

Oracle AutoVue 20.0.1

User's Manual

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Preface

The *Oracle AutoVue User Manual* explores AutoVue's key capabilities and is designed to help you get the most out of AutoVue's extensive functionality.

For the most up-to-date version of this document, go to the AutoVue Documentation Web site on the Oracle Technology Network at <http://www.oracle.com/technetwork/documentation/autovue-091442.html>.

Audience

This manual is intended for Oracle AutoVue end users.

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Related Documents

For more information, see the following documents in the Oracle AutoVue documentation library:

- *Installation and Configuration Guide*
- *Release Notes*
- *Supported Formats List*
- *ActiveX - Feature Matrix*
- *Product Variations - Feature Matrix*
- *Performance Related INI Options*
- *VCET API Manual*
- *Markup Control API Manual*

Conventions

The following text conventions are used in this document:

Convention	Meaning
boldface	Boldface type indicates graphical user interface elements associated with an action, or terms defined in text or glossary.
<i>italic</i>	Italic type indicates book titles, emphasis, or placeholder variables for which you supply particular values.
<code>monospace</code>	Monospace type indicates commands within a paragraph, URLs, code in examples, text that appears on the screen, or text that you enter.

Introduction

Oracle's AutoVue suite of enterprise visualization solutions provides native document viewing, markup and real-time collaboration capabilities that enable Web-based document review on hundreds of native document types, including 2D/3D CAD, EDA, Office and graphic formats. AutoVue enterprise visualization solutions address the challenge of accessing, reviewing and collaborating on documents, both internally and with globally dispersed teams and partners, in a secure and reliable manner without compromising security or precision. Improved team productivity, reduced errors, and accelerated innovation and time to market are just a few of the business benefits organizations can expect to achieve. This manual explores AutoVue's key capabilities and is designed to help you get the most out of AutoVue's extensive functionality.

Oracle AutoVue

Oracle AutoVue is a viewing and markup application that has been developed for both business and technical users. AutoVue is capable of displaying hundreds of file formats without the authoring application. Supported file types include Text, Office, Graphics, EDA, 2D Engineering Drawings, and 3D Models. Even contents of archive files can be viewed in AutoVue.

NOTE: Although not required, we recommend that file names include their standard file extensions. If the file extensions are not included, AutoVue may require additional time to load a file.

Oracle AutoVue is available in various product variations. Refer to the “AutoVue Product Variations – Feature Matrix” chapter at the end of this document to see what features are available for each variation.

Marking Up Documents

AutoVue has the ability to create markups for all its readable file formats without the document's authoring application. AutoVue creates markups for different formats without modifying the original file. You can attach comments, notes, and drawings to any file you display in AutoVue. This is called “marking up a file”, commonly known as annotating or redlining. A markup is an object or entity that you attach to a file. All markups are saved in a separate file called a *Markup* or a *Markup file*. When you display a file with its *markups*, the markups are overlaid as a layer on top of the drawing. AutoVue provides a variety of flexible, user-friendly entities. To name a few, there are circles, clouds, polygons, and leaders. You can also attach text to entities, insert a note for longer comments, add attachments, or add a stamp such as a company logo. In addition, you can create measurement markup entities and hyperlinks that link between the current file and other associated files or applications.

AutoVue Basics

This section introduces the basics of working in AutoVue; such as starting and exiting AutoVue, the help menu, changing the locale of AutoVue, displaying file versions information, and the AutoVue graphical user interface (GUI). The section on the AutoVue GUI discusses the included menu bars, tool bars, navigation panel, and navigation tree.

Starting AutoVue


To start AutoVue, you have one of two options:

TASK

1. From the Windows **Start** menu, select **Programs**, select **AutoVue, Desktop Version**, and then select **AutoVue, Desktop Version**.
2. From the Windows **Start** menu, select **Run**, and then enter the path and filename for *avwin.exe*.
3. Click **OK**.

STEP RESULT: The main AutoVue window appears.



RESULT:

After starting AutoVue, an AutoVue icon  appears in the system tray located on the far right side of the task bar. This icon is always accessible after exiting AutoVue since AutoVue continues running until you close AutoVue completely. Right-click the AutoVue icon to access the system tray Quick Menu. To resume working with AutoVue, double-click the icon or right-click the icon and select Restore AutoVue from the Quick menu.

Exiting AutoVue

To exit AutoVue:

TASK

- From the **File** menu, select **Exit**. The AutoVue icon remains in the system Tray. or
 - Click **Close**  in the upper right corner of the AutoVue window. The AutoVue icon remains in the system Tray. or
 - From the system tray, right-click the **AutoVue** icon  and select **Close AutoVue**. AutoVue closes completely and the icon disappears from the system tray.
-

Help

AutoVue provides a *Help* menu that allows you to easily access information on how to use AutoVue.

The About dialog box, which is accessed by selecting About from the Help menu, provides information about your current installation of AutoVue. Information such as the variation of AutoVue you are using, the version, the build number, and the current language that AutoVue is running is displayed. From the About dialog box, you can also change the localization of the AutoVue user interface.

Changing the Locale of AutoVue

AutoVue supports different *localizations* for the User Interface. AutoVue supports the following languages: English, French, German, traditional and simplified Chinese, Japanese and Korean.

NOTE: *When you change from one language to another, all toolbar customizations are reset to their default settings.*

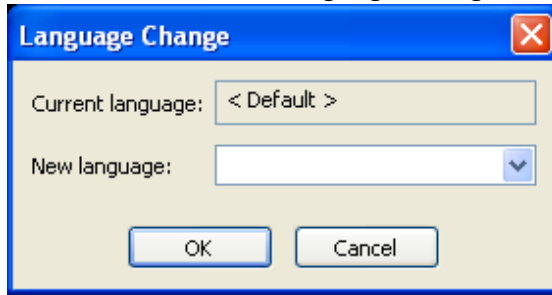
TASK

1. From the **Help** menu, select **About**.

STEP RESULT: The About dialog box appears displaying the language options.

2. Click **Change Language**.

STEP RESULT: The Language Change dialog box appears.



3. From the New Language list, select the language you want.
4. Click **OK**.
5. Click **OK** in the About dialog box.
6. Restart AutoVue for the new localization to take effect.

AutoVue Version Information

The About dialog displays AutoVue *version and build information*. You can also view the version, build number, and build date of components that are shipped with AutoVue. You can also export this information into a text file.

Viewing Version and Build Information

To view product version information:

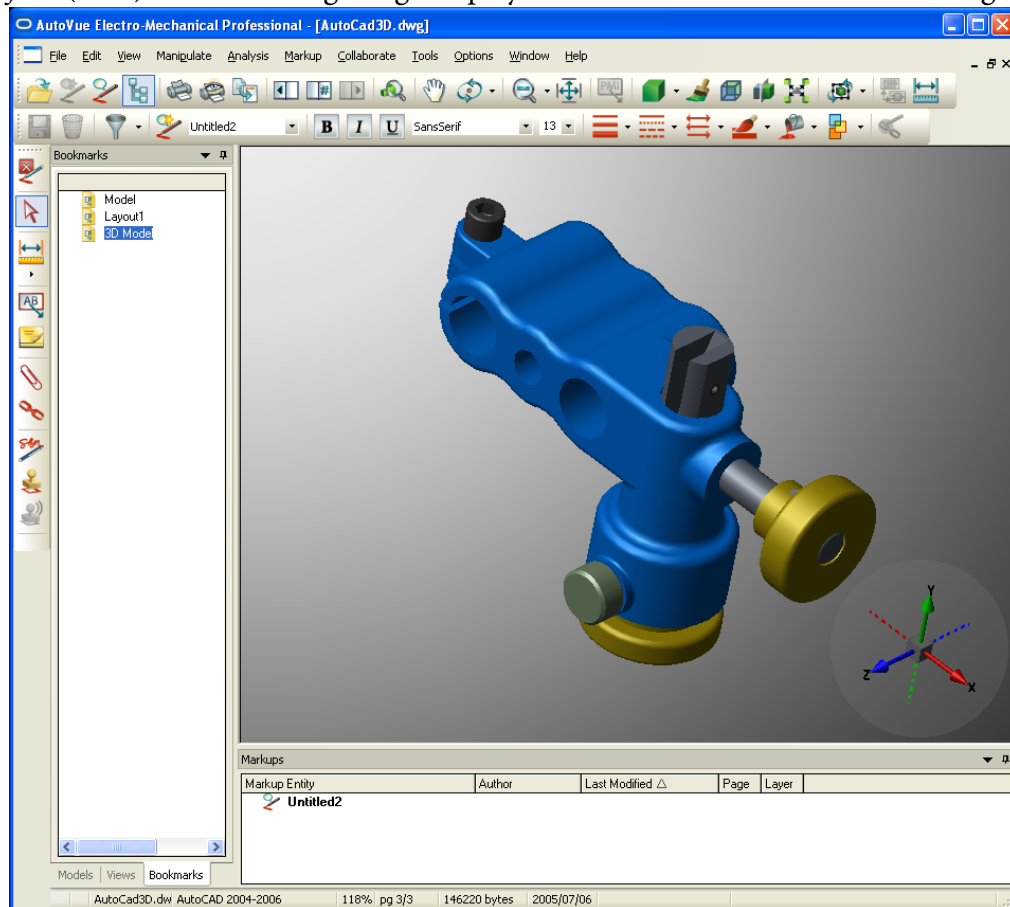
TASK

1. From the **Help** menu, select **About**.
STEP RESULT: The About dialog appears and displays the AutoVue version number and build date.
2. To view version and build information for components, click **Version Info**.
3. To export the version information click **Export**.
STEP RESULT: The Export dialog appears.
4. Navigate and select the directory to which you want to export the list.
5. Enter a file name (the default name is fverinfo.txt).

6. Click **Save**.
STEP RESULT: The list is exported to the specified file.
 7. When you are finished viewing the file version information, click **Close**.
 8. Click **OK** to exit the About dialog.
-

AutoVue Graphical User Interface

This section introduces you to the basics of working with AutoVue's *graphical user interface* (GUI). The following image displays the AutoVue GUI for a 3D drawing:



Menu Bar

The *menu bar* is the main access to all the menu commands. The selection of commands changes according to the tasks being accomplished by AutoVue. In addition, the menu bar

can be configured to include and exclude items via DDE scripting capabilities. For more information, refer to the AutoVue API Manual.

Toolbars

AutoVue has three *toolbars*: AutoVue toolbar, Markup Properties toolbar, and Markup Entity toolbar. The toolbars are fully adaptable to individual work preferences and can be easily customized, moved or removed. The toolbar buttons offer easy access to the options presented in the menus found in the menu bar.

Adding buttons to a toolbar

To add buttons to a toolbar:

TASK

1. Right-click anywhere on the toolbar and select **Customize Toolbar**.
STEP RESULT: The Customize dialog box appears.
 2. Select a menu option from the Categories list and a corresponding button from the Commands list.
STEP RESULT: A brief description of the button is provided in the Description field.
 3. Drag and drop the required buttons from the Commands list to the toolbar.
 4. Click **Close** to exit the Customize dialog box.
ADDITIONAL INFORMATION: To reset the toolbar to its default setup, right-click anywhere on the toolbar and select Reset All Toolbars.
-

AutoVue Toolbar

The *AutoVue toolbar* displays below the menu bar when you open AutoVue. It is the default toolbar and includes the most commonly used functions when viewing a file—open a file, create a new markup, print, zoom, and many others. Depending on the file that is opened, the toolbar buttons will change. For example, the following image shows the AutoVue toolbar when a 3D file is open:



Markup Properties Toolbar

The *Markup Properties toolbar* displays below the AutoVue toolbar when you enter Markup mode. It includes the available property and formatting options for the markup

entities—save markups, change font, fill type, line style, and many others. The following image shows the default Markup Properties toolbar:



Markup Entity Toolbar

The *Markup Entity toolbar* displays by default on the left side of the AutoVue window when you enter Markup mode. It includes all the available markup entities for the opened file. Depending on the file that is opened, the toolbar buttons will change. For example, the following image shows the Markup Entity toolbar when a 3D file is open:



Navigation Panel

The *Navigation Panel* displays by default on the left-hand side of the AutoVue workspace when you view a drawing. For example, when viewing an EDA drawing, it allows you to navigate through a list of component instances, nets, and the associated pins and net nodes (pins connected to a net) present in the current schematic drawing or Printed Circuit Board (PCB) design.

NOTE: To hide or display the Navigation Panel, click .

The columns displayed in the Navigation Panel are determined by the profile of the entity types in the current schematic drawing or PCB design. Lists can be sorted in order to group similar component instances.

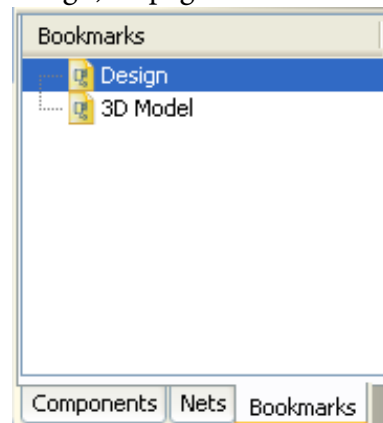
You can also use the Navigation Panel to select (highlight) single or multiple components or entities; zoom to a component or entity, and query entity information.

Components		
RefDes	Part Type	Location
U4	BQFPC6...	(29.000)
C1	CAPAE1...	(51.000)
C2	CAPC321...	(7.0000)
C3	CAPC321...	(7.0000)
C4	CAPC321...	(7.0000)
U1	SOP63P...	(50.000)
U2	SOP63P...	(50.000)
U3	SSOP50...	(50.000)
U5	SSOP50...	(50.000)
R4	RESC16...	(7.0000)
R3	RESC16...	(7.0000)
R2	RESC16...	(7.0000)
R1	RESC16...	(7.0000)
RN1	RESCAX...	(9.0000)
RN2	RESCAX...	(9.0000)
RN3	RESCAX...	(9.0000)
RN4	RESCAX...	(9.0000)
J1	MOL-541...	(30.000)
S1	SW_SNA...	(11.000)
S2	SW_SNA...	(49.000)
J2	USBAF	(23.000)
DISPLAY1	7SEGSM	(29.000)

Component Pins		
Name	Location	With
U5.1	(47.1500...	false
U5.2	(47.1500...	false
U5.3	(47.1500...	false
U5.4	(47.1500...	false
U5.5	(47.1500...	false
U5.6	(47.1500...	false

Bookmarks Tab

The *Bookmarks* tab lists links to specific views (Draft views, 2D plans, 3D Views of an EDA design) or pages or bookmarks structure that is saved in files such as PDF.



Navigate between the pages or files or views by clicking the appropriate link. Bookmarks lead to various views of CAD files such as AutoCAD file Model Space, the 3D model of a file and associated 2D engineering drafts.

If a plus sign appears to the left of a bookmark, click it to expand and view the lower bookmark levels. If a minus sign appears to the left of the bookmark, click it to collapse the lower bookmark levels.

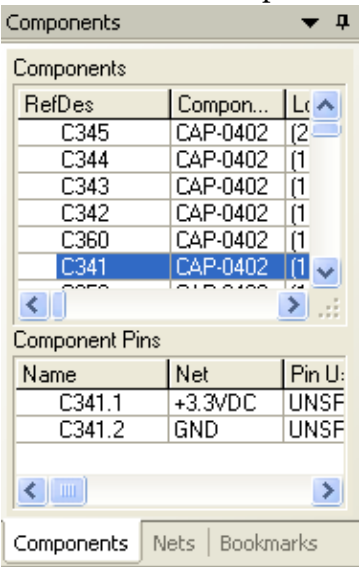
To go to a destination specified by a bookmark, click the bookmark text or the page icon located to the left of the bookmark text.

Components Tab

NOTE: The Components tab displays only for the EDA pages of a drawing.

The *Components tab* lists component instances and the associated pins. The top portion of the tab lists all the instances of the currently displayed page of the drawing. The lower portion of the tab lists the associated pins for selected instances.

When you select a component, it appears highlighted on the drawing. To select more than one component, press the **Shift** or **Control** key while selecting. All the selected components and their associated pins are highlighted.

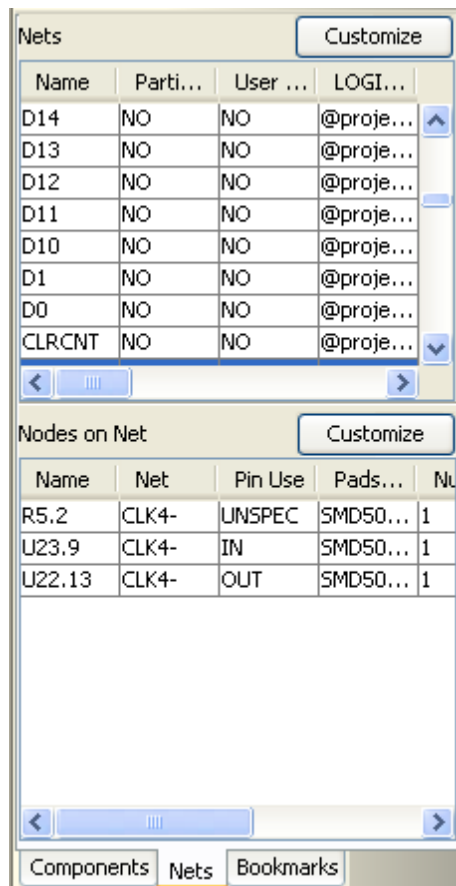


Nets Tab

NOTE: The Nets tab displays only for EDA drawings.

The *Nets tab* lists nets and associated net nodes (pins connected to a net). The top portion of the tab lists all the nets of the drawing's currently displayed page. The lower portion of the tab lists the associated net nodes for the selected nets.

All selected nets are highlighted on the drawing. To select more than one net, press the **Shift** or **Control** key while selecting. Alternately, in the Nets panel, you can click and drag in the Navigation Panel list to select multiple nets. The Nodes on Net panel lists the associated nodes for the selected nets.

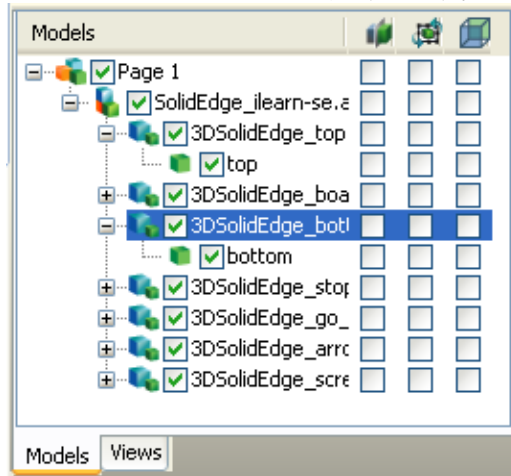


Models Tab and Model Tree

The *Models tab* displays the *Model Tree*. The tree displays the model's hierarchy, inter-relation of different parts, assemblies, and bodies. The tree also indicates if a required external reference (XRef) is missing. With the tree, you can select different parts and modify their attributes such as color, visibility, render mode or transformation.

See "Displaying XRefs" for more information on XRefs.

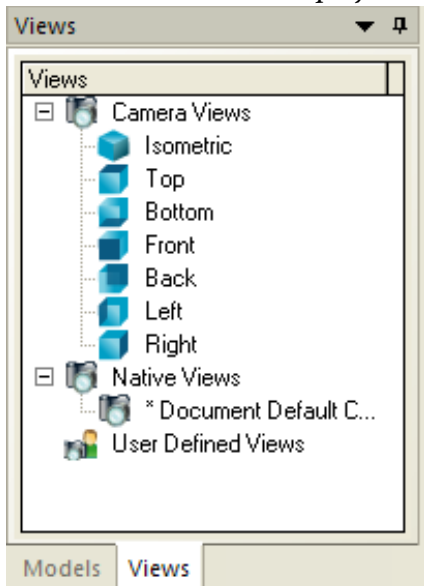
NOTE: The Models tab displays only for 3D drawings.



Views Tab









The *Views tab* lists the standard, native, and user-defined views. You can switch to a standard, native, or user-defined view, as well as add or delete user-defined views.

NOTE: The Views tab displays only for 3D drawings.



Markup Navigation Tree

When you are in Markup mode, a **Markup Navigation Tree** appears below the workspace. If the tree does not appear, from the **Options** menu, select **Show Panel**, and then select **Markup Panel**.

Markups					
Markup Entity	Author	Last Modified 	Page	Layer	
 Untitled2					
 Text	JohnDoe	04/09/2008 11:51:06 AM	1	0	
 Leader	JohnDoe	04/09/2008 11:51:08 AM	1	0	
 bnb	JohnDoe	04/09/2008 11:51:22 AM	1	0	
 Highlight	JohnDoe	04/09/2008 11:51:24 AM	1	0	
 Box	JohnDoe	04/09/2008 11:51:29 AM	1	0	
 Line	JohnDoe	04/09/2008 11:51:31 AM	1	0	

The tree displays a hierarchy of markups or comments created by users. You can navigate through the markups. A set of properties is generated for each markup. You can sort the markups in the tree according to each property by clicking the column headers. These properties are:

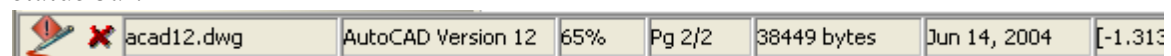
Property	Description
Markup Entity	Type of markup entity created.
Author	The name of the user who created the markup entity.
Last Modified	The date and time the markup entity was last modified.
Page	Page number of the original document on which the markup entity is created.
Layer	Markup layer on which the markup entity is created.




When a markup entity is created, it appears in the tree and the information is recorded and saved in the Markup file.


NOTE: Hover your mouse over an entity to display its author and date properties.


Status Bar


When in View mode or Markup mode, the *status bar* is located at the bottom of the main window. The fields displayed from left to right on the status bar are: marker, current active filename, file type, zoom factor, current page and total number of pages, current active file size, file creation date, and cursor's coordinate position. The following image shows the status bar:



There are three markers that may be present on the status bar: the *Markup Indicator* icon  indicates associated markups, the *Missing Resource* icon  indicate missing resources, and the *Substituted Resource* icon  indicates substituted resources.

The *Markup Indicator*  indicates that the current active file has associated markups. While in View mode, click the Markup Indicator icon to view the Markup Files dialog, then select a Markup file or group of Markup files to open.

The *Missing Resource icon*  indicates that some main resources required to properly display the current active file are not available. To identify the missing resources, click the Resource icon to display the Properties dialog.

The *Substituted Resource icon*  indicates that if a required resource is missing, AutoVue has replaced it with another resource. To identify the substituted resources, click the Substituted Resource icon to display the Properties dialog.

Quick Menus

One of the quickest ways to access options is through *Quick Menus* or context-sensitive shortcut menus. These are the menus you see when you right-click in the workspace, Markup Navigation Tree, and Model Tree. The available Quick Menu options depend on the location where you right-click.

Opening Files

You can open base files and Markup files from the File menu and Markup menu, respectively. Additionally, you can browse files in a directory, open files simultaneously, and view the contents of archive files.

Opening a File

TASK

1. From the **File** menu, select **Open**.

ADDITIONAL INFORMATION: From the **AutoVue** toolbar, you can also click **Open** .

STEP RESULT: The Open dialog box appears.

2. Enter a file name or browse to locate the file that you want to open.
3. Click **Open**.

RESULT:

The file appears in the AutoVue workspace.

Opening Multiple Files

By default, AutoVue keeps one file open at a time. If you open a new file, the currently opened file is replaced with the new file. However, it is possible to have *multiple files* open by creating windows within AutoVue.

TASK

1. From the **Window** menu, select **New Window**.

STEP RESULT: A new window appears on top of the opened file.

2. From the **File** menu, select **Open**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Open** .

STEP RESULT: The Open dialog box appears.

3. Enter a file name or browse to locate the file that you want to open.
4. Click **Open**.

RESULT:

The file appears in the new window.

NOTE: You can open as many windows as you like by repeating steps 1 through 4.

Switching to Another Window

From the Window menu, select the file that you want to display from the bottom of the list. The file is displayed and a check mark appears beside the filename in the Window menu list.

Viewing Multiple Files at the Same Time

You can arrange the files so you can view them at the same time.

From the Windows menu, select **Cascade** if you want the files to appear on top of each other, **Tile Horizontally** if you want the files to appear side by side, and **Tile Vertically** if you want the files to appear above each other.

Streaming Files

AutoVue generates streaming files for most supported file formats. When a file is accessed by AutoVue for the first time, a streaming file is generated. The streaming file contains file data and provides the benefit that AutoVue can access the streaming file much faster than it can the native file. As a result, rendering time is significantly faster when loading from the streaming file.

If the native file changes or its external resource files change or if an INI option changes, the streaming file is invalidated. In this case, AutoVue opens the display from the native file and generates a new streaming file.

Browsing Files

When you use the Browse option, all files and folders in the current directory are displayed in the Browse dialog box and the last file opened is highlighted.

TASK

1. From the **File** menu, select **Browse**.
STEP RESULT: The Browse dialog box appears listing the files located in the current active directory.
 2. To change the current active directory, type the full path of the directory in the text field at the top of the Browse dialog box.
STEP RESULT: The list of files contained in the directory appear.
 3. Select the file that you want to view.
STEP RESULT: The file is displayed in the AutoVue workspace.
 4. Continue selecting and viewing files as often as you like since the Browse dialog box floats over the other AutoVue windows.
 5. Click **Close** to exit the Browse dialog box.
-

RESULT:

The last file displayed remains open.

Archive Files

The full *archive file* directory displays in the AutoVue window. It is not necessary to decompress the file. Double-click a file to display it within AutoVue. If you would like to markup an archived file, the file must be made accessible to AutoVue in a decompressed form.

File Properties

From the **File** menu, select **Properties** to open the Properties dialog. The Properties dialog provides information specific to the current active file, such as filename, file size, date of creation, and file type. The file properties that you can view are:

Property Tab	Description
File Properties	Information specific to the current active file, such as filename, file size, date of creation, file type, number of pages, and x, y & z dimensions. It also indicates if the file is loaded from a streaming file.
Resource Information	Resources specific to the current active file, such as text font, shape file, line style, and external reference files.
Native Properties	Custom properties for file types, such as last person who saved the file, signature verification, and author comments.
DMS	If AutoVue is integrated with a backend DMS/PLM/ERP system, a DMS tab appears that lists file attributes retrieved from the backend system.




NOTE: The Properties dialog varies for each file format that is open. As a result, some property tabs may be absent when viewing a file.

Viewing File Properties

To view file properties:

TASK

- From the **File** menu, select **Properties**.
STEP RESULT: The Properties dialog appears.
- To view the file properties, click the **File Properties** tab.
STEP RESULT: The File Properties tab displays the file name, file size, date last modified, file type, and number of pages the document contains.
- To view resource information, click the **Resource Information** tab.
STEP RESULT: All included resources are listed in the Resource Information tab. If a resource that is required to fully display the file is missing, it is listed with the Missing Resource icon

. If a resource is found it appears with the Found Resource icon . If a required resource is missing and AutoVue substitutes it with another resource, the Substituted Resources icon  appears.

4. To view native properties, click the **Native Properties** tab.
 5. Click **OK** to close the Properties dialog.
-

Working With 2D Files

When working with vector and non-vector 2D files in AutoVue, you can instantly change how the active file is displayed in the workspace by choosing from a number of different view options. For example, you can zoom into an area of the drawing, magnify a part of the drawing, and rotate a file's orientation clockwise, counterclockwise, or flip a file's orientation along the horizontal or vertical axis.

When viewing 2D vector files, you have access to entity information, views saved in the drawing, and the ability to “intelligent snap” when performing measurements. All the information is obtained by AutoVue by referencing various sources; either internal files or external reference files that are located outside the file.

In addition to the features mentioned, this chapter provides detailed information on how you can modify AutoVue to suit your needs and preferences through a broad range of configuration options.

Searching Text

You can perform a *text search* on 2D vector and text-based documents.

AutoVue provides search options that you can use to customize a search. These options are as follows:

NOTE: *You cannot perform a text search on raster files.*

Option	Description
Match Whole Word Only	Match a complete word.
Match Case	Search for a word or text string with specific capitalization.
Up	Search backward in the document.
Down	Search forward in the document.

