

Alibre Design 8.2

User Guide

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1 Installation

Thank you for choosing Alibre Design! We welcome you to the ever growing community of professional engineers who have discovered the high value of Alibre Design's feature-rich modeling capability, unique team design features and modest price.

Alibre Design is easily installed on your computer. By following the directions in this chapter, you can be up and running in a matter of minutes.

This chapter describes:

- The system requirements for Alibre Design
- The contents of the Alibre Design CDs
- Installing the Alibre Design software
- Upgrading your client to a new software version
- Installing the Alibre Design Help system locally

1.1 System Requirements

The following requirements must be met to install and run Alibre Design.

Recommended operating systems

- Windows® XP Professional or Home Edition
- Windows 2000 Professional SP2 or later

Other supported operating systems¹

- Windows Me
- Windows 98 SE or later

Internet connection

An internet connection is not required for design and data management functions of Alibre Design.

An internet connection IS required to hold Team Design sessions, securely share data through the Repository, and use the Alibre Assistant for support.

- 56.6 kb minimum
- Recommended: DSL, Cable Modem, T1 or faster

Software requirements

- Internet Explorer 4.01 SP2 + Active Desktop or later
(6.0 or later recommended)
- Microsoft Virtual Machine
- Microsoft DirectX 9.0c

Hardware requirements

- Intel Pentium or equivalent processor; 800 MHz or faster
- 512 MB RAM
- 1024 x 768 screen resolution (800 x 600 for laptops)
- Video card with DirectX 9.0c support (64 MB or higher)
- 16-bit or high color
- 150 MB available hard disk space
- Virtual memory: 800 MB

- CD-ROM drive (to install from CD)
- Mouse or pointing device

Additional Recommendations for assemblies of 500 or more parts

- Intel Pentium or equivalent processor; 2 GHz or faster
 - 1024 MB RAM
 - 1000 MB Virtual Memory
1. Windows Me & Windows 98 SE are not recommended for use in production work. We strongly recommend switching to Windows XP or Windows 2000, which are better suited to the demands of production 3D design.

1.2 On the CD

The following items are on the installation CDs.

The Alibre Design Software & Multimedia Demo CD, is shipped with all versions of Alibre Design.

- Alibre Design 8.2 application
- Alibre Design Help
- Alibre PhotoRender (specific license key required)
- ModelPress Viewer
- Alibre Design Training Videos (2 hours of training)
- Microsoft Virtual Machine software, build 3809
- Internet Explorer 5.5
- DirectX 9.0c

Alibre Design Professional CD is shipped with Alibre Design Professional, along with CD1.

- ALGOR DesignCheck
- Alibre Part Library
- MecSoft VisualMill
- ModelPress Publisher

Alibre Design Expert CD is shipped with Alibre Design Expert, along with CD1.

- ALGOR DesignCheck
- Alibre Part Library
- MecSoft VisualMill
- Alibre PhotoRender Industrial Pack
- SprutCAM for Alibre Design
- Machinist Toolbox
- ModelPress Publisher
- MSC.visualNASTRAN Motion (in the U.S.A & Canada)

1.3 Installing and Uninstalling

To Install Alibre Design from the CD:

- 1** Insert the CD-ROM into the CD drive. Within a few seconds, a window appears. If the window does not appear, open the Windows Explorer, and double-click your CD-ROM drive icon to open the window.
- 2** Click **Install Alibre Design** from the selection list.
- 3** Click **Install Alibre Design 8.2**.
- 4** Review the system requirements; then click **Install Now**. InstallShield initializes and launches the Alibre Design installation wizard.
- 5** Click **Next**. The License Agreement page appears.
- 6** Read the Product License Agreement and select **I accept the terms in the license agreement**.
- 7** Click **Next**.
- 8** Specify a name and organization if desired.
- 9** Choose whether this installation is for all users or only you.
- 10** Click **Next**.
- 11** By default, Alibre Design is installed in the **Program Files** directory on the **C:** drive. If desired, select a different destination folder.
- 12** Select the language you prefer.
- 13** Click **Install**.
- 14** When the installation is complete, click **Finish** to exit the wizard.

To Uninstall Alibre Design:

- 1** From the **Start** menu, select **Control Panel** or **Settings – Control Panel** depending on your operating system.

- 2 Double-click Add/Remove Programs.
- 3 From the Currently installed programs list, select **Alibre Design**.
- 4 Click **Remove**.
- 5 At the prompt, click **Yes** to confirm that Alibre Design should be removed.
- 6 The uninstall program removes program files, folders, and registry entries. Repository files are not removed.
- 7 When the files are removed, the uninstall program may indicate that the process is complete. Click **OK**.

1.4 Upgrading

Alibre may periodically release software service packs or minor software releases (such as version 8.x or 8.2 SPx) to implement new features or resolve software issues. These releases are distributed via the Alibre Design server automatically. When an update is available, you are prompted to upgrade upon launching Alibre Design, if you are online. The process includes a small download and a brief installation.

1.5 Installing Alibre Design Help

You may install a local copy of Alibre Design Help that will be available when working offline. If you choose to install Help locally, you will need to reinstall Help after each upgrade to ensure that you have the most recent Help version.

To install the version of Help included on the CD:

- 1 Right-click your **My Computer** icon and select **Explore**.
- 2 Double-click the CD-ROM drive icon. Double-click **start_here.exe**.
- 3 Click **Install Alibre Design** from the menu.
- 4 Click **Install Alibre Design Help**.
- 5 Click **Install Now**. InstallShield initializes and launches the Alibre Design Help installation wizard
- 6 Follow the steps in the InstallShield Wizard to install Help locally.

2 Getting Started With Alibre Design

Alibre Design is a powerful mechanical design software application. You can use Alibre Design to create complex 3D designs and 2D drawings. In addition to its powerful modeling capabilities, Alibre Design contains a unique collaboration engine that allows engineering teams to work together simultaneously over the Internet to create, visualize, review and modify their designs and drawings. Alibre Design also allows users to directly share and manage all types of files with other users, using the Internet as a work platform.

This chapter provides a high-level overview of the basic design capabilities in Alibre Design and highlights the unique team design tools built into the application.

This chapter describes:

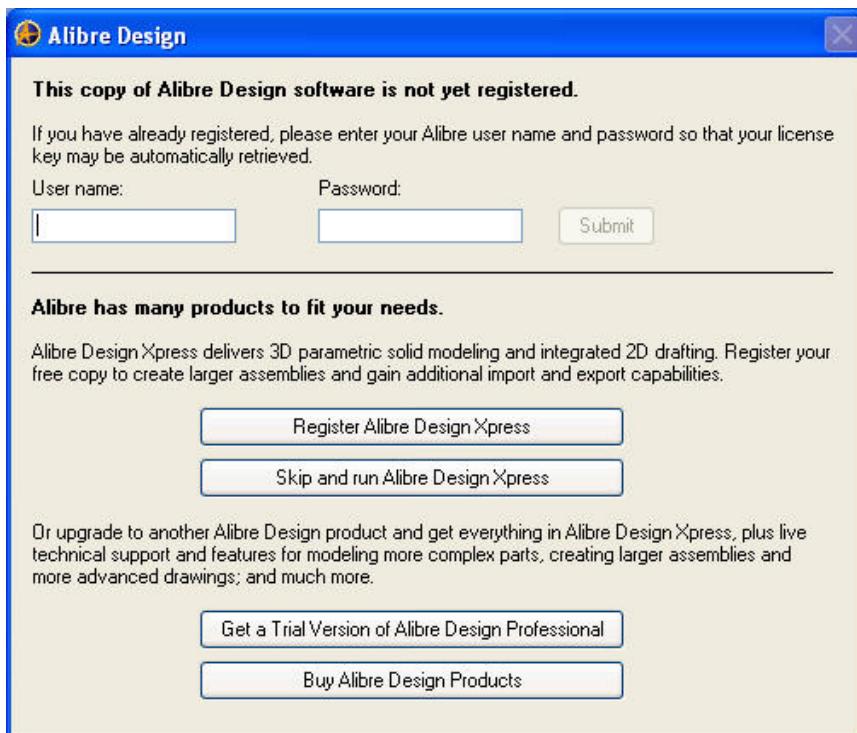
- The initial launch of Alibre Design
- The built-in tutorials
- An initial design session
- The major application windows: Home window, Repository, Message Center, Team Manager, and Design and Drawing Workspaces

2.1 Initial Launch of Alibre Design

To use the unique communication and collaboration tools in Alibre Design, you must be signed into the Alibre Design server. Users who have purchased Alibre Design Basic, Standard, Professional, and Expert who have current maintenance can sign into the Alibre Design server. Working while connected to the Alibre Design server is referred to as online mode. For these users, Alibre Design automatically starts in online mode - if a network connection exists - when launching the software for the first time. See Section 2.5 for information about changing the default startup mode.

To launch Alibre Design:

- 1 From the Start menu, select **All Programs > Alibre Design**. An initial startup screen is displayed.



If you have already registered with Alibre, Inc....

- 2 Enter the **user name** and **password** you obtained from Alibre, Inc..

Note: Shortly after purchasing Alibre Design, new customers receive an email from Alibre containing a user name and a temporary password. If you register and/or purchase online, your user name and password are displayed at that time.
- 3 Click **Submit**. If an internet connection is detected, your license key will be automatically retrieved from the Alibre server. Otherwise, you will see a message with a 5 character site key and instructions you need to follow to obtain a license key.
- 4 After several seconds, the startup screen will be replaced by the Alibre Design **Home Window**. Also, the **Set Default Workspace** dialog will appear.
- 5 If you are a new user, you may want the appropriate tutorial to display when you open a new workspace. If so, check **Display tutorials when opening new workspaces**.
- 6 Click **OK**. A new **Part workspace** is created and the corresponding introductory tutorial, **Modeling a Simple Part**, is automatically displayed (if you checked the option in step 5). The Home window remains open behind the workspace.

If you have not yet registered with Alibre, Inc....

- 2 Choose one of the available options:
 - **Register Alibre Design Xpress** - Register your free copy of Alibre Design Xpress to gain additional functions and the opportunity to participate in exclusive activities like the Xpress Design Contest

This button will direct you to a web form where you can register with Alibre, Inc. You will be provided with a user name and password to enter in the startup screen shown above.

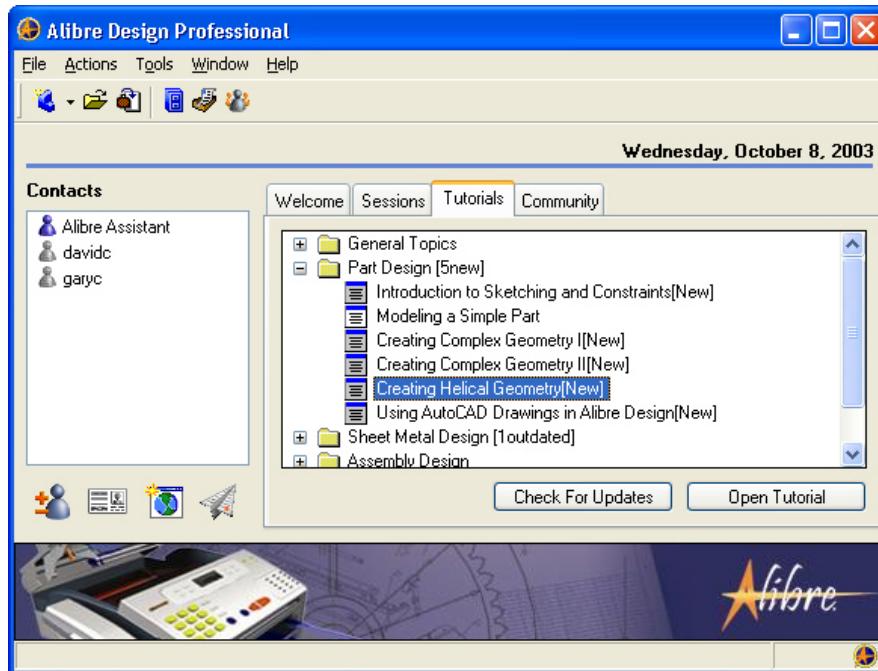
- **Skip and run Alibre Design Xpress** – Start using Alibre Design Xpress
- **Get a Trial Version of Alibre Design Professional** – Sign up for a 30 day fully-functional trial of Alibre Design Professional
- **Buy Alibre Design Products** – Purchase options for upgraded versions of Alibre Design

2.2 Integrated Tutorials

Alibre Design tutorials are fully integrated with the application. When the application is launched for the first time, a part modeling tutorial opens in conjunction with a part workspace.

Each workspace type (part, assembly, sheet metal, drawing) has a corresponding introductory tutorial. By default, whenever you create a new workspace, the corresponding introductory tutorial is also displayed. This behavior can be changed: see Section 2.4.6 for details.

In addition, all of the available tutorials are accessible from the **Tutorials** tab in the **Home Window**. See Section 2.4 for more information on the Home window.



Alibre will periodically release new tutorials and make these available in the **Tutorials** tab. When working online you can check for newly posted material by clicking the **Check for updates** button. When new tutorials are available, they can be downloaded to your computer and subsequently accessed while offline.

Note: To download new material when you see [New] or [Outdated] next to the title of a tutorial, right-click on the title and choose **Download**.

2.3 An Initial Design Session

The remainder of this chapter and all of Chapter 3 provide introductory overviews of the main application areas and the 3D design interface.

Before proceeding further in this User Guide, we strongly recommend that you work through two of the basic Part Design tutorials, *Introduction to Sketching and Constraints, and Modeling a Simple Part*.

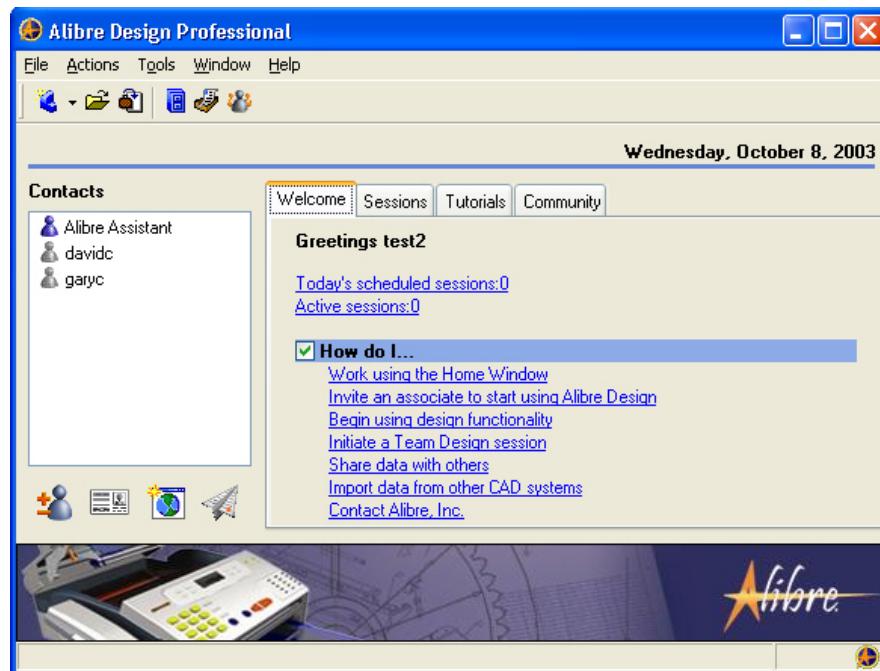
Working through these tutorials will give you a good understanding of the basic modeling terms and concepts used in Alibre Design. You will also become familiar with 3D viewing techniques in Alibre Design.

By completing these tutorials now you will be better prepared for the material in the subsequent sections.

2.4 The Home Window

Alibre Design is comprised of five main components: the **Home window**, the **Repository**, the **Message Center**, the **Team Manager**, and design and drawing areas referred to as **workspaces**. Each component opens in a separate window that you can independently resize, position or tile.

When you launch Alibre Design, the Home window appears first and is always open while Alibre Design is running. The Home window serves as the starting point for all other areas of Alibre Design. Whenever you have another Alibre Design window open, you can quickly access the Home window by clicking the Alibre Design icon in the lower right corner of any other Alibre Design window.



2.4.1 The Contacts List and Alibre Assistant

The **Contacts List** is a customizable listing of your personal contacts of Alibre Design users, support personnel and consultants.

The Alibre Assistant:

After signing into the Alibre Design server, you may see a contact called **Alibre Assistant** in the **Contacts** list. The Alibre Assistant corresponds to a support engineer at Alibre headquarters who can offer real-time technical help through Alibre Design. An Alibre support engineer is online and available for assistance whenever **Alibre Assistant** is visible in the contacts list.

To add an associate to your contacts list:

- 1 While working online, select **Add/Remove Contacts** from the **Actions** main menu. Or, click the **Add/Remove Contacts** icon on the Home window.
- 2 Select the desired name from the **Listed Users** area and click **Add**. Alternatively, if you know the user name of your associate, type it into the **Unlisted User** area and click **Add**.

- 3 Click **OK**. The added users will appear in your contact lists.

See also **Invite an associate to start using Alibre Design** located in the Welcome tab of the Home window.

2.4.2 The Welcome Tab

The **Welcome** tab hosts seven helpful links under the **How Do I** heading. These links lead to detailed information on important Alibre Design features. Access specific information about the Home window by clicking the **Work using the Home window** and **Invite an Associate to start using Alibre Design** links.

Note: The How Do I links may be hidden by removing the check mark next to the "How Do I" heading.

2.4.3 The Sessions Tab

The **Sessions** tab lists all active and scheduled team design sessions that you are involved in. You can view details about each session as well as join active sessions or schedule new sessions from this tab.

2.4.4 The Tutorials Tab

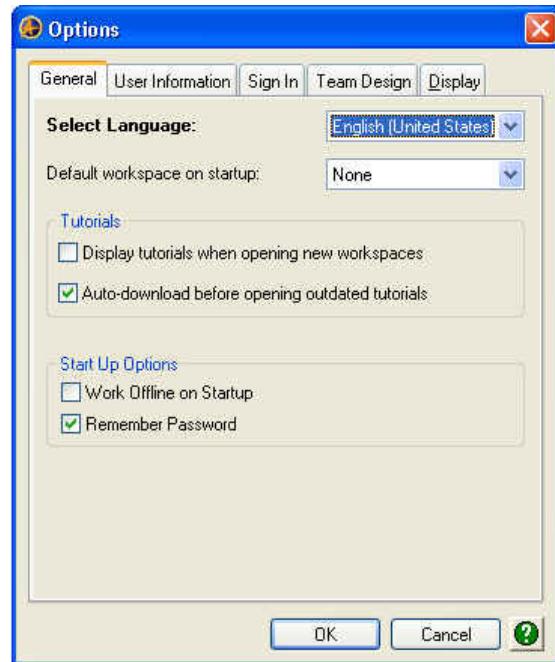
The **Tutorials** tab provides access to integrated tutorials within the software. Introductory tutorials are available for each type of Alibre Design workspace. You can download and open the tutorials directly from this tab. Alibre will frequently release new tutorials that will be made available via the Tutorials tab. When working online, you will be able to download any new tutorials by clicking the **Check for Updates** button in the **Tutorials** tab.

2.4.5 The Community Tab

The **Community** tab provides access to a set of resources to help you get the most out of Alibre Design, including authorized consultants, solution partners, international resellers, support, and training.

2.4.6 System Options

A number of system options can be set and modified from the Home window. From the **Tools** menu select **Options**. The **Options** dialog box appears.



General tab:

The **General** tab allows you to specify the language you are using; which type of workspace opens by default on startup (**Note:** You can select **None** so that no workspaces automatically open upon launching the software); whether or not a tutorial opens by default with the applicable workspace type; and whether or not updated tutorials will be automatically downloaded.

Also, you can change the startup options for working online and you can choose to have Alibre Design remember your password.

Note: The second and third options are also available in the **Set Default Workspace** dialog, which is displayed during initial startup of Alibre Design.

User Information tab:

Under the **User Information** tab, you may enter personal information such as name, company, email address, phone number, etc. Check the **Public** box next to any item to make that information available to other users. If you want to make your user name visible

to the entire Alibre Design community, select the **Display my user name in the public list** option.

Sign In tab:

The **Sign In** tab allows you to change your Alibre Design password anytime, and you can add an additional username and password for accessing the Alibre Design server.

Team Design tab:

The **Team Design** tab provides access to options associated with Team Design sessions.

Display tab:

Under the **Display** tab you can check your graphics settings by selecting the **Settings** button.

2.5 More on Working Online and Offline

As mentioned in Section 2.1, Alibre Design operates in online mode by default. However, Alibre Design can also be used offline in a standalone mode. All modeling and design related tasks are available in offline mode. However, all communication and collaboration tools are disabled. You can change the default startup mode to offline if desired.

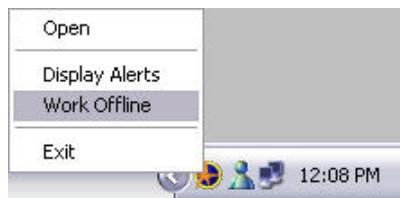
To change the default launch mode:

- 1 From the **Tools** menu in the **Home Window**, select **Options**. The **Options** dialog box will appear.
- 2 Click the **General** tab.
- 3 In the **Start Up Options** area, check the **Work Offline on Startup** checkbox to set the default startup mode to offline.
- 4 Click **OK**. The next time you launch Alibre Design, the software will start in the offline mode.

To switch between working online and offline:

When Alibre Design is running in online mode, the Alibre Design icon in the Windows system tray has an orange appearance. When offline, the icon has a gray appearance. To switch

between offline and online modes, right-click the Alibre Design icon in the Windows system tray and select **Work Online** or **Work Offline**.



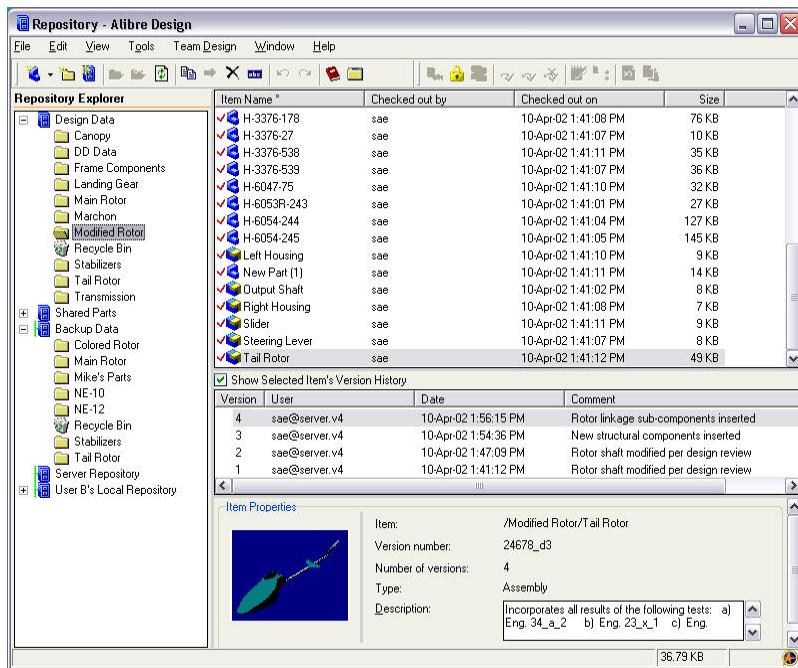
Note: When Alibre Design is in online mode, users may choose to be alerted when certain events occur. The alerts can be turned on and off by clicking **Display Alerts** in the right-click menu. Alerts occur when an associate signs into Alibre Design, a message or team session invitation is received, and an associate has shared a repository.



2.6 The Repository

Some versions of Alibre Design include a personal data vaulting and versioning system known as the **Repository**. Chapter 14 provides a detailed guide to the Repository.

To open the Repository from the Home window, select **Repository** from the **Window** menu. Or, click the Repository icon on the toolbar.



One local repository is available upon installation. The initial repository name matches your user name; however, the repository name can be changed, just right-click the repository and select **Rename**. To create additional local repositories, from the **File** menu, select **New Repository**. All repositories are displayed in the **Repository Explorer**. Data can be moved and/or copied between repositories.

A version history is automatically maintained for all files. Each time a file is saved, a new version is created in the Repository. To view version history, select a file; then check **Show Selected Item's Version History**. The version history shows which user created the version, the day and time the version was created, and any comments.

Note: Versioning can be disabled. From the **Tools** menu, select **Options** and deselect the **Always make new versions on save** option.

The **Item Properties** area displays summary information for each design as well as a preview image. The image displays the model in its orientation when it was last saved.

An important and versatile feature of the Repository is the ability to share data directly with other users anywhere in the world, easily and securely. The modern peer-to-peer architecture of Alibre Design makes this possible without having to store data on a central server. However, for users who desire a centralized location to store data, server-based repositories are also available from Alibre (for an additional charge).

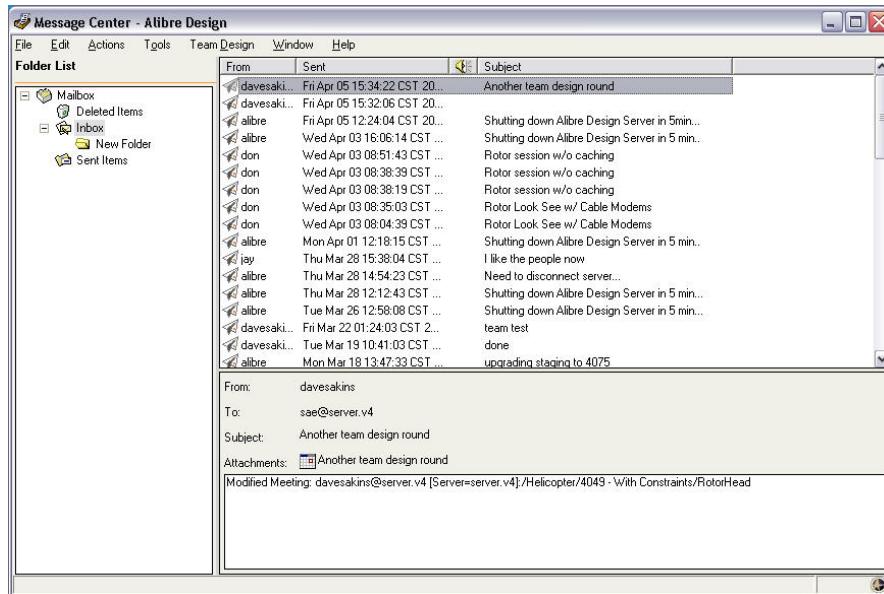
For detailed information about sharing data, see **Chapter 14**, as well as the **Share data with others** link on the Home window.

2.7 Message Center

In addition to this overview, a detailed guide to using the Message Center is provided in **Chapter 15**.

Messages can be sent to other Alibre Design users while working online. Messages are sent, viewed and managed in the Message Center. To access the Message Center, from the **Window** menu, select **Message Center**. Or, click its icon in the Home window toolbar.

The Message Center is organized similar to most email applications.



Incoming messages are stored in the **Inbox**, and outbound messages are logged in **Sent Items**. To create a new message, from the **File** menu, select **New Message**. Create new folders to organize messages, just right-click a folder and select **New Folder**. Most Message Center features are only available when working online.

Note: Messages can also be sent from the Home window. From the Actions menu, select Send Message. Or, right-click a contact in the Contact list and select Send Message.

To set options for receiving messages, from the **Tools** menu, select **Options**.



You may opt to be alerted of new messages by pop-up boxes and/or a system sound.

Even if the notification and alert options are turned off, a message icon in the lower right corner of the Home window indicates that a new message has arrived.



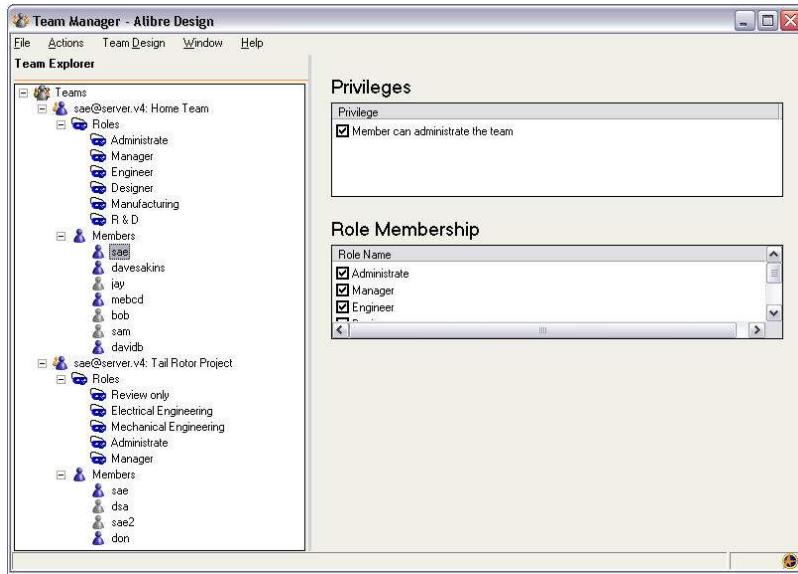
You may also forward messages to an email account. Three options are available: never forward messages, forward messages only when offline, and always forward messages.

2.8 Team Manager

In addition to this overview, a detailed guide to using the Team Manager is provided in [Chapter 16](#).

Alibre Design enables you to efficiently manage people and data by defining roles and teams for your contacts. Defined teams of users are ideal for projects involving multiple people.

Team administration is handled in the Team Manager window, which is only accessible when working online. To open the Team Manager from the Home window, from the **Window** menu, select **Team Manager**. Or, click its icon on the Home window toolbar.



By default, a Home Team is listed in the **Team Explorer** on the left side of the window. To create a new team, from the **Action** menu, select **Add Team**. To add members to a team, click a team name in the Team Explorer to select it; then, from the **Actions** menu, select **Add Team Member**. Team roles may also be added and assigned to team members. From the **Actions** menu, select **Add Team Role**.

Roles are typically used to control data access. For example, some team members may need permission to modify data while others only need permission to view data. An "Engineer" role could be set with more advanced permissions and a "Reviewer" role could be set with more limited access to the data. Permissions are set in the Repository, applied to individual files and folders. After you create a team and its associated roles, data can easily be shared.

with the entire team through the repository. See Chapter 14, Sections 16, 17 and 18 for more details on setting access permissions on repository data.

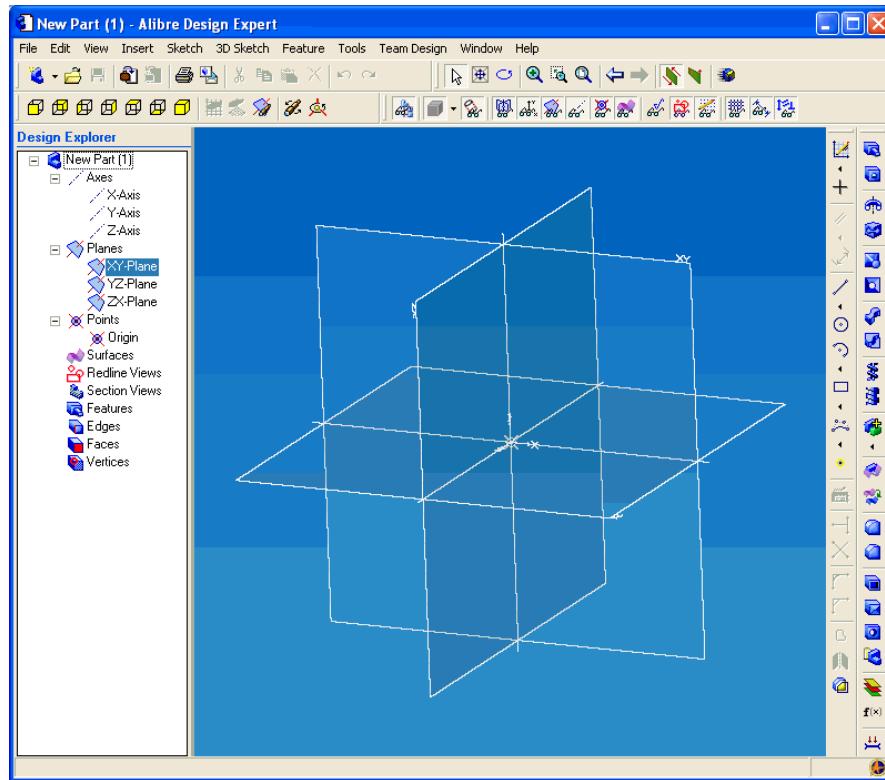
Teams can also be invited to a Team Design session, eliminating the need to send an invitation to multiple people. If a user is removed from a team, he or she is also removed from the team session. See Chapter 17 for more details on Team Design sessions.

2.9 Workspaces

In addition to this overview, more information about design workspaces is provided in **Chapters 3–11**.

If you have not already done so, we strongly recommend that you spend twenty minutes working through the introductory part design tutorial, *Modeling a Simple Part* before proceeding further in this User Guide. See Sections 2.1, 2.2 and 2.3 to get started.

All design related tasks, as well as all Team Design sessions, are carried out in windows referred to as workspaces. New workspaces can be opened from the Home window, the Repository, or other open workspaces. From the **File** menu, select **New** to open a new workspace. Five workspace types are available: part, sheet metal part (available only in Alibre Design Professional and Expert), assembly, drawing, and bill of materials.

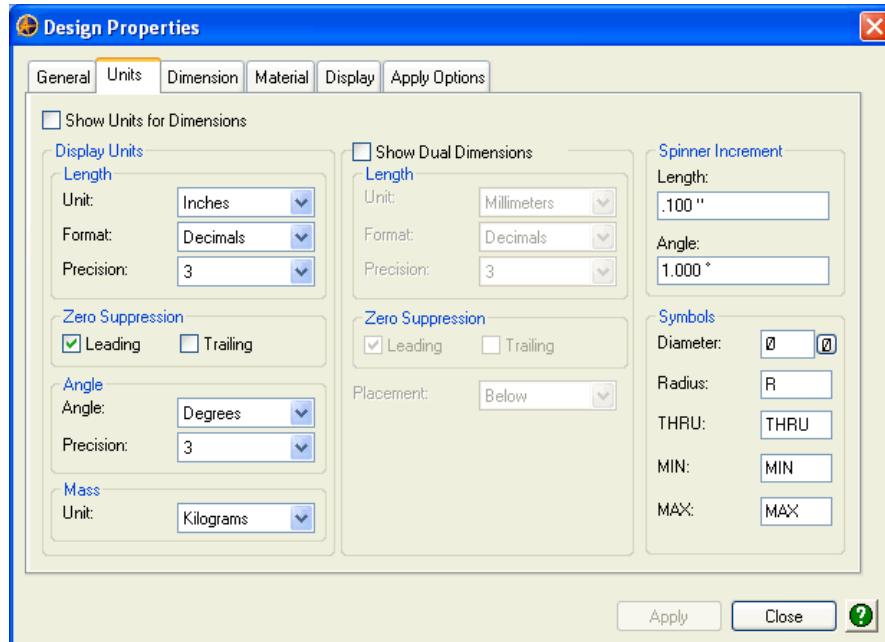


The **Design Explorer**, displayed on the left side of a part or assembly workspace, lists all reference geometry, such as planes and axes, and feature geometry associated with the design. Toolbars are located on top and to the right of the main workspace area.

For detailed information about creating new designs, please see **Chapters 3–11** and the **Begin using design functionality** link on the Home window.

The workspace can be customized a number of ways. From the **Tools** menu, select **Options**. Use the **Color Scheme** tab to change the background color of the main workspace area. Default colors can be selected and custom colors can be added. Additionally, the toolbars can be positioned and toggled on and off — from the **View** menu, select **Toolbars**.

To set workspace properties, from the **File** menu, select **Properties**. The Design Properties dialog box includes six tabs.



The **General Tab** is used for notes and comments related to the design

The **Units** tab is employed to set all design units and associated display format and precision. Several other unit display settings are also available.

The **Dimension** tab is used to control dimension properties such as size, dimension orientation and spacing, and arrowhead style.

The **Material** tab is used to set the density of the part material. This information is used when calculating physical properties of the part.

The **Display** tab is used to control the graphics display.

In the **Display Acceleration** field, you can set the options you wish to use when in Display Acceleration mode in parts and assemblies. Refer to Section 9.16 for more information on Display Acceleration.

The **Curve Smoothness** setting is applied on this tab to set the precision of the graphic display. Selecting the **Automatic** option provides the cleanest display of the model, with all edges appearing smooth and precise. Selecting the **Manual** option yields an approximate faceted display of the model. The **Manual** option also requires a **Minimal Circular Facets** number, which represents how many line segments make up a circle. The higher the facet setting, the more detailed the

display appears. The **Automatic** and **Manual** options are a tradeoff between performance and graphic display. Applying the Automatic option improves the graphic display but performance decreases slightly. Applying the Manual option improves performance but the graphic display degrades somewhat.

The **Apply Options** tab is used to apply Design Properties settings on a system-wide basis, or only to the current workspace.

3 Introduction to the Design Interface

Alibre Design is a parametric solid modeling system. Parametric solid modeling involves the application of dimensions and other parameters to define a 2D profile, and ultimately the 3D shape, of a part. These dimensions and parameters can be changed at anytime to easily modify designs and alter the shape of a model.

Parts are comprised of one or more features. A feature represents an individual shape and can either add material or remove material in a part. Examples of features include holes, fillets, cuts, and revolutions.

Alibre Design data consists of parts, assemblies, drawings and bills of materials (BOMs). Full associativity between parts, assemblies, drawings and BOMs automatically applies changes made in one document across the entire design.

You typically start a new part by first creating a sketch, which is a 2D profile of a 3D shape. The sketch will subsequently be used to create a base feature, always the first feature you create in a part. You can add as many additional features as necessary to fully define a part.

You can easily change the shape of a model by adding, deleting, editing, or reordering features.

You can generate 2D mechanical drawings and bills of materials at any time in the design process based on information in parts and assemblies.

You can also import parts, assemblies, and drawings from other CAD systems into Alibre Design. You can modify imported parts and assemblies or you can add new features to them.

This chapter describes:

- Workspaces and associated options
- The work area and manipulating the view
- The Design Explorer
- Selection methods
- Toolbars
- Getting Help

3.1 Workspaces

All design work in Alibre Design is done in windows called workspaces. You can open a part, sheet metal part (in Alibre Design Professional and Expert), assembly, drawing, or bill of materials workspace. Each workspace is displayed in a separate window; however, a drawing workspace can contain multiple drawing sheets. You can have as many workspaces open as needed.

3.1.1 Opening a New Workspace

To open a new part, sheet metal part, assembly, drawing, or bill of materials workspace:

- 1 On the Home window, from the **File** menu, select **New**.
- 2 Select **Part, Sheet Metal Part, Assembly, Drawing, or Bill of Materials**.

Or

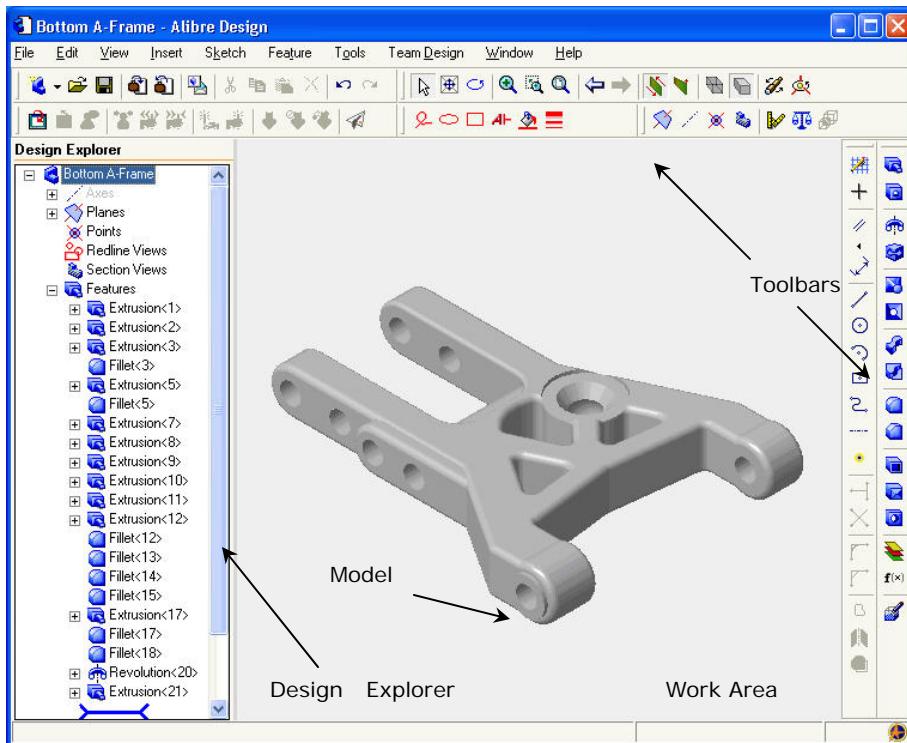
Click the new **Part, Sheet Metal Part, Assembly, Drawing**, or the **Bill of Materials** workspace icon on the Home window main toolbar. Note that sheet metal and drawings are not available in all versions of Alibre Design.



3.1.2 Workspace Terms

Alibre Design workspaces are divided into two distinct areas.

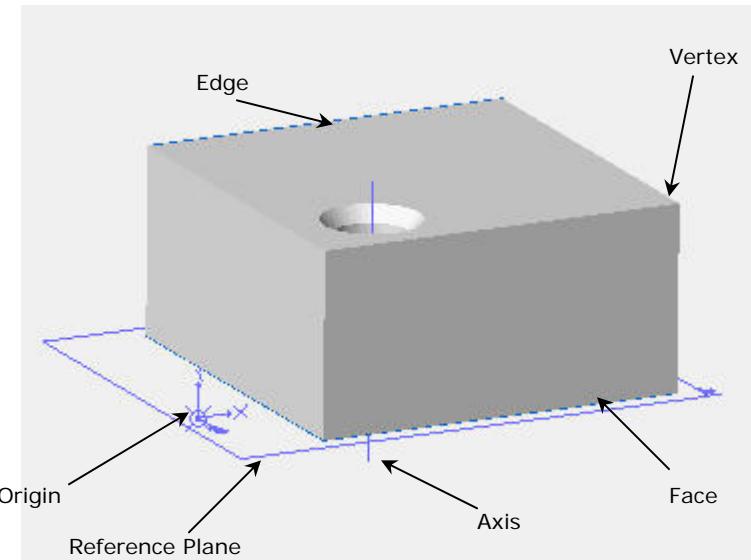
- The **Design Explorer** is located on the left side of the workspace and lists all information related to a design including reference geometry and feature geometry.
- The **work area** is the graphics canvas in which you create all parts, assemblies and drawings. Toolbars may be located above and to the right of the work area.



Part Workspace with default toolbars

3.1.3 Model Terms

The following terms are consistently used throughout the documentation and refer to geometric elements in a model (faces, edges, and vertices) as well as construction geometry (reference planes, axes, and the origin).



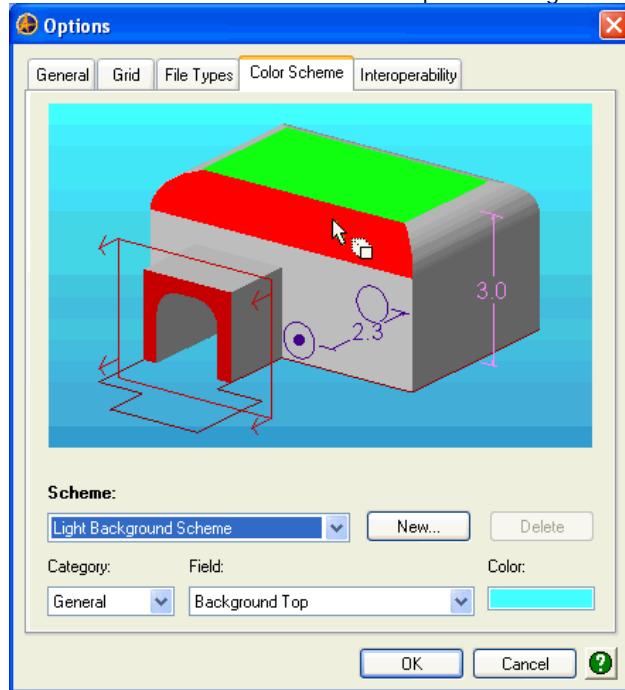
3.1.4 Work Area Color Scheme

You can change the color of the work area background and all other display components in a part or assembly workspace.

To change the color scheme:

- 1 In a part or assembly workspace, from the **Tools** menu, select **Options**.

- 2 Select the **Color Scheme** tab in the Options dialog box.



- 3 Choose one of the four predetermined color schemes, e.g. **Dark Background Scheme**, from the **Scheme** dropdown list, then click **OK** to apply the setting.

Or

Customize your display by creating a new color scheme.

To create a new color scheme:

- 1 Click **New**. The **New Color Scheme** dialog box appears.
- 2 Enter a name for the new custom color scheme, then click **OK**.

Note: You cannot modify the default color schemes. If you attempt to change a color without creating a new color scheme, you will automatically be prompted to create a new scheme.

- 3 Select a category to modify from the **Category** drop down list. Next, select a **Field** in that Category. The diagram will show the current color scheme.
 - 4 Click in the **Color** field to set a new color. Select the desired color, then select **OK** to apply the setting.
- Note: A gradient background can be achieved by setting different colors for the 'General – Background Top' and 'General – Background Bottom' fields. A solid background can be achieved by setting these to be the same color.
- 5 Click **OK** to finish creating the new color scheme.

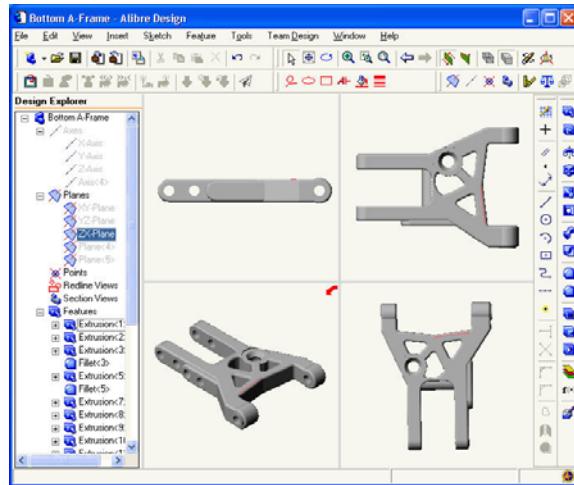
3.1.5 Multiple Views

You can split the work area into as many as four different views. You can zoom, rotate, and set the view mode in each view independently.

To split the work area into multiple views:

- 1 In a workspace, from the **Window** menu, select **Split View**.
- 2 Select **Horizontal**, **Vertical**, or **Both**. The Horizontal and Vertical options split the work area into two views. The Both option splits the work area into four views.

To select a view and make it active, click anywhere in the view border. A red arrow appears in the upper right corner of the active view.



3.1.6 Named Views

In part and assembly workspaces, you can use named views to control view display and manipulation. You can quickly change the display to a default view and add custom views using the **Orientations** command under the **View** menu or by selecting the **View**

Orientations tool on the View toolbar.



To apply a named view to the work area, double-click a named view, or select a named view and then click **Set**.

You can also add a custom view orientation by clicking **Add** and subsequently entering a view name.

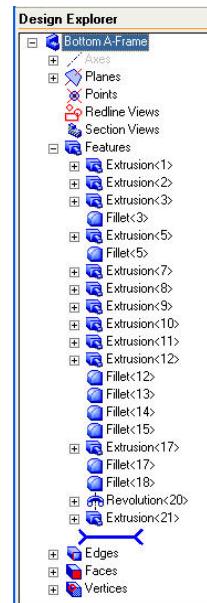
Default views are listed in blue, and custom views are listed in black.

3.1.7 Design Explorer

As previously described, each workspace consists of the work area and the Design Explorer. The Design Explorer's primary purpose is to track and list the structure of a part, assembly, or drawing. However, the Design Explorer can be used to accomplish numerous tasks.

Use the Design Explorer to:

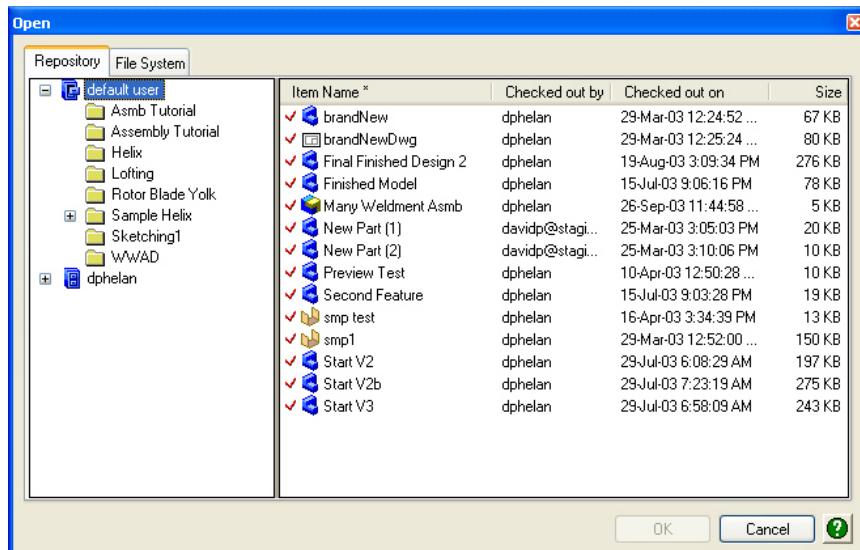
- Select items in the design by name.
- Suppress or hide selected features and parts.
- Hide planes and axes.
- Temporarily roll the model or assembly back to an earlier state: double-click a feature or part or use the rollback bar.
- Identify and change the order in which features are regenerated.
- Rename features: right-click a feature and select **Rename**.
- Delete features and parts.
- Toggle the display of section views on and off.
- Edit sketches: right-click a sketch and select **Edit**.
- Track and control the display of redline markups.
- Check the status of a feature or part to resolve errors.

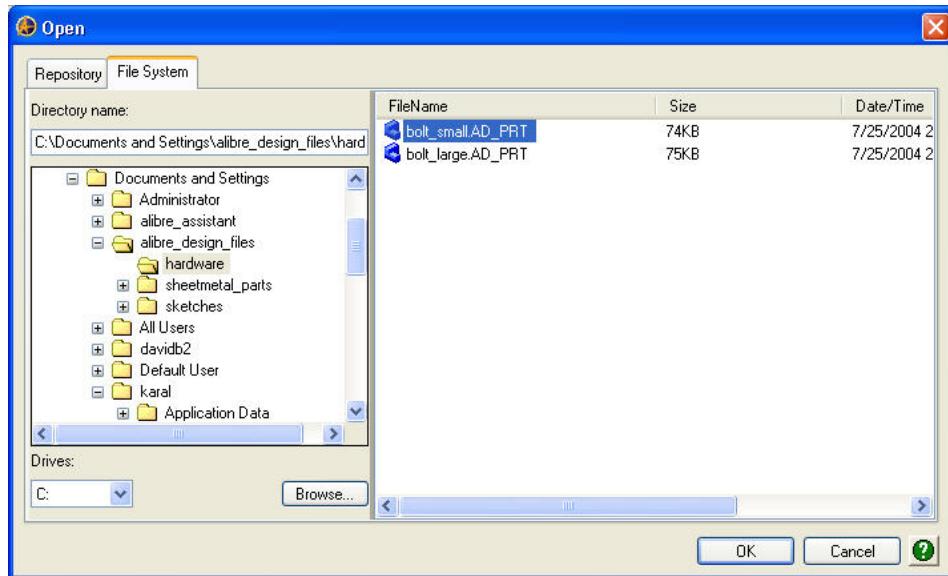


3.1.8 Document Browser

Alibre Design provides two different mechanisms to save and retain your design documents. All versions of Alibre Design support storing design documents as files in the Windows file system. In addition, some versions of Alibre Design include the **Repository**, a personal data vaulting and versioning system.

Whenever you save or open a design document in Alibre, you are presented with the **Document Browser**, shown below. Through the two tabs, **Repository** and **File System**, the Document Browser gives you access to both the Repository and the Windows File System.





On the File System tab, you can browse through any of the drives on your computer, or you can click the **Browse** button to browse to other areas such as Desktop, My Documents, and My Network Places.

3.2 Selection Methods

The majority of design tasks require that an item be selected. For example, a reference plane or planar face must be selected before you can begin sketching.

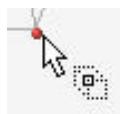
- **Normal Selecting** – To select faces, edges, vertices, etc., first click the Selection tool or choose **Select** from the **View** menu. Move the cursor over an item, and then click once to select the item. An item will change to red as you move the cursor over it and will change to yellow after you select it.

Note: Selection highlight colors will vary depending on the work area background color scheme used. This documentation describes properties associated with the **Dark Background Scheme**.

- **Selecting Multiple Items** – To select multiple items in the work area, hold the **Shift** key while you select the items. In cases where a multiple selection is required, you do not need to hold down the **Shift** key. In the Design Explorer,

holding the **Shift** key will select everything in between the items you selected. Hold the **Control** Key while selecting to choose only the items you click.

- **Selecting by Dragging** – In a sketch or drawing, select multiple items by dragging a selection rectangle around a group of items.
- **Using a Selection Filter** – Apply a filter to make selecting a specific item type easier. Filter on **Solid > Features, Faces, Edges, and/or Vertices**, in addition to **Surface > Surfaces, Faces, Edges, and/or Vertices**. To apply a selection filter, from the **Tools** menu, expand **Selection Filters** and choose the appropriate filter. You can also activate the **Selection Filters** toolbar and control filters easily with icons.
- **Face, Edge, and Vertex Cursors** – As you move the cursor over faces, edges, and vertices, the cursor will change to provide a visual indication of the item type.



Vertex Selection



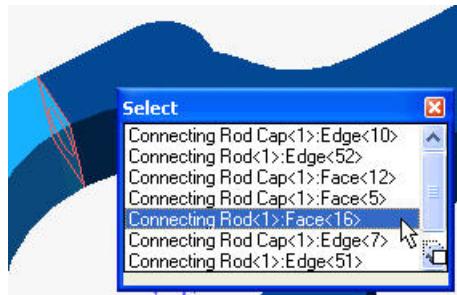
Edge Selection



Face Selection

- **Modifying a Selection** – Many dialog boxes require a selection or multiple selections in order to complete the command. The selected item name will often populate an area in a dialog box. To change the selection, either make another selection, which will override the previous selection, or right-click the item name in the dialog box, and choose **Clear Selections** or **Remove Selected Item(s)**.
- **Advanced Selector** – On occasion, other items may obscure the item you want to select. Use the **Advanced Selector** to accurately select an item. To use the Advanced Selector, move the cursor over the item, right-click it, and choose **Advanced Selector** from the pop-up menu. The **Select** dialog box appears containing all the items in the vicinity of the click location, including items hidden behind other faces. Select the item you want from the list.

Note: If available, you can also **Ctrl-click** with the middle mouse button to access the Advanced Selector.



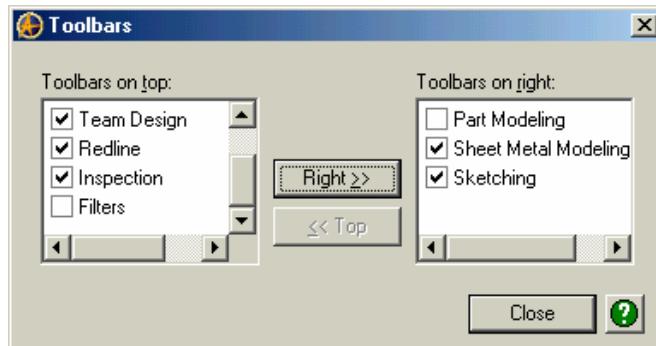
- **Selecting from the Design Explorer** – Select any item in the **Design Explorer** by clicking the item name. The associated item will change color in the work area. Select non-consecutive items in the Design Explorer by holding down the **Ctrl** key as you click. Select consecutive items in the Design Explorer by holding down the **Shift** key as you click.

3.3 Toolbars

You can control toolbar visibility as well as toolbar position in a workspace.

To turn toolbars on and off:

- 1 In any open workspace, from the **View** menu, select **Toolbars**. The Toolbars dialog box appears.
- 2 To hide a toolbar that is currently displayed, click the checked box next to the toolbar name.
- 3 To display a toolbar, click the empty check box next to the toolbar name.
- 4 Click **Close** to apply changes and exit the dialog box.



To move a toolbar between the top and right of the workspace:

- 1 In any open workspace, from the **View** menu, select **Toolbars**. The Toolbars dialog box appears.
- 2 To move a toolbar from the top of the workspace to the right of the workspace, select the toolbar name and then click **Right**.
- 3 To move a toolbar from the right of the workspace to the top of the workspace, click the toolbar name and then click **Top**.
- 4 Click **Close** to apply changes and exit the dialog box.

To drag a toolbar to a different position in an area:

- 1 Move the cursor near the vertical bar on the left side of the toolbar.
- 2 Click and hold the left mouse button and drag the toolbar to a new position within its respective area.

3.4 View Manipulation

You can change the perspective and orientation of the work area view by using various tools available from the View toolbar, the Orient View toolbar, the Visibility toolbar, and the View menu.

View Toolbar:



Orient View Toolbar:**Visibility Toolbar:**

- The **Pan** tool dynamically moves the current view around the work area. Click the icon and then click and drag the cursor around the work area.

Note: You can also pan by holding the **Shift** key and the left and right mouse buttons down simultaneously while moving the cursor around the work area.

- The **Rotate** tool dynamically rotates a part or assembly around a given point. Click the icon and then click and drag the cursor around the work area.

Note: You can also rotate by holding the left and right mouse buttons down simultaneously while moving the cursor around the work area.

- The **Zoom Mode** tool dynamically changes the scale of the work area view. Click the icon, hold the left mouse button down, and move the cursor up to zoom in, or down to zoom out.

Note: If available, you can use the mouse wheel to dynamically zoom in and out.

- The **Zoom to Window** tool changes the scale of the view so that a specified region fills the work area. Click the icon, click and drag a rectangle around an area with the cursor. Release the mouse button when the rectangle borders the correct area.

- The **Zoom to Fit** tool restores the view so that the entire design is displayed in the work area.

- The **Previous View** tool reorients the work area to views that preceded the current view. The **Next View** tool becomes available after the Previous View tool has been used.

- The **Orthographic** tool changes the display of the work area so that parallel edges, faces, etc. appear as infinitely parallel.

- The **Perspective**  tool changes the display so that all parallel edges, faces, etc. appear to converge into one point.
- The **Display Acceleration**  tool optimizes your display by simplifying the designs according to user-defined preferences. Refer to Section 9.16 for more information.
- The **Orient** tools  store default views of front, back, left, right, top, bottom, and isometric.
- The **Orient to Sketch Plane** tool  changes the display so the sketch plane is displayed. This is only available in sketch mode. This is a toggle between the front and back view of the sketch plane.
- The **Isometric to Sketch Plane** tool  changes the display so that it is an isometric view of the sketch plane.
- The **Orient to Plane** tool  allows you to reorient the view to a plane of your choosing.
- The **View Orientations** tool  stores default views (e.g. front, back, left, etc.) as well as custom views.
- The **Rotation Points** tool  stores default rotation points (the origin, model's center of mass and center of volume) as well as custom rotation points.
- The **Model Shading** tools change the display to wireframe (so that only the edges of the model are shown, or shaded (so that the faces of the model are shaded).



- The **Toggle** tools allow you



to

toggle on and off the following options: Design Explorer, Silhouette Edges, All Reference Geometry, Coordinate System, Planes, Axes, Points, Surfaces, Annotations, Redlines, Sketches, Grid, Sketch Dimensions, and Constraint Symbols.

3.5 Getting Help

Help can be accessed numerous ways while you are using Alibre Design.

- While online, you can access the Help system from any window with the **Help** menu or the **F1** key.
- You can also install the Help system locally so that Help is available when you are working offline. You may install Help from the CD or the Alibre support web site, www.alibre.com/support.
- Seven help modules related to basic functionality are accessible on the Home window in the Welcome tab. Click one of the seven links in the **How Do I** section to access these help modules.
- To access help related to a specific dialog box, click the **Help** button in the dialog box.
- When you mouse over an icon on a toolbar, a **Tooltip** will popup that identifies the function of the tool.
- The status bar in the lower left corner of a workspace displays hints related to completing a command and provides a brief description of a tool or function.
- While online, you can contact the **Alibre Assistant** from the **Contacts** list in the Home window. You can send messages, emails, start a chat conversation, and work in a Team Design session in real-time to ask technical questions and resolve issues. The Alibre Assistant is available from 8:00 am to 6:00 pm Central Time.
- Design tutorials are available by default in the Alibre Design Home Window on the Tutorials tab. Periodically, Alibre will release new tutorials and distribute them for download via the Tutorials tab. You can check for new materials in the Tutorials tab when you are signed into the Alibre Design server.

4 Sketching

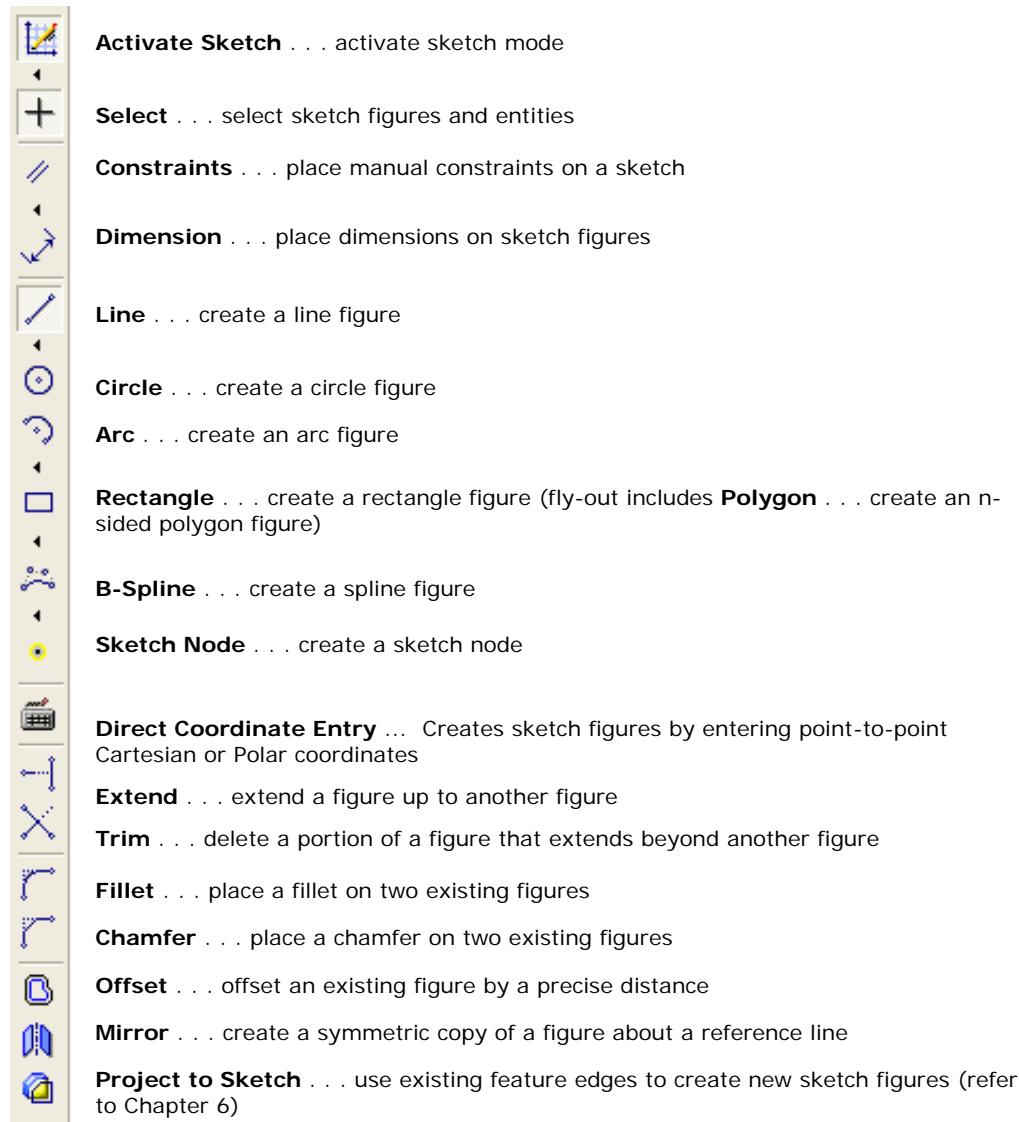
Sketching is fundamentally the most critical aspect of parametric solid modeling. The majority of features in Alibre Design begin with a sketch. A sketch is made up of one or more figures and provides the basic profile for a feature. Mastering sketching techniques is important and eventually leads to considerable time savings during modeling work.

This chapter describes:

- The Sketching interface
- Entering and Exiting Sketch Mode
- Using the Sketching toolbar, Sketch menu and sketching tools
- Creating sketch figures
- Dimensioning sketch figures
- Using the Equation Editor
- Adding and deleting sketch constraints
- Editing and deleting sketches

4.1 The Sketching Interface

The Sketching toolbar is shown by default on the right side of the workspace. Commonly used sketch tools are accessible on the Sketching toolbar.



Note: The **Extend**, **Trim**, **Fillet**, **Chamfer**, **Offset**, **Mirror**, and tools on the Sketching toolbar are only active when applicable sketch figures are in the sketch. For example, the **Extend** tool will not become active until at least two figures have been sketched.

The tools that are accessible on the Sketching toolbar are also accessible from the **Sketch** menu. The Sketch menu also contains tools that do not have a corresponding toolbar icon.

Ordinate Dimension . . . Creates ordinate dimensions in 2D drawings.

Auto Dimension . . . Automatically place dimension to place sketch

Figures > Ellipse . . . Creates an elliptical figure in sketch mode.

Figures > Elliptical Arc . . . Creates an elliptical arc in sketch mode.

Reference Figures . . . Create reference figures of various shapes.

Move . . . Move sketch figures from one location to another

Rotate . . . Rotate sketch figures about a center axis

Repeat > Linear . . . Creates a linear pattern of a sketch figure.

Repeat > Circular . . . Creates a radial pattern of a sketch figure.

Analyze . . . Determines if open ends, overlaps, or self-intersections exist in a sketch.

Insert > Axis . . . Inserts a 3D axis; sketch figures can be used as references.

Insert > Point . . . Inserts a 3D point; sketch figures can be used as references.

Create Custom Symbol . . . Creates a custom symbol using sketch figures.

Text > Field . . . Inserts a text field into sketches.

Text > Label . . . Inserts a text label into sketches.

4.2 Entering and Exiting Sketch Mode

4.2.1 Entering Sketch Mode

You must enter sketch mode before you can begin sketching.

To enter sketch mode:

- Select the **Activate 2D Sketch**  tool from the Sketching toolbar.
- Or
- From the **Sketch** menu, select **Activate 2D Sketch**.
- Or
- Right-click in the work area and select **Activate 2D Sketch** from the pop-up menu.
- Or
- Press **Ctrl + K** on the keyboard.

The Activate Sketch tool on the Sketching toolbar will always appear in the active state while in sketch mode. 

4.2.2 Exiting Sketch Mode

The same methods used to enter sketch mode can also be used to exit sketch mode.

To exit sketch mode:

- Select the **Select**  tool from the View toolbar.
- Or
- Create a feature from the sketched profile. For example, select a feature tool such as the **Extrude Boss**  tool from the Part Modeling toolbar.
- Or
- Select the **Regenerate**  tool from the Part Modeling toolbar, or from the **Feature** menu, select **Regenerate All** or press the **F5** key on the keyboard.

4.3 Sketch Figures

4.3.1 Line

To sketch a line:

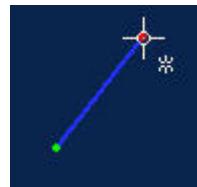
- 1 Select the **Line**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Line**; or right-click and select **Line** from the pop-up menu.
Position the cursor at the location you want to start the line.
- 2 Click in the Work Area to start the line and drag the cursor to sketch the line.
- 3 Click again to complete the line segment. You can continue to sketch additional line segments by clicking. Double-click or press **ESC** on the keyboard to end the line.



Note: During sketching, hints that provide step-by-step instructions are displayed in the lower left corner of the workspace.

To resize a line:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over a node at the end of the line.
- 3 Click, hold the mouse button, and drag the node to resize the line.
- 4 Release the mouse button.

**To move a line:**

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the line.
- 3 Click, hold the mouse button, and drag the line to a new location.
- 4 Release the mouse button.

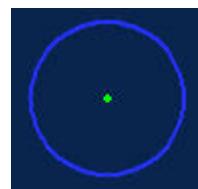
**To change the angle of a diagonal line:**

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over a node at the end of the line.
- 3 Click, hold the mouse button, and drag the node to change the angle.
- 4 Release the mouse button.

4.3.2 Circle

To sketch a circle:

- 1 Select the **Circle**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Circle**; or right-click and select **Circle** from the pop-up menu.
- 2 Position the cursor at the center point location.
- 3 Click in the Work Area to place the center of the circle and drag the cursor to sketch the circle.
- 4 Click again to complete the circle.



To resize a circle:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the circle.
- 3 Click, hold the mouse button, and drag the figure to resize the circle.
- 4 Release the mouse button.

To move a circle:

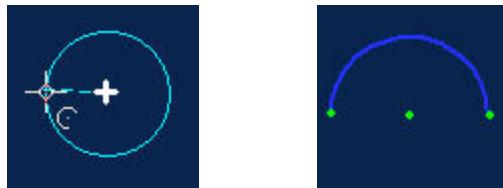
- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the center node of the circle.
- 3 Click, hold the mouse button, and drag the circle.
- 4 Release the mouse button.

4.3.3 Circular Arcs

You can sketch three different circular arc types: 1) **Center, Start, End**; 2) **Start, End, Radius**; 3) **Tangent-Start, End**.

To sketch a circular arc using **Center, Start, End**:

- 1 Select the **Circular Arc – Center, Start, End**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Circular Arc > Center, Start, End**; or right-click and select **Circular Arc** from the pop-up menu.
- 2 Position the cursor at the center of the arc.
- 3 Click to place the center of the arc.
- 4 Click a second time to start the arc.
- 5 Move the cursor to sketch the arc.
- 6 Click a third time to complete the arc.



To sketch a circular arc using **Start, End, Radius**:

- 1 Select the **Circular Arc – Start, End, Radius**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Circular Arc > Start, End, Radius**.
- 2 Position the cursor at the arc starting location.
- 3 Click to start the arc.
- 4 Click a second time to place the arc endpoint.
- 5 Move the cursor to size the arc.
- 6 Click a third time to complete the arc.

To sketch a circular arc using Tangent-Start, End:

- 1 Select the **Circular Arc – Tangent-Start, End**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Circular Arc > Tangent-Start, End**.
- 2 Click the line or circular arc that will be tangent to the new arc.
- 3 Move the cursor to size the arc.
- 4 Click a second time to complete the arc.

To increase or decrease a circular arc's diameter:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the circular arc.
- 3 Click, hold the mouse button, and drag the figure to resize the circular arc.
- 4 Release the mouse button.

To reshape a circular arc:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the circular arc center node.
- 3 Click, hold the mouse button, and drag the center node to reshape the circular arc.
- 4 Release the mouse button.

To move a circular arc:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Select the arc AND center node by dragging a selection rectangle around the entities.
- 3 Hold the **Shift** key, click and hold the mouse button, and drag the circular arc.
- 4 Release the mouse button.

4.3.4 Rectangles

You can sketch two different rectangle types: 1) **Rectangle by Two Corners**; 2) **Rectangle by Three Corners**.

To sketch a rectangle using Two Corners:

- 1 Select the **Rectangle by Two Corners**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Rectangle > Two Corners**.
- 2 Click to place one corner of the rectangle.
- 3 Move the cursor to sketch the rectangle.
- 4 Click a second time to locate the opposite corner of the rectangle.



To sketch a rectangle using Three Corners:

- 1 Select the **Rectangle by Three Corners**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Rectangle > Three Corners**.
- 2 Click to place one corner of the rectangle.
- 3 Move the cursor and click a second time to locate the other corner on the same end of the rectangle.
- 4 Move the cursor to adjust the rectangle length.
- 5 Click a third time to place the end of the rectangle.

To resize a rectangle:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over an edge or a node.
- 3 Click, hold the mouse button, and drag the sketch entity to resize the rectangle.

- 4 Release the mouse button.

To move a rectangle:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Select the entire rectangle by dragging a selection rectangle around it.
- 3 Hold the **Shift** key, click and hold the mouse button, and drag the rectangle.
- 4 Release the mouse button.

4.3.5 Spline Curves

Alibre Design provides NURBS (B-spline) curve functionality in sketches. The main distinguishing feature of the B-spline compared to "simple" splines is that it retains its shape at all times. That is, it can only translate and rotate rigidly in order to satisfy the constraints imposed on it.

Creation of NURBS curves by control points

Using this method, you specify a set of control points in the Work Area to define the B-spline. Existing points can be identified. A preview is shown as you move the mouse.

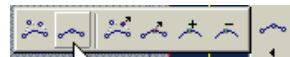


- 1 Select the **B-spline by control points** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Spline > Create > B-spline by control points**.
 - 2 Click to start the spline curve.
 - 3 Move the cursor and click a second time to place a control point.
 - 4 Move the cursor to shape the curve.
 - 5 Continue clicking to place additional control points.
- Note:** You may choose to specify one or more control points by invoking the direct coordinate entry tool and keying in the X, Y coordinates.
- 6 Double-click or hit escape to complete the spline curve.

Note: By making the last control point to be the same as the first control point, a closed B-spline curve can be created.

Creation of NURBS curves by interpolation

Using this method, you first specify a set of interpolation points in the Work Area that define the B-spline. A curve is then interpolated through the interpolation points.



