale these views or define an overall scale factor for the whole drawing. You can then derive projection views and section views from the primary views. Crosshatching for section views is generated automatically.

Annotation

Intelligent annotation tools allow you to add dimensions, callouts, and tolerance information in a context-sensitive manner. You can even add dimensions between different views of a part. Part views and annotations that reference model geometry are associative. If you modify a part, its drawing views will be updated to reflect the changes.

When you open a drawing, any referenced part files are opened, too. Therefore, when moving a drawing from one system to another, be certain to include all the part files.

Printing Drawings

You make hardcopies of drawings by printing the document. You can send scaled-down proof plots to the office printer and full-size plots to the intended plotter.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Using Sketches

A drawing contains a single workplane in which you can add as many sketches as you like. The tools for creating sketches and lines are the same as the sketch and line tools in a part window.

You can use sketches in a drawing to contain user-defined lines, such as a sheet border, header block, or a revision grid. You also use sketches to define section cuts for section views and clipping boundaries for partial views. Since a sketch can be used for a variety of purposes, give it a meaningful name, such as "Section A-A". This will make it easier to identify the sketch later.

The sketch commands are available on the **Drawing** menu. See *Introduction to Sketching* for more details.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

About Sheet Properties

When you create a drawing, you can enter some of its properties, such as size and scale. You can change these later by choosing the **Sheet Setup** command on the **Drawing** menu. For example, if you find that you cannot fit all the views you want on a drawing, you can either select a larger drawing or you can adjust the scale factor applied to all views.

Sheet Size

You choose the sheet size from a list of ISO and ANSI standard sizes or you can enter your own custom size. Alternatively, you can select another drawing to be used as a template, which also defines the sheet size. The sheet extents are shown in a drawing window as a dashed line. This will not appear in plots.

Scale

You can specify a scale factor for the drawing, which defines the default scale for all views on the drawing in terms of a ratio of paper to model. You can also set the scale for each view individually, but it is usually easier to set the default for the drawing. This way, you can usually avoid having to specify a scale each time you create a view. The view scale simply determines how large a view will appear on a drawing of a given size. It does not scale the geometry within a view.

Units

When you create models, you work in the units specified for *model distances*. Internally, all distances are standardized to SI units. This means that you don't need to know what units a model was created with when you create a view of it on a drawing. When you create lines in a drawing or specify the sheet size, you work in the units specified for *paper distances*. You can set both these units on the **Units** page.

Numeric values that you see on the drawing itself, such as dimension values, take their measurements directly from the model and specify their own units for display purposes. You can set the default units for numeric values using the **Number** page.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Using Formats

When you create a new drawing, or later using the **Sheet Setup** dialog, you can specify the sheet size or you can select an existing format. A format is, simply, another drawing, whose lines and text are superimposed on the drawing that references it. The format also defines the sheet size. When you change a format, any drawings that reference it will be updated when they are opened.

Normally, engineering drawings contain a border and sheet header block with drawing information, such as standards conformation, and a company logo. You can place all of these common lines and text objects on a set of drawings to be used as formats. Whenever you create a new drawing, you can select an existing format and see all the shared objects. Later, if you wish to alter the style of border or change the header block, you need only change the format drawing.

If you move a drawing to another system and it references a format drawing, be certain to include the format, too.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating Formats

You create a format by creating a new drawing and adding the lines and text you want to see on all drawings using that format. The content of the format drawing is not editable on a drawing that references it, so you avoid adding text that you will want to change for each drawing. You should create a different format for each sheet size you use.

When you define the border on your format drawing, depending on the printer or plotter device you are using, you will probably need to offset the border lines in by a small distance to allow for a margin. If you place the border lines directly on the sheet boundary, you may lose the upper or right border when printing. This is because the printable area is often slightly less than the physical paper size, due to the printer imposing its own margins.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Adding the Principle View

Typically, a drawing contains at least one principal view of a part; that is one of the standard elevations such as plan, front, left side and right side. Before you add a principal view of a part to a drawing, you first select the desired view direction in the open part window. The standard elevations are available on the **View** menu. You then choose the **Add Modeling View** command on the **Drawing** menu and select from a list of all the open parts in the current session.

You can add other views of the same part by repeating this procedure after reorienting the view in the part window. You can add views of any number of different parts to a drawing. Normally, the set of views for a particular part follows a standard convention, such as *first* or *third* angle projection. If you wish to create projected views, it is usually easier and more beneficial to use the **Add Projected View** command.

After you add a view, the **Select Views** command is active. In this mode, views are displayed with a surrounding dashed box. You select a view by selecting its bounding box and you reposition the view by dragging the box to a new location.

You change the properties of a view by double-clicking its boundary. You can control the display of hidden lines, scale the view, or specify a clipping boundary to create a partial view.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating Projected Views

Projected views are created from existing drawing views. You create a projected view by first selecting a side-on face onto which you want a projection to be made and then choosing the **Add Projected View** command on the **Drawing** menu.

Once created, a projected view remains associated to its defining view, including knowledge of the face from which the projection was made. You can reposition a projected view wherever you want by dragging its border. By holding the SHIFT key down while dragging, you can maintain the correct alignment for the projected view. Position the view accordingly to achieve first or third angle projection.

You can create angled auxiliary views by projecting from any side-on planar face. The orientation of the auxiliary view will always be maintained at the correct angle.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Adding Secondary Views

You can add isometric or trimetric views by activating the part window and choosing the desired view direction on the **View** menu. You can then add this view to the drawing, as you would a principal view.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Removing Hidden Lines

By default, hidden lines are shown as dashed lines. You can turn off the display of hidden lines for an individual view by selecting the view and adjusting its properties. You can redisplay the hidden lines later, if you wish.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Aligning Views

Normally, a projected view is shown aligned with its principal view along the axis of projection. If you reposition a principal view and you have projected views associated with it, the projected views are aligned automatically.

You can further adjust the position of a projected view by selecting it and then dragging its border while holding the SHIFT key down to maintain the alignment with its projection. If you reposition a projected view, you can quickly realign any other views that are meaningfully related to it by choosing the **Align Other Views** command on the **Drawing** menu.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Updating Views

All drawing views are associative. This means that they will be updated to reflect changes made to the underlying part. When you modify a part, all drawing views of it will initially be shown as a black wireframe rendering instead of the normal dark blue hidden line image. This indicates that the views on the drawing need updating. At this point, you can choose the **Update Views** command to regenerate the hidden line images in the drawing views.

When you update a view, you also regenerate any projected views associated with it. Any annotation that references the views is updated automatically.

In some situations, a view can become invalid if the objects used to define it are missing or invalid themselves. An invalid view will be shown with a green dashed bounding box. You can determine the reason for an invalid view by double-clicking its boundary in view selection mode and looking at its properties.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

About Section Views

A section view is defined by a principal or projected view - the exterior view - and a section cut through that view. You define the section cut using a sketch containing connected straight lines. You use the **Add Section View** command on the **Drawing** menu to create a section view and then specify the sketch to be used for the cut in the dialog window that is presented. Each section definition must live in its own sketch. There are rules that determine how the lines within the sketch generate the resulting section view.

There are two distinct types of section view in DesignWave; sectional views and cross-sections. *Sectional views* show the visible lines behind the section line and *cross-sections* do not show these lines.

Crosshatching is automatically generated where the solid is cut. Furthermore, if you section an assembly view, the crosshatch angle is alternated for each part in the assembly.

Section views are fully associative. If you move the exterior view from which the section is derived, the section view is aligned automatically; if you change the part or modify the section cut lines, you can regenerate the section views with the **Update Views** command. Just like projected views, section views know how they have been created and in which direction they should be aligned. You reposition a section view by dragging its border. If you hold the SHIFT key down while dragging, the alignment is constrained along an axis perpendicular to the principal section line.

Section Lines

You define the section cutting planes using straight lines connected in a chain. Each solid line defines a section cut. There is always one principal section line that extends outside the view. The projection of the resulting section view is perpendicular to this line. You can connect another section line at an angle to obtain an aligned section. You can create steps in the section lines to obtain offset sections, but you must join any adjacent stepped segments with a construction line.

To indicate to which side of the principal section line you want the section view to be projected, you append a construction line to the principal section line pointing away from the eye. This construction line also indicates the principal section line.

When you create the sketch for the section definition, remember to give it a meaningful name, such as "Section A-A", so that you can identify it from other sketches in the drawing.

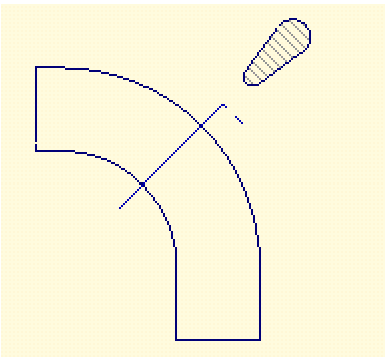
If you want the section line to pass through attraction points of model edges, such as the center of a hole or the midpoint of an edge, turn the **Edges** check box on temporarily. You will probably need to turn it off again to connect the section lines.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating Cross-sections

Sometimes, section views are used to describe a profile or 'slice' of a solid, without showing the visible lines behind the cutting plane. You can create a cross-section in the same way as you would a sectional view just by choosing the cross section option when the view is created. Alternatively, you can change a sectional view to a cross-section, or vice versa, by double-clicking the view boundary and changing its properties. After you make a change, choose the **Update Views** command to regenerate the section view.



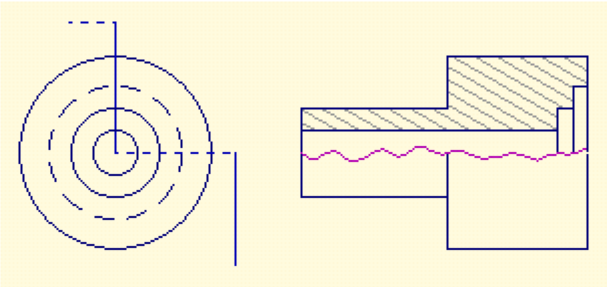
This example shows a cross-section through a swept solid. The view has been moved to a position coaxial with the section cut line.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating Half Sections

Half sections refer to section views that are only partially sectioned. You create a half section view by stepping the section definition outside the view at the point at which you want the sectioning to finish, and continuing with another solid section line. You must connect the solid lines with a construction line.