TASK

1. From the **Edit** menu, select **Find Text**.
STEP RESULT: The Find dialog appears.
 2. Enter the word or phrase that you want to find in the **Find What** field.
ADDITIONAL INFORMATION: If you are searching in a vector file, select a text string from the list.
 3. You can refine your search by selecting **Match Whole Word Only** or **Match Case**.
 4. Click **Find Next**.
STEP RESULT: AutoVue highlights the text and zooms into the text area.
 5. Click **Close** to close the Find dialog.
-

Selecting in 2D Files

The *Select* option is available when viewing graphic and non-graphic files.

NOTE: For non-graphic files, you can only select text.

TASK






1. From the **Edit** menu, select **Select**.



2. Make your selection:
 - In a graphic file, click the entity or click and drag the area you want to select.
 - To select a block of text, position the cursor at the beginning of the block, click and drag to highlight.
 3. To copy the selected data, from the **Edit** menu, select **Copy** or press the keyboard shortcut **Control+C**.
STEP RESULT: The data is sent to the clipboard.
 4. To paste the copied data, from the **Edit** menu, select **Paste** or press the keyboard shortcut **Control+V**.
-






2D Viewing Options

From the View menu, you can change how the active file is displayed in the workspace. For example, you can zoom into an area of the drawing, magnify a part of the drawing, and rotate a file's orientation clockwise, counterclockwise, or flip a file's orientation along the horizontal or vertical axis.

You can access these options from the View menu. The options are:

Menu	Sub-Menu	Description
Zoom	Zoom Box	Click and drag to draw a box around an object that you want to enlarge to fill the window. From the Autovue toolbar, you can also click  or right-click in work-space and select from pop-up menu.
	Zoom In	Zoom in by a factor of 2. From the AutoVue toolbar, you can also click  .
	Zoom Out	Zoom out by a factor of 2. From the AutoVue toolbar, you can also click  .
	Zoom Previous	Reverts to the previous zoom level. From the AutoVue toolbar, you can also click  . <i>NOTE: You can also right-click the workspace and select Zoom - Previous.</i>
	Full Resolution	Display the file at full resolution. From the AutoVue toolbar, you can also click  .

Menu	Sub-Menu	Description
Fit	Horizontal	Fit the image horizontally in the active window. The vertical dimensions of the image are zoomed proportionally but may be too large or small for the window. <i>NOTE: You can also right-click the workspace and select Zoom - Page Width.</i>
	Vertical	Fit the image vertically in the active window. The horizontal dimensions of the image are zoomed proportionally but may be too large or small for the window.
	Both	AutoVue finds the best fit for the current file with respect to both its vertical and horizontal dimensions. From the AutoVue toolbar, you can also click  . <i>NOTE: You can also right-click the workspace and select Zoom - Fit.</i>
Magnify Glass		Magnify an area of the file that is specified by the cursor location. See “Using the Magnify Glass” for more information.
Magnify Window		Magnify a selected area of a file and displays it in the Magnify Window. See “Using the Magnify Window” for more information.
Pan and Zoom Window		Display a close-up view of a particular area of a file while maintaining a view of the entire file. From the AutoVue toolbar, you can also click  . <i>NOTE: The Pan and Zoom Window option is disabled for text-based documents and spreadsheets.</i> See “Using the Pan and Zoom Window” for more information.

Menu	Sub-Menu	Description
Pan		Click and drag to move the drawing. To exit right-click. From the AutoVue toolbar, you can also click  .
<i>NOTE: The Rotate option is disabled for archived files, text-based documents and spreadsheets.</i>		
Rotate	Rotate Clockwise	Rotate the file 90 degrees clockwise. From the AutoVue toolbar, you can also click  .
	Rotate Counter Clockwise	Rotate the file 90 degrees counter clockwise. From the AutoVue toolbar, you can also click  .
Flip	Horizontal Axis	Flip the drawing on its horizontal axis. From the AutoVue toolbar, you can also click  .
	Vertical Axis	Flip the drawing on its vertical axis. From the AutoVue toolbar, you can also click  .

Using the Magnify Glass

The Magnify Glass view option displays the magnified area at the cursor location. You can view the details of a selected area of a file while maintaining a full view of the file.

TASK

1. From the **View** menu, select **Magnify Glass**.
2. Move the cursor to the area that you want to magnify.
3. Click and hold the left mouse button.

ADDITIONAL INFORMATION: To view different areas of the file, click and drag the mouse.

STEP RESULT: The magnified area appears in the Magnify Glass.

4. Right-click to exit Magnify Glass view.


Using the Pan and Zoom Window

The *Pan and Zoom Window* view option allows a close-up view of a particular area of a file while maintaining a full view of the file. The Pan and Zoom Window displays a miniature version of the file and has a movable box frame on top of the miniature to indicate the area of the file displayed in the AutoVue workspace.

NOTE: The *Pan and Zoom Window* option is disabled for text-based documents and spreadsheets.

TASK

1. From the **View** menu, select **Pan and Zoom Window**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Pan and Zoom Window** .

STEP RESULT: The Pan and Zoom window appears displaying a full view of the file.

2. From the **Options** menu, select **Dynamic** to view the changes in “real time”.
3. To view a close-up of a specific area of the file in the AutoVue workspace, minimize the frame box by clicking and dragging the frame handles.
4. To view a different area of the file in the AutoVue workspace, click and drag the frame box to the area that you want to view.

ADDITIONAL INFORMATION: If you perform a zoom function in the AutoVue workspace, the area displayed in the workspace is highlighted by the frame box in the Pan and Zoom Window.

5. From the **Pan and Zoom** menu, select **Exit** to close the window.
-

RESULT:

The last view performed remains in the AutoVue workspace.

Using the Magnify Window

The *Magnify Window* option displays the magnified area in a window that updates dynamically. You can click and drag the window to any location on your screen, view the details of a select portion of the displayed file while maintaining the display of the full file.

TASK

1. From the **View** menu, select **Magnify Window**.

STEP RESULT: The Magnify Window appears.

2. Move the cursor to the area that you want to magnify in the current active file.

3. Click once.

STEP RESULT: The area appears magnified in the Magnify Window.

4. Right-click to exit the Magnify Window.
-

Selecting Views

The *Views* option allows you to access different named views of a file.

TASK

1. From the **View** menu, select **Views**.

STEP RESULT: The Select a Named View dialog appears.

2. From the list, select the view that you want to display.
 3. Click **OK**.
-

RESULT:

The selected view is displayed.

NOTE: To return to the default view, select **Default View**.

Specifying a View Point

The *View Point* option allows you to render a drawing from a selected viewpoint.

TASK

1. From the **View** menu, select **View Points**.

STEP RESULT: The View Point dialog appears.

2. Enter the **X**, **Y** and **Z** coordinates for the view point with which you want to render the drawing.
 3. Click **OK**.
-

RESULT:

The drawing is displayed from the selected view point.

Working with 2D Vector Files



In addition to all the features that are available for generic 2D files, AutoVue provides the ability to access entity information, access views saved in the drawing, and the ability to “intelligent snap” when performing measurements.

AutoVue references various sources to obtain all data required to completely and accurately display vector files. These sources can be internal to the file—like layers, blocks, and overlays—or external reference files (XRefs) that are located outside the file.

Manipulating 2D Vector Files

The *Manipulate* menu option allows you to manipulate how the current active file is displayed. For example, you can select which layers, blocks, and external reference files (XRefs) to display.

The Manipulate menu options are:


Menu	Sub-Menu	Description
Visibility Control	Layers	Select and display different layers of a drawing. From the AutoVue toolbar, you can also click  . See “Displaying Layers” for more information.
	Blocks	Select and display a block from a drawing. From the AutoVue toolbar, you can also click  . See “Selecting Blocks” for more information.
	XRefs	Select the XRefs to display in the drawing. See “Displaying XRefs” for more information.
Overlays		Select an overlay to modify. See “Adding an Overlay” for more information.

Displaying Layers

Use this option to configure the layers to display of the current active file.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Select the Layers to Display dialog appears listing the layers and layer visibility for the current active file.


2. To sort the list of layers in the dialog, click **Name** to sort alphabetically or numerically, or click **Status** to sort by visibility.
 3. You can modify the visibility of a layer or layers by doing one of the following:
 - Deselect/select the Status check box beside the layers that you want to hide/display.
 - Click **All On** to display all the layers or click **All Off** to hide all the layers.
 - Select a layer from the dialog and then click **Toggle** to hide/display the layer.
 4. Click **OK** to close the dialog and to apply your changes.
-

Selecting Blocks

Use this option to select a block to display from the current active file.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Blocks**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Blocks** .

STEP RESULT: The Select a Block to Display dialog appears listing the blocks for the current active file.

2. From the list, select the block that you want to display.
3. Click **OK**.

STEP RESULT: The selected block is displayed.

Displaying XRefs

AutoVue references various sources to obtain all the data required to completely and correctly display files. These sources can be internal to the file, like layers and blocks. External reference files (XRefs) are located outside the file.

Use the XRefs option to display the external references of the current active file.



TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **XRefs**.
STEP RESULT: The Select the External References to Display dialog appears listing the external reference files associated with the current active file.
 2. You can modify the visibility of the XRefs by doing one of the following:
 - Deselect/select the Status check box beside the XRef that you want to hide/display.
 - Click **All On** to display all the XRefs or click **All Off** to hide all the XRefs.
 - Select a XRef from the dialog and then click **Toggle** to hide/display the XRef.
 3. Click **OK** to close the Select the External References to Display dialog.
-

RESULT:

The selected XRefs are displayed.






Displaying Details About Resources

Resource information for a file is displayed in the left of the AutoVue status bar. If a file is missing resources the *Missing Resource icon*  appears to the left of the AutoVue status bar. These missing resources can be XRefs, text fonts, linestyles, or shape files. If a missing resource has been substituted for another resource, the *Substituted Resource icon*  appears.

NOTE: *In order to correctly display a file, you must make sure all required resources are provided to AutoVue.*

See “Viewing Missing XRef Notification Icon” for information on missing XRefs.

TASK

1. Click the **Missing Resource icon**  or the **Substituted Resource icon**  on the AutoVue status bar.
ADDITIONAL INFORMATION: From the **File** menu, you can also select **Properties**.
STEP RESULT: The Properties dialog appears.
 2. Click the **Resource Information** tab to display details about missing or substituted resource information required to properly display this file.
ADDITIONAL INFORMATION: A green check mark  indicates the resources that AutoVue is able to access. A red  indicates that resources are not accessible. A yellow exclamation mark  indicates that a missing resource has been substituted for another resource.
 3. Click **OK** to close the Properties dialog.
-

Adding an Overlay

When working with 2D vector files, you can overlay other files over the current active file. You can also adjust an overlay, move an overlay, and scale an overlay by defining the X and Y coordinates and the scaling factor.

***NOTE:** When working with a raster file, it should be used as the base file because raster formats are opaque and would hide files underneath them.*

TASK

1. Open the file that you want to use as the base file for the overlay.
 2. From the **File** menu, select **Import File as Overlay**.
STEP RESULT: The Overlays dialog appears.
 3. Click **Add**.
STEP RESULT: The Please Select an Overlay File dialog appears.
 4. Select a file to overlay and then click **Open**.
 5. To add another file, repeat steps 3 through 4.
 6. When done, click **OK** to close the Overlays dialog.
-

RESULT:

The base file is displayed with the selected overlay files on top of it.

Modifying an Overlay

TASK

1. From the **Manipulate** menu, select **Overlays**.
STEP RESULT: The Modify Overlay dialog appears.
2. Select the overlay that you want to modify.
3. Click the **Action** that you want to apply to the overlay.
4. Click **Move** if you want to move the overlay. Click a point on the base file where you want to set the lower left corner of the overlay. Click another point where you want to set the upper right corner of the overlay.
ADDITIONAL INFORMATION: As you select the point to define the position of the upper right corner, you can resize the destination box.
5. Click **Scale** if you want to resize the overlay. Enter the XOffset and YOffset coordinates and/or the Scale Factor.
ADDITIONAL INFORMATION: XOffset and YOffset are relative to the base drawing and all options are displayed at their current values.

6. Click **Warp** to adjust the overlay. Click a point on the overlay and drag the cursor to where you want the overlay starting point. Click another point and drag the cursor to where you want the overlay to end.

ADDITIONAL INFORMATION: The overlay's size is scaled to accommodate the origin and destination points you defined

7. Click **OK**.

ADDITIONAL INFORMATION: To modify other overlays, repeat steps 2 to 4.

STEP RESULT: The changes are applied to the selected overlay.

Removing an Overlay

TASK

1. From the **File** menu, select **Import File as Overlay**.

STEP RESULT: The Overlays dialog appears.

2. Select the overlay that you want to remove.

3. Click **Remove**.

STEP RESULT: The overlay is removed from the list.

4. Click **OK**.


RESULT:

The overlay is removed from the display.

Analyzing 2D Vector Files

With the *analysis* functionality, you can measure entities, compare two files, or view drawing information. These options are available under the Analysis menu.

The following table lists all the options from the Analysis menu:

Menu	Sub-Menu	Description
Measure		Measure the distance, area, angle, and arc of an entity. From the AutoVue toolbar, you can also click  . See Also "Measuring in 2D Files" for more information.

Menu	Sub-Menu	Description
Compare		Compare two files. See “Comparing 2D Files” for more information.
Show Drawing Information	Select Single Entity	Scale or translate a file in order to compare files accurately. See Also “Viewing Details of a Single Entity” for more information.
	List Tags/Attributes	List block attributes and tags. See Also “Viewing Tags/Attributes” for more information.
	Entity Information	View information of a set of entities. See Also “Viewing Information for a Set of Entities” for more information.

Comparing 2D Files





AutoVue provides the ability to visually *compare* two files and display color-coded comparative data. When you compare two files, AutoVue displays three windows, the first containing the original file, the second containing the file you compared the original against, and the third containing the comparison results.

In the Comparison Result window you can specify whether you want to display only the additions, deletions or unchanged, or any combination of the three. To access these options, right-click in any window and select an option from the pop-up menu.

The comparison results are displayed in different colors to differentiate the results of the file comparison. The comparison options and corresponding colors are:

Option	Color	Description
View Additions	Green	Indicates the something has been added.
View Deletions	Red	Indicates that something has been deleted.
View Unchanged	Blue	Indicates that there is no change.

TASK

1. View the base file in AutoVue.
2. From the **Analysis** menu, select **Compare**.
STEP RESULT: The Open dialog box appears.
3. Enter the file name or browse to select the file that you want to compare with the current active file.
4. Click **OK**.
ADDITIONAL INFORMATION: If you apply a change from the View menu, all three windows display the synchronized change.
STEP RESULT: AutoVue displays three windows, the first displaying the original file, the second displaying the compare file and the third displaying the comparison results.
5. To access the Compare options, right-click in any of the windows.
ADDITIONAL INFORMATION: To maximize a window, click on the  button on the title bar of the window. To minimize a window, click on the  button. To restore the window, click on the  button.
STEP RESULT: A menu appears displaying the Compare options.
6. When comparing AutoCAD files, you can view a file without viewports. To do so, from the View menu, select View Without ViewPorts.
7. To maximize any of the windows, double-click the title bar of the window that you want to maximize. To restore the window, double-click the title bar.
8. To exit **Compare** mode, from the **File** menu, select **Exit Compare**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Exit Compare Mode** .

RESULT:

The original file appears in the workspace.

Drawing Information

The *Drawing Information* option is available with AutoCAD and MicroStation drawings, and is accessed from the Analysis menu. The Drawing Information options available are: Select Single Entity, List Tags/Attributes, and Entity Information.

Viewing Details of a Single Entity

The *Select Single Entity* option allows you to view information of a single entity.

TASK

1. From the **Analysis** menu, select **Show Drawing Information**, and then select **Select Single Entity**.
 2. Click the entity for which you want to view information.
 3. If you did not select an entity, a message appears informing you that no entities were found and prompts you to select again.
 4. The Get Entity Info dialog appears displaying the information for the selected entity. The Extended Data (XData) button appears if additional information exists for that entity. Click XData to view the information.
 5. Click **OK** to close the Get Entity Info dialog.
-

Viewing Tags/Attributes

The *List Tag/Attributes* option allows you to view information on block attributes and tags.

TASK

1. From the **Analysis** menu, select **Show Drawing Information**, and then select **List Tags/Attributes**.
 2. Click in the area of the file for which you want to view information on block attributes and tags.

ADDITIONAL INFORMATION: If there are no entities within the selected box, a message appears informing you that no entities were found and prompts you to select again.

STEP RESULT: The Block Attributes dialog appears displaying the attributes/tags for the selected entity.
 3. Click **OK** to close the Block Attributes dialog.
-

Viewing Information for a Set of Entities

The *Entity Information* option allows you to view information of a set of entities contained in a specific area of a drawing.

TASK

1. From the **Analysis** menu, select **Show Drawing Information**, and then select **Entity Information**.
2. Click and drag to highlight the entities for which you want to view entity information.

ADDITIONAL INFORMATION: If there are no entities within the selected area, a message appears informing you that no entities were found and prompts you to select again. To exit, right-click the mouse anywhere in the workspace.


STEP RESULT: The List Entities dialog appears displaying the information of all the selected entities.

3. Click **OK** to close the List Entities dialog.

Measuring in 2D Files

AutoVue provides the ability to perform *measurements in 2D files*. Measurement options vary between vector and non-vector files.

- For vector files, AutoVue provides the option to “snap” to fixed points on the drawing.
- For non-vector files, the Snapping Modes option is disabled. However, you can free snap to any point on the drawing.

AutoVue provides several measure options that you can choose from. You can access the measure options from **Analysis** menu and selecting **Measure**, or from the AutoVue toolbar, click **Measure** .

The measure options are as follows:





Name	Description
Distance	Measure the distance between two points. See “Distance in non-Vector Files” “Distance in Vector Files”

Name	Description
Area	Measure selected area. See “Area in non-Vector Files” “Area in Vector Files”
Angle	Measure the angle between selected points or between two lines. See “Angle in non-Vector Files” “Angle in Vector Files”
Arc	Measure an arc entity. See “Arc in non-Vector Files” “Arc in Vector Files”

2D Vector Snapping Modes

The available **snapping modes** allow you to click to precise geometrical points on a drawing. For example, if you select **Snap to End-Point** and you move the cursor over an end-point of a line, the end-point will be highlighted by a snap box.

The snapping modes allow you to snap to the mid, center, and end-points of an entity:

Button	Snap to	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near a linear component's end point.
	Mid-point	Geometric snap mode where a snap box appears when moving the cursor near the halfway point of a linear component.
	Center-point	Geometric snap mode where a snap box appears when moving the cursor near the center of an elliptical component.
	Free snap	Allow snapping at any point on the drawing.

NOTE: When selecting a snapping mode you also have the option to select all snapping modes (**All On**) or to turn off snapping modes (**All Off**)

The below table lists snapping locations for certain measurements:


Measurement	Snapping Location
Area	Snaps to a shape on the drawing.

Measurement	Snapping Location
Arc	Snaps to an arc on the drawing.
Angle	Snap to two non-parallel lines.

Distance in non-Vector Files

Use the *Distance* option to measure the distance between two specific points.


TASK

- From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears displaying the measurement options.
- Click the **Distance** tab.
- In the Measure Distance Units list, select a unit of measurement.
ADDITIONAL INFORMATION: If you want to measure the distance along a path, select the Cumulative check box.
- Click a point on the drawing to define the starting point.
- Click another point on the drawing to define the end point.
ADDITIONAL INFORMATION: If you selected **Cumulative**, continue clicking points along the path that you want to measure.
- Right-click to complete the measurement.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
STEP RESULT: The points are joined by a line. The measured distance, Delta-X, and Delta-Y appear in their respective fields in Distance tab.
- Click **Close** to close the Measurement dialog.

Distance in Vector Files

Use the *Distance* option to measure the distance between two specific points.

TASK

- From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears displaying the measurement options.
- Click the **Distance** tab.

3. Select the snapping modes that you want to use for measuring.
 4. To select all snapping modes click **All On**. To clear all snapping modes click **All Off**.
 5. From the Measured Distance Units list, select the units of measurement.
ADDITIONAL INFORMATION: If you want to measure the distance along a path, select Cumulative.
 6. Click a point on the drawing to define the starting point.
 7. Click another point on the drawing to define the end point.
ADDITIONAL INFORMATION: If you selected **Cumulative**, continue clicking points along the path that you want to measure.
 8. Right-click to complete the measurement.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
STEP RESULT: The points are joined by a line. The Measured Distance, Delta-X, and Delta-Y appear in the Measurement dialog.
 9. Click **Close** to close the Measurement dialog.
-

Calibrating Distance

Calibrate the distance measurement.


TASK

1. Measure distance between two points or measure cumulative distance.
 2. From the **Distance** tab, click **Calibrate**.
STEP RESULT: The Distance Calibration dialog appears displaying the measured distance.
 3. From the Units list, select the unit to which you want to calibrate the distance.
 4. Select **Calibrate to** and enter a value if you want to calibrate the distance to a value.
 5. Select **Set Factor** and enter a value if you want to calibrate the distance by a factor.
 6. Click **OK**.
STEP RESULT: The calibration result appears in the Measurement dialog.
 7. Click **Close** to close the Distance tab.
-

Area in non-Vector Files

Use the Area option to measure the area and perimeter of a region.


TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears displaying the measurement options.
 2. Click the **Area** tab.
 3. In the Area Units list, select a unit of measurement in which to measure the area of the region.
 4. In the Perimeter Units list, select a unit of measurement in which to measure the perimeter of the region.
ADDITIONAL INFORMATION: In the Net Area Result group, select **Add** to cumulate a net area result of different areas, select **Subtract** to subtract an area from the net area result, and select **Clear** to clear the Net Area Result field.
 5. Click a point on the drawing to define the starting point.
 6. Continue clicking points on the drawing to define the region you want to measure.
STEP RESULT: Each point is joined by a line. The area and perimeter measurements appear in their respective fields in the Area tab.
 7. Right-click to complete the measurement.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
 8. Click **Close** to close the Measurement dialog.
-

Area in Vector Files

Use the Area option to measure the area and perimeter of a region.

TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
2. Click the **Area** tab.
3. Select **Between Points** if you want to measure the area between points on a drawing.
ADDITIONAL INFORMATION: Snapping modes are displayed.

4. Select the snapping modes that you want to use for measuring: See “2D Vector Snapping Modes” for more information.


Option	Descriptions
All On	Select all snapping modes.
All Off	Clear all snapping modes.

5. Select **Shape** if you want to measure the area of a predefined shape on the drawing.
 6. From the Measured Area Units list, select the unit in which you want to measure the area.
 7. From the Perimeter Units list, select the unit in which you want to measure the perimeter.
 8. To cumulate a Net Area Result of different areas, click **Add**.
 9. To subtract an area from the Net Area Result, click **Subtract**.
ADDITIONAL INFORMATION: Select **Clear** to clear the Net Area Result.
 10. If you selected **Between Points**, click points on the drawing to define the area.
STEP RESULT: Each point is joined by a line. The area and perimeter measurements appear in the Measurement dialog.
 11. Right-click to complete the measurement.
 12. If you selected **Shape**, click the edge of the predefined shape.
STEP RESULT: The shape is highlighted and the area and perimeter measurements appear in the Measurement dialog.
 13. Click **Reset** to take another measurement.
 14. Click **Close** to close the Measurement dialog.
-

Angle in non-Vector Files

Use the Angle option to measure the angle between points on a drawing.

TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears displaying the measurement options.
2. Click the **Angle** tab.
3. Select a unit of measurement from the Units list.

4. Click points on the drawing to define the angle you want to measure.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

STEP RESULT: Angle arms appear with an arc connecting them. The angle measurement appears in the Angle tab.


5. Click **Close** to close the Measurement dialog.
-

Angle in Vector Files

Use the Angle option to measure the angle between points on a drawing.

TASK

1. From the **Analysis** menu, select **Measure**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .

STEP RESULT: The Measurement dialog appears displaying the measurement options.

2. Click the **Angle** tab.
 3. From the Angle Definition section, select one of the following:
 - **From 3 Points:** Measure the angle between three points.
 - **Between 2 Lines:** Measure the angle between two non-parallel lines.
 4. Select the snapping modes that you want to use for measuring.
 5. To select all snapping modes click **All On**, To deselect all snapping modes click **All Off**. See "2D Vector Snapping Modes" for more information.
 6. Select **Between 2 lines** if you want to measure the angle between two lines.
 7. From the Measured Angle Units list, select the unit.
 8. If you selected **From 3 Points**, click three points on the drawing to define the angle.
 9. If you selected **Between 2 Lines**, click two lines on the drawing to define the angle.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

STEP RESULT: Angle arms appear with an arc connecting them. The angle measurement appears in the Measurement dialog.
 10. Click **Close** to close the Measurement dialog.
-

Arc in non-Vector Files

Use the Arc option to define an arc in the drawing and calculate its arc center, radius, diameter, and arc length.

TASK

1. From the **Analysis** menu, select **Measure**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .

STEP RESULT: The Measurement dialog appears displaying the measurement options.

2. Click the **Arc** tab.
3. In the Arc Info section, select a unit of measurement from the Length Units list in which to measure the length of the arc.
4. In the Measured Angle section, select a unit of measurement from the Sweep Units list in which to measure the angle of the arc.
5. Click points on the drawing to define the arc.

ADDITIONAL INFORMATION: To take another measurement click **Reset**.

STEP RESULT: The points are joined by an arc. The measurements for center point coordinates, radius, diameter, arc length, start and end of angle, and sweep appear in the Arc tab.


6. Click **Close** to close the Measurement dialog.
-

Arc in Vector Files

Use the Arc option to define an arc in the drawing and measure its radius, center, arc length, start and end of angle, sweep, and diameter.

TASK

1. From the **Analysis** menu, select **Measure**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .

STEP RESULT: The Measurement dialog appears.

2. Click the **Arc** tab.
3. Select **From 3 Points** if you want to measure the arc between three points. Snapping modes are displayed
4. Click the snapping modes that you want to select as the points for the measurement.

ADDITIONAL INFORMATION: To select all snapping modes click **All On**. To clear all snapping modes click **All Off**. See "2D Vector Snapping Modes" for more information.

5. Select **Arc Entity** if you want to measure a predefined arc.

6. From the Arc Info Unit list, select the unit in which you want to measure the distance.
 7. From the Measure Angle Units list, select the unit in which you want to measure the angle.
 8. If you selected **From 3 Points**, click three points on the drawing to define the arc.
STEP RESULT: The points are joined by an arc. The measurements for center point coordinates, radius, diameter, arc length, start and end of angle and sweep appear in the Measurement dialog.
 9. If you selected **Arc Entity**, click the edge of the arc that you want to measure.
ADDITIONAL INFORMATION: To take another measurement click **Reset**.
STEP RESULT: The arc is highlighted. The measurements for center point coordinates, radius, diameter, arc length, start and end of angle and sweep appear in the Measurement dialog.
 10. Click **Close** to close the Measurement dialog.
-

Calibrating an Arc

TASK

1. Measure an arc in the drawing.
 2. From the **Arc** tab, click **Calibrate**.
STEP RESULT: The Radius Calibration dialog appears displaying the measured distance.
 3. From the Units list, select a unit of measurement to which you want to calibrate the distance.
 4. Click **Calibrate to** and enter a value if you want to calibrate to a value.
 5. Click **Set Factor** and enter a value if you want to calibrate by a factor.
 6. Click **OK**.
STEP RESULT: The calibration results appear in the Arc tab.
 7. Click **Close** to close the Measurement dialog.
-

Working with EDA Files

In addition to all the features that are available for generic 2D files, AutoVue also provides intelligent querying for EDA files; you can create/modify layer sets, generate BOM, verify EDA designs, perform intelligent measurements, analyze nets and components, set enhanced display options and global transparency, and much more. For information on the available features for EDA files, refer to the following sections.

See “Appendix B: Gerber Layer File Format” for information on the Gerber Layer File (GBL) format.

Navigation Panel

The *Navigation Panel* displays by default on the left-hand side of the AutoVue workspace when you view a drawing. For example, when viewing an EDA drawing, it allows you to navigate through a list of component instances, nets, and the associated pins and net nodes (pins connected to a net) present in the current schematic drawing or Printed Circuit Board (PCB) design.