- 1 Select the **B-spline by interpolation points** tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Spline > Create > B-spline by interpolation points**.
 - 2 Click to start the spline curve.
 - 3 Move the cursor and click a second time to place an interpolation point.
 - 4 Move the cursor to shape the curve.
 - 5 Continue clicking to place additional interpolation points.
- Note:** One or more interpolation points may be specified via the direct coordinate entry tool.
- 6 Double-click or hit escape to complete the spline curve.

Constraining B-spline curves

The following constraints are supported on the new B-spline curve:

- **Coincident constraint:** An existing reference point can be made coincident with a location on the B-spline by either dragging it onto the B-spline OR by using the sketch Coincident constraint tool.

Note: A point coincident to a B-spline is kept “floating”, meaning that the constraint system can move the point of coincidence to any other location along the curve. However any coincidence established at either endpoint of the B-spline remains fixed.
- **Tangent constraint:** can be placed between a B-spline curve and any other figure in the sketch that can participate in the constraint system.
Note: The note above on “floating” and “fixed” constraint applies to this constraint also.

- **Perpendicular constraint** can be placed between a B-spline curve and any other figure in the sketch that can participate in the constraint system.
Note: The note above on “floating” and “fixed” constraint applies to this constraint also.
- **Intersection point constraint** can be placed between a B-spline curve and any other figure in the sketch that can participate in the constraint system.
- **Fixed constraint** allows the B-spline curve to be locked in place.

Moving a spline curve

The main distinguishing feature of the B-spline is that it retains its shape at all times. That is, it can only translate and rotate rigidly in order to satisfy the constraints imposed on it. Other items to note include:

- Dragging a B-spline by its endpoint to existing sketch geometry introduces a “fixed” coincident constraint between the B-spline and that figure.
- Otherwise, dragging a B-spline to an existing reference point introduces a “floating” coincident constraint between the B-spline and that figure.
- Dragging an existing reference point to a B-spline introduces a coincident constraint between the point and the B-spline. It is “fixed” or “floating” depending on whether the point was dragged to the B-spline’s endpoint or an internal location respectively.

To move a spline curve rigidly:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Click and hold the mouse button on the curve, and drag the spline curve.
- 3 Release the mouse button to place the curve.

Trimming and extending spline curves

The B-spline curve can be trimmed using other figures (lines, arcs, circles, ellipses and B-splines).

Open figures like lines and arcs can be extended to the intersection point with a B-spline curve. However, the B-spline curve itself cannot be extended.

Spline curve shape modification

An assortment of methods is available to modify or tweak the B-spline curve's shape while still honoring all the constraints placed on the curve. These tools take advantage of the excellent local shape modification properties of B-splines.

To modify a spline curve's shape:

1. From the **Sketch** menu, select **Figures > Spline > Edit > Desired Action**

Possible Actions:

Move Control Points — Modify shape by moving a control point

Move Curve Points — Modify shape by moving a point on the B-spline curve to a new position.

Insert Knots — Insert new knots on the curve knot vector without changing curve shape.

Remove Redundant Knots — Remove existing knots that can be removed without changing the curve shape. Alibre Design will automatically remove any possible knots; then a dialog box will appear summarizing the results.

DXF/DWG import/export of NURBS curves

NURBS geometry defined by control points present in DXF/DWG files can be read and precisely represented by the same mathematical representation in Alibre Design.

DWG files can also contain splines that are defined by interpolation points. Alibre Design will read these and convert them into B-spline curves. While for most parts these curves will appear very similar to what they look like in AutoCAD, their shapes may not exactly match. Also, AutoCAD allows users to specify "tolerance" while defining splines by interpolation. Alibre Design will assume this value is always zero.

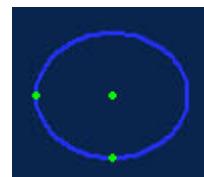
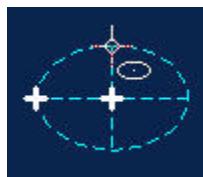
Using Offset on a spline curve

The offset tool can be used to select B-spline curves for creating an offset curve. As with other figures this offset will not be associative to the original B-spline curve. Reference section 4.5.5 for details on how to use the Offset tool.

4.3.6 Ellipses

To sketch an ellipse:

- 1 From the **Sketch** menu, select **Figures > Ellipse**.
- 2 Click to select the ellipse center.
- 3 Move the cursor and click a second time to place the major axis of ellipse.
- 4 Move the cursor and click a third time to place the minor axis of the ellipse.



To resize an ellipse:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the ellipse.
- 3 Click, hold the mouse button, and drag to reshape the ellipse.
- 4 Release the mouse button.

To move an ellipse:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Select the entire ellipse by dragging a selection rectangle around it.
- 3 Hold the **Shift** key, click and hold the mouse button, and drag the ellipse.
- 4 Release the mouse button.

4.3.7 Elliptical Arcs

To sketch an elliptical arc:

- 1 From the **Sketch** menu, select **Figures > Elliptical Arc**.
- 2 Click to select the elliptical arc center.
- 3 Move the cursor and click a second time to place the major axis of elliptical arc.
- 4 Move the cursor and click a third time to place the minor axis of elliptical arc.
- 5 Click a fourth time to start the elliptical arc.
- 6 Click a fifth time to complete the elliptical arc.



To resize an elliptical arc:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the elliptical arc or one of the associated nodes.
- 3 Click, hold the mouse button, and drag resize the elliptical arc.
- 4 Release the mouse button.

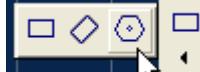
To move an elliptical arc:

- 1 Select the **Select**  tool from the Sketching toolbar.
- 2 Select the entire spline curve by dragging a selection window around it.
- 3 Hold the **Shift** key, click and hold the mouse button, and drag the elliptical arc.
- 4 Release the mouse button.

4.3.8 Polygons

To sketch an n-sided polygon:

- 1 From the **Sketch** menu, select **Figures > Polygon**; or select the **Regular Polygon**



tool from the Sketching toolbar. The Regular Polygon dialog box appears.

- 2 Enter the number of sides for the polygon.
- 3 Choose to measure the internal or external diameter.
- 4 Click in the sketch window to locate the center of the polygon. Move the cursor to drag the polygon to the desired size. Click again to place the polygon.
- 5 Choose **Apply**; then **Close**.

To move an n-sided polygon:

- 1 Select the Select tool from the Sketching toolbar.
- 2 Select the entire polygon by dragging a selection rectangle around it.
- 3 Hold the Shift key, click and hold the mouse button, and drag the polygon.
- 4 Release the mouse button.

4.4 Reference Figures and Sketch Nodes

Reference figures and sketch nodes are used as construction geometry. For example, a reference line can be sketched and subsequently used in a sketch mirror operation. You can also place dimensions and constraints in relation to reference figures and sketch nodes. Reference figures and sketch nodes are contained within a sketch but are only visible in sketch mode. Reference figures are displayed as green dashed lines in sketch mode. Reference figures can be created in the exact same shapes as normal sketch figures.

To sketch a reference figure:

- 1 From the **Sketch** menu select **Reference Figures** > and then select a figure type.
- 2 To sketch the reference figure, follow the same steps you would use to create a normal sketch figure.



You can move and resize a reference figure just as you would a normal sketch figure.

To place a sketch node:

- 1 Select the **Sketch Node**  tool from the Sketching toolbar; or from the **Sketch** menu select **Figures > Node**.
- 2 Click once to place a sketch node.



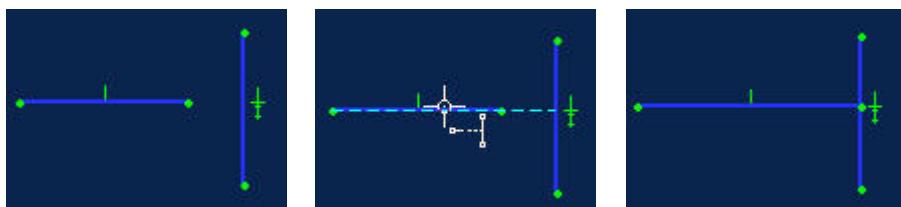
4.5 Working with Existing Sketch Figures

4.5.1 Extending Figures

You can use the **Extend** tool to extend a line or arc to meet another line, arc, circle, spline, or reference line.

To extend a sketch figure:

- 1 Select the **Extend**  tool from the Sketching toolbar; or from the **Sketch** menu select **Extend**; or right-click and select **Extend** from the pop-up menu.
- 2 Move the cursor over the line or arc that you want to extend.
A dashed preview will appear showing the direction of the extended entity. If the direction is incorrect, move the cursor to the opposite end of the entity.
- 3 To generate the extension, click once on the entity.



4.5.2 Trimming Figures

You can use the **Trim** tool to delete portions of sketch entities based on intersections with other entities. You can trim a line, arc, ellipse, circle, and spline that intersect with other lines, arcs, ellipses, circles, splines, and reference lines.

To trim a sketch figure:

- 1 Select the **Trim Figure**  tool from the Sketching toolbar; or from the **Sketch** menu select **Trim**; or right-click and select **Trim** from the pop-up menu.
- 2 Move the cursor over the portion of the sketch figure that you want to trim. The portion becomes highlighted.
- 3 Click the highlighted portion to delete it up to its intersection with another sketch figure. The entire figure will be deleted if it does not intersect with another figure.



4.5.3 Adding Fillets to Sketch Figures

You can use the **2D Fillet** tool to place a tangent arc at the intersection of two sketch figures and subsequently delete the corner. You can also place a fillet on non-intersecting figures; the figures will be extended and a fillet will be placed accordingly at the resultant intersection.

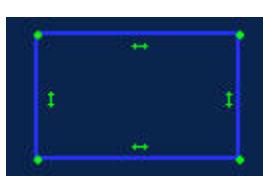
To add a 2D fillet to a sketch figure:

- 1 Select the **2D Fillet**  tool from the Sketching toolbar; or from the **Sketch** menu select **Fillet**. The **Fillet Figures** dialog box appears.
- 2 Select the first figure by clicking it. The figure name appears in the **Figures to Fillet** area in the dialog box.
- 3 Select the second figure by clicking it. The figure name appears in the **Figures to Fillet** area in the dialog box.
- 4 Enter the fillet radius value in the **Radius** box.
- 5 Click **Apply** to create the fillet.

The Fillet Figures dialog box remains open so you can continue to place fillets on other figures.

Note: Consecutive fillets with a diameter equal to the first fillet will not be dimensioned; instead an equal constraint will automatically be placed.

- 6 Click **Close** to close the Fillet Figures dialog box.



4.5.4 Adding Chamfers to Sketch Figures

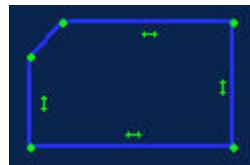
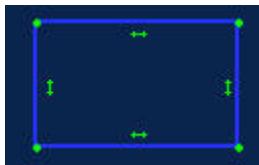
You can use the **2D Chamfer** tool to place a beveled edge at the intersection of two sketch figures and subsequently delete the corner. You can also place a chamfer on non-intersecting figures; the figures will be extended and a chamfer will be placed accordingly at the resultant intersection.

To add a 2D chamfer to a sketch figure:

- 1 Select the **2D Chamfer**  tool from the Sketching toolbar; or from the **Sketch** menu select **Chamfer**. The **Chamfer** dialog box appears.
- 2 Select the first figure by clicking it. The figure name appears in the **Figures to Chamfer** area in the dialog box.
- 3 Select the second figure by clicking it. The figure name appears in the **Figures to Fillet** area in the dialog box.
- 4 Enter the chamfer distance value in the **Distance** box.
- 5 Click **Apply** to create the chamfer.

The Chamfer dialog box remains open so you can continue to place chamfers on other figures.

- 6 Click **Close** to close the Chamfer dialog box.



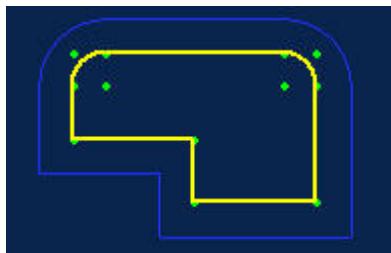
4.5.5 Offsetting Figures

You can use the **Offset** tool to automatically create sketch figures offset from another selected figure or sketch by a specified distance.

To offset a sketch figure or figures:

- 1 Select the **Offset**  tool from the Sketching toolbar. The **Offset** dialog box appears.

- 2 Select the figure(s) to offset either one at a time or drag a selection rectangle around all figures. The figure name(s) appears in the **Figures to Offset** area in the dialog box.
- 3 Enter the offset distance value in the **Distance** box.
- 4 If necessary, select the **Flip Direction** option to create the offset in the opposite direction.
- 5 Select a **Gap Type** (the default is **Natural**) . . .
 - **Natural:** Extends the figures along their natural curves until they intersect; for example, along a circle and along a straight figure.
 - **Round:** The offset figure will contain fillets on any corners.
 - **Extend:** Extends the figures in straight lines until they intersect.
- 6 Click **OK** to create the offset figure(s). The new figures are created and become part of the sketch.



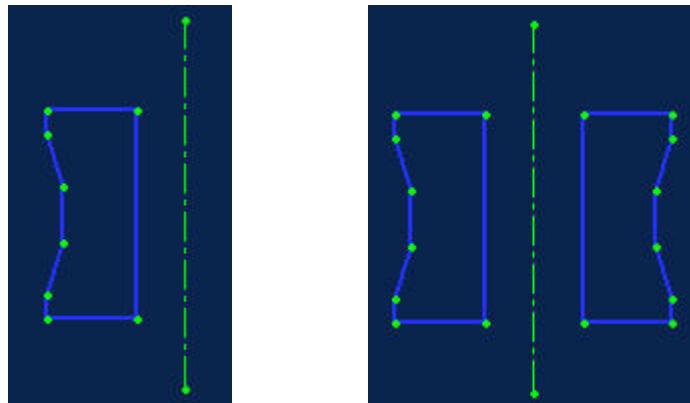
4.5.6 Mirroring Figures

You can use the Mirror tool to create copies of figures mirrored about another reference line or figure. A **Symmetric** constraint is automatically applied between the original figure and the mirror figure (refer to section 4.6 to learn more about constraints). If you change the original figure, the mirrored figure will also change.

To mirror a sketch figure:

- 1 Select the **Mirror**  tool from the Sketching toolbar. The **Mirror Figure** dialog box appears.
- 2 Select the figure(s) to mirror either one at a time or drag a selection rectangle around all figures. The figure name(s) appears in the **Figures to mirror** area in the dialog box.

- 3 Select the **Mirror Axis** to mirror the figure about. The mirror axis can be either a reference line or another sketch figure.
- 4 Click **OK** to create the mirrored figure.



4.5.7 Creating Sketch Figure Patterns

You can create linear and radial patterns of an existing sketch figure. Linear patterns can be created in one and two directions.

To create a linear pattern in one direction:

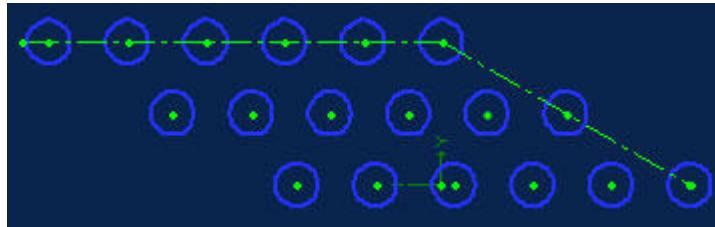
- 1 From the **Sketch** menu, select **Repeat > Linear**. The Linear Repeat dialog box appears.
- 2 Select the figure to be patterned.
- 3 Select the linear path for the first pattern direction. Lines, reference lines, axes, and edges can be used as the linear path.
- 4 Enter the appropriate value in the **Copies** field; this value includes the original figure.
- 5 Enter the appropriate **Spacing** value that controls the distance between each figure in the pattern.
- 6 If necessary, select the **Change Direction** option.

-
- 7 Click **OK** to create the pattern.



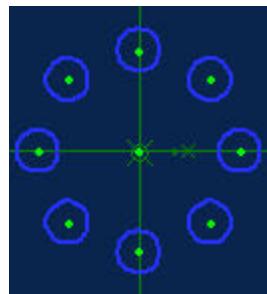
To create a linear pattern in two directions:

- 1 From the **Sketch** menu, select **Repeat > Linear**. The Linear Repeat dialog box appears.
- 2 Select the figure to be patterned.
- 3 Select the linear path for the first pattern direction. Lines, reference lines, axes, and edges can be used as the linear path.
- 4 Enter the appropriate value in the **Copies** field; this value includes the original figure.
- 5 Enter the appropriate **Spacing** value that controls the distance between each figure in the pattern.
- 6 If necessary, select the **Change Direction** option.
- 7 Select the linear path for the second pattern direction. The second linear path should not be parallel to the first.
- 8 Enter the appropriate value in the **Copies** field; this value includes the original figure.
- 9 Enter the appropriate **Spacing** value that controls the distance between each figure in the pattern.
- 10 If necessary, select the **Change Direction** option.
- 11 Click **OK** to create the pattern.



To create a radial pattern:

- 1 From the **Sketch** menu, select **Repeat > Circular**. The **Circular Pattern** dialog box appears.
- 2 Select the figure to be patterned.
- 3 Select the **Circular path center**. Axes, points, and existing edges on other features can be used as the circular path center.
- 4 Enter the appropriate value in the **Copies** field; this value includes the original figure.
- 5 Enter the angle that will separate each copy in the radial direction.
- 6 If necessary, select the **Change Direction** option.
- 7 Click **OK** to create the pattern.

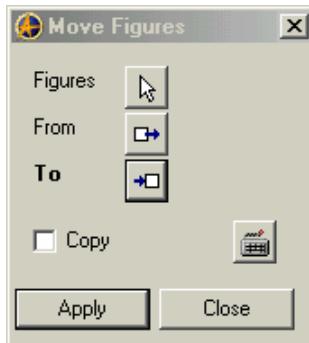


4.5.8 Moving and Rotating Sketch Figures

You can move existing sketch figures from one location to another, or rotate them about an axis.

To move sketch figures:

- 1 From the **Sketch** menu, select **Move**. The **Move Figures** dialog box appears.



- 2 Click the **Figures** selection button. Select the figures you want to move in the workspace.
- 3 Click the **From** button. Click a location in the workspace that you want to move the figures from.
- 4 Click the **To** button. Click the location in the workspace that you want to move the figures to.
- 5 Check **Copy** if you want the original figures to be copied to the new location.
- 6 Click **Apply**; then **Close**.

Note: If you want to move the figures using coordinates, click the **Direct Coordinate Entry**  button.

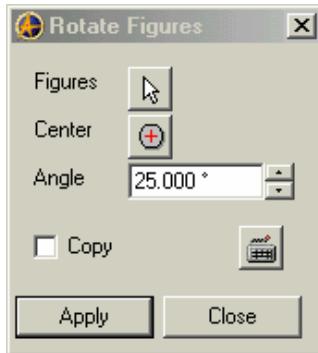
OR:

To move figures within a sketch after placement:

- 1 Select the sketch **Select**  tool from the Sketching toolbar.
- 2 Select the figures to be moved. If you are moving an entire sketch, from the **Edit** menu, select **Select All**, or press **Ctrl + A** on the keyboard. The figures become highlighted.
- 3 Hold the **Shift** key on the keyboard, and click and drag the sketch figure(s) to the new location.

To rotate sketch figures:

- 1 From the **Sketch** menu, select **Rotate**. The **Rotate Figures** dialog box appears.



- 2 Click the **Figures** selection button. Select the figures you want to rotate in the workspace.
- 3 Click the **Center** button. Click the location in the workspace around which you want to rotate the figures.
- 4 In **Angle**, set the angle of rotation.
- 5 Check **Copy** if you want the original figures to be copied to the new location.
- 6 Click **Apply**; then **Close**.

Note: If you want to move the figures using precise coordinates, click the **Direct Coordinate Entry** button.

4.6 Sketch Constraints

Figures in a sketch may be constrained to a size, orientation, and relationship to another 2D figure or 3D edge. Some constraints are used to apply relationships between figures (for example, perpendicular, tangent, parallel, equal size) or between figures and reference lines, planes, axes, vertices, and edges. Other constraints are applied to individual figures, including those that control a figure's orientation (for example, horizontal or vertical) and dimension.

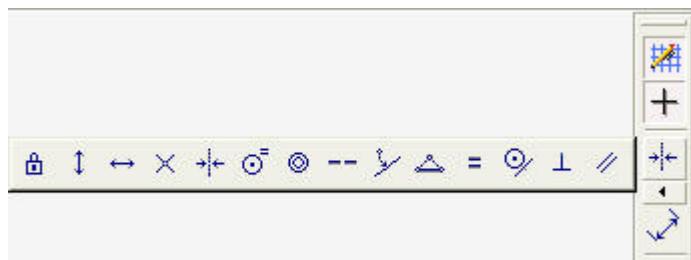
Some sketch constraints are applied automatically as figures are sketched. Horizontal, vertical, coincident, midpoint, tangent, intersection, and perpendicular constraint types are automatically placed depending on the type, size and orientation of the figure(s).

Sketch constraints are displayed in a bright green color and are shown in proximity to the applicable sketch figure.



Note: When you sketch a new figure on an existing figure or on a node on an existing figure (e.g. start a new line on the node at the end of an existing line), a **coincident** constraint is automatically applied. The coincident constraint is not displayed in this case. To break this coincident constraint, move the cursor over the coincident nodes, hold the **Ctrl** key, click and drag.

Constraints can also be applied manually to figures after they have been sketched with tools from the Constraints toolbar. The Constraints fly-out toolbar is available from the Sketching toolbar in part and drawing workspaces.



4.6.1 Constraint Types

Fourteen different constraints can be applied to sketch figures.



Fixed – Figures may be constrained to a fixed position in the sketch. After the constraint is applied, the node or figure may not be moved without first deleting the constraint.

Can be applied to: a node or any sketch figure



Vertical — One or more lines may be constrained to be vertical. Sketch nodes may also be constrained to be vertically aligned. Lines may be vertically constrained automatically as they are sketched or after placement.

Can be applied to: any line or any two nodes



Horizontal — One or more lines may be constrained to be horizontal. Sketch nodes may also be constrained to be horizontally aligned. Lines may be horizontally constrained automatically as they are sketched or after placement.

Can be applied to: any line or any two nodes



Intersection — Two figures may be constrained to intersect at a point.

Can be applied to: a point and any combination of arcs or lines



Symmetric — An axi-symmetric relationship may be defined between figures. After a symmetric constraint is applied, the figures are arranged axi-symmetrically and equidistant from a reference line or sketch line. The figures will become equal in size after the constraint has been placed.

Can be applied to: any two figures of like nature, e.g. two lines or two circles



Co radial — Figures may be constrained to share the same center point and same radius. Circles/arcs can be coradially constrained automatically during sketching or after placement.

Can be applied to: two or more arcs or circles



Concentric — Figures may be constrained to share the same center point. Circles/arcs can be concentrically constrained automatically during sketching or after placement.

Can be applied to: two or more arcs or circles



Collinear — Figures may be constrained so that they lie in the same line. Lines may be collinearly constrained automatically as they are sketched or after placement.

Can be applied to: a combination of two or more lines, axes, reference lines, edges



Coincident — A point can be constrained so that it lies on a figure.

Can be applied to: a point and any sketch figure



Midpoint – A node can be constrained so that it is fixed at the middle of a line. Midpoint constraints can be placed automatically as a figure is sketched or after a figure has been sketched.

Can be applied to: a node and a line or arc



Equal – Figures can be constrained to be equal in size. Equal constraints can be applied automatically during sketching or placed manually after a figure has been sketched.

Can be applied to: any two or more sketch figures



Tangent – Figures can be constrained to be tangent to a curve. Tangent constraints can be applied automatically during sketching or placed manually after a figure has been sketched.

Can be applied to: a curve and a line or two curves



Perpendicular – Lines can be constrained to be perpendicular to other linear entities. Perpendicular constraints can be applied automatically during sketching or placed manually after a figure has been sketched.

Can be applied to: two lines or a line and a circle or circular arc



Parallel – Lines can be constrained to be parallel to each other. Lines can be constrained parallel automatically during sketching or can be manually constrained after placement.

Can be applied to: at least two lines

4.6.2 Manually Applying Sketch Constraints

To manually apply a sketch constraint:

- 1 Click the small black arrow  on the Sketching toolbar to access the Constraint toolbar, or from the **Sketch menu**, select **Constraints**.

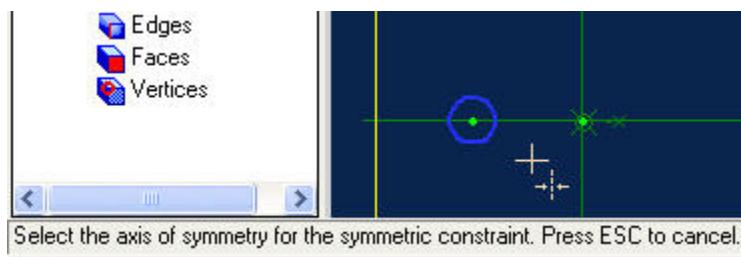
- 2 Select the applicable constraint tool. The cursor changes to show the corresponding constraint symbol. For example, after selecting the **Coincident** constraint tool,



cursor changes to

- 3 In the sketch, select the figures to constrain. Many of the constraint types require multiple selections. For example, applying a symmetric constraint first requires selecting a reference line or sketch line and then selecting two other sketch figures. Simply click all the required entities one-by-one to apply the constraint.

Note: When manually applying constraints, hints are displayed in the status bar in the lower left corner of the workspace. The hints provide step-by-step instructions to apply a constraint.



4.6.3 Deleting Constraints

Sketch constraints can be deleted at anytime regardless of how they were created, i.e. automatically or manually.

To delete a sketch constraint:

- 1 Select the sketch **Select**
- 2 Position the cursor over the sketch constraint you want to delete. The figure associated with the constraint is highlighted and the cursor displays the selected constraint symbol.



- 3 When the anchor symbol appears, right-click the constraint and select **Delete** from the pop-up menu. The constraint is deleted.

Note: For constraints applied to multiple figures, deleting the constraint from one deletes that constraint from all the other figures in that constraint group.

4.6.4 Controlling the Display of Sketch Constraint Symbols

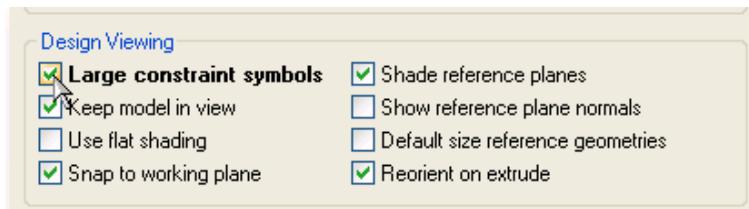
Sketch constraints symbol visibility can be turned on and off. In addition, you can control the size of the sketch constraint symbols.

To turn off sketch constraint visibility:

- 1 From the **View** menu, select **Constraint Symbols**, or press **Ctrl+Shift+C** on the keyboard. Figures will remain constrained when constraint symbols are hidden. This is a toggle on/off, so they visibility can be turned back on using the same command.

To change the size of the constraint symbols:

- 1 From the **Tools** menu, select **Options**. On the **General** tab, in the **Design Viewing** field, check **Large Constraint Symbols**. This will increase the size of the constraint symbols. Un-checking the option will return the symbols to their original size.



4.6.5 Checking the Status of a Sketch

Constraint status of an individual figure

There are several possible states that indicate whether a figure is constrained completely or not. A figure is constrained completely when zero degrees of freedom remain. These states are displayed in the workspace status bar when a sketch constraint tool is selected and the cursor is positioned over a figure.



Well-defined: A figure is fully constrained and dimensioned; there are no remaining degrees of freedom.

Under-defined: A figure is not fully constrained or dimensioned; figures can move unexpectedly as a result.

Over-defined: A figure has conflicting constraints and/or dimensions that may or may not cause an additional constraint to fail. If a constraint fails, delete one or more constraints or dimensions.

Fixed: A figure is fully constrained and the figure cannot be modified. Other figures can be constrained to it.

Not-changed: The indicated constraint was not applied to this geometry. The figure is dependent on another figure with conflicting constraints.

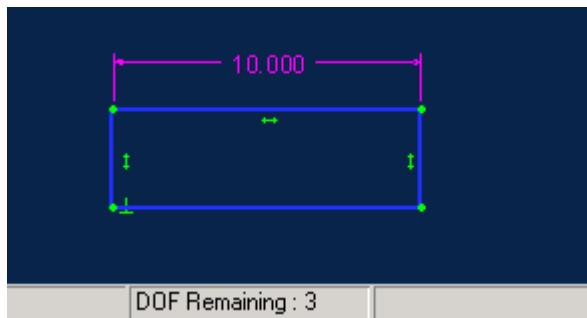
Not-consistent: The assigned dimension value(s) and constraints are in conflict and cannot be applied to the geometry.

Unknown: Occurs when a component of a constraint has been removed.

It is not necessary for figures in a sketch to be well-defined before you use the sketch to create a feature. However, it is good design practice in general to ensure that sketches are well-defined.

Status of entire sketch

By default, the number of remaining degrees of freedom (DOF) in the entire sketch is displayed in the status area in the lower right corner of a workspace. The DOF value will increase or decrease automatically as you sketch or delete figures, and add or delete dimensions and constraints. A fully defined sketch will have zero degrees of freedom. It is not required to fully define a sketch before it can be used in a feature operation.



To hide the DOF hints:

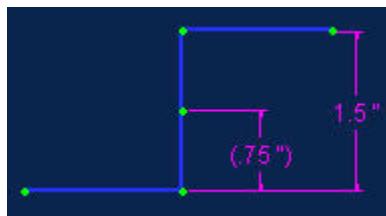
- 1 From the **Tools** menu, select **Options**. The Options dialog box appears.
- 2 Select the **General** tab if it is not already selected.
- 3 In the **Hints** area, unselect **DOF** hints.
- 4 Click **OK**.

4.7 Dimensioning Sketch Figures

Normally, to fully define and capture design intent in a sketch you must place dimensions on sketch figures. However, it is not required to dimension sketches before they are used to create features. Most importantly, sketch dimensions can easily be changed and modified at any time. Additionally, any dimensions you place in a sketch will in turn be displayed in the 2D drawing that is based on the part.

Two types of dimension states exist: driving and driven. Driving dimensions are used to define and constrain a figure. After driving dimensions have been placed on a figure, driven dimensions can also be added that are dependent upon the values of the driving dimensions. By default, driven dimensions are displayed in parentheses. Driven dimensions cannot be edited since they are dependent on driving dimensions. Subsequent changes to driving dimensions automatically update the driven dimensions.

In the figure below, the **1.5"** dimension is a driving dimension, and the **.75"** dimension is a driven dimension. If the **1.5"** dimension was changed, then the **.75"** dimension would automatically change as well.



4.7.1 Dimensioning Sketch Figures

Linear, radial, diametrical and angular dimensions may be created. Linear dimensions may be placed on a line, between two parallel lines, or between nodes. Diameter and radius dimensions may be created for circles and arcs. Angular dimensions may be created between two non-parallel lines.

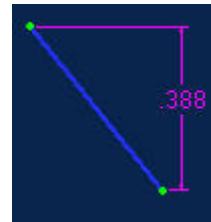
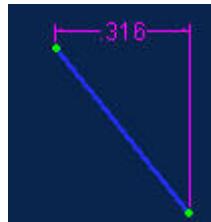
To dimension individual figures:

Follow the steps below to place length dimensions on lines, and diameter dimensions on arcs and circles.

- 1 Select the **Dimension** tool from the Sketching toolbar; or from the **Sketch** menu, select **Dimension**; or right-click in the work area and select **Dimension** from the pop-up menu.
- 2 Move the cursor over the figure you want to dimension. The figure is highlighted.
- 3 Click the figure to show a preview of the dimension. Move the cursor to move the dimension preview.

Note: You can press the **Esc** key on the keyboard to cancel the current dimension operation.

Depending on the type of figure you are dimensioning as well as where you move the preview dimension, a new dimension may be inferred. For example, depending on where the cursor is moved, three different dimensions could be placed on an angled line.



- 4 After the dimension has been positioned properly, click again. A dimension control box appears.



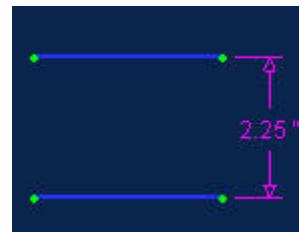
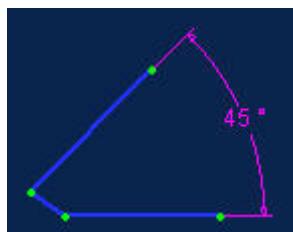
- 5 Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.

Note: You can enter fractions (e.g. **3/8**) and simple equations (e.g. **1.5 * 3**) into the dimension control box. You can also enter a value with units other than the current display units (e.g. **5 mm**). The value will be converted to the display units automatically. Supported unit abbreviations are "**,**", '**mm**', '**cm**', and '**m**'.

To dimension distances or angles between figures:

Follow the steps below to place distance dimensions or angular dimensions between figures, e.g. between lines, between nodes, between arc or circle center nodes, etc.

- 1 Select the **Dimension**  tool from the Sketching toolbar, or from the **Sketch** menu, select **Dimension**, or right-click in the work area and select **Dimension** from the pop-up menu.
- 2 Move the cursor over the first figure you want to dimension from.
- 3 Click the figure, a preview may appear but do not place the dimension at this time.
- 4 Move the cursor over the second figure you want to dimension to and click again. A new dimension preview appears.
- 5 Move the cursor to position the dimension and click a third time. A dimension control box appears.
- 6 Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.



4.7.2 Auto Dimensioning a Sketch

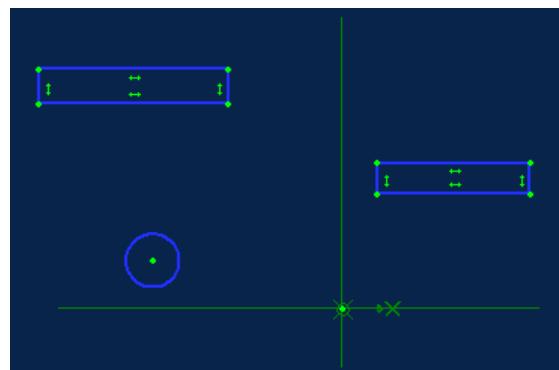
You can automatically place driving dimensions on an entire sketch or on a selected subset of sketch figures. The number of dimensions placed automatically will vary depending on the number of sketch constraints that exist in the sketch. If you want to automatically place as many dimensions as possible initially, it is recommended you minimize the use of sketch constraints.

From the **Sketch**, select **Auto Dimension**. The Auto Dimension dialog appears.

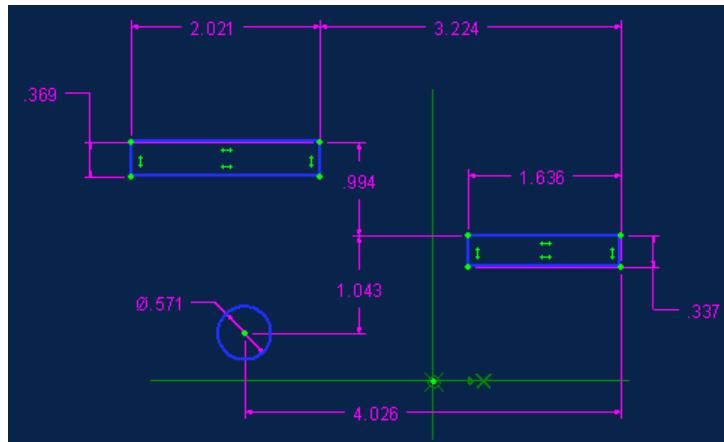


To auto dimension the entire sketch:

- 1 Choose the option **All figures** and click the **Apply** button.
- 2 Dimensions appear on the sketch. You can **Close** the dialog and modify any of the dimensions as required.



Before Applying Auto Dimension



After Applying Auto Dimension

To auto dimension a subset of the sketch figures:

- 1 Choose the option **Selected figures**.
- 2 Select one or more figures to dimension and click the **Apply** button.
- 3 Dimensions appear for the selected figures. You can either select and dimension additional figures or **Close** the dialog and modify the created dimensions.

4.7.3 Using Spinner Controls

When you place dimensions on sketch figures, you can enter the dimension value manually by typing a value. You can also use the spinner arrows to incrementally change a dimension value based on a pre-determined increment value.

To set the spinner increment:

- 1 From the **File** menu, select **Properties**. The Design Properties dialog box appears.
- 2 Select the **Units** tab if it is not already selected.
- 3 In the **Spinner Increment** area, enter a **Length** increment value, i.e. .125", .250", .375", etc.
- 4 Also enter an **Angle** increment value based on degrees.

- 5 Click **Apply** and then click **Close**.

To use the spinner arrows:

- 1 Select the **Dimension**  tool from the Sketching toolbar and dimension a figure.
- 2 When the dimension control box appears, click one of the two black arrow buttons to increase or decrease the dimension by the spinner increment value.



4.7.4 Using Equations in Dimensions

You can create dimensions using mathematical relations between dimensions or parameters, using dimension names as variables in the equations.

To use equations in dimensions:

- 1 From the **File** menu, select **Properties**. The **Design Properties** dialog box appears.
- 2 Select the **Dimension** tab.
- 3 Select the **Show Equations** option.
- 4 Click **Apply** and then click **Close**.
- 5 Place a dimension on a figure. The parameter name associated with the dimension is now displayed.



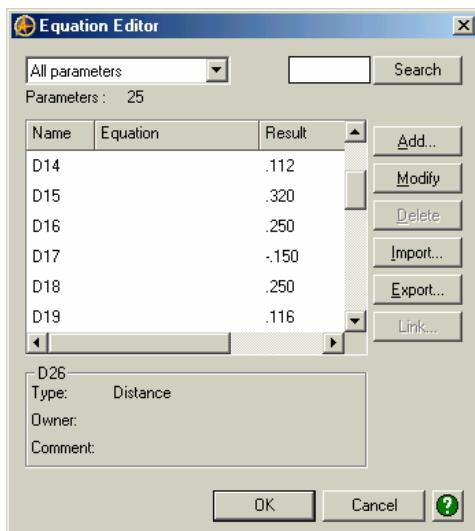
You can then create new dimensions by referencing existing parameters. You can then manage the equations using the **Equation Editor**.

To use the equation editor:

- 1 You can access the **Equation Editor** directly when placing dimensions. Press the **Edit Equation**  button on the dimension control box.



The **Equation Editor** dialog box appears in which you can add new equations and constants, use an existing equation to set the value of the dimension, or set the value equal to another dimension. The dimension parameters are listed under the **Name** column, the current values of the parameters are listed under the **Result** column, and the dimension type is listed under the **Type** column.



You can access the Equation Editor at anytime. From the **Tools** menu, select **Equation Editor**, or press **Ctrl + E** on the keyboard.

- 2 To modify an equation, select the equation from the list, and click **Modify**, or double-click in the field that you want to modify, i.e. name or equation.
- 3 If necessary, you can click **Add** to create new parameters. In the **Add Equation** dialog, specify the name of the new parameter; the type of parameter you want to create (Distance, Angle, Count, or Scalar); and the equation that will define its value. You can also add a comment if you desire. When you are finished, **Close** the **Add Equation** dialog. Type in the new variable name, new equation, or variable value and press **Enter** on the keyboard.

Chapter 4 - Sketching

- 4** Click **OK** in the Equation Editor dialog to apply the changes.

Functions available for use

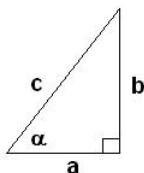
The following functions are available for use in equations in Alibre Design.

Function	Name	Description
abs(x)	absolute value	Returns the absolute value of the argument.
acos(x)	arc cosine	$\text{acos } (a/c) = \alpha \text{ in radians } x \leq 1$
asin(x)	inverse sine	$\text{asin } (b/c) = \alpha \text{ in radians } x \leq 1$
atan(x)	arc tangent	$\text{atan}(b/a) = \alpha \text{ in radians } x \leq 1$
cos(x)	cosine	Returns the cosine of an angle. The argument can be any valid numeric expression in radians. $\cos(\alpha) = a/c$
int(x)	integer	Returns the integer portion of the argument. The argument can be any valid numeric expression. If the argument is negative, int() returns the first negative integer less than or equal to the number. If the argument is a positive decimal number, such as 0.987, null is returned.
frac(x)	fraction	Returns just the decimal portion of the argument.
sign(x)	sign	Returns the sign of the argument. The argument can be any valid numeric expression. If the number is greater than zero, sign() returns 1, if the number returns 0, and if negative, the sign() returns -1.
sin(x)	sine	Returns the sine of an angle. The argument can be any valid numeric expression in radians. The sin() function takes an angle and returns the ratio of two sides of a right triangle. The ratio is the length of the side opposite the angle divided by the length of the hypotenuse. To convert degrees to radians, multiply degrees by $\pi/180$. To convert radians to degrees, multiply radians by $180/\pi$. $\sin(\alpha) = b/c$
sqrt(x)	square root	Returns the square root of the argument: $x \geq 0$

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$\tan(x)$	tangent	The argument can be any valid numeric expression that expresses an angle in radians. $\tan(\alpha) = b/a$
X^n	X^n	$x > 0$

In the table above, "x" is a real number; "n" is an integer; and "a," "b," "c" and " α " have the following relationship:



Dimensionality of equations

Each parameter has a specified dimensionality (that is, length, angle, scalar, count). When you write an equation for a parameter, the equation's dimensionality must match that of the parameter. If the equation's dimensionality is different, the equation will be displayed in red and a popup error message will appear when you rollover the equation.

For example, if D1 and D2 are the lengths of two line figures, you can write the equation, $D2 = D1 * 0.50$, which has the correct dimensionality (length), but you cannot write $D2 = D1 * D1$, because this equation has dimensionality "length squared".

4.7.5 Changing Sketch Figure Dimensions

To change the value of an existing dimension:

- 1 Select the sketch **Select** tool from the Sketching toolbar.
- 2 Move the cursor over the dimension. The dimension is highlighted.
- 3 Double-click the dimension. The dimension control box appears displaying the current dimension value.
- 4 Enter a new dimension value in the dimension and press **Enter** on the keyboard. The dimension is updated and the figure reflects the new dimension.

4.7.6 Deleting Sketch Figure Dimensions

To delete an existing dimension:

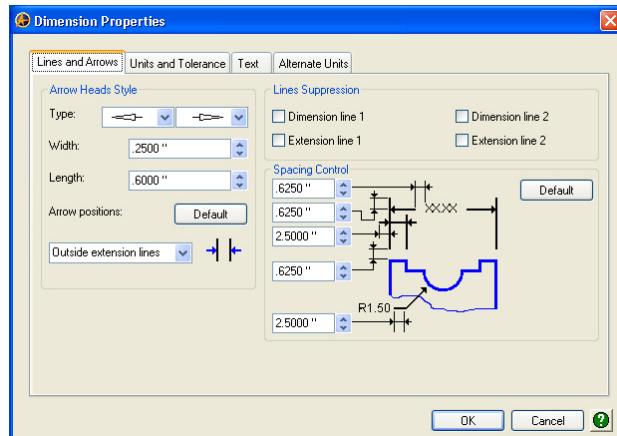
- 1 Select the sketch **Select**  tool from the Sketching toolbar.
- 2 Move the cursor over the dimension. The dimension is highlighted.
- 3 Select the dimension by clicking it.
- 4 Press the **Delete** key on the keyboard
Or . . . right-click and select **Delete** from the pop-up menu
Or . . . from the **Edit** menu, select **Delete**
Or . . . select the **Delete**  tool from the Standard toolbar

4.7.7 Modifying Sketch Dimension Properties

You can change individual dimension properties such as dimension line size and style, dimension value format and precision, dimension text size and orientation, dual dimension display, and tolerance information.

To modify sketch dimension properties:

- 1 Select either the sketch **Select**  tool or the **Dimension**  tool from the Sketching toolbar.
- 2 Move the cursor over the dimension, right-click, and select **Properties** from the pop-up menu. The **Dimension Properties** dialog box appears.



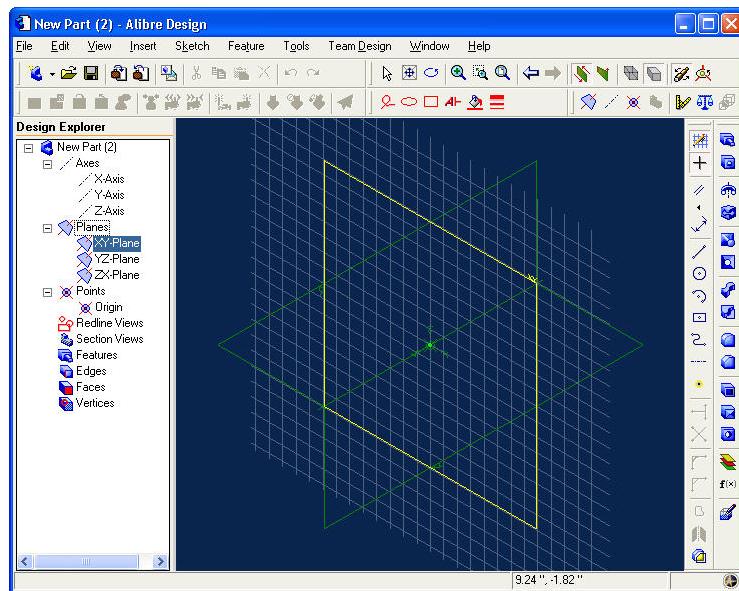
- 3 Select one of the four tabs: **Lines and Arrows**, **Units and Tolerance**, **Text**, and **Alternate Units**.
- 4 Select any changes or modifications to the dimension text and line settings.
- 5 Click **OK** to apply the changes.

4.8 Working in a Sketch

By default, a new part workspace contains three reference planes: **XY**, **YZ**, and **ZX** (Refer to Chapter 6 – Reference Geometry for details related to inserting additional reference planes). The default sketch plane is the **XY-Plane**; however, any reference plane can be used as the sketch plane. To select a different reference plane to be used as the sketch plane, simply select it either in the work area or the Design Explorer.

4.8.1 The Sketch Grid

Upon entering a sketch, by default a sketch grid will be displayed that can be used as reference during sketching.



The grid acts as an additional reference during sketching. You can customize the grid spacing as well as choose to automatically snap to grid during sketching (snap to grid is not on by default). Both grid display and snap to grid are optional and can be turned on or off at any time.

To turn the grid off:

- 1 From the **Tools** menu, select **Options**. The Options dialogue box appears.
- 2 Select the **Grid** tab.
- 3 Click in the **Display grid** check box to turn the grid off.
- 4 Click **OK** in the Options dialogue box.

To turn snap to grid on (only applies if the grid is displayed):

- 1 From the **Tools** menu, select **Options**. The Options dialogue box appears.
- 2 Select the **Grid** tab.
- 3 Click in the **Snap to grid** check box to turn the snap to grid on.
- 4 Click **OK** in the Options dialogue box.

4.8.2 Snapping to the Working Plane

As mentioned previously, after entering sketch mode the work area view will automatically re-orient so that it is parallel to the screen and normal to your view.

To turn Snap to the Working Plane off:

- 1 From the **Tools** menu, select **Options**.
- 2 Select the **General** tab if it is not already selected.
- 3 In the **Design Options** area, unselect the **Snap to working plane** option.
- 4 Click **OK** in the Options dialog box.

After turning Snap to Working Plane off, the work area view will remain in its current orientation as you activate a sketch.

To reorient the view back to the sketch plane:

While you are in sketch mode, you can reorient the view back to the sketch plane at any time.

From the **View** menu, select **Orient > To Sketch Plane**; or select the **To Sketch Plane** tool  from the Orient View Toolbar.

To reorient the view to the isometric of the sketch plane:

While you are in sketch mode, you can reorient the view to the isometric of the sketch plane at any time.

- 1 From the **View** menu, select **Orient > Isometric To Sketch Plane**; or select the **Isometric to Sketch Plane** tool from the Orient View Toolbar .

4.8.3 Cursor Dimension Hints

As you sketch new figures, dimensional properties are by default displayed near the cursor. For example, as you sketch a line, the line length and angle are displayed and updated automatically as you move the cursor. You can hide the cursor dimension hints if desired.

To hide the cursor dimension hints:

- 1 From the **Tools** menu, select **Options**.
- 2 Select the **General** tab if not already selected.
- 3 In the **Hints** area, unselect the **Cursor hints** option.
- 4 Click **OK**.

4.8.4 Cursor Display

For the most part during sketching, the cursor's appearance will change depending on which sketch tool is selected as well as the position of the cursor.

- The default cursor appears as  when the sketch **Select**  tool is selected.
- When another sketch tool is selected, the cursor will change to indicate the tool's function. For example, when the circle tool is selected, the cursor  appears as .
- The cursor also changes automatically depending on its position over an existing figure.



Node on Figure



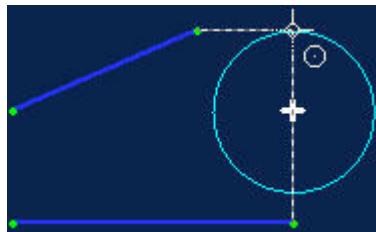
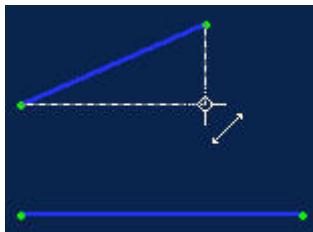
On the Figure



Midpoint of Figure

4.8.5 Inference Lines

During sketching, inference lines are displayed to provide a visual aid for aligning nodes. Inference lines appear as dashed lines and are automatically generated when your cursor is vertically or horizontally aligned with existing nodes or points, including the origin.



4.8.6 Direct Coordinate Entry

You can enter Cartesian and/or Polar coordinates while sketching to define start/endpoints, center points, and angular and radial values.

To use direct coordinate entry:

- 1 Select any sketch figure tool.
- 2 Right-click in the work area and select **Direct Coordinate Entry** from the pop-up menu,

OR from the **Sketch** menu, select **Direct Coordinate Entry**.



OR click the Direct Coordinate Entry icon from the toolbar. The **Direct Coordinate Entry** dialog box appears.

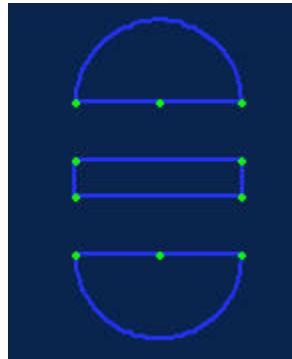
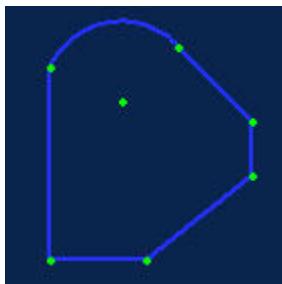
- 3 Select either the **Cartesian** or **Polar** tabs.
- 4 Select either the **Absolute** or **Relative** options. The **Absolute** option will define all nodes with respect to the origin at (0,0,0). The **Relative** option will define all new nodes with respect to the last node entered.
- 5 Enter the Cartesian or Polar coordinates depending on which system is being used.
- 6 Click **Set** to define a node.
- 7 Click **Close** when finished.

4.8.7 Right-click Menu

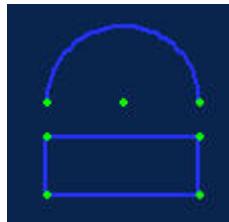
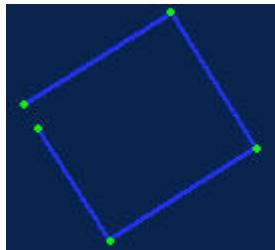
During sketching, a number of sketch tools are available from a menu that is quickly accessed by a right-click in the work area. Utilizing the right-click menu often provides the most efficient method in selecting a tool.

4.8.8 Open and Closed Sketches

Sketches define the profile of a 3D feature. Consequently, the majority of sketches are required to be closed before a 3D feature can be created. A closed sketch contains no open-ended figures. Closed sketches are required for all **Boss** and **Cut** features (refer to **Chapter 7** for more information related to features). The sketches illustrated below are examples of closed sketches.



An open sketch contains open-ended figures. Open sketches can only be used in **Thin Wall Boss** and **Thin Wall Cut** features (refer to **Chapter 7** for more information related to features). The sketches illustrated below are examples of open sketches.

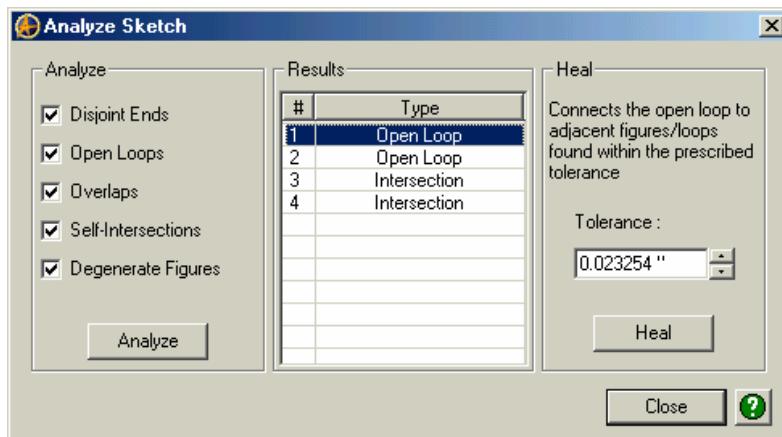


4.8.9 Checking Sketches for Open Ends, Intersections, Overlaps

For complicated sketches involving many figures and nodes, it may be helpful to check for open ends, intersections, and overlaps before creating a feature. Checking for these is also a valuable troubleshooting tool to resolve sketch problems.

To check a sketch for open ends:

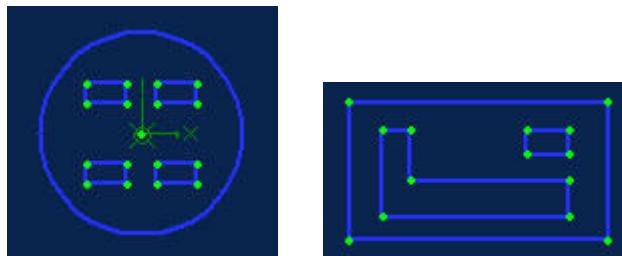
- 1 Select the sketch **Select**  tool from the Sketching toolbar.
- 2 Select either the entire sketch by dragging a selection rectangle over the desired sketch figures or select the appropriate sketch figures one at a time. The figures are highlighted after selection.
- 3 From the **Sketch** menu, select **Analyze**. The Analyze Sketch dialog box appears.



- 4 In **Analyze**, check the items you would like to search for; then click **Analyze**. The sketch errors will appear in the **Results** area of the dialog box.
- 5 Click the result you want to view, and the area will highlight in the part workspace. In some instances, you can use the **Heal** option to resolve the sketch.
- 6 Select the **Tolerance** option in the **Heal** area of the dialog box.
- 7 Enter a **Tolerance** value that is larger than the existing gap distance between the open nodes.
- 8 Once the Tolerance value is set, the **Heal** button will become active. Click the **Heal** button to resolve the figure.

4.8.10 Enclosed Figures

A simple way to reduce steps when modeling a part is to create a sketch with enclosed figures. Enclosed figures are figures that are sketched within the profile of another figure. Material will be removed from the enclosed figure profile when a feature is created (refer to Chapter 7 for more details). The sketches illustrated below contain enclosed figures.



4.8.11 Copying and Pasting Sketch Figures

You can cut, copy, and paste entire sketches or individual sketch figures within the same sketch or into new sketches altogether.

To copy and paste sketch figures within the same sketch:

- 1 Select the sketch **Select** tool from the Sketching toolbar.
- 2 Select the figures to be copied. The figures become highlighted.
- 3 From the **Edit** menu, select **Copy**, or press **Ctrl + C** on the keyboard.
- 4 From the **Edit** menu, select **Paste**, or press **Ctrl + V** on the keyboard, or right-click and select **Paste** from the pop-up menu.
- 5 The copied figure(s) are placed slightly offset from the originating figures.

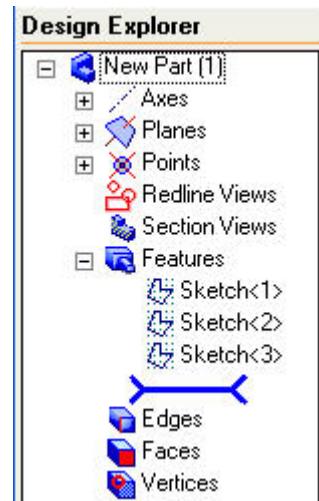
To copy and paste sketch figures into a new sketch:

- 1 Select the sketch **Select** tool from the Sketching toolbar.

- 2 Select the figures to be copied. If you are copying an entire sketch, from the **Edit** menu, select **Select All**, or press **Ctrl + A** on the keyboard. The figures become highlighted.
- 3 From the **Edit** menu, select **Copy**, or press **Ctrl + C** on the keyboard.
- 4 Exit sketch mode, select the new sketch plane, and enter sketch mode.
- 5 From the **Edit** menu, select **Paste**, or press **Ctrl + V** on the keyboard, or right-click and select **Paste** from the pop-up menu. The figures are pasted into the new sketch in the same orientation with respect to the origin as the originating sketch.

4.9 Sketches and the Design Explorer

After creating a sketch, and subsequently exiting sketch mode, sketches will be listed in the Design Explorer under the **Features** node in the order they were created.



4.9.1 Editing Sketches

Sketches can be edited at anytime.

To edit a sketch:

- 1 In the Design Explorer, right-click the sketch name and select **Edit** from the pop-up menu, or double-click the sketch name. The sketch appears in sketch mode.
- 2 Make the appropriate modifications to the sketch and exit sketch mode to apply the changes.

4.9.2 Renaming Sketches

Sketches are listed in the Design Explorer by default as **Sketch<1>**, **Sketch<2>**, etc. You can rename the sketches to provide relevant information.

To rename a sketch:

- 1 In the Design Explorer, right-click the sketch name and select **Rename** from the pop-up menu, or double-click the sketch name with a pause between the first and second click. The sketch name is highlighted and can be changed.
- 2 Type in the new sketch name.
- 3 Press **Enter** on the keyboard.

4.9.3 Deleting Sketches

You can delete a sketch as long as it has not been used to create a feature. To delete a sketch that has an associated feature, you must first delete the feature.

To delete a sketch:

- 1 In the Design Explorer, right-click the sketch name and select **Delete** from the pop-up menu, or select the sketch and press **Delete** on the keyboard.

5 3D Sketching

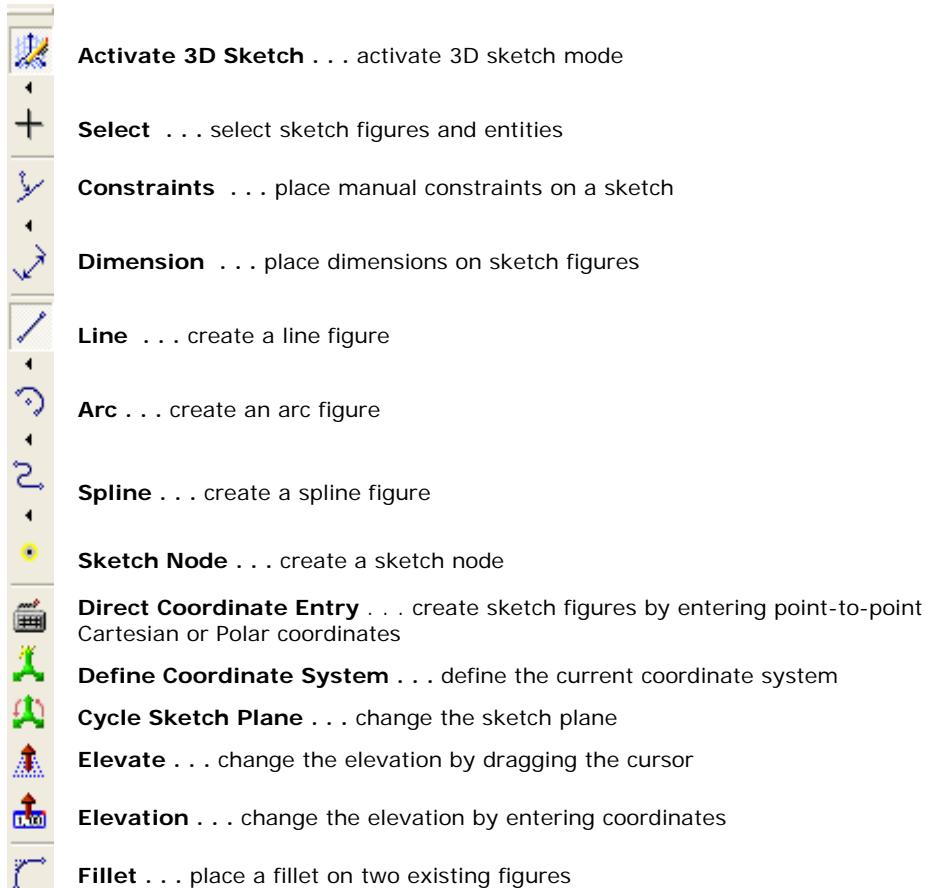
3D Sketching allows you to create guide curves for better control of lofts. In addition, using 3D sketches allows you to create sweeps that are ideal for modeling piping and cabling systems.

This chapter describes:

- The 3D Sketching interface
- The Current Coordinate System
- The Sketch Plane and Elevation
- Entering and Exiting Sketch Mode
- Creating sketch figures
- Dimensioning sketch figures
- Adding and deleting sketch constraints

5.1 The 3D Sketching Interface

The 3D Sketching toolbar is shown by default on the right side of the workspace. Commonly used sketch tools are accessible on the Sketching toolbar.



All of the tools accessible on the 3D Sketching toolbar are also accessible from the **3D Sketch** menu.

5.1.1 3D Sketching Context

3D sketching takes place in a separate environment from part modeling, just as 2D sketching. The 3D sketching environment has a dedicated toolbar and right-click menu, which allow you to access the 3D sketching functions.

You can modify the display of various 3D sketch items from the view menu.

To modify the display of sketch items:

- 1 From the **View** menu, select **Sketch Display**. The following items can be turned on or off in the display:
 - Grid
 - Sketch Dimensions
 - Constraint Symbols
 - Guide Lines
 - Current Coordinate System Indicator

Select an item to turn it on or off. A checkmark next to an item means it is visible (on). This is a toggle on/off.

To modify the view orientation:

From any orientation back to the sketch plane:

- 1 From the **View** menu, select **Orient > To Sketch Plane**; or select the To Sketch Plane tool  from the Orient View Toolbar.

From any orientation to the isometric view of the sketch plane:

- 1 From the **View** menu, select **Orient > Isometric To Sketch Plane**; or select the Isometric To Sketch Plane tool  from the Orient View Toolbar.

5.1.2 Current Coordinate System

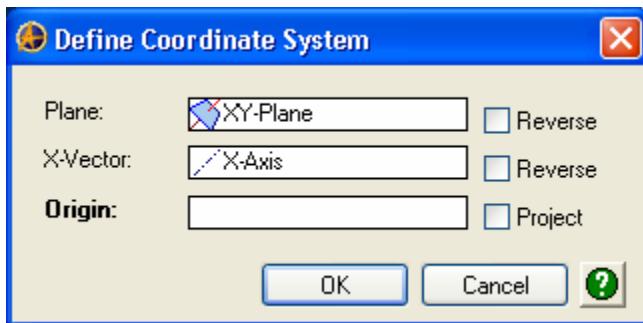
The Current Coordinate System

In the 3D sketching environment, all position-related data is entered with respect to the Current Coordinate System (CCS). The CCS is depicted graphically by a 3D coordinate system in the workspace. This graphical coordinate system is called the CCS indicator. Upon entering 3D sketch mode, the CCS is automatically created using the active plane, which becomes the XY-Plane of the CCS. The direction of the axes is determined automatically by the system.

To Define A New 3D Coordinate System:

The CCS can be changed using the Define 3D-Coordinate System command.

- 1 Select the **Define Coordinate System** tool from the Sketching toolbar; or from the **3D Sketch** menu, select **Define Coordinate System**. The Define Coordinate System dialog box appears.



Enter the following information:

Plane: Required – Enter the desired reference plane, planar face, or 2D sketch to be used as the plane. Checking **Reverse** will toggle which **Z-axis** direction is positive.

X-Vector: Optional – Enter the reference axis, linear edge, 2D sketch line, or 3D sketch line to be used for the X-Vector. Checking **Reverse** will toggle which **X-axis** direction is positive.

Origin: Optional – Enter a reference point, vertex, 2D node, or 3D node to be used as the origin. If **Project** is checked, the point is projected to the plane to become the origin; otherwise the plane is moved to intersect the point that becomes the origin.

Click **OK** to apply the changes.

To turn off the CCS indicator:

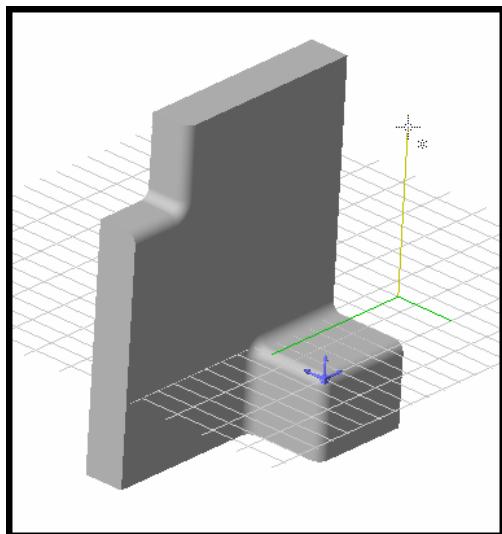
- 1 From the **View** menu, select **Sketch Display > Current Coordinate System Indicator**. This is an on/off toggle.

5.1.3 Sketch Plane, Guide Lines, and Elevation

The sketch plane is defined by any of the planes of the CCS and an elevation. The sketch plane can be offset along its normal by an elevation distance.

By default, the sketch plane is the XY plane of the CCS. The sketch plane can be altered by using the Cycle Sketch Plane Command.

Coordinate guides are displayed while in 3D sketch to indicate the current location of the cursor with respect to the CCS. Guides for the sketch plane are displayed on the sketch plane. A guide extending from the base plane to the current cursor position indicates the current elevation.



Coordinate Guides Shown with the Sketch Plane Grid

To change the sketch plane:

- 1 Select the **Cycle Sketch Plane** tool  from the Sketching toolbar; or from the **3D Sketch** menu, select **Cycle Sketch Plane**.

The sketch plane cycles to the next primary plane each time the tool is clicked. If the grid is turned on, it is displayed on the current base plane.

Note: This command can be issued during other commands.

Note: You can also cycle the sketch plane by pressing the Tab or "F" key on the keyboard. If you press one of these keys while in a figure creation command, the cursor will remain in the same position in space, rather than moving to an elevation of zero on the new sketch plane.

Elevation is controlled in two ways. The first method is to drag the mouse while in elevate mode. The second method is to use the Elevation Dialog.

To control elevation in Elevate mode:

- 1 Select the **Elevate**  tool from the sketching toolbar; or from the **3D Sketch** menu, select **Elevate**. This puts you into Elevate mode.

While in Elevate mode, the coordinates on the base plane remain constant.

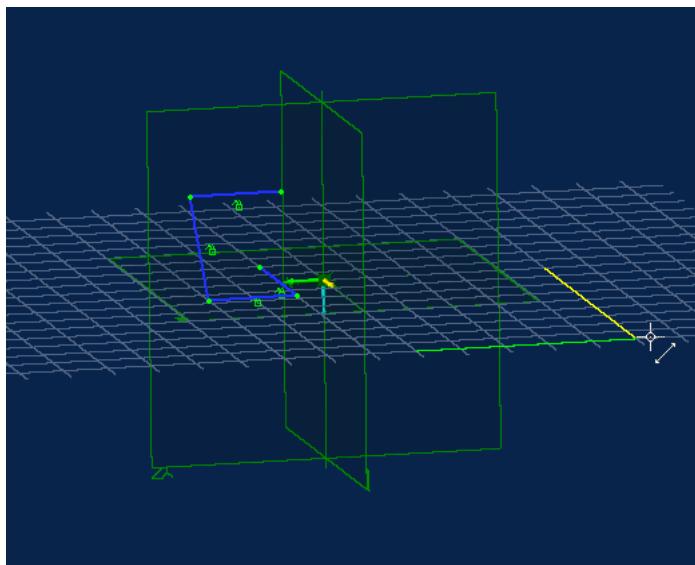
Click and drag the cursor to the desired height from the base plane.

To exit Elevate mode, select the **Elevate** tool again from the sketching toolbar, or from the **3D Sketch** menu, select **Elevate**.

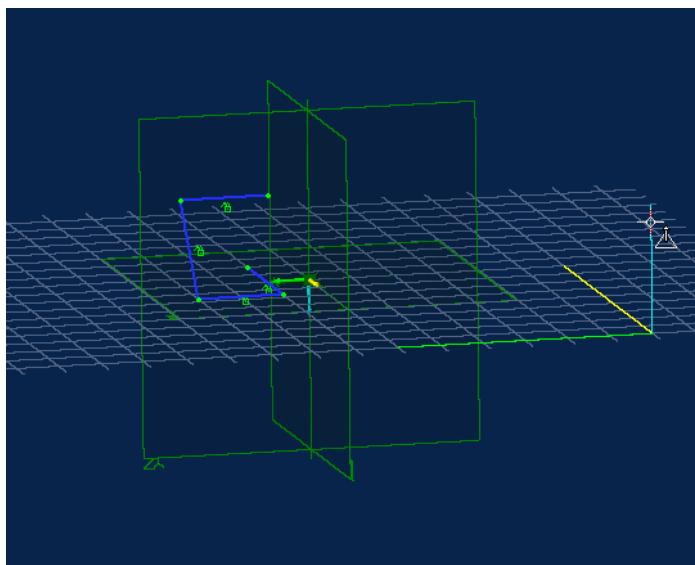
Note: To change the elevation of an existing figure, click and drag the figure while in Elevate mode.

Note: Elevate can be issued during other commands.

Note: While in a figure creation command, Elevate can be accessed at anytime by pressing the "E" key on the keyboard and holding it down while placing the figure.



Elevation height at the base plane



Elevation height changed by dragging cursor

To control elevation using Elevation Dialog:

- 1 Select the **Elevation**  tool from the sketching toolbar; or from the **3D Sketch** menu, select **Elevation**. The Elevation dialog box appears.
- 2 Enter the desired elevation in the dialog box. This dialog can remain open and in use during other commands.
- 3 Click the X in the upper corner of the dialog box to close when finished.

5.2 Entering and Exiting 3D Sketch Mode

5.2.1 Entering 3D Sketch Mode

You must enter 3D sketch mode before you can begin sketching.

To enter 3D sketch mode:



- Select the **Activate 3D Sketch**  tool from the Sketching toolbar.
- Or
- From the **3D Sketch** menu, select **Activate 3D Sketch**.

The activate 3D Sketch tool on the Sketching toolbar will always appear in the active state  while in 3D sketch mode.

5.2.2 Exiting 3D Sketch Mode

The same methods used to enter 3D sketch mode can be used to exit 3D sketch mode.

To exit 3D sketch mode:

- Select the **Select**  tool from the View toolbar.

Or

- Create a feature from the sketched profile. For example, select a feature tool such as the **Extrude Sweep** tool from the Part Modeling toolbar.

Or

- Select the **Regenerate**  tool from the Part Modeling toolbar, or from the **Feature** menu, select **Regenerate All**.

5.3 3D Sketch Figures

5.3.1 Line

To sketch a line:

- 1 Select the **Line** tool from the Sketching toolbar; or from the **3D Sketch** menu, select **Figures > Line**; or right-click and select **Line** from the pop-up menu.

Position the cursor at the location you want to start the line.

Click to start the line and drag the cursor to sketch the line.

Click again to complete the line segment. You can continue to sketch additional line segments by clicking. Double-click or press **ESC** on the keyboard to complete the line.

5.3.2 Arc

You can sketch three different circular arc types: 1) **Center, Start, End**; 2) **Start, End, Radius**; 3) **Tangent-Start, End**.

To sketch a circular arc using Center, Start, End:

- 1 Select the **Circular Arc – Center, Start, End**  tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Circular Arc > Center, Start, End**; or right-click and select **Circular Arc** from the pop-up menu.

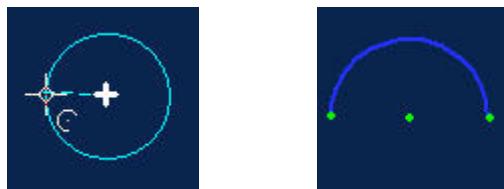
Click in the Work Area to place the center of the arc.

Click a second time to start the arc.

Move the cursor to sketch the arc.

Click a third time to complete the arc.

Note: The plane of the arc is defined by the three nodes. If the three nodes are collinear, the current sketch plane is used.



To sketch a circular arc using Start, End, Radius:

- 1 Select the **Circular Arc – Start, End, Radius** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Circular Arc > Start, End, Radius**.

Position the cursor at the arc starting location.

Click to start the arc.

Click a second time to locate the end of the arc.

Move the cursor to size the arc.

Click a third time to complete the arc.

Note: The plane of the arc is defined by the three nodes.

To sketch a circular arc using Tangent-Start, End:

- 1 Select the **Circular Arc – Tangent-Start, End** tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Circular Arc > Tangent-Start, End**.

Click a line or circular arc.

Move the cursor to size the arc.

Click a second time to complete the arc.

Note: The plane of the arc is defined by the tangent line passing through the nodes at the start and end of the arc.