This example shows a half section view through a shaft. One half of the shaft is sectioned and the other half shows the exterior. The wavy break line has been added as a separate sketch.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating Offset Sections

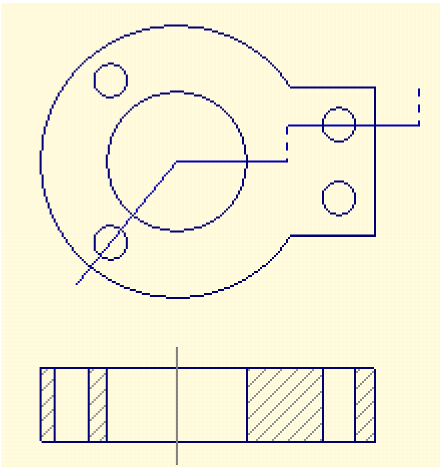
A section view is defined by a chain of section cut lines. You can create offset section views by adding offset segments to the section definition. You must join the adjacent offset lines by a construction line to maintain a contiguous series of lines defining the section cut. The construction segments do not contribute to visible sectioning in the resulting view. See *Creating Aligned Sections* for an example that includes an offset section.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating Aligned Sections

You can create an aligned section view by adding a segment to the section definition at an angle. The same rules apply for the section definition line chain: it must be contiguous with offset sections joined by construction lines. The direction of projection is defined by the principal section cut, which is identified by an additional construction line. The tail of the construction line points away from the eye.



This example shows how a section cut line can define an aligned and an offset section view.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Controlling Cross Hatching

Crosshatching is selectable as annotation. You can double click on an individual crosshatch object and change its angle from the horizontal, the line spacing or the phasing. A separate crosshatch object is generated for each component of an assembly. When an assembly view is sectioned, the crosshatch angle is automatically alternated from positive to negative for adjacent parts - a common practice when sectioning assemblies to distinguish the components. If you want to provide further distinction, you can change the crosshatch properties of any individual part.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Clipping Views

You can clip any existing drawing view to create a partial view. The clipping boundary is defined by a sketch containing straight, contiguous lines forming a closed boundary. Remember to create a new sketch for the clipping boundary and apply a name that will help you find it later. To clip a view, double click its boundary to view its properties and check the clip option, giving the name of the sketch used to form the boundary.

You can change the boundary definition at any time afterwards. As long as the sketch defines a valid closed boundary, you will see the view update automatically as you edit the lines in the boundary. Later, if you want, you can remove the clipping option.

See *Creating a Shortened View* for an example of how clipping can be used.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating a Detail Enlargement

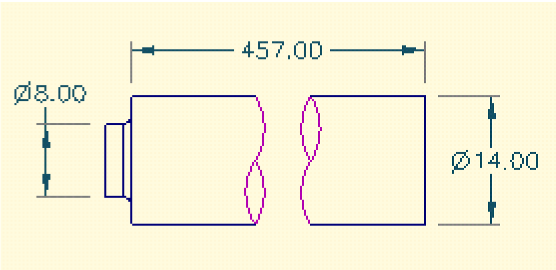
You create a detail enlargement by choosing the **Add Modeling View** on the **Drawing** menu to add a view. You then enlarge the view by double-clicking its boundary and setting the scale factor in its **Properties**. Now you can define a clipping profile in a new sketch and specify this in the enlarged view's properties page. See *Clipping Views* for details on how to clip a view.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating a Shortened View

You can create a shortened view by adding the view twice to the drawing. Scale and position the views so that the two ends show in their desired positions. You can define clipping boundaries for each view to obscure the remaining parts. If you do not wish the clipping boundaries to show in the drawing, you can use the **Sketches** dialog to turn off the visibility of the appropriate sketches. See *Clipping Views* for details on how to clip a view.



This example shows a shortened view of a long shaft. In fact, it is represented using two enlarged, clipped views. The visibility of the clipping sketches has been turned off and the break lines have been added as a separate sketch. The feature based dimensioning used in DesignWave ensures that the separation values you see are always correct for the model space, even though the actual features being referenced may be in different views.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

Creating Multi-Sheet Drawings

DesignWave does not impose any requirements or limitations on what views you choose to place on a drawing, or what constitutes a set of sheets defining a drawing. A drawing and a sheet are the same thing to DesignWave and each is stored as a file. You can define your own multi-sheet structure by storing the sheets making up a drawing set in their own folder.

For example, you can create a single drawing containing all the views of a simple assembly, including the individual part views and section views. Alternatively, you can place some views on one sheet and, say, conceptual views on another sheet.

See Also

{button ,AL(`drawings')} *Other topics on drawings*

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Notes can go here.

Introduction to Feature Modeling

You can build parts by creating features. Each feature can consist of a number of Boolean operations. The feature modeling process starts with the construction of a basic shape, a profile. You can construct a base feature by extruding, revolving, or sweeping the profile. After you construct the base feature, you can add or remove material from the base solid using a number of profile-based feature operations. You can use profile-based features like **Extrude Profile**, **Project Profile**, **Revolve Profile** or **Sweep Profile**. You can use **Extrude** and **Project Profile** to create bosses, pads, pockets, and holes. You can use **Sweep Profile** to create pipes, grooves, and lips. You can use **Revolve Profile** for the creation of curved pockets or bosses. You can also create a thin walled solid using **Hollow Solids** or you can use **Blend Edges** to create a blend, fillet or a chamfer.

Naming Features

Features have names. The name is used whenever the feature is shown in the browser view. Although DesignWave supplies a default name, such as "extrusion 2", when you create a new feature, it is usually worth your effort to give a more meaningful name that captures the intended use of the feature in the part. You can give a name for the feature at the time you create it or at any time afterwards by editing the properties of the feature.

Editing Features

You can edit Features by selecting them from the browser view and changing their **Properties** from property sheets. You can modify Profile-based features by changing the sketch. You can also **Condemn** or **Suppress** or Reorder features using the feature browser. After editing, condemning, suppressing or reordering the features, the part can be updated by selecting **Update Design** from the **Feature** menu in part window or by pressing the **Update Design** button on the toolbar.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Updating Your Model

You can edit features by changing feature properties or by changing the sketch it depends on. You can also suppress or condemn a feature in the feature browser. Feature browser also allows you to reorder the features and to move the Finish flag. When you modify features, **Update Design** from the **Feature** menu (or **Update Design** button on the toolbar) becomes active. At this point, you can either regenerate the model or make more changes.

You may find that some features fail to regenerate while updating. Failure can occur for valid reasons, such a radius that is too large, or the deletion from the model of faces and edges necessary to the feature. You can change the feature properties to repair the problem. You can also condemn or suppress the feature and update the rest of the features.

Stopping the Update Process

The Update dialog box displays during part update and indicates the progress of the update process. You can press the **Stop** button in the update dialog to pause the update process. Subsequently you can press the **Continue** button to proceed with the update process or press the **Stop** button again to confirm that you want to stop the update process.

In some situations, you may need to indicate whether or not you want to continue the update process, even though you did not press the Stop button. For example, suppose you draw a rectangle and extrude it. Then, you select one of the side faces to create a workplane, sketch a circle on it, and extrude it to create a boss. You can blend some of the edges after that. However, if you change the lines in the rectangular sketch for the first extrusion to arcs, the workplane for the boss cannot regenerate successfully, since the face geometry it references is no longer planar. At this point, you can stop the update process and mend the problem, or continue to update the rest of the features.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Using a Sketch More than Once

You can use the same sketch in multiple features. You may wish to use the same sketch in several situations, such as projecting the same sketch above and below workplane to create a throughole when the sketch is inside the volume of the solid.

Use of a sketch in multiple features can also be the result of an error.

- You might select the wrong sketch from the list when using the **Sketches** command.
- You might forget to create a new sketch before trying to create a feature.
- You might draw lines on the active workplane without creating a new sketch, which results in the lines being added to the active sketch.

If you select a sketch that has already been used, you receive a warning message since this is often done accidentally. Just confirm that you wish to proceed.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

About Profiles

In feature operations, profiles are sketches drawn on workplanes. All lines that you create live in sketches, which belong to workplanes.

You must have closed loops of nonintersecting lines for a valid profile. You may have inner loops that define “holes” in the profile, but no further loops inside those. Inner loops provide a quick way of producing thin walled shapes in the profile-based features, but multiple levels of nesting are not allowed. You should not draw inner loops that touch the outer loop or are coincident with any portion of the outer loop.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Modifying Profiles

You can modify profiles by adding new lines to the sketch or by deleting or modifying existing lines in the sketch. The resultant sketch should form a valid profile for the feature operation. You can use the **Activate Sketch** on the popup menu whenever you can see the sketch in the browser view.   When the sketch is activated **Select Lines** command in the toolbar becomes active and you are ready to modify the sketch.

You can also convert some of the lines in the sketch to construction lines using **Toggle Construction** on the **Lines** menu. Construction lines do not participate in the profile for the creation of features.

You can change the workplane to which a sketch belongs from the **Properties** dialog of the sketch. You can use this to move features from one face to the other.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Creating the Basic Shape

You can start constructing a part by creating a basic shape with a base feature. You can create the base feature by extruding a sketch, revolving a sketch, or sweeping a sketch along a path.

Generally you can sketch the outer contour of the part you are going to model and use that as the profile for the base solid. You can then add detailed features to the part by adding and removing material from the basic shape using other feature operations.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Adding Bosses, Pads, and Pockets

You can add bosses, pads, and pockets using the **Extrude Profile** command if you know the height for the pads and bosses and depth for the pockets. The contour of the pads and pockets are reflected in the sketches used.