NOTE: To hide or display the Navigation Panel, click .

The columns displayed in the Navigation Panel are determined by the profile of the entity types in the current schematic drawing or PCB design. Lists can be sorted in order to group similar component instances.

You can also use the Navigation Panel to select (highlight) single or multiple components or entities; zoom to a component or entity, and query entity information.

Components		
Components		
RefDes	Part Type	Location
U4	BQFPC6...	(29.000)
C1	CAPAE1...	(51.000)
C2	CAPC321...	(7.0000)
C3	CAPC321...	(7.0000)
C4	CAPC321...	(7.0000)
U1	SOP63P...	(50.000)
U2	SOP63P...	(50.000)
U3	SSOP50...	(50.000)
U5	SSOP50...	(50.000)
R4	RESC16...	(7.0000)
R3	RESC16...	(7.0000)
R2	RESC16...	(7.0000)
R1	RESC16...	(7.0000)
RN1	RESCAX...	(9.0000)
RN2	RESCAX...	(9.0000)
RN3	RESCAX...	(9.0000)
RN4	RESCAX...	(9.0000)
J1	MOL-541...	(30.000)
S1	SW_SNA...	(11.000)
S2	SW_SNA...	(49.000)
J2	USBAF	(23.000)
DISPLAY1	7SEGSM	(29.000)

Component Pins		
Name	Location	With
U5.1	(47.1500...	false
U5.2	(47.1500...	false
U5.3	(47.1500...	false
U5.4	(47.1500...	false
U5.5	(47.1500...	false
U5.6	(47.1500...	false

Customizing

In the Navigation Panel, you can sort a column, change column order, or hide/show a column.

TASK

1. To sort a column, click the column heading.
2. To change the column order or to show/hide a column, right-click a column heading and select **Customize**. The Customize Columns dialog box appears.
3. To show or hide a column, select the check box beside the column or columns you want to show.
4. Deselect the check box beside the column or columns you want to hide.

ADDITIONAL INFORMATION: To show all columns, click Show All. To hide all columns, click Hide All.

5. To change the column order, select the column you want to move, then click **Move Up** to move the column up in the list or click **Move Down** to move the column down in the list.
6. Click OK.

RESULT:

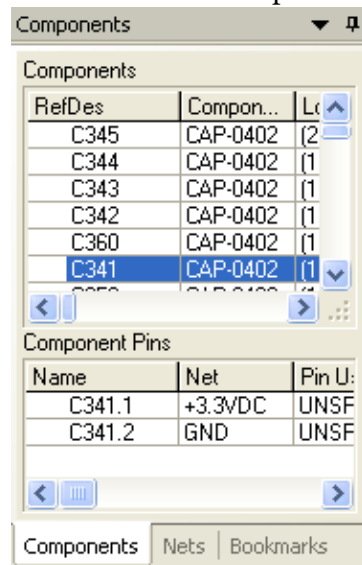
The changes appear in the Navigation Panel.

Components Tab

NOTE: The Components tab displays only for the EDA pages of a drawing.

The *Components tab* lists component instances and the associated pins. The top portion of the tab lists all the instances of the currently displayed page of the drawing. The lower portion of the tab lists the associated pins for selected instances.

When you select a component, it appears highlighted on the drawing. To select more than one component, press the **Shift** or **Control** key while selecting. All the selected components and their associated pins are highlighted.



Nets Tab

NOTE: The Nets tab displays only for EDA drawings.

The *Nets tab* lists nets and associated net nodes (pins connected to a net). The top portion of the tab lists all the nets of the drawing's currently displayed page. The lower portion of the tab lists the associated net nodes for the selected nets.

All selected nets are highlighted on the drawing. To select more than one net, press the **Shift** or **Control** key while selecting. Alternately, in the Nets panel, you can click and drag in the Navigation Panel list to select multiple nets. The Nodes on Net panel lists the associated nodes for the selected nets.

Nets

Customize

Name	Parti...	User ...	LOGI...
D14	NO	NO	@proje...
D13	NO	NO	@proje...
D12	NO	NO	@proje...
D11	NO	NO	@proje...
D10	NO	NO	@proje...
D1	NO	NO	@proje...
D0	NO	NO	@proje...
CLRCNT	NO	NO	@proje...

<

>

Nodes on Net

Customize

Name	Net	Pin Use	Pads...	Nu
R5.2	CLK4-	UNSPEC	SMD50...	1
U23.9	CLK4-	IN	SMD50...	1
U22.13	CLK4-	OUT	SMD50...	1

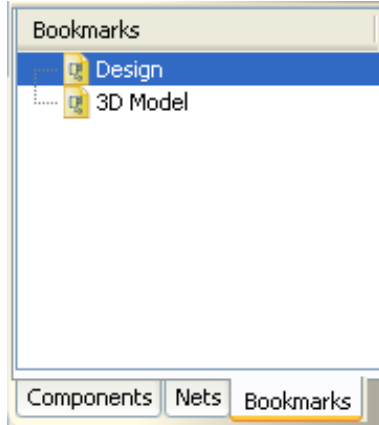
<

>

ComponentsNetsBookmarks

Bookmarks Tab

The *Bookmarks* tab lists links to specific views (Draft views, 2D plans, 3D Views of an EDA design) or pages or bookmarks structure that is saved in files such as PDF.



Navigate between the pages or files or views by clicking the appropriate link. Bookmarks lead to various views of CAD files such as AutoCAD file Model Space, the 3D model of a file and associated 2D engineering drafts.

If a plus sign appears to the left of a bookmark, click it to expand and view the lower bookmark levels. If a minus sign appears to the left of the bookmark, click it to collapse the lower bookmark levels.

To go to a destination specified by a bookmark, click the bookmark text or the page icon located to the left of the bookmark text.

Selecting Entities

Selecting an entity or entities is often the first step to many of the operations that you perform with EDA files. The following sections provide information on how to select entities from the workspace, Navigation Panel, and the Entity Search dialog. To specify which types of entities you can or cannot select, use the Entity Filter dialog.

See “Filtering Entity Types” for more information on the Entity Filter dialog.

Once an entity is selected, you can zoom to it in the workspace and perform other operations with the EDA file.

From the Navigation Panel

To select an entity from the Navigation Panel, perform the following steps:

To select one entity, click the component, net, associated pin, or net node in the Navigation Pane.

To select multiple entities, press the **Shift** or **Control** key while selecting.

The selected entity or entities are highlighted in the workspace. If the selected entity is small, a flash box appears indicating the location of the highlighted entity in the workspace.

See “Zooming to a Selected Entity”

From the Workspace

To select an entity from the workspace, perform the following steps:

Click an entity in the workspace to select it. To select more than one entity, press the **Shift** or **Control** key while selecting.

The selected entity or entities appear highlighted in the workspace and in the Navigation Panel. If the selected entity is small, a flash box appears indicating the location of the highlighted entity in the workspace.

You can also apply selection filters when selecting entities in the workspace.

See “Filtering Entity Types”

When you hover the mouse over an entity in the workspace, a tooltip with information about the entity’s attributes appears. The tooltip displays whether or not you select the entity.

From the Entity Search Dialog

To select an entity from the *Entity Search dialog*, perform the following steps:

TASK

1. Perform an attribute-based or entity type-based search using the Entity Search dialog.
 2. Select an entity or entities from the Entity Types list or the Attributes list of the Entity Search dialog. To select more than one entity, press the **Shift** or **Control** key while selecting.
-

RESULT:

The selected entity or entities appear highlighted in the workspace and in the Navigation Panel.

If the selected entity is too small, a flash box appears indicating the location of the highlighted entity in the workspace.

See “Searching Using Entity Search”

Filtering Entity Types

With the *Entity Filter* option, you can choose to display certain entity types while hiding others. You can also limit the types of entities you can select in the workspace.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Entity Filter**.

STEP RESULT: The Entity Filter dialog appears.

2. Under the Visibility column, perform one of the following:
 - Select the check box beside the entity types you want to display in the workspace.
 - Deselect the check box to hide the entity types.
3. Under the Selection column, perform one of the following:
 - Select the check box beside the entity types that you want to be able to select in the workspace.
 - Deselect the check box beside the entity types that you do not want to select.

ADDITIONAL INFORMATION: To select all entity types, select the column header check box. Deselect the column header check box to deselect all entity types.

4. Click **Apply** to apply the changes.
 5. Close **OK** to close the Entity Filter dialog.
-

RESULT:

Only the selected entity types remain displayed in the workspace.

When you click in the workspace, only the entity types checked in the Entity Filter dialog will be highlighted.

Zooming to a Selected Entity

To *zoom to a selected entity*, perform the following steps:

TASK

1. If you selected an entity from the workspace or Navigation panel, right-click and select **Zoom Selected**.
 2. If you selected an entity from the Entity Search dialog, right-click and select **Zoom Selected**.
-

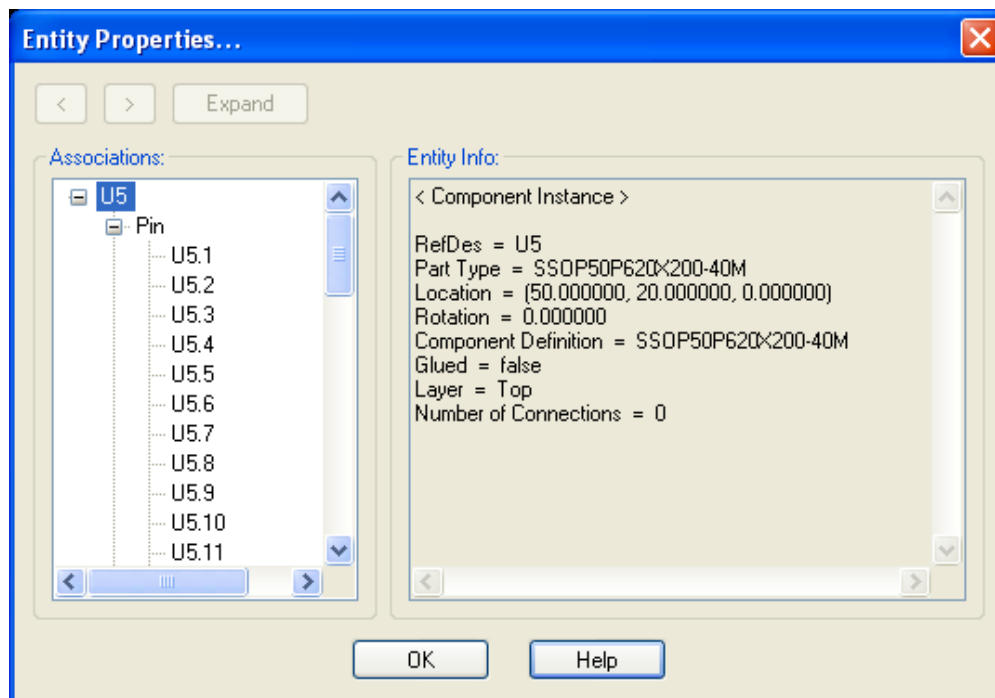
RESULT:

AutoVue zooms to the selected entity in the workspace.

Entity Properties

The *Entity Properties* dialog displays detailed information about any selected entity in the current schematic drawing or PCB design. To view the Entity Properties dialog, first select an entity from the workspace and then from the **Analysis** menu select **Show Entity Properties**.

NOTE: You can also double-click an entity in the workspace, or right-click an entity in the workspace, Navigation Panel, or Entity Search dialog, and then select **Show Entity Properties**.



On the left side of the Entity Properties dialog, a tree displays all the entities associated with the entity you selected. The entity you selected is the root of the tree, and all the associated entities (grouped by their type) are shown as its children.

The attributes of the entity display to the right of the tree, under Entity Info. In the tree, select any associated entity to display its attributes under Entity Info.

You can also view entity attributes in the workspace. When you hover the mouse over an entity in the workspace, a tooltip with commonly used entity information appears. You can turn these tooltips off or on from the Configuration dialog.

See “Configuring AutoVue” for more information on the Configuration dialog.

Viewing the Properties of an Entity

The Entity Properties dialog displays detailed information about any selected entity in the current schematic drawing or PCB design.

TASK



1. Select the entity in the workspace or from the Navigation Panel or Entity Search dialog.

2. Right-click and select **Show Entity Properties**.

ADDITIONAL INFORMATION: You can also double-click the entity in the workspace to view its entity information.

STEP RESULT: The Entity Properties dialog displays the selected entity and its associated entity.

3. Select any associated entity in the tree to display its attributes.
4. To view all associated entities for any given entity in the tree, select the entity and click **Expand**.

ADDITIONAL INFORMATION: To view the entity properties for the previous entity, click the back arrow . To return to the entity properties displayed before you clicked the back arrow, click the forward arrow .

5. Click **OK** to close the Entity Properties dialog.
-

Showing Net Connectivity

You can view the net connectivity of an entity such as a pin, via, or trace with the *Show Net Connectivity* option.

NOTE: *Show Net Connectivity* is disabled when more than one entity is selected.

TASK

1. Select an entity such as a pin, via, or trace from the workspace or the Navigation Panel.

STEP RESULT: The corresponding entity is highlighted in the workspace and in the Navigation Panel.

2. From the **View** menu, select **Show Net Connectivity**.

ADDITIONAL INFORMATION: From the workspace or Navigation Panel, you can also right-click the selected entity, and then select **Show Net Connectivity**.

RESULT:

The graphical entities belonging to the connected nets are highlighted.

Displaying the Entity Properties of a Net

You can view the properties of a net via the *Show Entity Properties* option.

NOTE: The Show Entity Properties is disabled when more than one net entity is selected.

TASK

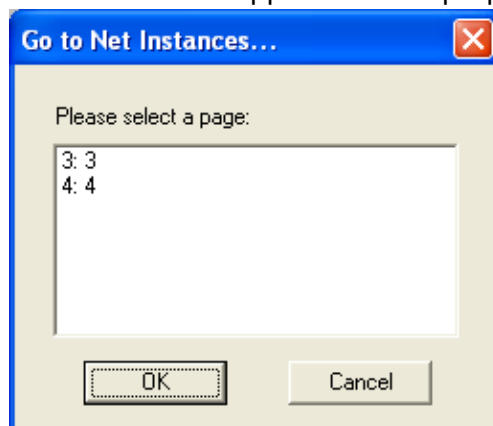
1. Select an entity such as a pin, via, or trace from the workspace or the Navigation Panel.
STEP RESULT: The corresponding net is highlighted in the workspace and in the Navigation Panel.
 2. From the workspace or Navigation Panel, right-click the selected net, and then select **Show Entity Properties**.
STEP RESULT: The Entity Properties dialog appears displaying the properties of the selected net.
 3. Click **OK** to close the Entity Properties dialog.
-

Show Net Instances

You can display the instances of a net in a multi-page file.

TASK

1. From the Navigation Panel, click the **Nets** tab and select the net you want to view.
STEP RESULT: The net is highlighted in the Navigation Panel and in the workspace.
2. From the **View** menu, select **Go to Net Instances**, or right-click the highlighted net from the Navigation Panel or workspace and select **Go to Net Instances**.
STEP RESULT: AutoVue highlights the instance or instances of the selected net. If the selected net appears on multiple pages the Go to Net Instances dialog appears.



3. Select the page on which you want to view the net instance, then click **OK**.
-

RESULT:

The selected page is displayed and the instance of the selected net is highlighted.

Navigating Design Hierarchy

AutoVue supports navigation through the hierarchical structure of a schematic drawing. A hierarchical block in a schematic is a symbol that refers to a child schematic.

Navigating using Descend Hierarchy

With the Descend Hierarchy option, you can navigate to a child schematic.

On the parent page of your schematic's design hierarchy, select any hierarchical block in the workspace or in the Navigation Panel, then right-click and select **Descend Hierarchy**.


AutoVue opens the page with the selected child schematic.

***NOTE:** You can also select the hierarchical block in the Entity Search dialog box, then right-click and select **Descend Hierarchy**.*

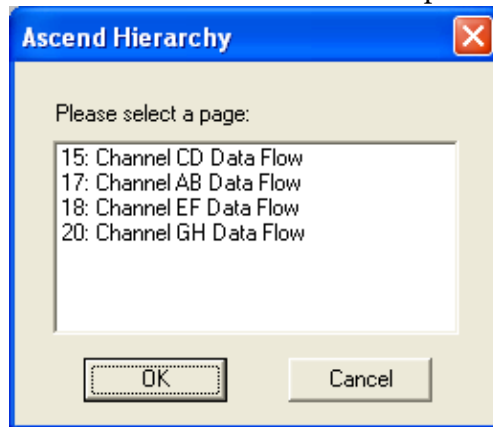
Navigating using Ascend Hierarchy

With the Ascend Hierarchy option, you can navigate from the child schematic to the parent page.

On the child page of your schematic's design hierarchy, select an entity in the workspace or in the Navigation panel and then from the **View** menu select **Ascend Hierarchy**. AutoVue returns to the parent page.


***NOTE:** You can also right-click an entity in the workspace or in the Navigation panel and select **Ascend Hierarchy** .*

If you select an entity that has multiple parents, the Ascend Hierarchy dialog appears displaying the parent pages. Select the Parent page you want to go to, then click **OK**. AutoVue returns to the selected parent page.





NOTE: You can also select the entity in the Entity Search dialog, then right-click and select **Ascend Hierarchy**.

Layers

When working with EDA files, AutoVue lets you view all the physical layers and associated logical layers of the EDA drawing; you can view all layer sets for that file, or create your own. Additionally, from the Layers option, you can manipulate certain layer attributes, such as visibility, color, polarity, and ordering. To display the Layers dialog, from the **Manipulate** menu, select **Visibility Control**, and then select **Layers**. From the AutoVue toolbar, you can also click **Layers** .

NOTE: For drawings that do not contain layers, the Layers menu option and button are disabled.

The Layers dialog has a Physical Layers section and a Logical Layers section. You can display and hide these sections by clicking Expand  and Collapse , respectively.

Physical and Logical Layers Sections

The *Logical Layers* section displays the layers in the order they display in the workspace. The *Physical Layers* section displays the layers in the order they appear in the layer stackup when manufactured.

The *Physical Layers* section displays a matrix with a mapping of physical layer names to entity types. Each row in the matrix corresponds to a physical layer in the board. Each column in the matrix lets you control the visibility of an entity type, such as pin, via, or

trace. The entity types that display depend on the entities available in the open file. Select a physical layer to change its visibility, layer order, and color. Select or deselect the entity type of a physical layer to affect its visibility separately from the other entities of the layer.

When you select a physical layer from the Physical Layers section, the associated logical layers are also selected in the Logical Layers section. You can also select individual logical layers from the Logical Layers section, or press the **Shift** or **Control** key to select multiple logical layers.

NOTE: One physical layer can be selected at a time.

The Logical Layers section displays a list of logical layers and the layer attribute options you can modify: visibility, order, color, and polarity.

To modify the attributes for both physical and logical layers, first select the layers you want to modify, and then use the settings to make the changes you want.

When you select physical layers and modify settings from the Layers dialog, a message displays at the bottom of the dialog to indicate your last action.

When you are satisfied with the changes, click **Apply** to make the changes in the workspace. You can also create layer sets to save your modified layer settings to reuse them later.

When you open a file, it displays all layer sets for that file. You can choose the layer set you need, or create your own.

Changing the Order of Layers

You can change the order in which layers display in the workspace. Specifically, you can change the z-order of layers.




TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers** .

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Layers dialog opens.

2. If the Logical Layers section is not open, click **Expand** .

3. Select the layer or layers that you want to move. To do so, you can do one of the following:
 - Select a physical layer from the Physical Layers section. Logical layers associated with the selected physical layer are also selected. When you make changes to a selected physical layer, the logical layers associated with it are also affected.
 - Select one or multiple logical layers from the Logical Layers section. To select multiple logical layers, press the **Shift** or **Control** key while selecting.
4. Click one of the following buttons:
 - Click **Bring to Front**  to move all selected layers to the front in the workspace. In the Logical Layers section, these layers move to the top of the list.
 - Click **Up**  to move selected layers up one layer.
 - Click **Down**  to move selected layers down one layer.

ADDITIONAL INFORMATION: You can also drag and drop selected logical layers in the Logical Layers section.
5. Click **Apply** to see the changes in the workspace.


STEP RESULT: The file now displays according to the layer scheme you arranged. Also, logical layers are renumbered in the Order column of Logical Layers section.
6. You can now save your changes as a user-defined layer set.
7. Click **Close** to close the Layers dialog.

Modifying Visibility for Physical Layers


Use the Layers dialog to hide or display specific physical layers in the workspace.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Layers dialog opens.
2. In the Physical Layers section, you can show or hide the following:

Option	Description
All physical layers	Click the All column header.
One physical layer	Select the check box to the left of the layer name. You can also select a layer and click Visibility  .
One entity type for all physical layers	Click the column header. For example, click the Trace column header to show or hide all trace entities of all physical layers.

Option	Description
One entity type for one physical layer	Click a specific check box.

- Click **Apply** to view the changes in the workspace.
- You can now save your changes as a user-defined layer set.
- Click **Close** to close the Layers dialog.


ADDITIONAL INFORMATION: Check boxes for physical layers can have four states: checked, unchecked, gray checked, and gray unchecked. Check boxes that are checked and gray indicate that the entities of a physical layer are neither all visible nor all hidden. Gray unchecked check boxes that you cannot select indicate that there is no entity of that type for that layer.

Modifying Visibility for Logical Layers


Use the Layers dialog to hide or display specific logical layers in the workspace.



TASK

- From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Layers dialog opens.

- If the Logical Layers section is not open, click **Expand** .
- In the Logical Layers section, you can show or hide the following:

Option	Description
One logical layer	Select the check box in the Visibility column  .
Multiple logical layers	To select more than one layer, press the Shift or Control key while selecting, and then click Visibility  .

- Click **Apply** to view the changes in the workspace.
 - You can now save your changes as a user-defined layer set.
 - Click **Close** to close the Layers dialog.
-

Modifying Polarity

From the Logical Layers section of the Layers dialog, you can invert the *polarity* of the selected layer (if polarity exists in the selected file).





TASK

1. Select the layer and then click the **Polarity** icon to toggle the polarity, or select/deselect the checkbox in the Polarity column.
 - When positive polarity is selected, the layer appears as is.
 - When negative polarity is selected, the layer image is inverted; the entities in the layer are displayed as transparent and the remainder of the layer appears as a solid color.
-

Changing Layer Color

You can modify the color of any physical or logical layer from the Layers dialog.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .
STEP RESULT: The Layers dialog opens.
2. If necessary, click **Expand**  to open the Logical Layers section.
ADDITIONAL INFORMATION: Select the layer or layers for which you want to change the color. You can select a physical layer from the Physical Layers section; logical layers associated with the selected physical layer are also selected. You can also select individual logical layers from the Logical Layers section, or press the **Shift** or **Control** key while selecting to select multiple layers.
3. After you select a layer, you can do one of the following to modify its color:
 - Click **Modify Color**  and then select a color from the palette. Use this button if you want to change the color of all logical layers associated with a selected physical layer, or to change the color of many selected logical layers at the same time.
 - Double-click a square in the Color column  of the Logical Layers section and then select a color from the palette. Use this option to change the color of individual logical layers.
STEP RESULT: The color of all selected layers changes in the Color column.
4. Click **Apply** to save the changes.
STEP RESULT: The display updates in the workspace based on the changes you made to the layer color.


5. You can now save your changes as a user-defined layer set.
 6. Click **Close** to close the Layers dialog.
-

Sorting Logical Layers


You can sort the list of logical layers in the Logical Layers section by name, visibility, color, or physical layer.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.


ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Layers dialog opens.

2. If the Logical Layers section is not open, click **Expand** .
 3. Click the column header of an attribute.
-

RESULT:

Layers are sorted according to the attribute header you click.

For example, if you click the Color  column header, the logical layers in the section are grouped according to their color.

NOTE: To restore the original sort order, click the **Order** column header.

Layer Sets


A layer set is comprised of all the physical and logical layers in the drawing. Layer sets differ in the attributes of the different layers; for example, z-order, visibility, and color. You can also define your own layer sets to control which layers you can view and print.

Viewing Layer Sets

AutoVue lists top, bottom, and default layer sets, as well as any layer set that is stored in the file. You can choose to display any layer set you need. You can also create user-defined layer sets and display them later. To view a layer set, take the following steps:

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.


ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Layers dialog opens.

2. From the Layer Sets list, select the layer set you want to display.
3. Click **Apply**.

STEP RESULT: The selected layer set displays in the workspace.

4. To restore the default layer set, from the Layer Sets list, select **(Default)**.
5. Click **Close** to close the Layers dialog.


ADDITIONAL INFORMATION: You can also view the layer sets from the Layers list beside the **Layers** button  on the AutoVue toolbar.

Creating User-Defined Layer Sets

You can define a layer set and save it for the duration of the session. The layer set you defined is added to the Layer Sets list in the Layers dialog and to the Layers list on the AutoVue toolbar. By default the Top, Bottom and Default layer sets and any other layer sets belonging to the file are listed in the Layer Sets list.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Layers dialog opens.

2. Click **Add**.

STEP RESULT: The Add Layer Set dialog appears.


3. Enter the layer set name.
4. Click **OK**.

STEP RESULT: The new layer set appears in the Layer Sets list.

5. To modify the attributes of the new layer set, select one or more layers and change the Visibility, Order, or Color.
6. Click **Apply** to save the changes and to display the new layer set in the workspace.

7. To define more layer sets, repeat steps 2 to 6.
 8. Click **Close** to close the Layers dialog.
-


RESULT:

The new layer set appears in the Layers list beside the **Layers Control** button  on the AutoVue toolbar.

Deleting User-Defined Layer Sets

To delete a user-defined layer set, perform the following steps:


TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .
STEP RESULT: The Layers dialog opens.
 2. From the Layer Sets list, select the user-defined layer set you want to delete.
ADDITIONAL INFORMATION: You can only delete user-defined layer sets.
 3. Click **Delete**.
ADDITIONAL INFORMATION: The layer settings of the deleted layer set remain displayed on the screen until you select another layer set.
 4. Click **Apply** to save the changes.
 5. Click **Close** to close the Layers dialog.
-

Saving User-Defined Layers Sets with Markups

You can save user-defined layer sets using Markup files.

TASK

1. Create custom layer sets.
2. From the **Markup** menu, select **New**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Markup** .
3. Create any markup entities that you need.

4. From the **Markup** menu, select **Save As**.
STEP RESULT: The Save Markup File As dialog appears.
5. Enter the markup information and click **OK** to close the Save Markup File As dialog.

RESULT:

The Layer sets you created are saved with the Markup file. When you reopen the file during a different session, you can view the Layer sets when you open the Markup file.

See “Markups”

Manipulating EDA Views

As with any 2D file, with the *View options*, you can instantly manipulate how the current active file is displayed. For example, you can rotate a file’s orientation clockwise or counter-clockwise by 90 degrees, and flip a file’s orientation horizontally, vertically or both simultaneously.

See “2D Viewing Options” for more information on View options.

AutoVue provides several ways to change the view size of a selected area of a file, display different views, layers and blocks of the current active file. You can also navigate from one page to another page of a multi-page file.

3D View


For some ECAD formats, AutoVue supports *3D views* of PCB boards.

TASK

1. In the Navigation Panel, click the **Bookmarks** tab.
2. Select **3D Model** from the navigation tree.

RESULT:

The 3D view of the PCB board is displayed in the workspace.

*NOTE: 3D view can also be accessed from the **View** menu, selecting **Page**, and then selecting **Next**, or from the AutoVue toolbar and then clicking **Next Page** .*

Cross Probing


Cross probing is the ability to select elements in the schematic and have them mapped to the corresponding components in the layout drawings and vice versa.

The following sections provide information on cross probing between two or more EDA files, displaying the net connectivity when cross probing, and cross probing between 2D and 3D views of the same file.

Cross Probing Between Two or More EDA Files



The *Cross Probe* option lets you select entities in the schematic drawing or the PCB design to highlight in all the open files.