To modify a circular arc:

See section 4.3.3 for detailed information on modifying the diameter, shape, and location of circular arcs.

5.3.3 Spline

Creation of NURBS curves by interpolation

Using this method, specify a set of nodes in the Work Area to define the spline. A curve is then interpolated based on the placement of the nodes.

- 1 Select the **Spline**  tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Spline**.

Click in the Work Area to start the spline curve.

Move the cursor and click a second time to place an interpolation node.

Move the cursor to shape the curve.

Continue clicking to place additional nodes and curve segments.

Note: One or more interpolation nodes may be specified via the direct coordinate entry tool.

Double-click or hit escape to complete the spline curve.

Note: If you place the final node at the same location as the first, the spline will be completed as a closed spline.

Editing a Spline Curve

In 3D Sketch, you can edit spline curves by selecting and dragging any of the nodes in the curve.

Inserting a Node Into a Spline Curve

- 1 From the **3D Sketch** menu, select **Insert Node into Spline**; or select the **Insert Node into Spline**  tool from the sketching toolbar.
- 2 Click the spline to place a node.
- 3 Continue clicking the spline to place as many nodes as desired.
- 4 Select the **Select**  tool to exit the Insert Node command.

5.4 3D Sketch Nodes

5.4.1 Placing a Sketch Node

To place a sketch node:

- 1 Select the **Sketch Node**  tool from the Sketching toolbar; or from the **3D Sketch** menu select **Figures > Node**.

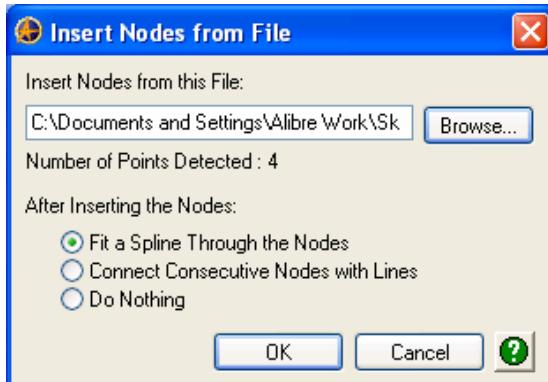
Click to place a sketch node.

5.4.2 Inserting Sketch Nodes From A File

You can insert sketch nodes from a comma delimited text file. These nodes can then be used to create features such as Sweeps.

To insert sketch nodes from a file:

- 1 From the **3D Sketch** menu, select **Figures > Insert from File**. The **Insert Nodes from File** dialog box appears.



Enter the required information:

File Name: Type in the file name or use the Browse button to designate the file containing the nodes.

Choose one of the following:

Fit a Spline Through the Nodes — the system will interpolate a spline through the nodes.

Connect Consecutive Nodes with Lines — each node will be connected by a straight line.

Do Nothing — the nodes will be placed in the sketch without connecting figures.

Click **OK** to insert the nodes.

5.5 Working with Existing 3D Sketch Figures

5.5.1 Adding Fillets

To add a fillet to a sketch figure:

- 1 Select the **Fillet**  tool from the Sketching toolbar; or from the **3D Sketch** menu select **Fillet**. The **Fillet Figures** dialog box appears.

Select the first figure by clicking it. The figure name appears in the **Figures to Fillet** area in the dialog box.

Select the second figure by clicking it. The figure name appears in the **Figures to Fillet** area in the dialog box.

Enter the fillet radius value in the **Radius** box.

Click **Apply** to create the fillet.

The Fillet Figures dialog box remains open so you can continue to place fillets on other figures.

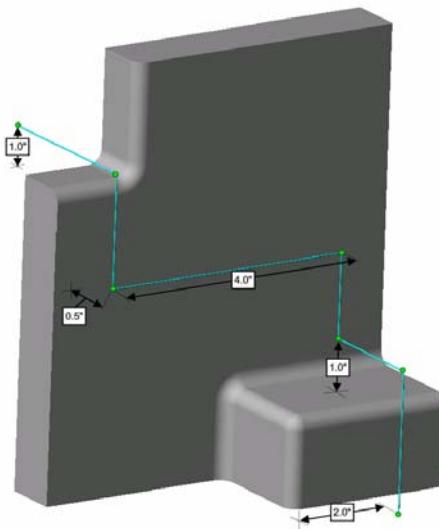
Note: Consecutive fillets with a diameter equal to the first fillet will not be dimensioned; instead an equal constraint will automatically be placed.

Click **Close** to close the Fillet Figures dialog box.

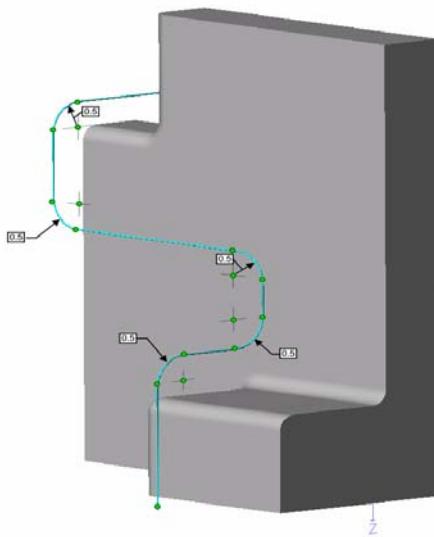
5.6 Dimensioning 3D Sketch Figures

To fully define the 3D sketch you must place dimensions on sketch figures. 3D sketch dimensions function similar to 2D sketch dimensions. It is not required to dimension sketches before they are used to create features. In addition, sketch dimensions can easily be changed and modified at any time.

Linear, radial, and angular dimensions can be created. Linear dimensions are placed on a plane running through an axis created by the endpoints and parallel with the screen at the time of creation. The dimension will always lie on this plane from that time forward. The dimension can be dragged along this plane from place to place. Radial dimensions may be created for circular arcs. Angular dimensions may be created between two non-parallel lines.



Examples of 3D Linear Dimensions



Examples of 3D Radius Dimensions

To dimension individual figures:

Follow the steps below to place length dimensions on lines and diameter dimensions on arcs.

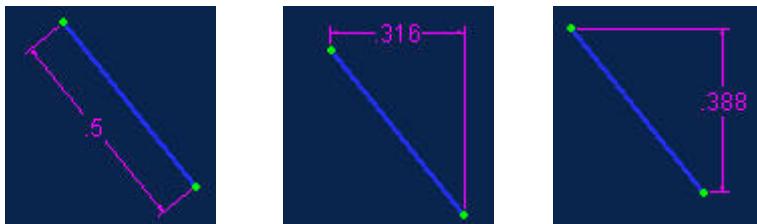
- 1 Select the **Dimension**  tool from the Sketching toolbar; or from the **3D Sketch** menu, select **Dimension**; or right-click in the work area and select **Dimension** from the pop-up menu.

Move the cursor over the figure you want to dimension. The figure is highlighted.

Click the figure to show a preview of the dimension. Move the cursor to move the dimension preview.

Note: You can press the **Esc** key on the keyboard to cancel the current dimension operation.

Depending on the type of figure you are dimensioning, as well as where you move the preview dimension, a new dimension may be inferred. For example, depending on where the cursor is moved, three different dimensions could be placed on an angled line.



After the dimension has been positioned properly, click again. A dimension control box appears.



Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.

Note: You can enter fractions (e.g. **3/8**) and simple equations (e.g. **1.5 * 3**) into the dimension control box. You can also enter a value with units other than the current display units (e.g. **5 mm**). The value will be converted to the display units automatically. Supported unit abbreviations are **”, ‘, mm, cm, and m**.

To dimension distances or angles between figures:

Follow the steps below to place distance or angular dimensions between figures, e.g. between lines, between nodes, between arc or circle center nodes, etc.

- 1 Select the **Dimension**  tool from the Sketching toolbar; or from the **3D Sketch** menu, select **Dimension**; or right-click in the work area and select **Dimension** from the pop-up menu.

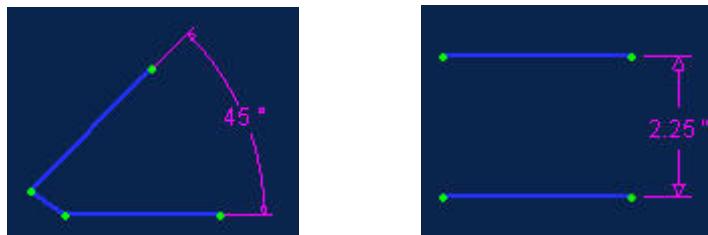
Move the cursor over the first figure you want to dimension from.

Click the figure; a preview may appear but do not place the dimension at this time.

Move the cursor over the second figure and click again. A new dimension preview appears.

Move the cursor to position the dimension and click a third time. A dimension control box appears.

Enter the appropriate dimension value in the box and press **Enter** on the keyboard. The dimension is defined.



5.7 3D Sketch Constraints

Figures in a 3D sketch may be constrained to a size, orientation, and relationship to another figure or model edge. These constraints are similar to their 2D sketch constraint counterparts.

5.7.1 Inferred Constraints

Location

During the creation of a figure a Location constraint will be applied automatically when a 3D node is created while hovering over the following objects:

- Reference Points
- Vertices
- An Existing 3D Node
- An Edge
- 2D Nodes (in visible 2D sketches that have not been used to create 3D geometry)
- 2D Figures (in visible 2D Sketch that have not been used to create 3D geometry)

This constraint is associative. If the object associated with the constraint moves, the corresponding 3D node moves with it. Inferred constraints can be broken by holding the CTRL key; then clicking and dragging the node away.

Fixed Direction

During the creation of lines that are horizontal or vertical with respect to the sketch plane, an implied fixed direction constraint will be placed on them.

5.7.2 Explicit Constraints

Explicit constraints are applied in a fashion analogous to their 2D equivalents through the use of a toolbar button or menu selection. The Constraints fly-out toolbar is available from the Sketching toolbar.



Coincident — A node can be constrained so that it lies on a figure, model edge, or planar face.



Fixed — Figures may be constrained to a fixed position in the sketch. After the constraint is applied, the node or figure may not be moved without first deleting the constraint.



Fixed Direction — The direction of a 3D line can be constrained so that it is held constant.



Parallel — Lines can be constrained to be parallel to each other. In addition, a line can be constrained to be parallel to a linear edge, a reference plane, or a planar face. For planes and faces, the constraint indicates that the line is perpendicular to the normal of the plane or face.



Perpendicular — Lines can be constrained to be perpendicular to other lines, reference planes, reference axes or planar faces. Splines can be constrained to reference planes and planar faces.

The behavior of lines and splines differs: Lines will be parallel to the normal of a plane or face. Splines will be set tangent at the start or end of the spline parallel to the normal of a plane or face.



Tangent — Figures can be constrained to be tangent to a curve. Tangent constraints can be applied automatically during sketching or placed manually after a figure has been sketched.



Tangent Continuous — An open 3D figure can be constrained to be continuous at its endpoint to another open 3D figure or an open edge.



Collinear — Figures may be constrained so that they lie in the same line.

5.8 Other 3D Sketch Functions

Other sketch options are similar to 2D sketch options. Please see sections 4.8 and 4.9 for information on the following:

The sketch grid

Snapping to the working plane

Cursor display

Editing sketches

Renaming sketches

Deleting sketches

6 Reference Geometry

Reference geometry consists of planes, axes, points, and surfaces, which are primarily used for feature construction aids. Reference planes serve as the default sketch planes. Axes are fundamental to creating features such as revolutions and patterns. The primary reference point in a workspace is the origin and is used extensively as a guide. Additional reference geometry can be added as necessary.

This chapter describes:

- Inserting new reference planes, axes, points, and surfaces
- Editing reference geometry properties
- Displaying reference geometry
- Deleting reference geometry
- Renaming reference geometry items in the Design Explorer

6.1 Reference Planes

By default, three reference planes are visible in a part and assembly workspace, the **XY-plane**, **YZ-plane**, and **ZX-plane**; any can be used as the sketch plane. Additional reference planes can be inserted in any orientation and also used as the sketch plane. You can modify the display options of the reference planes.

To modify plane display options:

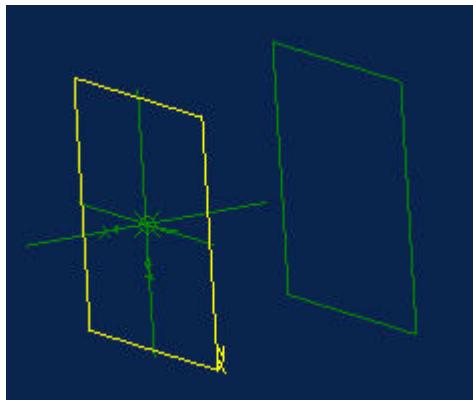
- 1 From the **Tools** menu, select **Options, General Tab**
- 2 Check **Shade reference planes** to see the reference planes slightly shaded in the workspace. Uncheck the box if you do not wish to see the shading.
- 3 Check **Show reference plane normals** to see a 3D arrow designating the direction of a reference plane's normal when you place the cursor over it. Uncheck the box if you do not wish to see the designating arrows.

6.1.1 Offset Plane

You can create a new reference plane parallel to an existing reference plane or planar face offset by a specified distance.

To create an offset plane:

- 1 Select the **Insert Plane**  tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog box appears.
- 2 Select the existing reference plane or planar face to offset.
- 3 Enter the offset **Distance** value. A preview of the new plane is displayed.
- 4 If necessary, select **Reverse** to create the plane in the opposite direction.
- 5 Click **OK** to create the plane.

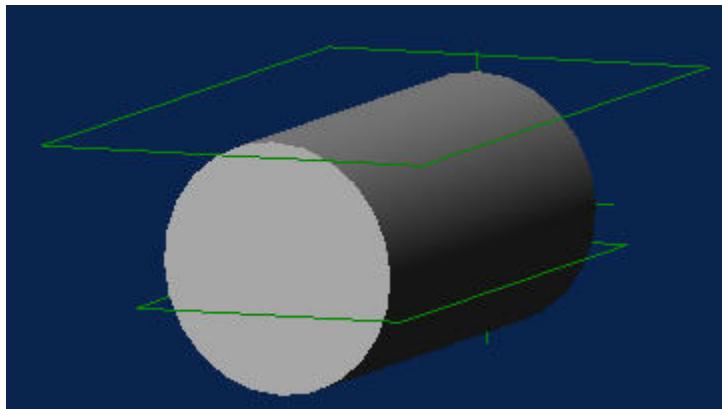


6.1.2 Tangent Plane

You can create a new reference plane parallel to an existing reference plane or planar face and tangent to an existing cylindrical face.

To create a tangent plane:

- 1 Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog box appears.
- 2 Select the existing reference plane or planar face to offset.
- 3 Select the cylindrical face that the new plane will be tangent to. A preview of the new plane is displayed.
- 4 Check the **Reverse** option to choose the alternative tangent point.
- 5 If desired, you can select the **Symmetry Axis** option to create the plane parallel to the original plane or face and through the axis of the cylindrical face.
- 6 Click **OK** to create the plane.



6.1.3 Angled Plane

You can create a new plane through an edge or axis at an angle to an existing reference plane or planar face.

To create an angled plane:

- 1 Select the **Insert Plane** tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog box appears.
- 2 Select an existing edge or axis.
- 3 Select an existing reference plane or planar face. A preview of the new plane is displayed.
- 4 Enter the **Angle** value.
- 5 If necessary, select **Reverse** to create the plane in the opposite direction.
- 6 Click **OK** to create the plane.

6.1.4 Parallel Plane Through a Point

You can create a plane parallel to an existing reference plane or planar face through an existing point.

To create a parallel plane through a point:

- 1 Select the **Insert Plane**  tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog box appears.
- 2 Select an existing reference plane or planar face.
- 3 Select an existing reference point. A preview of the new plane is displayed.
- 4 Click **OK** to create the plane.

6.1.5 Plane at Line and Point

You can create a plane through an edge or axis and a point or vertex.

To create a plane through a line and point:

- 1 Select the **Insert Plane**  tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog box appears.
- 2 Select an existing axis or edge.
- 3 Select an existing reference point or vertex. A preview of the new plane is displayed.
- 4 Select the **Containing Edge/Axis** option to create the plane through both the point and the axis/edge.

Select the **Normal to Edge/Axis** option to create the plane through the point and normal to the axis/edge.

- 5 Click **OK** to create the plane.

6.1.6 Three Point Plane

You can create a plane through three points or vertices.

To create a plane through three points:

- 1 Select the **Insert Plane**  tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog box appears.
- 2 Select an existing point or vertex.
- 3 Select a second point or vertex.
- 4 Select the third point or vertex. A preview of the new plane is displayed.
- 5 Click **OK** to create the plane.

6.1.7 Plane Normal to 3D Sketch or 3D Edge

You can create a plane normal to an open 3D sketch or normal to an edge of a solid model.

To create a plane normal to a 3D sketch or edge:

- 1 Select the **Insert Plane**  tool from the Inspection toolbar; or from the **Insert** menu, select **Plane**; or right-click in the work area and select **Insert Plane** from the pop-up menu. The **Insert Plane** dialog box appears.
- 2 Select an existing open 3D sketch or model edge. A preview of the new plane is displayed.
- 3 Check the **Other End** box to move the plane to the other end of the sketch or edge.
- 4 Click **OK** to create the plane.

6.2 Axes

Three axes are visible by default in part and assembly workspaces, the **X-axis**, **Y-axis**, and **Z-axis**. Additional axes can be inserted as needed.

6.2.1 Axis Through Axis or Edge

You can create an axis through an existing axis or linear edge.

To create an axis through an edge:

- 1 Select the **Insert Axis**  tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog box appears.
- 2 Select an existing axis or edge. A preview of the new axis is displayed.
- 3 Click **OK** to create the axis.

6.2.2 Axis Through Two Points

You can create an axis through two points or vertices. This method can be used to create an axis at an angle.

To create an axis through two points:

- 1 Select the **Insert Axis**  tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog box appears.
- 2 Select an existing point or vertex.
- 3 Select the second point or vertex. A preview of the new axis is displayed.
- 4 Click **OK** to create the axis.

6.2.3 Axis Using Cylindrical Face

You can create an axis using a cylindrical face as reference.

To create an axis using a cylindrical face:

- 1 Select the **Insert Axis**  tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog box appears.
- 2 Select an existing cylindrical face. A preview of the new axis is displayed.

- 3 Click **OK** to create the axis.

6.2.4 Axis Through Two Planes

You can create an axis at the intersection of two existing reference planes or planar faces.

To create an axis through two planes:

- 1 Select the **Insert Axis**  tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog box appears.
- 2 Select an existing reference plane or planar face.
- 3 Select a reference plane or planar face that intersects the first plane or face. A preview of the new axis is displayed.
- 4 Click **OK** to create the axis.

6.2.5 Axis Offset and Parallel to Axis or Edge

You can create an axis that is parallel to an existing axis or edge and offset by a specified distance.

To create an axis offset and parallel to an edge:

- 1 Select the **Insert Axis**  tool from the Inspection toolbar; or from the **Insert** menu, select **Axis**; or right-click in the work area and select **Insert Axis** from the pop-up menu. The **Insert Axis** dialog box appears.
- 2 Select an existing edge or axis.
- 3 Select a reference plane or planar face that passes through the edge or axis selected in step 1. A preview of the new axis is displayed.
- 4 Enter an offset distance.
- 5 If necessary select **Reverse** to create the axis in the opposite direction.
- 6 Click **OK** to create the axis.

6.3 Points

In part and assembly workspaces, the origin is the only 3D point displayed by default. Additional 3D points can be inserted as necessary.

To insert a new point:

- 1 Select the **Insert Point**  tool from the Inspection toolbar; or from the **Insert** menu, select **Point**; or right-click in the work area and select **Insert Point** from the pop-up menu. The **Insert Point** dialog box appears.
- 2 Select the appropriate point type below and follow the steps accordingly.

6.3.1 Point at Specified Coordinates

You can create a point using direct coordinate entry.

To create a point using direct coordinate entry:

- 1 Enter the 3D coordinates of the new point. The coordinates are based on the absolute coordinate system with the origin located at (0,0,0).
- 2 Click **OK** to create the point.

6.3.2 Point at Plane and Axis/Edge

You can create a point at the intersection of plane or planar face and an axis or edge.

To create a point at a plane and edge:

- 1 Select a reference plane or planar face.
- 2 Select an axis or edge. A preview of the new point is displayed.
- 3 Click **OK** to create the point.

6.3.3 Point at Axis/Edge and Axis/Edge

You can create a point at the intersection of two axes or edges.

To create a point at the intersection of two edges:

- 1 Select an axis or edge.

- 2 Select a second axis or edge. A preview of the new point is displayed.
- 3 Click **OK** to create the point.

6.3.4 Point at the Center of Circular Edge

You can create a point at the center of a circular edge.

To create a point at the center of a circular edge:

- 1 Select a circular edge. A preview of the new point is displayed.
- 2 Click **OK** to create the point.

6.3.5 Point at Vertex

You can create a point at a vertex.

To create a point at a vertex:

- 1 Select the vertex. A preview of the new point is displayed.
- 2 Click **OK** to create the point.

6.3.6 Point Along Edge

You can create a point along an edge at a specified location.

To create a point along an edge:

- 1 Select the edge. A preview of the new point is displayed.
- 2 Enter a **Ratio** value. A ratio of .5 will place the point at the midpoint of the edge, 1.0 will place the point at the end of the edge.
- 3 Click **OK** to create the point.

6.3.7 Point Between Two Points

You can create a point between two points at a specified location.

To create a point between two points:

- 1 Select a point or vertex.
- 2 Select a second point or vertex.
- 3 Enter a **Ratio** value. A ratio of .5 will place the point at the midpoint between the two existing points or vertices.
- 4 Click **OK**.

6.4 Reference Surfaces

You can insert surfaces as reference geometry, similar to application planes, in order to trim or extend solids to the surfaces. In addition, you can thicken a reference surface into a solid.

6.4.1 Inserting Reference Surfaces

Surfaces can be inserted from IGES or SAT files only.

To insert a surface:

- 1 Open the part workspace that you want the surface inserted into.

From the **Insert** menu, select **Surfaces**; or select the **Insert Surfaces** tool  from the Inspection toolbar. The Insert Surface dialog box appears.

Select the file that you wish to insert; then click **Open**. The Insert Surface Options dialog box appears, if you have that option turned on.

Note: To turn on the option to see the Insert Surface dialog box, From the **Tools** menu, select **Options> Interoperability Tab**. In the Insert Options section, check the box beside Show Options When Inserting.

Select the desired options; then click **OK**.

- **Stitch Adjoining Faces** takes faces that meet at a common edge and places

them in the same surface body. Each resulting lump becomes a surface.

- **None** inserts the surfaces, as they exist in the file. Each lump is a surface. The body is unchanged.
- **Unstitch to Standalone Surfaces** converts each face into a separate surface.
- **Heal** is same as healing for import of solid. It will recalculate inaccurate geometry in order to make the part more accurate upon import. It cleans up the body by making sure edges lie on faces, eliminates duplicate vertices, etc. Healing attempts to fix problems detected with the model by changing it. Most IGES files require healing to import properly.
- **Make Tolerant** will tag inaccurate geometry to enable more intelligent subsequent operations after import. Because the geometry of a tolerant model is allowed to be less precise, inaccurate or leaky data can often be imported using the Make Tolerant option. Making a model tolerant leaves its underlying geometry unchanged.

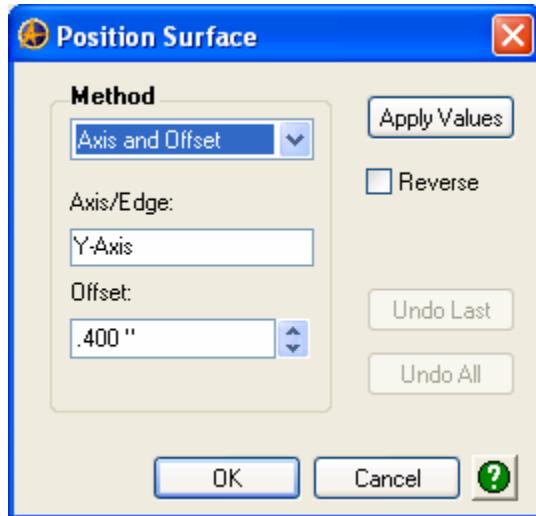
The surface is inserted into the workspace, and is listed in the Design Explorer under both Surfaces and Features.

6.4.2 Positioning Reference Surfaces

The default location of the inserted surface is such that its coordinate system matches the current part workspace coordinate system with the same origin, orientation, and scale. You can modify the location of the surface by positioning it.

To position a surface:

- 1 In the Design Explorer, right-click the surface you wish to position.
- 2 Select **Edit**. The **Position Surface** dialog box appears.



- 3 In **Method**, select the method you wish to use to position the surface.

Axis and Offset: The surface will be moved the offset distance in the direction of the specified axis or edge

Planar: Choose points or vertices to use for the start of the move and the end of the move. These points will be projected onto the chosen plane, and the surface will move parallel to the plane

Coordinates: Change the location of the surface by designating X, Y, and Z distances

Axis and Angle: Rotate the surface about the specified axis by the given angle

- 4 Fill in the appropriate data for the chosen method.

- 5 Click **OK** to apply the change.

6.4.3 Thickening Reference Surfaces

Once a reference surface has been inserted, you can thicken the surface to transform it into a solid. This will allow geometry such as holes and extruded cuts to be applied to the resulting solid. Only one surface can be thickened at a time.

To thicken a surface:

- 1 From the **Feature** menu, select **Thicken Surface**; or select the **Thicken Surface** tool  from the Part Modeling toolbar. The Thicken Surface dialog box appears.



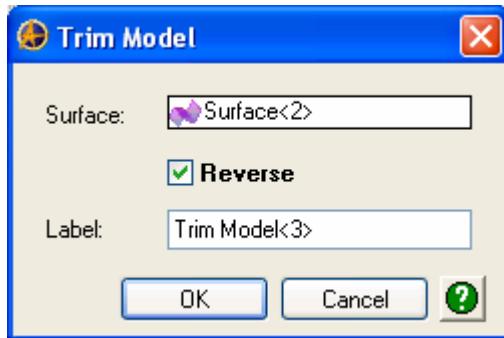
- 2 In **Surface**, select the surface you wish to thicken.
- 3 In **Thickness**, enter the desired thickness value.
- 4 In **Direction**, choose forward, reverse, or both sides. Arrows in the workspace will display the direction the thickness will be applied.
- 5 Click **OK** to apply the Thicken feature.

6.4.4 Trimming a Solid

You can trim a solid model with respect to a surface.

To trim a model:

- 1 From the **Feature** menu, select **Trim Model**; or select the **Trim Model** tool  from the Part Modeling toolbar. The **Trim Model** dialog box appears.



- 2 Select the **Surface** you wish to use to trim the model. Arrows in the workspace will show the direction the surface will trim. Check the **Reverse** box if necessary.
- 3 Click **OK** to apply the Trim Model feature. All features contained in the model will be trimmed.

6.4.5 Extruding to Geometry

You can use a reference surface in the “To Geometry” option of an extrusion or sweep. See Chapter 7 for information on extruding “To Geometry”.

The sketch you are extruding must lie completely within the outline of the surface or the extrusion will fail.

6.5 Reference Geometry Visibility

You can hide reference geometry on an individual basis, by group, or altogether.

6.5.1 Hiding Individual Reference Geometry Items

- 1 Select the reference geometry item in the Design Explorer or work area. You can also select multiple items. The selected items become highlighted in the Design Explorer as well as the work area.

- 2 Right-click in the work area and select **Hide** from the pop-up menu; or press **Ctrl + H** on the keyboard; or from the **Edit** menu select **Hide Selection**.

The selected reference geometry items are hidden. The text in the Design Explorer associated with the items will also become light gray.

6.5.2 Hiding Individual Reference Geometry Groups

You can hide groups of planes, axes, points, surfaces, or the world coordinate system independently.

- 1 From the **View** menu, select **References > Planes**, or **References > Axes**, or **References > Points**, or **References > Surfaces**, or **References > Coordinate System**, depending on which group you want to hide. The group becomes hidden, and the text in the Design Explorer associated with the items will also become light gray.

You can also press **Ctrl + Shift + P** on the keyboard to hide planes.

Note: Upon hiding planes as a group, all planes are hidden except for the plane that is currently selected. To hide all the planes at once including the selected plane, select the **Planes** node in the Design Explorer, right-click in the work area and select **Hide** from the pop-up menu.

6.5.3 Hiding All Reference Geometry Groups

You can hide all reference geometry groups at one time.

- 1 From the **View** menu, select **References > All**. All reference geometry groups become hidden except the plane that is currently selected. The text in the Design Explorer associated with the items will also become light gray.

6.6 Renaming Reference Geometry

New reference geometry items are by default sequentially named beginning with 4, e.g. **Plane <4>**, **Axis <4>**, **Point <4>**, etc. You can rename default reference geometry as well as inserted reference geometry items.

To rename reference geometry items:

- 1 Right-click the reference geometry item in the Design Explorer and select **Rename** from the pop-up menu. Or click the reference geometry in the Design Explorer

twice with a short pause between clicks. The name is highlighted and the cursor appears next to the name.

- 2** Type a new name.
- 3** Press **Enter** on the keyboard.

6.7 Deleting Reference Geometry

You can delete inserted reference geometry items. You cannot delete default reference geometry.

To delete an inserted reference geometry item:

- 1** Right-click the reference geometry item in the Design Explorer and select **Delete** from the pop-up menu; or select the reference geometry item in the Design Explorer or work area and press **Delete** on the keyboard.

6.8 Editing Reference Geometry Properties

You can modify the properties associated with an inserted reference geometry item.

To edit an inserted reference geometry item:

- 1** Right-click the reference geometry item in the Design Explorer and select **Edit** from the pop-up menu; or right-click the reference geometry item in the work area and select **Edit** from the pop-up menu. The dialog box associated with the item appears displaying the original properties.
- 2** Modify the properties as necessary.
- 3** Click **OK** to apply the change.

7 Feature Creation

Parts are modeled by creating features. Features are individual 3D shapes representing common mechanical design elements, like bosses and holes, which either create material or remove material in a part. Many features, such as extrude boss and revolve boss, require an associated sketch to define the 2D profile of the 3D shape. Other features, such as fillet and edge chamfer, can be created without a sketch and are applied to existing edges and faces.

This chapter describes:

- The part modeling interface
- Extrude boss and extrude cut features
- Thin wall boss and cut features
- Revolve features
- Loft features
- Sweep features
- Fillet and chamfer features
- Shell features
- Draft features
- Hole Features
- Catalog Features
- Mirroring features
- Creating feature patterns
- Managing features in the Design Explorer

7.1 The Part Modeling Interface

The Part Modeling toolbar is shown by default on the right side of the workspace. Commonly used modeling tools are accessible on the Part Modeling toolbar.



	Extrude Boss . . . create an extrude boss feature
	Extrude Cut . . . create an extrude cut feature
	Revolve Boss . . . create a revolve boss feature
	Revolve Cut . . . create a revolve cut feature
	Loft Boss . . . create a loft boss feature
	Loft Cut . . . create a loft cut feature
	Sweep Boss . . . create a sweep boss feature
	Sweep Cut . . . create a sweep cut feature
	Helical Boss . . . create a helical boss feature
	Helical Cut . . . create a helical cut feature
	Design Boolean . . . create Boolean feature
	Trim Model . . . create a fillet feature
	Thicken Surface . . . create a fillet feature
	Fillet . . . create a fillet feature
	Edge Chamfer . . . create an edge chamfer feature
	Shell . . . create a shell feature
	Draft Surface . . . create a draft surface feature
	Hole . . . create a hole feature
	Insert Catalog Feature . . . insert a feature that has been saved to the repository
	Layers . . . manage different layers within the part (refer to Chapter 9)
	Equation Editor . . . open the Equation Editor (refer to Chapter 4)
	Regenerate . . . regenerate the part to update changes (refer to Chapter 9)

The tools that are accessible on the Part Modeling toolbar are accessible from the **Feature** menu as well. The Feature menu also contains tools that do not have a corresponding toolbar icon.

Thin Wall Boss > Extrude . . . create a thin wall extrude boss feature

Thin Wall Boss > Revolve . . . create a thin wall revolve boss feature

Thin Wall Boss > Sweep . . . create a thin wall sweep boss feature

Thin Wall Cut > Extrude . . . create a thin wall extrude cut feature

Thin Wall Cut > Revolve . . . create a thin wall revolve cut feature

Thin Wall Cut > Sweep . . . create a thin wall sweep cut feature

Chamfer > Vertex . . . create a vertex chamfer feature

Mirror . . . mirror a feature about an edge or axis

Pattern > Linear . . . create copies of a feature in a linear pattern

Pattern > Circular . . . create copies of a feature in a circular pattern

Scale . . . scale a part model in any direction

Remove Face . . . delete a face (refer to **Chapter 8**)

Offset Face . . . offset a face by a specified distance (refer to **Chapter 8**)

Move Face . . . move a face by a specified distance (refer to **Chapter 8**)

Save Catalog Feature . . . save a feature to the repository for use in other models

7.2 Feature Terminology

7.2.1 Feature Types

Boss

Boss features are used to create or add material in a part. Generally, the first feature you create in a part will be a boss type feature. The first feature created in a part is called the **base feature**.

Cut

Cut features are used to remove material from a part. You cannot create a cut feature until at least one boss feature has been created. Consequently, a cut feature is never the base feature.

7.3 Extrude Boss and Extrude Cut

Although the extrude boss and extrude cut features are different in end result, the steps used to create them are identical.

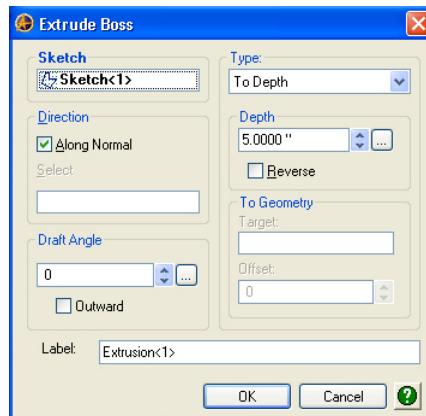
Extrude features, also referred to as **extrusions**, either create or remove material by extending a sketch in a linear direction by a specified distance.

7.3.1 Creating Extrude Boss and Extrude Cut Features

Extrude features require a closed sketch (refer to **Chapter 4**).

To create an extrude boss or extrude cut:

- With a sketch still active, select the **Extrude Boss**  tool or the **Extrude Cut**  tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Extrude** or **Cut > Extrude**. The **Extrude Boss** or the **Extrude Cut** dialog box appears.



- Make sure that the **Sketch** field is populated with the sketch you want to extrude.

- 3 Select a **Type** from the pull down menu. Follow the steps below depending on the Type.

To Depth, Mid Plane, Through All extrusions

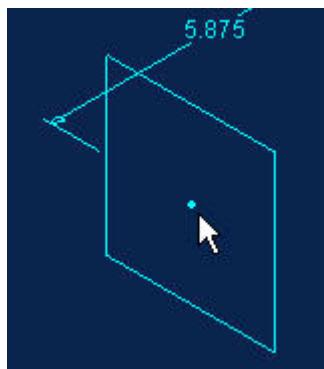
To Depth — Creates an extrusion of a specified depth on one side of the sketch plane.

Mid Plane — Creates an extrusion of a specified depth on both sides of the sketch plane. Half the extrusion length is proportioned to each side of the sketch plane.

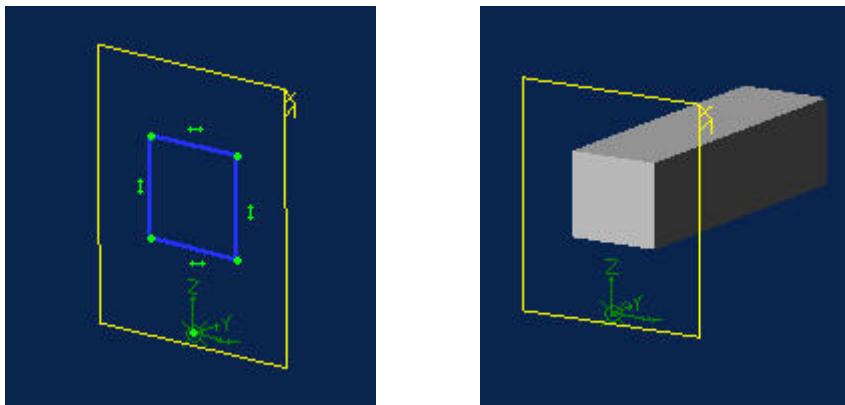
Through All — Applies only to extrude cut. Creates the cut through the entire solid in the specified direction.

- 4 Specify the extrusion **Depth** value. If using **To Depth**, you can select the **Reverse** option if necessary to create the extrusion on the opposite side of the sketch plane.

Note: When using the **To Depth** or **Mid Plane** options, you can dynamically resize the extrusion in the work area by dragging the node associated with the sketch profile. As you drag the node, the extrusion length will automatically increase or decrease increments based on the **Spinner Increment** value (**File > Properties > Units** tab).



- 5 To create the extrusion in a different direction other than normal to the sketch plane, deselect the **Along Normal** option. Then select a linear edge or axis to define the extrusion direction.
- 6 Specify a **Draft Angle** if required and click the **Outward** option if desired.
- 7 Enter a custom **Label** to modify how the feature name is displayed in the Design Explorer.
- 8 Click **OK** to create the extrusion.



To Geometry extrusions

To Geometry — Creates an extrusion up to another reference plane or face.

- 4 Select a **Target** by selecting a reference plane, surface, or face.
- 5 You can also specify an **Offset** value to create the extrusion up to a specified distance from the Target.
- 6 To create the extrusion in a different direction other than normal to the sketch plane, deselect the **Along Normal** option. Then select a linear edge or axis to define the new extrusion direction.
- 7 Specify a **Draft Angle** if necessary and click the **Outward** option if desired.
- 8 You can modify **Label** to control how the feature name is displayed in the Design Explorer.
- 9 Click **OK** to create the extrusion.

To Next extrusions

To Next — Creates an extrusion up to the nearest face(s) of the part.

- 4 Select the **Reverse** option if necessary to create the extrusion on the opposite side of the sketch plane.

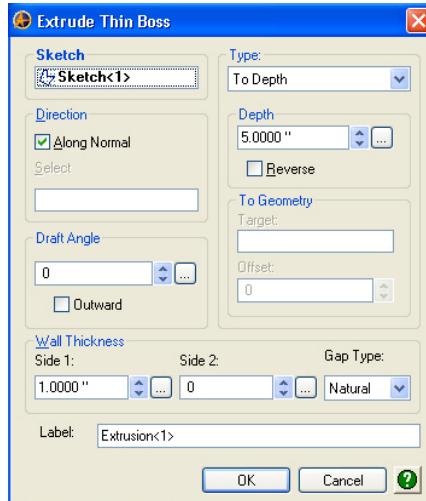
- 5 To create the extrusion in a different direction other than normal to the sketch plane, deselect the **Along Normal** option. Then select a linear edge or axis to define the new extrusion direction.
- 6 Specify a **Draft Angle** if necessary and click the **Outward** option if desired.
- 7 You can modify **Label** to control how the feature name is displayed in the Design Explorer.
- 8 Click **OK** to create the extrusion.

7.3.2 Creating Thin Wall Extrude Boss And Cut Features

Thin wall extrude features either create thin walled bosses or cuts by extending a sketch in a linear direction by a specified distance. As opposed to normal extrude boss and cut features discussed in section 7.3.1, thin wall extrusions can be created with open or closed sketches (refer to **Chapter 4** for more information about sketches).

To create a thin wall extrude boss or cut:

- 1 With a sketch still active, from the **Feature** menu, select **Thin Wall Boss > Extrude** or **Thin Wall Cut > Extrude**. The **Extrude Thin Boss** or the **Extrude Thin Cut** dialog box appears.

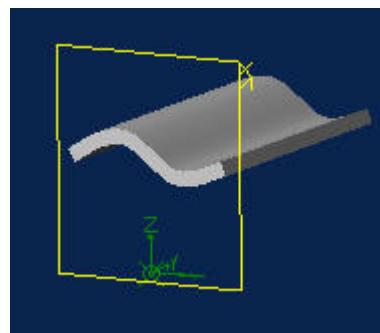
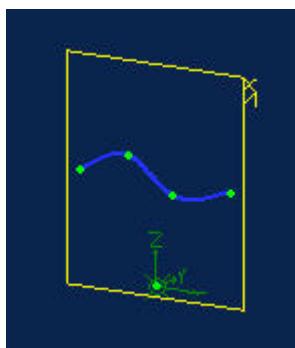


- 2 Select a **Type**. Refer to section 7.3.1 for information related to the Type condition.
- 3 Specify a **Length/Depth** value or select a **Target**.

4 Select a **Gap Type**:

- **Natural:** Extends the edges of the wall along their natural curves until they intersect; for example.
- **Round:** Creates fillets on any corners of the wall profile.
- **Extend:** Extends the edges of the wall beyond their endpoints in straight lines until they intersect.

- 5 Specify an extrusion direction. Select the **Along Normal** option to create the extrusion normal to the sketch plane. Deselect the **Along Normal** option and select an edge or axis to create the extrusion in a direction other than normal to the sketch plane.
- 6 To taper the extrusion, specify a **Draft Angle**. To change the draft orientation, select the **Outward** option.
- 7 Specify the **Wall Thickness** by entering values for the **Side 1** and **Side 2** wall thicknesses. Specifying a **Side 1** value will create or remove material inward from the sketch, and a **Side 2** value will create or remove material outward from the sketch.
- 8 Click **OK** to create the thin wall extrusion.



7.4 Revolve Boss and Revolve Cut

Although the revolve boss and revolve cut features are different in end result, the steps used to create them are identical.

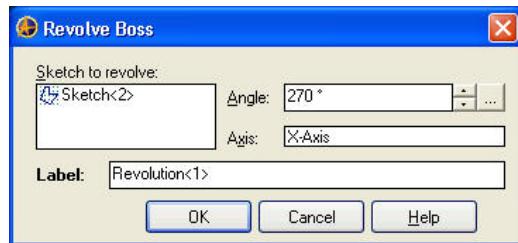
Revolve features, also referred to as **revolutions**, either create or remove material by revolving a sketch around a centerline.

7.4.1 Revolve Boss and Revolve Cut Features

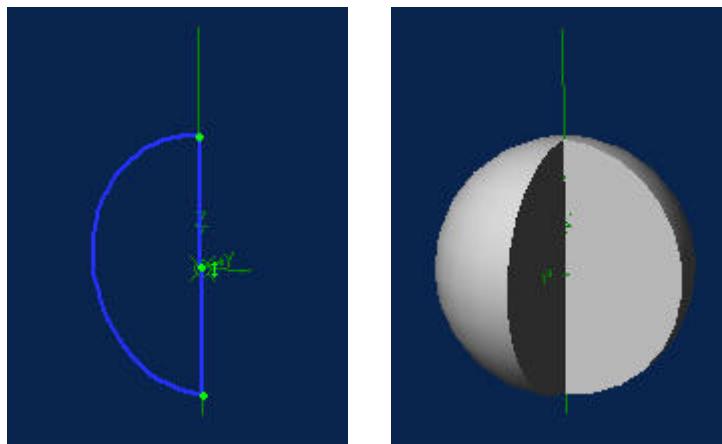
Revolve boss and cut features require a closed sketch (refer to Chapter 4 for more information about sketches).

To create a revolve boss or cut:

- 1 With a sketch still active, select the **Revolve Boss**  or the **Revolve Cut**  tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Revolve** or **Cut > Revolve**. The **Revolve Boss** or the **Revolve Cut** dialog box appears.



- 2 Make sure that the **Sketch to revolve** field is populated with the sketch you want to revolve.
- 3 Select an edge, axis, or sketch line as the **Axis** centerline.
- 4 Specify the rotation **Angle**.
- 5 Enter a custom **Label** to modify how the feature name is displayed in the Design Explorer.
- 6 Click **OK** to create the revolution.



7.4.2 Thin Wall Revolve Boss and Cut Features

Thin wall revolve features either create or remove a thin wall of material by revolving an open or closed sketch around a centerline.

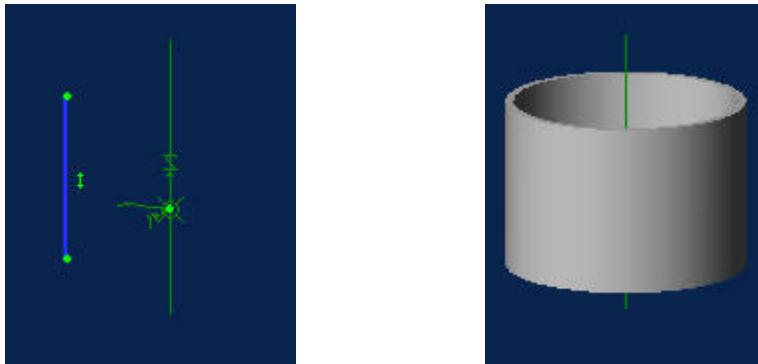
To create a thin wall revolve boss or cut:

- 1 With a sketch still active, from the **Feature** menu, select **Thin Wall Boss > Revolve** or **Thin Wall Cut > Revolve**. The **Revolve Thin Boss** or the **Revolve Thin Cut** dialog box appears.



- 2 Specify the rotation **Angle**.

- 3 Select an edge, axis, or sketch line as the **Axis** centerline.
- 4 Choose a **Gap Type**:
 - **Natural**: Extends the edges of the wall along their natural curves until they intersect.
 - **Round**: Creates fillets on any corners of the wall profile.
 - **Extend**: Extends the edges of the wall beyond their endpoints in straight lines until they intersect.
- 5 Specify the **Wall Thickness**. **Side 1** creates or removes material on the inward side of the sketch; **Side 2** creates or removes material on the outward side of the sketch.
- 6 Click **OK** to create the thin wall revolution.



7.5 Loft Boss and Loft Cut

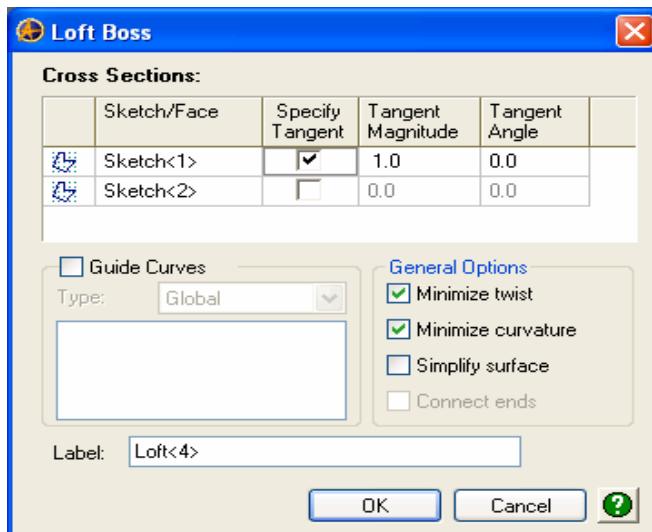
Although the loft boss and loft cut features are different in end result, the steps used to create them are identical.

Loft features, simply referred to as lofts, either create or remove material by forming a transition between sketches that reside on different planes.

You can use two or more sketches or existing faces to create lofts.

To create a loft boss or cut:

- 1 Sketch at least two profiles on two different planes. The sketches must be closed (Refer to **Chapter 4** for information about sketches). The planes do not have to be parallel. You may loft to a point by creating a sketch with exactly one node in it.
- 2 Select the **Loft Boss**  tool or **Loft Cut**  tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Loft** or **Cut > Loft**. The **Loft Boss** or the **Loft Cut** dialog box appears.



- 3 In the work area, select the sketches or faces to use in the loft. If you select inside the design explorer, you must press Shift to select multiple sketches or faces. As you make the selections, the corresponding labels appear in the **Cross Sections** box.

The sketches/faces must be listed in the order in which the loft will be created. To remove a sketch/face from the box, select it from the list and press **Delete** on the keyboard.

- 4 If desired, check the **Specify Tangent** box next to a sketch or face. When "Specify Tangent" is selected, local control of surface directions in the vicinity of the lofted sketches/faces is possible. If "Tangency Angle" is 0° , surface normals remain perpendicular to the sketch plane normal, or parallel to the adjoining face normals.
- 5 If **Specify Tangent** is checked, **Tangent Angles** can be specified for sketches (but not faces). Specify the desired angle in the **Tangent Angle** field. In this case, the lofted surface tangent would be at the angle specified with the sketch plane normal.

- 6 If Specify Tangent is checked, additional control is obtained using the Tangent Magnitude for each Cross Section. Tangent Magnitudes control the rate at which surfaces diverge from the cross section sketch planes or faces. As Tangent Magnitudes increase, surfaces diverge more slowly from surface tangents at the sketch cross sections. Specify the desired value in the **Tangent Magnitude** field. A value of 0 is equivalent to not using "Specify Tangent".
- 7 If desired, Guide curves can be specified to constrain the location or direction of the lofted surface that is generated. To specify Guide curves, check the **Guide Curves** checkbox, and select the sketches in the work area. 2D and 3D sketches may be used.

Guide curves are each exactly one open or closed loop, each touching all the profiles. Multiple guide curves can be used for Global and Local types, however, for the Tangent type, only one guide curve should be specified. As each curve is selected, the corresponding labels appear in the Guide Curves list box. To remove a Guide curve from the box, select it from the list and press delete key on the keyboard, or choose an option from the right-click menu.

Guide curve types:

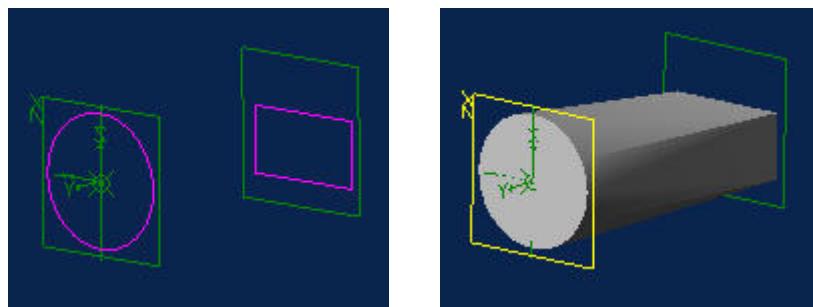
- **Global:** Creates "virtual" guide curves, providing the ability to make one guide curve globally affect the lofted surfaces.
- **Local:** These guide curves provide the ability to make one guide curve locally affect the lofted surfaces.
- **Tangent:** Constrains the take-off vectors on each profile based on a "path" curve. The resulting surface does not follow the path; rather a constant vector field is placed on each profile. The vector is defined as the tangent vector of the path curve at the point in which the curve intersects the profile's plane.

- 8 If necessary, select a loft creation option:

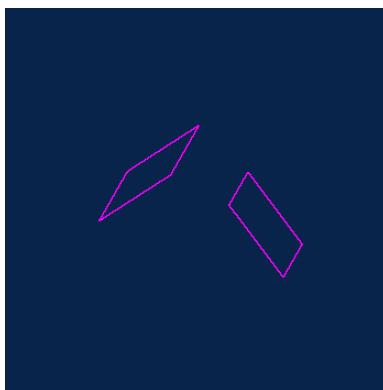
- **Simplify Surface:** Converts the resultant loft face from a spline type to an analytic type when possible. A simplified surface contains less data, is of higher quality, and is faster to process in subsequent modeling operations.
- **Minimize twist:** Aligns the profiles so that the start of the second sketch is aligned to the start of the first sketch.
- **Connect ends:** The first cross-section is treated as if it is also the last cross-section. This is not available when using Guide Curves, as the Guide Curves can be used to connect the first and last cross sections. You must have at least three cross-sections to use this option.

- **Minimize Curvature:** This determines the Tangent Magnitudes based on maximizing the minimum radius of curvature of the lofted body as a whole. This not only helps to create more pleasing surfaces but ensures greater ability to shell and blend lofted models. This is applicable only when take off factors are specified.

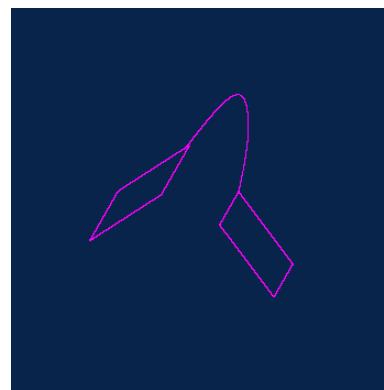
9 Click **OK** to create the loft.



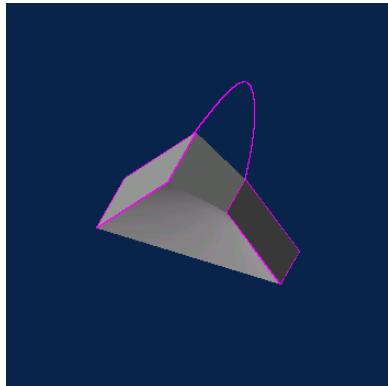
The following pictures illustrate the results of a loft creating using the same sketches, but with different types of guide curves.



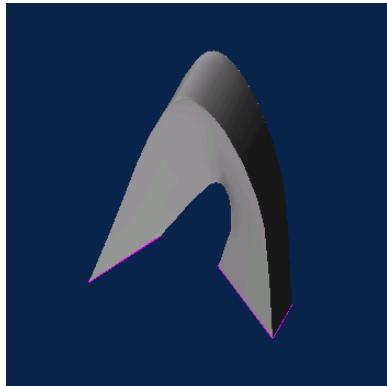
Sketches loft will be created with



Lofting sketches with guide curve sketch

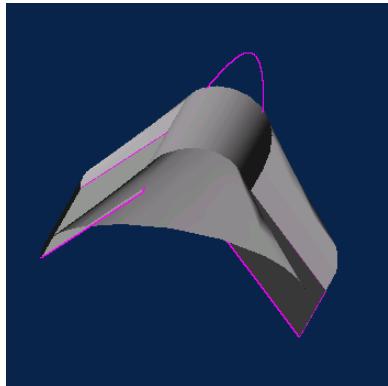


with no guide curve

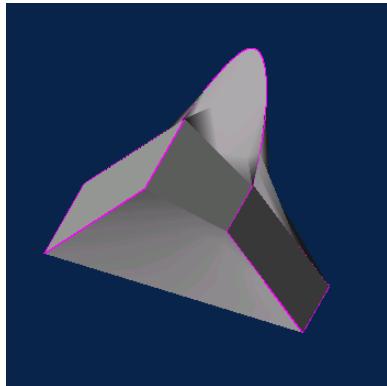


Loft with global guide curve

Loft



Loft with tangent guide curve



Loft with local guide curve

7.6 Sweep Boss and Sweep Cut

Although the sweep boss and sweep cut features are different in end result, the steps used to create them are identical.

Sweep features, simply referred to as sweeps, either create or remove material by moving a sketch along a path defined by a second sketch. The following guidelines should be followed when creating swept features:

- The sketch that defines the profile must be closed (refer to Chapter 4 for information about sketches).
- The sketch(es) that define the path can be open or closed but cannot be self-intersecting.
- The sketch path cannot lie on the same sketch plane as the profile.
- The sketch path must either start on the profile plane or pass through the profile plane.
- The sketch path can be multiple 2D or 3D sketches, as well as Edges of Parts or Surfaces.
- The sketch path must be continuous.

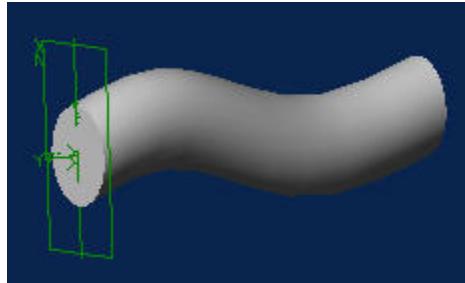
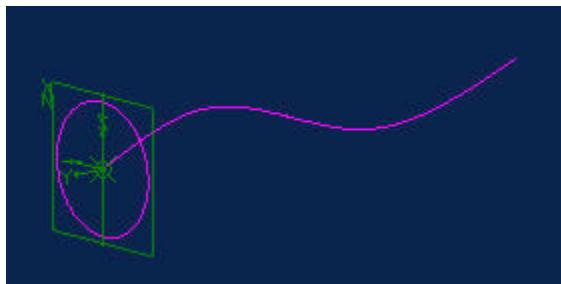
7.6.1 Sweep Boss and Sweep Cut Features

To create a sweep boss or cut:

- 1 Sketch the closed profile.
- 2 Sketch the path(s). The path must either start on or pass through the plane of the profile, but is not required to pass through the profile itself. The path cannot intersect itself.
- 3 Select the **Sweep Boss**  tool or the **Sweep Cut**  tool from the Part Modeling toolbar; or from the Feature menu, select **Boss > Sweep** or **Cut > Sweep**. The **Sweep Boss** or the **Sweep Cut** dialog box appears.



- 4 In the **Sketch to Sweep** field, select the profile sketch.
- 5 In the **Path Objects** field, select the path sketch(es). One or more sketches or edges may be selected.
- 6 Select a **Sweep Type**. You can create the sweep along the **Entire Path** or **To Geometry**.
- 7 If **To Geometry** was selected, click in the **Geometry Target** box, and select an existing plane, surface, or face in the work area. You can also specify a **Geometry Offset** value to create a gap between the target and the end of the feature.
- 8 If **Entire Path** was selected, click the **Rigid** check box if desired to force the profile to remain parallel to the profile's sketch plane through the sweep.
- 9 To place a draft on the sweep, specify a **Draft Angle**. Select the **Outward** option if necessary. You cannot create a draft if the **Rigid** option is applied.
- 10 Click **OK** to create the sweep.

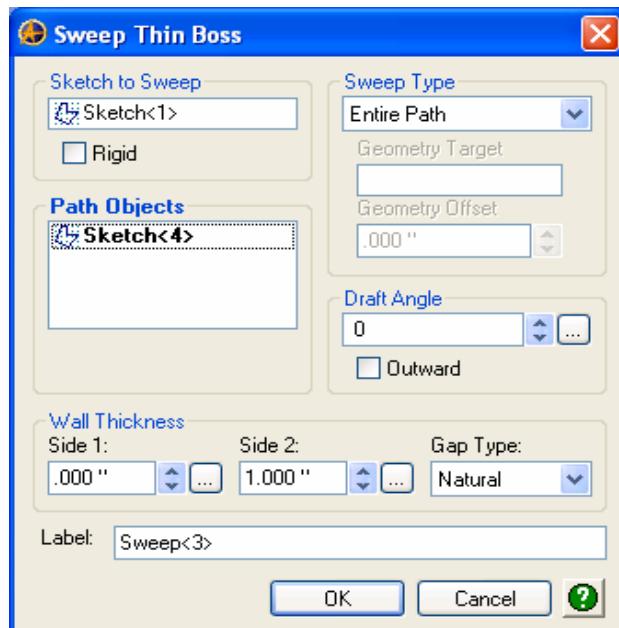


7.6.2 Thin Wall Boss Sweep and Cut Sweep Features

Thin wall sweep features either create or remove a thin wall of material by moving a closed or open sketch along a path defined by a second sketch.

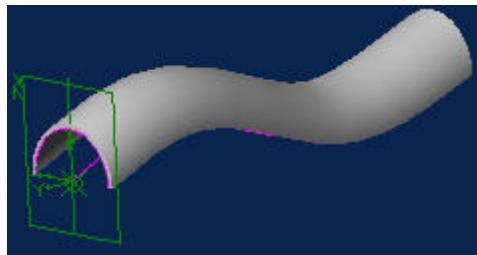
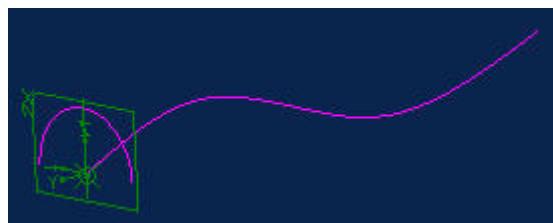
To create a thin wall boss or cut sweep:

- 1 Sketch a closed or open profile to define the sweep cross-section.
- 2 On a different plane, create a new sketch and sketch the path.
- 3 From the **Feature** menu, select **Thin Wall Boss > Sweep** or **Thin Wall Cut > Sweep**. The **Sweep Thin Boss** or the **Sweep Thin Cut** dialog box appears.



- 4 Select the **Sketch to sweep** and the **Path Sketch(es)**.
- 5 Select a **Sweep Type**. You can create the sweep along the **Entire Path** or **To Geometry**.
- 6 If **To Geometry** was selected, click in the **Geometry Target** box, and select an existing plane, surface, or face in the work area. You can also specify a **Geometry Offset** value to create a gap between the target and the end of the feature.
- 7 If **Entire Path** was selected, click the **Rigid** check box if desired to force the profile to remain parallel to the profile's sketch plane through the sweep.
- 8 Select a **Gap Type**:

- **Natural:** Extends the edges of the sweep wall along their natural curves until they intersect.
 - **Round:** Creates fillets on any corners of the wall profile.
 - **Extend:** Extends the edges of the wall beyond their endpoints in straight lines until they intersect.
- 9 Specify the **Wall Thickness**. **Side 1** creates or removes material on the inward side of the sketch, **Side 2** creates or removes material on the outward side of the sketch.
- 10 To taper the extrusion, specify a **Draft Angle**. To change the draft orientation, select the **Outward** option.
- 11 Click **OK** to create the thin wall sweep.



7.7 Helical Boss and Helical Cut

Although the helical boss and helical cut features are different in end result, the steps used to create them are identical.

Helical features, often referred to as helices, either create or remove material by automatically sweeping a cross section, represented by a sketch, along a helical path. The

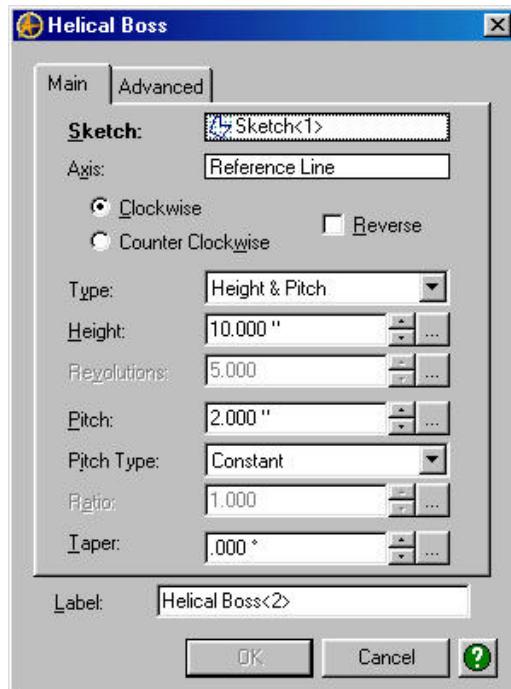
helical path is automatically created by the software and is driven by user specified parameters. Helical features are beneficial when modeling springs, internal and external threads, ball screws, worm gears, etc.

The following guidelines should be followed when creating helical features:

- The sketch that defines the cross-section of the helix must be closed (refer to Chapter 4 for information about sketches).
- The sketch that defines the cross-section of the helix must also contain a reference line that represents the helical axis.

To create a helical boss or cut:

- 1 Sketch the cross-section of the helix. The cross-section is not limited to a certain profile but must be a closed sketch.
- 2 From the **Sketching** toolbar, select the **Reference Line** tool.
- 3 Sketch a reference line of any length in the axial direction that the helix will be created in. This reference line will represent the axis of the helix.
- 4 If necessary, place a dimension between the reference line and sketch figure(s) created in step 1.
- 5 Select the **Helical Boss**  or the **Helical Cut**  tool from the Part Modeling toolbar; or from the **Feature** menu, select **Boss > Helix** or **Cut > Helix**. The **Helical Boss** or the **Helical Cut** dialog box appears, and the **Main** tab is initially displayed by default.



- 6 In the **Sketch** field, select the sketch representing the helical cross-section. The **Axis** field is automatically populated with the reference line corresponding with the sketch. If more than one reference line exists, you can choose which one you wish to use.
- 7 Select the direction of the helix, **Clockwise** or **Counter Clockwise**.
- 8 If necessary, click the **Reverse** check box to change the axial direction the helix is created in.
- 9 From the **Type** pulldown menu, select the appropriate helix type:
 - **Height and Revolution:** a helix is generated by specifying the overall feature height as well as the number of helical revolutions within the specified height.
 - **Height and Pitch:** a helix is generated by specifying the overall feature height as well as pitch. The **pitch** is defined as the distance from one point on the helix to a corresponding point on the next revolution measured parallel to the axis.
 - **Revolution and Pitch:** a helix is generated by specifying the number of revolutions as well the pitch.
 - **Spiral:** a flat helix is generated by specifying the number of revolutions as well

as pitch.

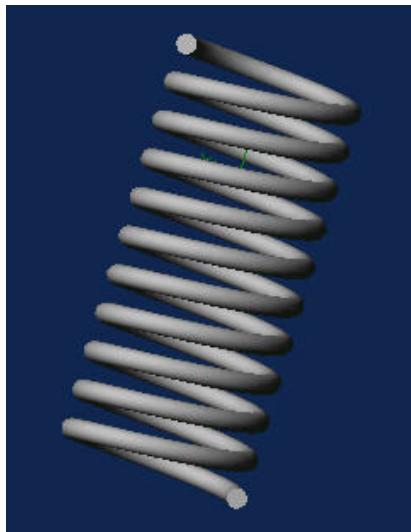
- 10 Specify the appropriate helix parameters (**height**, **pitch**, **revolutions**) depending on which type was selected. Note that if **Pitch** is specified you can also specify **Constant**, **Variable Ratio**, and **Variable End** pitch conditions.
 - **Constant**: Lets you maintain a constant distance. The "ratio" control is disabled.
 - **Variable End**: Specifies the pitch at the end of the helix.
 - **Variable Ratio**: Allows you to change the pitch from start to finish in a ratio such that pitch at the end will be: ratio x start pitch.
- 11 If applicable, specify a **Taper** angle to create a tapered helix.
- 12 If applicable, click the **Advanced** tab to specify start and end conditions other than the **Natural** default condition.
 - **Natural or Flat** Conditions: for each of the two ends of the coil. The ends can have dissimilar end conditions. If Flat is chosen, then you will also need to specify the Transition Angle and the Flat Angle.
Transition Angle: The distance (in degrees) over which the coil achieves the transition (normally less than one revolution). The example shows the top with a natural end and the bottom end with a one-quarter turn transition (90 degrees) and no flat angle.



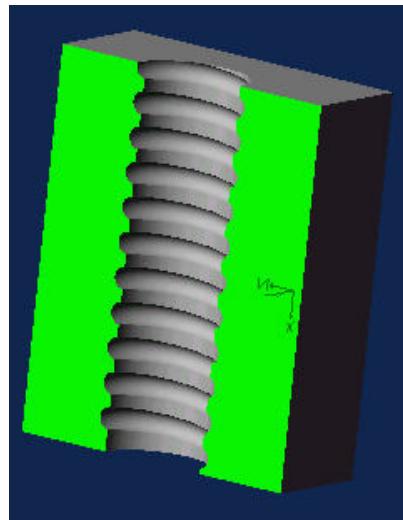
Flat Angle: The distance (in degrees) the coil extends after transition with no pitch (flat). Provides transition from the end of the revolved coil to a flattened end. The example shows the same coil as the Transition angle shown above, but with a half-turn (180 degree) flat angle specified.



- 13 Specify a **Parallel** or **Normal Profile Orientation** condition.
- 14 Click **OK** to create the helical feature.



Helical Boss Feature



Helical Cut Feature

7.8 Fillet

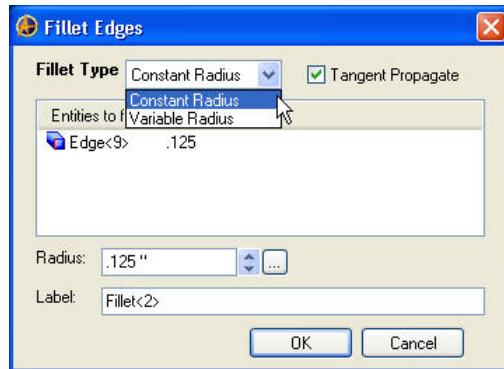
Fillet features create a rounded face on a part. You can place a fillet on individual edges, edge loops, and all edges of a face. You can create a constant radius fillet or a variable radius fillet.

7.8.1 Constant Radius Fillets

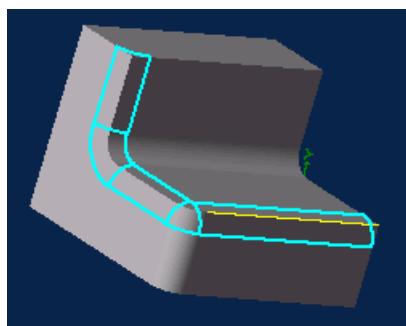
Constant radius fillets create a rounded face of constant radius on an edge.

To create a constant radius fillet:

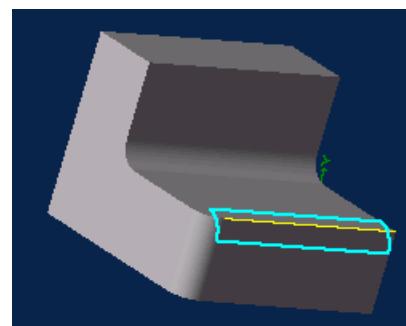
- 1 Select the **Fillet**  tool from the Part Modeling toolbar; or from the **Feature** menu select **Fillet**; or right-click and select **Add Fillet** from the pop-up menu. The **Fillet Edges** dialog box appears.



- 2 Select **Constant Radius** as the **Fillet Type**.
- 3 Select the edge(s) or face(s) to be rounded. Selecting a face will subsequently select all the edges associated with that face.
- 4 Unselect the **Tangent Propagate** option if necessary. The Tangent Propagate option creates a fillet on the selected edge as well as any other edges that form a path in which a tangent condition can be resolved.



Tangent Propagate On



Tangent Propagate Off

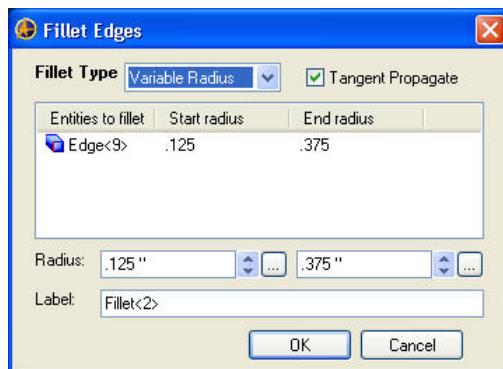
- 5 Specify the **Radius**.
- 6 Click **OK** to create the fillet.

7.8.2 Variable Radius Fillets

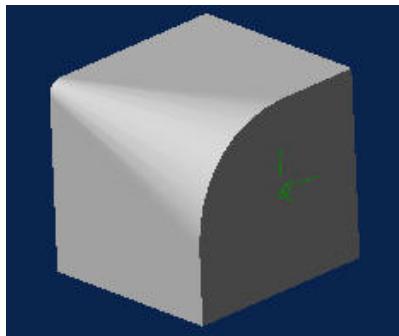
Variable radius fillets create a rounded face of variable radius on an edge. You specify a start radius and end radius and a smooth transition is made between the two along the selected edge(s).

To create a variable radius fillet:

- 1 Select the **Fillet**  tool from the Part Modeling toolbar; or from the **Feature** menu select **Fillet**; or right-click and select **Add Fillet** from the pop-up menu. The **Fillet Edges** dialog box appears.
- 2 Select **Variable Radius** as the **Fillet Type**.



- 3 Select the edge(s) or face(s) to be rounded. Selecting a face will subsequently select all the edges associated with that face.
- 4 Unselect the **Tangent Propagate** option if necessary. The Tangent Propagate option creates a fillet on the selected edge as well as any other edges that form a path in which a tangent condition can be resolved.
- 5 Specify the start **Radius** in the first radius field.
- 6 Specify the end **Radius** in the second radius field.
- 7 Click **OK** to create the fillet.



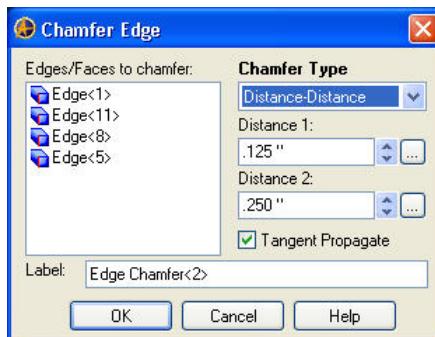
7.9 Chamfers

Chamfer features create a beveled face on a selected edge, face, or vertex.

7.9.1 Edge Chamfers

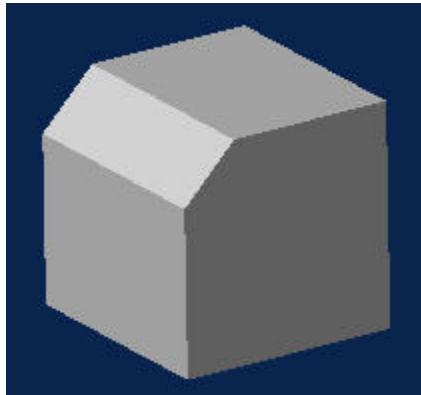
To create an edge chamfer:

- 1 Select the **Chamfer** tool from the Part Modeling toolbar; or from the **Feature** menu, select **Chamfer > Edge**; or right-click and select **Add Edge Chamfer** from the pop-up menu. The **Chamfer Edge** dialog box appears.



- 2 Select the edges or faces to chamfer.