You can also create the bosses by **Project Profile**, projecting a sketch onto a solid and adding material.   Similarly, you can create pockets by drawing a sketch inside the solid and perform projection with material removal.

With pads or bosses, you are often more interested in the position of the top face than the height of the feature. Sometimes the height is not even well defined, since the feature is not sited on a planar face. You can create a workplane where you want the top face to be, sketch a profile, and then project the profile down onto the part.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Projecting Profiles onto Your Model

You can use the **Project Profile** command to add ribs in a thin-walled part or to create througholes in a part.

Often when extruding a profile toward a part to add a feature, you find that, in order to extrude far enough at one point, the material would be too long somewhere else and poke through to the inside or out of the back of your part. In situations like this, you can project the profile to the part.

You can add ribs to a thin-walled part by sketching long, thin rectangles across the face of the part you want to open. The rectangles can hang outside the part. Projection takes care of the trimming for you.

You can sketch a profile on the surface of your part and then project it down the part, removing material. You can create a set of holes at one time.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Adding Revolved Features

You can add revolved features using the **Revolve Profile** command from **Feature** menu. You can use **Revolve Profile** to create spherical or toroidal pockets and protrusions in a part. You can also use **Revolve Profile** to create the base feature.

You should not sketch an axis of revolution that intersects the profile. You can use the sketch for axis of revolution as well as the profile in multiple features. The sketch for the profile and the sketch for the axis should be on the same workplane.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Adding Holes

You can create holes in a solid in a number of ways:

- If you know the extent of the solid, you can extrude the sketch of the hole by a distance greater than the extent of the solid and subtract material. You can use the **Symmetric about workplane** option for a throughole if the sketch is inside the volume of the solid.
- If you do not know the extent of the solid, you can project the profile to create a throughole. You might need to project above and below the workplane using the same sketch to produce a throughole, if the sketch is inside the volume of the solid.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Creating Pipes, Grooves, and Lips

You can create pipes, grooves, and lips on a solid by using the **Sweep Profile** command from the **Feature** menu. You can sweep a profile that is partly inside and partly outside the volume of the solid to create a lip or a groove. In these cases, the path along which the profile is swept, is often the contour of the solid or projection of the contour on a plane perpendicular to the plane of the profile.

You should create a sweep path that intersects the plane of the profile perpendicularly. Make certain the path does not have tight corners, otherwise DesignWave detects self-intersection. If there are multiple curves in the path, you should make the curves tangent continuous to successfully produce the elbows.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Adding Blends, Fillets, and Chamfers

You can add blends, fillets, or chamfers using the **Blend Edges** command from the **Feature** menu. You should blend edges of a part that have sharp corners that are difficult to manufacture or may lead to stress concentration when the part is under load. You might want to blend edges for aesthetic purposes, too.

The edges selected for a blend feature may not exist anymore while updating the part because you might have suppressed some features the blend depended on.

The faces generated by a blend feature may change considerably if you have the **Auto propagate** option turned ON and reorder blend features.

It is recommended that blending be the last set of operations performed on a part.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Creating Thin-Walled Parts

You can construct thin-walled parts using the **Hollow Solids** command from **Feature** menu by selecting the faces to remove. You should create the thin wall after creating the basic shape along with the shapes for which you want the wall to be built. Generally, it is recommended that you perform blending after creating the thin-walled part.

You should select all the tangent faces of the part for removal. If you intend to open up only some of the tangent faces, the operation may not succeed. You can select all the tangent faces by selecting a face and then selecting the **Add Tangent Faces** command on the **Select** menu of the part window.

Make sure the offset distance for the hollow feature does not exceed the extent of the solid and is a reasonable distance. Otherwise, the feature may fail.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Sketching on Faces

You can sketch on a planar face by selecting the face and selecting the **Plane of Object** command on the **Workplane** menu. This command generates a workplane coincident with the face and a new sketch. Then you can start drawing lines that are added to the sketch.   The workplane you created may fail to regenerate if the planar face it referenced before is not planar anymore during the update process.

If you do not create a workplane after selecting the face and you already have an active workplane coplanar with the face, you may start sketching thinking that you are creating a new sketch, but the newly drawn lines are added to the active sketch.   Later when you try to create a feature, you receive a warning that sketch is being used by multiple features.

The workplane that you create on a face has the same normal as that of the face. You should accordingly perform the operation above or below the workplane.

Note: The normal of a face always points outside the volume of the solid.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Using Constructed Workplanes

Sometimes you need to create bosses extruded or projected at an angle to an existing face or edge or a workplane. In such situations, you can construct a workplane that is either at an angle to a face or an existing workplane, or the mid-plane between two selected faces, or passing through selected edges.

If the face you want to draw the sketch on is not planar, construct a workplane by offsetting another workplane.

You can find the direction of the normal to the constructed workplane by looking at the arrowheads of the work axes. If the arrowheads are filled, you are looking from above the workplane; if they are not filled, you are looking at it from below the workplane. Use of this fact when making a decision to extrude or project a profile above or below a workplane.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Using the Workplane Browser

When you construct a workplane in terms of faces or edges of the model, the workplane is automatically repositioned, if necessary, when you use the **Update Design** command to update the design.   For example, if you have a workplane sited on the top face of an extruded profile, and you can use the **Properties** command to change the distance of the extrusion. When you do this, the workplane is updated when the design is updated so that it is still on that face, even though the face has moved.

You can use the workplane browser to find the workplane sequence that is guided by the order in which the features exist in the feature tree at that instance. You may name the workplanes according to their position in the browser.

The workplane browser is also useful for browsing sketches that are not part of any features that cannot be viewed from the feature browser.

The workplane that contains the active sketch is shown with its name in bold text.   If you expand that node in the browser, a list of sketches that belong to that workplane is displayed.   The active sketch is shown in bold.

You can also change the properties of the sketches and workplanes from the workplane browser. The workplane browser also displays workplanes with a mark if the workplane geometry cannot be determined.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Deleting Unused Workplanes

You can delete unused workplanes after you have finished constructing all the sketches for the part to unclutter the display and reduce the part size. Choose **Select WorkPlanes** from the toolbar and choose **Select All** from the **Edit** menu in the part window. All the workplanes in the document are highlighted. You can then press the **Delete** key on the keyboard or choose **Delete** from the **Edit** menu. At this point you receive the warning message: *Some of the selected objects are not allowed to be deleted.* This means that the workplanes that contain sketches cannot be deleted without deleting their dependents.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Browsing Your Features

You can find the makeup of a part in terms of features using the Feature browser. The browser provides the sequence of the feature operations at any instance of time. The browser also provides information about failed features, whether a feature is suppressed or condemned.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Examining Feature Properties

You can examine feature properties from the property sheet of that feature. The property sheet has two pages in it – one specific to a feature and the other is a generic feature page containing the name of the feature and **Suppressed** check box. If you get an error in creating the feature or during update and the error points to a specific feature parameter, you can examine this parameter from the feature property sheet.

You can also double-click on the edges or faces created by a particular feature to see the property sheet.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Making Changes to Your Features

You can make changes to your features in several different ways – you can modify the sketch for the feature after activating the sketch by adding, deleting lines or modifying the existing lines. You can also change the parameters of the feature from the property sheet of that feature. Property sheets and sketches for a feature can be selected from the feature browser.

You can also double-click on the edges or faces created by a particular feature to see the property sheet.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

About the Feature Browser

You can use the feature browser to view the sequence of features in a part. The browser also has a **Finish** flag and an **Update** flag. You can move the Finish flag to partially regenerate the feature tree. You cannot move the update flag – it moves depending upon the action you take. You can select a feature in the browser and select **Suppress**, **Condemn** from the popup menu. You can also select **Properties** to change feature properties such as name and parameters. If it is a failed feature you can select **Explain Problem** from the popup menu which displays a dialog box with the specific error message.

You can also double-click on each feature node to expand on the dependency list. For instance, extrusion 2 might expand to sketch 3, which, in turn, expands to workplane 4 , which depends on extrusion 1. Anywhere you see sketch in the browser you can select **Activate Sketch**, **New Sketch**, **Sketches** and **Properties** from the popup menu. You can use these commands to create a new sketch or modify an existing sketch. You can also get to the **Properties** of the workplane where you can change the name of the workplane or the offset distance.

You can find the edges or faces created by a particular feature by clicking on that edge or face and selecting **Synchronize Browser** from the **Select** menu in part window. The corresponding feature is highlighted in the browser.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Moving the Finish Flag

You can move the **Finish** flag  in the feature browser to update a portion of the feature list. You receive a message asking whether you want to update the design. You can update the model immediately or move the flag further before updating the model.

When you drag the **Finish** flag to a highlighted feature, you place it before that feature. Moving the Finish flag is an alternate way of regenerating the model without editing any feature properties.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Suppressing Features

You can suppress a feature by selecting the feature in the browser view and clicking on **Suppress** from the popup menu. The **Update** flag  of the browser jumps before the feature and **Suppress** is checked on the popup menu. At this time, **Update Design** is active also. The feature icons become

 from the original icon

You can also suppress a feature from the **Properties** sheet of a feature. Check the **Suppressed** check box in the **Feature** page. When you uncheck the box, you unsuppress the feature.

You may suppress features when they fail to update. You may also want to suppress all features of a particular type like all the blends and regenerate the model for rough analysis purposes. Suppressing a feature may lead to failures while updating other features, since some of the faces and edges in the model can become dormant.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Condemning Features

You can condemn a feature by selecting the feature in the browser and clicking on **Condemn** from the popup menu. When you condemn a feature all its dependent features are also condemned. The number of dependent features displays in a message box. The condemned features appear in the browser with a mark against them. The **Update** flag

jumps above the feature you condemn and **Update Design** becomes active.

When you update a part, if some feature fails, the update stops at that feature after reporting the problem. You have the option at that point to either mend the problem, suppress the feature, or condemn the feature. In some situations you might need to create some features solely for the purpose of constructing other features. You can condemn these features later on in the design process.