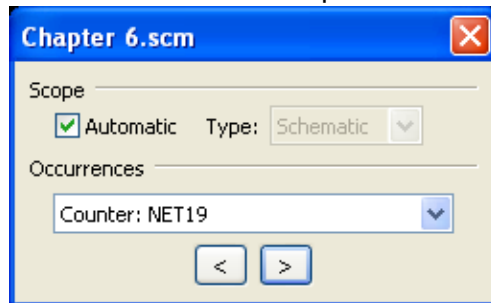
TASK

1. Open an EDA file to cross probe.
2. From the **Analysis** menu, select **Cross Probe**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Cross Probe** .
STEP RESULT: The Cross Probe dialog appears.
3. Click **Add File**.
4. In the Open dialog that appears, enter the file name or browse to locate the file you want to cross probe, then click **Open**.
ADDITIONAL INFORMATION: To add multiple files to the Cross Probe dialog repeat steps 3 and 4. When adding multiple files from the same directory, press **Shift** while making your selection.
5. Click **OK**.
STEP RESULT: Each file you select appears in a new window. A dialog appears for each new window which lets you change the view or page for each file. Each dialog has the Automatic option selected by default. When the Automatic option is selected for each dialog, you see different behavior depending on the files you cross probe.
6. If you cross probe a schematic and PCB, one window automatically displays a 2D PCB and the other window displays a schematic. The same is true if either file contains a PCB and a schematic. Whereas, if files A and B contain both a schematic and a PCB and they are cross probed, one of the files displays the schematic page and the other displays the 2D PCB.

7. If you cross probe two PCBs, one window automatically displays a 2D PCB and the other window displays a 3D PCB.
8. If you cross probe two schematics, both are set to schematic.
9. To change the view of a file in a window, do the following:
 - a Deselect the **Automatic** option in the dialog for the window you want to change.
 - b From the Type list, select **Schematic**, **PCB**, or **PCB 3D View**. These options are available in the Type list only when the views are present in the file.

STEP RESULT: The view of the file changes in the window.

10. To switch between multiple occurrences of a selected component, do the following:
 - a From the Navigation Panel, select a component in one file that has multiple component occurrences in the other file.
 - b From the dialog of the file with multiple component occurrences, select one of the components from the Occurrences list or click **Next**  or **Previous**  to switch between component occurrences.



11. Select any entity in the schematic drawing.

RESULT:


The same entity is highlighted in the PCB design.

Cross Probing Between 2D and 3D Views of the Same File

The *Cross Probe* option lets you select entities to highlight in the 2D and 3D views of the same file.

TASK

1. From the **Analysis** menu, select **Cross Probe**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Cross Probe** .

STEP RESULT: The Cross Probe dialog appears.

2. Click **Add File**.
3. In the Open dialog that appears, select the same file, then click **Open**.

4. In the Cross Probe dialog, click **OK**.
STEP RESULT: The file appears in a new AutoVue window. The views that display in each window depend on the contents of file you are cross probing.
 5. If the file does not contain a schematic, one window shows the 2-dimensional view of the PCB and the other shows the 3-dimensional view of the PCB.
 6. If the file contains a schematic as well as a PCB, the schematic view displays in one window and the 2D PCB view displays in the other window. In this case, take the following steps to see the 2D and 3D views of the PCB:
 - a In the window's dialog, deselect the **Automatic** option.
 - b Select a different view from the TYPE list. For example, you can switch between Schematic and PCB 3D View.
 7. Select an entity in the 3D view.
-

RESULT:

The same entity is highlighted in the 2D view of this file.

NOTE: You can select entities in the 2D view or 3D view of the file. The selected entities are highlighted in both open views of this file.

Showing the Net Connectivity when Cross Probing

To display the *net connectivity* of the design when cross probing, perform the following steps:

TASK

1. Select an entity or entities in any open file.
 2. Right-click and select **Show Net Connectivity**.
-

RESULT:

The net connectivity for the entity or entities that you selected are highlighted in all the files.

Zooming when Cross Probing

When you are cross probing files, you can *zoom* in on the entities you select.

Zoom in on a selected entity or entities:

- When cross probing between a schematic drawing and a PCB design.

- When cross probing between the 2D and 3D views of the same file.

TASK

1. Select an entity or entities in either one of the open files or views.

ADDITIONAL INFORMATION: If you are cross probing between a schematic drawing or a PCB design, you can select entities in either file. You can also select entities in any open view when cross probing the 2D view and 3D view of the same file.

2. Right-click and select **Zoom Selected**.
-

RESULT:

AutoVue zooms to the same entities in all the open file or views.

See Also “Configuring Zoom Behavior when Cross Probing” for more information.

Comparing a PCB with Artwork

You can visually compare PCB designs with artwork. In Compare mode for EDA files, EDA options such as Select Entity, Entity Search dialog and Entity Filter are available.

TASK

1. Open the PCB file that you want to compare with the artwork.

2. From the **Analysis** menu, select **Compare**.

STEP RESULT: The Open dialog appears.

3. Enter the file name or browse to locate the artwork file you want to compare with the PCB file.

4. Click **Open**.

STEP RESULT: The PCB-Artwork Comparison dialog appears.


5. From the box, select the PCB Physical layer that represents the artwork.

ADDITIONAL INFORMATION: For example, **TOP** or **BOTTOM**.

6. Click **OK**.

STEP RESULT: AutoVue displays three windows, the first displaying the original PCB drawing, the second displaying the Artwork drawing, and the third the Comparison Result window.

7. To exit Compare mode, from the **File** menu, select **Exit Compare**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Exit Compare** .

RESULT:

The original file appears in the AutoVue workspace.

Generating Bill of Material (BOM)

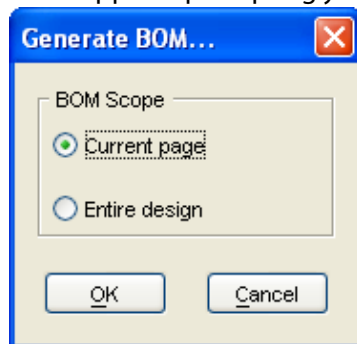
For EDA files, you can obtain a list of the components and parts required for manufacturing the item featured in the schematic drawing or PCB design.

The *Bill of Material (BOM)* report produces a count of the unique components or parts needed for manufacturing. It lists the quantity required, component name, reference designators, value (for resistors and capacitors), size, and descriptions. When determining the uniqueness of a component, the count considers only the attributes chosen for inclusion in the report output. It includes only those attributes that provide the most accurate and unique component quantities.

TASK

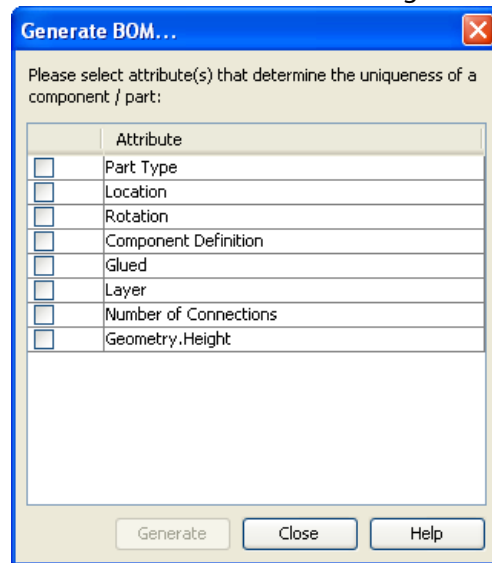
1. From the **Analysis** menu, select **Generate Bill of Material**.

STEP RESULT: If you are generating a BOM for a schematic with multiple pages, a dialog appears prompting you to select the BOM scope: Current page or Entire design.



2. Click **OK** after making your selection.

STEP RESULT: The Generating BOM dialog appears listing the attributes of the file.



3. Select the attributes that you want to appear in the report.
4. Click **Generate**.

ADDITIONAL INFORMATION: To sort a column by alphanumerical or alphabetical order, click on the column heading.

STEP RESULT: The Bill of Material dialog appears listing the count for each component or part possessing the attributes you selected. There are also columns displaying the values of the attributes you selected.

5. To save the BOM, click **Export**.
6. Specify the directory where you want to store the file, the file name and extension, then click Save.

ADDITIONAL INFORMATION: You can specify either .pdx (Product Data Exchange) or .csv (Comma Separated Values) for the extension.

STEP RESULT: AutoVue automatically saves the results in a file with the specified extension.

7. Click **Close** to close the Bill of Material dialog.

Design Verification

Design verifications are operations that check the electrical design of a PCB against a set of rules defining physical and electrical design constraints. Performing these checks prevent short circuits and process errors. The types of design rules are as follows:

Design Rule Type	Description
Clearances	The minimum spacing (air gaps) allowed between two sets of entities in the design. These sets can belong to the same type of entity or to different types.
Electrical	Targets certain properties of the nets.
Manufacturing	Targets the physical properties of a particular entity.

Design Rule Checks

For design rule checks that have values, you can set the global unit of measurement. The available design rule checks are:

Type	#	Design Rule Check	Description
Clearance	1	Minimum Clearance (Pad > Pad)	Specify the minimum distance between the pad outline and the outline of its drill hole on all physical layers. <i>NOTE: The two outlines (geometries) are part of the pin (pad) entity but on different logical layers.</i>
	2	Minimum Clearance (Pad > Trace)	Specify the minimum pad to trace clearance and pads on the same physical layer. <i>NOTE: Only the pad outline is considered. Does not take into account traces connected to a pin (pad) on the same net.</i>

Type	#	Design Rule Check	Description
	3	Minimum Clearance (Trace > Trace)	Specify the minimum trace to trace clearance on the same physical layer. <i>NOTE: Does not take into account directly connected traces.</i>
	4	Minimum Clearance (Component > Component)	Specify the minimum component to component clearance on the same physical layer (component top side or component bottom side). <i>NOTE: Only the logical layer that defines the actual outline of the component is considered.</i> <i>A component entity is usually made up of geometries on different logical layers (silkscreen, keepout, and so on.), but only the logical layer that defines the actual outline of the component is considered.</i>
Manufacturing	5	Minimum Annular Ring	Specify the minimum distance between the pad outline and the outline of its drill hole on all physical layers. <i>NOTE: The two outlines (geometries) are part of the pin (pad) entity but on different logical layers.</i>
	6	Minimum Pad Diameter	Specify the minimum diameter of a pad. <i>NOTE: Only applicable to rounded pads.</i>

Type	#	Design Rule Check	Description
	8	Acute Angle (in degrees)	Specify the minimum acute angle between connected traces on the same physical layer. <i>NOTE: Only two traces belonging to the same net are considered.</i>
	9	Minimum Drill Hole Size	Specify the minimum diameter of the drill hole size allowed. <i>NOTE: Only applicable to rounded pads.</i>
	10	Maximum Drill Hole Size	Specify the maximum diameter of the drill hole size allowed. <i>NOTE: Only applicable to rounded pads.</i>
Electrical	7	Maximum Via Count	Specify the maximum via count allowed for a net.
	11	Minimum Route Width	Specify the minimum route (trace) width allowed in the design.
	12	Maximum Route Width	Specify the maximum route (trace) width allowed in the design.
	13	Minimum Route Length	Specify the minimum route (trace) length allowed in the design.
	14	Maximum Route Length	Specify the maximum route (trace) length allowed in the design.
	15	Empty Nets	Select this check box if you want to check for any nets that are not connected to any pin, via, trace and power/ground plane.

Type	#	Design Rule Check	Description
	16	Un-routed Traces	Select this check box if you want to check for any trace segment that is not connected to another trace segment, pin, via on either end.
	17	Single Connection	Select this check box if you want to check for a net connected to only one pin. <i>NOTE: Power and ground nets are the exception.</i>
	18	Short Circuit	Select this check box if you want to check for intersections (on the same layer) of traces that belong to different nets. <i>NOTE: Trace entities will be divided into n sets where n is the number of nets in the design.</i>
	19	Unconnected Pin	Select this check box if you want to check for any pin (pad) that is not graphically intersected with any traces on the same physical layer.

Verifying a Design

The following steps describe how to verify a design.

TASK

- From the **Analysis** menu, select **Verify Design**.
STEP RESULT: The Verify Design dialog appears.
- To enable a design rule, select the associated check box in the **Design Rules** section of the dialog.

3. To disable a design rule, deselect the check box.

ADDITIONAL INFORMATION: You can sort the design rules by clicking the column headings for **Enabled**, **Description**, or **Value**.

4. To add a value to the selected design rule, double-click the corresponding row of the Value column and enter a value.

ADDITIONAL INFORMATION: The selected design rule must be enabled to be able to add a value.

5. In the Maximum Violations field, enter the maximum number of results you want to display in the Results list.
6. From the Distance Units list, select the unit you want to use as the unit of measure.
7. Click **Verify**.

ADDITIONAL INFORMATION: The Verify button changes to Stop. To stop the Design Verification process at any point, click **Stop**.

When the process is complete, the total number of errors found during the Design Verification process, up to the maximum number specified in the Maximum Violations field appear in the Results list.

8. To view the description of a violation result, select the violation from the Results list.

STEP RESULT: The description appears in the Description field. The description includes information such as the type of violation, the location where it occurs (x- and y-coordinates), the component or entity it affects, and the actual value measured.

When you select a violation result, AutoVue zooms to the entity or set of entities that were affected and highlights them on the drawing.

9. Continue to select violation results to view its description.

ADDITIONAL INFORMATION: Click **Reset** to restart a new check.

10. Click **Close** to close the Verify Design dialog.
-

Exporting the Design Verification Results

You can export the design verification results into a text file.

TASK

1. Verify the design.
2. In the Verify Design dialog, click **Export**.

STEP RESULT: The Export Results dialog appears.

3. Navigate and select the directory where you want to export the results.
4. Enter a file name.

5. Click **Save**.


STEP RESULT: AutoVue saves the design verification results in a .txt (text) or a .csv (Comma Separated Values) file listing each violation result and its description.

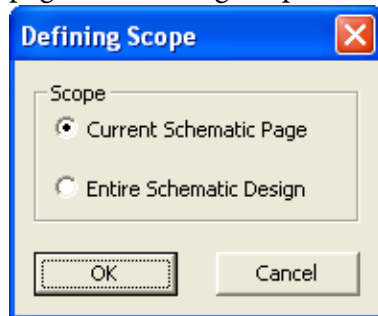
6. Click **Close** to close the Verify Design dialog.
-

Searching Using Entity Search

You can search for an entity in a schematic drawing or a PCB design using the *Entity Search* option. To filter entities, apply attribute, or entity type filters as search criteria. You can select entities from the results list to highlight them in the workspace and the Navigation Panel. You can also export the search results to a .csv file.

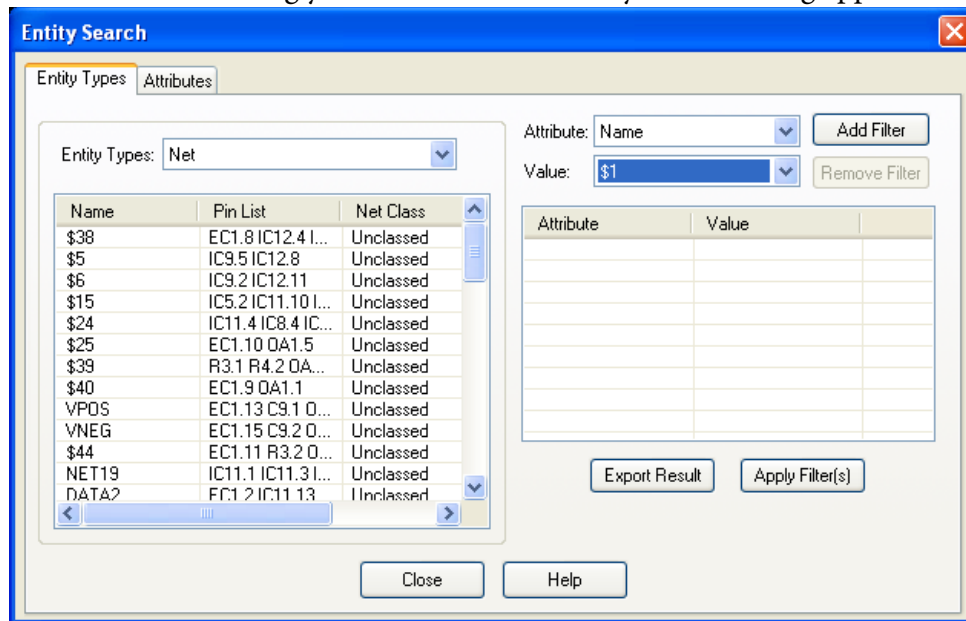
NOTE: If the selected entity is small, a flash box appears indicating the location of the highlighted entity in the workspace.

To open the Entity Search dialog, from the **Edit** menu, select **Entity Search**. From the AutoVue toolbar, you can also click **Entity Search** . If your schematic has multiple pages, a Defining Scope dialog appears.



Select Current Schematic Page to browse through only one page. Select Entire Schematic Design to browse through all schematic pages.

Click OK after making your selection. The Entity Search dialog appears.




Performing an Entity Type-based Search

The following steps describe how to perform an *entity type-based search*.

TASK

1. From the **Edit** menu, select **Entity Search**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Entity Search** .

STEP RESULT: The Entity Search dialog appears.

2. Click the **Entity Types** tab.
3. From the Entity Types list, select an entity type.

ADDITIONAL INFORMATION: To sort the list by attribute, click the attribute column heading that you want to sort by.

STEP RESULT: A list of attributes for the selected entity type appear.

4. If you want to search by attribute, select an attribute from the Attribute list.
5. If you want to search by value, select a value from the Value list.
6. Click **Add Filter**.

STEP RESULT: The selected attribute and its corresponding value appear in the list.

7. Repeat steps 4 and 5 until you have all the filters you want.

8. Click **Apply Filter(s)**.

ADDITIONAL INFORMATION: To remove a filter, select the line in the list containing the attribute and value you do not want to use as a filter, then click **Remove Filter**.

STEP RESULT: A list of entities matching the search criteria appear showing the column headers for the entity's available attributes.

9. To save the results, click **Export Result**.

STEP RESULT: The Save dialog appears.

10. Specify the file name and the directory where you want to store the file, then click **Save**.

STEP RESULT: AutoVue saves the results in a .csv (Comma Separated Values) file.

11. To view the properties of an entity, select the entity from the Entity Types list or the Attributes list, then right-click and select **Show Entity Properties**.

STEP RESULT: The Entities Properties dialog appears and displays the properties for the selected entity.

12. To show the net connectivity, click the **Attributes** tab, select the entity from the list, then right-click and select **Show Net Connectivity**.

STEP RESULT: The net connectivity for the selected entity is highlighted in the workspace.


13. Click **Close** to close the Entity Search dialog.

Performing an Attribute-based Search

The following steps describe how to perform an *attribute-based search*.

TASK

1. From the **Edit** menu, select **Entity Search**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Entity Search** .

STEP RESULT: The Entity Search dialog appears.

2. Click the **Attributes** tab.

3. From the Attributes list, select an attribute.

ADDITIONAL INFORMATION: To sort the list by Owner, Type, or Value, click the column heading that you want to sort by.

STEP RESULT: The Owner, Type, and Value of the selected attribute appear in the list.

4. If you want to search by owner, select an owner from the Owner Filter list.

5. If you want to search by value, select a value from the Value Filter list.

ADDITIONAL INFORMATION: You can also enter an Owner Filter or Value Filter in the text box.

6. Click **Apply Filter(s)**.

ADDITIONAL INFORMATION: Select an item in the result list and right-click to access options like Zoom Selected and Show Entity Properties.

STEP RESULT: A list of entities matching the search criteria appear displaying the Owner, Type, and Value columns.

7. To save the results, click **Export Result**.

STEP RESULT: The Save As dialog appears.

8. Specify the file name and the directory where you want to store the file, then click **Save**.


ADDITIONAL INFORMATION: AutoVue saves the results in a .csv (Comma Separated Values) file.

9. Click **Close** to close the Entity Search dialog.

Measuring in EDA Files

In EDA files, you can take measurements of distances, areas, arcs, and angles. When measuring, you have the option to “snap” to geometrical or electrical points on the drawing.

From the **Analysis** menu, select **Measure** to access the Measurement options.

NOTE: From the AutoVue toolbar, you can also click **Measure** .










The following table outlines the available measuring options:

Name	Description
Angle	Measure the angle between selected points.
Arc	Measure an arc entity.
Area	Measure a selected area.
Distance	Measure the distance between two points.
Minimum Distance	Measure the minimum distance between two nets. <i>NOTE: This option is only available for PCB files.</i>

EDA Snapping Modes

The *snapping modes* allow you to click to precise geometrical or electrical points. For example, when you select **Snap to Pin**, move the cursor over the pin you want to select until the pin is highlighted, then click. Highlight and click a second pin to measure the distance between them.


The snapping modes allow you to snap to the middle, center, and end-points of an entity, as well as a pin, via, and symbol. The following table outlines the available snapping modes:

Button	Snap To	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near the entity's end point.
	Mid-point	Geometric snap mode where a snap box appears when moving the cursor near the halfway point of a linear entity.
	Center-point	Geometric snap mode where a snap box appears when moving the cursor near the center of an entity.
	Pin	Electrical snap mode where a snap box appears when the cursor touches a pin.
	Via	Electrical snap mode where a snap box appears when the cursor touches a via.
	Symbol origin	Electrical snap mode where a snap box appears when the cursor touches a component.
	Free snap	Allows snapping at any point on the drawing.
	Net	Allows snapping to a net.
	Trace	Allows snapping to a trace.

Measuring Distance

Use the Distance option to measure the distance between two specific points.

TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
 2. Click the **Distance** tab.
 3. Select the snapping modes that you want to use for measuring.
 - To select all snapping modes click **All On**.
 - To deselect all snapping modes click **All Off**.
See “EDA Snapping Modes”
 4. From the Measured Distance Units list, select the unit in which you want to measure the distance.
ADDITIONAL INFORMATION: If you want to measure the distance along a path, select **Cumulative**.
 5. Click a point on the drawing to define the starting point.
 6. Click another point on the drawing to define the end point.
ADDITIONAL INFORMATION: If you selected Cumulative, continue clicking points along the path that you want to measure.
 7. Right-click to complete the measurement.
ADDITIONAL INFORMATION: To make another measurement, click **Reset**.
STEP RESULT: The points are joined by a line. The measured distance, Delta-X, Delta-Y and the “Manhattan Distance” appear in the Measurement dialog.
 8. Click **Close** to close the Measurement dialog.
-

Calibrating Distance

Calibrate the distance measurement.

TASK





1. Measure distance between two points or measure cumulative distance.
2. From the **Distance** tab, click **Calibrate**.
STEP RESULT: The Distance Calibration dialog appears displaying the measured distance
3. From the Units list, select the unit to which you want to calibrate the distance.

4. Select **Calibrate to** and enter a value if you want to calibrate the distance to a value.
5. Select **Set Factor** and enter a value if you want to calibrate the distance by a factor.
6. Click **OK**.
STEP RESULT: The calibration result appears in the Measurement dialog.
7. Click **Close** to close the Distance tab.

Measuring Minimum Distance

Use the Minimum Distance option to measure the minimum distance between two nets.

TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
2. Click the **Min. Distance** tab.
3. To measure the minimum distance between entities that are not on the same physical layer, select **Across Layers**.
ADDITIONAL INFORMATION: If a physical layer does not exist, this option determines whether or not the minimum distance should be calculated across logical layers.
4. Select **First Set**  to select the entities that you want to measure from.
5. Select the snapping modes that you want to select as the entities for the measurement.
ADDITIONAL INFORMATION: If you click **Net** , you cannot select any other type of entity.
See “EDA Snapping Modes”
6. Click the first set of entities on the drawing.
ADDITIONAL INFORMATION: To clear the last set of entities you selected, click **Clear Set**.
STEP RESULT: The entities are highlighted.
7. Select **Second Set**  to select the entities that you want to measure to.
8. Click the second set of entities on the drawing.
STEP RESULT: The entities are highlighted in a different color.
9. From the Measured Min. Distance Units list, select the unit in which you want to measure the distance.
10. Select **Zoom to Result**, if you want to zoom into the measurement on the drawing.

11. Click **Compute**.

ADDITIONAL INFORMATION: AutoVue does not measure the distance between layers (z-dimension).

STEP RESULT: The minimum distance from the first set of entities to the second set is represented by a line. The minimum measured distance, Delta X, Delta Y, and the Manhattan Distance appear in the Measure Min Distance section of the dialog.

12. Click **Close** to close the Measurement dialog.
-

Measuring Area

Use the Area option to measure the area and perimeter of a region.

TASK

1. From the **Analysis** menu, select **Measure**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .

STEP RESULT: The Measurement dialog appears.

2. Click the **Area** tab.
3. Select **Between Points** if you want to measure the area between points on a drawing. The snapping modes are displayed.
4. Select the snapping modes that you want to use for measuring.
5. Select **Shape** if you want to measure the area of a predefined shape on the drawing; the snapping modes are disabled.
6. From the Measured Area Units list, select the unit in which you want to measure the area.
7. From the Perimeter Units list, select the unit in which you want to measure the perimeter.
8. In the Net Area Result section of the dialog, select **Add** to cumulate a net area result of different areas.
9. To subtract an area from the Net Area Result, select **Subtract**.
10. Select **Clear** to clear the Net Area Result.
11. If you selected **Between Points**, click points on the drawing to define the area.

STEP RESULT: Each point is joined by a line. The area and perimeter measurements appear in the Measurement dialog.

12. If you selected **Shape**, click the edge of the predefined shape that you want to measure.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

STEP RESULT: The shape is highlighted. The area and perimeter measurements appear in the Measurement dialog.


13. Click **Close** to close the Measurement dialog.

Measuring an Angle

Use the Angle option to measure the angle between points on a drawing.

TASK

1. From the **Analysis** menu, select **Measure**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .

STEP RESULT: The Measurement dialog appears.

2. Click the **Angle** tab.
3. From the Angle Definition section, select one of the following:
 - **From 3 Points:** Measure the angle between three points.
 - **Between 2 Lines:** Measure the angle between two non-parallel lines.
4. Select the snapping modes that you want to use for measuring.
 - To select all snapping modes click **All On**.
 - To deselect all snapping modes click **All Off**.

See "EDA Snapping Modes"
5. Select **Between 2 Lines** if you want to measure the angle between two lines.
6. From the Measured Angle Units list, select the unit in which you want to measure the angle.
7. If you selected **From 3 Points**, click three points to define the angle.
8. If you selected **Between 2 Lines**, click two lines to define the angle.

ADDITIONAL INFORMATION: To take another measurement click **Reset**.

STEP RESULT: Angle arms appear with an arc connecting them. The angle measurement appears in the Measurement dialog.


9. Click **Close** to close the Measurement dialog.

Measuring an Arc

Use the Arc option to define or select an arc in the drawing and measure its arc length, start and end of angle, sweep, radius, center and diameter.

TASK

1. From the **Analysis** menu, select **Measure**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .

STEP RESULT: The Measurement dialog appears.

2. Click the **Arc** tab.
3. Select **From 3 Points** if you want to measure the arc between three points. Snapping modes are displayed.
4. Select the snapping modes that you want to use for measuring.
 - To select all snapping modes click **All On**.
 - To deselect all snapping modes click **All Off**.See "EDA Snapping Modes"
5. Select **Arc Entity** if you want to measure a predefined arc.
6. From the Arc Information section, select the unit in which you want to measure the distance of the arc.
7. From the Measured Angle Units list, select the unit in which you want to measure the angle.
8. If you selected **From 3 Points**, click three points to define the arc.

STEP RESULT: The points are joined by an arc. The measurements for center point coordinates, radius, diameter, arc length, start and end of angle and sweep appear in the Measurement dialog.

9. If you selected **Arc Entity**, click the edge of the arc that you want to measure.

ADDITIONAL INFORMATION: To take another measurement click **Reset**.

STEP RESULT: The arc is highlighted. The measurements for center point coordinates, radius, diameter, arc length, start and end of angle and sweep appear in the Measurement dialog.

10. Click **Close** to close the Measurement dialog.
-

Calibrating an Arc

TASK

1. Measure an arc in the drawing.
2. From the **Arc** tab, click **Calibrate**.

STEP RESULT: The Radius Calibration dialog appears displaying the measured distance.