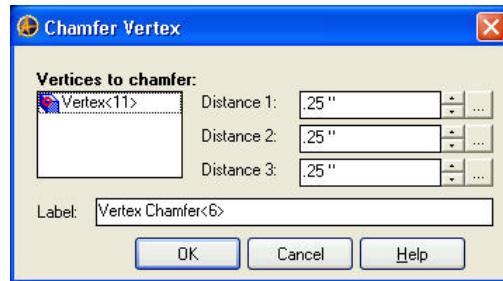
- 3 Select the **Chamfer Type** from the list:
 - **Distance – Distance:** specify two distances, one for either side of the chamfer edge.
 - **Angle – Distance:** specify a distance and angle for the chamfer.
 - **Equal Distance:** specify an equal distance for either side of the chamfer edge.
- 4 Specify the chamfer **Distance(s)** and/or **Angle** values accordingly.
- 5 Unselect the **Tangent Propagate** option if necessary. The Tangent Propagate option creates a chamfer on the selected edge as well as any other edges that form a path in which a tangent condition can be resolved.
- 6 Click **OK** to create the chamfer.



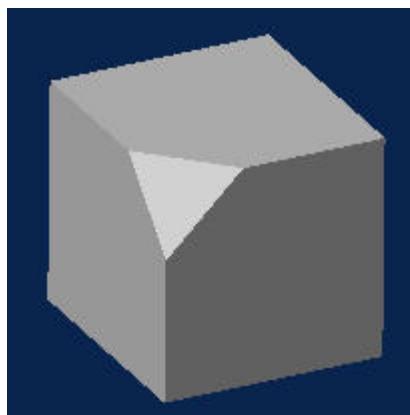
7.9.2 Vertex Chamfers

To create a vertex chamfer:

- 1 From the **Feature** menu, select **Chamfer > Vertex**. The **Chamfer Vertex** dialog box appears.



- 2 Select the vertex to chamfer.
- 3 Specify the three **Distance** values for the three chamfer edges.
- 4 Click **OK** to create the chamfer.

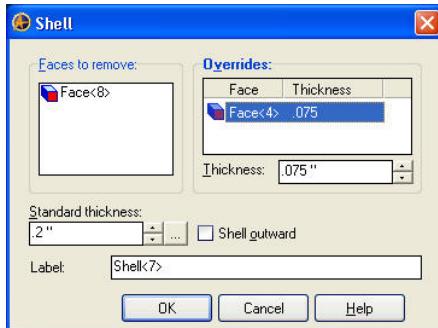


7.10 Shells

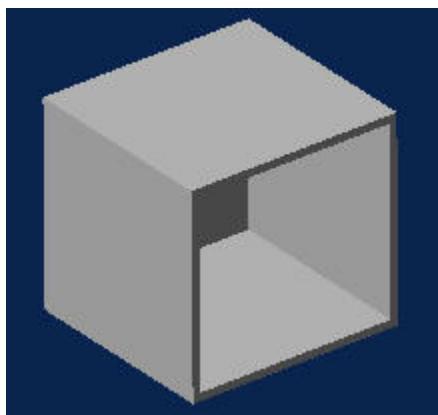
Shell features hollow out a part, removing the face(s) you select and leaving thin walls on the remaining faces.

To create a shell:

- 1 Select the **Shell** tool from the Part Modeling toolbar; or from the **Feature** menu select **Shell**. The **Shell** dialog box appears.



- 2 Select the **Faces to remove**. You can select multiple faces if necessary.
- 3 Specify the **Standard thickness** value, which defines the wall thickness after the part is shelled.
- 4 If required, you can also select faces to override, in which case you can specify a custom wall **Thickness**. To do so, click in the **Overrides** area, and select the face or faces to override in the Design Explorer or work area. You can then set a custom override **Thickness** for each applicable face.
- 5 Select the **Shell outward** option if you want to add external wall thickness.
- 6 Click **OK** to create the shell.

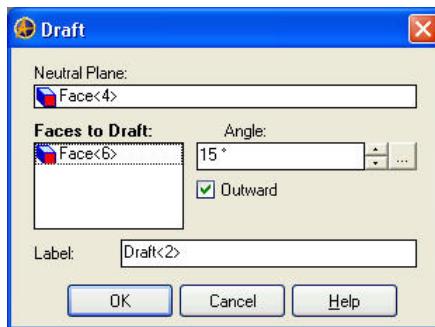


7.11 Draft Faces

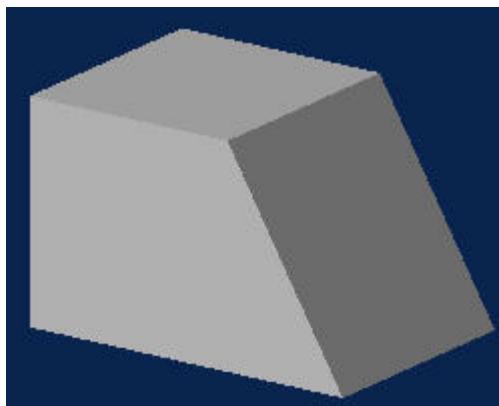
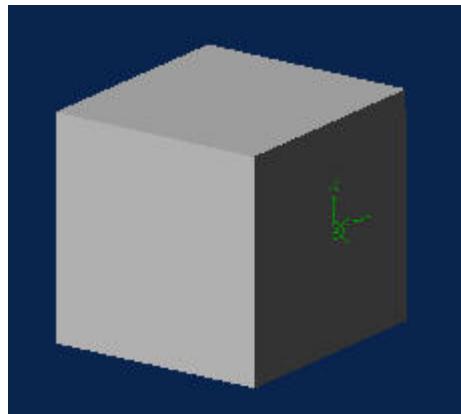
Draft surface features create a tapered face at a specified angle to another face in the model. You can create drafts as individual features or you can specify drafts while creating other features such as extrude bosses.

To create a draft:

- 1 Select the **Draft Surface**  tool from the Part Modeling toolbar; or from the **Feature** menu select **Draft**; or right-click and select **Add Draft** from the pop-up menu. The **Draft** dialog box appears.



- 2 Select the **Neutral Plane**. The draft angle is measured from the Neutral Plane. A reference plane or existing face can be used as the Neutral Plane.
- 3 Select the **Faces to Draft**.
- 4 Specify the draft **Angle**.
- 5 If necessary, select the **Outward** option to create the draft in the opposite direction.
- 6 Click **OK** to create the draft.

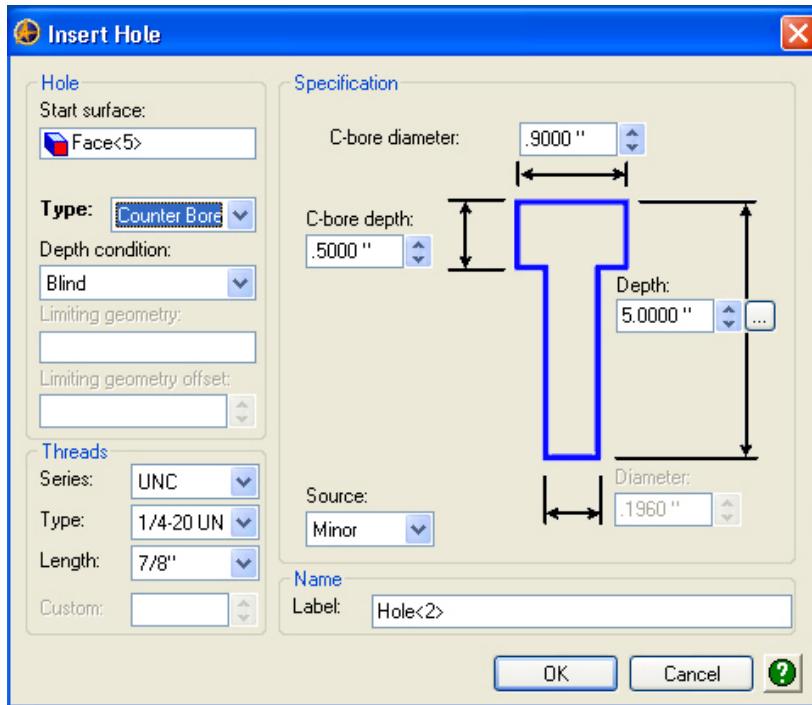


7.12 Holes

Hole features create standard holes, e.g. counter bored or counter sunk holes, in a part. You place hole features on planar faces and then place dimensions or constraints on the hole center to position the hole as required. In addition, you can add thread information, which can be shown on drawings.

To create a hole:

- 1 Select the **Hole**  tool from the Part Modeling toolbar; or from the **Feature** menu select **Hole**; or right-click and select **Add Hole** from the pop-up menu. The **Hole** dialog box appears.



- 2 Select the **Start surface**. A reference plane or planar face can be used as the start surface.

Note: You can place multiple holes simultaneously. With the Hole dialog box open, just click again on the start surface to place another hole.

- 3 Select the hole **Type** from the list. A number of standard hole types are available.
- 4 In the **Specification** field, enter the parameters associated with the hole type.
- 5 Select a **Depth condition** from the list:

- **Blind:** creates the hole to a specified depth.
- **To Limit Geometry:** creates a hole up to a specified face or plane. An offset can also be specified.
- **Through All:** creates a hole through the entire part.

- 6 Specify the depth parameters depending on which depth condition is being used.
- 7 If needed, choose the **Thread** type as described below.
- 8 Click **OK** to create the hole(s).

To add threads to a hole:

- 1 In the **Insert Hole** dialog, choose the desired thread **Series**. Alibre Design supports UNC, UNF, UNEF, UNS, Metric Course, Metric Fine, Metric Special, and NPT series threads.

Note: In addition to the pre-defined thread options, you can create your own thread definitions by editing the Alibre Design thread definition file, `alibre_unicode.thd`. You can use **Notepad** to edit this file. A definition of the file format is embedded within the file. This file is located in the folder **C:\Documents and Settings\All Users\Application Data\Alibre Design\System Files**.

- 2 Choose the **Type** thread desired.
- 3 Specify the **Length** of the thread. By default, the thread will extend the entire length of the hole.
- 4 Choose the **Source** to specify the diameter to use for modeling the hole. You may choose from the minor, pitch, major, and drill tap diameters.
- 5 Click **OK** to create the hole and define the thread.

Note: The thread information is now included with the hole data. While a graphical representation of the threads is not displayed on the model, when the 2D detailed drawing is created from the 3D design, the threaded hole information will automatically be called out in the applicable orthographic view. For information related to calling out the thread information in the 2D drawing, refer to section 11.7.2.

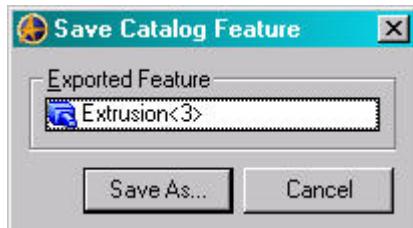
7.13 Catalog Features

Extrude boss and cut features, hole features, sheet metal cut and dimple features, and sketches can be cataloged and reused in the same and other designs.

7.13.1 Saving Catalog Features

To save a catalog feature:

- 1 From the **Feature** menu, select **Save Catalog Feature**. The Save Catalog Feature Dialog box appears.



Select the feature or sketch to save from the Design Explorer.

Click **Save As**. The **Save As** Dialog box appears.

Use the **Document Browser** embedded in the **Save As** dialog to navigate to the desired location in either the repository or file system. When you click **Save**, the feature will be saved to this location.

In **Name**, enter a name to give this feature.

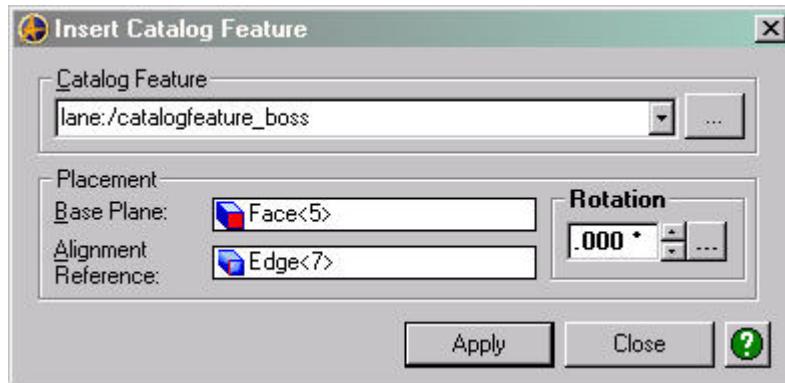
Click **Save** to complete the save.

Once you have a saved feature or sketch in the repository, you can create several instances of this feature in any model by using Insert Catalog Feature.

7.13.2 Inserting Catalog Features

To insert a catalog feature after it has been saved:

- 1 From the **Feature** menu, select **Insert Catalog Feature**. The Insert Catalog Feature dialog box appears.



- 2 Type in the saved name of the feature or sketch you want to use, or click the '...' button to bring up the 'Insert Design' dialog.
- 3 In the **Insert Design** dialog box, select the feature or sketch to use and click **OK**.
- 4 In **Placement**, select the 'Base Plane' field in the dialog. Select the face that you want to insert the feature or sketch on.
- 5 In **Alignment Reference**, select a linear edge to orient the catalog feature. How this feature is placed will depend on how it was created. When exported, the alignment reference is the bottom of the window of the sketch. Therefore, the "bottom" of the feature as it was originally sketched will be aligned with the edge selected here.
- 6 Enter a value for rotation if you need to rotate the imported operation. You can also use the 'Rotate' and 'Move' handles in the workspace to locate the feature — two dots appear on the preview. Move your cursor over them to get the move or rotate arrows. When you get the arrows, click and hold to drag the feature around.
- 7 Click **Apply** and **Close**. You will see in the new feature or sketch in the Feature Tree. The new operation and the sketch created by the catalog feature is a first class operation that can be edited/deleted.

7.14 Copying Existing Features

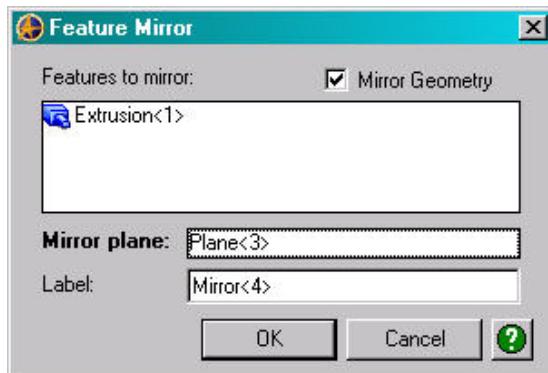
After features have been created, they can be mirrored or patterned to easily create copies.

7.14.1 Mirror Feature

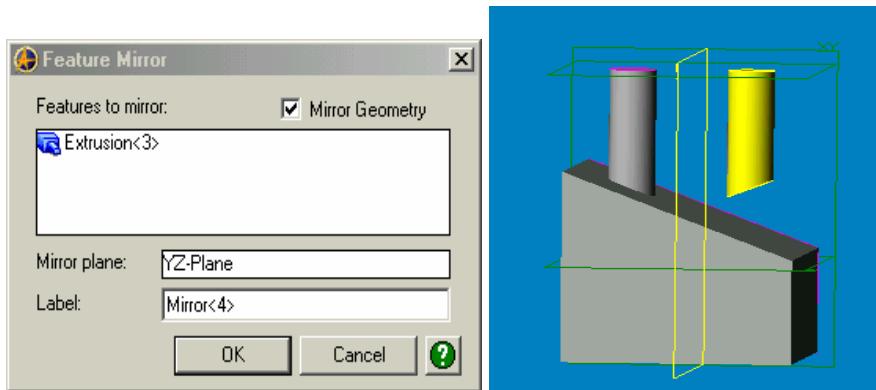
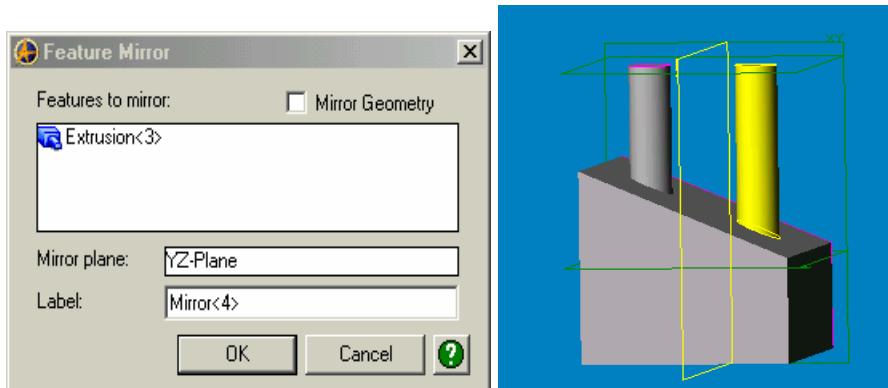
Mirror feature creates a copy of a feature mirrored about a reference plane or planar face. If the original feature changes, the mirrored copy will automatically change as well.

To mirror a feature:

- 1 From the **Feature** menu select **Mirror**. The **Feature Mirror** dialog box appears.



- 2 Select the **Features to mirror** from the Design Explorer.
- 3 Select the **Mirror plane**. You can use a reference plane or planar face.
- 4 Check the **Mirror Geometry** box if you want to mirror the geometry exactly. You will normally be concerned with this option when you are mirroring a feature that has been created using references to existing geometry such as "extrude to geometry". Below are two models. The first shows a mirror created with the Mirror Geometry box unchecked, and the second with the box checked.



5 Click **OK** to create the mirror.

7.14.2 Feature Patterns

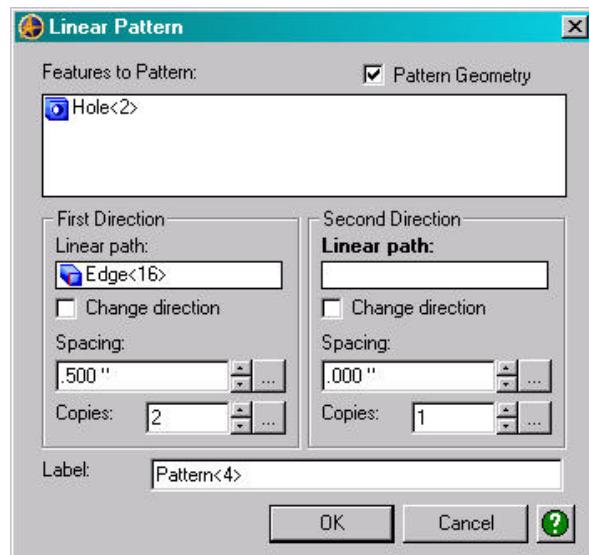
Feature patterns create copies of a feature by repeating it in a linear or circular array.

Linear Patterns

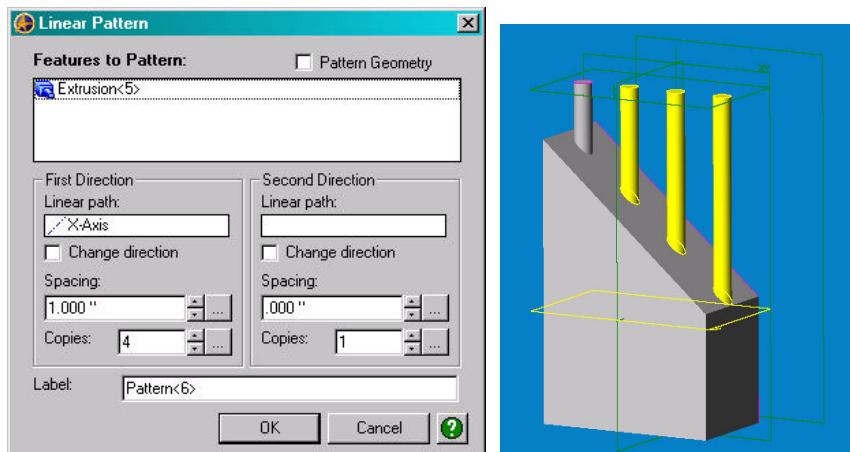
You can use a linear pattern to repeat a feature in one or two linear directions.

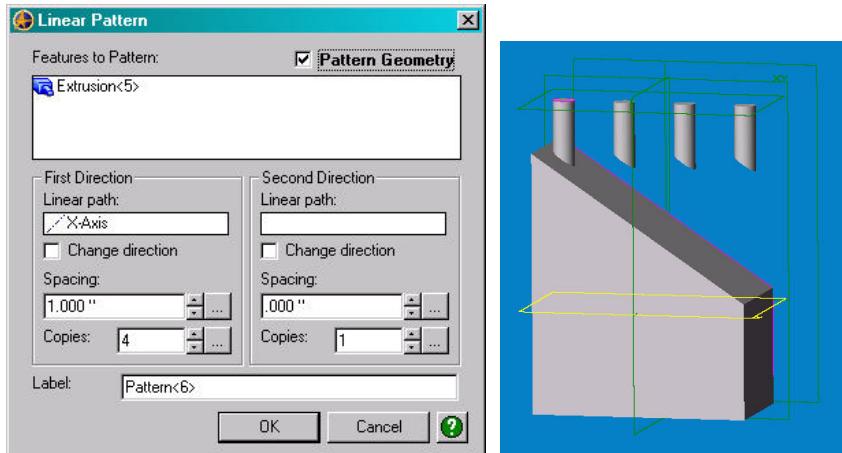
To create a linear pattern:

- 1 From the **Feature** menu, select **Pattern > Linear**. The **Linear Pattern** dialog box appears.



- 2 Select the features to be patterned.
- 3 Check the **Pattern Geometry** box if you want to pattern the geometry exactly. You will normally be concerned with this option when you are patterning a feature that has been created using references to existing geometry such as "extrude to geometry". Below are two models. The first shows a pattern created with the Pattern Geometry box unchecked, and the second with the box checked. The cylinder was sketched on the ZX Plane, and extruded to the sloped face.





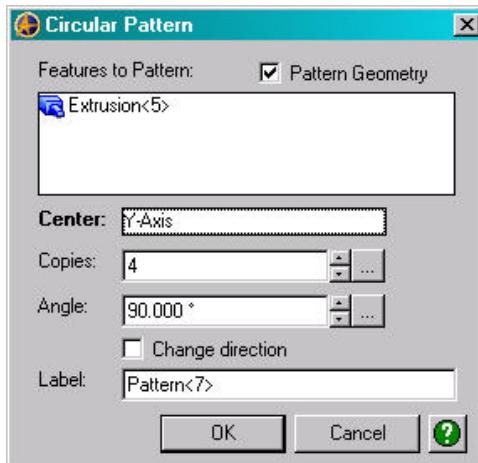
- 4 Select **Linear path** for the **First Direction**. An axis or linear edge can be used for the path.
- 5 Specify the number of **Copies** including the original feature.
- 6 Specify the **Spacing** between each copy.
- 7 Click the **Change direction** option to create the pattern in the opposite direction on the path.
- 8 If required, repeat steps **4-7** to create the pattern in a second direction as well.
- 9 Click **OK** to create the pattern.

Circular Patterns

You can use a circular pattern to repeat a feature in a radial direction around a centerline.

To create a circular pattern:

- 1 From the **Feature** menu, select **Pattern > Circular**. The **Circular Pattern** dialog box appears.



- 2 Select the features to be patterned.
- 3 Check the **Pattern Geometry** box if you want to pattern the geometry exactly. See the previous section on Linear Patterns for more information on Pattern Geometry.
- 4 Select the **Circular path center**. An axis or linear edge can be used as the center.
- 5 Specify the number of **Copies** including the original feature.
- 6 Specify the **Angle** in degrees to control the spacing between the copied features.
- 7 If necessary, select **Change direction** to create the pattern in the opposite radial direction.
- 8 Click **OK** to create the pattern.

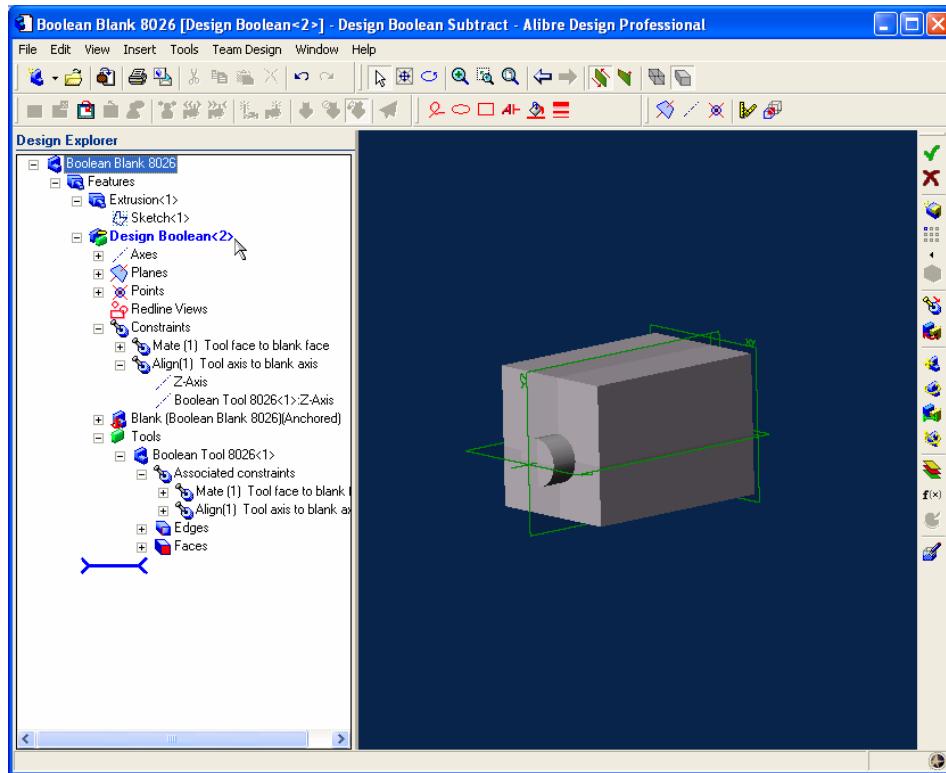
7.15 Design Boolean Features

You can use the Design Boolean Features to model parts for special applications such as packaging and mold design. This is accomplished by using a “tool” or “tools” (a collection of other parts or assemblies) to modify a “blank” (the current part). The creation of a Design Boolean is done in a special workspace called the Design Boolean Editor Environment. The Boolean feature is parametrically related to the tool part. Therefore, if the tool is modified

and saved, the Boolean feature will update upon reopening. You can also modify the dedicated assembly that contains the blank and the tool.

7.15.1 Design Boolean Editor Environment

Upon selecting a Design Boolean Feature, the Blank part workspace transforms into the Design Boolean Editor Environment. This special workspace exists within a part workspace, but resembles an assembly workspace regarding toolbars and options. Inside the editor environment you will create a dedicated assembly that contains the blank part and the tool(s), constrained as required for modifying the design. The editor environment workspace is shown below.



There are two ways to exit the editor environment. You can commit or discard the Boolean feature. Once you choose one of those options, the workspace is transformed back into the

blank part workspace. From the **Edit** menu, select Commit or Discard, or click the appropriate icon:



Commit



Discard

7.15.2 Creating Design Boolean Features

Although the Boolean Unite, Subtract, and Intersect features are different in end result, the steps used to create them are identical. The operations are the same as in set theory:

- Boolean Unite
- will add tool material to the blank part material.

- Boolean Subtract
- will remove tool material from the blank part material.

- Boolean Intersect
- will show the overlap of the tool material and the blank part material.

Before you begin:

You will need to have a part that you wish to modify. This will be your “blank”. In addition, you will need to have an existing part or assembly to use as a “tool”. The tool must be saved to the repository or file system.

To create a Boolean design feature:

- 1 Open the blank part.
- 2 From the **Feature** menu, select **Feature > Boolean > Subtract (or Unite, or Intersect)**. The **Insert Part/Subassembly** dialog box appears.

OR

Select the appropriate tool from the Design Boolean Feature fly-out on the Part



Modeling toolbar.

- 3 Highlight the part or assembly you wish to use for the tool; then click **OK**. The part workspace window transforms into the Design Boolean Editor window. Click in the Editor workspace to place the tool. Click **Finish** when you have the number of copies desired.

NOTE: You can repeat steps 2 and 3 to add as many parts or assemblies as required.

- 4 Constrain the tool(s) to the blank using the assembly constraints available in the Boolean editor workspace.
- 5 Click the **Commit** icon, or from the **Edit** menu, select **Commit** to complete the Boolean feature. The window transforms back into the blank part workspace, and the Boolean feature will be listed as a feature in the design explorer.

NOTE: If a part or assembly in the tool is hidden in the dedicated assembly, it will not participate in the Design Boolean feature until it is unhidden.

7.15.3 Editing Design Boolean Features

The Boolean feature is parametrically related to the tool part or assembly. Therefore, if the tool is modified and saved, the Boolean feature will update upon reopening (if you make changes to the tool inside the assembly context, the changes will automatically update without saving and reopening — see section 10.9.2 for more information on this). You can also modify the dedicated assembly that contains the blank and the tool by repositioning parts as well as adding or removing parts.

To edit a Boolean design feature:

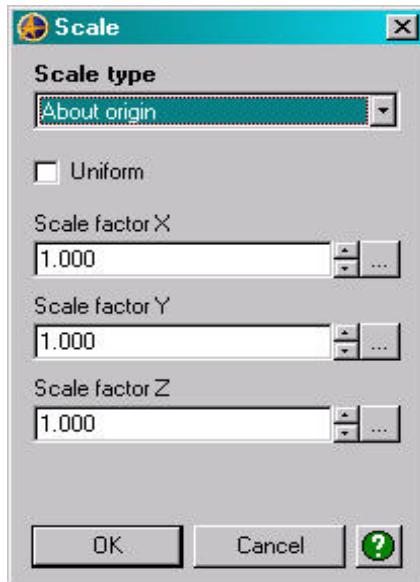
- 1 In the Design Explorer, right-click the Design Boolean feature you wish to modify.
- 2 Select **Edit**. The Design Boolean Editor Environment workspace will appear.
- 3 Make the necessary modifications to the dedicated assembly (remembering that hidden parts and assemblies will not participate in the Design Boolean feature); then from the **File** menu, select **Commit** to apply the changes. The workspace will transform back into the part workspace.

7.16 Scaling Parts

You can increase or decrease the size of a part by using the Scale feature. Note that sheet metal parts cannot be scaled – only non-sheet metal solid parts.

To scale a part:

- 1 From the **Feature** menu, select **Scale**. The Scale dialog box appears.



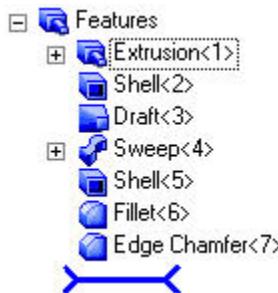
- 2 In **Scale Type**, choose About Origin or About Centroid.
- 3 Check **Uniform** if you want to uniformly scale the part in all 3 directions. If you want a different factor for each direction, leave uniform unchecked and type in the desired values in Scale Factor X, Y, and Z boxes. A scale factor of 1.0 leaves the model unchanged.

A feature is created in the design explorer when the scale operation is performed.

When modifying features before the scale operation, the model will roll back to the original dimensions.

7.17 Managing Features in the Design Explorer

After creating a feature it will be listed in the Design Explorer under the **Features** node. Additional features will be listed in the order in which they were created. This feature order defines the part's construction history.



By right-clicking a feature in the Design Explorer you can:

- **Edit** the features properties
- **SUPPRESS** the feature (refer to [Chapter 9](#) for more information)
- **Rename** the feature
- **Delete** the feature
- Check the **Status** of the feature

8 Sheet Metal Feature Creation and Part Parameters

Sheet Metal Parts are only available in Alibre Design Professional. Like other solid parts, sheet metal parts are modeled by creating features. Features are individual 3D shapes representing common mechanical design elements, like tabs and cuts, which either create material or remove material in a part. Some features, such as tab, require an associated sketch to define the 2D profile of the 3D shape. Other features, such as corner round and corner chamfer, can be created without a sketch and are applied to existing edges and faces.

This chapter describes:

- The sheet metal part modeling interface
- Sheet metal part parameters
- Tab features
- Flanges
- Closed Corners
- Cuts
- Corner Rounds and Chamfers
- Holes
- Unbend and rebend
- Flat patterns
- Catalog Features
- Mirroring Features
- Feature Patterns
- Managing Features in the Design Explorer

8.1 The Sheet Metal Part Modeling Interface

The Sheet Metal Part Modeling toolbar is shown by default on the right side of the workspace. Commonly used modeling tools are accessible on the Part Modeling toolbar.



- Flat Tab** . . . create a flat tab feature
- Flange** . . . create a flange feature
- Close Corner** . . . close a corner between two flanges
- Formed Dimple** . . . create a dimple or drawn cutout
- Cut** . . . create cut feature
- Round Corner** . . . round a corner
- Chamfer Corner** . . . chamfer a corner
- Hole** . . . create a hole feature
- Insert Catalog Feature** . . . insert a saved feature or sketch
- Unbend** . . . unbend one or more flanges
- Rebend** . . . rebend one or more unbent flanges
- Flat Pattern** . . . view the model in a flattened state
- Layers** . . . manage different layers within the part (refer to [Chapter 9](#))
- Equation Editor** . . . open the Equation Editor (refer to [Chapter 4](#))
- Regenerate** . . . regenerate the part to update changes (refer to [Chapter 9](#))

The tools that are accessible on the Sheet Metal Part Modeling toolbar are accessible from the **Feature** menu as well. The Feature menu also contains tools that do not have a corresponding toolbar icon.

Note: These features are all solid modeling features – using them in a sheet metal part may create a model that is incapable of unfolding to a flat pattern. Use with care. Refer to **Chapter 7** for information on creating each of these features.

Features > Boss . . . create extruded, revolved, sweep, or loft boss features

Features > Cut . . . create extruded, revolved, sweep, or loft cut features

Features > Thin Wall Boss . . . create a thin wall extruded, revolved, or sweep boss feature

Features > Thin Wall Cut . . . create a thin wall extruded, revolved, or sweep cut feature

Features > Fillet . . . create a fillet feature

Features > Draft . . . create a draft feature

Mirror . . . mirror a feature about an edge or axis

Pattern > Linear . . . create copies of a feature in a linear pattern

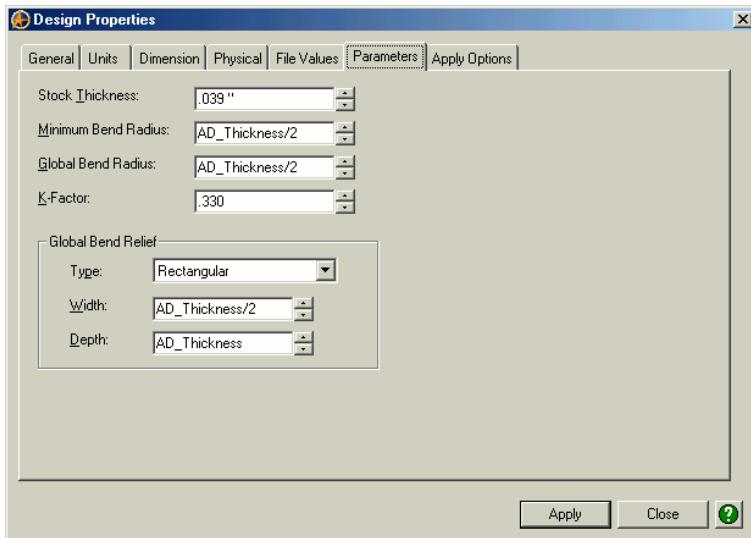
Pattern > Circular . . . create copies of a feature in a circular pattern

Save Catalog Feature ... save a feature to the repository for use in other models

8.2 Sheet Metal Part Parameters

Sheet metal parts have parameters that govern the way the part can be designed. The parameters are used as defaults in many sheet metal features. This allows a flat pattern to be resolved.

- 1 From the **File** Menu, select **Properties**. The Design Properties dialog box appears.
- 2 Click the **Parameters** Tab.



Set your sheet metal material parameters of:

- **Stock Thickness** — The thickness of the sheet of metal used to manufacture the part.
- **Minimum Bend Radius** — The minimum Bend Radius to be allowed in the model. If you attempt to enter a radius smaller, it will automatically go to this minimum. Default Value = thickness / 2.
- **Global Bend Radius** — The default bend radius used in all instances unless otherwise specified during feature creation. Default Value = thickness / 2.
- **K-Factor** — During a forming process, "elongation" of the material occurs — a changing of shape due to the radius pushing material into another location. The "elongation" is called the K-Factor, or bend deduction. K-factors can be determined by using material charts.
- **Global Bend Relief** — A small cut made in the material to prevent the bend radius from causing a distortion in the metal. Type can be **Rectangular** (produces a rectangle shape cut) or **Round** (produces a rectangular cut with a round on the short edge).

Default values:

Type – rectangular

Width – thickness

Depth – thickness*2

3 Choose **Apply**; then **Close**.

The parameters set here are accessible in the Equation Editor (refer to **Chapter 4**).

8.3 Tab

A Tab is the first feature created in a sheet metal part, and inserts flat stock based upon a sketch. This is the only allowed initial feature. Tab thickness is obtained from the parameters set in the sheet metal part parameters. The name of the thickness parameter is “**AD_thickness**”. The AD_thickness value corresponds to the stock thickness parameter specified in the Design Properties Parameters tab.

To create a tab:

- 1 Create a sketch and draw the profile you want to extrude.
- 2 From the **Feature** Menu, select **Tab**. The Tab dialog box appears.



- 3 Select the sketch you want to extrude.
- 4 Check **Reverse** if you want the Tab to project off the other side of the sketch plane from what the preview shows.
- 5 In **Label**, enter a unique name for this feature if desired.
- 6 Click **OK**.

8.4 Flange

A flange is a section of the sheet metal that is bent from the original flat shape. You can add a flange to any straight layer edge of flat stock material. The flange places a bend and a

rectangular (or trapezoidal for taper conditions) tab of flat stock material on the selected edge.

To create a flange:

On the Main Tab:

- 1 From the **Feature** Menu, select **Flange**, or select the **Flange** tool from the Sheet Metal toolbar. The flange dialog box appears.

- 2 In **Edge**, click the edge on the existing tab the flange will be attached to.

- 3 In **Alignment**, choose the inside the tab wall, outside the tab wall, or adjacent to the tab wall condition. Check the **Trim Side Bends** option if needed.

Note: Trim Side Bends can be checked only when the Bend Alignment Type is INSIDE or OUTSIDE. This option is applicable when the layer edge(s) adjacent to the selected edge have flanges. If this is true, then there will be setback part on the flat stock side, which would interfere with the bend on the adjacent edge. The distance by which the side bend is trimmed is equal to the setback of the current flange being made. If there are bends on both sides of the current flange being made then both the bends are trimmed if this option is selected. Note that the side bend may be in the opposite direction also.

- 4 In **Leg**, type the length of your flange, and choose if you want the length measured on the inside of the curve, the outside of the curve, or from the point where the bend ends.

- 5 Choose **Bend Only** if you do not want the flange wall, but just the bend that would be created for it. This option can be used for creating jogs.

- 6 Choose **Match Taper** if you want to specify that the taper of adjacent faces is continued through the flange.

- 7 In **Bend Angle**, type in the angle you want your flange with respect to the face you are adding it to. The bend angle is the "excluded" angle of the bend, measured as shown in the following:



Using a bend angle equal to or greater than 180° will disable inside/outside alignment options.

- 8 In **Bend Radius**, type in the radius for your bend. This will be the inside radius of the bend. The default value here is the global bend radius specified in the

parameters. Check **Reverse Bend** if you want the flange to go the opposite direction from what is showing in the preview.

On the Advanced Tab:

- 1 In **Bend Relief Type**, specify if you want No Bend relief, Rectangular Relief, or Rounded Relief.
- 2 If you choose to have a relief, enter the **Width** and **Depth**. (Default values are taken from the parameters set in the properties dialog box. Reference section 8.2)
- 3 **Bend Allowance – Choose Use K factor:** use the specified K-Factor to calculate allowance for the bend OR choose **Use unfold length:** set the unfold length used when unfolding this bend.
- 4 In **Label**, type in a unique name if desired.
- 5 Click **OK**.

8.5 Closed Corner

You can close corners where two bends meet in sheet metal bodies with appropriate extensions and square corner bend reliefs.

To close a corner:

- 1 From the **Feature** Menu, select **Closed Corner**. The closed corner dialog box appears.
- 2 In **Edge**, click an edge that is part of the corner you want to close.
- 3 In **Join**, choose the method that you want to use to join the two flanges.
- 4 In **Label**, enter a unique name for this feature if desired.
- 5 Click **OK**.

8.6 Dimple

To create a dimple:

- 1 From the **Feature** menu, select **Dimple**. The dimple dialog box appears.
 - 2 In **Sketch**, select the sketch you would like to use to create the dimple.
- Note:** Sketches used to create dimples can contain only 1 closed figure.
- 3 In **Depth**, Enter the depth of the dimple from the start face. The default is 2 times the thickness of the stock material.
 - 4 In **Draft Angle**, enter the angle if needed. Only positive angle values are allowed. If a negative angle is entered, the OK button will not activate.
 - 5 In **Sketch Alignment**, specify if the tool profile sketch is to be applied to the outside of the dimple or the inside, which impacts the offset and draft angles used to create the feature.
 - 6 Select **Cut Out Material** if you want to cut out the bottom of the dimple, resulting in a drawn cutout operation.
 - 7 In **Include Rounding**, you can include automatic fillets for the dimple operation. Die Radius is the user specified radius for external fillets created by the dimple operation. Punch Radius is the user specified radius for internal fillets created by the dimple operation.
 - 8 In **Round Profile Corners**, you can optionally round hard corners in the tool profile sketch.
 - 9 In **Label**, enter a unique name for the dimple feature if desired.
 - 10 Click **OK**.

8.7 Cut

Cutting removes material from a model based on the profile of a sketch. The cut will be punched through a single stock thickness of the sheet metal.

To Create a Cut:

- 1 From the **Feature** menu, select **Cut**. The Cut dialog box appears.
 - 2 In **Sketch**, select the sketch you wish to use for the cut.
- Note:** The sketch used in a cut must be placed on a sheet metal face.
- 3 In **Label**, enter a unique name for the cut feature if desired.
 - 4 Click **OK**.

8.8 Corner Rounds and Chamfers

You can create a round or chamfer on any corner of a sheet metal part.

8.8.1 Rounding a corner

- 1 From the **Feature** Menu, select **Corner Round**. The corner round dialog box appears.
- 2 In **Items to Round**, select the edges you want to round. You may also select faces, which will round all corners adjacent to that face. Any combination of faces and edges may be selected.
- 3 In **Radius**, type in the desired radius for the round. A preview will be shown of the round.
- 4 Click **OK**.

8.8.2 Chamfering a corner

- 1 From the **Feature** Menu, select **Corner Chamfer**. The Corner Chamfer dialog box appears.

- 2 In **Edges/Faces to Chamfer**, select the edges you want to chamfer. You may also select faces, which will chamfer all corners adjacent to that face. Any combination of faces and edges may be selected.
- 3 In **Chamfer Type**, select Distance-Distance, Angle-Distance or Equal-Distance.
- 4 Enter the desired values in the distance fields. A preview will be shown of the chamfer.
- 5 Click **OK**.

8.9 Holes

Alibre Design provides several standard holes. You can insert these standard holes on any planar face. The holes can be inserted to a specific depth ("blind"), through an entire model, or up to an intersection of a face. Holes are always inserted perpendicular to a face.

Refer to section 6.12 for comprehensive information on creating holes.

8.10 Unbend and Rebend

You can unbend then rebend one or more flanges at a time. A feature is added to the feature tree in the Design Explorer, allowing you to insert other features, such as a cut, between an unbend and rebend pair.

8.10.1 Unbending a flange

- 1 From the **Feature** menu, select **Unbend**.
- 2 In **Fixed Face/Edge**, choose a face or an edge to remain fixed throughout the operation.
- 3 In **Bends**, select the bends of the flanges to unbend, or choose **Select All Bends** to unbend all flanges.
- 4 Click **OK**.

8.10.2 Rebending a flange

- 1 From the **Feature** menu, select **Rebend**.
- 2 In **Unbent Bends**, select the bends to rebend or choose **Select All Unbent Bends** to rebend all the unbent flanges.

8.11 Flat Pattern

This feature creates a flat pattern view of the entire model. No modifications can be made while in flattened mode, and there is no editing access. If the model is saved while in flattened mode, the model will revert to normal mode before saving.

Note: When entering a Team Design Session on a sheet metal part, all participants will enter flattened mode.

To create a flat pattern:

- 1 From the **Feature** menu, select **Flat Pattern**. The flattened state will be displayed.
-OR-



- 1 Click the **Flat Pattern** tool.

To leave the flat pattern state:



- 1 Click the **Flat Pattern** tool.
- OR -

- 1 From the **Feature** menu, choose **Flat Pattern** (all other features will be grayed out, since no modifications can be made to the part while in the flattened state).

8.12 Catalog Feature

After features or sketches have been created, they can be cataloged and saved to the repository for use in other part models.

Refer to section 7.13 for comprehensive information on saving and inserting catalog features.

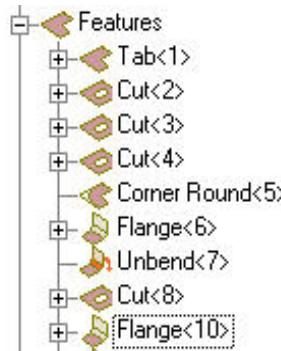
8.13 Copying Existing Features

After features have been created, they can be mirrored or patterned to easily create copies.

Refer to section 7.14 for comprehensive information on creating patterns and mirrors.

8.14 Managing Features in the Design Explorer

After creating a feature it will be listed in the Design Explorer under the **Features** node. Additional features will be listed in the order in which they were created. This feature order defines the part's construction history.



By right-clicking a feature in the Design Explorer you can:

- **Edit** the features properties
- **SUPPRESS** the feature (refer to **Chapter 9** for more information)
- **Rename** the feature
- **Delete** the feature

- Check the **Status** of the feature

9 Working with Parts

Parts are the fundamental component of a 3D design. Detail drawings are created from parts and assemblies are built by integrating multiple parts.

This chapter describes:

- Saving and Opening Parts
- Using the Design Explorer
- Editing, suppressing, and rolling back features
- Using the measurement tool
- Part display options
- Inserting 3D section views
- Using the **Project to Sketch** tool
- Applying color properties
- Calculating part physical properties
- Inserting part annotations
- Working with layers

9.1 Saving and Opening Parts

You save and open parts using the file system or the Repository.

9.1.1 Saving a New Part

- 1 Select the **Save**  tool from the Standard toolbar; or from the **File** menu select **Save**. The **Save** dialog box appears and a blue part icon  **New Part (1)** is displayed next to the new part (that has not been previously saved).

To save the part to the file system:

- 2 Click the **File System** tab.
- 3 Navigate to the file system folder in which you want to save the part.
- 4 To create a new folder at the currently selected location, click **New Folder**.
- 5 If desired, click **Advanced** to enter detailed comments about the part.
- 6 In the **Name** field, type the part name.
- 7 Click **Save** to save the part.

To save the part to the repository:

- 2 Click the **Repository** tab.
- 3 Navigate to the location in which you want to save the part. You can click the plus sign  next to a repository to expand it and display its folders. You can save the part directly under the selected repository or into any of the repository's folders.
- 4 To create a new folder at the currently selected location, click **New Folder**.
- 5 If desired, click **Advanced** to enter detailed comments about the part.
- 6 In the **Name** field, type the part name.
- 7 Select the **Save as type** from the list. The default type is the native **Alibre** format.
- 8 By default, the **Make new version for all** option is on. This option creates a new version of the design each time a save is completed. This option is ignored if the item is being saved for the first time. If you prefer to maintain one version of a design, deselect the **Make new version for all** option.

- 9 Click **Save** to save the part.

9.1.2 Opening a Part

You can open a previously saved part from the Home window, from the Repository or from an open workspace.

To open a part from the Home window or any workspace:

- 1 Select the **Open**  tool from the Standard toolbar; or from the **File** menu select **Open**. The **Open** dialog box appears.
- 2 Using the **Document Browser** embedded in the **Open** dialog, navigate through the repository or the file system to the location of the desired part.
- 3 Select the part from the item list and click **OK**; or double-click the part in the item list.

To open a part from the repository:

- 1 In the **Repository Explorer**, browse to the location the part is stored in. If necessary, click the plus sign  next to a repository to expand it and display the folders within.
- 2 To open the part, double-click the part in the item list; or right-click the part in the item list and select **Open** from the pop-up menu; or select the part in the item list and select the **Open**  tool from the Standard toolbar.

9.2 Using the Design Explorer

The Design Explorer on the left side of a workspace provides an outline of the part's design history and structure. The Design Explorer lists all features, sketches, faces, edges, vertices, planes, axes, points, section views, and redline views associated with a design.

The Design Explorer is displayed by default. You can hide the Design Explorer although it is recommended that you keep it displayed when working. To hide the Design Explorer, from the **View** menu select **Design Explorer**. A check mark next to a menu item indicates the item is currently displayed.

Numerous tasks can be accomplished from the Design Explorer:

- Moving the cursor over an item or selecting an item in the Design Explorer will subsequently highlight or select the corresponding item in the work area.
- Right-click a feature to edit its properties, rename the feature, delete a feature, suppress a feature, and check the status of a feature.
- Drag and drop features in the Design Explorer to reorder them. Changing the feature generation order changes the construction of the part.
- Click the plus or minus signs next to an item to expand or contract an item or item group. For example, click the plus sign next to a feature to see its associated sketch.

9.3 Modifying a Part

You can easily alter a part by editing sketches and features, suppressing features, reordering feature construction, and rolling back features.

9.3.1 Editing Sketches and Features

After the initial creation of a feature, you can always edit the original feature sketch or the defining properties of the feature.

To edit a feature sketch:

- 1 In the Design Explorer, click the plus sign next to the feature to expand it and display its associated sketch.
- 2 Right-click the sketch and select **Edit** from the pop-up menu; or double-click the sketch.

Or

- 1 Select the feature in the work area.
- 2 Right-click and select **Edit Feature Sketch** from the pop-up menu.

The feature sketch appears in sketch mode and you can edit the sketch.

To edit a feature:

- 1 In the Design Explorer, right-click feature and select **Edit** from the pop-up menu.

Or

- 1 Select the feature in the work area.
- 2 Right-click and select **Edit Feature** from the pop-up menu.

The dialog box associated with the feature type appears displaying the original feature properties. You can change the properties as required.

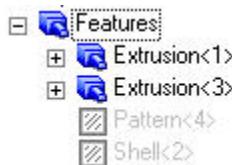
9.3.2 Suppressing Features

You can suppress features to temporarily remove them from a part. Suppressing features hides them in the display and prevents the feature from being used in any other modeling operations.

To suppress a feature:

- 1 Right-click the feature in the Design Explorer and select **Suppress** from the pop-up menu.

The feature becomes hidden in the work area. The feature is listed in gray in the Design Explorer and the **Suppress** icon  is also overlaid on top of the feature icon.



To remove the suppress state, right-click the suppressed feature and select **Suppress** from the pop-up menu.

9.3.3 Reordering Features

You can reorder features in the Design Explorer to alter the sequence in which features are created. Reordering features will change a part's construction.

To reorder features:

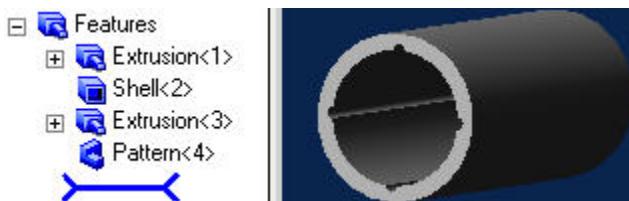
- 1 In the Design Explorer, click and drag a feature up or down in the feature list. When you drag a feature in the Design Explorer the cursor will change.



- 2 Release the mouse button when the feature has been dragged to the appropriate location in the feature list.

The part will regenerate to reflect the new construction order.

Original feature list:



After reordering features:



9.3.4 Rolling Back Features

You can roll back a part to an earlier state in the design. When you roll back to an earlier state, features below the rollback point become inactive. When a part is rolled- back, new features are inserted at the rollback point, not at the end of the feature list.

To roll back features in the Design Explorer:

- 1 In the Design Explorer, move the cursor over the blue **Feature History** line.

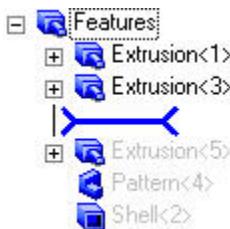
2 Click and drag the Feature History line above the features you temporarily want to disable.

3 Release the mouse button.

Or

1 Double-click the feature listed immediately above the features you want to disable.

The Feature History line moves to the rollback point, the features below the line become gray, and the part is rebuilt to only reflect the features above the Feature History line.



To roll features forward:

1 Drag the **Feature History** line below the feature or features you want to reactivate.

Or

Double-click the feature that you want to reactivate. All inactive features above the feature you double-click will also become active again.

Note: To simultaneously reactivate all disabled features, select the **Regenerate**



tool from the Part Modeling toolbar; or from the **Feature** menu select **Regenerate**; or press **F5** on the keyboard.

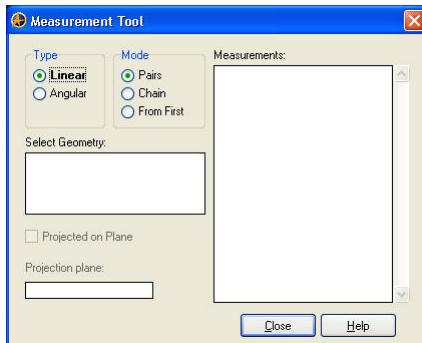
9.4 Using the Measurement Tool

You can use the measurement tool to measure items as well as distances and angles between edges, faces, vertices, planes, axes, and points in a design.

To take a measurement:

- 1 Select the **Measurement**  tool from the Inspection toolbar; or from the **Tools** menu select **Measurement Tool**; or press **Ctrl + M** on the keyboard.

The Measurement Tool dialog box appears.

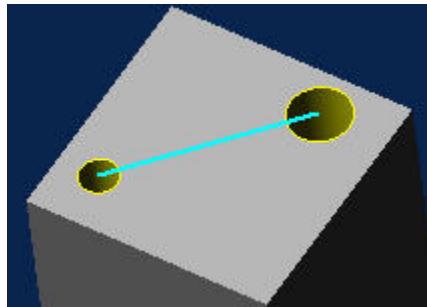


- 2 Select a measurement **Type**, either **Linear** or **Angular**.
- 3 Select a **Mode**:
 - **Pairs**: measurement is taken between two entities, e.g. distance from edge to edge
 - **Chain**: cumulative measurement is taken between sequential selections, e.g. distance along multiple edges
 - **From First**: measurement is taken between two entities, with first selection always remaining constant
- 4 Select an entity to measure or an entity to measure from.

Note: You can also pre-select entities and then open the Measurement Tool dialog box. When pre-selecting, you must hold the Shift key down to select multiple items.

- 5 If applicable, select an entity to measure to.

Note: To measure to or from the center of cylindrical shapes, select the associated cylindrical face.



- 6 Select the **Projected on Plane** option to project the measurement to another plane. Select the **Projection plane**.

A measurement preview line is displayed in the work area and measurement information is displayed in the **Measurements** area in the dialog box.

- 7 New measurements can be taken without closing the dialog box. Simply adjust the **Type** and **Mode** as necessary and select the new entities.
- 8 Click **Close** when finished.

9.5 Part Display Options

You can select from four different display modes to control how a part is displayed.

The default display type is **Shaded**. Other display options include **Wireframe**, **Shaded & Visible Edges**, and **Shaded & All Edges**.

To change the part display:

- 1 From the **View** menu, select **Display** and one of the four options:
 - **Shaded**: displays part in shaded mode, edges are not outlined.
 - **Wireframe**: displays part in wire frame mode, only edges are outlined and displayed.

Note: When viewing the display in wireframe, you can turn the silhouette edges of the model on or off. To do this, from the **View** menu, select **Display**. Check the **Silhouette Edges** option to turn

them on. You can also use the  tool to toggle them on and off.

- **Shaded & Visible Edges:** displays part in shaded mode, only visible edges are outlined.
- **Shaded & All Edges:** displays part in shaded mode, visible as well as hidden edges are outlined.

The part display is updated.

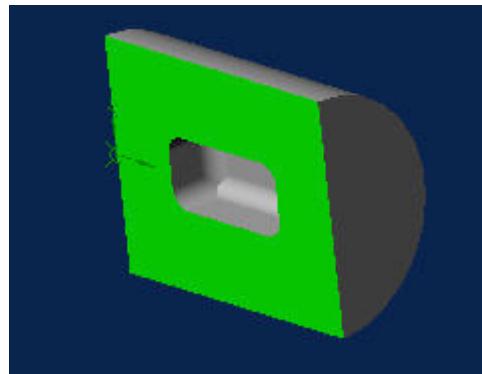
Note: You can quickly change between shaded and wireframe display modes by selecting the **Shaded**  and **Wireframe**  tools from the View toolbar.

9.6 3D Section Views

You can insert 3D section views into a part and subsequently take measurements on the sectioned face or use the sectioned face in modeling operations.

To insert a 3D section view:

- 1 Select the **Insert 3D Section View**  tool from the Inspection toolbar; or from the **Insert** menu select **3D Section View**; or right-click and select **Insert 3D Section View** from the pop-up menu. The **3D Section View** dialog box appears.
- 2 Select a **Slicing Plane**. A reference plane or planar face can be used. The slicing plane points to the section side that will remain visible.
- 3 If necessary, select **Reverse** to change the direction of the section view.
- 4 Specify an **Offset** to create the section view from a specified distance from the slicing plane.
- 5 Click **OK** to create the section view. The section view is also listed in the Design Explorer under the **Section Views** node.



Managing 3D section views:

You can turn section view visibility on and off as well as delete section views from the Design Explorer. To hide or display a section view, right-click the section view in the Design Explorer and select **View** from the pop-up menu. A check mark next to **View** indicates the view is currently displayed.

To delete a section view, right-click the section view in the Design Explorer and select **Delete** from the pop-up menu.

You can insert multiple section views into a part. However, you can only display one section view at a time.

9.7 Part Physical Properties

You can calculate and display the volume, mass, surface area, center of mass, inertial tensor, and principal axes of inertia of a part or assembly.

To set a density value:

- 1 From the **File** menu, select **Properties**. The **Design Properties** dialog box appears.
- 2 Select the **Material** tab.
- 3 In the **Material** field, select the desired material from the drop down menu, or select **Custom** to enter your own density value. (The density is only valid for a part, not an assembly.)
- 4 Click **Apply** and then **Close**.

To calculate physical properties:

- 1 Select the **Physical Properties**  tool from the Inspection toolbar; or from the **Tools** menu select **Physical Properties**. The **Physical Properties** dialog box appears.
- 2 Select an **Accuracy** setting.
- 3 Click **Calculate**. The physical properties are displayed. If necessary, you can copy the physical properties information from the dialog box and paste it into another document. The units displayed for these calculations can be set from the **File** menu by selecting **Properties**, and choosing the **Units** tab.
- 4 Click **Close** to close the dialog box.

9.8 Color Properties

You can add color to a part as well as control a part's reflectivity and opacity.

To apply color properties:

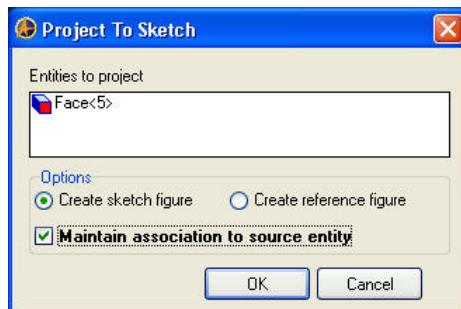
- 1 Right-click in the work area and select **Color Properties** from the pop-up menu; or from the **Edit** menu select **Color Properties**. The **Color Properties** dialog box appears.
- 2 Select the **Color** button to apply a color to the part, or the **Edge Color** button to apply a color to the part edges. The **Color** dialog box appears.
- 3 Select a color from the **Basic colors** area or create a **Custom** color. Click **OK** and the preview will update to reflect your selection.
- 4 Click **OK** to close the Color dialog box.
- 5 To add **Opacity** or **Reflectivity**, slide the controls appropriately. (Less opacity will make the part appear transparent, as glass or clear plastic. More reflectivity will make the part appear shiny, as metal or plastic.)
- 6 Click **OK** to apply the settings.

9.9 Using the Project to Sketch Tool

You can create new sketch or reference figures automatically by projecting existing edges onto a sketch plane.

To use Project to Sketch:

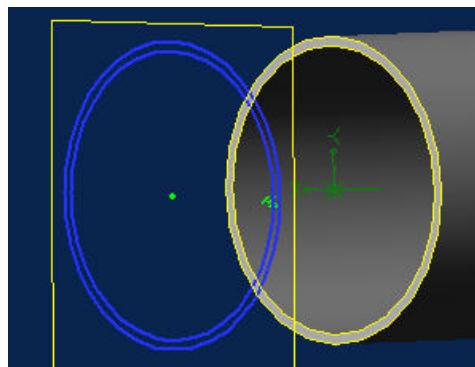
- 1 Select a sketch plane.
- 2 Enter sketch mode.
- 3 Select the **Project to Sketch** tool from the Sketching toolbar; or from the **Sketch** menu select **Project to Sketch**. The **Project to Sketch** dialog box appears.



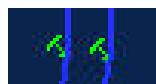
- 4 Select the existing feature edges that you want to project to the sketch plane.

Note: You can select all the edges on a face at once by selecting a face.

- 5 To create a new sketch figure, select the **Create sketch figure** option.
- 6 To create a new reference figure, select the **Create reference figure** option.
- 7 Select the **Maintain association to source entity** option if you want the new sketch or reference figure to reflect any changes made to the originating profile.
- 8 Click **OK** to create the new sketch or reference figure.



If you selected the **Maintain association to source entity** option, project-to-sketch constraint symbols are displayed on the new figures to indicate that they are constrained to the originating profile.



You will not be able to place dimensions on figures that were created with the Project to Sketch **Maintain association to source entity** option. You can, however, delete the constraints and subsequently add driving dimensions to the figure.

9.10 Layers

Layers can be used to create multiple design variations of a part or assembly. You can, for example, show or hide a specific feature by moving it to a separate layer and then showing or hiding that layer. Another example would involve creating a layer specifically for annotations and notes.

To add a layer:

- 1 Select the **Layer**  tool from the Part Modeling toolbar; or from the **Tools** menu select **Layers**. The **Layers** dialog box appears.
- 2 Click **Add**. The **Add Layer** dialog box appears.
- 3 Specify a layer **Name** and enter a custom **Comment** if necessary.
- 4 Click **OK** to create the new layer.

- 5 To set a layer as the current layer, select the layer from the list.
- 6 Click the **Make Current Layer** button.
- 7 Click **OK**. The new layer becomes the current layer.

Any features created after a new layer is set, will be assigned to that layer.

To turn layers on and off:

- 1 Open the Layers dialog box.
- 2 To hide or unhide a layer, click the check box next to the layer name. A check mark indicates the layer is currently being displayed.
- 3 Click **OK** to apply the changes.

9.11 Spreadsheet Driven Designs

You can create variations of a design by using a spreadsheet of parameters. Additionally, a single spreadsheet of parameters can be shared by multiple designs. The spreadsheet must be created in Microsoft Excel; both Excel 2000 and Excel XP are supported.

Before you can drive a design with a spreadsheet of parameters, you must first set up Excel by installing the Alibre Design Add-In in Microsoft Excel. Next, you may either create a design with a desired set of dimensions, or create a spreadsheet first with a set of varying parameters for those dimensions. Then you are ready to drive the design with the spreadsheet.

9.11.1 Setting Up Excel to Drive Designs

Before you can drive a design with a spreadsheet of parameters, you must first install the Alibre Design Add-In to Microsoft Excel.

To set up Excel:

- 1 Launch Alibre Design.
- 2 Launch Microsoft Excel.
- 3 From the **Tools** menu in Excel, select **Add-Ins**. The Add-Ins dialog box appears.

- 4 Click Browse. The Browse dialog box appears.
- 5 Browse to **Program Files > Alibre Design** on your PC's local C:/ drive and locate the file named "Alibre Design Add-In.xla." (This is the default location for Alibre Design files at installation.)
- 6 Click the file name.
- 7 Click **OK**. The Alibre Design Add-In appears in the Add-Ins dialog box with a checkmark next to it.
- 8 Click **OK**. In Excel, a new item appears in the Tools menu: Alibre Design Add-In > Control Parameters.

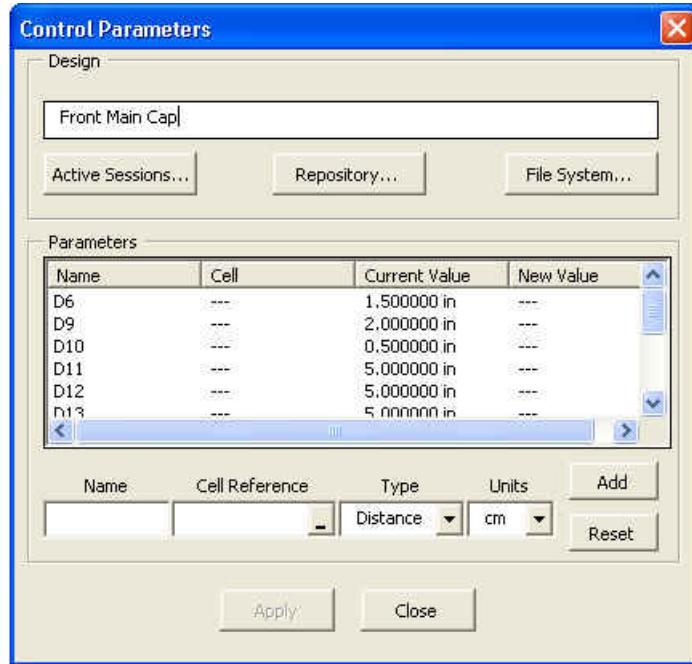
9.11.2 Driving Designs by Spreadsheet

You can use Microsoft Excel spreadsheets to store and manage design information for use in driving one or more designs in Alibre Design. You may either set up the spreadsheet first, or design the part first.

TIP: You must go through the steps in section 9.11.1 to set up Excel before you can drive a design.

To drive a design by spreadsheet:

- 1 Launch Alibre Design.
- 2 Open the desired part.
- 3 Launch Microsoft Excel.
- 4 Save the workbook (spreadsheet).
- 5 Enter the desired settings for the parameters. For each parameter, enter a value, type, and units. You may also enter a name and comments.
- 6 From the **Tools** menu in Excel, select **Alibre Design Add-In > Control Parameters**. The Control Parameters dialog box appears. The part name is displayed in the Design field and its dimensions appear in the parameters table.



- 7 In **Parameters**, click one of the parameters. Its name appears in the Name field.
- 8 For **Cell Reference**, click the field then click the cell on the Excel spreadsheet that the parameter should reference.
 - OR -
Click the button. The cell reference box appears. Click the desired spreadsheet cell; then click the button to return to the Control Parameters dialog box.
- 9 In **Type**, select the parameter type: distance, angle or count.
- 10 In **Units**, set the desired units of measurement (for distance and angle parameters).
- 11 Click **Modify**.
- 12 To add a new parameter, click **Reset**. The entry fields are cleared. In **Name**, type a name for the new parameter then follow steps 7-11 above.

- 13 Click **Close**. The open part in Alibre Design is modified to the new parameters.

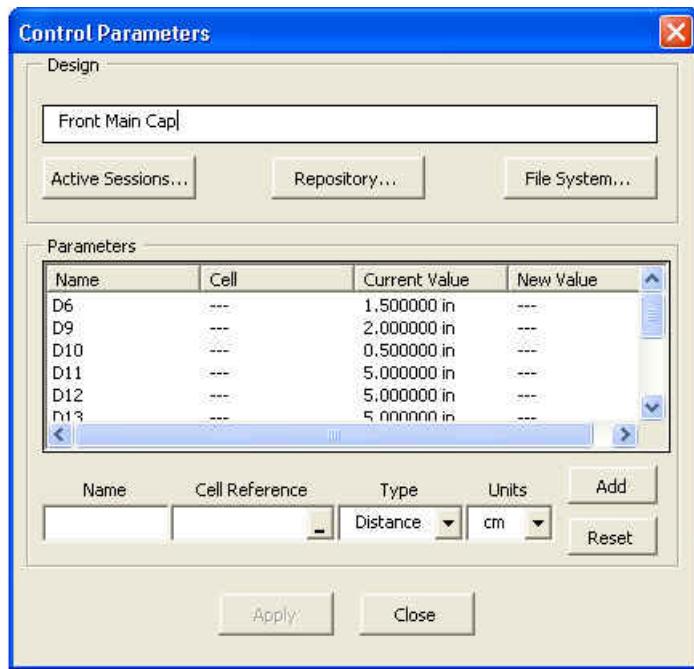
9.11.3 Modifying Spreadsheet Driven Parameters

Modify the parameters for spreadsheet driven designs directly in Microsoft Excel.

To modify spreadsheet driven parameters:

- 1 Launch Alibre Design.
- 2 Open the part that is linked to the spreadsheet.
- 3 Launch Microsoft Excel.
- 4 Open the spreadsheet of prepared parameters.
- 5 From the **Tools** menu in Excel, select **Alibre Design Add-In > Control Parameters**. The Control Parameters dialog box appears. (You may be prompted to save the workbook – the Excel file – before using it to control parameters.) The name of the open part is displayed in the Design field and its dimensions appear in the Parameters table.

TIP: IF several parts are open in Alibre Design, click **Active Sessions** in the Control Parameters dialog box. The Active Sessions dialog box appears. Select the desired part and click **OK**. Alternatively, you can click **Repository** or **File System** to find the desired design.



- 6 In **Parameters**, click of the parameters. Its name appears in the Name field.
- 7 For **Cell Reference**, click the field then click the cell on the Excel spreadsheet that the parameter should reference.

– OR –



Click the button. The cell reference box appears. Click the desired spreadsheet cell; then click the button to return to the Control Parameters dialog box.

Note: If the spreadsheet is moved, or any folders are renamed, the link from the cell to the part dimension will be broken. To reestablish the link, use the Equation Editor dialog box to re-link the spreadsheet to the part. (Reference section 9.11.4).

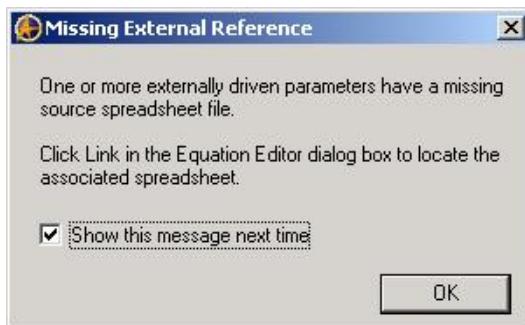
- 8 In **Type**, select the parameter type: distance, angle, or count.
- 9 In **Units**, set the desired units of measurement (for distance and angle parameters).

10 Click **Modify.**

- 11 To add a new parameter, click **Reset**. The entry fields are cleared. In **Name**, type a name for the new parameter; then follow steps 6-10 above.
- 12 Click **Close**. The open part in Alibre Design is modified to the new parameters.

9.11.4 Re-linking a Spreadsheet to a Part

If a spreadsheet file used in driving a design is moved or and folders are renamed in the path to their locations, their links to the parts they drive will be broken. The next time you open the Equation Editor dialog box, this warning will appear.



To reestablish the links between spreadsheets and parts, use the link button in the Equation Editor dialog box.

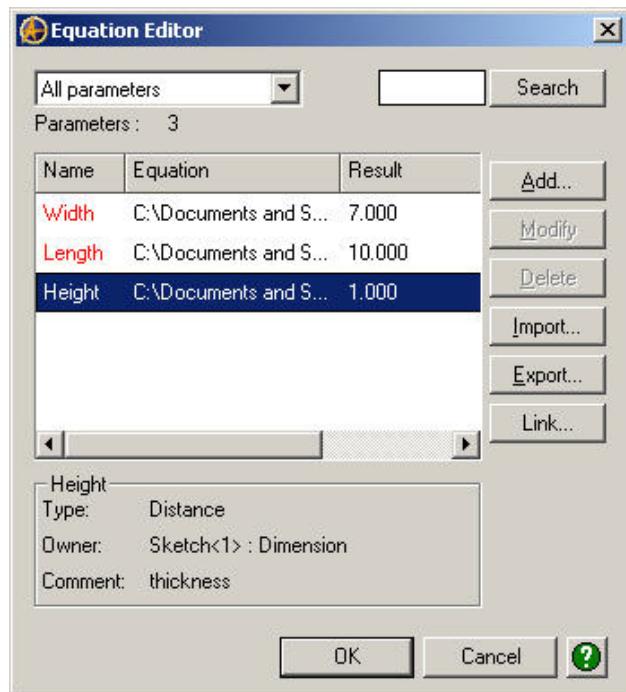
To reestablish the link between a part and a spreadsheet

- 1 With the affected part open, click the **Equation Editor** tool.

- OR -

From the **Tools** menu, select **Equation Editor**. The Missing External Reference dialog box appears.

- 2 Click **OK**. The Equation Editor dialog box appears.
- 3 The affected parameters are displayed in red.



- 4 Click an affected parameter and click **Link**. The Link Parameter dialog box appears.



- 5 Click **Windows Folders**; then **Browse** if the associated spreadsheet is in a Windows folder. Click **Repository**; then **Browse** if the associated spreadsheet is in a repository. The Browse for Spreadsheet File dialog box opens.

- 6 In **Look In**, browse to the location of the spreadsheet.
- 7 Click the spreadsheet's file name to select it.
- 8 Click **Open**. The new file path to the spreadsheet appears in the browse field.
- 9 Click **Apply to all such parameters** to apply the changes to all parameters driven by the same spreadsheet.
- 10 Click **OK**.
- 11 Repeat steps 4-10 if any affected parameters are driven by a separate spreadsheet.

9.12 Printing 3D Models

You can print the images from your 3D workspace. The print function will print all that is displayed, so you may want to hide planes, axes, and other items.