You can select all the features that you want to condemn from the browser and update the part in one shot. You can also use **Uncondemn All** from **Feature** menu to uncondemn the features.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Reordering Features

You can reorder a feature by selecting the feature in the Feature browser and dragging it in the browser view until another feature highlights in the browser. The dragged feature gets added to the browser before the highlighted feature. The part may change substantially when the features are reordered.

As an example you can start with a rectangular block as the base feature, hollow the block selecting the top face to be removed and put a throughole in the block using a sketch on the top face and extruding below workplane. A thin-walled solid is created with a hole at the bottom and its top open. If you now reorder the throughole, interchanging its position with that of the hollow, the resulting solid is quite different. The result is a thin-walled solid with a thin tube sticking up from the bottom of the solid.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Why Update Design can Fail

Update Design can fail due to geometric problems such as a taper angle that is too large for an extrusion, a face referenced by a workplane for its geometry is no longer planar, or a blend radius that is too large. Geometric problems can also arise if you create an invalid profile. The feature is shown in the feature tree with a mark against it to indicate that there is a problem with it.

Update Design can also fail due to relational problems. An example of this is when some edges become dormant in a blend feature or some of the faces become dormant for the hollow feature. A dormant error is shown with a mark against the feature in the browser.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

About Dormant Faces and Edges

When you update a model, there may be some faces and edges without any topological and geometric data associated with them. In other words, these faces and edges do not exist in the regenerated model in this instance. These entities are referred to as dormant entities. You can create dormant faces and edges by editing, suppressing, condemning, or reordering features.

For example, you can have a rectangular block as the base feature and a circular boss on the top face of the block. Next, you can blend the circular edge at the base of the boss. At this point of time, you will have three features in the browser – base feature, circular boss, and the blend. If you now **Suppress** the circular boss, the edge that you blended does not have any topological and geometric data associated with it anymore since the circular boss is suppressed. You receive an error message while regenerating the blend that objects that it depends on are currently dormant. If you **Unsuppress** the circular boss, the blend successfully regenerates as the circular edge "comes back to life" being associated with valid topological and geometric data.

Similarly, faces can become dormant too. In our example, if there was a workplane associated with the top circular planar face of the boss, after suppressing the boss the workplane cannot regenerate as the face that the workplane reference for its geometry is dormant now.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Moving a Sketch to a Different Workplane

You can change the workplane of the sketch by selecting **Properties** of the sketch and pressing the **Change** button. Subsequently you can select a workplane from the Workplane list in the **Choose Workplane** dialog box. The list contains all the workplanes that are not dependent on geometry produced by the sketch.

You can get to the property sheet of the sketch in a number of ways. You can use the **Sketches** command from the **Workplane** menu in the part window and press the **Properties** button in the **Sketches** dialog box. You can also select the sketch in the browser view and select **Properties** from the popup menu.

You might also want to copy a feature onto a different face rather than moving it, in which case you might need to **Copy** the lines of the sketch to clipboard and **Paste** them back onto the newly created workplane before performing the feature operation.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Blending all Edges Around a Face

You can select all the edges around a face by selecting the face and using the **Edges of Selected Faces** from the **Select** menu. You can also box-select all edges of the face. Once all the edges are selected you can use the **Blend Edges** from the **Feature** menu. The selected edges should not include any smooth edges. Smooth edges can arise because of previous blending operations.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Suppressing Features Within a Profile

You can have multiple closed loops in the profile as long as they are not intersecting. After placing the feature you can activate the sketch of the feature and toggle some of these closed loops into construction lines using the **Toggle Construction** on the **Line** menu. Closed loops that are turned into construction lines do not participate in the generation of the feature anymore and you can get the effect of a feature being suppressed. To unsuppress the feature, you can toggle construction lines again, after selecting the lines in the closed loops.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

Moving a Feature to a Different Face

You can create a workplane on a face by selecting the face and clicking **Plane of Object** on the **Workplane** menu. You can move a feature to a different face by creating a workplane on the face and moving the sketch for the feature to the new workplane.

See *Moving a Sketch to a Different Workplane* for more information.

See Also

{button ,AL(`featureModeling')} *Other topics on feature modeling*

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Notes can go here.

What is DesignWave?

Introducing DesignWave

DesignWave brings the power of parametric feature-based solid modeling, assembly modeling and feature-based drafting to the PC desktop as a ready-to-go productivity tool for design engineers. DesignWave offers dramatic gains over existing CAD technology, and marks the end of an era for 2D drafting systems for general-mechanical design.

With parametric feature-based solid modeling, you can create designs in less time, with fewer mistakes along the way. The feature-based drafting significantly enhances drawing productivity, so you can concentrate your efforts on design, rather than documentation.

Easy to Learn and Use

There is a common perception that 3D CAD is inherently difficult to use. DesignWave demonstrates that there is absolutely no reason why powerful high-end CAD technology cannot be harnessed as an easy-to-use design system on the PC desktop.

In fact, DesignWave is intuitive and discoverable. After just an hour or so of self-paced tutorial and exploration, you will feel confident and ready to start modeling and drafting real-life objects. So you do not have to use it everyday, all day long, to maintain a level of expertise.

General-Mechanical Design

DesignWave is designed for general-mechanical design and drafting. That is, commonplace hunks of solid material connecting functional form features of a mechanical or assembly-fit nature, and assemblies of such parts. The majority of mechanical products, or clearly identifiable sub-systems, fall into this category. It is easy to recognize those parts before you start. There is nothing more infuriating than getting 90% of the way through a 3D design, then coming across a limitation in the software.

See Also

{button ,AL(`intro')}   *Other topics in this introduction*

Setting Up DesignWave

Hardware

You must run DesignWave on a computer with an Intel or compatible CPU. The minimum hardware specification is a 90MHz Pentium with 16Mb RAM. For serious use, 32Mb RAM or more is recommended. DesignWave runs on Windows 95 or Windows NT v4.

You can check your processor and RAM by opening the **System** icon in the **Control Panel**.

`{button Take Me There,EF(`SysDM.cpl',`System,0')}`

Graphics

To run DesignWave, set your display for 16 bits of color or more (65536 colors, or high color). You can use 24 bits of color (true color), but DesignWave will not make use of the extra colors available. DesignWave will *not* work if the display is set for 8 bits of color (256 colors).

The recommended desktop size is 1024x768 pixels or larger for serious use, although DesignWave will run with any desktop size.

You can check all your display settings by opening the **Display** icon in the **Control Panel** and going to the **Settings** page. It is common for a display driver to add an extra page to this dialog, usually just before the **Settings** page. If this is the case for you, then you should use the settings on that page instead.

`{button Take Me There,EF(`Desk.cpl',`Display,3')}`

Printing

You can print DesignWave drawings to any raster printer or plotter. These include ink-jet, bubble-jet, and laser printers and plotters. Vector devices, such as pen plotters, are not supported.

See Also

`{button ,AL(`intro')}` *Other topics in this introduction*

Support

If you have a web browser, you can access the DesignWave support web page by pressing this button:

`{button Support Information,EF("index.htm",`',1)}`

See Also

`{button ,AL(`intro')}` *Other topics in this introduction*

About the User Interface

Objects

The word *object* is the generic term for any identifiable entity within DesignWave. Examples of objects include parts, solids, faces, edges, features, mating conditions, workplanes, sketches, lines, drawings, drawing views, dimension lines, leaders, callout groups and callouts.

Notice that some objects have no physical presence in their own right. You cannot *see* a feature, although you can see the faces and edges that are produced as a result. You cannot *see* a sketch, but you can see the lines that it contains. Even more abstract, is a mating condition, which you certainly cannot *see*, although you can see that the component is in the right place.

You can select objects that you *can* see in views or drawings. Object that you *cannot* see appear as icons in the browser, down the left side of the part window, to enable you to select them.

Object-Action

DesignWave uses an *object-action* paradigm. This means that you select your objects first, and then choose a command to operate on them. Commands on the menu are appropriate for the selected object. When the command has executed, the objects remain selected, ready for you to choose another command.

Modes

You choose a selection mode for the type of object you want to select. For example, you might be selecting lines, selecting edges, selecting faces, or selecting annotations. The **Select** menu lists the different selection modes, and the check mark next to the menu entry indicates the active interface mode. Selection modes are also available on the toolbar, where the depressed button indicates the active mode.

There are other modes that are not selection modes, such as creating circles or trimming line segments. When you are not in a selection mode, you can press the ESCAPE key to enter the selection mode most closely related to the mode that you are in. So, if you are creating circles, pressing ESCAPE puts you into the mode for selecting lines. As an alternative to pressing the ESCAPE key, you can click in the view without dragging.

See Also

{button ,AL(`intro')}   *Other topics in this introduction*

Finding Commands

The menus that you see on the menu bar depend on whether a part window or a drawing window is active. A window is active when its title bar is highlighted, usually in blue unless you have customized your desktop.

There are various short cuts that allow you to access commands quickly. Toolbars present graphical buttons that activate commands. Popup menus, displayed by clicking the right mouse button, offer a short list of common commands. Keyboard key combinations provide quick access to commands. These short cuts are a convenience and a productivity aid. You will quickly learn the short cuts that suit your style of working.

In general, all commands can be found on menus presented in the menu bar. The exceptions to this rule are the commands on browser popup menus - for example, the **Suppress** command in the feature browser. This is because these commands operate on the icon in the browser, whereas menu bar commands apply to the selected objects in the view. So if you have trouble finding a command, perhaps that is where it is.

See Also

{button ,AL(`intro')}   *Other topics in this introduction*

The Undo Command

If you make a mistake, or change your mind about what you just did, you can choose the **Undo** command on the **Edit** menu to go back one step. The main toolbar also has a button for **Undo** and the text next to it tells you what will be undone if you press it.

Undo affects changes to parts and drawings, but does not affect window commands. For example, if you use **Delete** to remove a line, and then rotate the view and zoom in to make things a little larger, the text of the **Undo** command still says "Delete". Choosing **Undo** will bring back your line, but the view rotation and zoom will not be affected.