3. From the Units list, select a unit of measurement to which you want to calibrate the distance.
 4. Click **Calibrate to** and enter a value if you want to calibrate to a value.
 5. Click **Set Factor** and enter a value if you want to calibrate by a factor.
 6. Click **OK**.
STEP RESULT: The calibration results appear in the Arc tab.
 7. Click **Close** to close the Measurement dialog.
-

Working with 3D Files

From AutoVue's 3D mode, you have many ways in which to manipulate the model. For example, you can select model parts to transform independently from the rest of the model, as well as modify the selected model part's visibility and attributes. 3D mode also gives you the ability to perform an entity search and compute mass properties, and it allows you to import 3D models into the current window and export files to other formats. Additionally, the Global Axes inform you of where the X, Y and Z axes are positioned throughout all the operations you perform in AutoVue.

The *Perspective* option shows objects in three dimension with distances, planes and curved surfaces adjusted to give a sense of depth. You have lighting options where you can adjust the overall lighting surrounding a model or just its source light.

You can also customize operations to suit your needs. For example, you can create and save your own views. Define cross sections and cut-throughs of 3D models. Create a customized three-axis coordinate system that you can set as the active coordinate system for your 3D files.

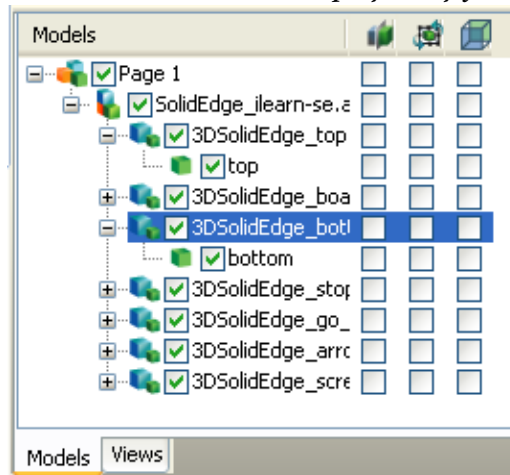
There are also features like Bill of Material (BOM), Interference Checking and Product and Manufacturing Information (PMI) Filtering.

Models Tab and Model Tree

The *Models tab* displays the *Model Tree*. The tree displays the model's hierarchy, inter-relation of different parts, assemblies, and bodies. The tree also indicates if a required external reference (XRef) is missing. With the tree, you can select different parts and modify their attributes such as color, visibility, render mode or transformation.

See "Displaying XRefs" for more information on XRefs.

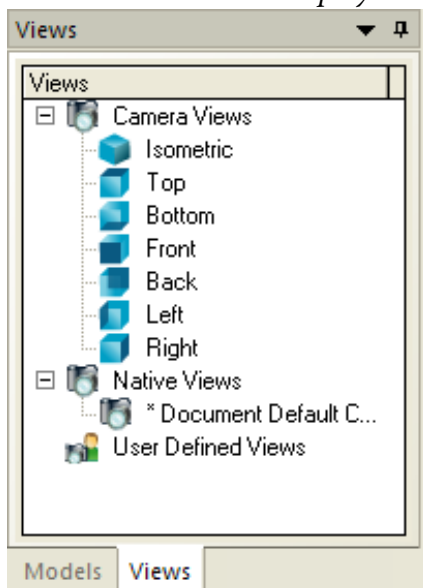
NOTE: The Models tab displays only for 3D drawings.



Views Tab

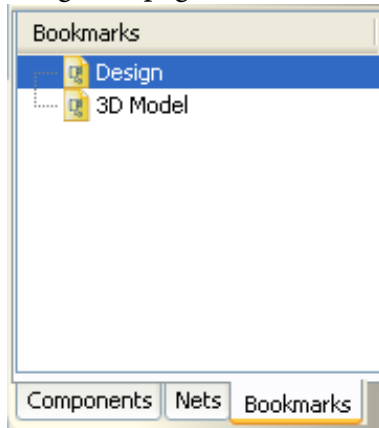
The *Views tab* lists the standard, native, and user-defined views. You can switch to a standard, native, or user-defined view, as well as add or delete user-defined views.

NOTE: The Views tab displays only for 3D drawings.



Bookmarks Tab

The *Bookmarks* tab lists links to specific views (Draft views, 2D plans, 3D Views of an EDA design) or pages or bookmarks structure that is saved in files such as PDF.



Navigate between the pages or files or views by clicking the appropriate link. Bookmarks lead to various views of CAD files such as AutoCAD file Model Space, the 3D model of a file and associated 2D engineering drafts.

If a plus sign appears to the left of a bookmark, click it to expand and view the lower bookmark levels. If a minus sign appears to the left of the bookmark, click it to collapse the lower bookmark levels.

To go to a destination specified by a bookmark, click the bookmark text or the page icon located to the left of the bookmark text.

Global Axes

By default, there is a three-axis representation in the lower left corner of the workspace. The **X-axis** is red, the **Y-axis** is green and the **Z-axis** is blue. When transforming a model or defining a viewpoint, all operations are applied with respect to these axes.

Selecting Model Parts

You can select model parts from the *Model Tree* or from the workspace to highlight them in the Model Tree and on the model. You can also select a model part and have all its identical parts appear highlighted on the model and in the Model Tree.

You can also configure the **Selection Highlight**.

Select Model Parts from the Workspace

TASK

1. Select a part or parts on the model in the workspace.
ADDITIONAL INFORMATION: To select more than one model part, press the **Shift** or **Control** key while selecting.
 2. To select the parent entities of a selected part, press the **Shift** key and select the part again.
STEP RESULT: A pop-up appears listing the parent entities of the selected part. Select an entity from the pop-up list. The selected part or parts appear highlighted on the model and in the Model Tree.
 3. To select a group of parts, from the **Edit** menu, select **Select**, then click and drag the mouse around the parts.
-

RESULT: The entities within the box are selected and highlighted in the Model Tree. If the selected entity is hidden in the Model Tree, the Model Tree expands to display the selected entity.

Selecting All Identical Parts of a Model

Select a model part in the workspace or from the Model Tree.

Right-click and select **Select Identical Parts**.

All identical parts are highlighted on the model and in the Model Tree. If there are no identical parts found, a message appears indicating “No identical parts found”. You can also select sub-assemblies and **Select Identical Parts** to display all identical sub-assemblies.

Re-Centering

The *Re-center* option repositions the model back to the center of the View window. You can use a model part as a central reference point to reposition a model. You can select several model parts to use as one collective central reference point.

Re-Centering a Model to a Selected Model Part

TASK

1. Select the model part or parts that you want to re-center.
 2. From the **View** menu, select **Re-Center**, and then select **Selected**.
-

RESULT: The model is repositioned using the selected model part or parts as the central reference point. The **Selected** option is only available when one or more model parts are selected.

Re-Center All

The **Re-Center All** option repositions the entire model back to the center of the AutoVue workspace.

TASK

1. From the **View** menu, select **Re-Center**, and then select **All**.

STEP RESULT: The model is repositioned to the center of the workspace.

Entity Reference

You can use an entity as a central reference point to reposition a model. The entities are as follows:

Entity	Description
Vertex	All vertices are highlighted. Select a vertex to use as the central reference point. A snap box appears when moving the mouse over a vertex.

Entity	Description
Edge	All edges are highlighted. Select an edge to use as the central reference point. A snap box appears when moving the mouse over an edge.
Midedge	Highlight all edges in the model. Select a mid-edge to use as the central reference point.
Arc Center	Highlight all arcs and ellipses in the model. Select an arc center to use as the central reference point. A snap box appears when moving the mouse over an arc or ellipses, indicating the center of the arc.
Face	Select a face to use as the central reference point. When moving the cursor along a model face, the face is highlighted.

Re-Centering a Model to an Entity

To re-center a model to an entity, perform the following:

TASK

1. From the **View** menu, select **Re-Center**, and then select **Entity**.
STEP RESULT: The Re-Center dialog appears.
 2. Select the entity that you want to use as the central reference point.
STEP RESULT: All instances of the entity are highlighted.
 3. Click an instance of the entity.
STEP RESULT: The model is repositioned using the selected entity as the central reference point.
 4. Close the Re-Center dialog.
-

Model Tree

The *Model Tree* displays the model's hierarchy, inter-relation of different parts, assemblies, bodies, and missing XRef notifications. With the tree, you can select different parts and modify their attributes such as color, visibility, render mode or transformation.

Expanding/Collapsing the Model Tree

You can expand the Model Tree to display child entities of a selected node or nodes.

You can also configure the level you want to expand the Model Tree.

TASK



1. Click the **Models** tab.
2. From the Model Tree, select the node or nodes that you want to expand, then right-click and select **Expand All Children**.

ADDITIONAL INFORMATION: To select more than one node, press the **Shift** or **Control** key while selecting.

STEP RESULT: The selected entities are highlighted. The Model Tree expands displaying the child entities of the selected nodes.

3. To collapse a node, select the node, then right-click and select **Collapse All Children**.

STEP RESULT: The Model Tree collapses the selected node.


RESULT: The Model Tree collapses the selected node. You can also expand a node by clicking . To collapse a node click .

Viewing Missing XRef Notification Icon

If the loaded drawing is missing XRefs, the **Missing XRef Notification** icon  appears in the Model Tree to the left of the missing XRef. To view XRefs are missing, do the following:


See Also “Displaying Details About Resources”

TASK

1. From the status bar, click the **Missing Resource** icon .

STEP RESULT: The Properties dialog appears. You can also view the missing XRef from the **File** menu, and then selecting **Properties**.

2. Click the **Resource Information** tab.

STEP RESULT: Any missing XRefs appear in the External Reference File Reference File Resources section of the dialog and are preceded by a .

3. Click **OK** to close the dialog.
-

Selecting Model Parts from the Model Tree

TASK

1. Click the **Models** tab.
2. Select the part or parts from the Model tree.

ADDITIONAL INFORMATION: To select more than one model part, press the **Shift** or **Control** key while selecting.

RESULT: The selected part or parts appear highlighted on the model and in the Model Tree.

Hiding Model Parts

You can hide specific parts of a model or display specific parts and hide the rest of the model.

TASK

1. Select the part or parts on the model or from the Model Tree.

ADDITIONAL INFORMATION: To select more than one model part, press the **Shift** or **Control** key while selecting.

STEP RESULT: The selected part or parts appear highlighted on the model and in the Model Tree.

2. To hide the selected part or parts, right-click a selected part on the model or from the Model Tree and select **Hide**.

STEP RESULT: The selected part or parts are hidden on the model.

3. To display the selected part or parts and hide the rest of the model, right-click a selected part on the model or from the Model Tree and select **Hide Rest**.

STEP RESULT: The selected parts are displayed in the workspace and the rest of the model is hidden.

Creating 3D Mockups

You can import other 3D Models into the current active file.

The imported files must be 3D and have similar dimensions.

TASK

1. From the **File** menu, select **Import File for Mockup**.
STEP RESULT: The Mockup dialog box appears.
 2. Click **Add**.
STEP RESULT: The Open dialog box appears.
 3. Enter the file name or browse to locate the file you want to import.
 4. Click **Open**.
STEP RESULT: The file appears in the Mockup dialog box. To import more than one file, repeat steps 2 to 4.
 5. Click **OK**.
-

RESULT:

The file or files you imported appear in the workspace. Once open, you can position these models in the desired manner using the Transformation tool, or align the models using Part Alignment, or run Interference Checks.

See Also “Transformation”

*** UNDEFINED CROSS-REF FORMAT [SeeAlsoList] ***

*** UNDEFINED CROSS-REF FORMAT [SeeAlsoList] ***

Deleting Models from a Mockup

The following steps explain how to delete models from a mockup.

TASK

1. From the **File** menu, select **Import File for Mockup**.
STEP RESULT: The Mockup dialog appears.
 2. Select the file or files that you want to remove.
 3. Click **Remove**.
STEP RESULT: The file or files are removed from the list.
 4. Click **OK** to close the Mockup dialog.
-

RESULT: The file or files disappear from the workspace.

Converting 3D Models to Other Formats

Sometimes you need to translate a 3D model to be able to use it with an application it was not created from. AutoVue provides several conversion file formats for you.

Conversion Options

Depending on the conversion type being performed, the available options in the Convert dialog box will vary. These options are:

Option	Description
Save As	Specify the name and path of the file in which the conversion is to be stored. This file is also known as the output file. You can use Browse to provide AutoVue with the file's path.
Convert to Format	A list of all the possible types of output file formats currently available for conversion. The available formats are: CALS GP4 Encapsulated Postscript (Raster) HP Laserjet Printer (PCL) Run Length RLC File PCX Bitmap PDF Stereolithography (STL) TIFF Virtual Reality Modeling Language (VRML) Windows Bitmap
Sub-Format	The Sub-Format list appears when you select TIFF or STL from the Convert to Format list. Select the Sub-Format from the list.
Color Depth	Select an option from the list: 1 = Black and white 4 = 16 colors 8 = 256 colors 24 = True color auto = AutoVue selects the color depth that best matches the original file. Option is not available for STL and VRML formats.
Positive Triangle Values	When selected, the model is translated so that all the vertex coordinate values are positive. Option is only available when Convert to Format is set to STL .

Option	Description
Convert Region	The area of the file to be converted. The available options are: All - converts the entire file Display - converts the image to fit on the output page For example, if you zoomed in on a particular region of the file, the zoomed portion of the file is converted. Extent - converts the entire extents of the file Selected - converts the selected model parts in the file Options vary depending on the selected format.
Convert Pages	The number of pages to be converted. The available options are: All - converts all pages of the file Current - converts the current page of the file Range - converts the pages of the file indicated in the range Option is only enabled when converting to TIFF .
X and Y	Specify X and Y to define the resolution for the converted file. X indicates the number of horizontal pixels Y is the number of vertical pixels for the current active file contents At times, AutoVue presets X and Y to match the specifications of the selected conversion file format.

Converting a 3D Model

TASK

- From the **File** menu, select **Convert**.
STEP RESULT: The Convert dialog appears.
- In the **Save As** field, enter the path and filename or click **Browse** to locate the directory where you want to convert the file to, then enter a file name.
- From the Convert to Format list, select the format you want to convert the file to.
ADDITIONAL INFORMATION: The Output and Convert area options change according to the type of format you choose.
- Select the output and convert options that you want to set for the file.
- Click **OK**.









RESULT:




The Conversion in Progress dialog appears. The dialog disappears when the file is converted successfully.

Manipulating Display of a 3D Model

AutoVue provides the flexibility to manipulate the display of a 3D model. You can rotate, scale, or translate a model or any selection of model parts. You can also navigate from one page to another page of a multi-page file.

You can access these options from the **View** menu. The options are:

Menu	Sub-Menu	Description
Zoom	Zoom In	Zoom in by a factor of 2. From the Autovue toolbar, you can also click  .
	Zoom Out	Zoom out by a factor of 2. From the Autovue toolbar, you can also click  .
	Previous	Return to the previous zoom level. From the Autovue toolbar, you can also click  or right-click in workspace and then select from the menu.
	Selected	Resize the object so that the objects selected fill the window.
	Dynamic Zoom	Click and drag the cursor up to zoom in or down to zoom out. From the Autovue toolbar, you can also click  .
	Zoom Box	Click and drag to draw a box around an object that you want to enlarge to fill the window. From the Autovue toolbar, you can also click  or right-click in workspace and then select from the menu.
Zoom Fit		Resize object to fit window. From the Autovue toolbar, you can also click  or right-click in workspace and then select from the menu.
Pan		Click and drag to reposition the model, then release the mouse button. From the Autovue toolbar, you can also click  or right-click in the workspace and then select from the menu.
Rotate		Click and drag to rotate the model on all three axes, then release the mouse button. From the Autovue toolbar, you can also click  or right-click in workspace and then select from the menu.

Menu	Sub-Menu	Description
Spin		Click and drag the object in the direction you want the object to spin continuously, then release mouse button. To stop the object from spinning, click once anywhere in the workspace. The spin velocity is determined by the speed at which you drag the mouse. From the Autovue toolbar, you can also click  .
Re-Center	All	Reposition the entire model back to the center of the AutoVue workspace. See “Re-Centering”
	Selected	Only available when one or more model parts are selected. The selected part or parts are considered as the central reference point by which the model is repositioned. See “Re-Centering”
	Entity	Select a model part or entity as a central reference point to reposition a model. See “Re-Centering”
Default View		Displays view of the model when it is initially loaded. You can also right-click in the workspace and then select from the menu. See “3D Views”
Camera Views		Displays different views of 3D models: Isometric, Top, Bottom, Front, Back, Left, Right, and View Points. You can also right-click in the workspace and then select from the menu. See “3D Views”
Views		Access native views of the model or access and create your own views. You can also right-click in the workspace and then select from the menu. See “3D Views”
Page	Next Page	Go to the next page of a multi-page file. From the Autovue toolbar, you can also click  .
	Previous Page	Go to the previous page of a multi-page file. From the Autovue toolbar, you can also click  .
	Page Number	Go to the specified page of a multi-page file. From the Autovue toolbar, you can also click  .

Display Attributes

AutoVue provides several different render methods to display a 3D CAD model. You can also adjust the degree of transparency, change color, or visibility to suit your preferences.

Render Modes


The choice of render mode varies between the level of detail and the render speed of the model. For example, a shaded model is three-dimensional and highly detailed but requires more time to render. The different render modes are as follows:

Method	Description
Wireframe	A skeletal model constructed using lines and curves that represent the model's "true" edges. All internal lines are visible.
Shaded	A solid model constructed of planes and surfaces. These surfaces are shaded to increase the illusion of three dimensions.
Hidden Line	A wireframe constructed with all internal lines hidden.
Silhouette	A wireframe constructed with all internal lines visible but including additional silhouette edges. These are not "true" edges but help to visualize the model.
Wire Polygons	A skeletal model constructed of unfilled polygons.
Shaded Wire	An outlined solid model constructed of planes and surfaces. The outline is a solid line and the model's surfaces are shaded to increase the illusion of three dimensions.
Reflective	A solid model constructed of planes and surfaces. These surfaces are have a reflective sheer to accentuate the model's shadows.
Reflective Wire	An outlined solid model constructed of planes and surfaces. The outline is a solid line and the model's surfaces have a reflective sheer to the model's shadows.

Changing the Render Mode

To change the render mode of the selected model or model parts, perform the following:

TASK

1. From the **Manipulate** menu, select **Visual Effects**, and then select **Rendering**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Render Mode** .
STEP RESULT: Render mode options are listed.
 2. Select a render mode from the list.
STEP RESULT: The selected render mode is highlighted.
-

RESULT: The model or selected model parts change to the selected render mode.

Changing the Visibility

You can choose to show or hide selected parts; this can be done from either the Model Tree or the workspace.

From the Model Tree, deselect the check box beside the model part or parts that you want to hide.

Conversely, select the check box beside the model parts that you want to set visible.

RESULT:


The selected model parts appear or disappear from the model.

NOTE: You can also hide model parts by selecting them on the model or from the Model Tree, then right-clicking and selecting **Hide**. Selecting **Hide Rest** displays the selected model parts and hides the rest of the model.

Changing Model Color

You can change the color of a specific model part or selected model parts.

TASK

1. Select the model or model parts whose color you want to change.
ADDITIONAL INFORMATION: If no part is selected, the change is applied to the entire model.
2. From the **Manipulate** menu, select **Visual Effects**, and then select **Color**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Color** .
STEP RESULT: The Color dialog appears.

3. Select the desired color.
 4. Click **Apply**.
-

RESULT: The selected part or parts change to the selected color.


NOTE: To set the model color to its default color, repeat steps 1 to 2 and click **Reset** in the Color dialog.

Adjusting the Transparency

You can adjust the degree of transparency of a model. This function only applies to shaded and shaded wire models.

TASK

1. Select the model or model parts whose transparency you want to adjust.
2. From the **Manipulate** menu, select **Visual Effects**, and then select **Transparency**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Transparency** .

STEP RESULT: The Set Transparency dialog appears.

3. To adjust the transparency, move the slider left or right.
4. Select **Apply Dynamically** if you want the transparency to simultaneously change with the movement of the slider.

ADDITIONAL INFORMATION: Alternatively, to adjust the transparency level, you can enter in the **Value** field a value from 0 to 1. Where **0** renders the model opaque (default state) and **1** renders it transparent.

5. Click **Close** to apply the changes and close the Set Transparency dialog.
-

Changing the Mesh Resolution

TASK

1. Select the parts that you want to change the mesh resolution.
 2. From the **Manipulate** menu, select **Visual Effects**, and then select **Mesh Resolution**. You then have the option to select **High**, **Medium**, or **Low**.
-

RESULT:

NOTE: Changing the mesh resolution will not affect the default Mesh Resolution set in the Configuration dialog box.

Light Settings

The default light setting consists of a white light at the 10 o'clock position with respect to the model. In the Lighting dialog, it appears as a grey ball of white light at the 10 o'clock position along the circumference of the larger sphere.

Ambient lighting is the overall lighting that surrounds an object. It provides constant illumination to every surface of the model. This type of lighting is particularly effective as a fill-light for surfaces not directly illuminated by a directional light source. You can set the intensity or source position; too high a setting tends to saturate the image and reduce its clarity.

Using the **Directional lighting** option, you can adjust the position of the light source on an object.

You can make the following modifications via the Lighting dialog:

- Set the intensity and source position of the ambient light.
- Set the direction of light.
- Add and remove a light source.
- Change light color, brightness, and specularly.

Setting Ambient Lighting

TASK

1. From the **Manipulate** menu, select **Visual Effects**, and then select **Lighting**.

STEP RESULT: The Lighting dialog appears.

2. Click and drag the **Ambient Light** sliding bar until you achieve the desired lighting.

STEP RESULT: The lighting automatically changes with the movement of the sliding bar. The **Custom** option is selected when you modify the ambient light.

3. To set the ambient lighting to its default setting, select **Default**.
 4. Click **Close** to close the Lighting dialog.
-

Setting Directional Lighting

Setting the directional lighting adjusts the position of the light source on the model. You can also add a new light source.

TASK

1. From the **Manipulate** menu, select **Visual Effects**, and then select **Lighting**.
STEP RESULT: The Lighting dialog appears.
 2. To change the direction of the light, click and drag the small ball until you achieve the desired lighting.
ADDITIONAL INFORMATION: You can also drag the white ball outside of the large ball.
STEP RESULT: The direction of the light reflected on the 3D model automatically changes with the movement of the white ball. The **Custom** option is selected when you adjust the position of the light.
 3. To set the light direction to its default setting, select **Default**.
 4. Click **Close** to close the Lighting dialog.
-

Adding a New Light Source

TASK

1. From the **Manipulate** menu, select **Visual Effects**, and then select **Lighting**.
STEP RESULT: The Lighting dialog appears.
 2. Select the **Two lights** option.
STEP RESULT: A new light source appears as a black ball in the 5 o'clock position.
 3. To add more light sources, right-click inside the square surrounding the ball and select **Create New Light Source** from the pop-up menu.
STEP RESULT: The Custom option is selected and the new light source appears as a white ball.
 4. Click and drag the small ball until you achieve the desired lighting.
 5. To restore the light source to its default setting, select **Default**.
 6. Click **Close** to close the Lighting dialog.
-

RESULT:

NOTE: You can have a total of eight light sources at a time.

Changing the Light Properties

TASK

1. From the **Manipulate** menu, select **Visual Effects**, and then select **Lighting**.

STEP RESULT: The Lighting dialog appears.

2. To change the properties of the light, such as color or brightness, right-click directly on the small ball and select **Light Properties**.

STEP RESULT: The Light Property dialog appears.

3. Select a color from the **Color** menu.

ADDITIONAL INFORMATION: You can drag the **Brightness** and **Specularity** slide bars to modify the brightness of the source and the brightness of the model's reflection to light, respectively.

4. Click **OK**.

ADDITIONAL INFORMATION: To set the light properties to its default setting, select **Default**.

STEP RESULT: The light properties modifications are applied.

5. Click **Close** to close the Lighting dialog.
-

Removing a Light Source

TASK

1. From the **Manipulate** menu, select **Visual Effects**, and then select **Lighting**.

STEP RESULT: The Lighting dialog appears.

2. Right-click directly on the small ball that you want to remove and select **Remove Light**.

STEP RESULT: The small ball disappears and the light change is reflected.

3. You can also select **Default** to restore the lighting properties to its default setting.

4. Click **Close** to close the Lighting dialog.
-

3D Views

You can display different views of 3D Models or create your own views. The following sections provide information on the default view of the model, the available camera views, native views, and user-defined views.

Default View

The **Default View** is the view of the model when it is initially loaded; usually it is the view in which the model was saved. In cases where there is no saved view, AutoVue loads the isometric view of the model.

To return to the default view of a 3D model, from the **View** menu, select **Default View**.

Setting Standard or Camera Views

Rather than entering Rotation mode, you can view various predefined rotations with the **Camera Views** option from View and Markup modes.

To do so, from the **View** menu, select **Camera Views**, and then select the pre-defined view you want to display.

The view options are Isometric, Top, Bottom, Front, Back, Left, and Right.

The model changes to the selected view. You can also click the **Views** tab, then click the view from the Standard Views tree, or right-click in the workspace and select **Camera Views**.

Setting Native Views

AutoVue displays views for a 3D file as saved in its native application. Native views are present only if the file contains saved views.

From the **View** menu, select **Views**, and then select **Native Views**.

The model re-orientes to the selected view. The view options vary depending on the file. Examples of the view options are Last Saved View, Presentation, Front, Back, Left, Right, Top, Bottom, Isometric, Trimetric, and Dimetric. You can also click the **Views** tab, and then click the view from the **Native Views** tree, or right-click in the workspace and select **Views**, and then select **Native Views**.

Creating a User-Defined View

You may want to create and save your own views. AutoVue allows you to define a view and add it to the **User Defined Views**. You can define a customized view in View or Markup mode. Any views you apply to displayed models during Markup mode are saved as part of the Markup file.

TASK

1. Apply your own views or transformation to the displayed model.

ADDITIONAL INFORMATION: The view states that you can apply and save for your defined view are Extents, Rotation, Model Transformation, Explosion, Render Modes, Color, Transparency, Visibility, Sectioning, Camera Settings and views involving Mockups.

2. From the **View** menu, select **Views**, select **User Defined Views**, and then select **Add View**.

ADDITIONAL INFORMATION: You can also right-click in the workspace and select **Views**, then select **User Defined Views**, and then select **Add View**.

STEP RESULT: The Add User Defined View dialog appears.

3. Enter a view name for the view that you want to define.
 4. Click **OK** to close the Add User Defined View dialog.
-

RESULT:

To see the view you defined, click the **Views** tab and select it from the User Defined Views tree or select **Views**, select **User Defined Views**.

NOTE: You can also alternate between **User Defined Views** and **Standard Views** without affecting your personalized views.

Deleting a User-Defined View

TASK

1. Under the User Defined Views tree, select the view that you want to delete, then right-click and select **Delete**.

STEP RESULT: A confirmation prompt appears.

2. Click **Yes**.
-

RESULT: The view disappears from the tree.

Displaying the Perspective Projection of a 3D Model

The **Perspective** option shows objects in three dimensions with distances, planes and curved surfaces adjusted to give a sense of depth that reflects the desired perspective to the eye.

*NOTE: You can use the **Perspective** option in any of the 3D Views since it is a projection mode rather than a view point.*

From the **Manipulate** menu, select **Visual Effects**, and then select **Perspective**.

The depth of the model changes.

Viewing a Model from a Particular View Point

To view a model from a particular point, you can specify a view point.

TASK

1. From the **View** menu, select **Camera Views**, and then select **View Points**.

ADDITIONAL INFORMATION: You can also right-click in the workspace and select **Camera Views**, and then select **View Points**.

STEP RESULT: The View Point dialog appears.

2. Enter the coordinates for **X**, **Y** and **Z**:
 - a In the Camera Position fields, enter the coordinates to determine the position of the camera lens.
 - b In the Target Position fields, enter the coordinates to determine the position of the 3D model viewed through the camera lens.
 - c In the Up Direction fields, change the coordinates to values between 0 and 1 to determine the upward orientation.
 3. Click **OK** to close the View Point dialog.
-


RESULT: The entity moves to the defined view point. The Global axes and the User Coordinates System change position to reflect the view point.