To print:

- 1 From the File menu, select **Print**.
- 2 If you desire to print using a white background, check the option **Use white background**.
- 3 Choose the printer and the number of copies, and click **OK**.

9.13 Annotations

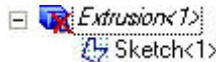
You can insert 3D annotations in a part and assembly workspaces. The following annotation types are supported in 3D workspaces:

- Notes
- Datums
- Datum Targets

- Feature Control Frames
- Surface Finishes
- Weld Symbols
- The methods used to insert annotations for drawing and model workspaces are the same. Refer to section 11.7 for detailed information related to inserting annotations.

9.14 Troubleshooting Failed Features

A feature can fail to generate properly due to numerous reasons. If a feature fails to generate correctly, a message is generated after the initial creation and a red X is displayed on the design icon in the Design Explorer, as illustrated below.



Common causes of feature generation failure include:

- The sketch used to define the feature profile contains open or overlapping figures (e.g. an extrude boss feature fails if the profile sketch contains any open ends)
- Improperly specifying the feature parameters (e.g. interchanging the path sketch and profile sketch involved in a sweep boss)
- Modifying an upstream feature causes a downstream feature to fail

To troubleshoot a failed feature:

- Right-click the feature and select **Status** from the pop-up menu. A dialog box appears containing information related to the cause of the feature failure.
- Right-click the feature and select **Edit**. The feature properties dialog box appears. Modify the conditions if applicable.
- If the failure is a result of an incorrect sketch, edit the sketch and correct accordingly.

9.15 Viewing Constituents

You can view the constituents of Alibre Design Parts, Sheetmetal Parts, Assemblies, Drawings, and Bills of Material.

To view constituents in the Repository:

- 1 In the **Repository Explorer**, browse to the location the item is stored in.
- 2 Right-click the item and select **Constituents** from the pop-up menu; or highlight the item and select **Constituents** from the **View** menu. The **Constituents** dialog box appears, showing all items that are related to the selected item.

To view constituents in the Windows File System:

- 1 In the Alibre Design Home Window, from the **Tools** menu, select **Show Constituents**. The **Show Constituents** dialog box appears.
- 2 Browse to locate the item in the dialog.
- 3 Click **Open**. The **Constituents** dialog box appears, showing all items that are related to the selected item.

-OR-

- 1 In the Windows File System, right-click an Alibre Design file and select **Constituents**. The **Constituents** dialog box appears, showing all items that are related to the selected item.

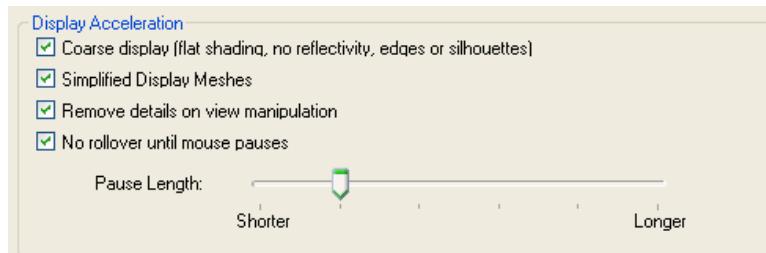
9.16 Display Acceleration

You can speed up the processing of large designs by entering Display Acceleration mode. This is achieved by simplifying the design according to user-defined options. This option can be used in designs (including parts, assemblies, sheetmetal parts, exploded assemblies, and Design Booleans), but not in drawings or BOMs. Some menu items will be unavailable while in Display Acceleration.

To set preferences for Display Acceleration:

- 1 In a part or assembly workspace, from the **File** menu, select **Properties**.

2 Select the **Display tab.**



Check your desired options for Display Acceleration:

- **Coarse Display** – Flat shading will be selected, no reflectivity will be used, visible and silhouette edges will not be shown.
- **Simplified Display Meshes** – The complexity of meshes will be reduced, which will reduce the visual precision of the parts.
- **Remove Details on View Manipulation** – During rotate, pan, and zoom operations, some faces and small parts will be left out of the display. These will be returned to the display after the operation is completed.
- **No rollover until mouse pauses** – If this option is checked, the you will not see any items highlight as you move the cursor until the mouse has paused for the designated amount of time. At that time, whatever item the cursor is paused over will highlight.

3 Choose **Apply, then **Close**.**

To enter Display Acceleration:

From the **View** menu, select **Display Acceleration**.

- OR-

Select the **Display Acceleration** tool  from the View Toolbar.

To exit Display Acceleration:

From the **View** menu, select **Display Acceleration**.

- OR-

Select the **Display Acceleration** tool  from the View Toolbar.

10 Assembly Design

You can create multi-part assemblies of varying function and complexity.

This chapter describes:

- Assembly design methodology
- Assembly design interface
- Assembly constraints
- Designing and editing parts in the context of an assembly
- Moving and rotating parts
- Checking for interferences between parts
- Hiding parts in an assembly
- Creating an exploded assembly view

10.1 Assembly Design Methodology

You can use two distinct assembly design methods, or a combination of both. The first method, often referred to as **bottom-up design**, involves creating each assembly part in an individual part workspace. After the parts have all been individually modeled, you can then insert them into an assembly workspace, and subsequently position and mate them correctly by inserting assembly constraints.

The second method, often referred to as **top-down design**, involves creating all the assembly parts in the assembly workspace. Using this method enables you to design parts while referencing other assembly parts.

Both methods have disadvantages and advantages. The bottom-up design methodology is perhaps the simpler of the two and enables you to manage the design more efficiently. The top-down design is somewhat more complex, but is valuable when the design of one part is heavily dependent on other parts.

You can also use a combination of the bottom-up and top-down design methods.

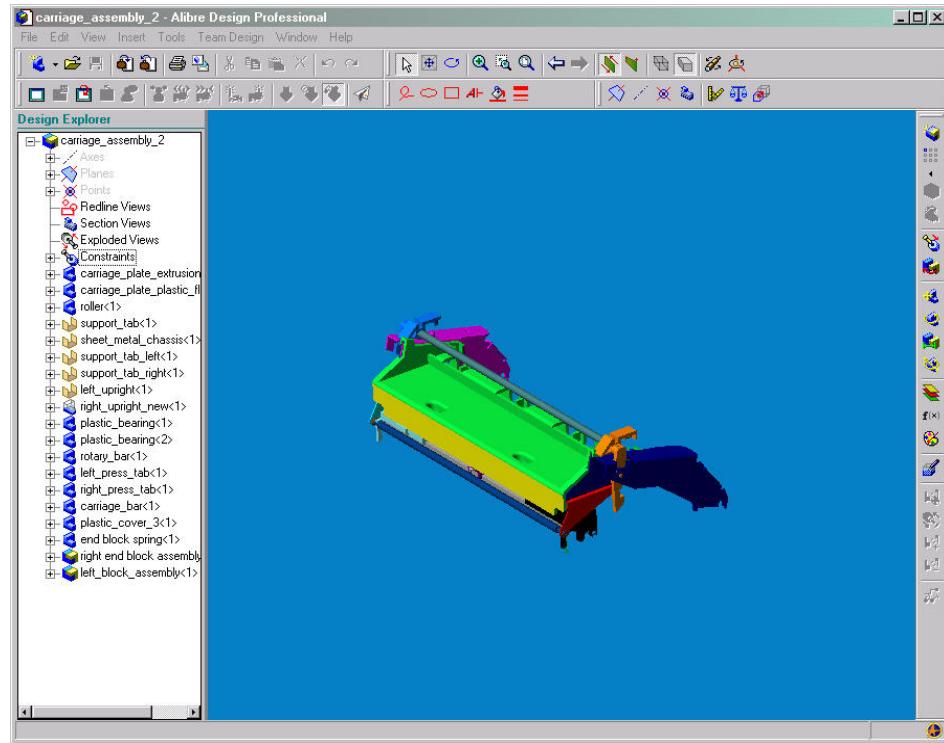
10.2 The Assembly Design Interface

Assemblies are designed in assembly workspaces. An assembly can be comprised of parts and other assemblies, referred to as **subassemblies**. The parts and subassemblies that constitute an assembly are referred to as **constituents**.

A typical assembly could have the following structure:

- **Top-Level Assembly**
 - Subassembly A
 - Part A1
 - Part A2
 - Part A3
 - Subassembly B
 - Part B1
 - Part B2
 - Part B3
 - Part C
 - Part D
 - Part E

Chapter 10 – Assembly Design



Assembly Workspace

The assembly workspace looks similar to the part workspace. However, the Sketching and Part Modeling toolbars are not displayed in the assembly workspace. Instead, the **Assembly Modeling** toolbar is displayed by default on the right side of the workspace.



Insert Part/Subassembly . . . insert a part into the assembly

Insert Pattern . . . pattern a part in the assembly

Insert Duplicate . . . insert a copy of a part in the assembly

Edit Part/Subassembly . . . edit a part or subassembly

Insert Assembly Constraint . . . manually insert an assembly constraint

Auto Constrain Mode . . . place mate and align constraints automatically

Move Part . . . move an individual part or parts

Rotate Part . . . rotate an individual part or parts

Precise Placement . . . position a part precisely

Minimum Motion Mode . . . use in conjunction with Rotate or Move to localize motion to selected part only.

Layers . . . refer to **Chapter 9**

Equation Editor . . . refer to **Chapter 4**

Color Properties . . . add color properties to parts

Regenerate . . . updates assembly constraints.

In assembly workspaces, the top-level assembly is listed first in the Design Explorer. Parts and subassemblies are listed at the bottom in the order in which they are inserted or created. The assembly  icon signifies an assembly or subassembly item. The part  icon signifies a part item.

The Design Explorer also lists the assembly reference geometry, assembly constraints, redline views, section views, as well as the faces and edges associated with each part.

You can click the plus  sign next to an item to expand it and see its associated details. For example, you will see a subassembly's constituents if you expand it.



You can insert the same part or subassembly multiple times into an assembly. When inserting duplicate parts or subassemblies, the item will be listed in the Design Explorer with its original name followed by a numeric label to indicate how many instances have been inserted, e.g. **Part1<1>**, **Part1<2>**, **Part1<3>**, etc.

10.3 Assembly Basics

You can create an assembly by inserting parts you have already designed, inserting imported parts, or designing new parts in the context of the assembly.

10.3.1 Opening a New Assembly and Inserting Existing Parts

- 1 Open a new assembly workspace. The assembly workspace and the **Insert Part/Subassembly** dialog box appear.
- 2 In the **Insert Part/Subassembly** dialog box, select the part to be inserted into the assembly.

Repository Tab:

- This tab is only accessible if you have a version of Alibre Design that enables Repositories.
- Press the Ctrl key as you select multiple components.
- You can select multiple components only in the same repository and folder.
- You can select and insert parts from any repository you have access to.

File System Tab:

- Select the drive you wish to browse through from the Drives drop down list.

- Select the folder the component is located in. The Directory field will update.
- If needed, you can use the Browse button to access non-mapped network locations.
- Select the components to insert into the assembly. Press the Ctrl key as you select multiple components. To select a series of items, hold the **Shift** key and select the first and last parts in the series.

Note: To start with a blank assembly workspace, click **Cancel** on the **Insert Part/Subassembly** dialog box.

- 3 Click **OK** to insert the part. A preview of the part(s) appears in the assembly workspace. The **Inserting** dialog box also appears.
- 4 Move the cursor to move the part(s) if necessary.
- 5 Click once to place the part(s) in the workspace.
- 6 If necessary, continue to click to insert duplicates.
- 7 Press **Esc** or click **Finish** in the **Inserting** dialog box to complete the insertion.

10.3.2 Anchored Parts

A part's position in an assembly can be fixed by **anchoring** the part in the work area. An

anchored part cannot be moved. When a part is anchored, the **Anchor**  icon is displayed on the part in the Design Explorer. You can anchor any part, as well as remove the anchor state from a part at any time.

To add or remove the anchor state:

- 1 Right-click a part in the Design Explorer and select **Anchor Part** from the pop-up menu.

10.3.3 Inserting an Existing Design Into an Open Assembly

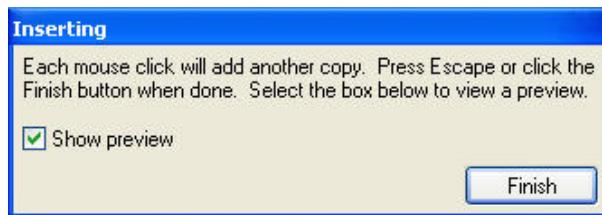
You can insert parts or sub-assemblies any time after an assembly has been opened.

To insert an existing design into an assembly:

- 1 From the **Insert** menu, select **Part/Subassembly**; or right-click in the work area and select **Insert Part/Subassembly** from the pop-up menu; or press **Ctrl + Shift + I** on the keyboard.

The **Insert Part/Subassembly** dialog box appears.

- 2 In the **Insert Part/Subassembly** dialog box, select a design to be inserted into the assembly. You can also select multiple items to be inserted simultaneously. Hold the **Ctrl** key when selecting multiple items.
- 3 Click **OK** to insert the design. A preview of the design(s) appears in the assembly workspace. The **Inserting** dialog box also appears.



- 4 To insert the design so that the design's origin is initially coincident with the assembly's origin, unselect the **Show preview** option.
- 5 If the **Show preview** option is left on, you can move the cursor to position the design(s) at any location if necessary.
- 6 Click once to place the design(s) in the workspace.
- 7 If necessary, continue to click to insert duplicates.
- 8 Press **Esc** or click **Finish** in the **Inserting** dialog box to complete the insertion.

10.3.4 Selecting Parts in the Assembly

You can select individual parts in the assembly from the Design Explorer or the work area. When you move your cursor over a part in the Design Explorer, the part is highlighted in the work area. When you select a part in the Design Explorer, the part is selected and highlighted in the work area, and vice versa.

To select a part in the work area:

- 1 Hold the **Ctrl** key and move the cursor over the part you want to select. The part highlights and the cursor changes to .
- 2 Click to select the part. The part is highlighted in the work area as well as the Design Explorer.



10.3.5 Inserting a Duplicate Design Into an Open Assembly

You can insert copies of a part or subassembly that you have already inserted into an assembly.

To insert duplicates into an assembly:

- 1 Select the design you want to insert a duplicate of. You can select the design in the Design Explorer or work area.
- 2 From the **Insert** menu select **Duplicate**. A preview of the part appears in the assembly workspace. The **Duplicating** dialog box also appears.
- 3 Move the cursor to move the design if necessary.
- 4 Click once to place the design in the workspace.
- 5 If necessary, continue to click to insert additional duplicates.
- 6 Press **Esc** or click **Finish** in the **Duplicating** dialog box to complete the insertion.
- 7 The duplicate appears and is listed in the Design Explorer.



10.3.6 Inserting a Pattern of Parts in an Assembly

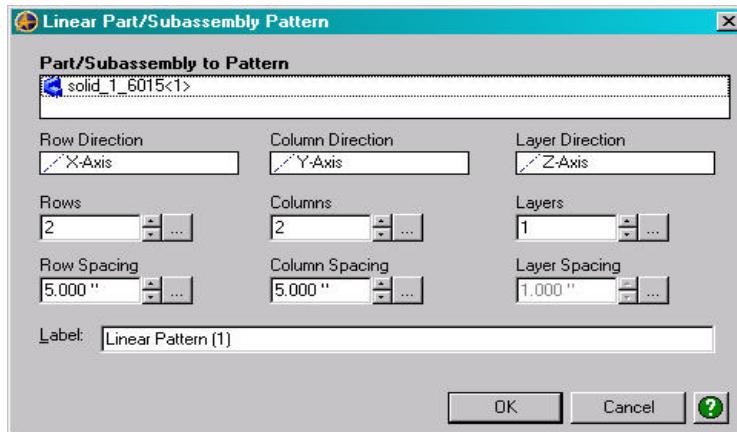
You can pattern a part or subassembly that you have already inserted into an assembly.

Linear Pattern

You can use a linear pattern to repeat a part in one, two, or three linear directions.

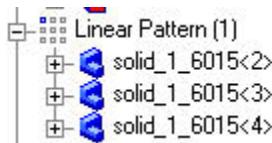
To create a linear pattern of a part in an assembly:

- 1 From the **Insert** menu select **Part/Subassembly Pattern > Linear**. The **Linear Part/Subassembly Pattern** dialog box appears.



- 2 In **Part/Subassembly to Pattern**, select the part you wish to pattern.
- 3 In **Row Direction**, select an edge, axis, or face that can be used to define an axis (as on a cone or cylinder) to set the direction for the row.
- 4 In **Column Direction**, select an edge, axis, or face that can be used to define an axis (as on a cone or cylinder) to set the direction for the column.
- 5 In **Layer Direction**, select an edge, axis, or face that can be used to define an axis (as on a cone or cylinder) to set the direction for the layer.
- 6 Enter the number of copies you want in each direction, including the original. A value of 1 will make no additional copies in that direction.
- 7 Enter the distance you want between each copy in the spacing fields.
- 8 In **Label**, enter a unique name for the pattern.
- 9 Choose **OK**.

The linear pattern will appear in the Design Explorer, with the parts in the pattern listed under it in the tree. These patterned parts can not be edited or moved, however, any of the patterned parts can be deleted by right-clicking them in the Design Explorer and choosing **Delete**.

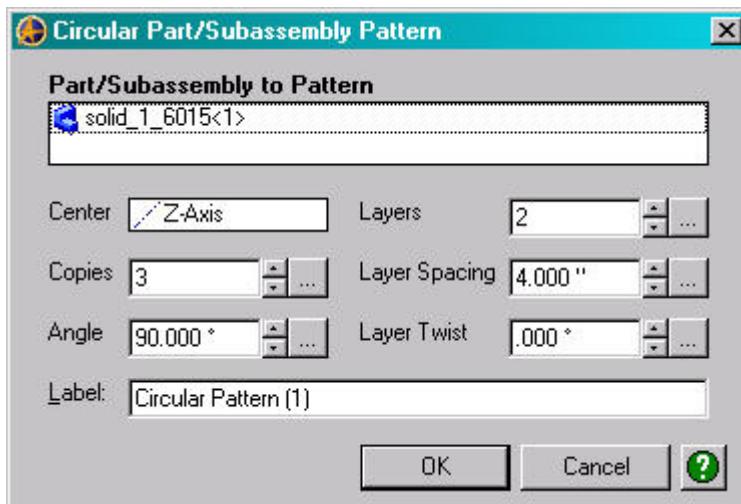


Circular Pattern

You can use a circular pattern to repeat a part in a radial direction around a centerline.

To create a circular pattern of a part in an assembly:

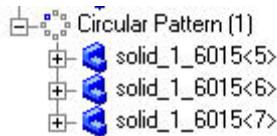
- 1 From the **Insert** menu select **Part/Subassembly Pattern > Circular**. The **Circular Part/Subassembly Pattern** dialog box appears.



- 2 In **Part/Subassembly to Pattern**, select the part you wish to pattern.
- 3 In **Center**, select the axis you want the parts to be patterned around from the Design Explorer.
- 4 In **Copies**, enter the number of copies you want, including the original.

- 5 In **Angle**, set the angle value.
- 6 In **Layers**, Enter the number of layers you want in the direction of your centerline. The same number of copies will be created on each layer.
- 7 In **Layer Spacing**, set the distance between each layer
- 8 In **Layer Twist**, set an angle to rotate the layers with respect to each other, if desired.
- 9 In **Label**, enter a unique name for the pattern
- 10 Click **OK**.

The circular pattern will appear in the Design Explorer, with the parts in the pattern listed under it in the tree. These patterned parts can not be edited or moved, however, any of the patterned parts can be deleted by right-clicking them in the Design Explorer and choosing **Delete**.



10.3.7 Moving and Rotating Parts Freely

You can move or rotate a part freely as long as it is not anchored or constrained in such a way that limits movement.

To move a part:

- 1 Hold the **Ctrl** key, select a part in the work area, and drag the cursor. The part moves as you drag the cursor. You can release the **Ctrl** key after the part begins to move.

Or

- 1 Select the **Move Part**  tool from the Assembly Modeling toolbar; or from the **Tools** menu select **Move Part**.
- 2 Move the cursor over the part you want to move.
- 3 Click and drag the cursor to move the part.

To rotate a part:

- 1 Move the cursor over the part to rotate.
- 2 Hold the **Ctrl** key, select a part in the work area holding both mouse buttons, and drag the cursor. The part rotates as you drag the cursor. You can release the **Ctrl** key after the part begins to rotate.

Or

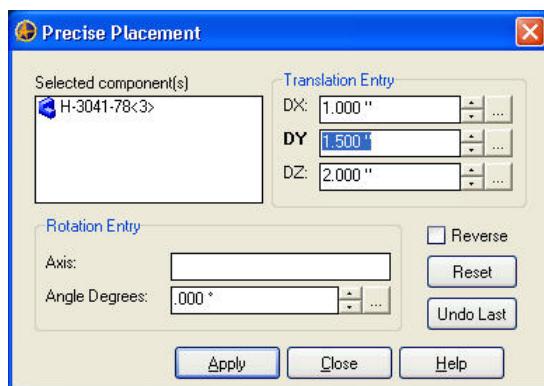
- 1 Select the **Rotate Part**  tool from the Assembly Modeling toolbar; or from the **Tools** menu select **Rotate Part**.
- 2 Move the cursor over the part you want to rotate.
- 3 Click and drag the cursor to rotate the part.

10.3.8 Moving and Rotating Parts Precisely

You can move or rotate a part precisely if necessary as long as a part is not anchored or constrained in such a way that limits movement.

To move or rotate a part a precise distance or angle:

- 1 Select the **Precise Placement**  tool from the Assembly Modeling toolbar; or from the **Tools** menu select **Precise Placement**. The **Precise Placement** dialog box appears.



- 2 Select the part to move or rotate.

- 3 To move a part, in the **Translation Entry** area, specify the relative translation distance by entering values for **DX**, **DY**, and **DZ**.

To rotate a part, in the **Rotation Entry** area, select the rotation **Axis** and specify the relative rotation **Angle** in degrees. The rotation axis can be an axis or linear edge on a part.

- 4 Click **Apply** to move or rotate the part.
- 5 Click **Close** to close the dialog box.

10.3.9 Moving Parts to Articulate Assembly Physical Motion

To position a part or subassembly

- 1 Hold the **Ctrl** key on the keyboard and click the part or subassembly. The item is highlighted.
- 2 While holding the **Ctrl** key and left-mouse button down, drag the part or subassembly to the new position. You can release the **Ctrl** key after the part/subassembly begins to move.

-OR-

- 1 From the **Tools** menu, select **Move Part**.
- 2 Click the **Minimum Motion**  tool on the Assembly Modeling toolbar
- 3 Click and drag the part or subassembly. The part or subassembly moves a minimum number of parts.

10.3.10 Hiding a Part

You can hide individual parts in an assembly.

- 1 Right-click a part in the Design Explorer and select **Hide** from the pop-up menu; or select a part in the work area (**Ctrl + click**) and right-click and select **Hide** from the pop-up menu. The part is hidden in the work area and is listed in gray in the Design Explorer.



To unhide the part, right-click the part in the Design Explorer and select **Hide** again.

10.3.11 Changing a Part's Display

You can control the display of an individual part.

To change a part's display:

- 1 Right-click a part in the Design Explorer and select **Wireframe**, **Shaded**, **Shaded & Visible Edges**, or **Shaded & All Edges** from the pop-up menu. The part display change is reflected in the work area.

To change the display back to **Shaded** (default), right-click the part in the Design Explorer and select **Shaded**.

10.3.12 Applying Color Properties to a Part

You can apply different color properties to each part in an assembly.

To apply color properties to a part:

- 1 Select a part either in the Design Explorer or the work area.
- 2 Select the **Color Properties**  tool from the Part Modeling toolbar; or right-click in the work area and select **Color Properties** from the pop-up menu; or from the **Edit** menu select **Color <Part Name> Properties**. The **Color Properties** dialog box appears.
- 3 Select a color and set the **Reflectivity** and **Opacity** levels as necessary.
- 4 Click **OK** to apply the properties.

Note: You can apply the same color properties to the entire assembly. Select the top-level assembly in the Design Explorer, or from the **Edit** menu select **Select All**. Follow steps 1-4 above to apply the color properties.

10.3.13 Checking Part Physical Properties

You can check physical part properties on an individual part basis or the assembly basis.

To check part/assembly physical properties:

- 1** Right-click a part or assembly in the Design Explorer and select **Properties** from the pop-up menu; or select a part in the Design Explorer or work area, right-click and select **Properties** from the pop-up menu.

The **Measurement Tool** dialog box appears and lists the physical properties for the selection.

- 2** Click **Close** on the dialog box when finished viewing the properties.

10.3.14 Viewing Part Reference Geometry

You can view the reference geometry of a part while in an assembly. These features can then be used to constrain the part to the assembly if desired.

To view part reference geometry:

- 1** Right-click the part in the Design Explorer
- 2** Choose **Show Reference Geometry**. The part reference features will appear in the design explorer and in the model window.

To hide part reference geometry

- 1** Right-click the part in the Design Explorer
- 2** Choose **Show Reference Geometry** (This will be checked if reference geometry is shown. Choosing it again will uncheck the option, hiding the reference geometry).

10.4 Assembly Constraints

You can insert assembly constraints to precisely position and mate parts with respect to each other in an assembly. Assembly constraints also dictate how parts move or rotate with respect to other parts.

A part is initially unconstrained and has six degrees of freedom when first inserted into an assembly. You can move or rotate an unconstrained part in any direction. As you place assembly constraints on a part, the degrees of freedom are reduced and you begin to limit how a part can be moved or rotated. A fully constrained part has zero degrees of freedom and its movement and/or rotation depend on the movement/rotation of the part or parts it is constrained to.

10.4.1 Assembly Constraint Types

You can use five different types of assembly constraints: **mate**, **orient**, **angle**, **align**, and **tangent**. Each constraint type is valid for specific combinations of items. You can apply constraints to the following items:

- Reference planes and axes
- Linear edges
- Planar faces
- Cylindrical faces
- Spherical, conical, and toroidal faces

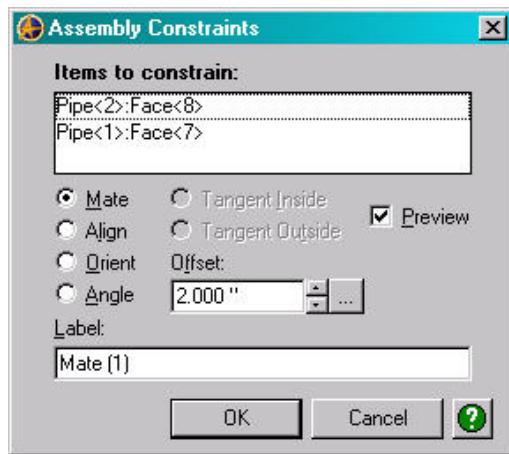
You will always select two items when applying a constraint. The following table summarizes which constraint types can be applied to various combinations of items.

	Plane	Cylinder	Line	Sphere
Plane (planar face or reference plane)	Mate Orient Angle Align			
Cylinder (cylindrical face)	Tangent Align	Align Tangent Orient		
Line (linear edge or axis)	Align Orient	Align Tangent	Align Orient	
Sphere (spherical face)	Tangent	Align Tangent	Align	Align Tangent

10.4.2 Inserting Assembly Constraints

To insert assembly constraints:

- 1 Click the **Insert Assembly Constraint** tool from the Assembly Modeling toolbar; or right-click in the work area and select **Insert Assembly Constraint** from the pop-up menu; or from the **Insert** menu select **Assembly Constraint**. The **Assembly Constraints** dialog box appears.



- 2 Select the first edge, face, plane, or axis. The selected item is highlighted in the work area and is listed in the **Surfaces to constrain** area.
- 3 Hold the **Shift** key and select the second edge, face, plane, or axis. The selected item is highlighted in the work area and is listed below the first selection in the **Surfaces to constrain** area.

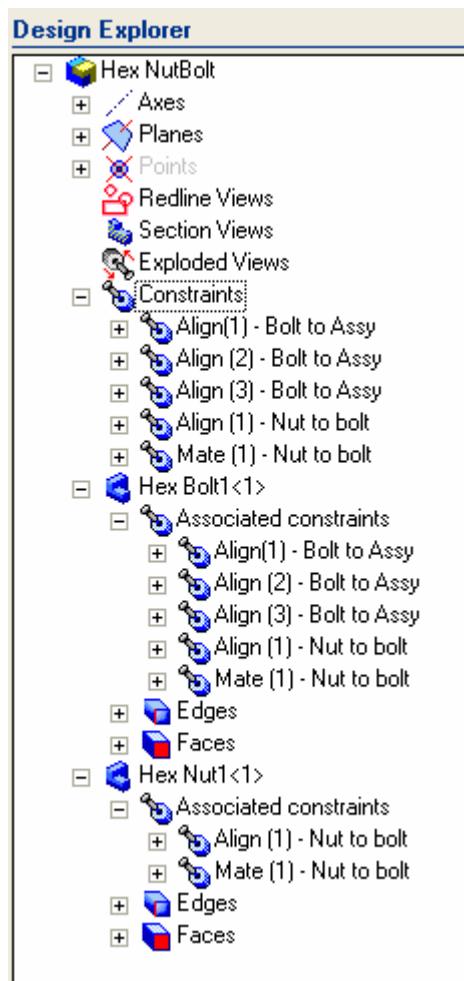
Note: If you prefer, you can pre-select the two entities and then open the **Assembly Constraint** dialog box. The **Surfaces to constrain** area will be populated with the selections.

- 4 Select the appropriate constraint type. Only the constraint types that are valid for the selected items are available.
- 5 If you are applying the **Angle** constraint, specify the angle in degrees.
- 6 If you are applying the **Tangent** constraint, select **Inside** or **Outside**.
- 7 You can specify an **Offset** value if you are applying a **Mate**, **Align**, or **Tangent** constraint.
- 8 Select the **Preview** option to check the result before applying it. If the preview shows the parts in the incorrect position, modify the constraint properties accordingly.

- 9 Click **OK** to apply the constraint.

10.4.3 Managing Assembly Constraints

Each new assembly constraint that you insert will be listed under the **Constraints** node in the Design Explorer, and/or under the constituent that it references. You can choose how you would like to display the constraints. By default, the constraint label contains the names of the two entities the constraint applies to.



If you move the cursor over a constraint or select a constraint in the Design Explorer, the applicable entities will highlight in the work area.

To change how constraints are viewed:

- 1 From the **View** menu, select **Assembly Constraints**.
- 2 Check **As List** to see the constraints listed in the Design Explorer under the Constraints node. This is a toggle on/off option.
- 3 Check **With Component** to see the constraints listed under the parts they relate to. This is a toggle on/off option.

Both options can be checked at the same time, allowing the constraints to be shown in both locations in the design explorer.

To change a constraint's properties:

You can change constraint properties such as offset distance or angle after a constraint has been applied.

- 1 Right-click the constraint in the Design Explorer and select **Edit** from the pop-up menu. The **Assembly Constraints** dialog box appears.
- 2 Modify the appropriate parameters.
- 3 Click **OK** to apply the changes.

To rename a constraint:

You can customize the constraint label in the Design Explorer.

- 1 Right-click the constraint in the Design Explorer and select **Rename** from the pop-up menu; or double-click the constraint with a short pause between clicks. The text cursor appears in the label.
- 2 Type in a new label.
- 3 Press **Enter** on the keyboard.

To delete a constraint:

- 1 Select the constraint and press **Delete** on the keyboard; or right-click the constraint in the Design Explorer and select **Delete** from the pop-up menu.

The constraint is deleted.

10.4.4 Using the Auto Constrain Mode Tool

You can use the **Auto Constrain Mode tool** to quickly place **mate** and **align** constraints. A **mate** constraint is applied if planar faces or linear edges are selected. An **align** constraint is applied if cylindrical faces or edges are selected.

To use the Auto Constrain Mode tool:

- 1 Select the **Auto Constrain Mode**  tool from the Assembly Modeling toolbar; or from the **Tools** menu select **Auto Constrain**; or press **Ctrl + Shift + C** on the keyboard.
- 2 Select the first entity.
- 3 Hold the **Shift** key on the keyboard and select the second entity.

The constraint is applied and is listed in the Design Explorer.

You will remain in auto constrain mode until you select a different tool.

10.4.5 Failed Assembly Constraints

A constraint may fail due to the following reasons:

- Constraint was initially applied incorrectly, e.g. applying an align constraint to a planar face and a non-planar face.
- The constraint creates an over-defined condition.
- A condition changes after the constraint was applied, e.g. deleting a part from an assembly or the geometry in which the constraint applies is modified.

A failed constraint is displayed in italics in the Design Explorer. In the example below, the Align constraint has failed.



It is recommended you resolve a failed constraint as soon as the condition occurs.

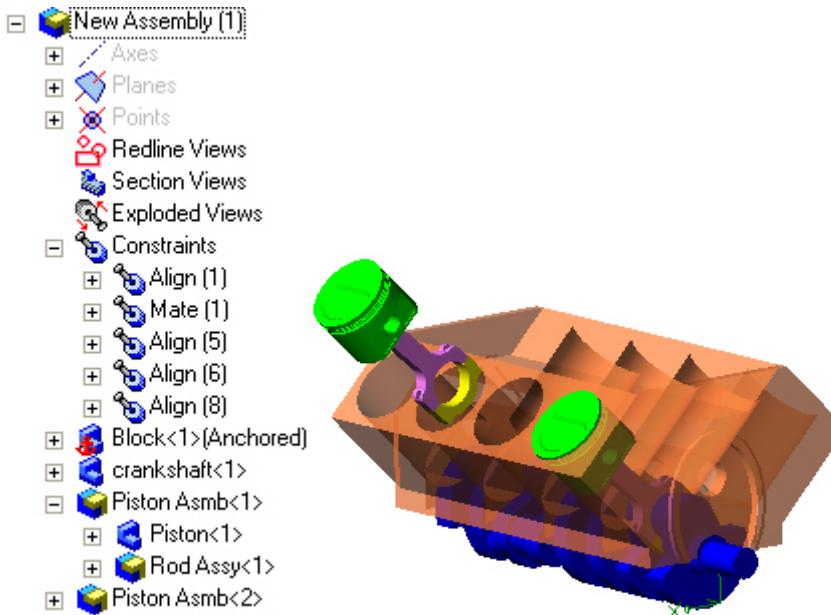
To troubleshoot and resolve a failed constraint:

- Right-click the constraint and select **Status** from the pop-up menu. A dialog box appears containing information related to the cause of the constraint failure.
- Right-click the constraint and select **Edit**. The Assembly Constraint dialog box appears. Modify the constraint conditions if applicable.
- In some cases, you may need to delete the constraint altogether and reapply the constraint.

10.5 Flexible Subassemblies

By default, subassemblies inserted into an assembly are rigid. That is, a subassembly is treated as a rigid body for purposes of positioning and constraining it in the context of the owning assembly. Consequently, if multiple instances of a subassembly are inserted into an assembly, the components in each instance will be in identical relative positions.

You can easily convert a subassembly from being rigid to being flexible. When a subassembly is made flexible, all of the parts in that subassembly can be positioned and constrained independently within the context of the top level assembly.



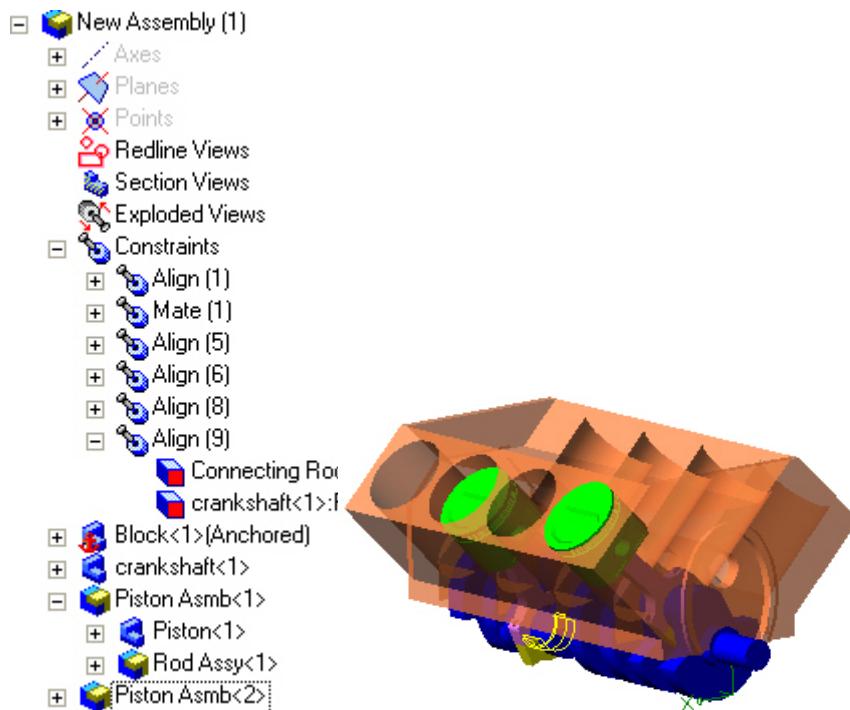
In the example shown here, two instances of a piston assembly are placed in the engine assembly. By default, these subassemblies are rigid. Since each piston subassembly is treated as a rigid body, it is not possible to properly constrain them such that each piston is

aligned with its respective cylinder and each connecting rod is aligned with the crankshaft. This is because in order to completely satisfy the set of constraints, the piston in the second subassembly must move relative to its connecting rod. You can easily accomplish this by making the second subassembly flexible.

To make a subassembly flexible:

- 1 Right-click the desired subassembly in the Design Explorer and click the **Make Flexible** toggle on the pop-up menu.

Each part in the subassembly can now be constrained independent of the other parts in the subassembly. In the example above, it is now possible to properly locate each piston subassembly, as shown here:



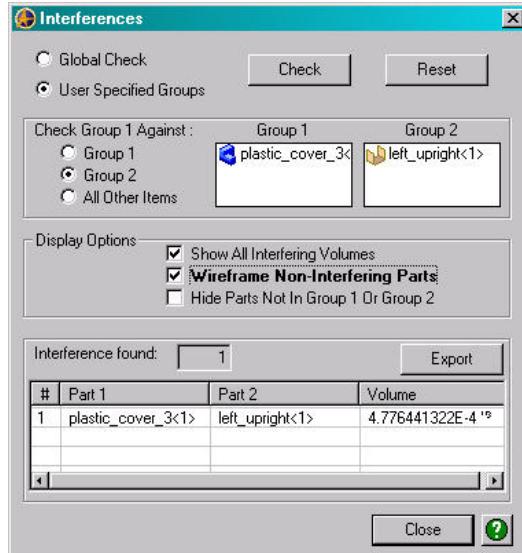
You can again make a subassembly rigid by turning off its **Make Flexible** toggle.

10.6 Checking for Interferences

You can check for interferences between selected assembly parts, selected groups of parts, or all assembly parts.

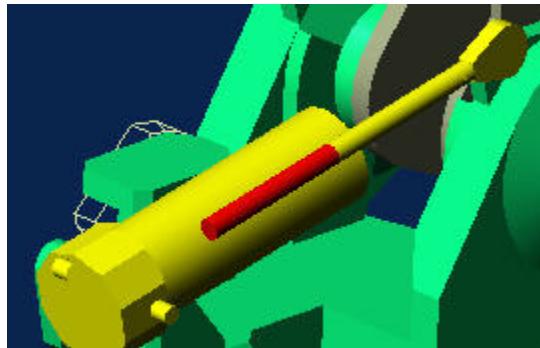
To check for interferences:

- Select the **Check for Interferences** tool from the Inspection toolbar; or from the **Tools** menu select **Check for Interferences**. The **Interferences** dialog box appears.



- Select **Global** or **User Specified Groups**. The **Global** option checks for interferences considering all parts of the assembly. The **User Specified Groups** option checks only the parts you select.
- If you selected **User Specified Groups**, you must select at least one part in **Group 1**. You can then choose to check for interferences between the selected Group 1 part(s) against:
 - Group 1**: checks for interferences within the selected **Group 1** parts (requires at least two selections)
 - Group 2**: checks for interferences between **Group 1** parts and parts selected in **Group 2**

- **All Other Items:** checks for interferences between **Group 1** parts and all other parts in the assembly
- 4 Click the **Check** button. The number of interferences detected is listed in the **Interference found** box. Each interference will also be listed individually specifying which parts are interfering as well as the interfering volume.
- 5 To further analyze interferences, select interference **Display Options**:
- **Show All Interfering Volumes:** displays the detected interferences in red
 - **Wireframe Non-Interfering Parts:** displays the parts that are not interfering in wireframe display mode
 - **Hide Parts Not in Group 1 or Group 2:** hides any parts that were not in the Group 1 or Group 2 selections
- 6 To export the interference information to a .CSV file, click the **Export** button.



10.7 Inserting an Exploded View

You can insert exploded views of the assembly during the design process. The exploded view allows you to display a configuration of the assembly with the parts separated. You can use the auto explode mechanism, manually separate parts, or a combination of both. You can create multiple exploded views in an assembly workspace. The exploded views are saved with the design and can subsequently be used as views in drawings.

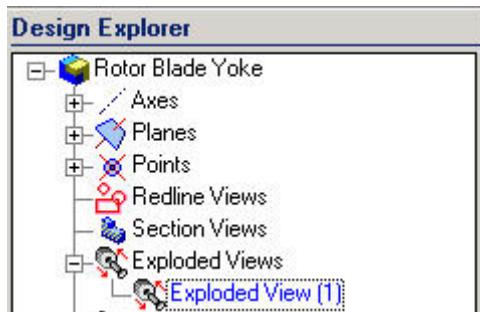
The assembly parts must be constrained before the exploded view can be used. The exploded view however has no effect on the interpretation of assembly constraints.

10.8.1 Inserting an Exploded View Using Auto Explode Mode

You can create multiple exploded views in an assembly workspace. You can auto explode an assembly to quickly create an exploded view of an assembly. **Mate** and **align** assembly constraints must be applied before an exploded view can be created.

To insert an exploded view using Auto Explode mode:

- 1 From the **Insert** menu, select **Exploded View**. The assembly changes to the exploded view. An Exploded View item is listed in blue under the **Exploded View** node in the Design Explorer. By default, exploded views will be labeled **Exploded View(1)**, **Exploded View(2)**, etc. You can rename an exploded view if desired (right-click the view and select **Rename** from the pop-up menu).

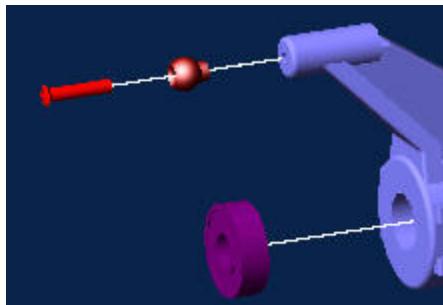


The majority of the assembly modeling tools become dimmed and the explode view tools become available on the Assembly Modeling toolbar.



- 2 To automatically explode the assembly, click the **Auto Explode Assembly** tool from the Assembly Modeling toolbar; or from the **Tools** menu, select **Auto Explode Assembly**. The assembly is exploded. The distances placed between assembly parts are automatically calculated based on part size and orientation.

By default, trail lines are displayed in the work area which provide a visual guide between a part and the part it is constrained to.



- 3 To hide the trail lines, unselect the **View Part Trails**  tool on the Assembly Modeling toolbar; or from the **View** menu, unselect **Exploded View Trails**.
- 4 To increase or decrease the explode distance, click the **Expand Explosion**  tool (to increase) or the **Contract Explosion**  tool (to decrease) from the Assembly Modeling toolbar; or from the **Tools** menu, select **Expand Explosion** or **Contract Explosion**.
- 5 Continue to expand or contract the exploded view until the desired view is achieved.
- 6 To restore a part back to its original position, right-click the part and select **Restore To Default Position** from the pop-up menu.
- 7 To exit the exploded assembly view and return to normal assembly mode, right-click the exploded view in the Design Explorer and select **Exit Exploded View** from the pop-up menu; or from the **Edit** menu, select **Exit Exploded View**; or right-click in the work area and select **Exit Exploded View** from the pop-up menu.

The assembly is returned to its normal view.

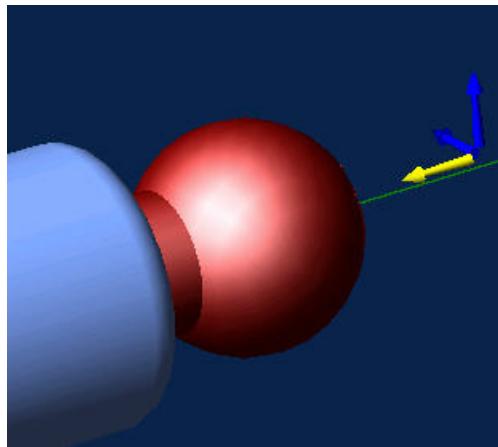
10.8.2 Inserting an Exploded View Using Manual Explode

You can create multiple exploded views in an assembly workspace. You can use the manual explode mode to create a custom exploded view of an assembly. Assembly constraints must be applied before an exploded view can be created. You can use manual explode mode in conjunction with auto explode mode.

To insert an exploded view using Manual Explode mode:

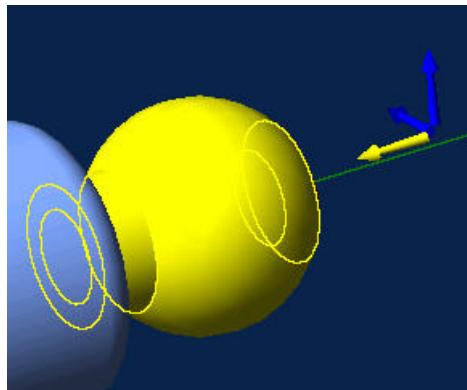
- 1 From the **Insert** menu, select **Exploded View**. An Exploded View item is listed in blue under the **Exploded View** node in the Design Explorer. By default, exploded views will be labeled **Exploded View(1)**, **Exploded View(2)**, etc. You can rename an exploded view if desired (right-click the view and select **Rename** from the pop-up menu).

- 2 To enter Manual Explode mode, select the **Manual Explode Mode**  tool from the Assembly Modeling toolbar; or right-click in the work area and select **Manual Explode Mode** from the pop-up menu; or from the **Tools** menu, select **Manual Explode Mode**.
- 3 To specify the explode direction, select an axis, reference plane, model edge, model planar face, or model cylindrical face. You can specify a different explode direction for each part you want to separate. Reference arrows will subsequently be displayed near your selection.

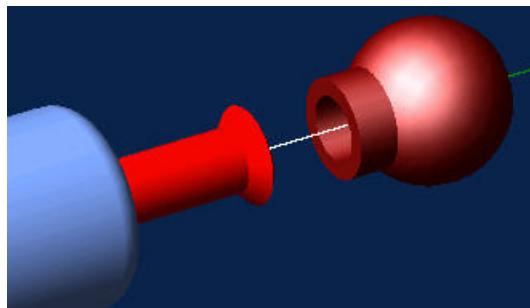


The reference arrows allow you to specify the direction in which you want to manually move a part. The arrow that is currently selected is displayed in yellow and defines the direction the part will move in. To select a different direction, select a different arrow. To remove the reference arrows, click anywhere in any open space in the workspace.

- 4 Select the part you want to move by holding the **Ctrl** key and clicking the part. The part is subsequently highlighted. To select multiple parts to move simultaneously, continue to click additional parts while holding the **Ctrl** key.



- 5 Click and drag the part(s) in the direction defined by the yellow reference arrow. You can also move the part(s) in the direction opposite the arrow is pointing.
- 6 Release the mouse button to complete the separation.



By default, trail lines are displayed in the work area which provide a visual guide between a part and the part it is constrained to. To hide the trail lines, unselect the **View Part Trails**



tool on the Assembly Modeling toolbar; or from the **View** menu, unselect **Exploded View Trails**.

- 7 To exit Manual Explode Mode, unselect the **Manual Explode Mode**  tool from the Assembly Modeling toolbar; or from the **Tools** menu, unselect **Manual Explode Mode**.
- 8 To automatically increase or decrease the explode distance after moving a part or parts, click the **Expand Explosion**  tool (to increase) or the **Contract**

Explode  tool (to decrease) from the Assembly Modeling toolbar; or from the Tools menu, select **Expand Explosion** or **Contract Explosion**.

Continue to expand or contract the exploded view until the desired view is achieved.

- 9 To restore a part back to its original position, right-click the part and select **Restore To Default Position** from the pop-up menu.
- 10 To exit the exploded assembly view and return to normal assembly mode, right-click the exploded view in the Design Explorer and select **Exit Exploded View** from the pop-up menu; or from the **Edit** menu, select **Exit Exploded View**; or right-click in the work area and select **Exit Exploded View** from the pop-up menu.

The assembly is returned to its normal view.

10.8.3 Viewing and/or Editing an Exploded View

You can view and edit an exploded view at anytime.

To view and/or edit an exploded view:

- 1 In the Design Explorer, right-click the Exploded view you want to edit and select **Edit** from the pop-up menu; or select the Exploded view and from the **Edit** menu, select **Edit Exploded View**.

The Exploded view being edited is listed in blue text in the Design Explorer and is displayed in the work area.



- 2 If necessary, make any necessary changes to the Exploded view.
- 3 To return to the normal assembly view, right-click the Exploded view being edited and select **Exit Exploded View** from the pop-up menu.

10.8.4 Deleting an Exploded View

To delete an exploded view:

- 1 Make sure you are not currently editing any Exploded views.

- 2 In the Design Explorer, right-click the Exploded view you want to delete and select **Delete** from the pop-up menu; or select the Exploded view in the Design Explorer, and press **Delete** on the keyboard.

10.8.5 Duplicating an Exploded View

You can create a copy of an existing exploded view.

To duplicate an existing exploded view:

- 1 Make sure you are not currently editing any Exploded views.
- 2 In the Design Explorer, right-click the Exploded view you want to duplicate and select **Duplicate** from the pop-up menu; or select the Exploded view in the Design Explorer, and from the **Insert** menu, select **Duplicate Exploded View**.

A copy of the Exploded view is listed in the Design Explorer.



10.8 Saving and Opening an Assembly

You can save an assembly and its constituents to the same local repository, to different local repositories, to remote repositories, or to the file system.

10.8.1 Saving a New Assembly

- 1 Select the **Save**  tool from the Standard toolbar; or from the **File** menu select **Save**. The **Save** dialog box appears.

The top-level assembly is listed first in the item tree in the left-most column of the dialog box.

The following parts/subassemblies will be listed under the top-level assembly:

- Any new parts that were created in the assembly.

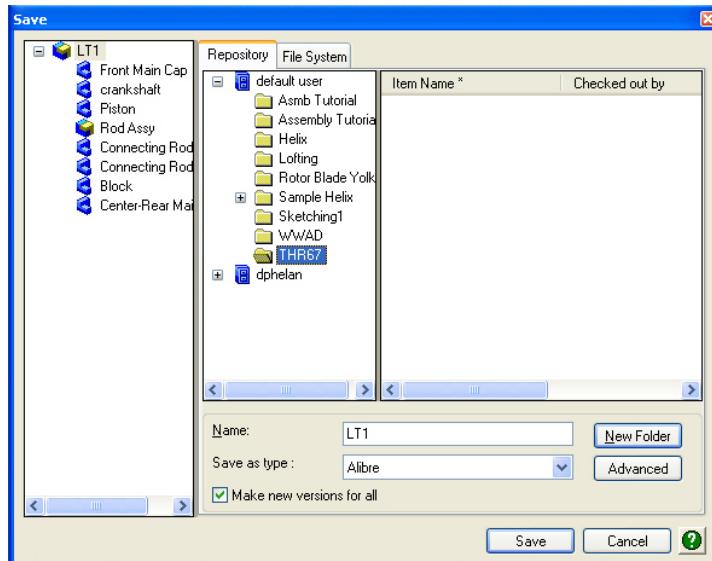
- Any existing parts or subassemblies that were inserted into the assembly and subsequently edited.

The assembly icon is displayed next to the top-level assembly.

A blue part icon and/or colored assembly icon are displayed next to new parts and sub-assemblies, respectively, that have not been previously loaded from the Repository (this includes new parts and imported parts/subassemblies).

A gray part icon and/or gray assembly icon are displayed next to parts and sub-assemblies, respectively, that have been previously saved to the Repository and have been edited in the context of the assembly.

Existing items that were inserted into the assembly but were not edited will be saved with the top-level assembly but will not be listed in the Save dialog box.



- 2 You can specify a name for the new assembly and each new part or sub-assembly that you created in the assembly. Click each individual item in the item tree and then enter a name for the item in the **Name** field.

To save an assembly to the file system:

- 3 Click the **File System** tab.
- 4 Navigate to the file system folder in which you want to save the assembly.

Note: If you are saving an item as native Alibre STEP, you can specify a different save location for each item. To do so, select the item in the item tree and then select the appropriate file system folder. If you do not specify locations for each item, new assembly items will be saved to the location used for the top-level assembly.

- 5 To create a new folder at the currently selected location, click **New Folder**.
- 6 If desired, click **Advanced** to enter detailed comments about the assembly.
- 7 In the **Name** field, type the assembly name.
- 8 Click **Save** to save the assembly.

To save an assembly to the repository:

- 3 Click the **Repository** tab.
- 4 Select a repository from the Repository Explorer and/or click the plus sign  next to a repository to expand it and display the folders within. Select a folder as the save location if desired.

Note: If you are saving an item as native Alibre STEP, you can specify a different save location for each item. To do so, select the item in the item tree and then select the appropriate repository and/or folder. If you do not specify locations for each item, new assembly items will be saved to the location used for the top-level assembly.

- 5 If desired, click **Advanced** to enter detailed comments about the assembly.
- 6 In the **Name** field, type the assembly name.
- 7 Select the **Save as type** from the list. The default type is the native **Alibre** format.
- 8 By default, the **Make new version for all** option is on. This option creates a new version of the design each time a save is completed. If you prefer to maintain one version of a design, unselect the **Make new version for all** option.
- 9 Click **Save** to save the assembly.

10.8.2 Opening an Assembly

You can open a previously saved assembly from the Home window, from the Repository or from an open workspace.

To open an assembly from the Home window or any workspace:

- 1 Select the **Open**  tool from the Standard toolbar; or from the **File** menu select **Open**. The **Open** dialog box appears.
- 2 Using the **Document Browser** embedded in the **Open** dialog, navigate through the repository or the file system to the location of the desired assembly.
- 3 Select the assembly from the item list and click **OK**; or double-click the assembly in the item list.

To open an assembly from the repository:

- 1 In the **Repository Explorer**, browse to the location the assembly is stored in. If necessary, click the plus sign  next to a repository to expand it and display the folders within.
- 2 To open the assembly, double-click the assembly in the item list; or right-click the assembly in the item list and select **Open** from the pop-up menu; or select the assembly in the item list and select the **Open**  tool from the Standard toolbar.

10.8.3 Manually Updating Parts/Sub-assemblies

By default, when you open an assembly, the latest versions of all the assembly's constituents are automatically opened. However, if desired, you can choose to manually update parts/sub-assemblies upon opening an assembly. Consequently, if any parts/sub-assemblies have changed since the last time you opened the assembly, you will be prompted to manually update the modified constituents to the latest version.

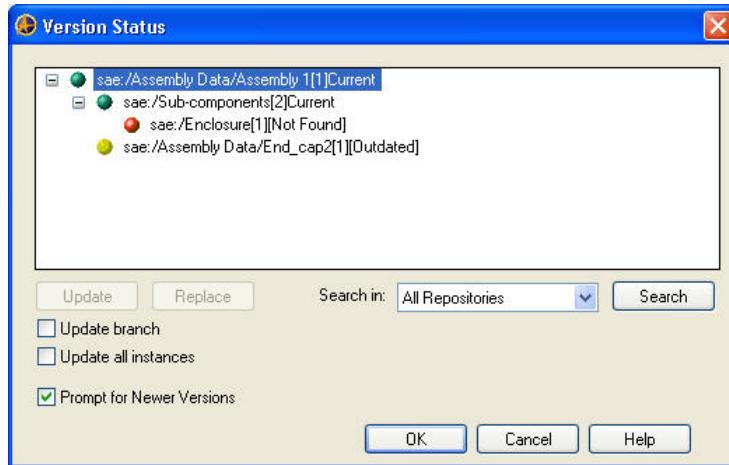
To turn on the prompt for newer versions option:

- 1 In any open workspace, from the **Tools** menu, select **Options**. The **Options** dialog box appears.
- 2 Click the **General** tab if it is not already selected.
- 3 In the **Design** area, click the **Prompt for newer versions** check box.

- 4 Click **OK**.

To manually update an assembly constituent:

- 1 Open the assembly. If any constituents have changed since last opening the assembly and need to be updated, the **Version Status** dialog box appears.



A **green** node displayed next to a constituent indicates that the constituent's version is current. A **yellow** indicates that a newer version of the constituent exists. A **red** node indicates that the constituent cannot be found.

- 2 To update an outdated constituent so that the most current version is used, select the constituent in the list and click **Update**.
- 3 To update a sub-assembly and its constituents, select the sub-assembly in the list, click the **Update branch** check box, and then click **Update**.
- 4 To update a constituent that is used in multiple sub-assemblies, select the constituent in the list, click the **Update all instances** check box, and then click **Update**.
- 5 To update a constituent that cannot be found, select the constituent in the list, and click **Search** to automatically search for the constituent in your repositories as well as any repositories that are currently shared to you. If the constituent is found, the node will turn yellow and you will need to update it.
- 6 To replace a constituent that cannot be found with a different constituent, select the constituent from the list, and click **Replace**. The **Insert Design** box appears. Select a constituent from the Repository Explorer and click **OK**.

- 7 Click **OK** in the Version Status dialog box to open the assembly.

10.9 Editing and Designing Parts in the Assembly

As previously mentioned in this chapter, you can edit existing parts as well as design new parts in the context of the assembly. You can reference existing geometry while editing existing parts and designing new parts.

10.9.1 Creating a New Part Within an Assembly

To create a new part in the assembly:

- 1 From the **Insert** menu select **New Part** or **New Sheet Metal Part**; or press **Ctrl + Shift + N** (for new part) on the keyboard. The work area changes from assembly edit mode to part edit mode. The Sketching and Part Modeling toolbars are displayed on the right side instead of the Assembly Modeling toolbar. All other parts in the assembly will be displayed semi-transparently.

Note: New Sheet Metal Part is only available in Alibre Design Professional and Alibre Design Expert.

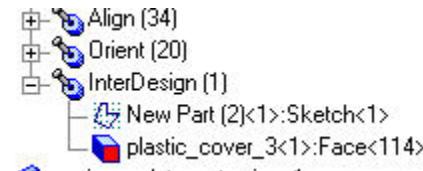
Additionally, the new part is listed in blue in the Design Explorer to signify that it is currently being edited.



- 2 Construct the part features using the same techniques used to create a part in a part workspace.

You can use edges and faces on other parts as the sketch plane in the new part. You can also use edges and faces on other parts as references or targets as you model the new part. You can also project entities from existing assembly parts onto a sketch plane in the new part to create sketch or reference figures.

While in part edit mode, if you use a face on an existing part as a sketch plane or to create a new reference plane, an **inter-design constraint** will automatically be created. This constraint will be displayed in the Design Explorer under the **Constraints** node when you switch back to assembly edit mode. The inter-design constraint label by default lists the two related parts (the parent part and the associated part).



After an inter-design constraint has been created, any change made to the parent part will automatically update the associated part. To break the interdependency, delete the inter-design constraint in assembly edit mode.

- 3 You can switch back to assembly edit mode at any point. To do so, right-click the part listed in blue in the Design Explorer and select **Edit Root Assembly** from the pop-up menu; or right-click the top-level assembly in the Design Explorer and select **Edit Part/Subassembly** from the pop-up menu.

10.9.2 Editing a Part in an Assembly

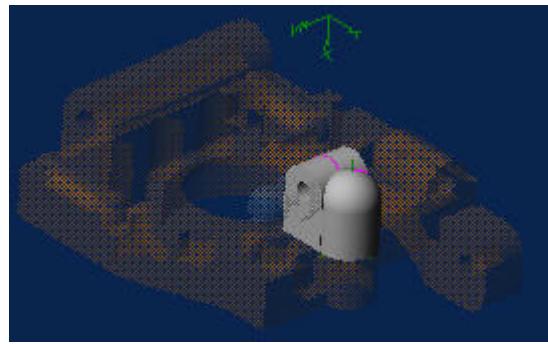
You can edit a part without leaving the context of the assembly. While editing a part in assembly mode, you can reference geometry on other parts while sketching or creating new features.

You can also edit a part in an independent workspace as well. After saving and closing the workspace, the changes made to the part will automatically be reflected in the assembly.

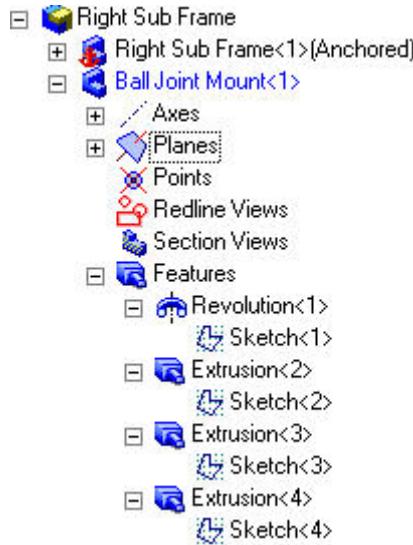
To edit a part in the context of the assembly:

- 1 Right-click the part in the Design Explorer and select **Edit Part/Subassembly** from the pop-up menu; or select the part in the Design Explorer or work area and from the **Edit** menu select **Edit <Part Name>**.

The work area changes from assembly edit mode to part edit mode. The Sketching and Part Modeling toolbars are displayed on the right side of the work area instead of the Assembly Modeling toolbar. All other parts in the assembly remain visible but are displayed semi-transparently. The part you are editing remains fully shaded.



The part being edited is listed in blue in the Design Explorer and all the associated reference geometry, sketches, and features can be accessed.



- 2 Edit the part sketches or features just like you would in a part workspace. You can also add new features if required.
- 3 After making the necessary changes, return to assembly edit mode by right-clicking the part being edited in the Design Explorer and select **Edit Root Assembly**; or right-click the top-level assembly in the Design Explorer and select **Edit Part/Subassembly**.

To edit a part in an individual workspace:

- 1 While holding the **Shift** key, right-click the part you want to edit in the Design Explorer and select **Edit Part/Subassembly**.

The part opens in a separate workspace.
- 2 Make any necessary changes to the part. The changes will be applied to the assembly automatically.
- 3 When finished editing, close the part workspace. You will be prompted to save the part if changes have been made. Save the part if you want to save the changes.

You are returned to the assembly workspace and can continue working.

10.10 Importing Parts into an Assembly

You can import parts and subassemblies into an assembly and subsequently use them in the design. You can import **STEP**, **SAT**, and **IGES** files into an assembly. To learn more about importing data refer to **Chapter 13**.

To import a part into an assembly:

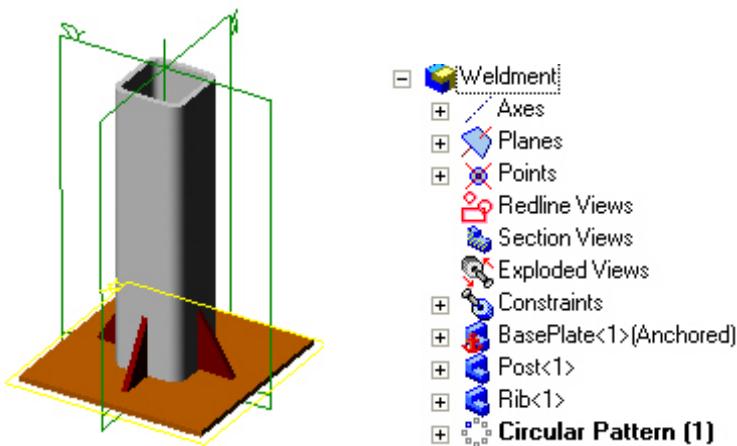
- 1 Select the **Import**  tool from the Standard toolbar; or from the **File** menu select **Import**. The **Import File** dialog box appears.
- 2 Browse to the location of the file that you want to import.
- 3 Select the file to import.
- 4 Click **Open**.

The part/subassembly appears in the assembly workspace and is listed in the Design Explorer. You can constrain the imported parts to existing parts and vice versa.

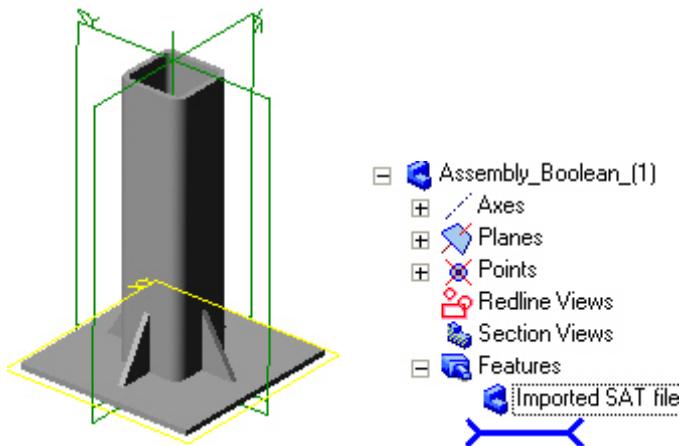
10.11 Joining Parts & Removing Material in an Assembly

You can use the **Assembly Boolean** command to either join multiple parts together into a new single part or to remove material from parts in an assembly. The result is one or more new parts that can be added into the existing assembly or placed in a new workspace.

To join parts in an assembly:



- 1 From the assembly workspace, select the **Assembly Boolean** command from the **Tools** main menu. The **Assembly Boolean** dialog appears.
- 2 Select the parts to join from the Design Explorer or the 3D work area. These will be listed in the **Blanks** field.
- 3 Check the **Join All Blanks** option.
- 4 If you want the resulting joined part to be placed in the original assembly, check the option **Insert results into current assembly**. If you want the joined part in a separate workspace, uncheck this option.
- 5 Keep the **Tools** field empty. Click **OK**. The selected parts are joined together. The joined part is represented in the Design Explorer as a new part with an **Imported SAT File** feature.



To remove material in an assembly:

You remove material from parts in the assembly by specifying one or more parts that will be used as tools for cutting away material. From the assembly workspace, select the Assembly Boolean command from the Tools main menu. The Assembly Boolean dialog appears.

- 1 Select the parts from which you want to remove material from the Design Explorer or the 3D work area. These will be listed in the **Blanks** field.
- 2 Click in the **Tools** field and then select the parts you want to use for cutting.
- 3 Use the **Join All Blanks** option if you want to unite all the blanks into a single part.
- 4 If you want the resulting cut parts to be placed in the original assembly, check the option **Insert results into current assembly**. If you want the cut parts in separate workspaces, uncheck this option.
- 5 Click **OK**. The selected blank parts are cut by the selected tool parts. Each cut blank is represented in the Design Explorer as a new part with an **Imported SAT File** feature.

11 Drawings

You can create 2D drawings of the parts and assemblies you create. Standard 2D views can be created automatically from the part or assembly. Custom views can be created based on the standard views already present in the drawing.

This chapter describes:

- Opening a new drawing workspace.
- Selecting a drawing template.
- Setting the drawing scale.
- Inserting standard views into the drawing.
- Adding dimensions to the drawing views.
- Adding notes and annotations to the drawing.
- Creating a custom drawing template.
- Inserting custom views.
- Saving a drawing.

11.1 Creating a New Drawing

11.1.1 Opening a New Drawing

You can open a new drawing from the Home window, Repository, or any open workspace.

To open a new drawing from the Home window, repository, or any workspace:

- 1 Select the **New Drawing**  tool from Standard toolbar; or from the **File** menu, select **New > Drawing**.

The **New Sheet Properties** dialog box appears.

11.1.2 Selecting a Drawing Template

You can select from a number of standard drawing templates, your own custom templates, or a blank drawing of varying size. **ANSI**, **DIN**, and **ISO** drawing templates are supported.

To select a drawing template:

- 1 In the **New Sheet Properties** dialog box, select **Template** or **Blank Sheet**.
- 2 If you selected **Template**, also select a standard drawing template from the list. To use a custom drawing template not listed, click **Browse** and select the template from the **Custom Drawing Template** dialog.

OR, if you selected **Blank Sheet**, select a sheet size from the list.

Note: If you have previously **browsed** to another template folder, you can use the **Default** button to reset the template list back to the system template folder.

- 3 In the **Default View Scale** area, specify the scale to use in the drawing. The scale can be changed later if necessary.
- 4 To create a drawing with no models associated with it, check the **Create An Empty Drawing** box.
- 5 Click **OK**.

If you selected a standard or custom template, the **Fill In Text** dialog box might appear.

11.1.3 Specifying Standard Drawing Information

When using standard or custom drawing templates, you can enter standard text information such as drawn by, drawn date, and drawing number.

To specify standard drawing information:

- 1 In the **Fill In Text** dialog box, select the **DRAWN** item from the **Select Tag Field** list and then type the appropriate information in the text box to the right.
- 2 Repeat for **DRAWN DATE** and **DWG NO.**.
- 3 Click **OK**. The drawing workspace and **Insert Design** dialog box appear.

Note: To leave these fields blank, click **OK** without entering any information.

11.1.4 Selecting the Model

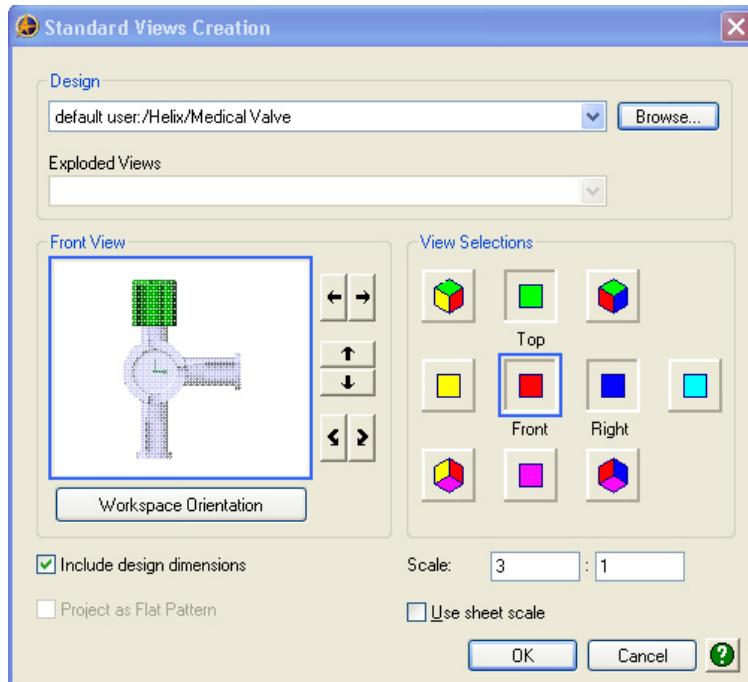
You can create 2D drawing views automatically using the views from a part or assembly.

To select the model:

- 1 In the **Insert Design** dialog box, use the Document Browser to select the part or assembly from which the 2D drawing views will be created.
- 2 Click **OK**. The **Standard Views Creation** dialog box appears and the selected design populates the **Design** field.

11.1.5 Inserting Standard Views

You can select which standard views you initially want to create in the drawing. You can insert additional standard views later as necessary.



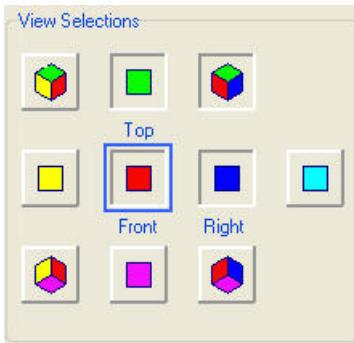
To insert the standard views:

- 1 A preview of the selected part or assembly is shown in the **Front View** preview window. To select a different part, click the **Browse** button to make a new selection.
- 2 Select the view that you want to use as the **Front View** in the drawing. You can use the arrow buttons to reorient the view in the preview window.



You can also click the **Workspace Orientation** button to select from default and custom views created in the part or assembly workspace.

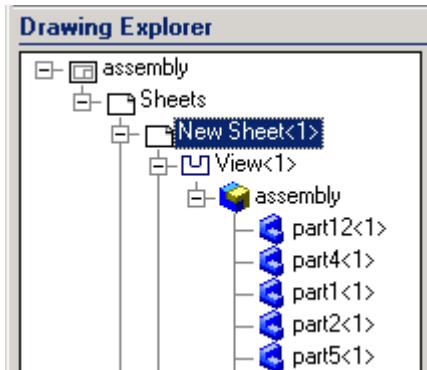
- 3 In the **View Selections** area select the views that you want to insert into the drawing. The default views are **Front**, **Top**, and **Right**. To add or remove a view, click the view's corresponding button.



- 4 In the case of assemblies, if any exploded views were created in the assembly workspace, you can also select an exploded view from the **Exploded Views** list.
- 5 Check the **Include design dimensions** box if you want to display the driving dimensions from the model.
- 6 The default value for the view **Scale** is automatically computed by Alibre Design so that the new views fit cleanly onto the sheet. As an alternative, you may override this value by entering an explicit value or by choosing to use the **Sheet scale**.
- 7 Click **OK** to create the views. Previews of the selected views are displayed in the work area. Note that the cursor is essentially tied to the front view. As you move the cursor, the views will all move together.
- 8 Move the cursor to dynamically re-position the front view to the correct location within the drawing sheet.
- 9 Click to place the front view. The corresponding views are also placed.

Note the dimensions that were used to create the part are displayed automatically in the corresponding view.

The **Drawing Explorer** on the left side of the work area lists the sheets associated with the drawing as well as the views associated with each sheet. The associated design is also listed under the view.