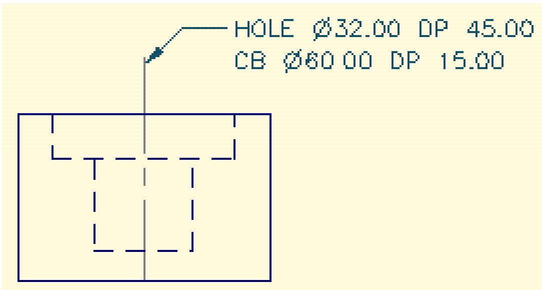
See Also

{button ,AL(`intro')} *Other topics in this introduction*

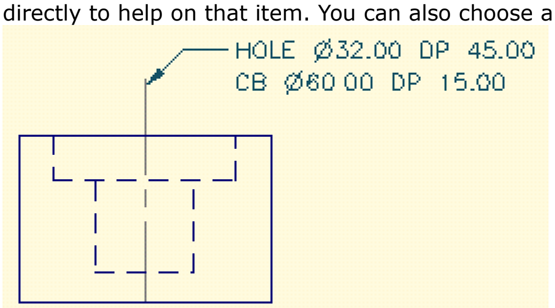
Using Context-Sensitive Help

DesignWave comes with extensive on-line help. The **Help Topics** command offers you a **Contents** page, an **Index** page for keyword searches, and a **Find** page for a full search of the help text. In addition, there is context-sensitive help for commands, messages and dialog windows.

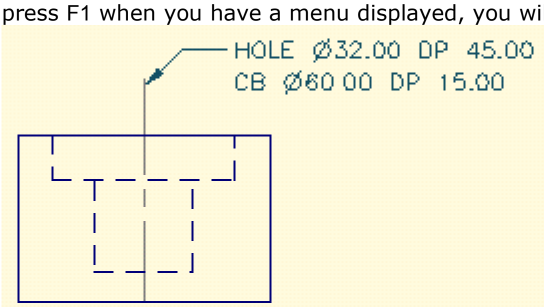
There are three ways to access context-sensitive help:



Enter the help mode. To do this, you can choose the **Context Sensitive Help** command on the **Help** menu, or you can press the SHIFT-F1 key combination. You will see the cursor change to indicate that you are now in the help mode. In this mode, you can click on anything you can see to jump directly to help on that item. You can also choose a command from a menu to get help on that command.



You can press the F1 key at any time you need help about your situation. If a message window comes up, pressing F1 gives you further explanation of the message. If you press F1 when you have a menu displayed, you will get help for the highlighted command.



When a dialog window is displayed, you can press the help button at the top right of the window to enter the help mode. You will see the cursor change to indicate that you are now in the help mode. In this mode, you can click on any control in the dialog window to get a popup message explaining what that control is for. Alternatively, you can select the **What's This?** command from the popup menu when you are over the control.

When a dialog window is displayed, the context-sensitive help provided is for each dialog control in turn. For help on using the command that displayed that dialog window, you should press the CANCEL button to dismiss the dialog, press the SHIFT-F1 key combination, and then select the same command again.

See Also

{button ,AL(`intro')} *Other topics in this introduction*

Creating Objects

To create common objects, such as lines and dimensions, you must first enter the appropriate creation mode. The window toolbar offers the creation modes, and the button that is depressed indicates the current mode. You create these objects by dragging in the window. As you drag the cursor, DesignWave shows you dynamic feedback of what will be created when you release the mouse button. To cancel the creation, press the ESCAPE key.

Some objects use dialog window input instead of creation modes. For example, there is no creation mode for creating an extruded profile feature, but there is a command called **Extrude Profile** instead.

See Also

{button ,AL(`intro')} *Other topics in this introduction*

Selecting Objects

Preselection Highlighting

As you move the cursor over objects in the window, the objects light up to indicate that they are selectable. If you are selecting lines, only lines light up. If you are selecting edges, only edges light up. This is called preselection highlighting and it means you know exactly what will be selected before you commit yourself to clicking the mouse button.

Selecting Several Objects

If you press the left mouse button down and release it, the prehighlighted object is selected and highlighted in red. If instead, you drag the mouse while the button is down, you can box-select all those objects that lie within the selection rectangle that appears.

If you hold the SHIFT key down, and press then release the mouse button without dragging, the pre-highlighted object is added to the selection if it is not already selected. Otherwise, it is removed from the selection. If you hold the SHIFT key down while box-selecting, the new objects are added to the selection, even if some of them were already selected.

To cancel a box-select when you are in the middle of dragging the box, press the ESCAPE key. To clear the selection, click on the background where there is no selectable object.

See Also

{button ,AL(`intro')}   *Other topics in this introduction*

Modifying Objects

You can modify the objects you created by dragging in the window (for example, lines and dimensions) by selecting and dragging. Dragging from different places on the object often produces different modification behaviors. For example, dragging the end point of a straight line repositions that end point, whereas dragging the middle of the line bends the line into an arc.

See Also

{button ,AL(`intro')}   *Other topics in this introduction*

Editing Properties

Objects have properties that can be edited. A circle has a center and a radius. A dimension callout has units, number of decimal places, and various other properties to do with its appearance.

To change the properties of an object, select it, then choose the **Properties** command from the **Edit** menu. You can also access the **Properties** command by double-clicking on the object. For objects that appear in the browser, such as sketches and features, the popup menu gives you a short cut to the **Properties** command for the object in the browser under the cursor.

See Also

{button ,AL(`intro')} *Other topics in this introduction*

Using Viewing Commands

The viewing commands - **Autoscale**, **Zoom In**, **Half Scale** and **Pan** – allow you to change the scale and center of your view scene. They affect only the active window, so if you have another window displaying the same subject, that other window is unaffected. See *Creating Additional Windows* for more information. You can use viewing commands at any time, except when a dialog window is displayed.

If a part window is active, you can also rotate the view using the arrow key accelerators. Commands are also offered on the **View** menu for this.

See Also

{button ,AL(`intro')}   *Other topics in this introduction*

Creating Additional Windows

The **New Window** command on the **Window** menu opens another window showing the same part or drawing. With two windows, you can look at the part or drawing in two different ways at the same time.

If you have several windows displaying the same part or drawing subject, changes made to the subject in one window are immediately reflected in the other windows. Likewise, the current selection is shown highlighted in all windows. The interaction mode is also the same for all windows.

The viewing commands - **Autoscale**, **Zoom In**, **Half Scale**, and **Pan** - do not affect other windows displaying the same subject. So you can have one window showing the entire subject, and another - perhaps to the side of it - zoomed in to show some detail. You can make selections in either window.

With two part windows displaying the same part, you can use the **Shaded** command on the **View** menu to show the part as a shaded image in one window, while displaying is a wire frame image in the other. You can also rotate the views so that each window shows the part from a different viewing direction.

See Also

{button ,AL(`intro')}   *Other topics in this introduction*

Splitting a Part Window

For part windows, you can use the **Split** command on the **Window** menu to break up the view and reveal additional panes. You can produce four panes by clicking within the viewing area, or two panes by clicking above, below, or off to one side of the window.

With four panes, the main view appears in the bottom right pane. The other three panes are linked by a 3rd angle projection scheme. If you zoom or pan in one of these panes, the linked panes are automatically adjusted to maintain the projection scheme. The top-left pane shows a plan view, the same view displayed for the main view if you choose the **Plan** command on the **View** menu.

With two panes, the main view moves to the right or below the new pane. The new pane shows a plan view, the same view displayed for the main view if you choose the **Plan** command on the **View** menu. You can zoom and pan the two panes independently.

Working with Split Windows

You can rotate only the main view. Although you can zoom and pan the other panes, you cannot rotate them. Since you can work in any of the panes, you can select objects or create lines in whichever pane is the most convenient. You cannot create lines or otherwise use workplane points in a pane displaying a side view of the workplane.

Unsplitting a Window

To unsplit a window, you can either drag the pane dividers off the window, or you can double-click on them. Double-clicking on the intersection of the dividers in a four pane window returns the window to a single pane, Double-clicking on just one of the dividers removes that divider, producing a two-pane window.

See Also

{button ,AL(`intro')} *Other topics in this introduction*

Setting User Options

The interaction required when creating objects in DesignWave is kept to a minimum by taking many of the inputs from settings that you can establish using the **Options** command on the **Tools** menu. For example, the number of decimal places used for all dimensions created can be set up here.

You can override these settings by using the **Properties** command after you have created the object.

The settings belong to you as user preferences. If someone else logs onto your computer, they will have their own settings.

See Also

{button ,AL(`intro')} *Other topics in this introduction*

Saving Your Work

Parts and Drawings

You can use the **Save** command on the **File** menu to save what you are working on. The title bar of the part window or drawing window shows the name of the part or drawing displayed in that window. If you have never saved it, the title bar will show Part3 or Drawing3 where “3” is a number that is incremented for each new part or drawing you create.

An asterisk after the name, as in Bracket.cvp *, tells you that changes to the part or drawing have been made since it was opened. New parts and drawings have an asterisk too. When you see an asterisk, this means that the part or drawing needs saving. You can save it at any time or use the **Close** command on the **File** menu and discard the changes.

Sessions

The **Save Session** command on the **File** menu saves the entire session. When you save your session, any open parts and drawings that have changed are saved, as well as a record of open files and window positions. You can open the session later and continue where you left off.

If you work on many projects, you can use sessions to represent the various projects. The name you give for the session can identify the project. If you only work on one project, you might simply name the session work.cvs.

See Also

{button ,AL(`intro')} *Other topics in this introduction*

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Notes can go here.

Using Measurements

Use the **Measure** command to determine the length, radius or diameter of an object, and the distance between two objects. This includes the measurement of the distance between a geometric object and the active workplane or work axes.

To measure an object, choose the **Measure** Command on the **Tools** menu. Then select a measurement and choose the **Add** button to display the active measured value in the text field of the dialog. The values in the text field are then available on the numeric list boxes of other commands, for example, the extrude distance numeric list box.