Configuring Layers

Use this option to configure which layers to display from the current active file.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **Layers**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Layers** .

STEP RESULT: The Select the Layers to Display dialog appears and lists the layers and layer visibility for the current active file.

2. To sort the list of layers in the dialog, click the **Name** column header to sort alphabetically or numerically, or click the **Status** column header to group by visibility.
 3. Select the check box beside the layers that you want to display.
 4. Deselect the check box beside the layers that you want to hide.
 5. Click **OK** to close the Select the Layers to Display dialog.
-

RESULT: The selected layers are displayed.

Entity Properties

You can view properties such as visibility, color, transparency, mass properties, and extents of a model or model parts.

General Attributes

The **Attributes** tab displays a model or model part's attributes. The list of attributes varies depending on the model. Some of the viewable **General** attributes are as follows:

Attribute	Description
Color	Color of the selected model part.
Density	The density of the model or selected model parts.
Name	The model part name or the displayed page name of the model.
Render Mode	The dynamic rendering used for displaying model or model part. For example, Shaded , Shaded Wire and Wireframe .

Attribute	Description
Transparency	The value between 0 and 1 representing the model or model part's degree of transparency. 0 = opaque 1 = transparent
Visibility	The value True (visible) or False (invisible) for a model or model part.

If you are displaying the 3D view of an EDA design, AutoVue displays the general attributes mentioned above. In addition, AutoVue displays attributes specific to the Printed Circuit Board (PCB), such as component name, board side, component class and device type.

If a 3D file has Product and Manufacturing Information (PMI) and a PMI entity is selected, the PMI attributes are displayed along with the above general attributes. Some of the viewable PMI attributes are X-Axis, Y-Axis, font color, text font name, and tolerance type.

Viewing Attributes

TASK

1. To view the attributes of specific model parts, select the parts from the model.
2. To view the attributes of the entire 3D model, make sure that no parts are selected.
3. From the **Analysis** menu, select **Show Entity Properties**.

ADDITIONAL INFORMATION: You can also right-click in the workspace and select **Show Entity Properties**, or in the Model Tree, select the model part or parts, then right-click and select **Show Entity Properties**.

STEP RESULT: The Entity Properties dialog appears.

4. Click the **Attributes** tab.

STEP RESULT: The attributes of the selected model parts are displayed in a hierarchical tree.

5. Click **Close** to close the Entity Properties dialog.

Viewing Mass Properties

The Mass Properties tab displays the precise measurements for mass, volume, area, center of gravity, moments of inertia, and inertia tensor for any model or selection.

TASK

1. Select the part or parts for which you want to calculate the mass properties. To select multiple parts, press the **Shift** or **Control** key while selecting.

2. To view the mass properties of the entire 3D model, make sure no parts are selected.

ADDITIONAL INFORMATION: You can also make your selection after you open the Entity Properties dialog.

3. From the **Analysis** menu, select **Show Entity Properties**.

ADDITIONAL INFORMATION: You can also right-click in the workspace and select **Show Entity Properties**, or in the Model Tree, select the model part or parts, then right-click and select **Show Entity Properties**.

STEP RESULT: The Entity Properties dialog appears.

4. Click the Mass Properties tab to view the properties of the selected mass.

STEP RESULT: When a mass property cannot be calculated, "N/A" displays in red for that property. When this is the case, you can also click **Error Report** for a list of mass properties that could not be calculated.

5. To change the density, change measurement units, or configure computation of inertia tensor, click **Options**.

STEP RESULT: The Options dialog appears.

6. Click **OK** to close the Options dialog.

7. Click **Close** to close the Entity Properties dialog.

Configuring Mass Properties

From the Mass Properties tab, you can change the density, the measurement units or configure the reference point of inertia tensor.

TASK

1. From the **Analysis** menu, select **Show Entity Properties**.

ADDITIONAL INFORMATION: You can also right-click in the workspace and select **Show Entity Properties**, or in the Model Tree, select the model part or parts, then right-click and select **Show Entity Properties**.

STEP RESULT: The Entity Properties dialog appears.

2. Click the Mass Properties tab.

3. Click **Options**.

STEP RESULT: The Options dialog appears.

4. To change the density, enter a value in the **Density** field.

5. To change the density units, select the unit for **Mass** and **Length** from their respective lists in the Units section.

6. To apply the density to model parts with unknown density, select **Use Only for Parts with Unknown Density**.

7. To apply the density to all model parts, select **Use for All Parts**.
 8. To change the display units, select the unit for **Mass** and **Length** from their respective lists in the Display Units section.
 9. To compute the inertia tensor based on output coordinate system, select **Output Coordinate System Origin**.
 10. To compute the inertia tensor based on the center of gravity, select **Center of Gravity**.
 11. Click **OK** to close the Options dialog.
STEP RESULT: Mass properties are instantly calculated and displayed in the Mass Properties tab. When a mass property cannot be calculated, "N/A" displays for that property. When this is the case, you can also click **Error Report** for a list of mass properties that could not be calculated.
 12. To save the changes, click **Save As**.
STEP RESULT: The Save Mass Properties As dialog appears.
 13. Specify the path where you want to store the file and enter the file name, then click **Save**.
STEP RESULT: AutoVue saves the results in a .txt file.
 14. Click **OK** to close the Entity Properties dialog.
-

Viewing Extents

The Extents tab displays the orientation coordinates and center coordinates of the X, Y, and Z axes, as well as the width, height, and depth measurements for any model or selection.

TASK

1. Select the part or parts for which you want to view Extents. To select multiple parts, press the **Shift** or **Control** key while selecting.
2. To view the Extents of the entire 3D model, make sure no parts are selected.
ADDITIONAL INFORMATION: You can also make your selection after you open the Entity Properties dialog.
3. From the **Analysis** menu, select **Show Entity Properties**.
ADDITIONAL INFORMATION: You can also right-click in the workspace and select **Show Entity Properties**, or in the **Model Tree**, select the model part or parts, then right-click and select **Show Entity Properties**.
STEP RESULT: The Show Entity Properties dialog appears.
4. Click the Extents tab.

5. You have three extents options:
 - a Select **Transformed** if you want to view the extents after you have transformed the model.
 - b Select **UnTransformed** if you want to view the extents of the model without transformation.
 - c Select **Oriented** if you want to view the X, Y, and Z coordinates of the reoriented model or selection.
 6. Click **Close** to close the Entity Properties dialog.
-

PMI Entities

A 3D file's Product and Manufacturing Information (PMI) is composed of annotations that are included in design files. These annotations indicate the limits and constraints that must be observed during the production of the object displayed in the 3D model.

At the highest level, PMI provides information about dimensions, feature control frames, weld specifications, and surface finishes. This information is based upon key design features which are specified through datum targets, measurement points, reference geometry (for example, construction lines, surfaces, and objects), or the geometry inherent in the object.

The following terms can be used interchangeably with the term PMI:


- Datums Cosmetics
- Dimensioning Cosmetics
- Geometric Tolerances (GTOLs)
- Geometric Dimensioning and Tolerance (GDT or GD&T)
- Functional Tolerance Annotation (FTA or FTA&A)

PMI Filtering

Use PMI Filtering to select which types of product and manufacturing information to display.

TASK

1. From the **Manipulate** menu, select **Visibility Control**, and then select **PMI Filtering**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **PMI Filtering** .

STEP RESULT: The PMI Filtering dialog appears and displays all PMI types.

2. In the Tree column, select the check box beside the items that you want to display in the Model Tree.
 3. In the View column, select the check box beside the items that you want to display on the 3D model.
 4. Deselect the check box beside the items that you want to hide.
ADDITIONAL INFORMATION: Click **All** to show all items, or click **None** to hide all items.
 5. Click **OK** to close the PMI Filtering dialog.
-

RESULT:

Only items selected in the **Tree** column are displayed in the Model Tree. Only items selected in the **View** column are visible on the 3D model. Visibility settings are saved and will be restored the next time you open the PMI filtering dialog.

NOTE: AutoVue supports default PMI visibility as saved in the file. To restore default PMI visibility, click **Default** in the PMI Filtering dialog.

Aligning to a PMI Entity

From the Model Tree, right-click the PMI item that you want to align to and select **Align To**. AutoVue aligns to the selected PMI item.

Go to a Displayed PMI Entity

From the Model Tree, right-click the PMI item that you want to go to on the model and select **Go To**. AutoVue zooms to the selected PMI item.

PMI Configuration Entities

The Views, Captures, Reference/Grouping entities refer to specific configurations of the model. These PMI configuration entities are listed in the Model Tree and are activated by right-clicking the entity and then selecting **Activate**.

Views

The View configuration entity displays predefined views, and highlights associated PMI entities.

TASK

1. Expand the Views tree to display the defined views.
 2. Right-click the selected view, and then select **Activate**.
-

RESULT:

The model and PMI entities display in the workspace as defined in the selected view. The PMI entities are also highlighted in the Model Tree.

Captures

The Captures configuration entity displays predefined views.

TASK

1. Expand the Captures tree to display the defined captures.
 2. Right-click the selected captures, and then select **Activate**.
-

RESULT: The model and PMI entities display in the workspace as defined in the selected capture.

Reference Frames

The Reference Frames configuration entity highlights the grouped PMI entities.

TASK

1. Expand the Reference Frames tree to display the predefined reference frames.
 2. Right-click the selected reference frame, and then select **Activate**.
-

RESULT: The grouped PMI entities are highlighted in the Model Tree.

PMI Hyperlinks

PMI entities can also include hyperlinks. To fire a PMI hyperlink, do the following:

TASK

1. From the Model Tree or workspace, right-click the PMI hyperlink entity.
2. From the menu, select **Follow Hyperlink**.

STEP RESULT: The 3D Hyperlink dialog appears if there are multiple hyperlinks associated with the selected PMI entity.

3. From the 3D Hyperlink dialog, select a link and then click **Fire**.
-

RESULT: The selected link opens.

Manipulating a 3D Model


You can rotate 3D CAD models along a specific axis. You can scale a model part up and down, as well as translate the model.

In Manipulate Mode, you can resize, translate, and/or rotate selected parts of the model. When you select one or more model parts, AutoVue displays a model-size representation of the Global Axes going through the selected part or parts of the 3D model.

Panning a Model Along the X, Y and Z-Axis

TASK

1. From the **Manipulate** menu, select **Manipulate**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Manipulate** .


STEP RESULT: You are now in Manipulate mode.

2. Select the model part or parts that you want to pan.

ADDITIONAL INFORMATION: To select more than one part, press the **Shift** or **Control** key while selecting.

STEP RESULT: A model-size representation of the Global Axes appears through the model part or parts and are encased by a three-dimensional box.

3. Click and hold the mouse button on the arrow at the end of the axis that you want to pan.


4. Drag the mouse to where you want to move the part or parts.
5. To exit Manipulate mode, click **Manipulate** , or from the **Manipulate** menu, select **Manipulate**.

ADDITIONAL INFORMATION: To restore the default state of a part of the model, from the **Manipulate** menu, select **Transform**, and then select **Reset Selected**. To restore the default state for the whole model, select **Reset All**.

Rotating a Model Along the X, Y and Z-Axis

TASK

1. From the **Manipulate** menu, select **Manipulate**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Manipulate** .

STEP RESULT: You are now in Manipulate mode.

2. Select the model part or parts that you want to rotate.

ADDITIONAL INFORMATION: To select more than one part, press the **Shift** or **Control** key when selecting.

STEP RESULT: A model-size representation of the Global Axes appears through the model part or parts and are enclosed by a three-dimensional box.

3. Click and hold the mouse button on the sphere at the end of the axis you want to rotate.

ADDITIONAL INFORMATION: The initial mouse movement determines which of the two axes will mark the site of rotation. If the axis you selected does not rotate around the right axis for the rotation that you want to make, click the sphere again and move the mouse in a different direction.

4. Clicking on an axis sphere allows rotation around one of the other two axes.
5. Move the mouse to rotate the model by the selected axis.

ADDITIONAL INFORMATION: If you want to rotate the selection freely, as done in Rotate Mode, press the **Control** key during rotation.


6. To exit Manipulate mode, click **Manipulate** , or from the **Manipulate** menu, select **Manipulate**.

ADDITIONAL INFORMATION: To restore the default state of a part of the model, from the **Manipulate** menu, select **Transform**, and then select **Reset Selected**. To restore the default state for the whole model, select **Reset All**.

Scaling a Model Along the X, Y and Z-Axis

TASK

1. From the **Manipulate** menu, select **Manipulate**.


ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Manipulate** .

STEP RESULT: You are now in Manipulate mode.

2. Select the model part or parts that you want to scale.

ADDITIONAL INFORMATION: To select more than one part, press the **Shift** or **Control** key while selecting.

STEP RESULT: A model-size representation of the Global Axes appears through the model part or parts and are encased by a three-dimensional box.

3. Click and drag a corner cube of the box up or down to scale the selected model part.
4. To exit Manipulate mode, click **Manipulate** , or from the **Manipulate** menu, select **Manipulate**.

RESULT:

To restore the default state of a part of the model, from the **Manipulate** menu, select **Transform**, and then select **Reset Selected**. To restore the default state for the whole model, select **Reset All**.

Part Alignment

With the **Part Alignment** option, you can select a point on a model's vertex, edge, or face to align to another model or model part.

The **Mobile** point determines the precise point of alignment for the model part. The **Fixed** point on the model determines the precise fixed point to which the mobile part aligns.

When aligning models parts, you can select from one of the following Constraint Types:

Option	Type	Description
Constraint	Coincident	The Mobile part is positioned so that the selected point on the mobile part coincides with the selected part on the Fixed part.

Option	Type	Description
	Parallel	Align the Mobile point so that it is parallel to the Fixed point.
	Perpendicular	Align the Mobile point so that it is perpendicular to the Fixed point.
	Concentric	Align the Mobile point so that it has the same center as the Fixed point

Part Alignment Constraints

The table below lists the available **Constraint Types** for different combinations of part alignment.

	Point	Line	Arc	Plane	Sphere	Cylinder	Cone
Cone	Coin Con	Par Per	Con	Per Con	Coin Con	Par Con	Coin Con Par
Cylinder	Con	Par Con	Par Con	Per	Coin Con	Par Con	
Sphere	Con	Coin Con	Con	Coin	Coin Con		
Plane	Coin	Per	Coin Par	Coin Par			
Arc	Con	Per Con	Par Con				
Line	Coin	Coin Par					
Point	Coin						

NOTE: The letters in the table indicate the **Constraint Type**:

Where:

Coin- **Coincident**

Par- **Parallel**

Per- **Perpendicular**


Con- **Concentric**

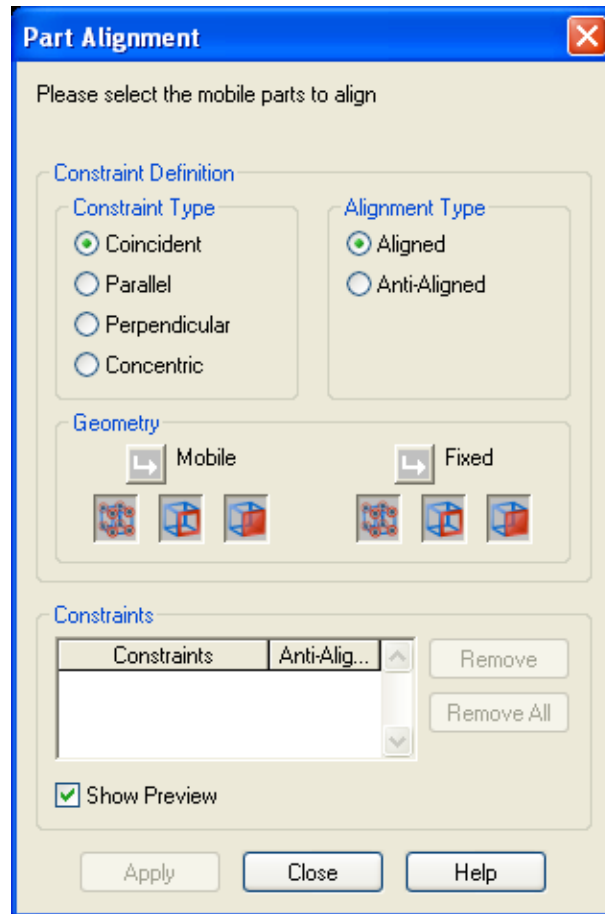
Aligning Model Parts

NOTE: To see a preview of the alignment before applying the change, select the **Show Preview** check box.


TASK








1. From the **Manipulate** menu, select **Align Parts**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Align Parts** .



STEP RESULT: The Part Alignment dialog appears.

2. Select the Constraint Type that you want to perform.
3. Select the Alignment Type.
 - Select **Aligned** if you want to align the Mobile point on the model part to the Fixed point on the model
 - Select **Anti_Aligned** if you want to align the Mobile point on the model part so that it is opposite to the Fixed point on the model.
4. Select a model part from the workspace. The **Mobile** arrow  is enabled.


5. Click **Vertex** , **Edge** , or **Face**  to select the type of geometrical point you want to click on the model part to be aligned.
STEP RESULT: All instances of the selected geometrical point appear highlighted.
6. Click the geometrical point on the selected model part.
STEP RESULT: The geometrical point is highlighted and the **Fixed** arrow  is enabled.
7. Click **Vertex** , **Edge** , or **Face**  to select the type of geometrical point you want to click on the model part to remain fixed.
8. Click the geometrical point on the model.
9. To view a preview of the alignment, select **Show Preview**.
STEP RESULT: The selected alignment appears in the workspace.
10. To apply the constraints, click **Apply**.
ADDITIONAL INFORMATION: To remove a constraint, select the Constraint that you want to remove, then click **Remove**. To remove all part alignments, click **Remove All**.
STEP RESULT: The model part is aligned according to the selected options. The Constraint Type and the types of geometrical points aligned appear under Constraints section of the dialog.
11. Click **Close** to close the Part Alignment dialog.
ADDITIONAL INFORMATION: A prompt will appear if you did not apply the changes before clicking **Close**.
12. To restore the default state of a part of the model, from the **Manipulate** menu, select **Transform**, and then select **Reset Selected**. To restore the default state for the whole model, select **Reset All**.

Transformation

With the **Transformation** option, you can rotate, scale, or translate a model or any selection of model parts, by using the illustrated buttons or by entering X, Y or Z values.

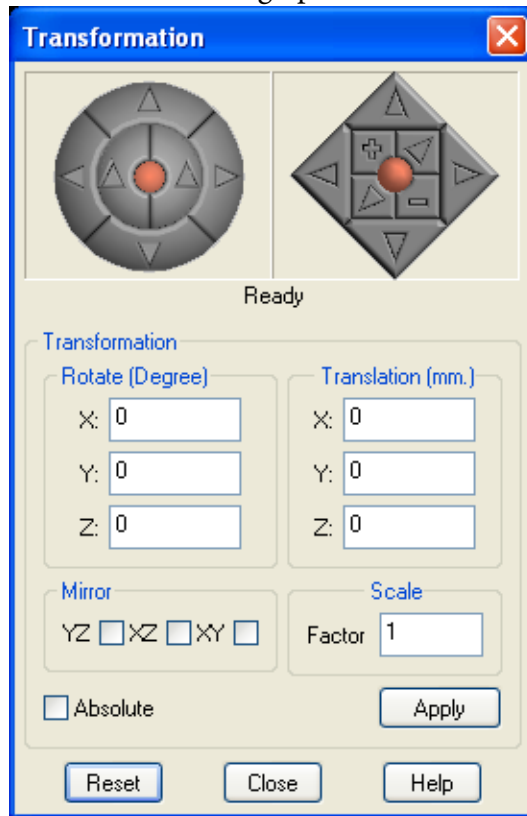
From the **Manipulate** menu, select **Transform**, and then select **Define**.

The Transformation dialog box appears.

NOTE: From the AutoVue toolbar, you can also select **Transform** .


- The buttons on the left correspond to the rotational moves along the three axes.

- The buttons on the right correspond to the translational moves along the three axes and the scaling up and down.



Transforming a Model Using Illustration Buttons

TASK

1. From the **Manipulate** menu, select **Transform**, and then select **Define**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also select **Transform** .
STEP RESULT: The Transformation dialog box appears.
2. Select the model part or parts that you want to rotate, translate, or scale.
ADDITIONAL INFORMATION: To select more than one model part, press the **Shift** or **Control** key while selecting. If no part is selected, transformation is applied to the entire model.
3. Use the rotate or translate buttons to transform the model.

4. To set the translate, rotate and scale increments, click **Options**. The **Options** dialog box appears.
 - a Enter the Translate Increment value in inches.
 - b Enter the Rotate Increment value in degrees.
 - c Enter the **Scale Increment** value, then click **OK**.

ADDITIONAL INFORMATION: When you click a rotation, translation or scale button, the model transforms in increments of the values entered in the **Options** dialog box.
5. Click **Reset** to set the model to its original transformation.
6. Click **Close** to close the Transformation dialog box.


STEP RESULT: The transformation state remains displayed.
7. To restore the default state of a part of the model, from the **Manipulate** menu, select **Transform**, and then select **Reset Selected**. To restore the default state for the whole model, select **Reset All**.

Resetting the Transformation of a 3D Model

You can reset a model or selected parts to their original transformation.

TASK

1. View the transformed 3D Model.
2. To reset the entire model to its original transformation, from the **Manipulate** menu, select **Transform**, and then select **Reset All**.
3. To reset selected model parts to their original transformation, from the **Manipulate** menu, select **Transform**, and then select **Reset Selected**.


ADDITIONAL INFORMATION: You can also reset selected model parts. To do so, from the Model Tree, deselect the check box beside the parts in the **Transformation** column .

STEP RESULT: The model or selected parts changes to their original transformation.

Transforming a Model by Setting Values

TASK

1. From the **Manipulate** menu, select **Transform**, and then select **Define**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also select **Transform** .

STEP RESULT: The Transformation dialog box appears.

2. Select the model part or parts that you want to rotate, translate or scale.
ADDITIONAL INFORMATION: To select more than one model part, press the **Shift** or **Control** key while selecting.
 3. To rotate the model by set values, enter the X, Y and Z value.
 4. To translate the model by set values, enter the X, Y and Z value.
 5. To view a mirror image, in the Mirror section of the dialog box, select a check box.
 6. To scale the model by a factor, in **Factor** field, enter a scale factor.
 7. Click **Apply**.
 8. To apply absolute transformation instead of incremental, select the **Absolute** check box.
 9. Click **Apply**.
 10. To repeat a transformation, deselect **Absolute**, then continue clicking **Apply**.
ADDITIONAL INFORMATION: To set the model to its original transformation click **Reset**.
 11. Click **Close** to close the Transformation dialog box.
-

RESULT:

The transformation state remains displayed.

NOTE: To restore the default state of a part of the model, from the **Manipulate** menu, select **Transform**, and then select **Reset Selected**. To restore the default state for the whole model, select **Reset All**.

Sectioning

With the Sectioning feature, you can view the cross-section and cut-through of 3D models. In the Define Section dialog, you can define the section plane position, section plane orientation, and the cut-through.

Section Plane Options

In the Define Section dialog, you can use the following options to define the orientation of the section plane:

Option	Description
XY Plane	Section plane is oriented along the XY plane.
YZ Plane	Section plane is oriented along the YZ plane.
XZ Plane	Section plane is oriented along the XZ plane.
From 3 Vertices	Select three vertices on the object to define the orientation of the section plane.
From Face Normal	Select a face on the object to define the orientation of the section plane perpendicular to the face.
From Edge Tangent	Select an edge on the object to define the orientation of the section plane perpendicular to the edge's tangent.
Define a Plane	Define X, Y and Z coordinates to orient the section plane.

Cut Options


In the Define Section dialog, you can use the following cut options to define the cut-through of an object:

Option	Description
Don't Cut	Display the object without a cut.
Cut	Object displayed is cut along the section plane.
Invert	Reverse the selection, display the other part of the object.
Show Both	Restore the cut part of the object.
Edges Only	Display only the edges of the object along the section plane.

Defining the Section Plane and Cut-through

TASK

1. From the **Manipulate** menu, select **Section**, and then select **Define**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Sectioning** .

STEP RESULT: The Define Section dialog appears.

2. From the Section Plane list, select the orientation for the section plane.
3. From the Cut Options list, select the cut option that you want.
4. To define the plane position, click and drag the **Plane Position** slider to the position you want.
5. Select **Dynamic**, if you want the plane to move simultaneously with the slider.
6. Select **Show Plane**, if you want the section plane visible.
7. Select **Fill**, if you want the section plane filled.

ADDITIONAL INFORMATION: The fill color can also be set from the Configuration dialog.

STEP RESULT: The section plane is displayed as filled and the **Section Area** is calculated.

8. From the Measured Units list, select the unit in which you want to measure the section surface.
 9. Click **Close** to close the Define Section dialog.

ADDITIONAL INFORMATION: Once you have defined the section plane and closed the Define Section dialog, among other options, you can choose whether to enable, remove, or invert a defined section from the **Section** menu.
-


Exporting Section Edges

You can export a copy of the section plane and save it as a **.dxf** or **.dwg** file.

TASK

1. Define a section plane.
 2. From the **Manipulate** menu, select **Section**, and then select **Export**.

ADDITIONAL INFORMATION: If the Define Section dialog is already open, click **Export**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Sectioning** .

STEP RESULT: The Export Section As dialog appears.
 3. Enter a file name and select a directory location to export the file.
 4. From the Save As Type list, select whether to save the file as **.dxf** or **.dwg** format.
 5. Click **Save**.

ADDITIONAL INFORMATION: To remove a section plane, from the **Manipulate** menu, select **Section**, and then select **Remove**.
-



Exploding

You can use the Explode option to better understand the structure of an assembly and to analyze the dismount capability. You can also save the exploded view, as well as obtain a printed document of the exploded product for further analysis.

Explode Options

From the Explode dialog, you can define how the explosion is to be performed and to visually explode or implode a model.


The options that you can set are:

Option	Description
Maximum Depth	Define up to which level you want to explode. All entities from the root level to the specified level are exploded. All entities on the other levels will not be exploded. If you are exploding: Whole model - the number of available levels depends on how many explodable levels exist in the model. For example, if there are four levels, with level 2 being the first level under the main assembly, then level 3 and level 4 are added to the list. Selected parts - the number of available levels to be added to the list depends on the selected virtual tree.
Animated	When selected, animation is shown while exploding or imploding the model.
Show Arrows	When selected, arrows are displayed in an explosion assembly. The arrow starts from an exploded entity and points to the center point of its parent.
Scroll Explode	Dragging the slider gradually displays the progression of the explosion to show the explosion dynamics. Explode - pressing  brings the explosion level to the (n-1) level if current explosion level is between n-1 (inclusive) and n level (exclusive). Implode - pressing  brings the explosion level to the n level if current explosion level is between n-1 (exclusive) and n level (inclusive).

Exploding a 3D Model

TASK

1. From the **Manipulate** menu, select **Explode**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also select **Explode** .

STEP RESULT: The Explode dialog box appears.

2. Select the parts that you want to explode.

ADDITIONAL INFORMATION: To select more than one model part, press the **Shift** or **Control** key while selecting. To explode the entire model, ensure that no parts are selected.

3. From the Maximum Depth list, select the level to which you want the model to explode.

4. Select **Animated** if you want to see an animated view of the explosion.

ADDITIONAL INFORMATION: This option is selected by default.

5. Select **Show Arrows** if you want to display the arrows.

6. To explode the model, click the  button.


STEP RESULT: The whole model or the select model parts explode.

7. To implode the model, click the  button.

ADDITIONAL INFORMATION: To manually change the explosion state, click and drag the scroll bar.

8. Click **OK** to close the Explode dialog box.
-

RESULT:

The explode state remains displayed, and the **Transformation** column  in the Model Tree is updated indicating the exploded parts.

See “Resetting the Transformation of a 3D Model”

Saving an Exploded View of a 3D Model

From the Explode dialog box, you can save the exploded view of the 3D model.

TASK

1. Explode a 3D Model.

2. Click **Save View** in the Explode dialog box.

STEP RESULT: The Add User Defined View dialog box appears.

3. Enter a view name.

4. Click **OK**.

STEP RESULT: The view is saved and appears in the **Views** tab under the User Defined Views tree.