NOTE: Alibre Design supports both **First Angle** and **Third Angle** projection methods. You can set the projection method for a drawing in the **Detailing** tab of the **Drawing Properties** dialog. Click **Properties** in the **File** main menu to bring up the Drawing Properties dialog.

11.2 Saving and Opening a Drawing

11.2.1 Saving a New Drawing

- 1 Select the **Save**  tool from the Standard toolbar; or from the **File** menu select **Save**. The **Save** dialog box appears. The drawing icon will be displayed next to the drawing name in the item list. If you made changes to driving dimensions in the drawing, the associated part also will be displayed in the item list. A gray part icon will be displayed next to the part.

To save a drawing to the file system:

- 2 Click the **File System** tab.
- 3 Navigate to the file system folder in which you want to save the drawing.
- 4 To create a new folder at the currently selected location, click **New Folder**.
- 5 If desired, click **Advanced** to enter detailed comments about the drawing.
- 6 In the **Name** field, type the drawing name.

- 7 Click **Save** to save the drawing.

To save a drawing to the repository:

- 2 Click the **Repository** tab.
- 3 Navigate to the location in which you want to save the drawing. You can click the plus sign  next to a repository to expand it and display its folders. You can save the drawing directly under the selected repository or into any of the repository's folders
- 4 To create a new folder at the currently selected location, click **New Folder**.
- 5 If desired, click **Advanced** to enter detailed comments about the drawing.
- 6 In the **Name** field, type the drawing name.
- 7 Select the **Save as type** from the list. The default type is the native **Alibre Design** format.
- 8 By default, the **Make new version for all** option is selected. This option creates a new version of the drawing each time a save is completed. If you prefer to maintain one version of a drawing, deselect the **Make new version for all** option.
- 9 Click **Save** to save the drawing.

11.2.2 Opening a Drawing

Opening a drawing

You can open a previously saved drawing from the Home Window, from the Repository or from an open workspace.

To open a drawing from the Home window or any workspace:

- 1 Select the **Open**  tool from the Standard toolbar; or from the **File** menu select **Open**. The **Open** dialog box appears.
- 2 Using the **Document Browser** embedded in the **Open** dialog, navigate through the repository or the file system to the location of the desired part
- 3 Select the drawing from the item list and click **OK**; or double-click the drawing in the item list.

To open a drawing from the repository:

- 1 In the **Repository Explorer**, browse to the location the drawing is stored in. If necessary, click the plus sign  next to a repository to expand it and display the folders within.
- 2 To open the drawing, double-click the drawing in the item list; or right-click the drawing in the item list and select **Open** from the pop-up menu; or select the drawing in the item list and select the **Open**  tool from the Standard toolbar.

Immediately after opening a drawing, you have the ability to perform limited functions in the drawing, while the design(s) related to the drawing continue to load. Large drawings containing numerous designs can take some time to fully load. The following functions are available as soon as the drawing is visible:

- Print and Print Preview
- Zoom and Pan
- View options such as Toggle Annotations and Toggle Redlines
- Send snapshot by email
- Selection Filter Commands

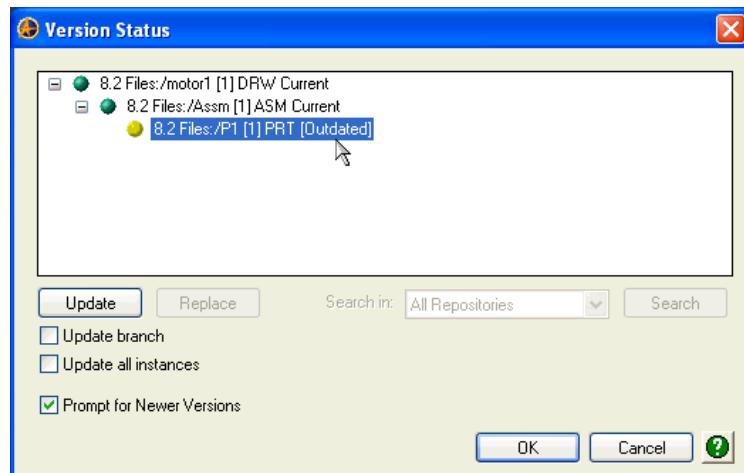
11.3 Working in a Drawing

11.3.1 Drawing Mark-Up Mode

Drawing mark-up mode allows you to load a drawing without loading the underlying designs for the drawing. You will enter mark-up mode if the designs are unavailable, or if you have modified any of the designs and choose not to update the drawing. The features available in mark-up mode are all of the features listed in Section 11.2.2, as well as Insert Annotations and Insert Redlines. One benefit to Drawing Mark-up Mode is that you can send another user a drawing to review without sending the design files.

Opening from the repository: Drawings with designs saved with versions

When you open a drawing from the repository that has outdated versions associated with it (meaning you have modified a design and saved it as a new version), you will see the Version Status dialog box.



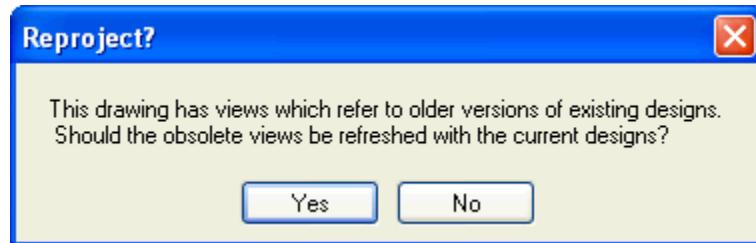
The designs that have been modified and saved will be marked as Outdated. You can choose to update the parts, then select **OK**;

-OR-

Select **OK** without updating the parts. If you update the designs, the drawing will open in normal edit mode. If you choose not to update the designs, the drawing will open in mark-up mode and you will have use of limited features.

Opening from the repository: Drawings with designs saved without versions

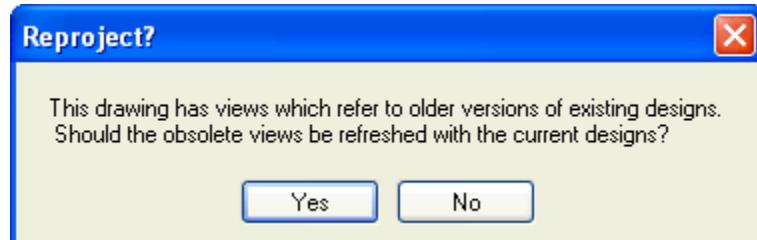
When you open a drawing from the repository that has an outdated design that was saved without versions, the drawing file will open, and you will be prompted to **Reproject** the outdated designs.



If you choose **Yes**, all of the outdated designs will be updated and the drawing will open in normal edit mode. If you Choose **No**, the outdated designs will not be updated, and the drawing will open in mark-up mode, allowing you the use of limited features.

Opening drawings from the file system

When you open a drawing from the file system that has an outdated design, the drawing file will display, and you will be prompted to **Reproject** the outdated designs.



If you choose **Yes**, all of the outdated designs will be updated and the drawing will open in normal edit mode. If you Choose **No**, the outdated designs will not be updated, and the drawing will open in mark-up mode, allowing you the use of limited features.

11.3.2 Renaming Sheets & Views

You can rename the sheet and corresponding views to convey relevant design information.

To rename a sheet or view:

- 1 Right-click a sheet or view in the Drawing Explorer and select **Rename** from the pop-up menu.
- 2 Type the new name.
- 3 Press **Enter**. The sheet or view name is updated.



11.3.3 Changing the Drawing Template

You can change the drawing template after the initial drawing creation.

To change the drawing template:

- 1 Right-click the sheet in the Drawing Explorer and select **Change Template** from the pop-up menu. The **New Sheet Properties** dialog box appears.
- 2 Select the **Template** option and select a template from the list. To use a custom drawing template not listed, click **Browse** and select the template from the **Custom Drawing Template** dialog. **Note:** If you have previously browsed to another template folder, you can use the **Default** button to reset the template list back to the system template folder.

Or

Select the **Blank Sheet** template option and select a sheet size from the drop down list.

- 3 Change the Default View Scale if necessary.
- 4 Click **OK**.
- 5 If a Standard template was selected, the **Fill In Text** dialog box appears. Complete the applicable fields and click **OK**. The sheet is updated and the new template is displayed in the work area.

11.3.4 Deleting Views

You can delete views at any time.

To delete a view:

- 1 Right-click the view in the Drawing Explorer and select **Delete** from the pop-up menu; or select a view in the work area or Drawing Explorer and press **Delete** or select the **Delete**  tool from the Standard toolbar.

Note: Any views that were created from the deleted view, such as auxiliary views, will also be deleted.

11.3.5 Hiding Views

You can hide/unhide views at any time. A hidden view is not displayed in the 2D work area. It is displayed in the Drawing Explorer but is grayed out.

To hide a view:

- 1 Right-click the view in the Drawing Explorer or the 2D work area and select **Hide** from the pop-up menu.

To unhide a view:

- 1 Right-click the view in the Drawing Explorer and select the **Hide** toggle (which should be marked with a check) from the pop-up menu.

11.3.6 Drawing Selection Filters

By default, you can select any item in a drawing. When you move the cursor over an item in the work area, the item is highlighted. You can select the following items individually in a drawing workspace:

- Parts
- Part edges and vertices
- Dimensions
- Sketches
- Annotations
- Redlines
- Views

As you work in a drawing workspace, you may find it advantageous to select a certain group or groups of items as opposed to all items. In this case, you can apply selection filters and specify which item groups you want to select.

To use selection filters:

On the **Filters** toolbar (illustrate below), select the tools corresponding to the item groups you want to be able to select. A filter is applied when the corresponding tool is in the pressed state.



In this example, the **Views**, **Annotations**, and **Dimension** filters are on. Consequently, only items in these groups can be selected in the work area. Additionally, when you move the cursor in a view, only the view and any corresponding dimensions and annotations will be highlighted.

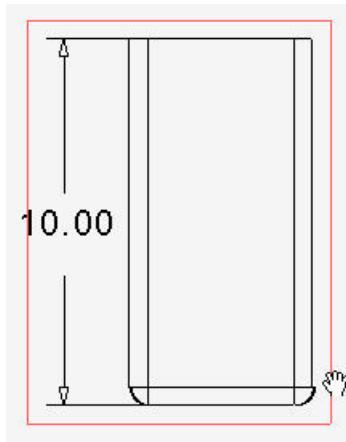
You can also access and apply **Selection Filters** from the **Tools** menu. If a check mark is displayed next to a filter, the filter is currently being applied.

11.3.7 Moving Views on the Sheet

After initially placing the views, you can re-position views on the sheet as necessary. Note that the standard views are initially aligned. Moving the front view will cause related standard views to move. You can break the view alignment if necessary.

To move a view:

- 1 Select the **View** selection filter if it is not already being applied.
- 2 Select the **Select**  tool from the View toolbar if it is not already selected.
- 3 Move the cursor over the view. A red view boundary appears and the cursor changes.



- 4 To reposition a view or the views, click and hold the mouse button, and move the cursor. The selected view and all associated views move.
- 5 Release the mouse button to place the views.

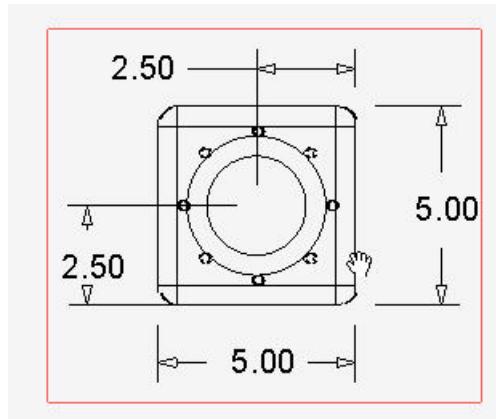
Note: You can break an individual view's dependence on other views by right-clicking the view and un-checking the **Align** toggle in the pop-up menu. You can again establish the alignment by re-checking the **Align** toggle.

11.3.8 View and Sheet Boundaries

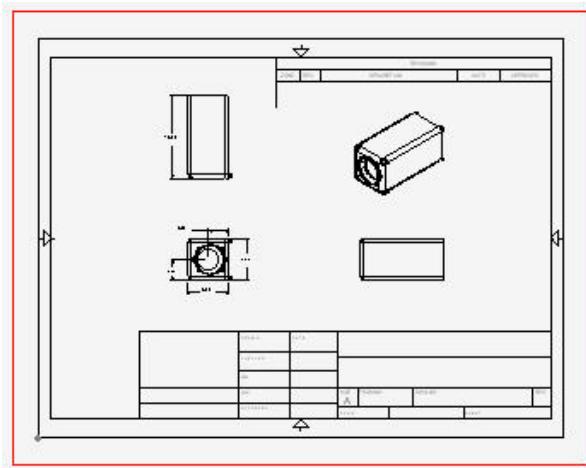
A red view boundary is highlighted as you move the cursor over a view. The view boundary is displayed both in and out of sketch mode. The boundary size is calculated automatically

based on the extents of the view. Consequently, you cannot change the size of the view boundary.

You can only work on a view (e.g. add dimensions, sketch figures, etc.) when the view boundary is displayed around it. The view boundary indicates the view is active. Any items added to the view when it is active will be associated with the view. If the view is moved, the inserted items will move with it.



A sheet boundary is also highlighted when a sheet is selected and you enter sketch mode. The sheet boundary indicates the sheet is active. Any items added to the sheet when it is active will be associated with the sheet. The boundary size is based on the extents of the entire sheet.



11.3.9 Changing the View Scale

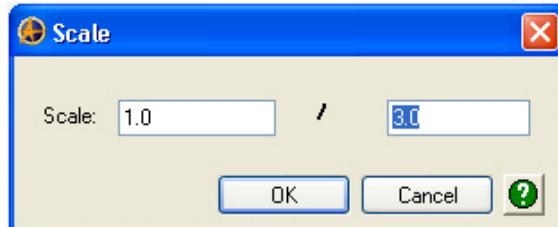
You can change the default view scale or the scale of individual views after they have been placed. A changed default view scale is applied to any views that are inserted subsequent to the scale change. Views already in the sheet will not be effected. Changing the scale of an individual view will subsequently change the scale of its dependent views.

To change the default view scale:

- 1 In the Drawing Explorer, right-click the sheet you want to change the default view scale in and select Default View Scale from the pop-up menu. The Scale dialog box appears.
- 2 Modify the scale as required.
- 3 Click **OK**. The default view scale is changed.

To change the scale of a view:

- 1 Right-click the view and select **Scale** from the pop-up menu. The **Scale** dialog box appears.



- 2 Specify the new scale.
- 3 Click **OK**. The view scale is updated.

11.3.10 Line Display in Views

You can control how individual views are displayed. You can show or hide **hidden lines** and **tangent edges**. By default, tangent edges are shown, and hidden lines are not displayed.

To show or hide hidden or tangent edge lines in a view:

- 1 Right-click a view in the Drawing Explorer, or right-click a view in the work area and select one or both of the following: **Show Hidden Lines**, or **Show Tangent Edges**. A check mark displayed next to the option indicates the item is currently being displayed.

11.3.11 Centerlines and Centermarks

The following apply to centermarks and centerlines in 2D drawings:

- Displayed by default for holes in 2D drawings.
- Can be modified globally or on an individual basis.
- Dimensions can be placed between centermarks/centerlines and any other applicable item in a view.
- Can be individually deleted, edited, or placed on a different layer.
- Can be inserted on a per view basis.
- Can be inserted individually for projected circular or cylindrical geometry.

To turn off automatic centermark and centerline display:

- 1 From the **File menu**, select **Properties**. The **Drawing Properties** dialog box appears.

- 2 Click the **Detailing** tab.
- 3 Under **View Creation Options**, uncheck **Centerlines** and/or **Centermarks**.
- 4 Click **Apply** and **Close**.

To modify global centermark and centerline properties:

- 1 From the **File** menu, select **Properties**. The **Drawing Properties** dialog box appears.
- 2 Click the **Detailing** tab.
- 3 Under **Centerlines**, modify the **Centermark Style**, **Short Dash**, **Extension**, and **Gap** as needed.
- 4 Click **Apply** and **Close**.

To modify individual centermark and centerline properties:

- 1 Select the **Select** tool from the **Viewing** toolbar.
- 2 Move the cursor over the centermark or centerline to be modified.
- 3 Right-click the centermark or centerline and select **Edit** from the pop up menu. The **Centerline Properties** dialog box appears.
- 4 Modify the **Centermark Style**, **Short Dash**, **Extension**, and **Gap**.
- 5 If a centermark is being edited, you can rotate it by specifying a **Direction** angle.
- 6 Click **OK** to accept the changes.

To delete a centermark or centerline:

- 1 Select the **Select** tool from the **Viewing** toolbar.
- 2 Move the cursor over the centermark or centerline to be deleted.

- 3 Right-click the centermark or centerline and select **Delete** from the pop up menu.

OR

Right-click the figure and select **Remove Center** from the pop up menu.

To insert a centermark or centerline on a per view basis:

- 1 Select the **Select** tool from the **Viewing** toolbar.
- 2 Move the cursor over the view in which centermarks and centerlines are to be added.
- 3 Right-click the view and select **Insert Centers**.

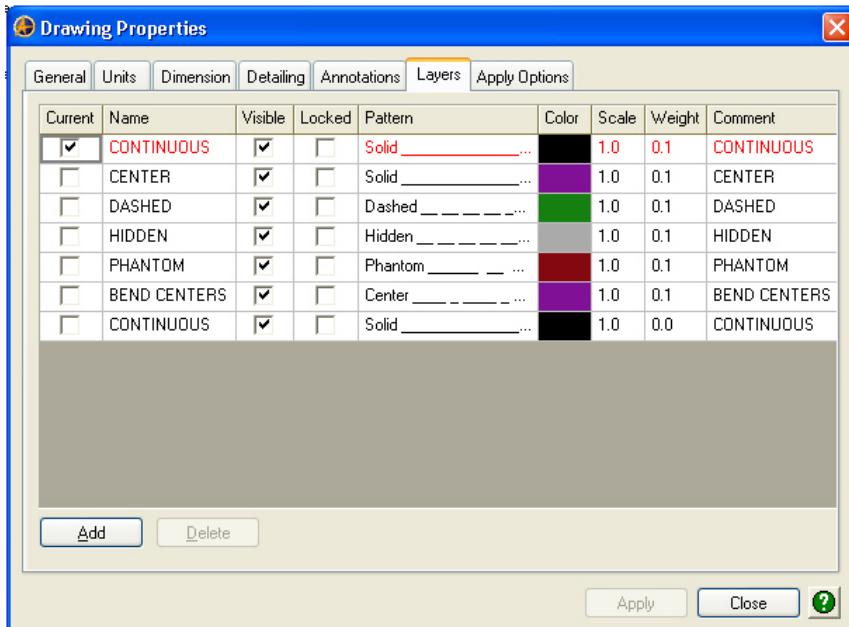
Note: To insert the centerlines for individual holes, select the hole; then right-click and choose **Insert Center**.

To insert a centerline on projected circular or cylindrical geometry:

- 1 Select the **Select** tool from the **Viewing** toolbar.
- 2 Move the cursor over the figure to insert a centerline on.
- 3 Right-click the figure and select **Insert Center** from the pop up menu. The centerline is displayed on the figure.

11.3.12 Layers

In drawings, model views are displayed and detailed using a variety of layers. The use of different layers is often dependant on the drafting and detailing standards defined by your organization. In Alibre Design, there are six predefined layers which can be modified to meet your standards. Additional layers can be added to your pallet as needed. The layer settings are accessible from the Layers tab in the Drawing Properties dialog box (**File > Properties > Layers** tab).



The layer attributes are described as follows:

- **Current** [checkbox]: Layer to be used for new drawing items. Only one layer can be designated as current.
- **Name**: The layer name.
- **Visible** [checkbox]: When checked, all drawing items assigned to that layer are visible. Clear the checkbox to hide items in a particular layer.
- **Locked** [checkbox]: When checked, the corresponding layer is locked and changes cannot be made to layer attributes.
- **Pattern**: A preview of the layer line style. Click to access a menu of additional styles.
- **Color**: The layer color in the drawing. Double-click the colored box to access the Color dialog box and select a different color.
- **Scale**: Maximum length of line segments in dashed lines.
- **Comment**: Insert a comment to indicate the layer purpose.

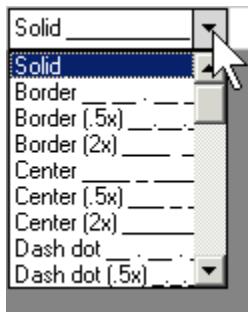
Any items that are inserted or created in a drawing workspace (e.g., sketch figures, annotations, dimensions, etc.), are displayed using the current layer's attributes. Only one layer can be current at a time. If you change the pattern, visibility, color, or scale of a layer used in the drawing, all existing drawing items will be updated with the new settings.

To select a different layer as the current layer:

- 1 Click the **Current** check box corresponding with the layer you want to make current.
- 2 Click **Apply**.
- 3 Click **Close**.

To change the pattern

- 1 Double-click the appropriate **Pattern** entry. An arrow appears in the cell.



You can scroll through the list, or type the first letter of the desired pattern name. For example, type the letter "i" to jump to the first ISO style. Continue pressing the "i" key to page through all of the ISO styles.

- 2 Select the pattern you want from the list.
- 3 Check **Current** to use this layer for new items.
- 4 Click **Apply** to implement the new settings.

Note: For advanced users who want to customize the available line patterns in Alibre Design: You can modify the predefined line patterns that ship with Alibre Design by editing the text file, alibre_unicode.lin. You can use **Notepad** to edit this file. A definition of the file format is embedded within the file. This file is located in the folder **C:\Documents and Settings\All Users\Application Data\Alibre Design\System Files**.

To change the color:

- 1 Double-click the appropriate Color cell. The **Color** dialog box appears.
- 2 Select a preset color or click **Define Custom Colors** to create a specific color.
- 3 Click **OK** to close the Color dialog box.
- 4 Click **Apply** to implement the new settings.

To change the layer name, scale, weight, or comment:

- 1 Click the cell containing the text or value you want to change. A gray box borders the cell.
- 2 Click again. A blinking cursor appears. You can now edit the contents of the cell.
- 3 Click **Apply** to implement the new settings.

To add a layer:

- 1 Click **Add**. A new row appears at the bottom of the table, temporarily named **New Line Style**.

Current	Name	Visible	Locked	Pattern	Color	Scale	Weight	Comment
<input checked="" type="checkbox"/>	CONTINUOUS	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Solid _____	#000000	1.0	0.1	CONTINUOUS
<input type="checkbox"/>	CENTER	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Solid _____	#800080	1.0	0.1	CENTER
<input type="checkbox"/>	DASHED	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Dashed _____	#008000	1.0	0.1	DASHED
<input type="checkbox"/>	HIDDEN	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Hidden _____	#A9A9A9	1.0	0.1	HIDDEN
<input type="checkbox"/>	PHANTOM	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Phantom _____	#8B0000	1.0	0.1	PHANTOM
<input type="checkbox"/>	New Line Style?	<input checked="" type="checkbox"/>	<input type="checkbox"/>	Solid _____		1.0	0.1	New Line Style?

- 2 Modify the layer attributes as necessary, including the layer **Name**.
- 3 Click the **Current** check box to use this style for new items.
- 4 Click **Apply** to implement the changes.
- 5 Click **Close** when finished.

To delete a layer:

- 1 Select the layer you want to delete from the list.
- 2 Click **Delete**.

To change the layer of an existing figure, dimension, or annotation:

- 1 Right-click the item you want to reformat and select **Set Layer** from the pop-up menu. The **Layers** dialog box appears.
- 2 In the Current column, select the layer you want to move the selected item to.
- 3 Click OK to apply the change.

When the layer for an annotation or dimension is changed, the entire dimension/annotation including figures and text will be rendered in the layer's color. All figures and leaders will be rendered with the line pattern of the layer (continuous, dashed, etc.).

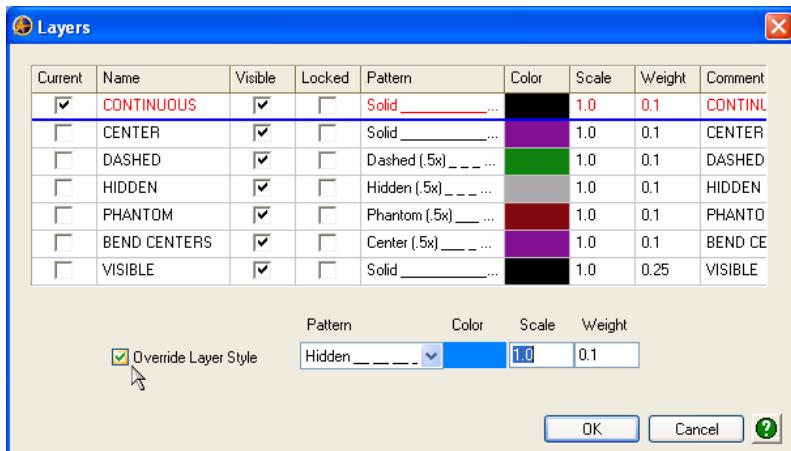
The only exception to this rule is the **Text Note** annotation, for which the text is always displayed using the font color specified in the Text Note dialog box. The leader will still be rendered in the color and line pattern associated with the layer.

To override the layer properties of a figure without changing the layer:

You can change the properties of any figure on a layer without changing the layer that the figure resides on. You do this by overriding the current layer style:

- 1 Right-click the figure you wish to change the properties of.

Select **Set Layer**. The Layers dialog box appears.



Check the **Override Layer Style** box; then select the Pattern, Color, Scale, and Weight for the figure.

Click **OK** to apply the changes. The properties of the figure will be changed, but the figure will still reside on the original layer.

11.3.13 Adding Sheets

You can add additional sheets to the drawing as needed.

To add a sheet to the drawing:

- 1 Select the **Insert New Sheet**  tool from the Detailing toolbar; or from the **Insert** menu select **New Sheet**. The **New Sheet Properties** dialog box appears.
- 2 Follow the steps outlined in sections **11.1.2** and **11.1.3** to specify the information included in the new sheet. After inserting a new sheet into the drawing, the sheet will be listed in the Drawing Explorer under the **Sheets** node.



- 3 The new sheet will initially be blank. Refer to section **11.5** for information related to inserting views.
- 4 To switch between sheets, select a sheet in the Drawing Explorer to view it.
- 5 To delete a sheet, right-click the sheet in the Drawing Explorer and select **Delete** from the pop-up menu.

11.3.14 Moving a View to Another Sheet

You can move a view to another sheet in a drawing.

To move a view to another sheet:

- 1 Right-click the view in the work area or Drawing Explorer and select **Move** from the pop-up menu. The **Select Target Sheet** dialog box appears listing the drawing sheets you can move the view to.
- 2 Select the sheet you want to move the view to.
- 3 Click **OK**. The view is moved to the specified sheet and is listed under the target sheet in the Drawing Explorer.

11.3.15 Hiding Parts in a View (Assemblies Only)

In a drawing of an assembly, you can hide parts in a view.

To hide a part in a view:

- 1 Right-click the part in the work area view and select **Hide Part** from the pop-up menu.

Or

- 1 In the Drawing Explorer, click the plus sign  next to the applicable view.
- 2 Click the plus sign  next to the assembly name to expand the list of associated parts.
- 3 Right-click the part and select **Hide** from the pop-up menu.

The part is hidden in the view and is dimmed in the Drawing Explorer. To unhide the part, right-click the hidden part in the Drawing Explorer and unselect **Hide**.

11.3.16 Inserting Images in a Drawing

You can insert images, such as logos, into drawings. Several popular file types are supported: JPG, GIF, TIF, BMP, RLE, DIB, RLE, EMF, WMF, PNG, JPE, JPEG, JFIF, and TIFF. Alibre Design does not support JPEG 2000 or lossless JPEGs, as well as some types of TIFF files.

When you save the drawing, the images are compressed and stored with the drawing. Opening the drawing opens all the images inside it.

When you export a drawing in the Alibre Design STEP format, the images are compressed and saved with it. When the STEP file is opened, the images are decompressed and loaded into the file.

When you save a drawing as a template after inserting an image, the image will remain part of the template when it is used again in another drawing. See section 11.6 for information on using custom templates.

To insert an image in a drawing:

- 1 From the Insert menu, select Image. The Select Image dialog box appears.
- 2 Click Look In to browse to the saved location of the image you want to insert.
- 3 Click the image's file name.
- 4 Click Open. The image appears in the drawing workspace and the cursor changes to a dotted crosshairs icon.
- 5 Click the crosshairs icon on the drawing sheet to place the upper-left-hand corner of the image. The image is placed in the selected location.
- 6 To resize the image, right-click it and select Scale from the pop-up menu. The Scale dialog box appears.
- 7 To make the image smaller, decrease the value on the left. To make the image larger, decrease the value on the right.
- 8 Click and drag the image to place it as necessary.

Note: You may also move images into Alibre Design by importing them to a repository or by dragging them from a Windows folder onto the Home window. A blank drawing sheet opens with the image placed on it.

11.3.17 Printing a Drawing

You can print one, all, or a specified list of sheets in a drawing. You can also print just a portion of the current sheet.

To print:

- 1 From the File menu, select **Print**.
- 2 Use the **Sheet range** to specify what you want to print:
 - All sheets in the drawing.

- The currently displayed portion of the current sheet.
 - The entire current sheet.
 - Only the sheets checked in the print dialog.
- 3 Specify the number of copies you want to print.
- 4 If you want the printed drawing to be fit to the size of the paper, check the **Scale to fit** option.
- 5 If you want to print using only black and white, check the **Print black and white** option.
- 6 Click **OK**.

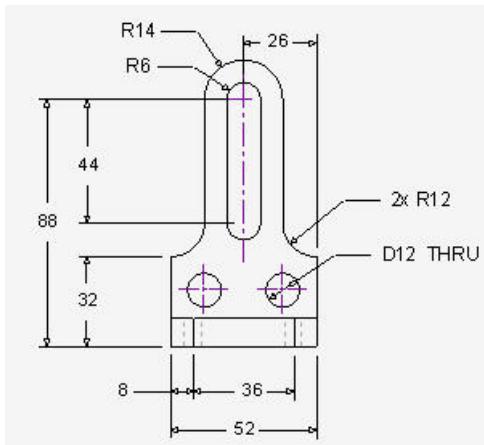
To see a print preview of the current sheet:

- 1 From the **File** menu, select **Print Preview**. The **Print Preview** window appears with a preview of the drawing.

11.4 Dimensioning

Typically, as you create features in part mode you place driving dimensions that define the associated sketch profiles. Additionally, dimensions that define a feature's size, such as extrusion depth, are also considered driving dimensions.

When you create a drawing based on the part, you can choose to display the **driving** dimensions from the part automatically on the applicable view in a drawing (see section 11.1.5). If you do not choose to display driving dimensions upon creating the drawing you can always do so later.



You can also manually insert additional dimensions on a drawing view. These user-added dimensions are referred to as **Reference** dimensions. Reference dimensions are displayed the same as driving dimensions. However, you can display a reference dimension in parentheses to distinguish it from driving dimensions. Right-click the reference dimension and select **Properties** from the pop-up menu. Select the **Units and Tolerance** tab, select the **Display As Reference Dimension** option, and click **OK**.

If a driving dimension is changed in part mode, the associated dimension in the drawing will automatically get updated. You can also edit driving dimensions in the drawing. If you change a driving dimension in the drawing, the part is automatically updated.

Reference dimensions cannot be edited or changed. However, reference dimensions will update upon the modification of driving dimensions. You can insert additional reference dimensions in drawings to further clarify the design intent.

11.4.1 Placing Additional Dimensions on a View

You can use the same methods to add dimensions to a drawing as you use to dimension sketches in part mode (refer to [Chapter 4](#)).

To place dimensions in a drawing:

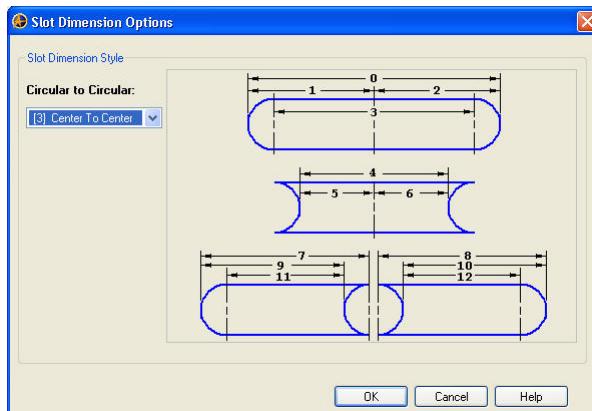
- 1 Select the **Dimension** tool from the Sketching toolbar.
- 2 Select the figure to dimension. A dimension preview appears, press **Esc** if the wrong figure was dimensioned.
- 3 When the preview displays the correct dimension location, click to place the dimension. The newly placed dimension is a reference dimension.

11.4.2 Dimensioning Slots and Holes

You can easily control how dimensions related to slots and holes are created and displayed.

To insert a dimension related to a slot or hole:

- 1 Select the **Dimension**  tool from the Sketching toolbar.
- 2 Select the first circular/radial figure or line to be dimensioned from. A dimension appears but do not click to place it.
- 3 Select a second circular/radial figure or line to be dimensioned to. The **Slot Dimension Options** dialog box appears.

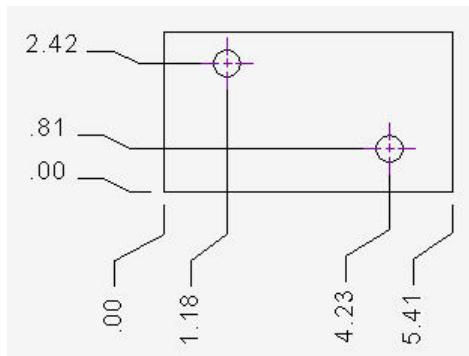


- 4 From the **Circular to Circular** or **Linear to Circular** list, select a dimension type from the drop down list. The numbers in the list correspond to the graphical key in the dialog box.
- 5 Click **OK**. The dimension is previewed in the work area.
- 6 Move the cursor to correctly position the dimension.
- 7 Click to place the dimension.

11.4.3 Placing Ordinate Dimensions on a View

To place ordinate dimensions:

- 1 Right-click in the work area and select **Ordinate Dimension** from the pop-up menu; or from the **Sketch** menu select **Ordinate Dimension**.
- 2 Select the **Baseline** figure.
- 3 Select the **Origin** on the baseline figure.
- 4 Drag the origin's dimension line away from the model and click to place it.
- 5 Select the entities (edges or points) you want to dimension using the same ordinate. As you select each entity, the dimension is placed in the view aligned to the origin.



Note: You can insert additional ordinate dimensions to a chain after initial placement. Select the Dimension tool, pick the baseline dimension, and then select the new dimension location to add the new dimension to the chain.

11.4.4 Modifying Driving Dimension Values

As previously described, you can change the values of driving dimensions in drawings. A change to a driving dimension in a drawing will be reflected in the part automatically.

To modify a driving dimension:

- 1 Make sure the **Dimension** selection filter is on (refer to section 9.3.4 for more information).
- 2 Select the **Select**  tool from the View toolbar if it is not already selected.
- 3 Move the cursor over the dimension. The dimension turns red.

-
- 4 Double-click the dimension. The dimension control box appears.



- 5 Enter a new value in the box and press **Enter**.

Changes made to driving dimensions in the drawing will be reflected in the part after the drawing has been saved and the part is opened.

11.4.5 Dimension Properties

You can set default dimension properties in the **Units** and **Dimension** tabs on the **Drawing Properties** dialog box (**File > Properties**). You can also set dimension properties on individual dimensions.

To modify the properties of an individual dimension:

- 1 Make sure the **Dimension** selection filter is on (refer to section 11.3.5 for more information).
- 2 Select the **Select**  tool from the View toolbar if it is not already selected.
- 3 Right-click a dimension and select **Properties** from the pop-up menu. The **Dimension Properties** dialog box appears.
- 4 To change properties related to dimension lines, extension lines and arrowheads, select the **Lines and Arrows** tab.
- 5 To change properties related to units, the display of the dimension value, and the tolerance format, select the **Units and Tolerance** tab.
- 6 To change properties related to the dimension text font, supplemental text, and text placement and orientation, select the **Text** tab.
- 7 To change properties related to dual dimensioning, select the **Alternate Units** tab.
- 8 Click **OK** to apply the changes.

11.5 Inserting Additional Views

In addition to the standard views, you can insert additional custom views to further clarify design intent. You can insert **auxiliary**, **detail**, **section**, and **exploded** views. You can also insert flat pattern views of sheet metal parts in Alibre Design Professional.

11.5.1 Standard View

You can insert an additional standard view as needed.

To insert a standard view:

- 1 Select the **Standard View**  tool from the Detailing toolbar; or from the **Insert** menu select **Standard View**. The **Insert Design** dialog box appears.
- 2 Follow the steps outlined in sections **11.1.4** and **11.1.5** to insert an additional standard view.

11.5.2 Auxiliary View

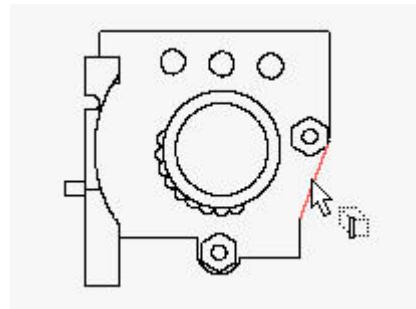
An auxiliary view is created by projecting an orthogonal view normal to a linear edge or sketch line in an existing view.

To create an auxiliary view:

- 1 Select the **Auxiliary View**  tool from the Detailing toolbar; or from the **Insert** menu select **Auxiliary View**.
- 2 Select a linear edge or straight sketch line on an existing view. The edge selected from the parent view must reside on a face that is perpendicular to the plane of the screen.

Note: To use a sketch line as the projection line, the sketch line must reside on the view. To do this, first select the view you wish to use to create the auxiliary view. Then select the **Activate 2D Sketch** tool from the sketching toolbar and sketch a line. The line can intersect the view boundaries. Exit Sketch Mode, then follow steps 1 and 2 above to create the new view. (After the auxiliary view is created, the sketch line can be deleted.)

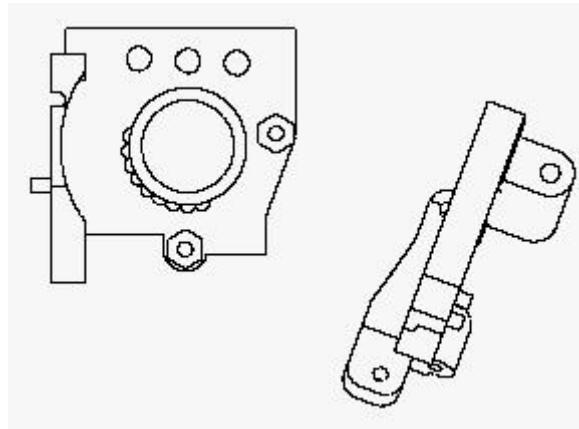
- 3 A preview of the auxiliary view is displayed.



Move the cursor to position the auxiliary view to the correct location.

- 5 Click to place the auxiliary view. The auxiliary view is placed on the sheet, aligned to the edge from which it was created. You can only move the auxiliary view in the direction normal to the edge from which it was created.

Note: To break the alignment between the auxiliary view and the parent view, right-click the view that is currently aligned, and select **Align**. Selecting Align will uncheck that option, and you can then drag the view unconstrained.

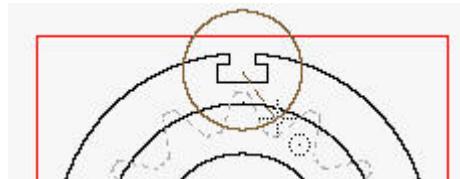


11.5.3 Detail View

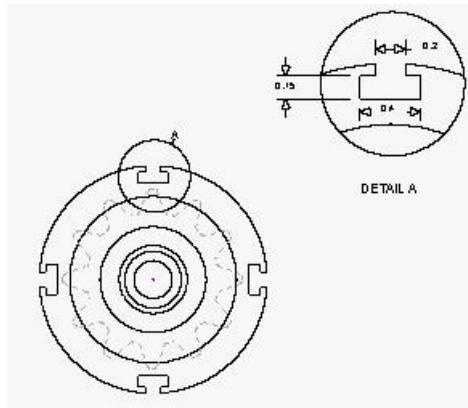
A detail view is a view that shows a portion of an existing view at an enlarged scale.

To create a detail view:

- 1 Select the **Sketch Mode**  tool from the Sketching toolbar.
- 2 Sketch any closed figure enclosing the area that you want to detail.



- 3 Select the **Detail View**  tool from the Detailing toolbar; or from the **Insert** menu select **Detail View**.
- 4 Select the sketched circle. A preview of the detail view appears.
- 3 Move the cursor to position the detail view appropriately. The detail view can be placed anywhere on the sheet. Click to locate the view. The default scale for the detail view is the same as the sheet scale. To enlarge the detail view, right-click it and select **Scale** from the pop-up menu.

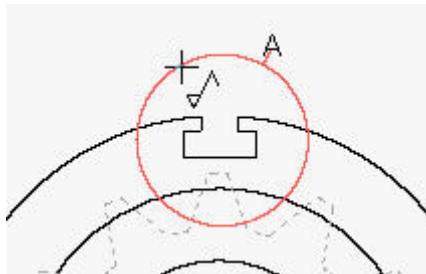


You can place dimensions on the detail view. You can also change both the detail circle label and the detail note. To change either, double-click the text or right-click the text and select **Edit** from the pop-up menu. The **Note** dialog box appears containing the original text. Enter the new text and click **OK**.

Also, in the **Detailing** tab of the **Drawing Properties** dialog, you can pre-define the border style and font used for detail circles and detail view labels.

To change the detail location:

- 1 Select the **Select**  tool from the View toolbar if it is not already selected.
- 2 Move the cursor over the detail circle annotation. The annotation is highlighted and the cursor changes.



- 3 Click and drag the detail circle annotation to the new location.
- 4 Release the mouse button to reposition the circle. The detail view is updated automatically to reflect the detail circle position change.

To change the detail area size:

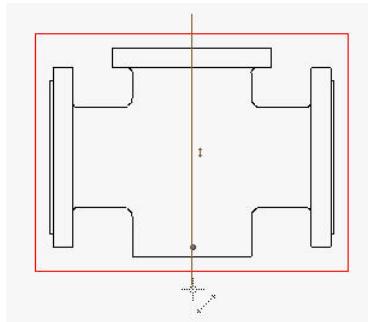
- 1 Select the **Select**  tool from the View toolbar if it is not already selected.
- 2 Move the cursor over the detail circle annotation. The annotation is highlighted and the cursor changes.
- 3 Double-click the annotation. The **Detail View Annotation** dialog box appears and the circle center node is displayed.
- 4 Move the cursor over the circle.
- 5 Click and drag to resize.
- 4 Release the mouse button when the circle is resized appropriately.
- 5 Click **OK** in the Detail View Annotation dialog box. The detail view is updated automatically to reflect the change in size of the detail area.

11.5.4 Section View

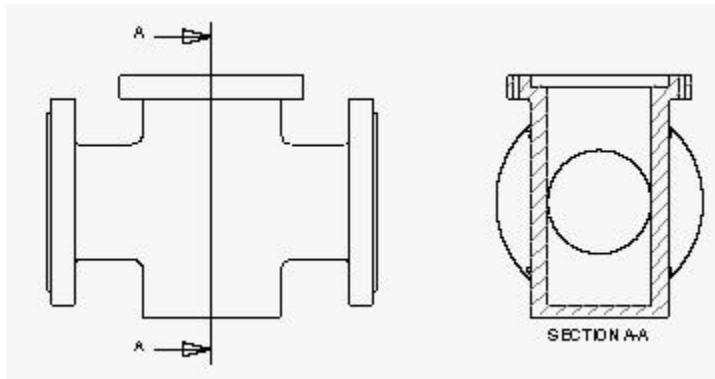
A section view represents a 2D cross-section of a model. Section views are created from other views and are dependent on them. You can create a normal section view or a stepped section view.

To create a normal section view:

- 1 Sketch a straight line across the view to define the section location. Make sure you are sketching in the view by selecting the view in the Design Explorer before entering Sketch Mode.



- 2 Select the **Section View**  tool from the Detailing toolbar; or from the **Insert** menu select **Section View**.
- 3 Select the line that you sketched in step 1. A preview of the section view appears in the drawing.
- 4 Drag the section view preview to the appropriate location on the sheet. You will only be able to move the view in a direction normal to the section line.
- 5 Click to place the view.



Note: In the **Detailing** tab of the **Drawing Properties** dialog, you can pre-define the display style for the section line, the font for the section line label, and the default hatch pattern style.

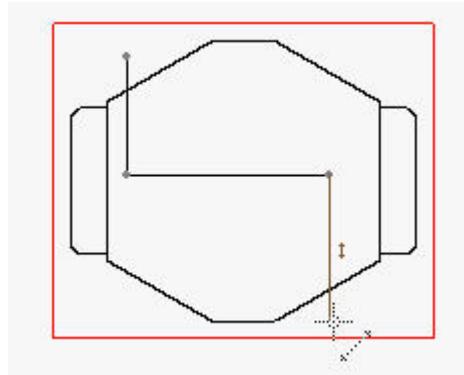
Note: For advanced users who want to customize the available hatch patterns in Alibre Design: You can modify the predefined hatch patterns that ship with Alibre Design by editing the text file, alibre_unicode.pat. You can use **Notepad** to edit this file. A definition of the file format is embedded within the file. This file is located in the folder **C:\Documents and Settings\All Users\Application Data\Alibre Design\System Files**.

To modify a section view:

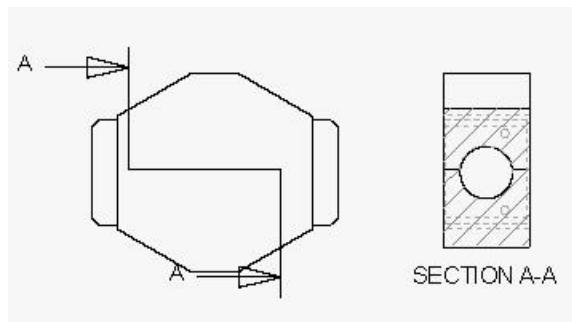
- To edit the letter label on the section line or the note on the section view, double-click the item, or right-click and select **Edit** from the pop-up menu. The **Note** dialog box appears containing the original text. Enter the new text and click **OK**.
- To change the cut direction, right-click the section view and select **Reverse Section View** from the pop-up menu.
- The section view crosshatch pattern is set in the **Detailing** tab on the **Design Properties** dialog box (**File > Properties**). You can modify the crosshatch pattern for an individual section view. Right-click the section view and select **Change Cross Hatch** from the pop-up menu. The **Hatch Properties** dialog box appears. Select a new crosshatch pattern from the list or modify the **Scale** or **Angle**. Click **OK** to apply the changes.

To create a stepped section view:

- 1 Sketch a series of line segments across the view to define the section location.



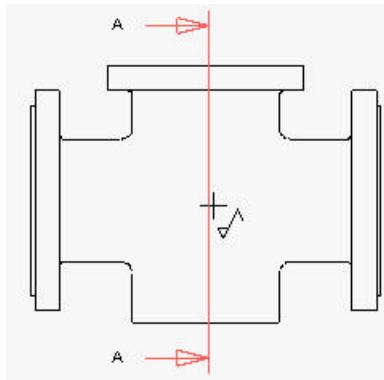
- 2 Select the **Section View**  tool from the Detailing toolbar; or from the **Insert** menu, select **Section View**.
- 3 Select any of the line segments that you sketched in step 1. A preview of the section view appears in the drawing.
- 4 Drag the section view preview to the appropriate location on the sheet. You will only be able to move the view in a direction normal to the section line.
- 5 Click to place the view.



To redefine the section location (either normal or stepped):

- 1 Select the **Select**  tool from the View toolbar if it is not already selected.

- 2 Move the cursor over the section line annotation. The annotation is highlighted and the cursor changes.



- 3 Click and drag the line to the desired location.
- 4 Release the mouse button to place the line. The section view updates automatically based on the new section line location.

To change the direction of the section view:

- 1 Right-click the section view in the work area and select **Reverse Section View** from the pop-up menu.

Or

Right-click the section view in the Drawing Explorer and select Reverse Section View from the pop-up menu.

The section line is flipped and the section view is displayed from the opposite direction.

To change the hatch pattern for all parts in the section view:

- 1 Right-click the section view in the work area and select **Change Cross Hatch** from the pop-up menu; or right-click the section view in the Drawing Explorer and select **Change Cross Hatch** from the pop-up menu.

The **Hatch Properties** dialog box appears.

- 2 Select a new cross hatch pattern from the **Pattern** pull down menu.
- 3 Specify the cross hatch **Scale**.

- 4 Specify the cross hatch **Angle**.
- 5 If desired, you can also modify the **color**, **line weight**, and **offset** distance for the hatch pattern.
- 6 Click **OK** to apply the new cross hatch pattern.

Note: You can also access the cross hatch settings from the Drawing Properties dialog box. From the **File** menu, select **Properties** and then select the **Detailing** tab in the dialog box.

To change the hatch pattern for individual parts in a section view:

- 1 In the Drawing Explorer, expand the assembly under the section view to reveal the individual parts displayed in the section.
- 2 Right-click the desired part and choose **Change Cross Hatch** from the pop-up menu.
The **Hatch Properties** dialog box appears.
- 3 Modify the hatch pattern properties as desired.
- 4 Click **OK** to apply the new cross hatch pattern to the selected part.

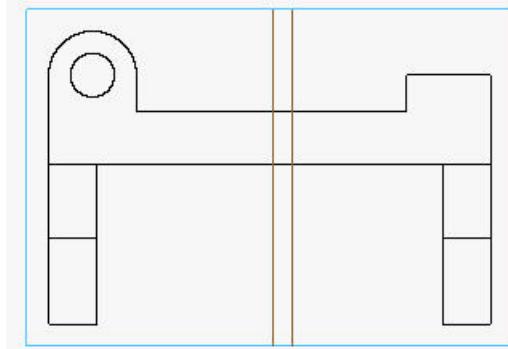
11.5.5 Broken View

You can create a broken view in a drawing of a long part that has a uniform cross-section. Creating a broken view is useful when you want to display a part with a larger scale on a smaller size drawing sheet.

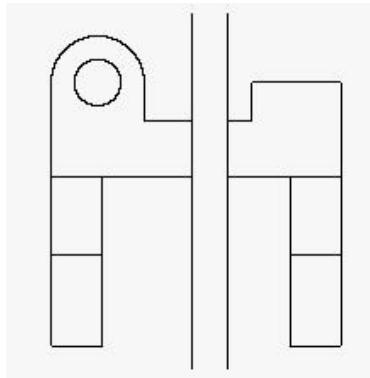
To insert a broken view:

- 1 Select the  tool from the View toolbar if it is not already selected.
- 2 Right-click the view you want to break in the Drawing Explorer or the work area and select **Create Broken View** from the pop-up menu; or select the view and then from the **Insert** menu, select **Broken View**; or select the view and then select the  tool from the Detailing toolbar.

Two break lines appear in the view and the **Broken View** dialog box appears.



- 3 Select a break line **Style**. You can use **Straight**, **Zig**, or **Curve** break lines.
- 4 Drag the break lines to the appropriate break locations in the view.
- 5 Specify the break **Width**.
- 6 Specify a break line **Angle** if desired.
- 7 Click **OK**. The part is displayed with a break in the geometry.



Reference dimensions and part dimensions associated with the broken area reflect the actual value.

To modify the broken view:

- 1 Select the **Select**  tool from the View toolbar if it is not already selected.

- 2 Move the cursor over one of the break lines.
- 3 Right-click the break line and select **Edit** from the pop-up menu. The **Broken View** dialog box appears.
- 4 Change the **Style**, **Width**, or **Angle** as necessary.
- 5 Drag the break lines to redefine the break position.
- 6 Click **OK** when finished.

To restore the broken view to its original state:

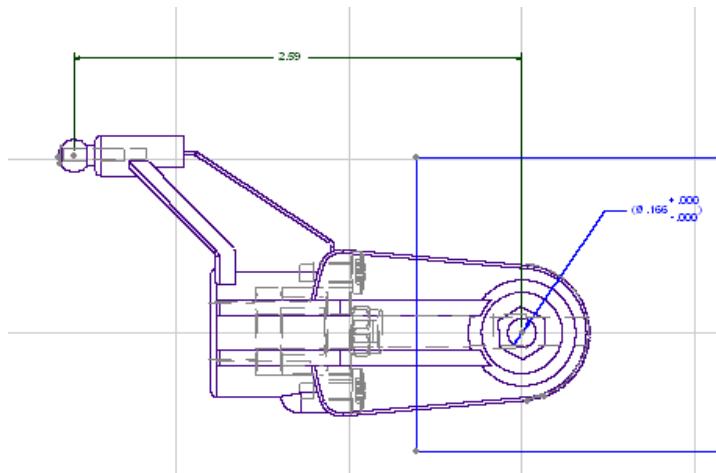
- 1 Select the **Select**  tool from the View toolbar if it is not already selected.
- 2 Move the cursor over one of the break lines.
- 3 Right-click the break line and select **Delete** from the pop-up menu. The break lines are deleted and the view is restored to its unbroken state.

11.5.6 Partial View

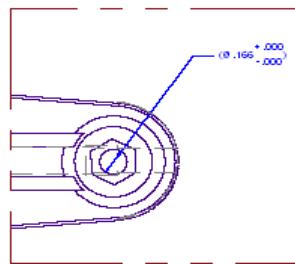
You can modify an existing view to create a partial view. A partial view allows you to only show a portion of an existing view.

To insert a partial view:

- 1 Select the **Select**  tool from the View toolbar if it is not already selected.
- 2 Select the view that you wish to transform into a partial view, then select the **Sketch Mode** tool from the Sketching toolbar. Partial views can be created from any other view, including primary and dependent views.
- 3 Sketch any closed figure enclosing the area that you wish to keep in the partial view.



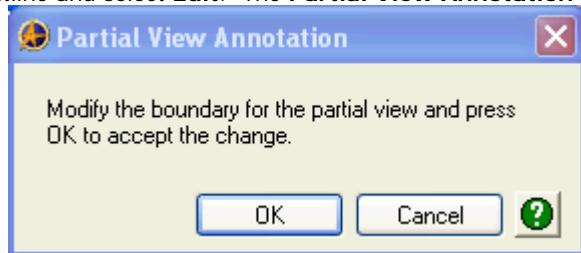
- 4 Select the **Partial View** tool from the Detailing toolbar; or from the **Insert** menu select **Partial View**.
- 5 Select the sketched closed figure. The view will transform into a partial view.



To change the partial view area size:

- 1 Select the **Select** tool from the View toolbar if it is not already selected.
- 2 Move the cursor over the partial view area dashed outline.

- 3 Right-click the dashed outline and select **Edit**. The **Partial View Annotation**



dialog box appears.

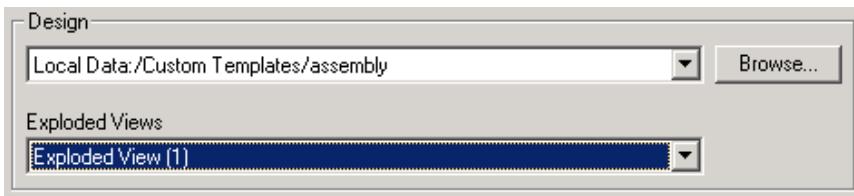
- 4 Make any necessary changes to the partial view outline. You can only modify the exiting sketch lines; you cannot sketch new ones.
- 5 In the Partial View Annotation dialog box, select **Cancel** to discard the changes, or **OK** to update the partial view.

11.5.7 Exploded View

You can insert a 2D exploded view representation of any exploded view you created in the assembly workspace.

To create an exploded view of an assembly:

- 1 Select the **Standard Views**  tool from the Detailing toolbar; or from the **Insert** menu, select **Standard Views**. The **Standard Views Creation** dialog box appears.
- 2 In the **Design** area, select the assembly item from the drop down list that you want to insert an exploded view of; or click **Browse** to select the assembly from the Document Browser. Any exploded views that were saved with the assembly are subsequently listed in the Exploded Views drop down list.
- 3 Select the appropriate exploded view to insert.



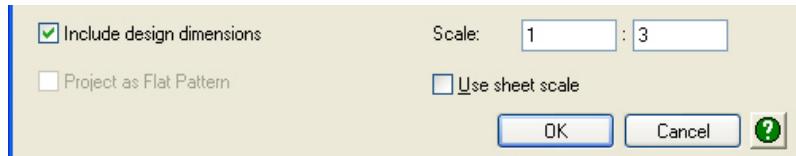
- 4 Use the arrow buttons in the **Front View** area as well as the View Selection buttons to select the appropriate exploded view orientation(s).
- 5 Click **OK**. A preview of the exploded view appears in the work area.
- 6 Move the cursor to position the view.
- 7 Click to place the exploded view on the sheet.

11.5.8 Flat Pattern View of a Sheet Metal Part

You can insert a flat pattern representation of any sheet metal part you created. Bend lines will be shown in a flat pattern view, and they can be used to create dimensions.

To create a flat pattern view of a sheet metal part:

- 1 Select the **Standard Views**  tool from the Detailing toolbar; or from the **Insert** menu, select **Standard Views**. The **Standard Views Creation** dialog box appears.
- 2 Follow steps 1-4 in section 11.1.5 for creating standard views. Then check the Project as Flat Pattern box as shown below:



- 3 Click **OK**. A preview of the flat pattern view appears in the work area.
- 4 Move the cursor to position the view.

- 5 Click once to place the view on the sheet.

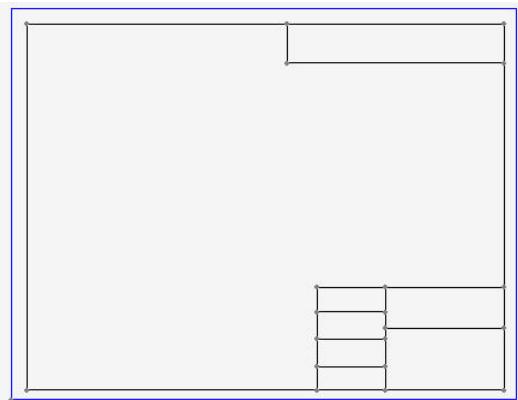
11.6 Custom Templates

You can create a custom drawing template that meets your design process requirements. A custom template can be created in two different ways: as a symbol that you insert into a drawing, or saved as a drawing that you select from the **New Sheet Properties** dialog.

11.6.1 Creating a Custom Template

To create a custom template:

- 1 Open a new drawing workspace. The **New Sheet Properties** dialog box appears.
- 2 Select **Blank Sheet**.
- 3 Select a sheet size from the pull-down list.
- 4 Click the **Create An Empty Drawing** check box.
- 5 Click **OK**. An empty drawing workspace appears.
- 6 In the work area, sketch the border and title block using the sketch tools.



- 7 Create text labels or standard information such as your company's name and address. From the **Sketch** menu select **Text > Label**. Text labels can be used for making title block labels such as **Scale**, **DRW**, **PART NO.**, etc.

- 8 Set up fields that collect data from the user for variable information such as Drawn by, Date, Designed For, Drawing Number, Scale, etc. From the **Sketch** menu select **Text > Field**. Text fields can be used for making title block fields to receive data that corresponds with SCALE, DRW, PART NO., etc.
- 9 Save the custom template as a drawing (section 11.6.3) or as a symbol (section 11.6.4).

11.6.2 Customizing an Existing Template

To customize an existing template:

- 1 Open a new drawing workspace. The **New Sheet Properties** dialog box appears.
- 2 Select the **Template** option and select a template from the list. To open a custom template not listed, click **Browse** and select the template from the **Custom Drawing Template** dialog. **Note:** If you have previously **browsed** to another template folder, you can use the **Default** button to reset the template list back to the system template folder.
- 3 Click **OK** in the Sheet Properties dialog box.
- 4 If the **Fill In Text** dialog box appears, complete any applicable fields.
- 5 Click **OK** when finished with the Fill In Text dialog box. The **Insert Design** dialog box appears.
- 6 Click **Cancel**.
- 7 Right-click the sheet in the Drawing Explorer and select **Activate Sketch on Sheet** from the pop-up menu.
- 8 Move the cursor over the drawing border. The border is highlighted.
- 9 Right-click and select **Explode Symbol** from the pop-up menu. The template is exploded into individual segments.
- 10 Make the necessary changes to the template.
- 11 From the **Sketch** menu, select **Activate Sketch** to leave the sketch mode.
- 12 Save the custom template as a drawing (section 11.6.3) or as a symbol (section 11.6.4).

11.6.3 Saving and Using a Custom Template as a Drawing

To save a custom template as a drawing:

- 1 From the **File** menu, select **Save As**. The Save As dialog box appears.
- 2 Navigate to the desired location in either a repository or the file system.
- 3 Specify a name for the drawing template.
- 4 Click **Save**.

To use a custom template drawing:

- 1 Start a new drawing.
- 2 In the **New Sheet Properties** dialog, choose **Template**, and select the **Browse** button.
- 3 In the **Custom Drawing Template** dialog, browse through your repository or file system to find the desired template, which you have already saved, and click it.
- 4 Click **OK** in the **Custom Drawing Template** dialog (The OK button is slow to activate here).
- 5 Click **OK**.
- 6 You will be prompted to fill in any default and user added text if you put any text fields in your template. Enter the desired information; then click **OK**. Your drawing format will appear, with the prompt to select a model.

Note: The next time the **New Sheet Properties** dialog is invoked, the **Template** list will be populated with all the custom templates located in the folder you last browsed to in step 2 above. You can use the **Default** button to reset this list to the default template folder.

11.6.4 Saving and Using a Custom Template as a Symbol

To save a custom template as a symbol:

- 1 Select the **Activate Sketch**  tool from the Sketching toolbar.
- 2 From the **Sketch** menu select **Create Custom Symbol**. The Create Custom Symbol dialog box appears.
- 3 In the workspace, drag a selection box around the figures, fields, and labels that you want to include in the template. These elements appear in the **Figures to include** list.
- 4 In the dialog box, move your cursor to **Anchor Point** and click in the **X** or **Y** boxes.
- 5 Click in the work area where you want the bottom left corner of the template to appear in a new Drawing workspace. The coordinates appear in the **Anchor Point** area and are relative to the origin. Maintaining a (0,0) anchor point is acceptable.
- 6 Click **OK**. The **Save Custom Symbol** dialog box appears.
- 7 In the Repository Explorer, browse to the appropriate save location and select a repository and/or folder.
- 8 Specify a name for the drawing template
- 9 Click **Save**.

To use a custom template symbol:

- 1 Open a new drawing workspace. The **New Sheet Properties** dialog box appears.
- 2 Select **Blank Sheet**, and the corresponding sheet size.
- 3 Click the **Create An Empty Drawing** check box.
- 4 Specify the drawing **Scale**.
- 5 Click **OK**. A blank drawing workspace appears.
- 6 From the **Sketch** menu select **Activate Sketch**.
- 7 From the **Sketch** menu select **Insert Custom Symbol**. The **Select Custom Symbol** dialog box appears.
- 8 Browse the Repository Explorer and select the custom symbol you created to use as a template.

- 9 Click **OK**.
- 10 In the work area, select a location corresponding to the anchor point defined in the custom template. If your anchor point was defined at (0, 0), pick the sketch node representing the origin of the sheet. The custom template appears in the work area.
- 11 If any fields were created with the custom template, a dialog box appears containing the field properties. Enter the text associated with the fields.
- 12 Click **OK**.

11.7 Annotations

You can insert various annotation types into a part, assembly, or drawing workspace to describe and clarify design and manufacturing information. You can insert notes, datums and datum targets, feature control frames, surface finishes, weld symbols, and balloon callouts.

To insert annotations in any workspace, from the **Insert** menu select **Annotations** and then select from the available annotation types.

In a drawing workspace, the annotation tools are also displayed on the Detailing toolbar.



- Note** . . . insert a note annotation
- Datum** . . . insert a datum annotation
- Datum Target** . . . insert a datum target annotation
- Feature Control Frame** . . . insert a feature control frame notation
- Surface Finish** . . . insert a surface finish annotation
- Weld** . . . insert a weld symbol annotation
- Callout** . . . insert a balloon callout annotation

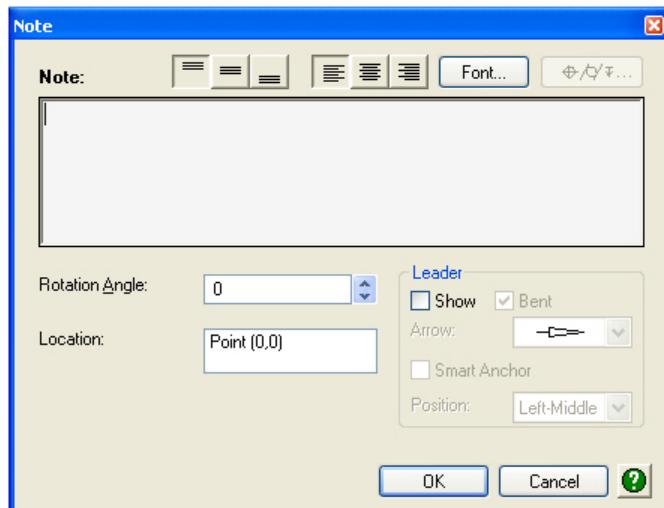
You can pre-define certain display characteristics for annotations, including arrow types and sizes, text font, and the shape of balloon callouts, in the **Annotations** tab of the **Drawing Properties** dialog.

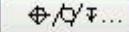
11.7.1 Note

You can create a free-floating note or a note with a leader pointing to an edge or face. The note can contain both text and symbols.

To insert a note:

- 1 In any workspace, from the **Insert** menu select **Annotation > Note**; or in a drawing workspace, select the **Note**  tool from the Detailing toolbar. The **Note** dialog box appears.



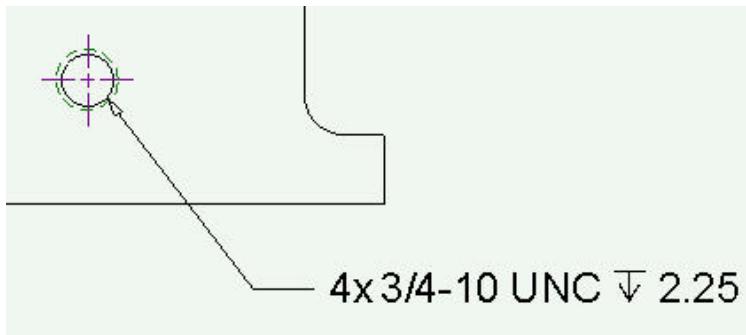
- 2 In the **Note** area, type the annotation text. You may align the text vertically and horizontally by clicking the text alignment icons above the **Note** field.
- 3 Click **Font** to specify the font, font style, size, color, and effects.
- 4 To insert a symbol, click the **Symbols**  button. The **Insert Alibre Design Symbols** dialog box appears. Click a symbol to insert it. Click **Close** to close the symbols box.
- 5 Specify a **Rotation Angle** if required.
- 6 If you want to include a leader, click the **Show** option in the **Leader** area.
- 7 Select the **Bent** option if you want the leader line to have a short horizontal segment near the annotation.
- 8 From the **Arrow** pull-down menu, select the arrow type you want to use.
- 9 From the **Position** pull-down menu, select the position in which the text will be placed in relation to the leader.

- 10 Select **Smart Anchor** if desired. The smart anchor option will automatically adjust the position of the text if the leader is re-positioned. The smart anchor option overrides the position selection in step 9.
- 11 Click in the **Location** box and then click in the work area to select the location of the note. If you are using a leader, click a figure or face to attach the note to. The selection coordinates are displayed in the **Location** box.
- 12 Click **OK** to finish placing the note.

Note: In drawings, if you insert a note while a view is active, the note will be attached to that view. If you insert a note while the sheet is active, the note will be attached to the sheet.

11.7.2 Displaying Hole Callouts and Threads in Views

Threaded hole information, if applied in the 3D design, will automatically be called out upon creation of views in the 2D drawing. Cosmetic threads, also displayed by default, are represented graphically by dashed lines.



The default callout includes the number of identical holes, the type of thread, and the thread depth.

You can control whether hole callouts and cosmetic threads are automatically created with new views. The options to control this are under **View Creation Options** in the **Detailing** tab of the **Drawing Properties** dialog.

**To manually apply a hole callout:**

- 1 Move the cursor over the hole. The hole is highlighted.
- 2 Right-click and select **Insert Hole Callout** from the pop-up menu. The hole callout is displayed.
OR
- 1 Select the hole.
- 2 From the **Insert** menu, select **Hole Callout for Hole**.

Note: You can show the callouts for an entire view by right-clicking a view in the Drawing Explorer, or right-click a view in the work area and select **Insert Hole Callouts**.

To edit the hole callout:

- 1 Move the cursor over the hole callout. The cursor changes and displays the annotation symbol.
- 2 Right-click the hole callout and select **Edit** from the pop up menu. The **Hole Callout** dialog appears.
- 3 Modify the **Callout Note** as necessary.
- 4 Modify the **Leader** parameters as necessary.
- 5 Click **OK** when finished.

To manually apply cosmetic threads:

- 1 Move the cursor over hole. The hole is highlighted.
- 2 Right-click and select **Insert Cosmetic Threads** from the pop-up menu. The cosmetic threads are displayed for the selected hole.

OR

- 1 Select the hole.
- 2 From the **Insert** menu, select **Cosmetic Thread for Hole**.

Note: You can show the cosmetic threads for an entire view by right-clicking a view in the Drawing Explorer, or right-click a view in the work area and select **Insert Cosmetic Threads**.

Note: You can show the cosmetic threads as a $\frac{3}{4}$ circular figure by checking the **Three Quarter Circle** box in the View Creation Option section of the Detailing tab in the Drawing Properties dialog.

To delete a cosmetic thread:

- 1 Move the cursor over the cosmetic thread symbol. The cursor changes and displays the annotation symbol.
- 2 Right-click the cosmetic thread and select **Delete** from the pop up menu.

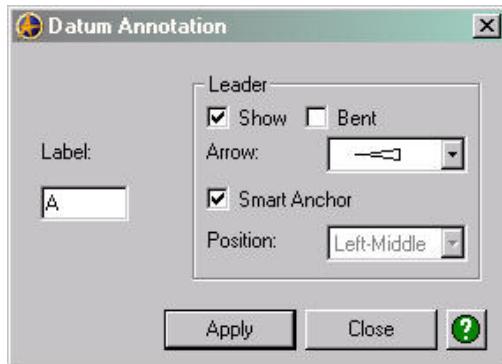
11.7.3 Datums

In part or assembly workspaces, you can attach datum annotations to faces of models (but not to vertices or edges). In drawing workspaces, you can place datum annotations at any location.

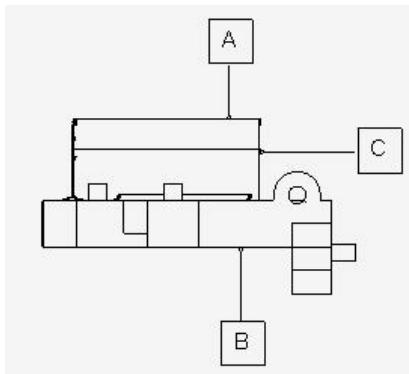
To insert a datum annotation:

- 1 In any type of workspace, from the **Insert** menu select **Annotation > Datum**. In a drawing workspace, select the **Datum**  tool from the Detailing toolbar.

The **Datum Annotation** dialog box appears.



- 2 In the **Datum Label** box, specify the letter you wish to start the datum series with.
- 3 Select the **Show** option if want to use a leader with the datum. Select the **Bent** option if you want the leader line to have a short horizontal segment near the annotation.
- 4 From the **Position** pull-down menu, select the position in which the text will be placed in relation to the leader.
- 5 Select **Smart Anchor** if desired. The smart anchor option will automatically adjust the position of the text if the leader is re-positioned. The smart anchor option overrides the position selection in step 4.
- 6 From the **Arrow** pull-down menu, select the type of arrow to use with the leader.
- 7 In the work area, select the face, edge, or view to attach the annotation. If a leader is used, two clicks are required.
- 8 Continue to click to populate a series of datums.
- 9 Click **Apply** and **Close** when finished.



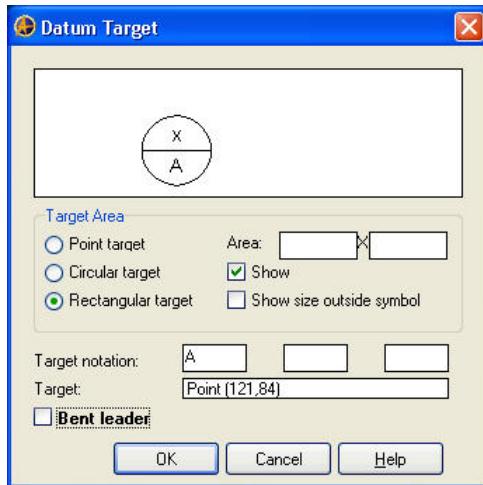
Datums can be resized and repositioned by clicking the text and dragging. This will change the length of the leader, as well as reposition the datum box. Datums can also be moved to another location by clicking the arrow, as opposed to the text.

11.7.4 Datum Targets

In part or assembly workspaces, you can attach datum target annotations to faces of models (but not to vertices or edges). In drawing workspaces, you can place datum annotations at any location.

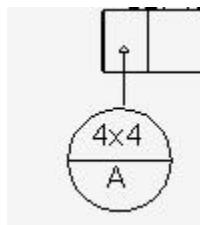
To insert a datum target:

- 1 In any type of workspace, from the **Insert** menu select **Annotation > Datum Target**. In a drawing workspace, select the **Datum Target** tool from the Detailing toolbar. The **Datum Target Annotation** dialog box appears.