You can invoke the calculator from the dialog to perform mathematical operations on measured values before using them in a subsequent command.

See Also

{button ,AL(`misc')} *Other topics in this section*

Tips for Working with Large Models

Minimize windows you don't need

Whenever you make changes to a model in one window, the changes are automatically displayed in any other drawing and assembly windows in which that model can be seen. If you have many windows open with complex parts or drawings, the time needed to update the windows can be significant. Minimized windows and windows hidden behind a maximized window are skipped during window update. So, when working with large models or with lots of models simultaneously, either minimize unused windows, or maximize the window you are working in to improve performance. The changes you make will be applied as a batch when the window is revealed later.

Suppress features that are not important for the time being

You can use the **Suppress** command on the popup menu inside the feature browser to take selected features out of service. While features are suppressed, **Update Design** takes less time since there are fewer features to process. Later, you can deselect the **Suppress** command to return the features to service.

Build a large model in stages

Each time you make a change to a model, the model regenerates from scratch. You can avoid this by creating a new part and using the **Add Component** command on the **Assembly** menu as a means of check-pointing your design at a meaningful state.

Create your model and, at a good breaking point, save it. Now create a second part and use the **Add Component** command on the **Assembly** menu to add the basic design part as a component. There is no need to position the component in any way. Choose the **Use Component** command on the **Feature** menu to turn the component into a derived model that you can develop further.

Now you can add the detail features to the second part. Changes to the second part will only regenerate from the state of the model as it was brought in through **Add Component**. Changes to the original part can be effected into the second part using **Update Design** on the modeling toolbar.

See Also

{button ,AL(`misc')} *Other topics in this section*

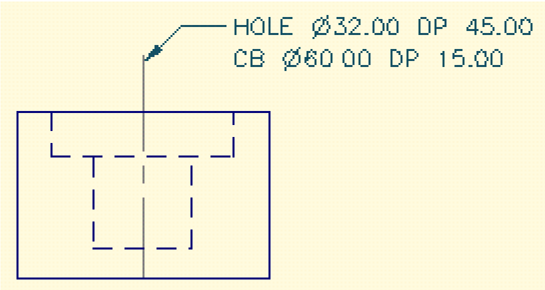
Importing and Exporting Parts

Importing and exporting parts allows you to save parts in formats that can be used by other applications. You can import and export a model in Parasolid™ transmit file format and output in VRML format. You export a model in Parasolid™ or VRML format using the **Save Copy** or **Save As VRML** commands, respectively, from the **File** menu. You can import a Parasolid™ transmit file using the **Open** command from the **File** menu. An existing Parasolid transmit file must be specified as the input.

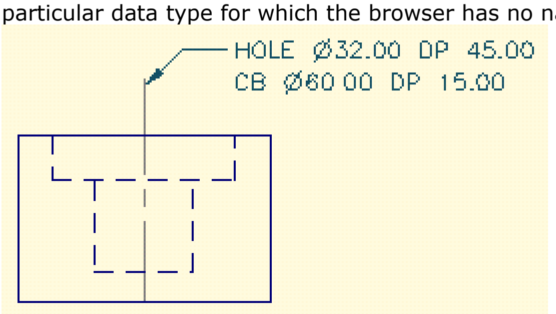
The ability to import and export models in Parasolid™ format allows you to share models with an application that supports the Parasolid™ transmit file format. The **Import** command allows you to read in another modeler's Parasolid™ transmit file and use it in advanced DesignWave environments. You can read in a model to add more features to it, produce a drawing from it, or use it in an assembly. The exporting of a DesignWave model in Parasolid™ transmit file allows you to work with your model in other applications. This would include other Parasolid™ based modelers as well as other applications, such as translators, that take the Parasolid transmit file format as input.

DesignWave provides fully automatic output of solids and assemblies in VRML V1.0 format. This allows you to build virtual worlds, or share virtual mockups over the company intranet, or perhaps even with customers over the Internet. You can view DesignWave parts with your favorite VRML viewer to produce marketing quality pictures.

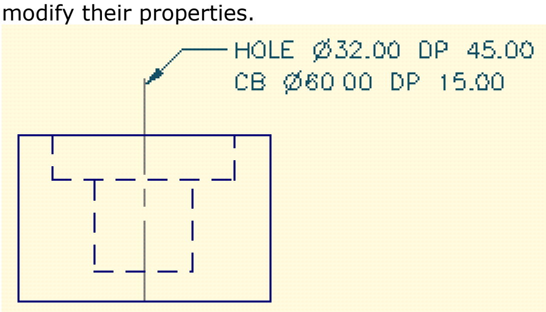
There are three basic types of VRML viewer:



Browser plug-ins - browser extensions designed to handle a particular data type for which the browser has no native support. A plug-in runs as if it is part of the browser itself.



Separate viewer applications - A separate VRML viewer application is a program that reads VRML files and displays objects, allowing you to manipulate them, and perhaps modify their properties.



Virtual world navigators - Virtual world navigators are the 3D space equivalent of 2D page-oriented web browsers. They display VRML world scenes, and let you move within them.

See Also

{button ,AL(`misc')} *Other topics in this section*

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Notes can go here.

Introduction to Sketching

All lines that you create live in sketches, which belong to workplanes or drawings.

A sketch can contain any number of lines. A line can be a straight line, an arc, or a circle. You use sketches to describe a profile to be used when creating a feature. Sketches are also used to define the cut when creating a section view on a drawing and a clipping profile when creating a partial view. Commands that use sketches as input often have rules that must apply to the lines in the sketch for that sketch to be valid.

The Active Sketch

When you create a new line, it is automatically added to the active sketch. You can change the active sketch at any time by using the **Sketches** command on the **Workplane** menu for a part window or on the **Drawing** menu for a drawing window. You can also use the **Activate** command on the popup menu whenever you can see the sketch in the browser view.

Naming Sketches

Sketches have names. The name is used whenever the sketch is shown in the browser view. Although the system generates a default name, such as "sketch 3", it is good to rename your sketch with a name that captures the intended use of the sketch. You can name the sketch when you create it or later by editing the properties of the sketch.

Hiding Sketches

The **Sketches** command also allows you to hide sketches. This de-clutters your view, and also stops points from lines in other sketches being inadvertently selected as you create new lines in the active sketch.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Using the Work Axes

Each workplane has a pair of work axes that defines a 2D (x,y) coordinate system for the workplane. Although the lines you create in sketches are 3D lines, they are described in terms of 2D coordinates in the workplane. DesignWave never asks you to enter a (x,y,z) coordinate.

The work axes belong to the workplane, not the active sketch. If you activate another sketch in the same workplane, the work axes are the same. The work axes of another workplane generally have a different position and orientation.

The work axes are yours to move around and orient. You can move and orient the work axes using the **Reposition Axes** command. You can also use the **Transform Axes** command to translate or rotate them by a specific amount.

Local Coordinate System

When you see (x,y) points displayed in the dialog bar, these are always relative to the work axes. Similarly, when you use the **Properties** command to examine or modify the geometry of a line, the 2D points shown in the dialog window are relative to the work axes.

Local Set Square

When creating straight lines, if you hold the SHIFT key down, the line snaps to the closest of the two work axes directions. This provides a quick and easy way for you to lay down parallel and perpendicular lines. See *Creating Lines* for more information.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Selecting Existing Points

Attraction Points

Whenever you are invited to select points in the active workplane, existing points are pre-highlighted as black squares as you move the cursor over them. These are called *attraction points*. Large black squares indicate precise points: end-points, mid-points, intersection points, and centers of circles or arcs. Small black squares indicate a point on the line close to the cursor.

Circles and Arcs

For circles and arcs, if the cursor is slightly on the inside of the line, the center point is offered. If the cursor is slightly outside the line, the point on the circumference is offered. When creating straight lines, a point on the circumference is interpreted as a tangent – see *Creating Lines* for more information.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Selecting Grid Points

When you select a point that is not an attraction point, DesignWave snaps it to an invisible grid so that you get sensible values for X and Y. The grid is completely automatic, so you do not need to manually set a grid size anywhere. DesignWave selects a grid size based on the zoom factor of the view or drawing. To get a finer grid, just choose **Zoom In** from the popup menu on the right mouse button. To get a coarser grid, choose **Half Scale** instead.

How the Grid Works

First of all, DesignWave looks to see what units you are working in. You can set the units using the **Options** command on the **Tools** menu, and going to the **Units** page. For part windows, the units for model distances are used. For drawing windows, the units for paper distances are used.

Then DesignWave determines a grid size based on these units and using a multiplier of 1, 2, or 5 with some power of ten. For example, if you are working in centimeters, you get grid sizes from the following series:

... 0.01cm, 0.02cm, 0.05cm, 0.1cm, 0.2cm, 0.5cm, 1cm, 2cm, 5cm, 10cm, ...

whereas if you are working in inches, you get grid sizes from this series instead:

... 0.01in, 0.02in, 0.05in, 0.1in, 0.2in, 0.5in, 1in, 2in, 5in, 10in, ...

The grid size DesignWave chooses is the finest one from the series that can be used effectively to select grid points easily using freehand movement of the mouse. The grid size is about 3 pixels on the screen.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Creating Lines

Creating Straight Lines

You create straight lines by choosing the **Straight** command from the **Line** menu. This puts DesignWave into an interface mode in which you can create straight lines just by dragging in the workplane. As you move the cursor around in the view, DesignWave prehighlights lines and also offers attraction points for end-points, mid-points, centers, and intersection points involving those lines, so you can use these points directly. See *Selecting Existing Points* for more on attraction points.

To create a straight line, press the left mouse button down for the start point, drag to the end point, and release the button. Attraction points and grid points are offered while you are dragging too, so you can select a precise end-point for your line. If you hold the SHIFT key down while dragging, DesignWave constrains the line so as to lie parallel to the closest work axis direction. You can still select from points off this line, and DesignWave will project points onto the line.