5. Click **OK** to close the Explode dialog box.

RESULT:

The exploded state is saved.

See “Resetting the Transformation of a 3D Model”

User-Defined Coordinate Systems

You have the option of creating customized three-axis coordinate systems. As with the **Global Axes** and the **World Coordinate System**, the user-defined axes are represented by a red X-axis, a green Y- axis, and a yellow Z-axis.

See “Global Axes”

When you set a User-Defined Coordinate System (UCS) as the active axes, operations such as measuring, transforming a model, or defining a view point are applied with respect to these axes.

You can create multiple UCS’s for a 3D file. For multiple-page 3D files, you can even set active a different UCS for each page.

Position Options

When defining a UCS, you can define the point where you want to place the user coordinate system.

The options are:

Position	Description
(x, y, z) Coordinates	If selected, enter a value to specify where the three axes are positioned.

Position	Description
Vertex	If selected, all vertices are highlighted on the model. A snap box appears when you move the cursor on a vertex. Click the vertex point where you want to place the user coordinate system. The x , y and z coordinate values for the vertex are displayed. The UCS axes move to the new position.
Edge	If selected, all edges are highlighted on the model. A snap box appears when you move the cursor near an edge. Click at a point on an edge where you want to place the user coordinate system. The x , y and z coordinate values for the edge point are displayed. The UCS axes move to the new position.
Midedge	If selected, all edges are highlighted on the model. A snap box appears when you move the cursor near the halfway point of an edge. Click at the mid-edge point where you want to place the user coordinate system. The x , y and z coordinate values for the mid-edge point are displayed. The UCS axes move to the new position.
Arc Center	If selected, all arcs and circles are highlighted on the model. A snap box appears when you move the cursor near the center of a circle or arc. Click on the arc center where you want to place the user coordinate system. The x , y and z coordinate values for the arc center are displayed. The UCS axes move to the new position.
Face	If selected, when you move the cursor along a model face, the face is highlighted. Click at a point on the face where you want to place the user coordinate system. The x , y and z coordinate values for the arc center are displayed. The UCS axes move to the new position.

Orientation Options

When defining a UCS, you can define the orientation for the coordinate system.

The options are:

Position	Description
(x, y, z) Coordinates	If selected, enter a value to specify the UCS orientation according to one or more of the three axes.
Edge Tangent	If selected, all edges are highlighted on the model. A snap box appears when you move the cursor near an edge. First select the axis that will determine the UCS orientation. Click at a point on an edge to define the orientation. The x , y and z values for the edge point are displayed. The UCS axes are reoriented to the selected edge.

Position	Description
Face Normal	If selected, when you move the cursor along a model face, the face is highlighted. First select the axis that will determine the UCS orientation. Click at a point on an face to define the orientation. The x , y and z values for the face point are displayed. The UCS axes are reoriented to the selected face.
2 Vertices	If selected, all vertices are highlighted on the model. A snap box appears when you move the cursor on a vertex. First select the axis that will determine the UCS orientation. Click two vertex points to define the orientation. The x , y and z values for the vertices are displayed. The UCS axes are reoriented to the selected vertices.
3 Vertices	If selected, all vertices are highlighted on the model. A snap box appears when you move the cursor on a vertex. First select the axis that will determine the UCS orientation. Click three vertex points to define the orientation. The x , y and z values for the vertices are displayed. The UCS axes are reoriented to the selected vertices.

Deleting a User Coordinate System

TASK

- From the **Analysis** menu, select **Set User Coordinate System**.
STEP RESULT: The User Coordinate Systems dialog appears.
- From the Coordinate System list, select the trihedral coordinate system that you want to delete.
- Click **Delete**.
ADDITIONAL INFORMATION: The UCS disappears from the list.
- Click **OK** to close the User Coordinate Systems dialog.

Modifying a User-Defined Coordinate System

TASK

- From the **Analysis** menu, select **Set User Coordinate System**.
STEP RESULT: The User Coordinate Systems dialog appears.
- From the Coordinate Systems list, select the UCS that you want to modify.
STEP RESULT: The settings you configured for the UCS are displayed.
- To change the name, highlight the name and enter the new one.

4. To change the position of the UCS, click the Position tab, then from the Define From list, select the point where you want the UCS axes to appear.
 5. To change the orientation, click the Orientation tab, then from the Define From list, select the point where you want to orient the UCS.
 6. To show the UCS axes, select **Show Trihedron**. To hide the UCS axes, deselect the check box.
 7. To change the UCS that it is relative to, select the UCS from the Relative To list.
 8. Click **OK** to close the User Coordinate Systems dialog.
-

RESULT: If you modified the position or orientation, the UCS axes dynamically move to the new position. A change in Show Trihedron option will make the UCS axes appear or disappear accordingly. Additionally, a name change will display the new name in the Coordinate Systems list.

Activating a Coordinate System

When you set a User-Defined Coordinate System (UCS) as the active axes, operations such as measuring, transforming a model, or defining a view point are applied with respect to these axes.

TASK

1. From the **Analysis** menu, select **Set User Coordinate System**.
STEP RESULT: The User Coordinate Systems dialog appears.
 2. From the Coordinate System list, select the trihedral coordinate system that you want to use.
 3. Click **Set Active**.
 4. Click **OK** to close the User Coordinate Systems dialog.
-

RESULT: The newly active trihedral coordinate system appears bolder and larger on the model.

Defining a User Coordinate System

TASK

1. From the **Analysis** menu, select **Set User Coordinate System**.
STEP RESULT: The User Coordinate Systems dialog appears.

2. Click **New**.

STEP RESULT: The name of the User-Defined Coordinate System (UCS1 for the first one) appears in the Coordinate Systems list.

3. To change the name that you want for the UCS, in the **Name** field, enter a new name.
4. Click the Position tab to define the origin of the UCS.
5. From the Define From list, select the point where you want the UCS axes to appear.
6. Click **Apply**.
7. Click the Orientation tab to define the orientation of the UCS.
8. From the Define From list, select the point where you want to orient the UCS.
9. Click **Apply**.
10. To hide the UCS axes on the model, deselect the **Show Trihedron** check box.

ADDITIONAL INFORMATION: This option is selected by default.

11. Click **OK** to close the User Coordinate Systems dialog.

ADDITIONAL INFORMATION: If there is more than one coordinate system for this file, select which coordinate system the new UCS will be relative to from the Relative To list. Otherwise, the new UCS is calculated with respect to the World Coordinate System.

RESULT:

The newly active trihedral coordinate system appears bolder and larger.

See “Modifying a User-Defined Coordinate System”

“Activating a Coordinate System”

Interference Checks

You can check for spatial interference of any two model parts.

Interference Check Options



With the **Interference Checking** option, you can perform different interference checks.

These checks are:

Check	Description
All Against All	Check for interference of any of the parts in the model.
Set Against the Rest	Check where a model part or set of model parts interfere with other model parts.
Set Against Itself	Check where a model part or set of model parts interfere with itself.
Set 1 Against Set 2	Check where a model part or set of model parts interfere with another model part or set of model parts.

Performing an Interference Check

TASK

- From the **Analysis** menu, select **Check for Interference**.
STEP RESULT: The Interference Checking dialog appears.
- From the Scope options, select the type of interference check that you want to perform.
- If you selected **Set Against the Rest** or **Set Against Itself**, select one or more model parts on the model or from the Model Tree.
ADDITIONAL INFORMATION: To select more than one model part, press the **Shift** or **Control** key while selecting.
ADDITIONAL INFORMATION: To clear the last set of selected model parts, click **Clear**.
STEP RESULT: The model parts appear in the Selected Set list and are highlighted on the model and in the Model Tree.
- If you selected **Set 1 Against Set 2**, click the **Set 1** arrow , then select one or more model parts on the model or from the Model Tree.
ADDITIONAL INFORMATION: To clear a selected part, press the **Control** key and click on the part on the model. To clear the last set of selected model parts, click **Clear**.
STEP RESULT: The model parts appear in the Set 1 list and are highlighted on the model and in the Model Tree.
- Click the **Set 2** arrow , then select one or more model parts on the model or from the Model Tree.
STEP RESULT: The model parts appear in the Set 2 list and are highlighted in a different color on the model and in the Model Tree.
- In the **Minimum Clearance Distance** field, enter a distance. The default value is 0.
ADDITIONAL INFORMATION: Adding a minimum clearance distance not only checks if the two model parts interfere, but also checks if they are located within a set minimum distance.
- From the Units list, select the unit for the clearance distance.

8. To configure the interference check results, click **Options**.
STEP RESULT: The Options dialog appears.
9. Click **Compute** in the Interference Checking dialog.
ADDITIONAL INFORMATION: Section plane is not taken into consideration when computing an interface check. When the process is complete, all the instances of interfering pairs appear in the Results list. The number of interfering pairs found appears in the Description box.
ADDITIONAL INFORMATION: To view a pair's interference information, click the corresponding Results line.
STEP RESULT: The **Compute** button changes to **Stop**. To stop the Interference Checking process at any point, click **Stop**. The information appears in the Description box and AutoVue zooms into the interfering area.
10. To save the results, click **Export**.
STEP RESULT: The Export Results dialog appears.
11. Specify the file name and the directory where you want to save the file.
12. Click **Save** to close the Export Result dialog.
ADDITIONAL INFORMATION: To perform another interference check click **Reset**, then click the type of interference check.
STEP RESULT: AutoVue saves the results in a **.csv** (Comma Separated Values) file.
13. Click **Close** to close the Interference Checking dialog.

Interference Check Results Options

With AutoVue, you can configure the results of an Interference Check.

The configuration options are:





Option	Select	Description
Actions	Zoom To	Once the interference check is computed and a result set is selected from the list of results, AutoVue zooms to the result.
	Show Intersection	Once the interference check is computed and a result set is selected from the list of results, AutoVue shows where the intersection of model parts occur.

Option	Select	Description
Selected Pair	Opaque	The selected pair from the list of results appear opaque.
	Transparent	The selected pair from the list of results appear transparent.
	Hide	The selected pair from the list of results appear hidden.
Other Parts	Transparent	The rest of the model appears transparent while the selected pair is displayed.
	Hide	The rest of the model is hidden while the selected pair is displayed.

- **Actions:** Provides a close-up look at how model parts interfere.
- **Selected Pair:** Configure if the selected pair from the Results list should display opaque, transparent or hidden.
- **The Rest:** Configure if the other parts of the model should display transparent or hidden while the selected pair is displayed.

Comparing 3D Files

AutoVue provides the ability to compare two 3D files or two sets of entities from the same file or different files and display color-coded comparative data. When you compare two files, AutoVue displays a **Compare Tree** and three windows, the first containing the base file, the second containing the file you compared the base against, and the third containing the comparison results.

The Compare Tree displays the model's hierarchy with a State column displaying different icons representing the results of the file comparison. The icons indicate if an model part has been added , modified , moved  deleted .



In the **Comparison Result** window, results are displayed in different colors to differentiate the results of the file comparison. The comparison options and corresponding colors are:

Option	Color	Description
View Additions	Green	Indicates that something has been added.
View Deletions	Red	Indicates that something has been deleted.
View Unchanged	Blue	Indicates that there is no change.

Comparing 3D Files

The following steps describe how to compare files:

TASK

1. View the base file.
2. From the **Analysis** menu, select **Compare**.
STEP RESULT: The Open dialog appears.
3. Enter the file name or browse to locate the file you want to compare with the current active file.
4. Click **Open**.
STEP RESULT: AutoVue displays the Compare Tree and three windows. The first window displays the base file, the second displays the compare file, and the third displays the comparison results.
5. To view the properties of a modified or moved entity in the base file and in the compare file, select the modified  or moved  entity from the tree, then right-click and select **Entity 1 Properties**.
STEP RESULT: The Entity Properties dialog appears displaying the properties of the entity in the base file.
6. Select the entity again, then right-click and select **Entity 2 Properties**.
STEP RESULT: The Entity Properties dialog appears displaying the properties of the entity in the compare file.
7. To compare the results of the entity in the base file to the compare file, select the entity from tree, then right-click and select **Compare Results**.
STEP RESULT: The Compare Results dialog appears.
8. To view the attribute differences, click **Attributes**.
9. To view the transformation difference, click **Transformation**.
ADDITIONAL INFORMATION: If there are no attribute or transformation differences, the buttons are disabled.
10. Click **OK** to close the Compare Results dialog.

11. To exit Compare mode, from the **File** menu, select **Exit Compare**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also select **Exit Compare** .

RESULT:

The base file appears in the workspace.

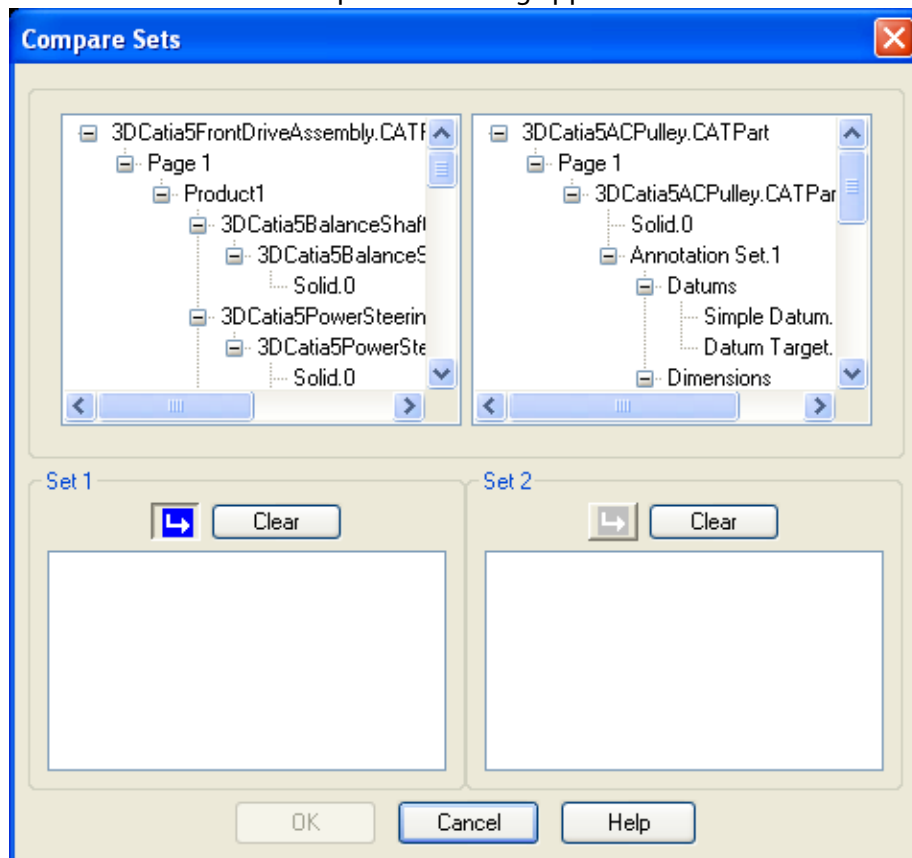
Comparing Entity Sets

From the file comparison, you can compare a set of entities from one file against a set of entities from the another file. You can also compare sets of entities from the same file.

TASK

1. Select the files you want to compare.
2. From the **Analysis** menu, select **Compare Sets**.

STEP RESULT: The Compare Sets dialog appears.



3. Click  **Set 1**.

4. Select the entities from either the tree on the left or the tree on the right.

ADDITIONAL INFORMATION: The tree on the left is the base file and the tree on the right is the compare file.

STEP RESULT: The selected entities appear in the Set 1 list.

5. Click  **Set 2**.

6. Select the entities from the other tree that you did not select for Set 1.

ADDITIONAL INFORMATION: To compare entities from the same file, select the entities for Set 1 and Set 2 from either the base file or the compare file.

STEP RESULT: The selected entities appear in the Set 2 list.

7. Click **OK**.

STEP RESULT: The Compare Tree and the three windows are updated with the set comparison results.

8. To restore the comparison files, from the **Analysis** menu, select **Compare Files**.
-

RESULT:

The files appear in the three windows.

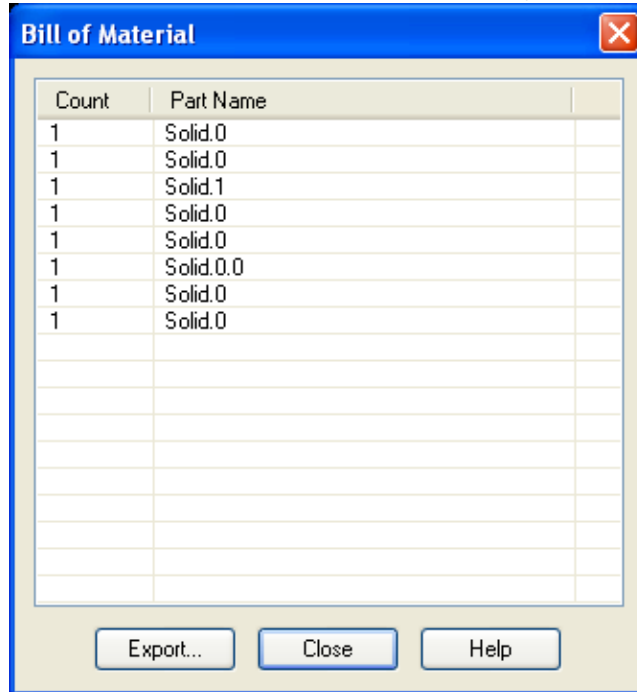
Generating a Bill of Material

For 3D files, you can obtain a list with a count of the parts required for manufacturing the item featured in the file. To generate a *Bill of Material* (BOM):

TASK

1. From the **Analysis** menu, select **Generate Bill of Material**.

STEP RESULT: The Bill of Material dialog appears.



2. To sort the list numerically, click the Count column header.
3. To sort the list alphabetically, click the Part Name column header.
4. To view a part or parts on the model, select the parts from the Part Name column.

ADDITIONAL INFORMATION: To select more than one part, press the **Shift** or **Control** key while selecting.

STEP RESULT: The parts are highlighted on the model and in the Model Tree.

5. To save the Bill of Material, click **Export**.

STEP RESULT: The Export BOM dialog appears.

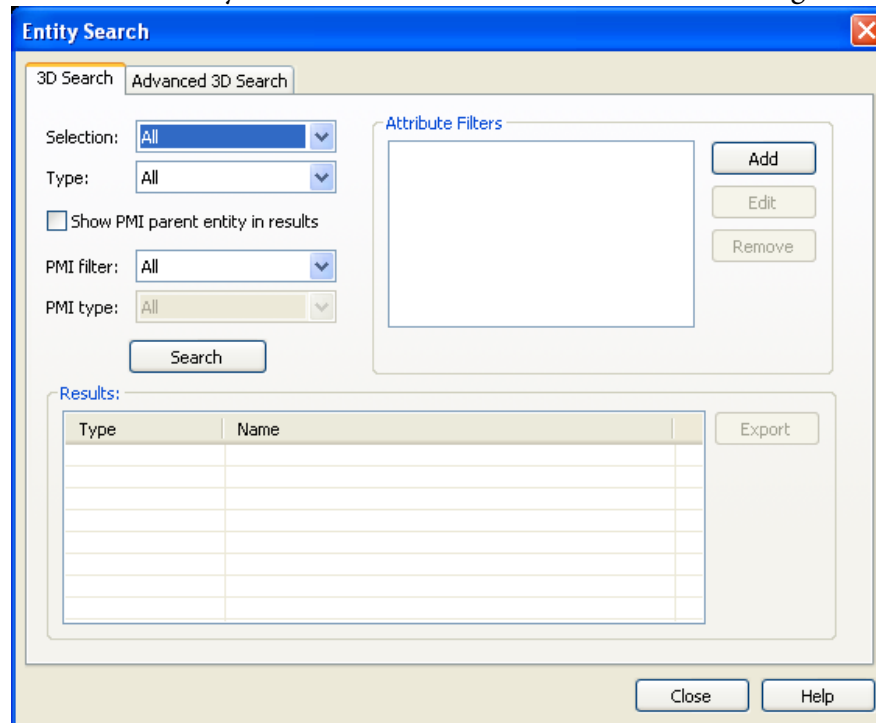
6. Enter a file name or browse to locate the directory where you want to store the file, then click **Save**.

STEP RESULT: AutoVue saves the results in a **.csv** (Comma Separated Values) or **.xml** file.

7. Click **Close** to close the Bill of Material dialog.
-

Entity Search

AutoVue allows you to search for entities in a 3D model using the Entity Search dialog.



You can search the whole model or selected entities on the model, perform a 3D text search, apply attributes, entity type or PMI filters as search criteria to filter entities. You can also search for entities by the size of an entity and location, or select entities from the results list and have them appear highlighted in the workspace and in the Model Tree.


The search options are:

Where	Option	Description
3D Search tab	Selection	Specify if you want to search the entire model or selected parts or parts that are not selected.
	Type	Specify the type of entity you want to search. For example, part, body, assembly or PMI.
	Show PMI parent entity in results	When selected, displays only the parent entity of a PMI entity that matches search criteria. If cleared, displays PMI entities that match search criteria in the Results dialog. Option is enabled if Type is set to PMI or All and file has PMI entities.
	PMI Filter	Filters the entities that have PMI information. Option is enabled if Type is set to PMI or All .
	PMI Type	Specify the specific PMI attribute you want to search. Option is disabled if PMI Filter is set to All .
	Containing Text	Specify a text string that you want to search. AutoVue searches for a specified string in attribute names and attribute values.
	Attribute Filters	Specify attributes to search for in 3D model. See “Performing an Attribute-based Search”
Attribute Filter dialog	Name	Specify the specific attribute you want to search. Some of the available attributes: Color Density Display_Mode Filepath Layer_ID Name Translucency Visibility <i>NOTE: The attributes vary depending on the file being viewed.</i> See “Performing an Attribute-based Search”
	Any Value	When selected, AutoVue searches for any value of the selected attribute. The attribute value options are disabled. If cleared, specify the values for the selected attribute you want to search. The attribute value options correspond to the selected attribute.

Where	Option	Description
Advanced 3D Search tab	Volume	Specify the size of an entity, the minimum and maximum dimensions. The dimensions of an entity boundary box must be between the specified dimensions to meet the search criteria.
	Location	Specify an axes aligned boundary box. The boundary box of an entity must be inside it to meet the search criteria.

Performing a Search

TASK

- From the **Edit** menu, select **Entity Search**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Entity Search** .
STEP RESULT: The Entity Search dialog appears.
- Click the **3D Search** tab.
- From the Selection list, specify what parts of the model that you want to search.
 - To search the entire model, select **All**.
 - To search specific parts of a model, select **Selected** and then select the parts on the model.
 - To search the model parts that are not selected, select **Not selected**.
- From the Type list, select the type of entity that you want to search.
ADDITIONAL INFORMATION: For 3D Models that have PMI information, the PMI search features are enabled.
- Select the **Show PMI parent entity in results** check box if you only want the parent entity of the selected PMI entity or entities to appear in the Results list.
- Deselect the check box if you want all PMI entities to appear in the Results list.
ADDITIONAL INFORMATION: The option is enabled if **PMI** or **All** are selected from the Type list and the file has PMI entities.

7. From the PMI Filter list, select the type of entities that you want to search.
 - To search all entities, select **All**.
 - To search entities with PMI, select **With PMI**.
 - To search entities without PMI, select **Without PMI**.

8. From the PMI Type list, select the type of entity that you want to search.

ADDITIONAL INFORMATION: The PMI Type list is disabled if **All** is selected from the PMI Filter list.

9. Click **Search** to perform the entity search.
-

RESULT:

The search results appear in the Results list.

Performing an Attribute-based Search

From the Entity Search dialog, you can search for entities by their attributes, such as color, density, shading, and so on.

TASK

1. From the **Edit** menu, select **Entity Search**.

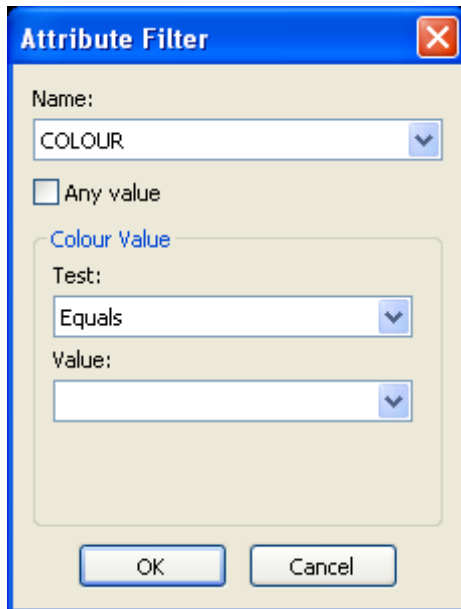
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Entity Search** .

STEP RESULT: The Entity Search dialog appears.

2. Click the **3D Search** tab.
3. Select the search criteria that you want to use for your search.

4. Click **Add**.

STEP RESULT: The Attribute Filter dialog appears.



5. From the Name list, select the attribute that you want to search.
STEP RESULT: The Value list appears corresponding to the selected attribute.
6. Select the **Any Value** check box if you want to search by any value; the associated value options are disabled.
7. Deselect the **Any Value** check box if you want to search by a specific value; from the Condition set condition for the search, and from the Values list select the values that you want to search.
8. Click **OK**.
ADDITIONAL INFORMATION: To add more than one Attribute Filter, repeat steps 4 to 8.
9. To edit an Attribute Filter, select the filter and click **Edit**.
STEP RESULT: The Attribute Filter dialog appears allowing you to edit the filter.
10. To disable an Attribute Filter, deselect the check box beside the filter or filters that you want to disable.
11. To enable an Attribute Filter, select the check box beside the filter or filters.
12. To remove an Attribute Filter, in the Entity Search dialog, select the filter and click **Remove**.
STEP RESULT: The Attribute Filter disappears from the list.
13. In the Entity Search dialog, click **Search**.
STEP RESULT: A list of entities matching the search criteria appear in the Results list displaying the name and type of the entities.
14. Click **Close** to close the Entity Search dialog.

Performing an Advanced 3D Search

From the Entity Search dialog, you can search for entities using volume or location filters. *Volume* specifies the size of the entity: you can specify the minimum and maximum dimensions an entity must be between. *Location* specifies an axes aligned boundary box: you can specify the dimensions for an axes aligned boundary box that an entity must be inside.

TASK

1. From the Entity Search dialog, click the **Advanced 3D Search** tab.

The screenshot shows the 'Entity Search' dialog box with the 'Advanced 3D Search' tab selected. The dialog is divided into two main sections: 'Volume' and 'Location'. Both sections have an 'Enabled' checkbox. The 'Volume' section includes 'Minimum Dimensions' and 'Maximum Dimensions' fields for X, Y, and Z, a 'Completely contained' checkbox, and a 'Units' dropdown menu. The 'Location' section includes 'Point 1' and 'Point 2' fields for X, Y, and Z, a 'Fits completely' checkbox, and a 'Units' dropdown menu. At the bottom are 'Close' and 'Help' buttons.

Section	Field	Value
Volume	Enabled	<input type="checkbox"/>
	Minimum Dimensions X	0
	Minimum Dimensions Y	0
	Minimum Dimensions Z	0
	Maximum Dimensions X	1
	Maximum Dimensions Y	1
	Maximum Dimensions Z	1
Completely contained	<input checked="" type="checkbox"/>	
Units	in.	
Location	Enabled	<input type="checkbox"/>
	Point 1 X	0
	Point 1 Y	0
	Point 1 Z	0
	Point 2 X	1
	Point 2 Y	1
	Point 2 Z	1
Fits completely	<input checked="" type="checkbox"/>	
Units	in.	

2. To search for entities by **Volume**, select the **Enabled** check box.
ADDITIONAL INFORMATION: If you want to search for entities that match all three dimensions, select the **Completely contained** check box. If the check box is deselected, then an entity only has to match one dimension in order to appear in the Results list.
3. Enter the minimum and maximum dimensions in their respective fields.
4. From the Units list, select the unit you want to set for the **Volume** dimensions.
5. To search for entities by location, select the **Enabled** check box.
ADDITIONAL INFORMATION: If you want to search for entities that match all three dimensions, select the **Fits completely** check box. If the check box is deselected, then an entity only has to match one dimension to appear in the Results list.
6. From the Units list, select the unit you want to set for the Location dimensions.
7. Enter the dimensions for **Point 1** and **Point 2**.
8. Click the 3D Search tab.