The top area of the Datum Target dialog box previews the annotation as you build it.

- 2 Select the target type: **Point**, **Circular**, or **Rectangular**.
- 3 In the **Area** fields, specify the target size.
- 4 Select the **Show** option to show or hide the datum target.
- 5 Select the **Show size outside symbol** option if desired.
- 6 In the **Target notation** fields, specify the datum reference label(s).
- 7 Click in the **Target** field.
- 8 In the work area, select the face, edge, or view to attach the annotation. Drag the target away from the selection to lengthen the leader.
- 9 Select the **Bent Leader** option if necessary.
- 10 Click to anchor the annotation.
- 11 Click **OK**.



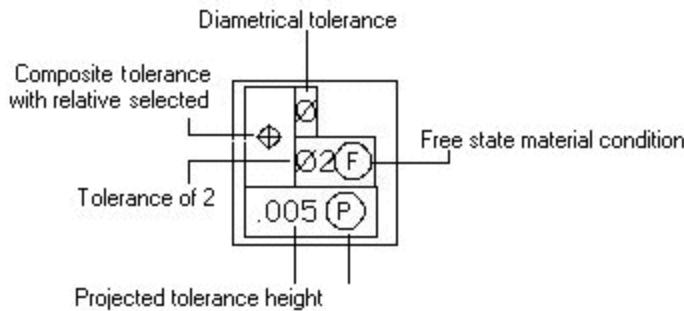
The datum target symbol can be repositioned by clicking the text and dragging. This will change the length of the leader, as well as reposition the datum target symbol. Datum targets can also be moved to another location by clicking the arrow, as opposed to the text.

11.7.5 Feature Control Frames

The geometric tolerance annotations let you specify a reference frame that contains all the geometric tolerance information for a selected surface or feature. The annotations support both the ANSI Y14.5 M-1982 and the 1994 standards.

Example

ANSI-Y14.5 M-1982 Standard



You also include datum references if the geometric tolerance is related to a datum.

You reference primary, secondary, and tertiary datum reference frames with the following material conditions for each geometric tolerance:

(M) MMC—maximum material condition

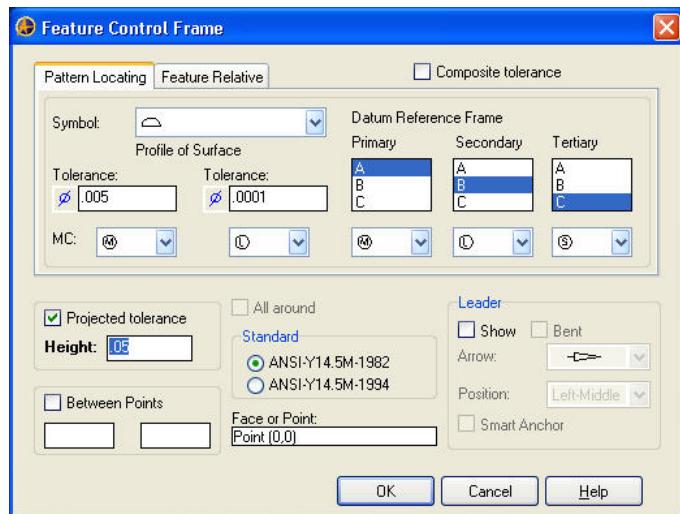
(L) LMC—least material condition

(S) RFS—regardless of feature size

(F) Free State—not limited by state position.

To create a feature control frame:

- 1 In any type of workspace, from the **Insert** menu select **Annotation > Feature Control Frame**. In a drawing workspace, select the **Feature Control Frame** tool from the Detailing toolbar. The **Feature Control Frame** dialog box appears.



- 2 If the geometric tolerance is related to a datum, specify as many as three datum references that form the **Datum Reference Frame**. For the primary, secondary, and tertiary datums:
 - Click the **Datum** reference letter. (The reference letters were created when you inserted the Datum annotations.)
 - Click the **MC** arrow for the datum, and select the material condition for that datum.

- 3 Click the **Symbol** arrow, and select one or more of the displayed tolerance symbols.

The geometric tolerance symbols indicate controls for form, profile, orientation, location, and runout.

Symbol Control	Type
	Orientation Angularity
	Form Circle
	Location Concentricity
	Form Cylindricity
	Form Flatness
	Orientation Parallelism
	Orientation Perpendicularity
	Location Position
	Profile Line Edge
	Profile Surface
	Runout Simple
	Form Straightness
	Locations Symmetry
	Runout Total

- 4 Specify the allowed **Tolerance** values.

- 5 Select the **Diametrical Tolerance**  symbol if the tolerance is associated with a diameter zone.
- 6 Select a material condition from the **MC** pull-down menu for each tolerance value.
- 7 Repeat steps 2-6 on the Feature **Relative** tab if you want to specify a stacked feature control frame. This allows you to see a different symbol for each row. If you want a composite tolerance instead (one symbol shared across rows), check the **Composite Tolerance** checkbox.
- 8 Select the **Projected Tolerance** option if required and specify the **Height** of the projected tolerance zone.
- 9 Select the **Standard** that you want to use for the tolerance symbol. Alibre Design supports ANSI Y14.5 M-1982 and 1994. The preview shows the symbol for the selected standard.
- 10 Select the **Between Points** option to call out a tolerance between two points. Enter labels for the two points.
- 11 Select the **Show** option to include a leader, the **Bent** option if desired, and select an **Arrow** type.
- 12 Select **Smart Anchor** if necessary.
- 13 Click in the **Face** or **Point** field.
- 14 In the work area, select a face, edge, or location to place the annotation.
- 15 Click **OK**.

11.7.6 Surface Finish Symbol

In drawing, part, or assembly workspaces you can specify the surface texture of a face by using a **Surface Finish Symbol**.

To create a surface finish symbol:

- 1 In any type of workspace, from the **Insert** menu select **Annotation > Surface Finish**. In a drawing workspace, select the **Surface Finish**  tool from the Detailing toolbar. The **Surface Finish** dialog box appears.

2 From the **Symbol** pull-down menu, select the machining method for the surface finish.

3 From the **Lay Direction** pull-down menu, select the direction of the surface pattern.

4 In the **Roughness** area:

Specify a value for the **Maximum** allowable height deviation from the surface mean plane.

Specify a value for the **Minimum** allowable height deviation.

Specify a value for the average **Spacing** of roughness peaks.

Specify a value for the roughness **Sampling** length.

5 Select the symbol **Standard** that you want to use. Alibre Design supports ANSI Y14.16, ISO 1302, and JIS Symbols.

6 In **Material Removal**, specify the value for the amount of stock to be removed by the machining method that you selected.

7 In the **Waviness** area:

Type a value for the **Waviness** (peak-to-valley height) of the waves.

Type a value for the **Spacing** between adjacent peaks.

8 If you want to specify the **Production Method** to be used for the surface finish, type it in the box provided.

9 If you want to include a leader:

Select **Show**.

Select **Bent** if you want the line to have a short horizontal segment near the annotation.

Click the **Arrow** down arrow, and select a style.

10 Click in the **Face or Point** field.

11 In the work area, select the face or edge to attach the annotation to, drag the annotation, and click to place it.

12 Use the **Rotation Angle** to change the angle at which the symbol is displayed.

13 Check the **Flip Text** box if you desire to flip the text 180°.

Note: The Flip Text box will automatically become checked if the rotation angle goes above 90°. However, you can uncheck the box if you do not want the text flipped.

14 Click **OK**.

11.7.7 Weld Symbol

To create a weld symbol:

- 1 In any type of workspace, from the **Insert** menu select **Annotation > Weld**. In a drawing workspace, select the **Weld**  tool from the Detailing toolbar. The **Weld** dialog box appears.
- 2 Click the **Far** or **Near** tab, depending on where you want the annotation placed in relation to the design.
- 3 From the **Finishing method** pull-down menu, select the method that you want to specify.
- 4 From the **Contour** pull-down menu, select the shape that you want for the weld surface.
- 5 Specify a **Groove angle** value in degrees.
- 6 Specify a **Root opening** value.
- 7 In the **Weld symbol** area, select the weld symbol from the pull-down menu. You can also type text into the boxes on both sides of the weld symbol.
- 8 In the **Joint with spacer** area, select the spacer type from the pull-down menu.
- 9 Select the applicable weld placement options:
 - **All around**
 - **Field or site weld**
 - **Display pointing down** (enabled if you select Field or site weld)
 - **Stagger weld** (enabled if you select a fillet for both the Near and Far tabs)
- 10 In the **Specification process** area, type any additional instructions to be included with the weld symbol.
- 11 If you are specifying both a **Near** and a **Far** weld, click the other tab, and repeat the steps for the other weld.
- 12 If you want to include a leader:

- Select **Show**.
- Select **Bent** if you want the line to have a short horizontal segment near the annotation.
- In **Arrow**, select an arrow style.

13 Click in the **Face or Point** field.

14 In the work area, click a face or edge to attach the weld symbol to.

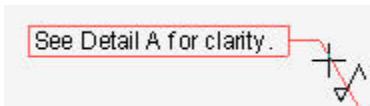
15 Click **OK**.

11.7.8 Editing and Deleting Annotations

You can edit or delete an annotation anytime after it has been created and placed.

To edit or delete an annotation:

- 1 Select the **Select**  tool from the View toolbar.
- 2 Move the cursor over the annotation. The annotation is highlighted and the cursor includes the symbol icon when it is over the annotation that can be edited or deleted.



- 3 Right-click the annotation and select **Edit** or **Delete** from the pop-up menu. In the **Edit** case, the **Annotation** dialog box appears.
- 4 Make the necessary changes to the annotation properties.
- 5 Click **OK**.

12 Bills of Material

You can create a bill of material (BOM) for an assembly, as well as a part if necessary. You can create a custom BOM, or create a new BOM from a template.

The bill of material is fully associative to the assembly and/or drawing. A change made in the assembly (e.g. adding or removing parts and subassemblies) is automatically applied to the BOM.

You can launch the BOM workspace directly from the drawing. Changes made in the BOM workspace will be updated automatically in the drawing. Manual changes to the BOM are not reflected in the associated assembly (or part).

This chapter describes:

- Creating a new BOM
- Creating a BOM template
- Inserting a BOM view into a drawing
- Working with a BOM in a drawing
- Editing a BOM
- Customizing the BOM display
- Printing and exporting BOM data

12.1 Specifying BOM Data

You can specify BOM related properties for a part or for an assembly that you want to treat as a part for BOM purposes. Consequently, these properties can be displayed in a BOM of any assembly that contains the part or assembly.

To save BOM data with a part:

- 1 From the **File** menu, select **Properties**. The **Design Properties** dialog box appears.
- 2 Select the **General** tab.
- 3 Scroll through the **Property** list to find the applicable BOM property.
- 4 To enter BOM property data, click in the corresponding value field. The cursor appears.
- 5 Type in the appropriate text.
- 6 Continue specifying value fields as required.
- 7 Click **Apply**.
- 8 Click **Close**.
- 9 Save the part.

When you add the part to an assembly, and subsequently create a BOM, the BOM data will automatically be displayed.

To save BOM data with an assembly:

- 1 Follow the procedure above for a part but also check the option **Treat as part in BOM**, which is found in the **General** tab of the **Design Properties** dialog.

Now, this assembly will be treated as a part whenever it is encountered in a BOM. It will appear as an item in the BOM. Also, the BOM properties assigned to it will be reported in the BOM. The parts contained in this assembly will **not** appear as separate items in the BOM.

12.2 Creating Bills of Material

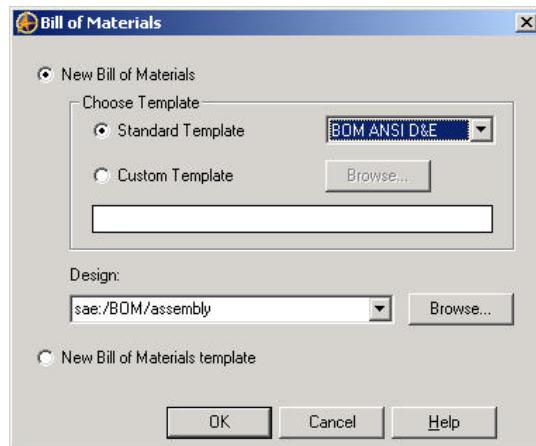
You can create a new Bill of Materials for an assembly, and a part if necessary. You must first create the design and save it before a BOM can be created.

12.2.1 Creating a New BOM

To create a new Bill of Materials:

- 1 In the Home window, Repository, or any workspace, from the **File** menu, select **New > Bill of Materials**; or, select the **Bill of Materials** icon from the main toolbar.

The Bill of Materials dialog box appears.



- 2 Select the **New Bill of Materials** option.
- 3 In the **Choose Template** area, select the **Standard Template** or **Custom Template** option.
- 4 If you are using a Standard Template, select the appropriate template size as well.

Note: Alibre Design includes the BOM ANSI ABC and BOM ANSI D&E bill of materials templates ready for use. The BOM ANSI ABC template is for use with drawing templates ANSI A Portrait, ANSI A Landscape, ANSI B, and ANSI C. The BOM ANSI D&E template is for use with drawing templates ANSI D and ANSI E.

- 5 If you are using a Custom Template, click **Browse**. The **Select Bill of Materials Template** dialog box appears. Select the appropriate template and click **OK**.
- 6 In the **Bill of Materials** dialog box **Design** area, click **Browse**. The **Choose Design Part or Assembly** dialog box appears.
- 7 Select the assembly or part for which you want a bill of materials.
- 8 Click **OK** in the Choose Design Part or Assembly dialog box.
- 9 Click **OK** in the Bill of Materials dialog box.
- 10 The BOM workspace appears containing the BOM data. When using a new BOM template, the first row will be blank by default. The default headers are **Item Number**, **Part Number**, **Quantity**, and **Part Name**.
- 11 Modify the BOM as required (refer to section 11.4 for detailed info related to working in a BOM workspace).

- 12 From the **File** menu, select **Save**; or select the **Save**  tool from the main toolbar. The **Save** dialog box appears.
- 13 In the Document Browser, select the location in which you want to save the BOM.
- 14 Enter the BOM **Name**.
- 15 Click **Save**. The BOM can now be opened independently or inserted into a drawing if necessary.

For information related to inserting a BOM into a drawing, refer to section 11.3.1.

12.2.2 Creating a Custom BOM Template

You can create a custom BOM template to meet your own design, purchasing, and production requirements and specifications.

To create a custom BOM template:

- 1 In the Home window, Repository, or any workspace, from the **File** menu, select **New > Bill of Materials**; or select the **New Bill of Materials**  icon from the main toolbar. The Bill of Materials dialog box appears.
- 2 Select the **New Bill of Materials** template radio button.

- 3 Click **OK**. A New Bill of Materials workspace appears. The workspace contains one empty row by default. The default column headers are **Item Number**, **Part Number**, **Quantity**, and **Part Name**.

	Item Number	Part Number	Quantity	Part Name
Header	Item Number	Part Number	Quantity	Part Name
1	1			

Note: If you choose to leave a blank row in the custom template, the blank row will be listed first any time you use the custom template. Delete the blank row if you do not want to include it in the custom template.

- 4 Modify the table as necessary to meet your requirements. For information about modifying BOM tables, refer to section 12.4.
- 5 Select the **Save** tool from the Standard toolbar; or from the **File** menu, select **Save**. The **Save** dialog box appears.
- 6 In the Document Browser, select the location in which you want to save the custom template.
- 7 Specify a **Name** for the custom template.
- 8 Select **Alibre** as the **Save as** type.
- 9 Click **Save**.

12.3 Working With a BOM in a Drawing

12.3.1 Inserting a BOM View Into a Drawing

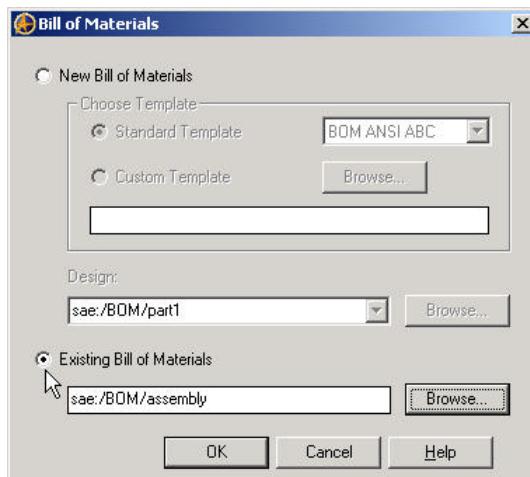
You can insert a new BOM view or existing BOM view into a drawing. You can only insert one BOM view per drawing. However, you can insert the BOM view into any drawing sheet. When you insert a BOM view into a drawing, you are automatically linking the BOM to the drawing. Linking a BOM to a drawing creates an association between the BOM data and the

drawing itself. You can link a BOM to a drawing without actually inserting the view into a sheet. For more information about linking a BOM, refer to section 12.3.2.

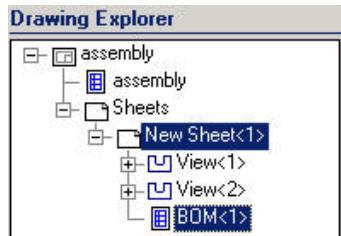
To insert an existing BOM view into a drawing:

- 1 Select the **Insert Bill of Materials**  tool from the Detailing toolbar; or from the **Insert** menu, select **Bill of Materials View**.

The **Bill of Materials** dialog box appears.



- 2 Click the **Existing Bill of Materials** radio button.
- 3 Click **Browse**. The **Choose Design Part or Assembly** dialog box appears. In the Document Browser, navigate to the location containing the BOM.
- 4 Select the BOM item and click **OK**.
- 5 Click **OK** in the Bill of Materials dialog box. A preview of the BOM view appears in the work area and is listed in the Drawing Explorer view list.



- 6 Move the cursor to position the BOM view on the sheet and click to place the view.

You can move a BOM view just like any other drawing view.

To insert a new BOM view into a drawing:

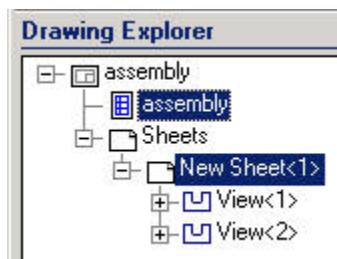
- 1 Select the **Insert Bill of Materials** tool from the Detailing toolbar; or from the **Insert** menu, select **Bill of Materials View**. The **Bill of Materials** dialog box appears.
- 2 Click the **New Bill of Materials** radio button.
- 3 Select a **Standard Template**.
Or
Select a **Custom Template**. Click **Browse** to select the custom template.
- 4 In the **Design** area, click **Browse**. The **Choose Design Part or Assembly** dialog box appears.
- 5 In the Document Browser, navigate to the location containing the design.
- 6 Select the design and click **OK**.
- 7 Click **OK** in the Bill of Materials dialog box. A preview of the BOM view appears in the work area and is listed in the Drawing Explorer in the sheet view list.
- 8 Move the cursor to position the BOM view on the sheet and click to place the view.

12.3.2 Linking a BOM to a Drawing

Linking a BOM to a drawing creates an association between the BOM data and the drawing itself. You can link a BOM to a drawing without actually inserting the BOM view into a sheet. This is useful if you want to display item callouts in a drawing but do not want to display the BOM data in the sheet. Before you can insert callout balloons, you must at minimum link a BOM to a drawing. Inserting a BOM view into a drawing automatically links the BOM to the drawing.

To link a BOM to a drawing:

- 1 From the **Tools** menu, select **Bill of Materials > Link**; or in the Drawing Explorer, right-click the drawing name and select **Link Bill of Materials** from the pop-up menu. The **Bill of Materials** dialog box appears.
- 2 To link a new BOM, select the **New Bill of Materials** option. Specify the **Standard Template** or **Custom Template** option, and select a size or template accordingly. Click **Browse** to select the applicable design.
- 3 To link an existing BOM, select the **Existing Bill of Materials** option.
- 4 Click **Browse**. The **Choose Design Part or Assembly** dialog box appears.
- 5 In the Document Browser, navigate to the location of the BOM.
- 6 Select the applicable BOM in the item list.
- 7 Click **OK**. The BOM name appears in the Bill of Materials dialog box.
- 8 Click **OK**. The BOM item appears in the Drawing Explorer under the drawing name.



You can insert callout balloons after the BOM has been linked to the drawing.

12.3.3 Unlinking a BOM from a Drawing

Unlinking a BOM from a drawing removes all association between the BOM and drawing. If the BOM view has been inserted into the sheet, unlinking a BOM will delete the BOM view from the sheet automatically.

To unlink a BOM from a drawing:

- 1** In the Drawing Explorer, right-click the BOM item beneath the drawing name and select **Unlink** from the pop-up menu; or from the **Tools** menu, select **Bill of Materials > Unlink**.

If a BOM view exists in the drawing, the **Unlinking Bill of Materials** dialog box appears.

- 2** In the **Unlinking Bill of Materials** dialog box, click **Yes**. If applicable, the BOM view is deleted from the drawing sheet, and the association between the BOM and the drawing is broken.

12.3.4 Editing a BOM

You can open the BOM workspace directly from the drawing and subsequently edit the BOM attributes.

To edit the BOM from the drawing:

- 1** Select the **Select**  tool from the View toolbar.
- 2** Move the cursor over the BOM view in the work area and double-click; or in the Drawing Explorer, right-click the BOM item and select **Edit Bill of Materials** from the pop-up menu; or in the Drawing Explorer, double-click the BOM item.

The BOM workspace appears.

- 3** In the BOM workspace, edit the BOM as necessary (refer to section 11.4 for information related to working in a BOM workspace). The BOM view in the drawing will update automatically.
- 4** Close the BOM workspace when finished. You do not need to save the changes before you close the BOM workspace. Any changes made to the BOM will be saved when you save the drawing.

12.3.5 Moving the BOM View on the Sheet

You can move a BOM view after it has been inserted into a sheet.

To move a BOM view on a sheet:

- 1 From the **Tools** menu, select **Selection Filters > Views**, if it is not already selected.
- 2 Select the **Select**  tool from the View toolbar.
- 3 Move the cursor over the view in the work area. The view is highlighted and the cursor changes.



Item Number	Quantity	Part Number	Part Number
1	1	a2	ba_r2
2	1	a4	ba_r4
3	1	a6	ba_r6
4	1	a5	ba_r5
5	1	a13	ba_r13
6	1	a12	ba_r12
7	1	a10	ba_r10
8	1	a1	ba_r1
9	1	a7	ba_r7
10	1	a8	ba_r8
11	3	a9	ba_r9
12	1	a11	ba_r11
13	1	a3	ba_r3

- 4 Click and drag the view to the desired location on the sheet.
- 5 Release the mouse button to place the view.

12.3.6 Hiding the BOM View

You can hide the BOM view in a sheet.

To hide a BOM view:

- 1 In the Drawing Explorer, right-click the BOM item listed under the sheet and select **Hide** from the pop-up menu.

Or

- 1 Select the **Select**  tool from the View toolbar.
- 2 Move the cursor over the BOM view in the work area, right-click, and select **Hide** from the pop-up menu.

The view is hidden in the work area and the associated text is dimmed in the Drawing Explorer.



To show the view, right-click the dimmed BOM item in the Drawing Explorer and unselect **Hide** from the pop-up menu.

12.3.7 Deleting the BOM View

You can delete a BOM view from a sheet at anytime.

To delete a BOM view:

- 1 In the Drawing Explorer, right-click the BOM item listed under the sheet and select **Delete** from the pop-up menu.

Or

- 1 Select the **Select**  tool from the View toolbar.
- 2 Move the cursor over the table in the work area, right-click, and select **Delete** from the pop-up menu.

The table is deleted from the work area and Drawing Explorer.

Note: The BOM is still associated with a drawing after you delete a table from a sheet. You must unlink the BOM from the drawing to remove all association between the BOM and drawing. Refer to section 11.3.3 for information related to unlinking a BOM from a drawing.

12.3.8 Moving a BOM View to Another Sheet

You can move the BOM view from one sheet to another sheet in the drawing if necessary.

To move the BOM view from one sheet to another:

- 1 Select the **Select**  tool from the View toolbar.
 - 2 Right-click the BOM view in the work area or the Drawing Explorer and select **Move** from the pop-up menu. The **Move** item is enabled only when the drawing contains at least two sheets.
- The **Select Target Sheet** dialog box appears.
- 3 From the **Target Sheet** list, select the sheet you want to move the BOM view to.
 - 4 Click **OK**. The BOM view is listed under the target sheet in the Drawing Explorer and appears in the target sheet work area.

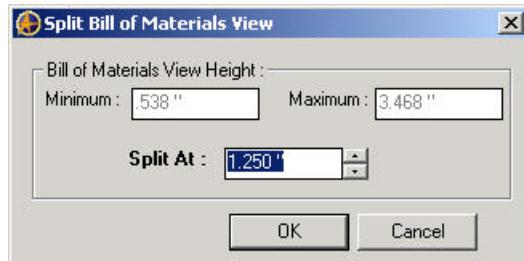
12.3.9 Splitting a BOM View

You can split a BOM view into multiple smaller views if necessary. This is useful if a view is too long to fit onto a sheet.

To split a view:

- 1 From the **Tools** menu, select **Bill of Materials > Split View**; or right-click the BOM view in the work area or Drawing Explorer and select **Split View** from the pop-up menu.

The Split Bill of Materials View dialog box appears.



The **Minimum** and **Maximum** values are specific to the table being split. The **Minimum** value represents the combined width of the header row and the widest row in the table. The **Maximum** value represents the combined width of the header row and all the rows in the table except for the last row.

- 2 In the **Split At** field, specify a split value. This value must fall between the **Minimum** and **Maximum** view height values.
- 3 Click **OK**. The view is split into multiple views.

You can move the views independently on the sheet. You can also move individual views onto a different sheet if necessary. However, if you delete or hide one view, the rest of the views will be deleted or hidden as well. If you add a row to the BOM after it has been split, the row will be added to the last BOM view.

To restore the view back to its original configuration, enter a **Split At** value outside the **Minimum – Maximum** range.

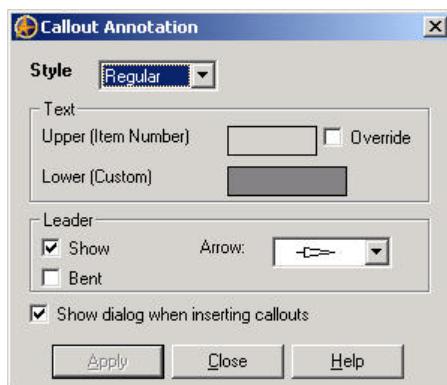
12.3.10 Adding Callout Balloons

After you have linked a BOM with a drawing or inserted a BOM view into a drawing, you can add callout balloons to drawing views.

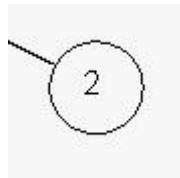
To add callout balloons:

- 1 Select the **Callout**  tool from the Detailing toolbar; or from the **Insert** menu, select **Annotation > Callout**.

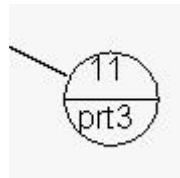
The **Callout Annotation** dialog box appears.



- 2 Select **Regular** or **Split** from the **Style** pulldown menu. The **Regular** style will by default display only the item number in the callout balloon. The **Split** style will divide the callout balloon into halves. The upper half displays the item number by default, and the lower half displays custom information.

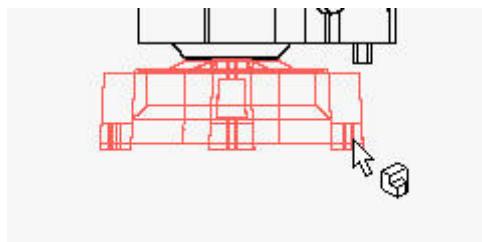


Regular Callout

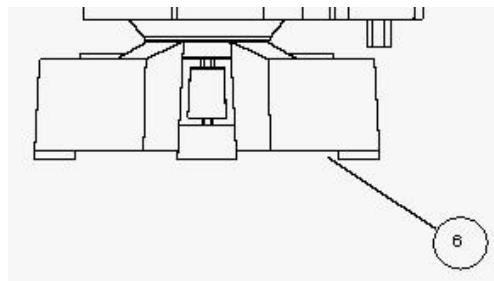


Split Callout

- 3 If necessary, select the **Override** option to manually enter an item number.
- 4 If the **Split** type was selected, enter custom text in **Lower (Custom)** text box area.
- 5 In the **Leader** area, select:
 - **Show** if you want to display a leader with the callout balloon.
 - **Bent** if you want to display a bent leader with the callout balloon.
 - An **Arrow** type from the pull down menu.
- 6 Move the cursor over a part in the drawing view. The part is highlighted.



- 7 Click once to create the callout balloon. The callout balloon appears.
- 8 Drag the balloon to the appropriate position and click to place.
- 9 Click **Apply** in the **Callout Annotation** dialog box. The balloon is placed.



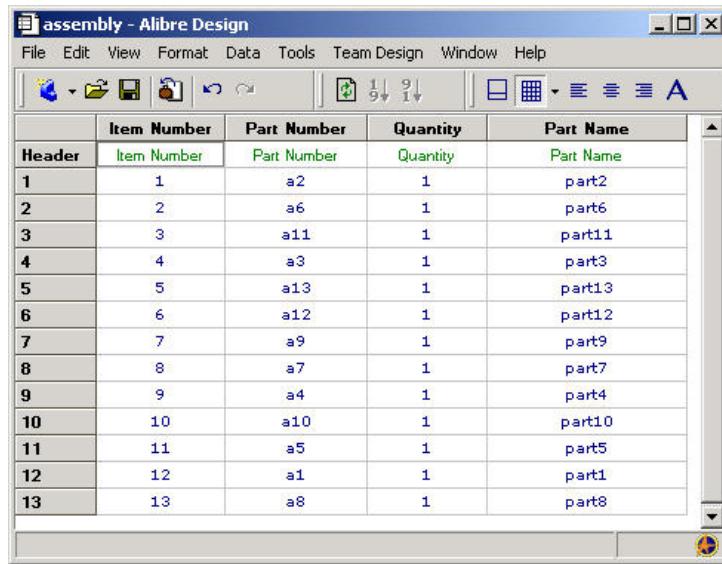
10 Continue to select parts and click **Apply** to add additional balloons.

11 Click **Close** when finished.

Note: The Callout Annotation dialog box appears by default when you insert a callout. If desired, you can specify the callout settings you want to use in the dialog box, and then unselect the **Show dialog when inserting callouts** option. You will then be able to insert callouts quickly without using the dialog box. After selecting the **Callout** tool, simply click a part in a view to create the callout balloon and click to place it. To turn the dialog box option back on, from the **Tools** menu, select **Options**. In the **General** tab, select the **Show dialog when inserting callouts** option.

12.4 Working in a BOM Workspace

All work related to creating or editing a BOM is performed in a BOM workspace. The BOM workspace displays the bill of material data in tabular format similar to a spreadsheet.



The screenshot shows the Alibre Design software window titled "assembly - Alibre Design". The menu bar includes File, Edit, View, Format, Data, Tools, Team Design, Window, and Help. The toolbar contains icons for opening files, saving, printing, and other functions. A status bar at the bottom shows "1 9 1". The main area displays a Bill of Materials table with the following data:

Header	Item Number	Part Number	Quantity	Part Name
1	1	a2	1	part2
2	2	a6	1	part6
3	3	a11	1	part11
4	4	a3	1	part3
5	5	a13	1	part13
6	6	a12	1	part12
7	7	a9	1	part9
8	8	a7	1	part7
9	9	a4	1	part4
10	10	a10	1	part10
11	11	a5	1	part5
12	12	a1	1	part1
13	13	a8	1	part8

In a BOM workspace you can:

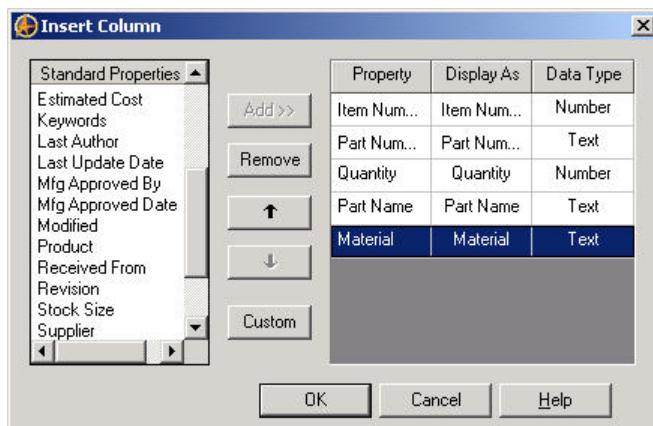
- Add/delete rows & columns
- Resize rows & columns
- Hide rows
- Change data and header font properties
- Automatically re-sequence data
- Override design values
- Set column header and data alignment
- Print BOM data
- Export BOM data to a .CSV file
- Append (add) rows
- Organize data by dragging and dropping columns and rows
- Change the table display orientation
- Sort data in ascending or descending order
- Control how a BOM will be displayed in a drawing

12.4.1 Adding and Deleting Columns in a BOM

You can add standard or custom columns to a BOM as well as delete columns as needed.

To add a column to a BOM:

- 1 From the **Edit** menu, select **Insert Column**; or right-click in the table area and select **Column > Insert** from the pop up menu. The **Insert Column** dialog box appears.

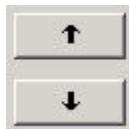


Note: In the Insert Column dialog box, you can edit the fields under the **Display As** column. You cannot edit the **Property** or **Data Type** fields when using standard headers.

- 2 In the **Standard Properties** area select a column header from the twenty-seven standard column headers listed.
- 3 Click the **Add** button to move the standard header to the column list.
- 4 To create a column with a custom header, click the **Custom** button. The custom column is automatically added to the column list.

Property	Display As	Data Type
Item Num...	Item Num...	Number
Part Num...	Part Num...	Text
Quantity	Quantity	Number
Part Name	Part Name	Text
Material	Material	Text
Column1	Custom	Text ▾

- 5 Edit the **Display As** field for the custom header as necessary.
- 6 In the **Data Type** column, select the column data format: **Text**, **Date**, or **Number**.
- 7 Columns will be displayed in the BOM in the order in which they are listed in the Insert Column dialog box. To move a column up or down in the list, select a row, and then click either the up arrow or down arrow.



- 8 Click OK.

To delete a column from a BOM:

- 1 Click the table header of the column you want to delete. The entire column is highlighted.

Item Number
Item Number
2
3
4
5
6
7
8
9
10
11
12
13
14

- 2 Right-click in the table area and select Column > Delete from the pop up menu; or from the **Edit** menu, select **Delete**.

12.4.2 Adding and Deleting a Row in a BOM

You can add rows in a BOM as needed. You can also delete a row at any time that has been manually inserted. However, in order to delete a row that was generated automatically from a design (i.e., a part in the assembly), you must first delete the associated part in the assembly. The Quantity value must then be updated to zero before the row can be deleted.

To add a row:

From the **Edit** menu, select **Append Row**; or right-click in the table area and select Row > Append from the pop up menu. A row is added to the end of the table.

To delete a row:

- 1 Click the table row number you want to delete. The entire row is highlighted.
- 2 Right-click in the table area and select **Row > Delete** from the pop up menu; or from the **Edit** menu, select **Delete**.

12.4.3 Hiding a Row

You can hide rows in a BOM. Hidden rows are not displayed in the BOM view in the drawing.

To hide a row:

- 1 Click the table row number you want to hide. The entire row is highlighted.
- 2 Right-click in the table area and select **Row > Hide** from the pop up menu; or from the **Format** menu, select **Row > Hide**. The row becomes hidden.

A distinct line is displayed between the rows that border above and below the row that has been hidden. In the illustration below, row 11 has been hidden.

9		10	a4	1	part4
10		11	a10	1	part10
12		13	a1	1	part1
13		14	a8	1	part8

To display hidden rows:

- 1 From the **View** menu, select **Hidden Rows**. The hidden row is displayed and the corresponding table row number field is orange.

10		11	a10	1	part10
11		12	a5	1	part5
12		13	a1	1	part1
13		14	a8	1	part8

To unhide a row:

- 1 If hidden rows are displayed, select the table row number of the hidden row. The entire row is highlighted.

Or

If hidden rows are not displayed, select table row number above the hidden row, and then hold the **Shift** key and select the table row number below the hidden row. Both selected rows become highlighted.

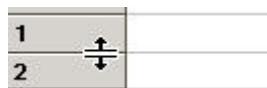
- 2 Right-click in the table area and select **Row > Unhide** from the pop up menu; or from the **Format** menu, select **Row > Unhide**.

12.4.4 Resizing Rows and Columns

You can resize rows and columns to customize the look of the table.

To resize a row or column by dragging:

- 1 In the row number or column header area, move the cursor near the edge of the row or column you want to resize. The cursor changes appearance.



- 2 Click and drag the row or column border to change the respective width or height.
- 3 Release the mouse button when finished resizing.

To resize a row by specifying a row height value

- 1 Click the row number you want to resize. The entire row is highlighted.
- 2 Right-click in the table area and select **Row > Height** from the pop up menu; or from the **Format** menu, select **Row > Height**. The **Row Height** dialog box appears.



- 3 Specify the Row height value.
- 4 Select the Apply to all rows options if desired.

-
- 5 Click OK.

To automatically adjust a column's width:

- 1 In the main column header area, move the cursor over the right edge of the column you want to automatically resize.
- 2 Double-click the right edge. The column width automatically adjusts to the widest field in the column.

Or

- 1 Click the table Header of the column you want to automatically resize. The entire column is highlighted.
- 2 Right-click in the table area and select **Column > AutoFit** from the pop up menu; or from the **Format** menu, select **AutoFit Column**. The column width automatically adjusts to the widest field in the column.

12.4.5 Adjusting Column Header and Data Alignment

You can adjust the alignment of column headers and column data independently or together.

To adjust column header alignment:

- 1 Select any field in the column in which you want to change the header alignment.
- 2 Right-click in the table area and select **Header Alignment > Left** or **Center** or **Right**.

Or from the **Format** menu, select **Header Alignment > Left** or **Center** or **Right**.

Or select the **Align Left** , **Center** , or **Align Right**  tool from the View toolbar.

The column header alignment changes.

To adjust column data alignment:

- 1 Select any field in the column in which you want to change the data alignment.
- 2 Right-click in the table area and select **Data Alignment > Left** or **Center** or **Right**.

Or from the **Format** menu, select **Data Alignment > Left** or **Center** or **Right**.

Or select the **Align Left** , **Center** , or **Align Right**  tool from the View toolbar.

The column data alignment changes.

To adjust column header and data alignment together:

- 1 Click the table header of the column you want align. The entire column is highlighted.
- 2 Right-click in the table area and select **Data Alignment > Left** or **Center** or **Right**.

Or from the **Format** menu, select **Data Alignment > Left** or **Center** or **Right**.

Or select the **Align Left** , **Center** , or **Align Right**  tool from the View toolbar.

The column header and data alignment changes.

12.4.6 Moving Rows and Columns in a Table

You can drag and drop rows and columns to reposition data within the table.

To move a row or column:

- 1 Move the cursor over a table column header or row number.



- 2 Click the column header cell or row number cell and drag to the new table position.
- 3 Release the mouse button to complete the move.

12.4.7 Sorting Data in Ascending or Descending Order

You can sort column data in ascending or descending order.

To sort data in ascending or descending order:

- 1 Click the table header of the column you want to sort. The entire column is highlighted.
- 2 To sort the column data in ascending order, select the **Sort Ascending**  tool from the main toolbar; or from the **Data** menu, select **Sort > Ascending**.

Or

To sort the column data in descending order, select the **Sort Descending**  tool from the main toolbar; or from the **Data** menu, select **Sort > Descending**.

The data is sorted accordingly.

12.4.8 Changing the Header Display Orientation

You can change the table display so that the header is located at the bottom of the table and the row numbers increase going up the table. By default, the header is located at the top of the BOM table and the row numbers increase going down the table.

To change the header display orientation:

- 1 Select the **Bottom Up Display**  tool from the View toolbar; or From the **View** menu, select **Bottom Up Display**.

The column headers are positioned at the bottom of the table and the row numbers increase going up the table.

- 2 To return to the default orientation, repeat step 1.

12.4.9 Customizing Header and Data Font Properties

You can change the font properties associated with column headers and tabular data. You cannot change font properties for individual items in the table.

To customize header font properties:

- 1 Right-click in the table area and select **Header Font** from the pop-up menu; or from the **Format** menu, select **Header Font**. The **Font** dialog box appears.
- 2 Modify the **Font**, **Font Style**, **Size**, **Effects**, **Color**, and **Script** as desired.
- 3 Click **OK**.

To customize data font properties:

- 1 Right-click in the table area and select **Header Font** from the pop-up menu; or from the **Format** menu, select **Header Font**. The **Font** dialog box appears.
- 2 Modify the **Font**, **Font Style**, **Size**, **Effects**, **Color**, and **Script** as desired.
- 3 Click **OK**.

12.4.10 Overriding Design Values

When you create a BOM, the table contains information based on the design, e.g. part number, part name, quantity, etc. These items are referred to as **design values** since they are dictated by the design. You can manually override design values in a BOM workspace. Overriding a design value in the BOM workspace, will not have an effect on the actual design.

To override a design value:

- 1 Select the field containing the design value that you want to override.
- 2 Change the value as necessary and press **Enter** on the keyboard. The cell containing the overridden value becomes blue.

10	11		a10	1	part10
11	12		a5	5	part5
12	13		a1	1	part1

To restore a value to its design value:

- 1 Select the cell containing the overridden value.
- 2 Right-click the cell and select Use Design Value from the pop-up menu; or from the **Edit** menu, select **Use Design Value**.

The cell's value is restored to the value dictated by the design.

12.4.11 Modifying the BOM View Style

You can control how the BOM view is displayed in the drawing. You can choose to show or hide row and column lines, only column lines, only row lines, or no lines. The table style setting only applies to the BOM view in the drawing. Table lines are always visible in the BOM workspace regardless of which table style setting is used. By default, row and column lines are visible.

To modify the table style:

- 1 Click the **Options** arrow on the View toolbar to display the Table Style drop down toolbar.



- 2 From the toolbar, select:
 - **No Lines:** the BOM table will be displayed without lines.
 - **Row Lines:** the BOM table will be displayed with row lines only.
 - **Column Lines:** the BOM table will be displayed with column lines only.
 - **Row, Column Lines:** the BOM table will be displayed with column and row lines.

Or

From the **Format** menu, select **Table Style > No Lines** or **Row Lines** or **Column Lines** or **Row, Column Lines**.

12.4.12 Resequencing Data

You can resequence (reorder) data in a table after you delete or move rows. Resequencing a BOM will reset the **Item Numbers** so that they are sequentially numbered correctly in the order listed.

To resequence a BOM:

- 1 From the **Data** menu, select **Resequence**. The **Item Numbers** are reordered.

	Item Number	Quantity	Part Number	Part Name
Header	Item Number	Quantity	Part Number	Part Number
1	1	1	a2	part2
2	7	1	a4	part4
3	2	1	a6	part6
4	9	1	a5	part5
5	4	1	a13	part13
6	5	1	a12	part12
7	8	1	a10	part10
8	10	1	a1	part1
9	6	1	a7	part7
10	11	1	a8	part8
11	12	1	a9	part9
12	13	1	a11	part11
13	3	1	a3	part3

Before Resequencing

	Item Number	Quantity	Part Number	Part Name
Header	Item Number	Quantity	Part Number	Part Number
1	1	1	a2	part2
2	2	1	a4	part4
3	3	1	a6	part6
4	4	1	a5	part5
5	5	1	a13	part13
6	6	1	a12	part12
7	7	1	a10	part10
8	8	1	a1	part1
9	9	1	a7	part7
10	10	1	a8	part8
11	11	1	a9	part9
12	12	1	a11	part11
13	13	1	a3	part3

After Resequencing

12.4.13 Updating the Table

You can have the assembly workspace open and the BOM workspace open simultaneously. Consequently, you can update a BOM table after making changes to an assembly. Before you can update the BOM, you first must save any changes made to the design. The design does not need to be open in order to update the table. You cannot update the BOM when it is being edited within the context of the drawing.

To update the table:

- 1 From the **Data** menu, select **Update Table**.

Or

Select the **Update Table**  tool from the Edit toolbar.

The table is updated to reflect any recent changes made in the design.

12.4.14 Exporting a BOM

You can export a BOM table as a .csv file. You can open .csv files in any spreadsheet application or text editor.

To export a BOM:

- 1 Select the **Export File**  tool from the Standard toolbar; or from the **File** menu, select **Export**. The **Export File** dialog box appears.
- 2 Select the **Save in** location.
- 3 Specify a **File name**.
- 4 Select a **Save as type**.
- 4 Click **Save**.

12.4.15 Printing a BOM

You can print a BOM table by itself directly from the BOM workspace.

To print a BOM table:

- 1 From the **File** menu, select **Print**. The **Print** dialog box appears.
- 2 Select the appropriate printer.
- 3 Specify the print layout.
- 4 Click **Print**

13 Importing and Exporting Data

Alibre Design's import and export functionality enables interaction with data from other CAD systems. Furthermore, machine tools and rapid prototyping can be driven from exported data created in Alibre Design. The STEP format is an ISO standard that is driven by industry. Alibre Design's native format is STEP AP 203 and AP 214, two protocols in the STEP standard. Alibre's STEP schema allows for the creation of parametric features along with geometry. This powerful capability and platform choice allows the data created in Alibre Design to be available for use beyond Alibre Design itself.

This chapter describes:

- Importing data from other CAD systems
- Interoperability settings
- Exporting data from Alibre Design

13.1 Importing Data

13.1.1 Supported File Types

A variety of file formats are supported for interoperating with data from other CAD systems. The import data types and their attributes are:

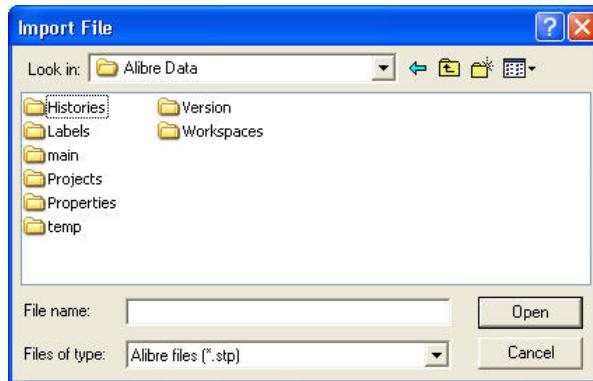
- **STEP AP 203/214** (*.stp, *.step, *.ste)
 - STEP (Standard for the Exchange of Product model data). An ASCII format set and driven by industry that all major CAD tools have adopted.
- **Alibre STEP** (*.stp)
 - Feature-based STEP data inherent from Alibre's STEP schema
- **SAT** (*.sat)
 - ACIS file format
- **IGES** (*.igs)
 - Initial Graphics Exchange Specification. An ANSI-standard format.
- **DWG** (*.dwg)
 - Standard file format for saving vector graphics from within AutoCAD.
- **DXF** (*.dxf)
 - For drawing interchange format. An ASCII or binary file format of an AutoCAD drawing file for exporting AutoCAD drawings to other applications or for importing drawings from other applications.
- **3DM** (*.3dm)
 - Rhino Program file format

13.1.2 Importing a File

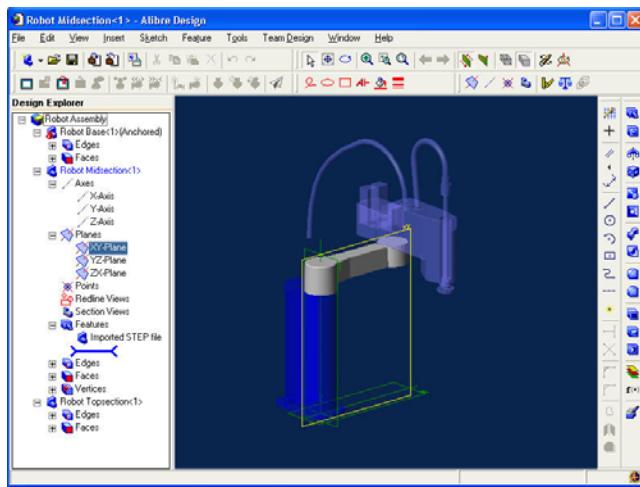
You can import 3D parts and assemblies in **STEP**, **IGES**, or **SAT** formats. You can import 2D drawings in the **DXF** or **DWG** formats.

To import a file:

- 1 Select the **Import**  tool from the Standard toolbar; or from the **File** menu select **Import**. The **Import File** dialog box appears.



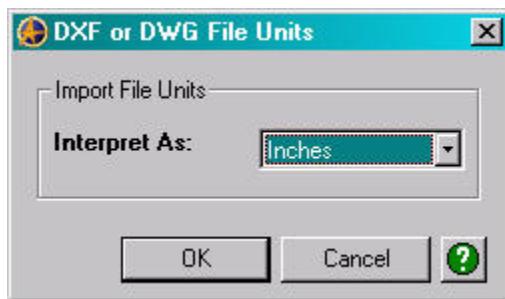
- 2 Browse to the location on the disc where the file is located and select the file.
- 3 Click **Open**. The **File Import Options** dialog box appears if you are importing STEP (non-native Alibre), IGES, and SAT files (see section 12.2 for more information about import options).
- 5 Select the applicable import options.
- 6 Click **OK**. The data appears in the workspace:



Note that the part is displayed in the Design Explorer with the label indicating the file type, e.g. **Imported STEP file** or **Imported SAT file**. Any feature created on the imported model will appear after this entry in the feature history tree.

To import a DXF or DWG file:

- 1 Select the **Import** tool from the Standard toolbar; or from the **File** menu select **Import**. The **Import File** dialog box appears.
- 2 Browse to the location on the disc where the file is located and select the file.
- 3 Click **Open**. The **DXF or DWG File Units** dialog box appears.



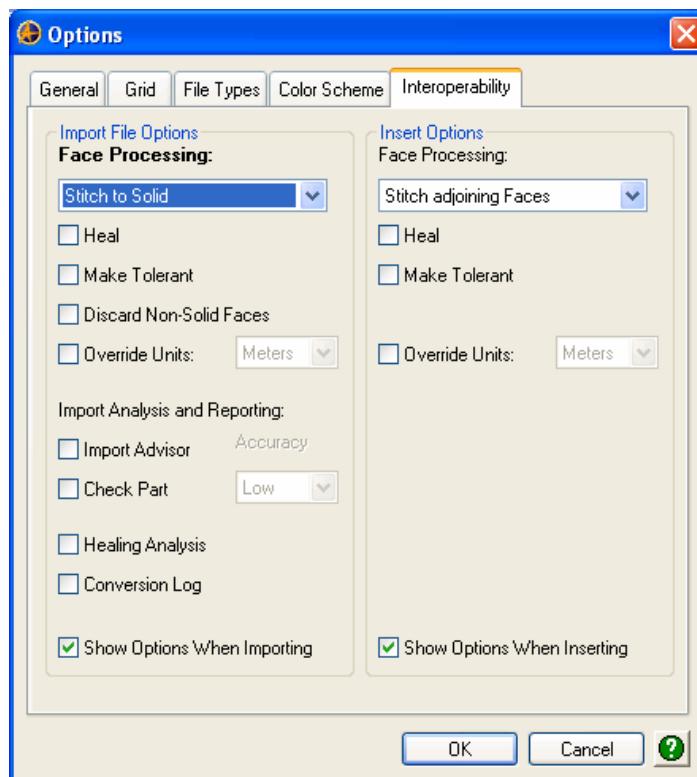
- 4 Select the units for the imported file.
- 5 Click **OK**. The file opens in a drawing workspace.

13.2 Import Settings and Import Advisor

Alibre Design's import settings provide various options to apply when importing data. As mentioned in section 13.1, the **Import Options** dialog box will appear by default as you import data. You can also set default import options from any workspace.

To set default import options from a workspace:

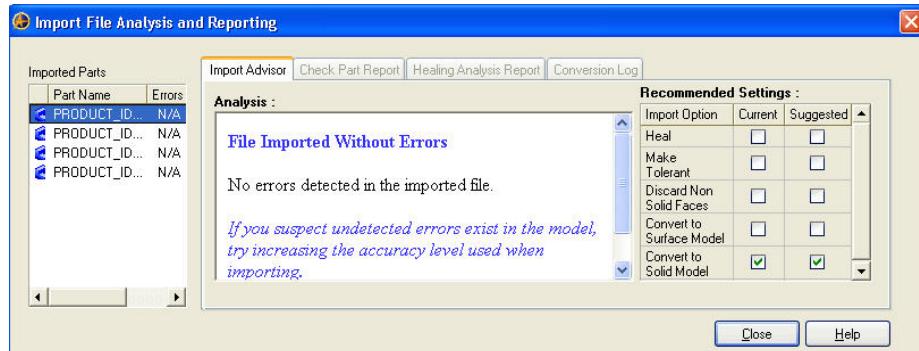
- 1 From the **Tools** menu select **Options**. The **Design Options** dialog box appears.
- 2 Select the **Interoperability** tab.



The Interoperability tab contains different selections that can be applied for imported data:

The selections for import are as follows:

- **Heal.** Recalculates inaccurate geometry in order to make the part more accurate upon import.
 - **Make Tolerant.** Tags inaccurate geometry for more intelligent subsequent operations.
 - **Discard Non-Solid Faces.** Discards faces that are not part of a solid. These faces may have been created for reference.
 - **Unstitch To Standalone Faces.** Converts a solid part to a set of faces. This can improve the visual representation of the part. This option is not recommended if changes will be made to the solid.
 - **Stitch To Solid.** Converts a surface model to a solid part.
 - **Override Units.** Converts the units to those specified.
 - **Note:** All import options can operate simultaneously, with the exception of **Convert To Surface Model** and **Convert To Solid Model**.
- 3 Select **Import Advisor** to generate an import summary after the file has been imported. With this option selected, a dialog appears after import displaying errors, suggestions and options that can be changed for the specific import issue.



- 4 Select the **Check Part** option to obtain information about the integrity of the file.
- 5 If **Import Advisor** or **Check Part** is checked, select an **Accuracy** level for the report:
- **Low:** Fast error checks.
 - **Medium:** Slower error checks plus D-cubed curve and surface checks.
 - **High:** Slower warning and error checks plus D-cubed and surface checks.

- **Very High:** Warning and error checks plus edge convexity change point and face/face intersection checks.
- 6 Select Healing Analysis if Heal is checked under Import Options. This produces a report on any data corrected as a result of the Heal command.
 - 7 Select **Conversion Log** to view a report about import.
 - 8 Check **Do Not Show Options When Importing** to bypass this dialog in the future. Options set in the Design Options dialog will be used automatically.
 - 9 Check **Set As Default Import Options** to use these settings for future imports. Settings in the Design Options dialog box will be updated.

If any Analysis and Reporting options are selected, the **Import File Analysis and Reporting** dialog box appears.

Note: To save report data, highlight the text and press **Ctrl-C** to copy. The text can be pasted into another application for viewing or printing.

The tools that are available through the import options will provide a high degree of success in working with data from other CAD systems. For issues that are elusive, the Alibre Assistant should be contacted.

13.3 Exporting Data

You can export Alibre Design native data using a number of formats.

13.3.1 Supported File Types

The table below summarizes the file types you can use to export data from Alibre Design.

Drawing	Alibre Design file (*.stp) AutoCAD DWG file (*.dwg) AutoCAD DXF file (*.dxf) JPEG Image file (*.jpg) Bitmap file (*.bmp) Enhanced Metafile file (*.emf)
Assembly	Alibre Design file (*.stp) AP 203 file (*.stp) AP 214 file (*.stp) ACIS 3.0-R10 files (*.sat)

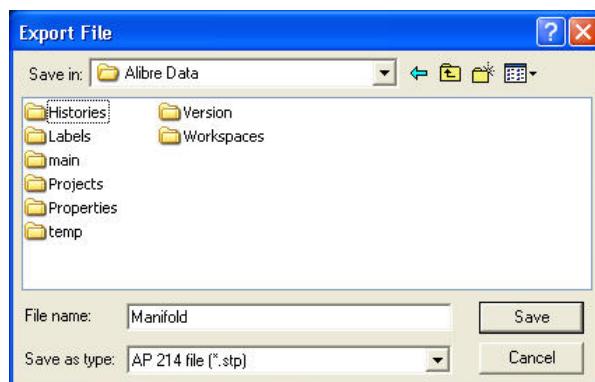
STL file (*.stl)
JPEG Image file (*.jpg)
Bitmap file (*.bmp)

Part Alibre Design file (*.stp)
AP 203 file (*.stp)
AP 214 file (*.stp)
ACIS 3.0-R10 files (*.sat)
IGES file (*.igs)
STL file (*.stl)
JPEG Image file (*.jpg)
Bitmap file (*.bmp)

13.3.2 Exporting a File

To export a file:

- 1 Select the **Export**  tool from the Standard toolbar; or from the **File** menu select **Export**. The **Export File** dialog box appears.

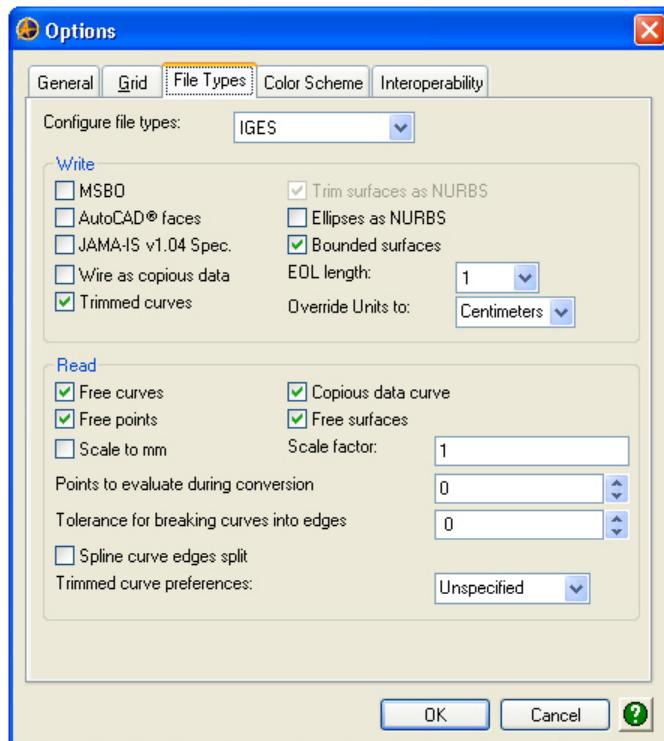


- 2 Browse to the location you want to export the file to.
- 3 Specify the **File name**.
- 4 From the **Save as type** menu, select the appropriate file format to use.

- 5 Click **Save**.

13.4 Special Options for IGES and STL Files

Alibre Design has special option settings for **exporting IGES** and **STL** files and **importing IGES** files. These special settings are found in the **File Types** tab of the **Options** dialog. You can access the Options dialog by selecting **Options** from the **Tools** main menu in any 3D workspace.



14 The Repository

The Repository is personal data vault and versioning system that lets you store and control access to project-related data. The Repository is available in certain versions of Alibre Design.

This chapter describes:

- Local repositories
- Repository items
- Setting permission and notification policies for repository items
- Sharing repositories
- Managing repositories
- Depositing and withdrawing items
- Opening items
- Version history
- Check-in/check-out
- Copying items
- Creating repository folders
- Caching
- Running Alibre Design as a service

14.1 Repository Overview

By default, your first repository is initialized on your local hard drive during installation. You may create additional local repositories as needed. Current subscribers may also purchase a repository on the Alibre Design server for centralized storage of team data.

Anything stored in a repository, including parts, symbols, subassemblies, assemblies, drawings or other files, are referred to as items.

Repositories are used to:

- Store, retrieve and share designs and drawings created with Alibre Design.
- Store, retrieve and share other project-related files.
- Access and manage items in a familiar directory structure.
- Track changes to items through built-in version history.
- **Copy** and **Paste** or **Move** items between repositories and folders.
- Manage access privileges for other Alibre Design users.

You may give other teams and users specific access rights to a repository and its contents. You can also decide who gets an automatic notification when items in the repository are changed. To activate access privileges, you must also share the repository with these teams and users. A shared local repository is only available to others when you are signed into Alibre Design on that computer. However, once shared, a server repository is available to other users even when you are not signed in.

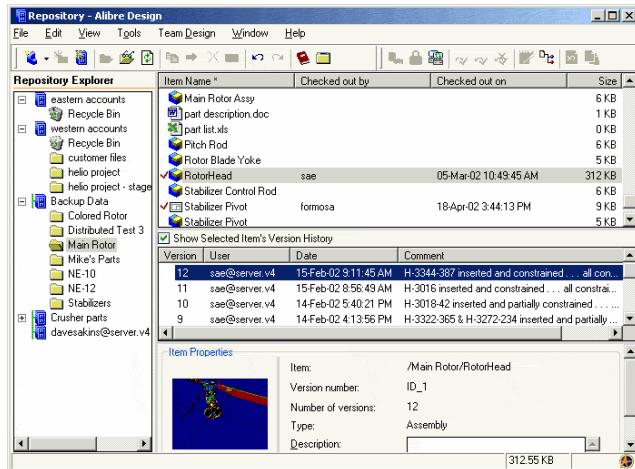
14.1.1 Opening the Repository

To open the Repository from the Home window:

- 1 Select the **Repository**  tool from the Standard toolbar; or from the **Window** menu, select **Repository**. The **Repository** window appears.

To open the Repository from any other area:

- 1 From the **Window** menu, select **Repository**. The **Repository** window appears.



14.1.2 Repository Explorer

The Repository Explorer, on the left side of the window, lists all repositories that you have access to. This includes repositories that have been shared with you. Local repositories are displayed in the Explorer with the local repository icon. Server repositories or repositories that others have shared with you are displayed in the Explorer with the remote repository icon. With appropriate access rights, you can create, rename, copy, delete and label folders within these repositories.

14.1.3 Item List

When a repository or folder is selected, the contents are listed in the **Item List** to the right of the Repository Explorer. When you select a folder, the number of items in the folder and disk space used is displayed along the bottom of the Repository window.

The item list displays basic information including item name and size, as well as checked in/checked out status info. When selected in the item list, additional information about an item is displayed below, including the version history, a thumbnail preview image, and item properties (e.g. name, location, number and description).

14.1.4 Menu Bar

Depending on access policies, you can use the menu bar to:

- Check items in or out. The check in/out process ensures that only one person at a time can change the data.
- Create or open parts, assemblies and drawings.

- Create folders to help you organize data.
- Copy, delete and rename repository folders and items. You can also undo and redo these actions.
- Open, change, manage and store files created outside Alibre Design.
- Empty the repository recycle bin or restore items from it.
- Refresh the repository and item lists.
- Set up permission and notification policies for repositories.
- Assign a label to the latest version of the selected item or all items in the selected folder or repository.
- Rollback to a selected version in an item's version history.
- Join a Team Design session in progress.

14.2 Local Repositories

By default, upon installation, every Alibre Design user is set up with one repository on the local hard drive. Additional local repositories may be created as needed. Local repositories exist on the computer where they were created.

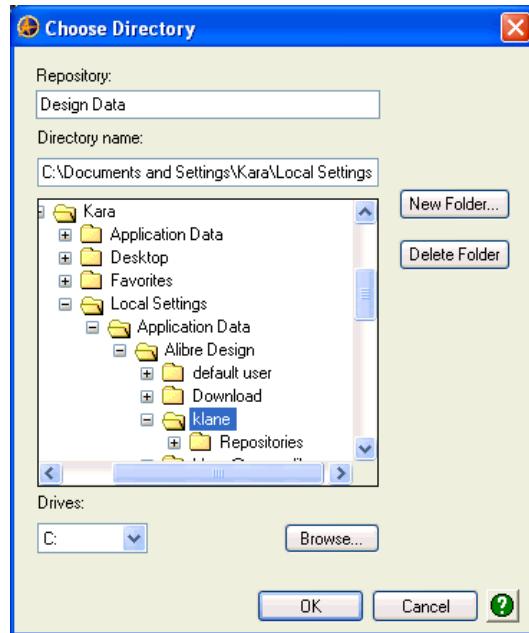
You may give other users specific access rights to content in your repositories. Your local repository is only available to others when you are signed into Alibre Design on the computer where the repository resides.

You can move and delete local repositories as well.

14.2.1 Creating a Local Repository

To create a local repository:

- 1 Select the **New Repository**  tool from the Standard toolbar; or from the **File** menu select **New Repository**; or right-click in the Repository Explorer and choose **Create Repository**. The **Choose Directory** dialog box appears.



- 2 Enter a name for the new repository in the **Repository** field.
- 3 From the **Drives** drop down menu, select the drive you want to store the repository on. You can select a network drive if necessary. Use the **Browse** button to search through non-mapped network drives.
- 4 Browse to select a location for the repository on the selected hard drive. The selected location will be shown in the **Location** field.
- 5 Click **New Folder** if necessary.
- 6 When finished, click **OK**. The new repository will appear in the Repository Explorer.

14.2.2 Moving a Local Repository

You may change the storage location of a local repository. This may be necessary if disk space becomes a limiting issue.

To move a local repository:

- 1 In the Repository Explorer, select the repository you want to move.
- 2 Select the **Move**  tool from the Standard toolbar; or right-click in the Explorer area and select **Move** from the pop-up menu; or from the **Edit** menu select **Move**. The **Choose Director** dialog box appears.
- 3 Browse to select a new location. You may also create a new directory using the **New Folder** button.
- 4 When finished, click **OK**. The repository is now stored in the indicated directory.

14.2.3 Deleting a Local Repository

You may delete all local repositories, but you cannot delete repositories that have been shared to you. The contents of a deleted repository cannot be recovered.

To delete a local repository:

- 1 In the Repository Explorer, select the repository you want to delete.
- 2 Select the **Delete**  tool from the Standard toolbar; or right-click in the Explorer area and select **Delete** from the pop-up menu; or from the **Edit** menu select **Delete**. The **Confirm Repository Delete** dialog box appears.
- 3 Click **Yes**.
- 4 A second **Confirm Repository Delete** dialog box appears. If absolutely certain, type **OK** in the text box.
- 5 Click **OK**. The repository and all its contents are permanently deleted. They cannot be recovered.

14.2.4 Renaming a Repository

You may rename a local repository.

To rename a repository:

- 1 In the Repository Explorer, right-click the repository and select **Rename** from the pop-up menu. The **Rename Repository** dialog box appears.

- 2 Enter a new repository name.
- 3 When finished, click **OK**. The repository is renamed.

14.3 About Repository Items

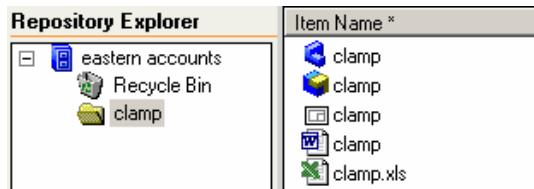
Anything stored in a repository, including parts, symbols, subassemblies, assemblies, drawings, bills of material, or other files, is referred to as an item. An item may contain notes, version history and comments.

14.3.1 Item Types

Item types are distinguished by icons:

Folders		
Parts		Parts created in Alibre Design
Assemblies		Assemblies created in Alibre Design
Drawings		Drawings created in Alibre Design
Custom symbols		Custom symbols can only be opened when in a drawing workspace. They can be renamed, copied and deleted in the repository. See Chapter 9 for more about custom symbols.
Unknowns		Other items are represented by the icon associated with the originating application, if known; otherwise a question mark is used.
BOM		Bill of Material created in Alibre Design

Items of different types can have the same name. You can give the same name to items of the same type only if they are in different folders.



14.3.2 Item Properties

When you save a part, assembly or drawing for the first time, you are prompted for certain properties.

- **Item Name:** Equivalent to a file name. Items can be renamed.
- **Version Number:** A number that you assign to the version you are saving. Displayed in the item's version history, described below.
- **Description:** A text description. Displayed in the property area of the Repository when the item is selected.
- **Notes:** Text notes attached to an item, indicated by a symbol on the icon . An item can have multiple notes. Access the notes by right-clicking the item and selecting **Add/View Notes**.

Other properties include:

- **Checked Out status:** Indicates whether you, or another user, have the item checked out. You cannot edit an item that is checked out by another user.
- **Version History:** Lists the version number, user name, date saved, and any comment entered when the version was saved. Displayed when you select the item and select **Show Selected Item's Version History** option.
- **Security/Notifications:** You assign access and notification policies for other users.

14.3.3 Selecting Items

To select one item or folder, click the item or folder.

To select multiple items, press the **Ctrl** key as you select each item.

To select all items in a folder, click the top item; then press **Shift** and click the last item.

Note: You can select multiple items, but you cannot select multiple folders.

14.4 Depositing and Withdrawing Other Files

Files created outside Alibre Design may be stored in the Repository (e.g., Word documents, Excel documents, files from other CAD systems, etc.). These files are referred to as **Other items** in the Repository environment. The items can be launched in their associated program via the Repository, checked in/out, changed and saved. Item properties such as notes and version history are also available.

14.4.1 Depositing Other Items

You can deposit **Other** type items from your operating system file structure into the Repository.

To deposit an item:

- 1** In the Repository Explorer, select the repository and folder where you want to deposit the file.
- 2** Right-click in the Explorer and select **Deposit** from the pop-up menu; or from the **File** menu select **Deposit**. The **Deposit to Current Folder** dialog box appears.
- 3** Browse to the location the file is stored in.
- 4** Select the file.
- 5** Click **Open**. The file is stored in the selected repository location.

Check Out the item if you do not want users with access to the folder to make changes to the file. Use **Check In** to make the item available again.

14.4.2 Withdrawing an Item

You can withdraw **Other** type items from the Repository into your operating system file structure.

To withdraw an item:

- 1** Select the item you wish to withdraw. (To retrieve an earlier version of the file, show the version history and select the version you want to retrieve.)
- 2** Right-click and select **Withdraw** from the pop-up menu; or from the **File** menu select **Withdraw**. The **Withdraw Selected Item** dialog box appears.
- 3** Select the location where you want to withdraw the item to.
- 4** Click **Save**. A copy of the item is saved to the new location.

14.5 Opening a Repository Item

You can open the items created in Alibre Design, as well as items you have deposited in the Repository.

14.5.1 To Open an Item

- 1 Select the part, assembly, drawing or other item that you want to open.

- 2 Double-click the item; or from the **File** menu select **Open**; or select the **Open** tool on the Standard toolbar and click the Repository tab in the Open dialog.

An Alibre Design part, assembly, or drawing opens in a workspace.

An **Other** item will open in the originating application. For example, if you open a Word document, the item will be opened in Microsoft Word.

14.5.2 Opening a Folder

- 1 In the Repository Explorer, click the plus sign  to expand a repository and/or any top-level folders, if necessary.
- 2 Select the folder you want to open. The items within the folder appear on the right in the items list.

14.5.3 Opening an Item That is Checked Out

You can view an item that someone else has checked out by opening a read-only copy of it. You cannot make changes to the read-only copy; all the editing commands and toolbar buttons in the workspace are disabled.

To view a read-only copy of an item:

- 1 In the Repository Explorer, browse to the repository or folder that contains the item.
- 2 Click the item to select it.
- 3 Right-click the item and select **Open Read-only**; or from the **File** menu select **Open Read-only**.

14.6 Adding/Viewing Notes for a Repository Item

You can add notes to a repository item. Unlike version comments, which are associated with a specific version of the item, notes belong to the item itself. You can add multiple notes, and you can remove notes.

Note: You cannot rename or edit a note, but you can remove it and replace it with a new note.

14.6.1 Adding a Repository Note

To create a repository note:

- 1 Select the item to which you want to add a note.
- 2 Right-click the item and select **Add/View Notes** from the pop-up menu; or select the **Add/View Notes**  tool from the **Repository Tools** toolbar; or from the **Tools** menu, select **Add/View Notes**. The **Note History** dialog box appears.
- 3 Click **Add**. The **Add New Note** dialog box appears.
- 4 Type a **Subject** for the note, and type the text of the note.
- 5 Click **OK**. A listing for the note appears in the item's Note History.
- 6 Click **OK** to close the Note History dialog box. The icon for the item now includes a small image of a note  to indicate that at least one note has been attached.

14.6.2 Viewing a Repository Note

If a repository item has notes, a small note image is displayed at the bottom right of its icon



To view a note:

- 1 Select the item with a note.
- 2 Right-click the item and select **Add/View Notes** from the pop-up menu; or select the **Add/View Notes**  tool from the **Repository Tools** toolbar; or from the **Tools** menu select **Add/View Notes**. The **Note History** dialog box appears.

- 3 Select the note that you want to view. The box at the bottom of the dialog box displays the text of the selected note.
- 4 Click **OK**.

14.6.3 Removing a Repository Note

To remove a repository note:

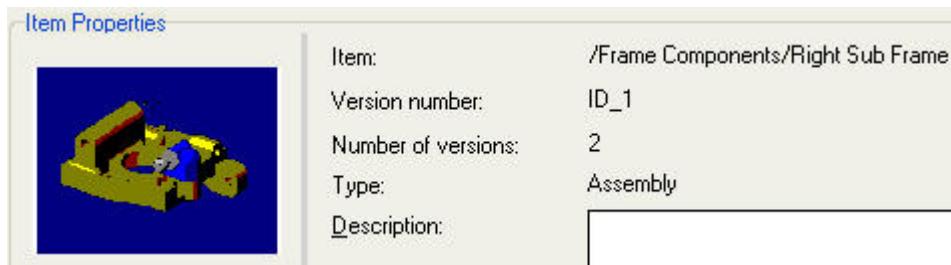
- 1 Select the item from which you want to remove a note.
- 2 Right-click the item and select **Add/View Notes** from the pop-up menu; or select the **Add/View Notes**  tool from the **Repository Tools** toolbar; or from the **Tools** menu select **Add/View Notes**. The **Note History** dialog box appears.
- 3 Select the note that you want to remove.
- 4 Click **Delete**.
- 5 Click **OK**.