Creating Tangent Lines

As a special case for creating straight lines only, DesignWave interprets selecting an attraction point on the circumference of a circle or circular arc as a tangency requirement. You can select a point on the circumference by placing the cursor close to, but just outside, the line. Placing the cursor just inside a circular line selects the center point. As you drag away from the circular line, DesignWave gives you dynamic feedback of the tangent line. If you drag towards another circular line, again just outside it, the end point is tangent.

If you want to use a point on the circumference directly, and not have DesignWave infer a tangency requirement, you can create a line that cuts across the circumference first, then select the intersection point produced. DesignWave interprets intersection points and end-points as a stronger requirement than tangency.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Using Construction Lines

Construction lines in a sketch are used for a number of reasons. In case of modeling, construction lines in the sketch are ignored as far as the profile definition is concerned. This allows you to use construction lines freely to define the underlying profile for a feature. When defining the cut line for a section view, a construction line is used to indicate the principal section line and which side of it to generate the section view. Construction lines will not appear when you print the drawing.

See Also

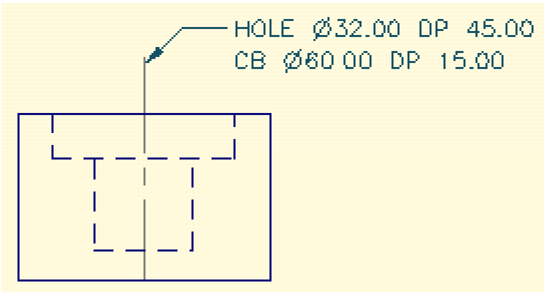
{button ,AL(`sketching')} *Other topics on sketching*

Modifying Lines

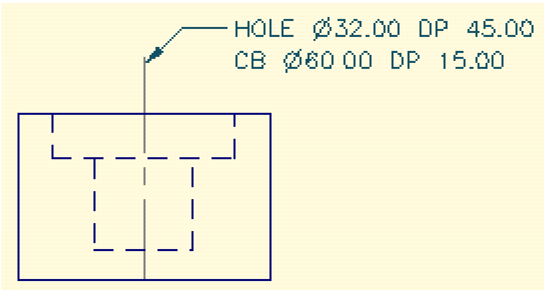
Modifying Straight Lines

To modify a straight line, you must first select it. You can select lines by choosing the **Lines** command from the **Select** menu. If you are currently in another interface mode for lines - such as creating straights, deleting segments, or filleting lines - you can press the ESCAPE key instead.

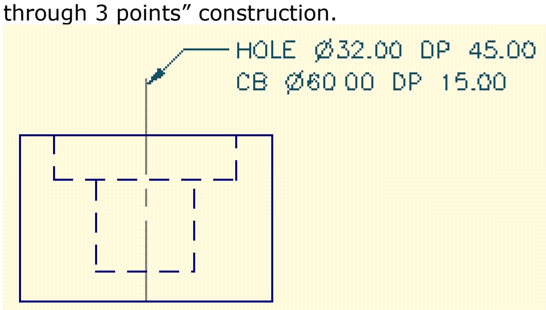
The following modifications are possible by dragging parts of the line:



Dragging an end-point repositions that end-point.



Dragging from near the mid-point bends a straight line into a circular arc that passes through the point at which you release the mouse button. This provides you with an “arc through 3 points” construction.



Dragging an end-point with the SHIFT key held down extends or shortens the line, while maintaining its underlying geometric form. As you drag, you can select points away from the line, and DesignWave will project them perpendicularly to the line. If you drag to a point on another line that is not an end point or mid-point, the line being modified is extended to intersect with that other line.

To move a line, you should use either the **Transform** command or **Shift Point-to-Point** command, which are on the Edit menu.

Editing Line Properties

You can change line properties by selecting a line from the sketch and using the **Properties** command on the **Edit** menu. As an alternative you can also double-click on the line to display the **Properties** dialog window.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Editing Line Geometry

You can modify line geometry in several ways. One way is to modify it using the **Properties** dialog box. To interactively edit the line geometry, drag one of its endpoints to the desired location. You can also bend a straight line into an arc by pulling it from the middle. Different cursors indicate the intent of the modification.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Using the Clipboard with Lines

You can use the clipboard commands - **Copy**, **Cut**, and **Paste** - on the **Edit** menu to move or copy lines. When lines are placed on the clipboard using **Copy** or **Cut**, DesignWave makes a note of the where the work axes are. When you **Paste** the lines, they are added to the active sketch in the same place relative to the current work axes.

This allows you to do several things:

You can copy lines from another sketch, even one from another part. Perhaps you are designing a part that has a face that will bolt to a similar face in a mating part. You can **Copy** the lines you want from the profile sketch in the mating part, and **Paste** them into the sketch you are working on.

You can move lines to a different sketch. Perhaps you have created several lines, and then you suddenly realize the wrong sketch is active. Just select and **Cut** the lines, then activate the correct sketch and **Paste** the lines into it.

You can reposition lines by repositioning the work axes. For example, you can move a 2D feature in your profile, such as a keyhole, to some new place with a new orientation. First, you place the work axes to represent a local origin and orientation for the 2D feature. Next, you select and **Cut** the lines. Then, you reposition the work axes, and **Paste** the lines into their new position. The topic *Using the Work Axes* describes various ways to position the work axes.

You can produce copies of lines in different positions in the same sketch. This gives a convenient means to create pattern of features. The **Duplicate** command under the **Edit** menu gives you the flexibility to create rectangular or circular patterns of sketches.

You can produce mirror images of lines by reversing the work axes before you **Paste**. Use this technique to make symmetric profiles.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Pasting Line Properties

You can copy and paste line properties onto other lines. Use this technique to create colinear lines, concentric arcs and circles, or equal length/radii segments.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Using Existing Model Geometry

Sketch geometry can reference existing model entities. To find the attraction points of model edges, turn the **Edges** check box on. The system highlights the endpoints and mid-points of edges.

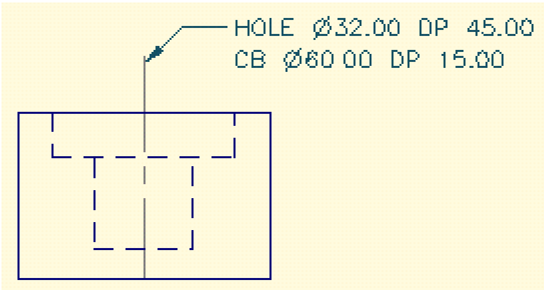
See Also

{button ,AL(`sketching')} *Other topics on sketching*

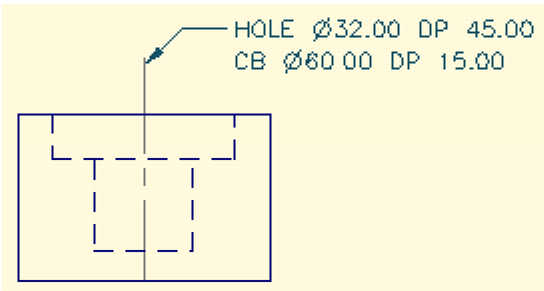
Fixing Sketches

Depending on what the sketch is to be used for, there are often rules about what constitutes a valid sketch. In particular, there are rules that must apply for a sketch to be a valid profile for feature modeling – see *About Profiles* for details. And there are rules that must apply for valid section cut sketches – see *About Section Views* for details.

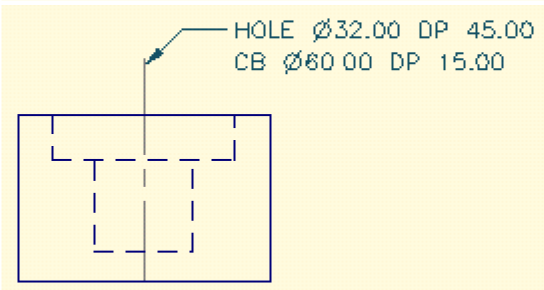
There are some techniques for examining and fixing sketches that are invalid. See the following topics:



Checking Connectivity.



Finding Gaps Between Lines.



Dealing With Overlapping Lines.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Checking Connectivity

Some sketches are not valid for feature operations because they do not form closed profiles. You can check the connectivity of a sketch by selecting a line segment and using the **Add Connected Lines** from the **Select** menu. Any lines in the sketch not picked up by this command are not connected.

Note: It is perfectly valid to have unconnected construction lines.

See *Fixing Sketches* for related information.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Finding Gaps Between Lines

Sometimes sketches that look connected may not be, due to gaps between lines. To find the gaps, use the **Fillet Lines** command under the **Line** menu. Use a random fillet radius to examine all the corners. If, at a corner, the lines that form the corner do not highlight, you have located a gap between lines. Use the **Zoom In** command to explore the corner further. Then fix the gap by connecting the two lines by dragging one of them or modifying them explicitly using the **Properties** pages.

The topic *Modifying Lines* describes various ways to modify lines.

See *Fixing Sketches* for related information.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Dealing With Overlapping Lines

Sometimes sketches are invalid profiles because they contain overlapping line segments. An easy way to determine the stray lines is to use **Select Lines** command and manually select all the lines that form the valid profile. Do not use **Add Connected Lines**, since it might pick up overlapping lines that are completely identical. Once the lines have been selected, use the **Transform** command from the **Edit** menu and **Rotate** the selection by 15 degrees. Then refresh the screen by using any view command such as **Autoscale** (or any of the arrow keys). If you find stray geometry, delete it. Then use the **Select All** command followed by **Transform** and give a negative -15 degrees rotation to get back to your original geometric location.

See *Fixing Sketches* for related information.

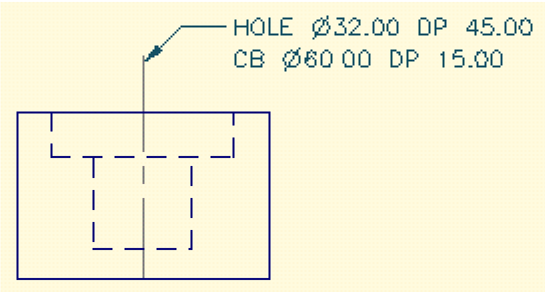
See Also

{button ,AL(`sketching')} *Other topics on sketching*

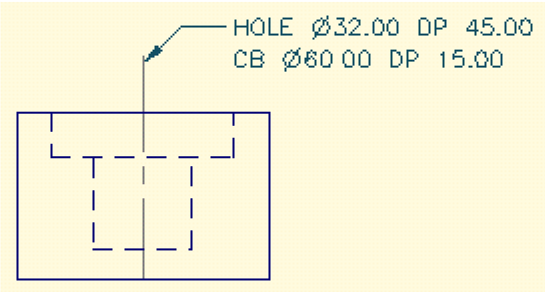
Managing Your Sketches

You can manage sketches by using the **Sketches** command under the **Workplane** menu. A dialog box appears, listing all the sketches in the active part. Use this dialog to activate a particular sketch, delete a sketch, or change the properties of a sketch. You can also use the **Hide Other Sketches** command to reduce the screen clutter.

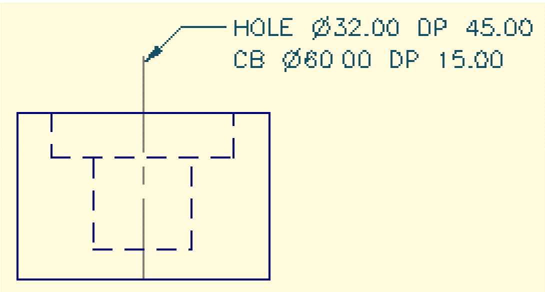
About the active sketch:



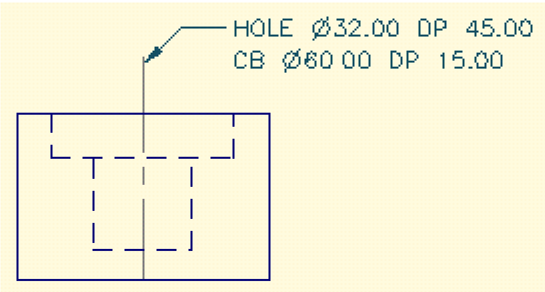
Newly created lines are added to the active sketch.



Construction lines are visible.

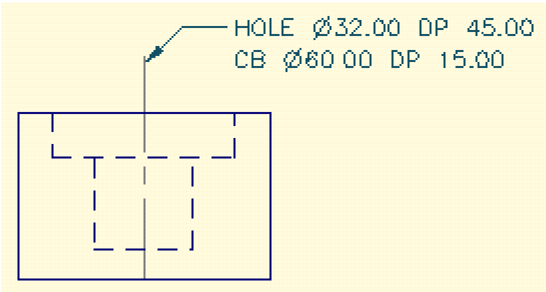


All lines in the sketch are selectable.

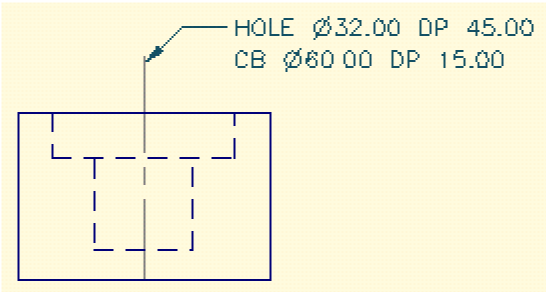


Attraction points involving lines are selectable.

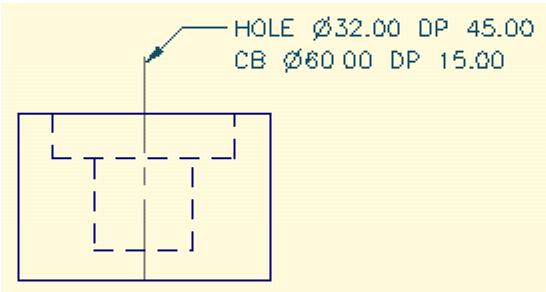
About visible sketches other than the active sketch:



Construction lines are not visible.

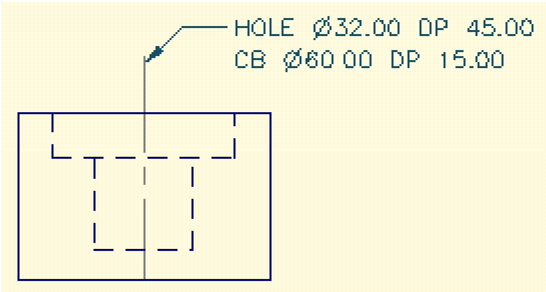


Lines in the sketch are not selectable.



Attraction points involving lines are selectable.

About invisible sketches:



No lines in the sketch are shown.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

Deleting Unused Sketches

You can delete a sketch that is not being used for any feature or section view by examining the number of times it has been used. To do this, use the **Sketches** command from the **Workplane** menu. The *Delete* button on the **Sketches** dialog is active only if the sketch is not being used. To find out how many times a particular sketch has been referenced, click on the sketch name and get its Properties. The properties dialog shows its number of uses.

See Also

{button ,AL(`sketching')} *Other topics on sketching*

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Notes can go here.

Concurrent Design with DesignWave

Concurrent Design

DesignWave supports concurrent design through its implementation of inter-file references and search paths. By straightforward use of the directory structure, you can support an environment where a design evolves while respecting the concepts of an ECO and approval process. This brings you an environment that supports team-based design and the concurrent development of a product.

With concurrent design, you can work on the drawing of a model at the same time a colleague is changing the part model. While your session is active and the models that it references are loaded, you are insulated from changes made to those models. The next time you open the drawing, you can choose the **Update Views** command on the **Drawing** toolbar to update your drawing to reflect changes made to the parts displayed. This process is identical for assemblies, as well as parts.

Concurrent Design Scenario

Imagine you have several people working as a team on the design of a product. In addition to creating parts at the same time, team members can also produce the drawings required to manufacture the parts. Early in the project, team members are working individually in their own private directories and do not see the work of their colleagues.

When a part or drawing is complete, you move it to a common "product" directory on the network. This common directory is in the search path of each team member, as specified with the **Options** command on the **Tools** menu. When a part or drawing is in this common area, the other members of the team can see it.

Assembling of the parts can begin at any time after multiple parts have been checked in. After you have added all of the parts to the assembly, you can add the assembly to the shared directory. This provides all members of the team access to the assembly.

In the meantime, you can respond to an ECO on your part by copying it back to your local private directory and working on it. Changes will not affect the shared design. When you are satisfied with your changes, you can submit your part to an approval process. Your team members can copy your part to their local directory to see your changes, or add your private directory ahead of the shared product directory in the Search Path. Once approved, you move your part to the shared directory, and all other parts and drawings can be updated to reflect your changes.

See Also

{button ,AL(`teamDesign')}   *Other topics in this section*

Sharing Information

Inter-File References

When you add a view to a drawing, DesignWave creates a reference to the part file. The same is true when you add a component to an assembly.

You can share files with your team members by defining a common directory on the network for product parts and drawings. You use this shared “product” directory as the set of approved files for a project that you are working on collaboratively. Each team member would also have a private local directory for making changes. By working in a local private directory, your changes would not affect any of the other accepted product parts and drawings. Once you are satisfied with your changes, you move them to the shared directory on the network. This makes your changes visible to all of the approved parts and drawings, and they can be updated to reflect your changes.

By adding the shared “product” directory to each the team member’s search path, the team member has access to all of the parts and drawings related to the product. This reduces the amount of copied parts and drawings, as well as the potential logistical problem of keeping them up-to-date and synchronized.

File Locking

DesignWave does not support file locking. This means that if several people have copied a part to their local directory, the last person to move the part back into the shared “product” directory overwrites the other changes. Only the last team member's changes will be reflected in the accepted design.

Check Out and Check In

You can avoid problems by using the read/write privileges of file system to control concurrent access to a file. A team member who copies a file to the local directory should change the access privileges to read-only on the common part. In this way, another team member who copies the part is notified that changes cannot be written because the part is read-only. You can adopt the practice of checking the permissions of the file as part of the process of copying it to a local directory, which provides the team with a mechanism for knowing when another team member is currently working on a file.

See Also

{button ,AL(`teamDesign')}   *Other topics in this section*

Using Search Paths

Search Path

When you open an assembly or drawing, other files that are referenced are loaded automatically. You are at liberty to move files around on your disk. As a consequence, a referenced file may no longer be found in the pathname that it was known to be in at the time the assembly or drawing was saved. When locating referenced files, if they have moved, the directory search path is examined in order to locate the file.

You can set up your search path using the **Options** command from the **Tools** menu. On the **Directories** page you can list the folders to be searched in order when locating files. Your search path is remembered between sessions.

Finding Files

When a file has to be resolved, it is looked for in the following places, in order:

1. At the pathname know at the time the assembly or drawing was saved.
2. In the same directory as the referencing assembly or drawing.
3. In each of the directories listed in the search path in turn.

If the file still cannot be found, you are prompted proceed assuming the file is deleted, or to cancel the open.

See Also

{button ,AL(`teamDesign')}   *Other topics in this section*

Understanding Versions

File Versions

Each time you save a part or drawing, an internal version number is incremented for that file. If an assembly or drawing is opened, and a referenced file is found to have a different version number, update occurs automatically to reflect any changes made between those versions.

The version will typically have increased, but by using search paths, it is possible to encounter an earlier version of the file. Update will still occur automatically.

A Warning about Version Branching

If two people take a root version of a file and start independently producing higher versions as two separate branches, you should note carefully that files from one branch are incompatible with files from the other branch. If you save an assembly using a part file version from one branch, you must ensure that your search path will not allow a version from the other branch to be loaded. Otherwise errors will occur and parts could get corrupted.

Version branching in this way is not recommended. If you want to evaluate alternative designs, you should use derived models instead. See *Using Derived Models to Capture a Process* for details.

See Also

{button ,AL(`teamDesign')}   *Other topics in this section*