14.7 Previewing a Repository Item

You may view a thumbnail preview of items in your repository. Previews are not available for non-native items.

To preview a repository item:

- 1 In the Repository Explorer, browse to the folder containing the item that you want to preview. The list of available items appears.
- 2 Select the item that you want to preview. The thumbnail preview appears in the **Item Properties** area. The thumbnail preview represents the item's display orientation during the last save operation.



Note: You must save items individually in a workspace before a preview is displayed in the Repository. For example if you create parts in the context of an assembly, only the top-level assembly will be previewed in the repository. You must open and save each part in the assembly in order to generate a preview image in the repository.

14.8 Renaming a Repository Item

You can rename items or folders in a repository. When renamed, references to the item or folder are automatically adjusted. To rename an item or folder, you must have **Write** privileges. Its name may be the same as another item in the repository folder (or even the folder name), as long as the items are different types (for example, part or assembly). You cannot rename an item that is checked out.

To rename an item:

- 1 Select the item that you want to rename.
- 2 Right-click the item and select **Rename** from the pop-up menu; or from the **Edit** menu select **Rename**. The **Rename Item** (or **Rename Folder**) dialog box appears.
- 3 Type the new name. The name must meet the naming requirements (see below).
- 4 Click **OK**.

Naming requirements for repository items:

- The name cannot exceed 80 characters.
- The name cannot contain any of the following characters: \ / : * ? " < > | or the tab character.
- The name cannot begin with a period.
- The path name cannot exceed 255 characters.

14.9 Viewing an Item's Version History

To view an item's version history:

- 1 Select the item to view the version history of.
- 2 Select the **Show Selected Item's Version History** option at the bottom of the Item area. The item's version history information appears.

Item Name *	Checked out by	Checked out on	Size
✓ RotorHead	sae	05-Mar-02 10:49:45 AM	312 KB
Stabilizer Control Rod			6 KB
<input checked="" type="checkbox"/> Show Selected Item's Version History			
Version	User	Date	Comment
12	sae@server.v4	15-Feb-02 9:11:45 AM	H-3344-387 inserted and constrained . . . all constrai...
11	sae@server.v4	15-Feb-02 8:56:49 AM	H-3016 inserted and constrained . . . all constrai...
10	sae@server.v4	14-Feb-02 5:40:21 PM	H-3018-42 inserted and partially constrained
9	sae@server.v4	14-Feb-02 4:13:56 PM	H-3322-365 & H-3272-234 inserted and partially ...

14.10 Rolling Back to a Previous Version

You can roll back to a previous version of an item. This action removes all versions that are more recent than the version marked for rollback. This action cannot be reversed or undone. An item cannot be rolled back while it is checked out.

To rollback an item:

- 1 Select the item you want to roll back.
- 2 Select the **Show Selected Item's Version History** option at the bottom of the Item area.
- 3 Click the version that you want to roll back to.
- 4 From the **Tools** menu click **Rollback**; or select the **Rollback**  tool from the Repository Tool toolbar. A confirmation message appears.

- 5 Click **Yes** to proceed. All versions later than the selected version are permanently deleted.

14.11 Purging Previous Versions of an Item

Purging permanently removes all but the most recent version of a repository item.

To purge an item:

- 1 Select the repository item that you want to purge.
- 2 From the **Tools** menu select **Purge**; or select the **Purge**  tool from the Repository Tools toolbar. A confirmation message appears.
- 3 Click **Yes** to permanently remove all versions of the item except the most recent version. You cannot restore the older versions after removing them.

14.12 Undoing a Check Out

Undoing a check out voids any temporary changes made while you had the item checked out. It does not cancel changes that you have saved to the item's version history.

To undo a check out:

- 1 Select the item for which you want to undo the check out. A checked out item is indicated by a checkmark to the left of the item name.

Item Name *	Checked out by	Checked out on	Size
<input checked="" type="checkbox"/> part1	sae	22-Jul-02 5:42:23 PM	89 KB

- 2 Right-click the item and select **Undo Check Out** from the pop-up menu; or from the **Tools** menu select **Undo Check Out**; or select the **Undo Check Out**  tool from the Repository Tools toolbar.

14.13 Copying and Moving Repository Items

Copying an item creates a duplicate of the item in the specified location. You can copy folders, parts, assemblies, drawings, and external items. When you copy a folder, both the folder and its contents are duplicated. It is not necessary to have write privileges to copy an item to your repository.

Moving an item changes the location that the item is stored in and accessed from. You can move folders, parts, assemblies, drawings, bills of material, custom symbols and external items.

However, if you want to share an item with a colleague or team, the best method is to share the repository. See section **14.16** for more information about repository sharing.

14.13.1 Copying an Item

To copy a single item:

- 1 Select the item that you want to copy.
- 2 Click, drag, and drop to the target location.

Or

- 1 Right-click the item and select **Copy** from the pop-up menu; or from the **Edit** menu select **Copy**; or select the **Copy Items**  from the Repository Tools toolbar. The **Copy Item** dialog box appears.
- 2 Select the repository or folder into which you want to copy the item.
- 3 Type a name if you want to give the copy of the item a new name. The name must meet the naming requirements (see section **12.8**).
- 4 Click **OK**.

Copied assemblies

If you copy an assembly in a repository, the location information for the assembly and its constituents changes to the new location, regardless of whether you moved it from another server or locally.

Since the copied assembly's constituents do not rely on whether the original location is online, they can be opened and changed at any time, provided the appropriate rights have been assigned and it is available to be checked out.

14.13.2 Moving an Item

You must have **Delete** permission on the item being moved and **Write** permission to the folder or repository where the item is being moved to.

- Moving an assembly or drawing does not move the associated constituents with it.
- Moving a folder includes the folder's contents.

To move an item:

- 1 Select the item that you want to move.
- 2 Hold the **Shift** key and drag the item to the target location.
Or
- 1 Right-click the item and select **Move** from the pop-up menu; or select the item and from the **Edit** menu, select **Move**. The **Move Item** dialog box appears.
- 2 Select the target location.
- 3 Click **OK**.

The item is moved to the specified location.

14.14 Deleting a Repository Item

You can delete folders and items in your repository. You can also delete an entire local repository if you own it.

Deleting an item not only deletes all versions of the item. To delete an item, you must have **Delete** privileges. You cannot delete an item that is checked out.

To delete a repository item:

- 1 Click the item that you want to delete.
- 2 Press **Delete** on the keyboard; or right-click the item and select **Delete** from the pop-up menu; or from the **Edit** menu select **Delete**.

Note: If you have been assigned policies since opening Libre Design, you may need to perform a **Refresh** operation in the Repository for those policies to be

active. Until you Refresh, you may not be able to perform any saving, moving, copying, etc. of secure folders.

To restore a deleted item:

- 1 Expand the repository in which the item was deleted.
- 2 Click the **Recycle Bin**.
- 3 Right-click the deleted item in the item list and select **Restore** in the pop-up menu.

The item is restored back to its original location.

14.15 Repository Folders

Folders in the repository are similar to other folders on your computer. They are used to organize your data. You can determine the total size and number of contents in a folder or repository through the **Folder Properties**.

You can create, rename, copy and delete folders in your repository. Copying a folder creates a duplicate of the folder and its contents in a location that you specify. When you copy a folder that contains assemblies or drawings, all parts and subassemblies referenced in those items are duplicated in the new folder. Deleting a folder removes both the folder and all of its contents.

14.15.1 Creating a Folder

To create a folder:

- 1 Right-click the folder or repository that will hold the new folder and select **Create New Folder**. Or, select a repository or folder then select **New Folder** from the **File** menu. The **New Folder** dialog box appears.
- 2 Enter a name for the folder. Review the naming requirements if necessary (section **12.8**).
- 3 Click **OK**.

14.15.2 Copying a Folder

To copy a folder:

- 1 Select a folder and drag it to a new repository or folder. A progress meter will appear while the copy takes place. It will display an estimate of the remaining time needed to complete the copy.

Or

- 1 Right-click the folder and select **Copy**. Or, select the folder and choose **Copy** from the **Edit** menu. The **Copy Folder** dialog box appears.
- 2 Select the repository and/or folder where you want to copy the folder.
- 3 Click **OK**. A progress meter will appear while the copy takes place. It will display an estimate of the remaining time needed to complete the copy.

14.15.3 Deleting a Folder

To delete a folder, you must have **Delete** privileges.

To delete a folder:

- 1 Right-click the folder and select **Delete**. The **Send to Recycle Bin** dialog box appears.
- 2 Click **Yes**. The contents of the folder are moved to the Recycle Bin in that repository.

14.16 Sharing and Unsharing Repositories

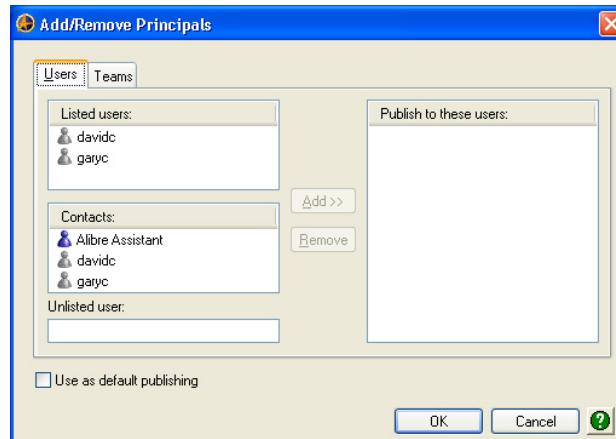
You can select which teams and individual users may view and access data in your repositories. You must be the owner of the repository to share it.

Sharing a repository allows others only to see the repository. Assigning permissions to repository items will allow others subsequently to see and access data in the shared repository.

To share a repository to a listed team or user:

- 1 Select the repository that you want to share.

- 2 Right-click the repository and select **Sharing/Security** from the pop-up menu; or select the **Sharing/Security** tool from the Repository Tools toolbar; or from the **Tools** menu select **Sharing/Security**. The **Sharing/Security** dialog box appears and the **Sharing** tab is active.



- 3 Click the **Add/Remove** button to bring up the **Add/Remove Principles** dialog. Use this dialog to select and **Add** individual users and teams. To select multiple entries, press and hold the **Ctrl** key as you select.
- 4 Click **OK** to close the **Add/Remove Principles** dialog.

- 5 Click **Apply** and **Close**.

To unshare a repository to a team or user:

You can unshare a repository by following the same procedure as above, except use the **Remove** button in the **Add/Remove Principles** dialog.

14.17 Setting Permission Policies for Repository Items

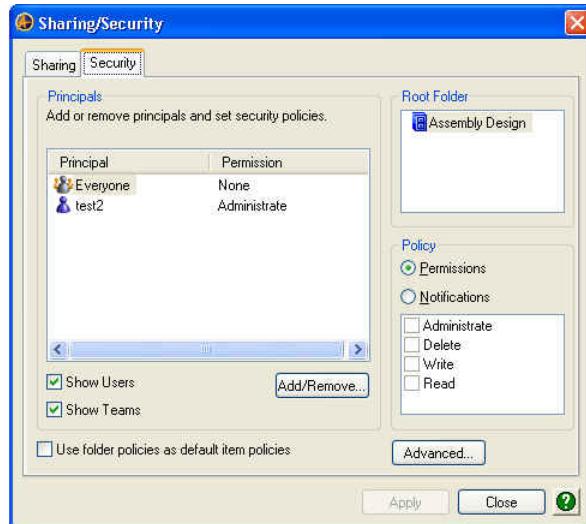
You can grant other users secure access to items and folders in your repositories by setting **Permission Policies** in the **Sharing/Security** dialog. Permission options include:

Administratate, Delete, Read and Write. Users granted **Administratate** rights may grant access rights to other users. You must be working **online** to modify the permission policies of your data and you must have already shared the repository to the users you are granting data access.

To simplify the assignment of permission policies, grant rights to a team role instead of individual users. Then add and remove users from that team role. To assign permissions for a team, you must have administrative privileges for the team. See **Chapter 16** for more information about teams and roles.

To modify the permission policies:

- 1 Select the folder or item for which you want to assign permissions.
- 2 Right-click the folder or item and select **Sharing/Security** from the pop-up menu; or select the **Sharing/Security**  tool from the Repository Tools toolbar; or from the **Tools** menu select **Sharing/Security**. The **Sharing/Security** dialog box appears and the **Security** tab is active.



- 3 Click the **Add/Remove** button to bring up the **Add/Remove Principals** dialog. Use this dialog to select and **Add** or **Remove** individual users and teams. To select multiple entries, press and hold the **Ctrl** key as you select.
- 4 Click the **Permissions** radio button to display the four access rights: Administrate, Delete, Read and Write. Check the desired options.
- 5 Click **Apply** and **Close**.

Note: If you select a folder or repository, you can choose to apply the folder's (or repository's) permission policies as the default settings for any new items created in the folder. Alternatively, you can click the **Advanced** button and explicitly define the default permission policies to be used for newly created items. Finally, on the **Apply Options** tab in the **Advanced** dialog, you can choose to apply these default policies to existing items or to apply them recursively to subfolders. You can also choose to **Replace/Combine** any existing policies by/with the new policies.

14.18 Assigning Notification Policies for Repository Items

Notification policies determine when a user is notified, via an automatic system message, of a specific activity associated with an item. You can specify that a user or team receive notification of the following events:

- Administrate

- Check in (not available for folders and repositories)
- Check out (not available for folders and repositories)
- Delete
- Write

Note: To assign notification policies to a team role, you must have administrative privileges for that team.

To assign notifications:

- 1 Select the folder or item for which you want to assign notifications.
- 2 Right-click the folder or item and select **Sharing/Security** from the pop-up menu; or select the **Sharing/Security**  tool from the Repository Tools toolbar; or from the **Tools** menu select **Sharing/Security**. The **Sharing/Security** dialog box appears and the **Security** tab is active.
- 3 Click the **Add/Remove** button to bring up the **Add/Remove Principals** dialog. Use this dialog to select and **add** or **remove** individual users and teams. To select multiple entries, press and hold the **Ctrl** key as you select.
- 4 Click the **Notifications** radio button to display the five types of notifications (three for folders and repositories). Check the desired options.
- 5 Click **Apply** and **Close**.

Note: If you select a folder or repository, you can choose to apply the folder's (or repository's) notification policies as the default settings for any new items created in the folder. Alternatively, you can click the **Advanced** button and explicitly define the default notification policies to be used for newly created items. Finally, on the **Apply Options** tab in the **Advanced** dialog, you can choose to apply these default policies to existing items or to apply them recursively to subfolders. You can also choose to **Replace/Combine** any existing policies by/with the new policies.

14.19 Repository Snapshots

You can create a snapshot copy of a local repository by saving it as a single file on the Windows file system. Then you can copy the snapshot to another computer with Alibre Design installed and insert the repository into the Alibre Design environment.

To create a repository snapshot file:

- 1 In the repository browser, right-click the repository for which you want a snapshot and select **Save Snapshot of Repository** from the right mouse pop-up. Or, click the desired repository and select **Save Snapshot of Repository** from the **Tools** main menu.

The **Save Snapshot of Repository** dialog appears.

- 2 Specify a name and file system folder for the snapshot file.
- 3 Click **OK**.

To create a repository from a snapshot file:

- 1 In the repository browser, right-click the repository list and select **Create Repository from Snapshot** from the right mouse pop-up. Or, select **Create Repository from Snapshot** from the **Tools** main menu.

The **Create Repository from Snapshot** dialog appears.

- 2 Specify the snapshot file.
- 3 Specify a file system folder to be used for the newly inserted repository.
- 4 If desired, specify a different name for the new repository.
- 5 Click **OK**. The repository and the items in it are now available for use in Alibre Design.

14.20 Caching

To avoid delays in loading model data before a Team Design session you may cache items in advance. Caching an item stores a copy of the item in the temporary system memory and reduces the model load time.

You can cache parts, assemblies and drawings. If an assembly has constituents, the constituents are also cached. You can cache individual items, or you can cache multiple items by selecting a folder.

Note: The repository with the item must have been shared to you. You must have at least read permissions for the repository, folder and item.

When an item is cached, the icon for that item changes based on the status of the caching process.



Red arrows: Caching has been requested, but the item is not yet cached.



Green arrows: The item and all of its dependencies (if any) are cached.



Blue arrows: The item is cached, but one or more of its dependencies (version history or constituents) are not yet cached.

14.20.1 Caching Options for Items

From the Item Cache Setting dialog box (**Repository > File > Caching**), in addition to caching the item, options include:

- Cache most recent versions only
- Cache the entire version history
- Cache constituents (only used with assemblies)
- The cache priority order when caching more than one item.

14.20.2 Caching Options for Folders

The folder cache settings may be applied to all item types or just one.

14.21 Caching Repository Items

Caching is useful for items residing in a shared repository that you want to be able to open quickly. Caching is also beneficial for users who plan to participate in a Team Design session.

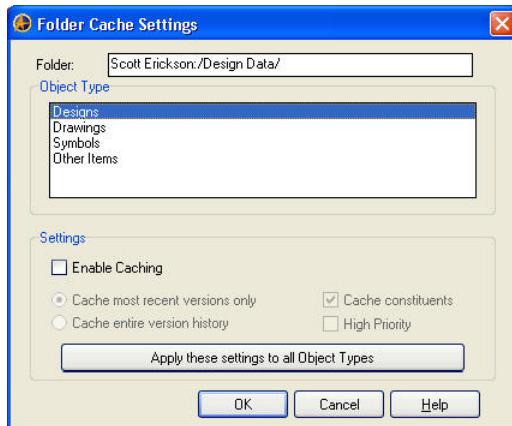
Caching an item stores a copy of the selected item in your temporary system memory. A cached item is physically stored in your local repository but it will not be listed in your repository. You must continue to access the item from the repository from where you cached it.

14.21.1 Caching a Repository Folder

To cache a repository folder:

- 1 Select the folder to be cached.

- 2 Right-click the folder and select **Caching** from the pop-up menu; or select the **Caching** tool from the Repository Tools toolbar; or from the **Tools** menu select **Caching**. The **Folder Cache Settings** dialog box appears.



- 3 Select the item type to cache in the **Object Type** area.
- 4 In the **Settings** area, select **Enable Caching**.
- 5 Select the **Cache most recent versions only** option if desired.
Or select the **Cache entire version history** option.
- 6 If assemblies are involved, select **Cache constituents** if you want to cache the constituents as well.
- 7 Select the **High Priority** option if you want an item type to be cached before other items.

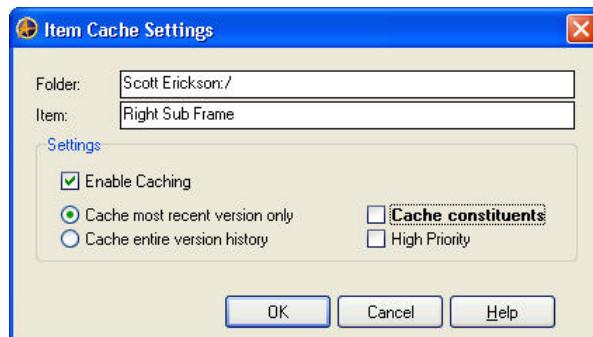
Note: Using the **High Priority** option for an item requires that item to be cached before other items selected are cached. Use **High Priority** for parts or assemblies that are needed first.

- 8 Click **Apply these settings to all Object Types** if you want all object types selected in the **Object Type** area of this dialog box to have the same settings.
- 9 Click **OK**.

14.21.2 Caching a Repository Item

To cache an item:

- 1 Select the item to be cached.
- 2 Right-click the item and select **Caching** from the pop-up menu; or select the  **Caching** tool from the Repository Tools toolbar; or from the **Tools** menu select **Caching**. The **Item Cache Settings** dialog box appears.



- 3 In the **Settings** area, select **Enable Caching**.
- 4 Select the **Cache most recent versions only** option if desired.
Or select the **Cache entire version history** option.
- 5 If you are caching an assembly, select the **Cache constituents** option if you want to cache the constituents as well.

Note: The **Current Item Status** area provides information on the item you are caching such as whether the item is currently cached and whether the version history was cached with the item or not.

- 6 Click **OK**.

14.21.3 Disabling Caching Repository Items

If you no longer want a repository item cached, disable the caching of the item.

To disable items cached in a repository folder:

- 1 Select the folder which you want to disable the caching of items.
- 2 Right-click the folder and select **Caching** from the pop-up menu; or select the  **Caching** tool from the Repository Tools toolbar; or from the **Tools** menu select **Caching**. The **Folder Cache Settings** dialog box appears.
- 3 In the **Settings** area, clear **Enable Caching**.
- 4 Click **OK**.

To disable caching of a repository item:

- 1 Select the item that you want to disable the caching of.
- 2 Right-click the item and select **Caching** from the pop-up menu; or select the  **Caching** tool from the Repository Tools toolbar; or from the **Tools** menu select **Caching**. The **Folder Cache Settings** dialog box appears.
- 3 In the Settings area, clear **Enable Caching**.
- 4 Click **OK**.

NOTE: When caching is disabled for an item, its icon no longer displays its cached status.

15 The Message Center

The Message Center is used to manage, send and retrieve voice and text messages. There are two types of messages stored in the Message Center: messages sent by other users, and messages that report activity associated with repository item notifications. New messages are sent to the Inbox in the Message Center. Additionally, you may have new messages forwarded to an email account.

Messages can be organized using folders, similar to email applications. Messages contain the following information: user name of sender, date and time the message arrived, subject of the message, and a recording or text. In addition, team session invitations include an attachment containing the session details.

This chapter describes:

- Opening the Message Center
- Reading messages
- Sending messages
- Organizing messages
- Setting message options

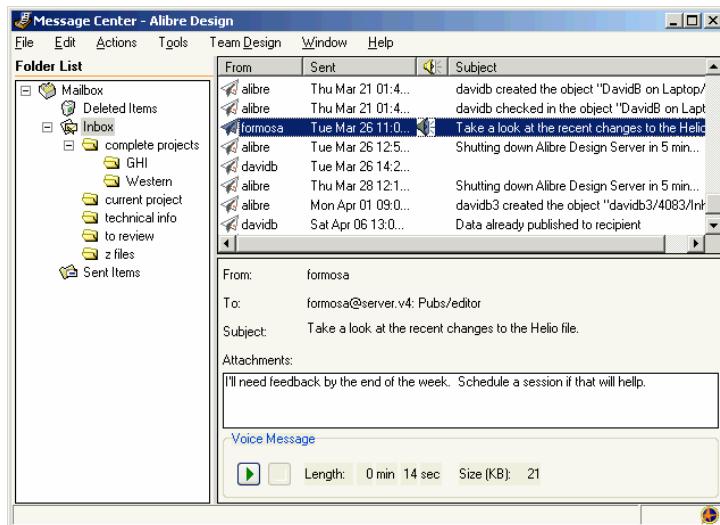
15.1 Opening the Message Center

To open the Message Center from the Home window:

- 1 Select the **Message Center**  tool from the Standard toolbar; or from the **Window** menu select **Message Center**. The **Message Center** window appears.

To open the Message Center from any other area:

- 1 From the **Window** menu, select **Message Center**. The **Message Center** window appears.



15.2 Retrieving Messages

You can retrieve messages when working online. For recorded messages, you will need speakers or headphones.

15.2.1 Reading a Text Message

To read a text message

- 1 Select the **Inbox**.

- 2 Select a message. The text of the message appears at the bottom of the window.

15.2.2 To Play a Recorded Message

- 1 Click the green arrow to start the message.



- 2 Click the black square to stop or pause the message.



15.3 Sending Messages

Messages can be sent from the Message Center, the Home window and any open workspace. Messages can be sent any time you are working online. Recipients working online will be notified of new messages immediately. Otherwise, the message will be delivered to their Inbox in the Message Center upon their next sign-in.

15.3.1 Creating a New Message From the Home Window

To send a message from the Home window:

- 1 From the **Actions** menu, select **Send Message**. Or right-click a contact and select **Send Message**. The **To** field will be populated automatically, and more recipients can be added.



Or click the **Send Message** icon.

15.3.2 Creating a New Message from the Message Center

To create a new message:

- 1 From the **File** menu, select **New > Message**.

Or from the **Actions** menu, select **New Message**.

Or right-click the message list and select **New Message**.

The **New Message** dialog box appears.



15.3.3 Working With the New Message Dialog Box

- 1 Click **To:**. The **Recipients** dialog box appears.
- 2 Select the teams and users to whom the message will be sent. Click the **Teams** tab to select teams. Click the **Members** tab to select members. **Ctrl + click** to select multiple users or teams.

Note: To write to an unlisted team or user, type the team or user name in the **Unlisted** field.
- 3 Click **Add**. Selected users or teams appear in the recipients list.
- 4 Click **OK**. The selected users and teams appear in the To field.
- 5 Type a subject (optional).
- 6 Type or record a message to activate the Send button.
- 7 Click **Send**.

15.3.4 Recording a Voice Message

- 1 Click the **Record**  button to begin recording your message. Voice messages may last as long as 60 seconds.

- 2 Click the **Stop**  button when finished recording. The number of seconds recorded appears and the **Send** button becomes active.

15.4 Replying to Messages

To reply to messages:

- 1 Right-click a message.
- 2 Select **Reply** or **Reply to All**.
- 3 Type a subject (optional).
- 4 Type or record a message.
- 5 Click **Send**.

Note: You may only reply to a message from the Message Center. You cannot reply to notifications.

15.5 Deleting Messages

Deleting a message moves it to the **Deleted Items** folder. Deleting a message from the Deleted Items folder permanently removes the message.

To delete a message:

- 1 Expand the folder that contains the message.
- 2 Select the message and press **Delete** on the keyboard; or right-click the message and select **Delete Message** from the pop-up menu; or from the **Edit** menu select **Delete**. The message moves to the **Deleted Items** folder.

To empty the Deleted Items folder, click the folder and select **Empty Deleted Items** from the **Tools** menu.

15.6 Using Folders in the Message Center

15.6.1 Creating a New Folder

To create a new folder:

- 1 Select the location for the new folder.
- 2 Right-click and select **New Folder** from the pop-up menu; or from the **File** menu select **New Folder**. A folder appears with the temporary name “New Folder” highlighted.
- 3 Type a name for the folder
- 4 Press **Enter**.

15.6.2 Deleting a Folder

To delete a folder:

- 1 Select a folder.
- 2 Press **Delete** on the keyboard; or right-click a folder and select **Delete Folder** from the pop-up menu; or from the **Edit** menu select **Delete**. The folder is moved to the Deleted Items folder.

15.6.3 Moving a Folder into Another Folder

- Select and drag the folder over another folder.

15.6.4 Renaming a Folder

To rename a folder:

- 1 Select a folder to highlight it.
- 2 Click again to make the folder name editable.
- 3 Type a new name for the folder.
- 4 Press **Enter**.

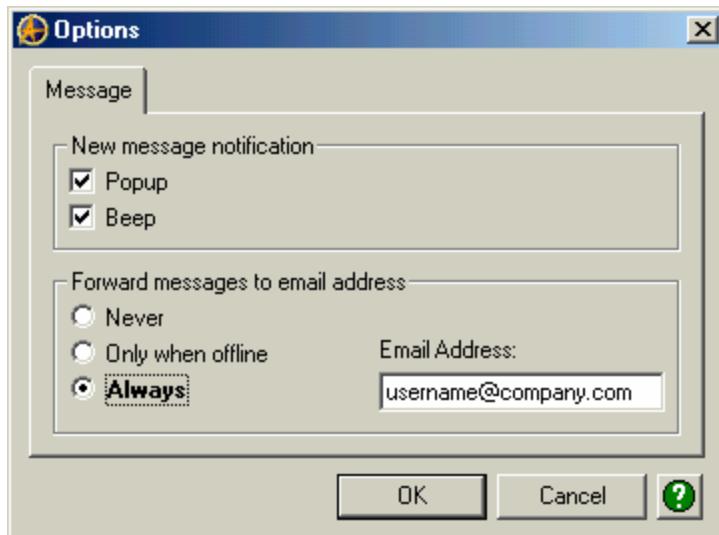
15.7 Setting Message Options

Message options affect how you are notified of messages. When working in Alibre Design, you may choose to be alerted to new messages through a pop-up window or sound.

In addition, you may choose that messages be forwarded to an email account, all the time, only when you are offline, or never. Forward copies of your messages to an email account to be notified of changes to designs, even when you are signed out of Alibre Design.

To set message options:

- 1 From the **Tools** menu select **Options**. The **Message** options dialog box appears.



- 2 Select the **Popup** option to receive an alert via a pop-up window.
- 3 Select the **Beep** option for an audible alert.
- 4 To forward Alibre Design messages to your email account, select **Never**, **Only when offline**, or **Always**.

Note: With **Never** selected, messages are not forwarded. The **Only when offline** option only forwards them when you are signed out of Alibre Design. Selecting **Always** forwards all messages to email account you specify.

- 5 Additionally, if you selected the **Only when offline** or **Always** option, type an email address.
- 6 Click **OK**.

16 The Team Manager

Alibre Design enables you to efficiently manage people and data by defining roles and teams for your contacts. Teams are ideal for projects involving multiple people. Team administration is handled in the Team Manager window, which is only accessible when working online.

The Team Manager is where teams and roles are created, members are added to teams and roles are assigned to members.

Teams published to you also appear in the Team Manager. All members and roles are displayed, but may only be modified by members granted administrative privileges.

This chapter describes:

- Opening the Team Manager
- Creating and deleting teams
- Creating and deleting team roles
- Assigning roles to team members
- Publishing teams

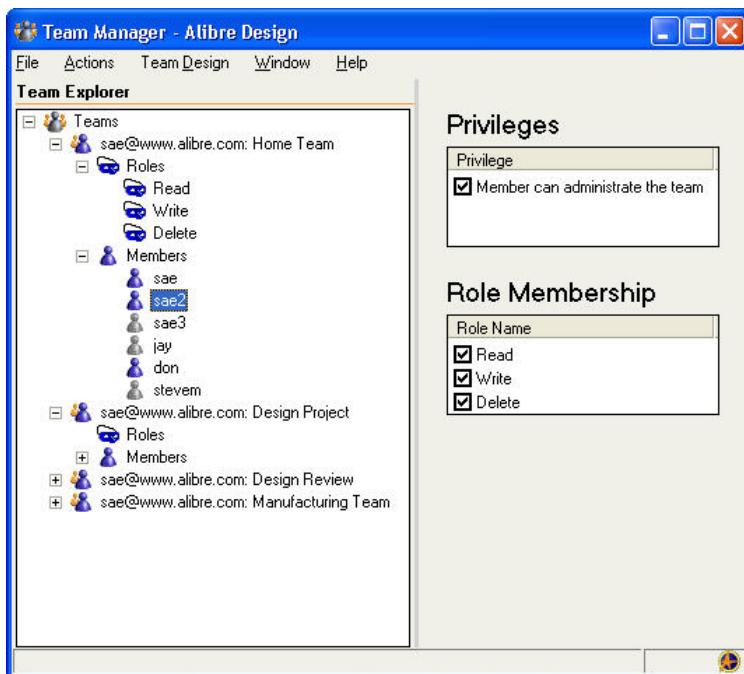
16.1 Opening the Team Manager

To open the Team Manager from the Home window:

- 1 Select the **Team Manager**  tool from the Standard toolbar; or from the **Window** menu select **Team Manager**. The **Team Manager** window appears.

To open the Team Manager from any other area:

- 1 From the **Window** menu, select **Team Manager**. The **Team Manager** window appears.



16.2 Creating and Deleting Teams

In the Team Manager, one team by default, your Home team, is displayed in the Teams Explorer with **Roles** and **Members**. You are the default member of your Home team.



To add a team:

- Right-click **Teams** and select **Add Team** from the pop-up menu.
- Or select **Add Team** from the **Actions** menu.

To delete a team:

- Right-click the team and select **Remove Team**.

Or

- 1 In the Team Explorer, select a team.
- 2 Select **Remove Team** from the **Actions** menu.

To add a member to a team:

- 1 Expand the team so that **Roles** and **Members** are visible.
- 2 Right-click **Members** and select **Add Team Member** from the pop-up menu; or from the **Actions** menu select **Add Team Member**. The **Add Member** dialog box appears.
- 3 Select a user from the Listed users or Contacts list, or type a user name in **Unlisted user** box.
- 4 Click **Add**. The user name is added to the **Members** list.
- 5 Click **OK**. The new member appears in the Team Explorer.

Note: To add members to a team, you must be the creator of the team or have administrative rights for it.

To remove a team member:

- Right-click the member and select **Remove Team Member** from the pop-up menu.

Or

- 1 In the Team Explorer, select the member.
- 2 From the **Actions** menu, select **Remove Team Member**.

16.3 Creating and Deleting Team Roles

Roles are typically used to control data access. For example, some team members may need permission to modify data while others only need permission to view data. An “Engineer” role could be set with more advanced permissions and a “Reviewer” role could be set with more limited access to the data. Permissions are set in the Repository. (For more information about the Repository, refer to chapter **13**).

Team roles provide a way to group team members for the purposes of sharing data and establishing access permissions to data. Combinations of permissions and notifications can be established to limit whether other users may view or modify repository items.

After you create a team and its associated roles, data can easily be shared with the entire team by sharing a repository to the team. Teams can also be invited to a Team Design session, eliminating the need to send an invitation to multiple people.

If a user is removed from a team, they are also removed from the team session.

Roles may be set to determine whether other users may:

- Check out and make changes to items.
- Overwrite changes made by other users.
- Delete.
- Be notified of changes.
- Be notified if users with administrative privileges are added.
- View the changes as new versions are checked in.

To create and delete team roles, you must be the creator of a team or have administrative privileges for it.

Members with administrative privileges may:

- Add and remove team members
- Add and delete roles
- Modify which roles are assigned to members.
- Grant access privileges for repository items to teams and roles, through the Repository.

To add a role to a team:

- 1 Expand the team so that **Roles** and **Members** are visible.
 - 2 Right-click **Roles** and select **Add Team Role** from the pop-up menu; or from the **Actions** menu select **Add Team Role**. The **Add Team Role** dialog box appears.
- Note:** If the command is dimmed, you do not have administrative privileges and may not add or remove roles.
- 3 Enter a name for the role.
 - 4 Click **OK**.

To remove a team role:

- Right-click the role and select **Remove Team Role** from the pop-up menu. The role is deleted.

Or

- 1 In the Team Explorer, select the role.
- 2 Select **Remove Team Role** from the **Actions** menu. The role is deleted.

To assign roles to a team member:

- 1 Expand the team so that **Roles** and **Members** are visible.
- 2 Expand **Members** so that the members are visible.
- 3 Select a team member. All roles in that team appear under **Role Membership** in the Team Manager.
- 4 Click to check or un-check a role. Checked roles are applied to the selected member.

16.4 Publishing a Team

You can publish one of your teams to a user or another team, which causes the team to appear in their Team Manager, and in the Teams list in dialog boxes, for publishing their own teams and team sessions and for sharing repositories.

Note: You must have administrative privileges for a team to publish it.

To publish a team:

- 1 Select a team.
- 2 From the **Actions** menu select **Publishing**. The **Publishing** dialog box appears.
- 3 Select a team from the Teams tab or a user from the Users tab. Ctrl-click to select multiple teams or users.
- 4 Click **Add**.
- 5 Click **OK**.

17 Team Design Sessions

Team Design sessions let you work online with other Alibre Design users to view, edit and provide feedback on designs. By reviewing designs and proposed changes together in real time, you can discuss issues and reach agreement faster, and eliminate delays inherent in traditional design-approval-redesign cycles.

Team Design sessions occur in a secure environment—only invited users can participate. The person who initiates the session, the leader, has full control over who attends and each user's level of participation. As the leader, you invite and admit users and assign each attendee viewer or editor status. During the session, all participants can see design changes in real time and insert comments, but only those with editor status will be able to make design changes. Additionally, you can lead or join any number of concurrent team sessions.

Team Design sessions help you work quickly and efficiently with your associates and other users. Alibre Design provides a number of tools to facilitate communication during a session including text chat, voice chat, private messages, reference arrows, redline and markup, and view manipulation and reorientation.

This chapter describes:

- Leading and joining a team session
- Working in a session
- Scheduling a session
- Inserting redlines and markups
- Leaving a session
- Saving the work done in a session
- Emailing the chat record

17.1 Leading a Team Session

When you start a team session, you are the session leader. In the Team Design explorer, you are distinguished as the leader by the crown icon .

The leader is responsible for:

- Inviting participants.
- Accepting or rejecting applicants.
- Designating participants as editors or viewers.
- Issuing free passes.
- Adding parts, assemblies and drawings to the session.
- Saving changes made to the design during the session.
- Ending the session.

If you want to lead a new team session while already participating in one, launch the new session from another workspace or the Home window.

17.1.1 Leading a Session from the Home Window

You can initiate a team design session directly from the Home window.

To lead a session from the Home window:

- 1 From the **Actions** menu, select **Lead Session**; or right-click in the **Contacts** area and select **Lead Session**.

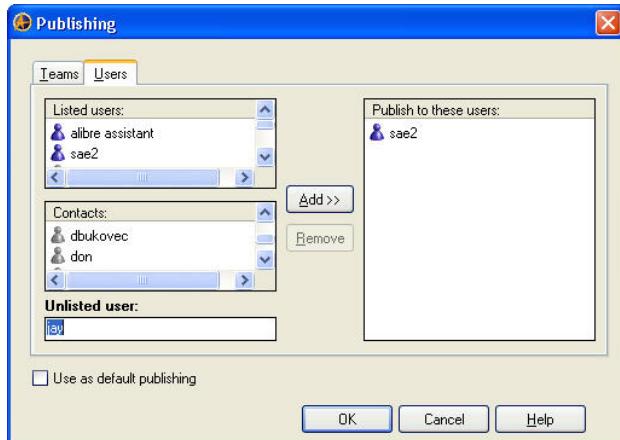
Note: To quickly start a chat session, right-click a user in the Contact list and select **Lead Session > No Data**. The Chat Window immediately opens and an invitation is sent to that contact.



Or click the **Lead Session** button.

- 2 Choose the type of data that will be used to start the session. Additional data can be added after the session begins.
 - **No Data:** the session will start without data, in the Chat Window.
 - **New Part/Assembly/Drawing:** the session will start in an empty workspace of the selected type.

- **Repository Item:** the session will start with the design selected through the **Select Item** dialog box. The **Publishing** dialog box appears.



- 3 In the **Listed teams** area or **Listed users** area, select the teams or users to invite. Ctrl-click to select multiple teams/users. If a specific team or user does not appear in the list, type it in the **Unlisted team** or **Unlisted user** box.
- 4 Click **Add**. The name appears in the **Publish to these teams** or **Publish to these users** area.
- 5 Select **Use as default publishing** to use the same group of teams and users for subsequent team sessions.
- 6 Click **OK**. If **Repository Item** was selected in step 2, the **Select Item** dialog box appears, otherwise the team session starts and invitations are sent.
- 7 If the **Select Item** dialog box appears, browse and select a design.
- 8 Click **OK**. The Team Design session starts with the selected design and invitations are sent.

17.1.2 Leading a Session from a Workspace

To lead a session from a workspace:

- 1 Open the design with which you want to start the session. You may also choose to start with a new part, assembly or drawing.
- 2 From the **Team Design** menu select **Lead Session**. Or select the **Lead Team Design Session**  tool from the Team Design toolbar. The **Publishing** dialog box appears.
- 3 Select the teams and users to invite to the session.
- 4 Click **OK**. The Team Design session starts and invitations are sent.

17.1.3 Accepting or Rejecting a Session Applicant

Once a Team Design session has been published, invited users can apply to join the session. The leader must then admit each applicant and assign editor or viewer status. Alternatively, the session leader can issue free passes, which allow invited users to bypass the request process. The leader may also change each participant's status during the session.

To accept a session applicant:

- 1 When a user applies to join your session, the **Leader Controls** dialog box will appear. Users who are ready to join the Team Design session are listed under the **Applicants** tab.



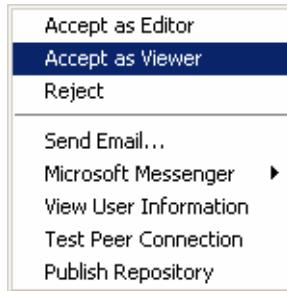
Note: You can also access the Leader Controls dialog box from the **Team Design** menu by selecting **Leader Controls**.

2 Select each user name and choose from the following:

- **Accept as Editor:** The user is admitted to the session and can hold the baton to edit the design.
- **Accept as Viewer:** The user is admitted to the session as a viewer. The participant cannot edit the design, but can observe the session, enter comments in the Chat Window and join in the conversation.
- **Reject:** The user is not admitted to the session, but the invitation to join is still open. The user can apply to join again. If the user's participation is not desired, it is best to remove the invitation.

3 Click **Close**.

You may also bypass the Leader Controls dialog box. Just right-click the user name under **Applicants** in the **Team Design** explorer and select the appropriate status. The result is the same.



17.1.4 Leader Controls: Toggling the Status of a Participant

The leader of a Team Design session may change the status of a participant at any time. This may be done with the Leader Controls dialog box or through the Team Design explorer.

To toggle a participant's status:

- In the Team Design explorer, right-click the user name under the **Participants** list and select **Toggle Status** from the pop-up menu. The status icon changes from a cap to a hard hat or vice versa.

Or

- 1 From the **Team Design** menu select **Leader Controls**. The **Leader Controls** dialog box appears.
- 2 Click **Participants**.
- 3 Select the participant whose status you want to change. Ctrl-click to select multiple user names.
- 4 Click **Toggle Status**. The status icon changes from a cap  to a hard hat  or vice versa.

17.1.5 Leader Controls: Removing a Participant

The leader of a Team Design session may remove a participant from the session at any time. This may be done with the Leader Controls dialog box or through the Team Design explorer. The participant can reapply to the session unless you remove that participant from the Publishing list.

To remove a participant:

- Right-click the user name under **Participants** list in the Team Design explorer and select **Remove** from the pop-up menu. If the participant has the baton, it is returned to you. A message in the Chat Window notes that the participant has left the session.

Or

- 1 From the **Team Design** menu select **Leader Controls**. The **Leader Controls** dialog box appears.
- 2 Select the **Participants** tab.
- 3 Click the name of the participant you want to remove.
- 4 Click **Remove**. If the participant has the baton, it is returned to you. A message in the Chat Window notes that the participant has left the session.

17.1.6 Leader Controls: Free Passes

The leader of a Team Design session can issue free passes to specific users, which allow them to bypass the admittance process. Users who have been issued a free pass are immediately accepted into the session when they join.

The session leader can

- Assign a free pass.
- Change the editor/viewer status of a user's free pass.
- Remove a free pass.

To assign a free pass:

- 1 From the **Team Design** menu select **Leader Controls**. The **Leader Controls** dialog box appears with the Free Pass Holders tab displayed.
- 2 Click **Issue Passes**. The **Issue Free Passes** dialog box appears with a list of users to whom you have published the session.



- 3 Select the users to whom you want to issue a free pass. Ctrl-click to select multiple users.
- 4 Select **Add as Viewer** or **Add as Editor**. The dialog box closes and the user appears in the Free Pass Holders list in the Leader Controls dialog box with the appropriate status icon.

To change the status of a user's free pass:

- 1 From the **Team Design** menu select **Leader Controls**. The **Leader Controls** dialog box appears with the Free Pass Holders tab displayed.
- 2 Click the member whose status you want to change.
- 3 Click **Toggle Status**.

To remove a free pass:

- 1 From the **Team Design** menu select **Leader Controls**. The Leader Controls dialog box appears with the Free Pass Holders tab displayed.
- 2 Click the member you want to remove.
- 3 Click **Remove**. The member may still apply to join the session, but as leader you must approve or reject the application.

17.1.7 Publishing a Session in Progress to Additional Users

Once a session is in progress, the leader may invite additional users and teams through the Publishing dialog box. At the same time, the leader can also effectively un-invite users or teams.

To publish a session in progress to additional teams or users:

- 1 When leading a team session, from the **Team Design** menu select **Publishing**. The **Publishing** dialog box appears with the current teams and users listed.
- 2 Click the **Teams** tab or the **Users** tab, as needed.
- 3 In the **Listed** area, select the team or user you want to add to the team session.
- 4 In the **Unlisted** area, type the name of any unlisted team or user you want to add to the team session.
- 5 Click **Add**. The names appear in the **Recipients** box.
- 6 Click **OK**. Your team session will be visible to invited users in the Sessions tab in the Home window or in the Join Session dialog box.

To remove teams or users from a session:

- 1 When leading a team session, from the **Team Design** menu, select **Publishing**. The Publishing dialog box appears with the current teams and users listed.
- 2 Click the **Teams** tab or the **Users** tab, as needed.
- 3 In the **Recipients** box, select the team or user you want to remove from the team session.
- 4 Click **Remove**.

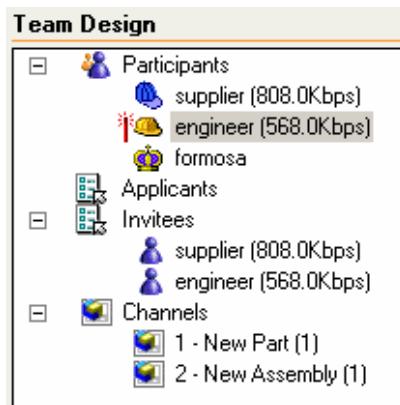
- 5 Click **OK**. The team session is no longer available to those users.

17.1.8 Adding a Design to a Team Session

The leader of a team session can add and remove parts, assemblies and drawings at any time. Workspaces that are part of a specific team session are listed in the **Team Design** explorer and menu as **channels**. The active workspace/channel is marked with a check. Each workspace is also listed in the **Windows** menu.

To add a design to an active session:

- 1 From the **Team Design** menu select **Open in Session**.
- 2 Select **New Part/Assembly/Drawing** to open an empty workspace of the selected type; or select **Repository Item** to open an existing design through the **Select Item** dialog box.
The added workspace appears in the **Team Design** explorer under **Channels**.



To add an open design to an active session

- 1 Open a part, assembly or drawing. The design opens in an appropriate workspace.
- 2 In that same workspace, from the **Team Design** menu, select **Add to Session**.
- 3 If you are currently leading more than one team session, the **Add to Session** dialog box appears so that you can choose a session. Otherwise, the item immediately becomes a part of the active session and is available to participants.

17.1.9 Removing a Design from an Active Session

To remove a design from an active session:

- 1 From the **Team Design** menu select **Remove from Session**. The **End Session** dialog box appears with the message: **Are you sure you want to remove this item from your Team Design session?**.
- 2 Click **Yes**. The item is removed from the session, but remains open.

Or

 - From the **File** menu, select **Close**. The workspace closes and is removed from the session.

Note: If you select **Remove from Session** when only one part is active in a Team Design session, the session will end. You will be prompted with the End Session dialog box.

17.1.10 Ending a Team Design Session

The leader of a Team Design session is the only participant who can end the session. The leader can end the session at any time by closing or removing the last item from the session, or by selecting End Session from the Team Design menu or toolbar.

To end a session and continue to work with the design:

- 1 From the **Team Design** menu, select **End Session**; or select the **End Session** tool from the Team Design toolbar. The **End Session** dialog box appears with the message, **Are you sure you want to end this team session?**
- 2 Click **Yes**. Participants are notified and their workspace closes. As the leader, the Team Design tools disappear, but the workspace remains open.
- 3 To close the workspace, select **Close** from the **File** menu.

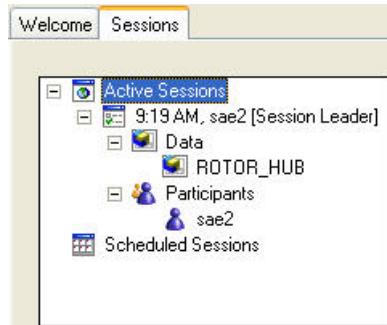
To end a session and exit the design:

- 1 From the **File** menu, select **Close** until all items associated with the Team Design session are closed. On the last item, the **End Session** dialog box appears with the message: **Are you sure you want to end this team session?**
- 2 Click **Yes**. The file closes and the session ends. Participants are notified that the session has ended.

17.2 Joining and Leaving a Team Design Session

17.2.1 Joining a Team Design Session

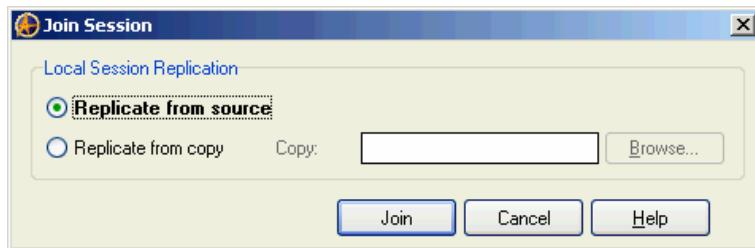
When you are invited to a Team Design session, you must join the session to participate. If you have been invited to a Team Design session, it will be listed in the **Sessions Explorer** in the Home window Sessions tab.



Sessions to which you have been invited are also listed in the **Join Session** dialog box, which is accessible from the **Team Design** menu in any other workspace, including the Repository, Team Manager and Message Center.

To join a Team Design session:

- 1 From the Home window click the **Sessions** tab;
- 2 Right-click the session in the Sessions Explorer and select **Join Session** from the pop-up menu; or double-click the session in the Sessions Explorer. The **Join Session** dialog box opens. All current Team Design sessions that have been published to you appear in the list.
- 3 Select the session that you want to join.
- 4 In the **Local Session Replication** area, select **Replicate from source** or **Replicate from copy**.
 - **Replicate from source**: When the session starts, a copy of the session leader's data is transferred to your computer.
 - **Replicate from copy**: To choose this option, you must have already copied the session leader's data to one of your repositories.



- 5 Click **Join**. The **Joining Team Design Session** dialog box appears with the message: **Waiting for confirmation**. This box will remain open until the leader accepts you as an editor or viewer, or your application is rejected. If you are accepted, the session will load. If you are rejected, you will receive a notification.

Note: You can also access the **Join Session** dialog box by selecting **Join Session** from the **Team Design** menu in the Repository, Team Manager, Message Center or any workspace. Or, in a workspace you can select the **Join Team Design Session**  tool from the Team Design toolbar.

17.2.2 Leaving a Team Design Session

As a participant, you can leave a Team Design session at any time. You are not responsible for saving changes to the model or drawing. When you leave, the session will continue until the leader ends it.

To leave a Team Design session:

- From the **Team Design** menu select **Leave Session**; or select the **Leave Team Design Session**  tool from the Team Design toolbar.

17.3 Scheduled Team Sessions

Scheduling a team session allows the participants to plan time for the session and, when given permission, preview the data. Additionally, participants can pre-cache the data for a faster start. Sessions can only be scheduled from the **Home window**.

17.3.1 Scheduling a Team Session

To schedule a team session:

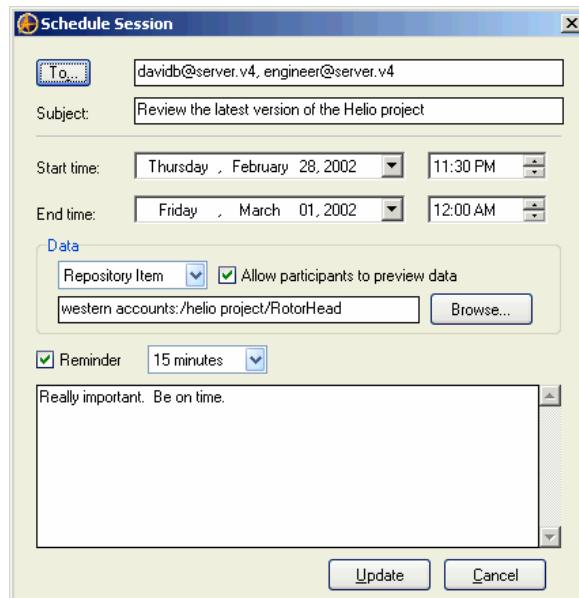
- In the Session tab in select the **Schedule Session** button.

Or from the **Actions** menu, select **Schedule Session**.

Or right-click the **Scheduled Sessions** entry in the Sessions explorer and select **Schedule Session** from the pop-up menu.

Or right-click the Contacts area and select **Schedule Session** from the pop-up menu. You can also right-click a contact to pre-populate the **To** field with that user.

The Schedule Session window appears.



- 2 Click the **To** button to invite users. The **Recipients** dialog box appears. Add users or teams and select **OK**.
- 3 Specify the **Start time**.
- 4 Specify the **End time**.
- 5 From the **Data** menu, select from the following choices:
 - **No Data:** the session will start without data, in the Chat Window. Data can be added later.

- **New Part/Assembly/Drawing:** the session will start in an empty workspace of the selected type.
 - **Repository Item:** the session will start with the design selected through the Browse button.
- 6 If **Repository Item** is selected, check **Allow participants to preview data** to give Read Only access to invitees to whom you have not previously shared the data.
- 7 If Repository Item is selected, click **Browse** to select the design. The **Select Item** dialog box appears. Browse to and select the design and click **OK**.
- 8 Set the session **Reminder** if desired.
- 9 Enter information about the session in the text box.
- 10 Click **Send**. The invitation is sent to invitees.

17.3.2 Accepting and Declining a Scheduled Session

When you are invited to a scheduled session, the session appears in the Sessions Explorer and is embedded in a message sent to your Message Center.

To accept an invitation:

- From the **Actions** menu, select **Scheduled Session > Accept**.
Or right-click the session entry in the Session explorer and select **Accept**. **Accepted** appears next to your user name under the **Participants** node.
Or
- 1 To review a scheduled session before accepting, right-click the entry in the Session explorer and select **Open**.
 - 2 Click **Accept**. **Accepted** appears next to your user name under Participants.

To decline an invitation:

- 1 From the **Actions** menu, select **Scheduled Session > Decline**.
Or right-click the session entry in the Session explorer and select Decline.

Or to review a scheduled session before accepting, right-click the entry in the Session explorer and select **Decline**.

The **Decline Session Invitation** dialog box appears.

- 2 To also delete the session from the Session explorer, select **Decline and delete the session invitation**. The session will be removed from the Session explorer.

Or select **Decline** the invitation for now to keep the invitation in the Session explorer. You can choose to accept the session at another time.

- 3 Click **OK**. A notification is sent to the organizer. If the scheduled session has not been deleted, **Declined** appears next to your user name under the **Participants** node.

17.4 Working in a Team Design Session

Participants in Team Design sessions are granted viewer or editor status by the session leader when they are admitted to the session. To avoid conflict, a participant must be an editor and hold the baton to edit a design. Since the baton holder has the ability to edit the model, the baton cannot be passed to a participant with **View** status—the command is unavailable. If a participant with View status should need to edit the model, the leader can change a participant's status at anytime.

Leader		Crown	Starts the session, invites and admits participants, assigns status and saves work completed. Also, can edit the design while in possession of the baton.
Viewer		Cap	Cannot edit the design, but has other capabilities. See Participant capabilities below.
Editor		Hard Hat	Can edit the design when in possession of the baton. Can also save work.
		Baton	Identifies who has the baton. Only the participant holding the baton can edit the model or drawing.

Participant capabilities

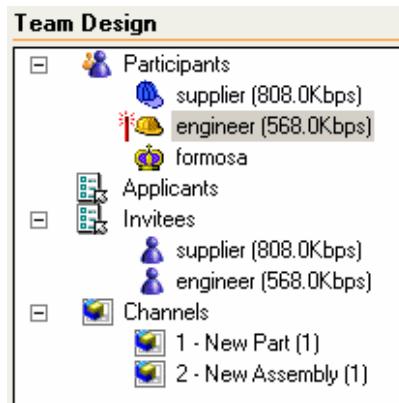
Editors holding the baton have complete control over the design.

Participants who are not holding the baton, regardless of viewer/editor status, can perform the following functions:

- Use the Team Design Explorer to reorient your view of the workspace to that of another participant, follow a participant's view and send a message to a participant.
- Use the commands available in the Team Design menu and toolbar including Follow Baton Holder and Reorient to Baton Holder.
- Independently modify your view of the design with the Rotate and Zoom tools. Zoom tools include: Zoom to Selection, Zoom to Window, and Zoom to Fit.
- Pan your workspace horizontally, vertically, or diagonally across the screen.
- Explode a constrained assembly. (Available only with assemblies.)
- Show/hide the Design Explorer.
- Insert redlines.
- Place and rotate arrows on the model.
- Use the Chat Window and show/hide the Chat Window.

17.4.1 The Team Design Explorer

During a Team Design session, participants are listed in the Team Design Explorer located below the Design Explorer. Icons indicate the leader, the baton holder and a participant's viewer/editor status. When you are leading a session, an Applicants list and Invitees list are also displayed.



Using the Team Design explorer

Right-click Menu Command	Description				
Pass Baton	Pass the baton to the selected participant.	X	X	X	
Send Message	Send a message to the selected participant during a Team Design session. The message will not be recorded in the Chat Window.	X	X	X	
Toggle Status	A Leader Control: toggle the status of a participant between viewer and editor. When that participant has the baton, it is returned to the leader.			X	
Remove	A Leader Control: remove the participant from the Team Design session.			X	
Take Baton	A Leader Control: take immediate control of the baton.			X	
Reorient to Participant	Your view changes to match that of the selected participant at that moment.	X	X	X	
Follow Participant	Your view updates continually so that it is the same as the view seen by the selected participant.	X	X	X	

17.4.2 The Baton

To facilitate Team Design, Alibre Design has implemented the concept of the baton. The baton holder can pass the baton to any editor or the leader at any time.

- Only one participant can hold the baton at a time.
- Only participants with editor status are eligible to hold the baton.
- At any time, the session leader may take the baton without the permission of the baton holder.

Requesting the baton

A baton request alerts the current holder that another participant would like to edit the design. Editors can request the baton at any time. The option to request the baton is not available to viewers.

To request the baton:

- Select **Request Baton** from the Team Design menu; or click the **Request Baton** button in the Chat Window; or select the **Request Baton**  tool on the Team Design toolbar. The baton holder is notified of the request by the **Baton Request** dialog box. The request is also recorded in the Chat Window.

The holder can then choose whether to pass the baton.

Passing the baton

The baton holder may pass the baton to any participant with editor status at any time.

To pass the baton:

- 1 In the Team Design Explorer, right-click the participant to whom you want to pass the baton.
- 2 Select **Pass Baton**. The baton is passed immediately. The participant is notified by the **Baton Received** dialog box, and the event is recorded in the Chat Window.

You may also select the participant in the Team Design Explorer; then click the **Pass Baton**  tool on the Team Design toolbar.

Note: If you have received a formal Baton Request, you may pass that participant the baton by selecting **Pass Baton to <username>@<servername>.com** from the **Team Design** menu.

Taking the Baton

The leader of a Team Design session can take the baton from any participant at any time. It may be necessary to take the baton if the current holder is disconnected from the session, or has been called away and forgot to give it to someone else. The **Take Baton** command is only available to the session leader.

To take the baton:

- 1 From the **Team Design** menu, select **Leader Controls**. The Leader Controls dialog box appears.



- 2 Click the **Participants** tab.
- 3 Click **Take Baton**. The baton is passed to you immediately and the event is recorded in the Chat Window.

Note: You may also right-click the baton holder in the Team Design explorer and select **Take Baton**.

17.4.3 Reorienting to Another Participant's View

When working in a Team Design session, it is useful to view the design from the same orientation as another participant. Participants can choose to view the design from the same

orientation as a specific participant or the baton holder. And, when you have the baton you may reorient all the other participants to your view.

While participating in a Team Design session you may choose to view the design from the same orientation as the baton holder. You can either **Reorient to Baton Holder** or **Follow Baton Holder**, or **Reorient to Participant** or **Follow Participant**.

When you **Reorient to Baton Holder**, or **Reorient to Participant**, your view of the workspace changes to match what is seen by the baton holder, or the selected participant, at that moment. When the baton holder or participant modifies the view, your view will remain the same. To maintain a synchronized view, you must either use the **Reorient to Baton Holder** command repeatedly (or Reorient to Participant) or choose the **Follow Baton Holder** command (or Follow Participant command).

To reorient to the baton holder:

- From the **Team Design** menu, select **Reorient > To Baton Holder**. Your view changes to match that of the baton holder.



Or select the **Reorient to Baton** tool on the Team Design toolbar.

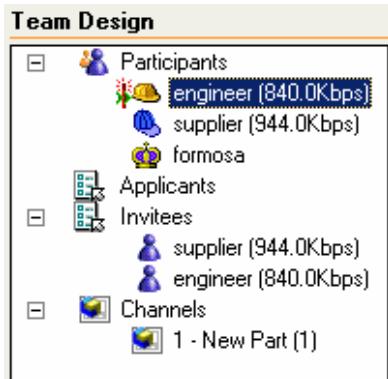
To follow the baton holder:

- From the **Team Design** menu, select **Reorient > Follow Baton Holder**.



Or select the **Follow Baton Holder** tool on the Team Design toolbar.

A green arrow overlaid on top of the baton icon in the Team Design explorer indicates the participant to whom your view is synchronized. Each time the baton holder modifies the view, your orientation changes to match. Once the baton is passed, your view will continue to be synchronized to whomever is holding the baton.



Note: When **Follow the Baton Holder** is active, a checkmark appears in the right-click menu. Clear the checkmark to stop following the baton holder.

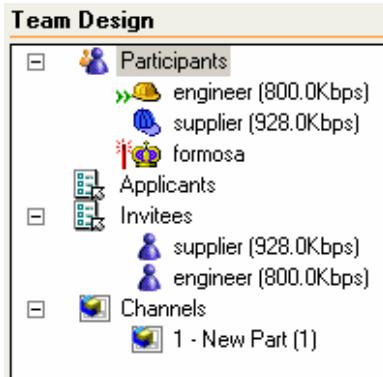
To reorient to a participant:

- 1 In the Team Design Explorer, right-click the participant to whom you want to reorient and select **Reorient To Participant** from the pop-up menu. Your view changes to match that of the selected participant.

To follow a participant:

- 1 In the Team Design Explorer, right-click the participant you want to follow and select **Follow participant** from the pop-up menu. As the participant's orientation changes, your orientation changes to match it.

A green arrow overlaid on top of the baton icon in the Team Design explorer indicates the participant to whom your view is synchronized.

**To reorient all others:**

- From the Team Design menu, select **Reorient > All Others**.



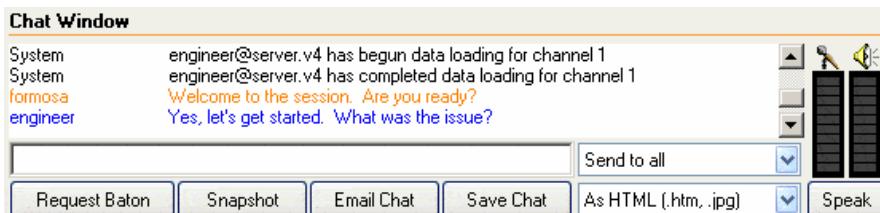
Or select the **Reorient All** icon  on the Team Design toolbar.

All participants' views change to match your current orientation. To lead participants through a series of views, you must reorient them each time you alter the view.

17.4.4 The Chat Window

Chat sessions are available when you work online. Each Team Design session includes a Chat Window, which records both system messages and participant messages. System messages are automatic and record baton events, participant administration events and note each change made to the design.

When working with multiple designs in a Team Design session, participant messages are visible in all channels, while system messages only appear in the channel in which they are invoked.



To show or hide the Chat Window in a Team Design session, select **Chat Window** from the **Team Design** menu to toggle the display. A checkmark indicates that the Chat Window is visible.

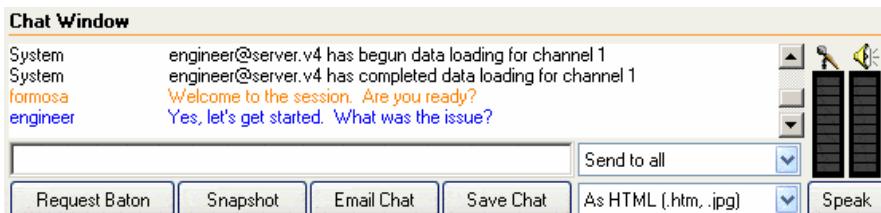
Additionally, you can

- Take periodic snapshots of ongoing work for documentation purposes.
- Save the chat session as a text file or HTML file. Store the file in a repository with other design documentation or email the chat to associates who may want a record of the session.
- Resize the Chat Window.
- Send a voice message.
- Request the baton, if you are an editor.

Note: Participant messages are color-coded. As participants join, each is assigned a unique message color.

Saving a Chat Session

You can save the chat history of a team session as a text file or HTML file to review at a later date and/or forward to the other participants. The file can be stored in the repository with other design documentation.



To save a chat session:

- 1 Select the format for the file from the drop down menu next to the **Save Chat** button.
- 2 Click **Save Chat**.
- 3 Choose a location, and enter a file name.

-
- 4 Click **Save**.

To send a private message:

When working in a team session, you may send a private message to another participant.

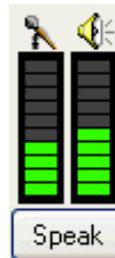
- 1 Right-click a participant in the Team Design explorer and select **Send Message**.
Or select **Send Message** from the Team Design menu; or click the **Send Message**  tool on the Team Design toolbar. The **New Message** dialog box appears. The message is already addressed to the selected participant. You may add additional recipients.
- 2 Type or record a message.
- 3 Click **Send**. Messages sent to participants will be delivered to their Message Center.

17.4.5 Voice Chat

When suitably equipped with speakers and microphones, users can speak to each other through the Chat Window. The meter on the right of the Chat Window shows the volume of the microphone and the speakers.

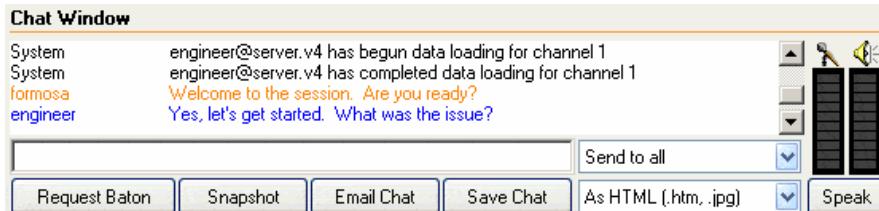
The meter labeled with a microphone represents the volume of your voice when speaking into the microphone. The meter labeled with a speaker represents the volume of incoming communication.

Green color bars indicate an acceptable level.
Yellow indicates an inadequate volume.
Red indicates a volume that has reached an unacceptable level and is distorting.



To voice chat with team members:

- 1 Click and hold the **Speak** button or press the **F12** key.
- 2 Speak into the microphone. All session participants will hear.



- When done speaking, release the Speak button.

Note: When the Chat Window is hidden, you can still use voice chat by pressing **F12**.

17.4.6 Redlines

Redlines let you point out or draw attention to specific aspects of a design. Redlines can also be used to document suggested design changes; saving a design as a new version will capture any redlines and can subsequently be reviewed later if necessary.

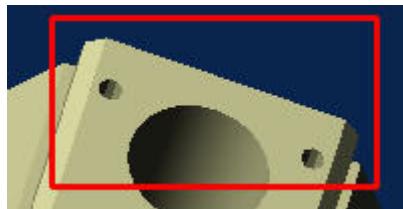
Redlines are associated with orientations and are only visible when the associated orientation is displayed. Redlines cannot be moved or edited. Any participant can insert redlines, but redlines can only be deleted by the baton holder.

Use redlines to:

- Draw a shape (circle, rectangle or freehand) around a specific area of the design.
- Insert a note or comment.

To insert a redline:

- From the **Insert** menu, select **Redline > Freehand, Ellipse, or Rectangle**; or select the **Freehand, Ellipse, or Rectangle** tool from the Redline toolbar.
- Drag a shape around the area you want to discuss or highlight. Or for a note, enter text in the **Note** dialog box. Click in the workspace to set the leader; then drag to place the note.

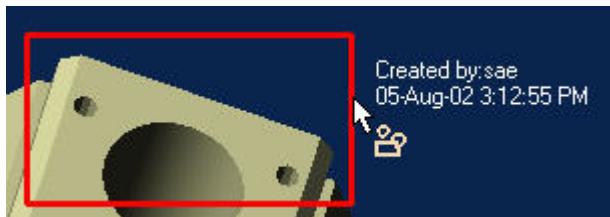
**To hide or show redlines:**

- From the **View** menu, select **Redlines**. If checked, redlines are visible.

Note: To see a specific redline, make sure redlines are not hidden; then double-click the entry in the Design Explorer. This will reorient the view so that the redline is visible.

To display the redline author and date created:

- Drag the pointer over the redline. The author's name and date created appears.

**To pick a different color for subsequent redlines:**

- From the **Insert** menu, select **Redline > Color**; or select the **Redline Color**  tool from the Redline toolbar. The **Color** dialog box appears.
- Select a new color.
- Click **OK**.

To set the redline width:

- 1 From the **Insert** menu, select **Redline > Width**; or select the **Redline Width** tool from the Redline toolbar. The **Width** dialog box appears.
- 2 The default width is **9**. Enter a new width (1 is the lowest allowable setting).
- 3 Click **OK**.

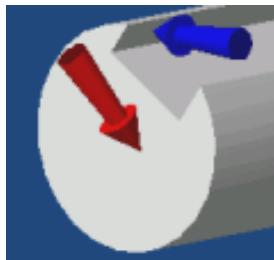


To delete a redline:

- 1 In the Design Explorer, right-click the redline and select **Delete** from the pop-up menu; or right-click the redline in the work area and select **Delete** from the pop-up menu.

17.4.7 Reference Arrows

Each participant in a Team Design session can use one 3D reference arrow in the workspace as a visual aid for communication. You can place, move or rotate your arrow anytime.



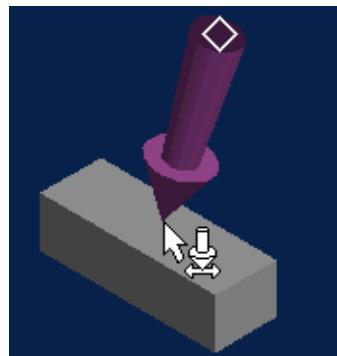
- You can show and hide arrows.
- You can attach a reference arrow to any part of a model or drawing.
- You can move your own arrow; you cannot move other participants' arrows.
- The arrow is only visible during the session; it does not become part of the design.
- The color of your arrow is assigned by Alibre Design to ensure that each participant's arrow is unique.

Two types of pointers are displayed when you are placing and moving reference arrows:



Appears when your pointer is over a location where you can place the arrow.

- Click anywhere on the model or drawing to place the arrow. Or you can drag the arrow to reposition it on the model or drawing.



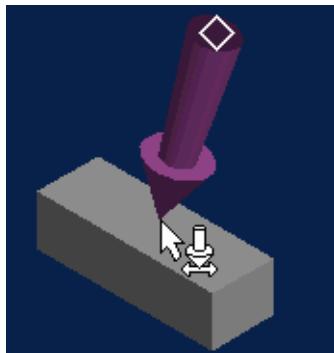
Appears when your pointer is over a location where you cannot place an arrow.



Note: 3D arrows cannot be placed on sketches.

To place your reference arrow:

- 1 From the **Team Design** menu, select **Place Arrow**; or select the **Place Reference Arrow**  tool from the Team Design toolbar.
- 2 Click the model where you want to place the arrow. The arrow appears.



To show or hide reference arrows:

- From the **Team Design** menu select **Show Arrows**. When checked, arrows are visible.

Or select the **Show Reference Arrows**  tool on the Team Design menu.

To move your reference arrow:

- 1 From the **Team Design** menu, select **Place Arrow**; or select the **Place Reference Arrow**  tool from the Team Design toolbar.
- 2 Drag the arrow to where you want to attach it. Or click the model or drawing where you want to attach the arrow. The arrow moves.

To remove reference arrows:

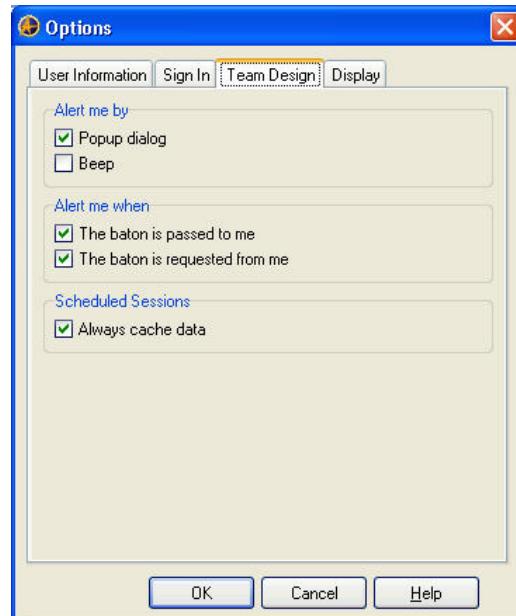
- From the **Team Design** menu, select **Remove Arrow**; or select the **Remove Reference Arrow**  tool from the Team Design toolbar.

Note: You can only remove your own reference arrow.

17.5 Setting Alert Options

You can choose to receive alerts about events occurring during Team Design sessions. You can also select how to receive them.

- 1 In the Home window, from the **Tools** menu select **Options**. The **Options** dialog box appears.
- 2 Click the **Team Design** tab.



- 3 In the **Alert me by** area, select how you want to receive alerts.
 - Popup dialog
 - Beep
- 4 In the **Alert me when** area, select which baton events trigger an alert.
- 5 Click **OK**.

Note: Regardless of your alert settings, you will receive chat messages about baton events

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