9. Click **Search**.

STEP RESULT: A list of entities matching the search criteria appears in the Results list.

10. Click **Close** to close the Entity Search dialog.

Saving Search Results

TASK

1. Perform a 3D entity search.

2. Click **Search**.

STEP RESULT: A list of entities matching the search criteria appear in the Results list displaying the type and name.

3. To save the results, click **Export**.

STEP RESULT: The Save As dialog appears.

4. Specify the path where you want to store the file and enter the file name, then click **Save**.


RESULT:

AutoVue saves the results in a **.csv** (Comma Separated Values) file.

Measuring in 3D Files

AutoVue provides the ability to perform measurements in 3D files. When measuring, AutoVue provides the option to “snap” to different entity types on the model.

From the **Analysis** menu, select **Measure** to access the Measurement options.

NOTE: From the AutoVue toolbar, you can also click **Measure** .

The following table outlines the available measuring options:




Name	Description
Angle	Measure the precise angle between any three vertices or any two edges, planes, faces or any combination of these entity types.

Name	Description
Arc	Measure the precise radius, length and angle of any arc and calculate the center point.
Distance	Measure the precise distance between any two vertices, edges, mid-edges, arc centers, faces or any combination of these entity types.
Minimum Distance	Measure minimum distance between any two vertices, edges, mid-edges, arc centers, faces or any combination of these entity types.
Edge Length	Measure the precise length of a edge.
Face Surface	Measure the precise surface area.
Vertex Coordinates	Provide the coordinates of each vertex.

3D Snapping Modes

The *snapping modes* available allow you to select or snap to different entity types on a model. For example, if you select **Vertex**, all vertices are highlighted and when you move the cursor over a vertex, a snap box appears.

The snapping modes allow you to snap to vertices, edges, faces, planes and arcs.




Button	Snap to	Description
	Edge	Edges are highlighted on the model. When moving the cursor over an edge, a snap circle appears.
	Face	Faces are highlighted when you move the cursor over a face and a snap triangle appears.
	Vertex	Vertices are highlighted on the model. When moving the cursor over a vertex, a snap box appears.

Measuring Distance

Use the *Distance* option to measure the precise distance between two vertices, edges, mid-edges, arc centers, faces or any combination of these entity types. When measuring the

distance between faces, if the faces are parallel, AutoVue compares the shortest distance between the faces.

TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog box appears.
 2. Click the **Distance** tab.
 3. From the Measured Distance Units list, select the unit in which you want to measure.
 4. Click the **From**  field to select the entity type that you want to measure from.
 5. Select the snapping modes that you want to use for measuring.
STEP RESULT: All entities of the selected entity type are highlighted on the model.
 6. On the model, select the entity you want to measure from.
ADDITIONAL INFORMATION: If you want to take more than one measurement from the same starting point, click **Fix Position**.
 7. Click the **To**  field to select the entity type that you want to measure to.
 8. Select the snapping modes that you want to select as the end point for the measurement.
STEP RESULT: All entities of the selected entity type are highlighted on the model.
 9. On the model, select the entity you want to measure to.
ADDITIONAL INFORMATION: To take another measurement click **Reset**.
STEP RESULT: The points are joined by a line. The measured distance, Delta X, Delta Y and Delta Z appear in the Measurement dialog box.
 10. Click **Close** to close the Measurement dialog box.
-

Calibrating Distance

Calibrate the distance measurement.

TASK




1. Measure distance between two points or measure cumulative distance.
2. From the **Distance** tab, click **Calibrate**.
STEP RESULT: The Distance Calibration dialog appears displaying the measured distance
3. From the Units list, select the unit to which you want to calibrate the distance.
4. Select **Calibrate to** and enter a value if you want to calibrate the distance to a value.

5. Select **Set Factor** and enter a value if you want to calibrate the distance by a factor.
 6. Click **OK**.
STEP RESULT: The calibration result appears in the Measurement dialog.
 7. Click **Close** to close the Distance tab.
-

Measuring Minimum Distance

With the **Minimum Distance** option you can measure the minimum distance between model parts, as well as any two points from the selection sets: vertices, edges, mid-edges, arc axes, arc centers, faces or any combination of entity types.

TASK


1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
2. Click the **Min. Distance** tab.
3. Click  **Set 1**.
4. From the Snapping Mode section, select one of the following:
 - Select **Entity** if you want to measure the distance between model parts. Snapping modes are disabled.
 - Select **Geometry** if you want to measure the distance between entity types. Snapping modes are enabled.
5. If you selected **Entity**, select a part or parts on the model. If you selected **Geometry**, select the snapping modes that you want to use for measuring.
ADDITIONAL INFORMATION: To reset a set, click **Clear**. To clear items from a set, select the items and press the Delete key. To deselect a part or entity type on the model, press the Control key and left-click the part or entity type.
STEP RESULT: The model parts appear in the list under **Set 1** and are highlighted on the model and in the Model Tree. All entities of the selected entity type are highlighted on the model.
6. Click  **Set 2**.
7. Repeat step 5.
STEP RESULT: The model part appears in the list under **Set 2**.
8. From the Units list, select the unit in which you want to measure the distance.
9. Click **Compute**.
STEP RESULT: The minimum distance from the first set of entities to the second set is highlighted by a line. The measured minimum distance, X, Y, and Z coordinates for **Position 1** and X, Y and Z coordinates for **Position 2** appear in the Measurement dialog.

10. To take another measurement click **Reset**.
11. Click **Close** to close the Measurement dialog.

Measuring an Angle

Use the **Angle** option to measure the precise angle between any three vertices or any two edges, planes, faces or any combination of these entity types.


TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
2. Click the **Angle** tab.
3. Select the snapping modes that you want to use for measuring.
STEP RESULT: Entities of the selected types are highlighted on the model.
4. To measure the angle between an entity type and a plane, select the **With Plane** check box, and then select the plane from the list.
5. From the Units list, select the unit in which you want to measure the angle.
6. Click two points on the model to define the angle.
STEP RESULT: Angle arms appear with an arc connecting them.
7. Click again to complete the measurement.
STEP RESULT: The angle is highlighted and the value of the angle measurement appears in the dialog.
8. To take another measurement click **Reset**.

Measuring an Arc

Use the **Arc** option to measure the precise radius, length and angles of any arc on the model. You can also calculate the center point location.

TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
2. Click the **Arc** tab.

3. Select **Arc Entity** if you want to measure a predefined arc.
STEP RESULT: All arc and circles are highlighted on the model. Snapping modes are disabled.
 4. Select **From 3 Points** if you want to measure the arc between three points.
STEP RESULT: Snapping modes are enabled.
 5. Select the snapping modes that you want to use for measuring.
 6. From the Units list, select the unit in which you want to measure the arc distance.
 7. If you selected **From 3 Points**, click three points to define the arc.
STEP RESULT: The points are joined by an arc. The measurements for center point coordinates, radius, diameter, arc length, start and end of angle and sweep appear in the Measurement dialog.
 8. If you selected **Arc Entity**, click the edge of the arc that you want to measure.
ADDITIONAL INFORMATION: To take another measurement click **Reset**.
STEP RESULT: The arc is highlighted. The measurements for center point coordinates, radius, ratio, arc length, start and end of angle and sweep appear in the Measurement dialog.
 9. Click **Close** to close the Measurement dialog.
-

Calibrating an Arc


TASK

1. Measure an arc in the drawing.
 2. From the **Arc** tab, click **Calibrate**.
STEP RESULT: The Radius Calibration dialog appears displaying the measured distance.
 3. From the Units list, select a unit of measurement to which you want to calibrate the distance.
 4. Click **Calibrate to** and enter a value if you want to calibrate to a value.
 5. Click **Set Factor** and enter a value if you want to calibrate by a factor.
 6. Click **OK**.
STEP RESULT: The calibration results appear in the Arc tab.
 7. Click **Close** to close the Measurement dialog.
-

Measuring Vertex Coordinates

The **Vertex Coordinates** option provides the coordinates of vertices on the model.


TASK

1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
 2. Click the **Vertex** tab.
STEP RESULT: All vertices on the model are highlighted.
 3. From the Units list, select the unit in which you want to measure.
 4. Move the cursor over the highlighted vertex that you want to measure.
STEP RESULT: The **X**, **Y** and **Z** coordinates appear in a tooltip.
 5. Select the vertex.
ADDITIONAL INFORMATION: To take another measurement click **Reset**.
STEP RESULT: The **X**, **Y** and **Z** coordinates appear in the Measurement dialog.
 6. Click **Close** to close the Measurement dialog.
-

Measuring the Length of an Edge

Use the **Edge Length** option to measure the length of any edge on the model.

TASK


1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
 2. Click the **Edge Length** tab.
STEP RESULT: All edges on the model are highlighted.
 3. From the Units list, select the unit in which you want to measure the edge length.
ADDITIONAL INFORMATION: If you want to cumulate the measurement of more than one edge, select **Cumulative**.
 4. Click the edge that you want to measure.
ADDITIONAL INFORMATION: To take another measurement click **Reset**.
STEP RESULT: The edge length measurement appears in the Measurement dialog.
 5. Click **Close** to close the Measurement dialog.
-

Measuring Face Surface

Use the **Surface** option to measure the surface area of an entity face or an entire entity on the model.

TASK

1. From the **Analysis** menu, select **Measure**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .

STEP RESULT: The Measurement dialog appears.

2. Click the **Surface** tab.
3. Select **Face Surface** if you want to measure the surface area of a face on an entity.
4. Select **Entity Surfaces** if you want to measure the surface area of an entire entity.
5. From the Units list, select the unit in which you want to measure the surface.

ADDITIONAL INFORMATION: If you want to cumulate the measurement of more than one surface, click **Cumulative**.

6. If you selected **Face Surface**, move the cursor on the model to highlight a face surface, then click the face surface that you want to measure.

STEP RESULT: The face surface is highlighted. The measured surface appears in the Measurement dialog.

7. If you selected **Entity Surface**, click the entity that you want to measure.

STEP RESULT: The entity, as well as all the faces belonging to the body are highlighted. The measured surface of the body appears in the Measurement dialog.

8. To measure the face surface of the entity, click on the highlighted body.

STEP RESULT: A list appears displaying the entity and its parents.

9. Select the entity or a parent from the list.

ADDITIONAL INFORMATION: To measure the face surface of a parent, select the parent from the list.

STEP RESULT: The selected entity is highlighted and the measured surface appears in the Measurement dialog.

10. To take another measurement click **Reset**.
 11. Click **Close** to close the Measurement dialog.
-

Configuring AutoVue

Use the configuration options to configure the AutoVue workspace for different groups of file formats or for all files in general. For example, you can set different background colors for EDA, 2D, 3D, or Office files. You can also set paths to locate external resources such as fonts, symbols, XRefs, or configure measurement options.

To access the configuration options, from the **Options** menu, select **Configure**. The Configuration dialog appears. To implement your changes and to close the Configuration dialog, click **OK**.

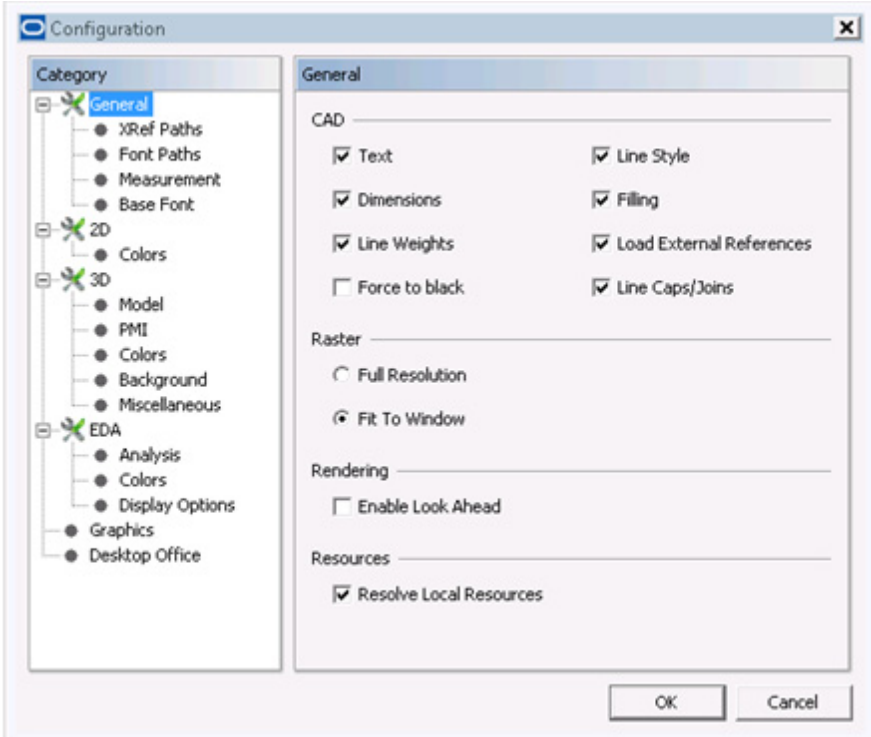
General Options

To access the **General** configuration options:

TASK

1. From the **Options** menu, select **Configure**.

STEP RESULT: The Configuration dialog appears.



2. From the Configuration tree, select **General**.

RESULT: The **CAD**, **Raster**, and **System Options** appear.

Configuring Options for CAD Files

Configure how you want to display text, dimensions, line styles, and so on for CAD files.

Option	Description
Text	Select to display text entities. Clear to hide text entities.

Option	Description
Dimensions	Select to display all dimensional entities. Clear to hide all dimension entities.
Line Weights	Select to display varying line thicknesses. Clear to make all lines appear equal, with a width of 1 pixel. No line weights display for any line.
Force to Black	Select to force all colors of a drawing to black. Clear to display the file in color.
Line Style	Select to display dotted and dashed lines. Clear to display all lines as solid.
Filling	Select to display filled entities as filled rather than just an outline. Clear to hide filling for filled entities.
Load External References	Select to display external references (XRefs) automatically. Clear to keep external references from displaying automatically.

Raster Files

Choose how you want the raster file to display:

Select **Full Resolution** to display raster files at full resolution.

Select **Fit to Window** to display raster files to fit the current window.

System Options

Select **Enable system tray hibernation** if you want the AutoVue icon to remain in the system tray even after you close AutoVue.

Double-clicking this icon lets you quickly reopen AutoVue, access recently-used files, and so on. To completely exit AutoVue when this option is selected, you need to right-click the AutoVue icon in the system tray and select **Close AutoVue**.

Deselect **Enable system tray hibernation** if you want AutoVue to quit completely when you close it.

Click **Registry** to open the AutoVue File Registration dialog box. From the list, select the file extensions you want to associate with AutoVue. Click **OK** to implement your changes and to close the dialog box.

Configuring Paths

Configure the paths for XRefs, fonts, symbols, or markups.

When working with files that need external resources, such as fonts or XRefs, you may need to specify the path to these external resources if they do not exist in the same location as the base file.

You may also need to access symbol libraries that are stored in a non-default location or you may need to configure from where AutoVue reads and locates Markup files.

Path	Description
XRefs	The directory paths for any external reference files associated with 2D, 3D or EDA files. See “Configuring XRef Paths”
Font	The directory paths for fonts required by AutoVue’s vector files. See “Configuring Font Paths”
Symbol	The directory path for symbol libraries used by AutoVue’s Markup entities. See “Configuring Symbol Paths”
Markup	The directory paths for Markup files. See “Configuring Markup Paths”

Configuring XRef Paths

XRef Paths are the directory paths for any external reference files associated with 2D, 3D or EDA files.

TASK

1. From the Configuration tree, expand **General**, and then select **XRef Paths**.
2. Click **Add**.
STEP RESULT: The Browse for Folder dialog box appears.
3. Enter the directory path, or scroll to locate the directory where the external reference files are located.
4. To browse all subdirectories below the current path, type two asterisks ** at the end of the file path. For example, **C:\samples**** will result in browsing all subdirectories below “samples”.
5. To browse one subdirectory below the current path, type one asterisk * at the end of the file path. For example, **C:\samples*** will result in browsing one subdirectory below “samples”.

6. Click **OK**.

ADDITIONAL INFORMATION: To add more than one path to the list, repeat steps 3 to 5.

STEP RESULT: The directory path appears.

7. To change the order, select the path you want to move, then click **Up** or **Down** to move the path to where you want it in the list.
 8. To remove a path, select the path and click **Remove**.
-

Configuring Font Paths

Font Paths are the directory paths for fonts required by 2D, 3D, EDA, or Office files.

TASK

1. From the Configuration tree, expand **General**, and then select **Font Paths**.
2. Click **Add**.

STEP RESULT: The Browse for Folder dialog box appears.

3. Enter the directory path or scroll to locate the directory where the external font files are located.
4. To browse all subdirectories below the current path, type two asterisks ** at the end of the file path.

ADDITIONAL INFORMATION: For example, **C:\samples**** will result in browsing all subdirectories below "samples".

5. To browse one subdirectory below the current path, type one asterisk * at the end of the file path.

ADDITIONAL INFORMATION: For example, **C:\samples*** will result in browsing one subdirectory below "samples"

6. Click **OK**.

ADDITIONAL INFORMATION: To add more than one path to the list, repeat steps 3 to 5.

STEP RESULT: The directory path appears.

7. To change the order, select the path you want to move, then click **Up** or **Down** to move the path to where you want it in the list.
 8. To remove a path, select the path and click **Remove**.
-


RESULT:

The selected path disappears from the list.

Configuring Symbol Paths

Symbol Paths are the directory paths for symbols libraries used by AutoVue's Markup entities.


TASK

1. From the Configuration tree, expand **General**, and then select **Symbol Paths**.
 2. Select the **Auto** option if you want to use the default directory to store the Symbol libraries.
 3. Select the **User-Defined** option if you want to select a directory to store the Symbol libraries. Enter the directory path where you want to store the Symbol files or click **Browse**  to locate the directory.
-

Configuring Markup Paths

Markup Paths are the directory paths for Markup files associated with the currently loaded document.

TASK

1. From the Configuration tree, expand **General**, and then select **Markup Paths**.
 2. Select the **Auto** option if you want to use the default directory to store the markup. AutoVue, by default, stores the Markup files in an **avred** subdirectory at the location of the base file.
 3. Select the **User-Defined** option if you want to select a directory to store the markup. Enter the directory path where you want to store the markup, or click **Browse**  to locate the directory.
-

RESULT: All markups will now be saved to and read from this new location.

Measurement

The **Measurement** options let you define the default measurement units and the number of decimal places.

From the Configuration tree, expand **General**, and then select **Measurement** to display the options you need.

The available options are as follows:

Option	Description
Significant Digits	Lets you specify the number of decimal digits displayed for each measurement. Enter a number from 1 to 18.
Default Units	Lets you specify the default font to use for AutoVue. If a file does not have native file units, the default file unit is used for the file.
Measurement Units	Lets you specify the default font to use for measurements. This list has the <i>Native File Unit</i> option in addition to the standard units. You can select Native File Unit if you wish to measure in the native units of the file.

Configuring the Base Font for Text Files

The Base Font option lets you define the font properties for text files.

TASK

1. From the Configuration tree, expand **General**, and then select **Base Font**.
2. Select a font from the Font list.
3. From the Font Style list, select **Regular**, **Bold**, **Italic**, or **Bold Italic**.
4. Select font size from the Size list.
5. From the Effects options, you can add a strikeout or underline to the text.

ADDITIONAL INFORMATION: The base font is previewed in the Sample area.

Configuring Streaming File Support

The streaming file is generated when AutoVue opens a native file for the first time. AutoVue generates the streaming file to decrease the loading time of files. As a result, when the file is opened subsequent times, AutoVue reads from the streaming file instead of the native file.

This option allows you to configure streaming file support.

NOTE: *If the native file changes, the streaming file will be invalid and AutoVue will open from the native file.*

TASK

1. Select the **Enable Streaming File Support** check box.

STEP RESULT: The Browse for Folder dialog box appears.

2. Browse and select a folder to use for storing streaming files and then click **OK**.

ADDITIONAL INFORMATION: The Enable Streaming File Creation check box is selected by default when you enable streaming file support.

STEP RESULT: The selected folder path appears in the Cache Folder field.

3. Select the **Enable Cache Size Limit** check box to limit the size of the streaming file cache folder (the default value is 0). Enter the cache size in the field to the right of the check box.
-

RESULT:

NOTE: The streaming file settings changes will take effect after the application is restarted.

Configuring AutoVue for 2D Files

You can configure background color, snap and overlay extents settings for 2D files.

TASK

1. From the **Options** menu, select **Configure**.

STEP RESULT: The Configuration dialog appears.

2. Select **2D** in the tree.

STEP RESULT: The **2D** options appear.

Snap Settings

In measurement mode, when you move the cursor within a predetermined snap radius, the snap box appears for the entity to be selected. To change the snap radius, change the value in the **Snap Radius** field. The snap radius is configured in pixels.

Configuring Colors

The **Colors** settings let you modify colors for 2D files.

From the tree, expand **2D**, and then select **Colors** to display the following options:

Option	Description
Background	Change the color of the background for 2D files.
Measurement	Modify the color you want displayed when taking measurements on 2D files.

Configuring AutoVue for 3D Files

There are configuration options that you can set to customize the work environment when working with 3D files.

TASK

- From the **Options** menu, select **Configure**.
STEP RESULT: The Configuration dialog box appears.
- Select **3D** in the tree.
STEP RESULT: The **Rendering** and **Dynamic Rendering** options appear.

Rendering

The **Rendering** options let you modify the manner in which the model is rendered.

Changing these options affects the level of detailed displayed. The **Rendering** options are:

Option	Description
Smooth Shading	Set by default. Turning this option off renders curved surfaces of shaded models as a series of flat surfaces. The level of detail is thus reduced but render speed is increased. Only affects shaded models.
Back-Face Removal	If selected, instructs AutoVue not to render the back faces of the model being displayed. This increases the render speed but the model appears less realistic while in motion. Only affects shaded models.

Option	Description
Precise Hidden Line	Instruct AutoVue to precisely calculate and render all hidden lines in a model. Selecting this option decreases render speed.

Dynamic Rendering

The **Dynamic Rendering** options let you select the rendering mode for a model in motion.

The options you can select are:

Menu Option	Description
Fast Frame	Model spins or rotates at a fast rate. The level of detail is reduced, which enables faster rendering when the model is in motion.
Wire Frame	Model is displayed as wire framed during rotation or spinning.
Flat Shading	Smooth shading is not performed on curved surfaces while the model is in motion.
Wire Polygons	Render the model in wire polygon mode while in motion.
Vertex Cloud	Model is displayed as a skeleton of vertices when in motion.
Bounding Box	Model parts are enclosed by bounding boxes when the model is in motion.
Current Render Mode	Model is rendered in the same mode whether moving or static.

Model

The **Model** configuration options let you control the streaming method, the dynamic load mesh resolution, and the visibility of model parts.

From the Category tree, expand **3D**, and then select **Model** to load the **Loading**, and **Default Mesh Resolution** options.

Loading

TASK

1. Select the **Load Mesh Data** option to load available mesh data.

2. Deselect **Load Mesh Data** to load BREP data.

ADDITIONAL INFORMATION: You need to reload the file for your changes to take effect.

Default Mesh Resolution

The **Default Mesh Resolution** slider lets you modify the number of polygons drawn when displaying a model.

Drag the slider between **Low**, **Medium** and **High** resolution levels.

A low resolution level means fewer polygons are drawn giving lower detail and higher render speed.

The **Default Mesh Resolution** slider only affects shaded models and wire polygon mode.

PMI Filtering

From the tree, expand **3D**, and then select **PMI** to display the PMI options.

The **PMI Filtering** options let you configure which types of product and manufacturing information to display. The check boxes in the **Tree** column let you select which PMI entity types display in the 3D model tree. The check boxes in the **View** column let you select which PMI entity types display in the workspace.

The following **PMI Text Rendering Style** options let you define the style for the PMI text:

Option	Description
Native Setting (from file)	PMI text displays with the default setting.
3D	PMI text displays in 3-dimensions. It may not always face you.
Flat-to-screen	PMI text always faces you.

Configuring Color

With the **Color** options, you can configure the color to allow easier viewing of 3D file details.

From the Configuration tree, expand **3D**, and then select **Colors** to view the available Color options. They are grouped under **Common**, **Section Highlight**, and **Geometry Highlight**, as follows:

Common

Option	Description
Background	Set the background color for all views (except the PCB 3D view) and file types.
Selection	Set the color when selecting an entity.

Section Highlight

Option	Description
Edges	Set the color for the section edges when defining Section Cut Options.
Fill	Set the fill color when defining Section Cut Options.

Geometry Highlight

Option	Description
Vertex	Set the color for vertices when highlighted during measurement and markup.
Face	Set the color for faces when highlighted during measurement and markup.
Edge	Set the color for edges when highlighted during measurement and markup.

Configuring Background

With the **Background** options, you can select a gradient or an image to display in the background. From the Configuration tree, expand **3D**, and then select **Background**. The Background Gradient and Background Images options appear.

For the Background Gradient option

Select either **Plain**, **Directional Gradient**, or **Radial Gradient**. The background is previewed to the rights of the options.

For the Background Images option

This feature adds a single or multiple background images to the workspace.

TASK

1. Click **Add**.
STEP RESULT: The Background Image dialog appears.
 2. Click on the ellipses to the right of the Image File field to browse to the image file.
ADDITIONAL INFORMATION: You can only select **.bmp**, **.jpeg**, or **.img** files.
 3. From the Stretch Type list, select whether to leave the image as is, **Stretch to fill**, **Zoom to fit**, or **Zoom to fill**.
 4. From the Position in the Background list, select where to position the image and then click **OK**.
 5. To add multiple background images, repeat steps 1 through 4.
 6. You can change the order of the background images. To do so, select a file from the Background Images list and then click **Up** to move an image to the front and **Down** to send an image to the back.
 7. To edit the background image, select the file from the Background Images list and then click **Edit**.
STEP RESULT: The Background Image dialog appears. Edit as required.
 8. To remove a background image, from the Background Images list, select the file and then click **Remove**.
 9. Click **OK** when done.
-

RESULT:

NOTE: The selected background image is implemented.

Miscellaneous

From the Configuration tree, expand **3D**, and then select **Miscellaneous** to display the Miscellaneous option:

Display

The **Show Global Axes** option is active by default. Disable this option to remove the axes that display in the bottom-right corner of the workspace.

You can change the size of the global axes by moving the Size of Axes slider. Move the slider to the left to minimize the axes and to right to maximize.

Model Tree

Enter a value in the **Expand Level** field to define the level at which you want the Model Tree to display when opening a 3D file. The default level is 3.

Alternately, select **Expand On Demand** to collapse the whole tree.

Manipulator

Enable the **Align with current UCS** option to automatically align the manipulators with the current User-defined Coordinate System.

Highlight Selection

The following **Highlight Selection** options let you define how AutoVue indicates that an object is selected.

Option	Description
Highlight Bounding Box	Selection is enclosed in a wireframe box.
Highlight Entity	Selection is indicated by changing color.

Configuring AutoVue for EDA Files

There are configuration options that allow you to customize your work environment when working with EDA files. To access the EDA configuration options, from the **Options** menu, select **Configure**. In the Configuration dialog that appears, select **EDA** in the tree.

Customizing Selections

Configure how you want selected components to be highlighted. From the Configuration tree, select **EDA**. There are two selection options: **Highlight Entity** and **Dim Unselected**.

Highlight Entity


Activate this option to highlight all entities that you select. This option is enabled by default.

NOTE: The default highlight color is yellow.

See Also “Modifying Colors”

Dim Unselected

Activate this option to dim all the entities that are not selected. Selected entities retain their original entity color.

From the AutoVue toolbar, you can also click **Dim Unselected** .

When you select Dim Unselected, you can set the dimness level for entities that are not selected. Drag the slider to the right to increase the dimness level and to the left to decrease the dimness level. The icon to the right of the dimness settings in the Configuration dialog lets you preview the dimness level.

Additionally, with Dim Unselected, you can select the **Thicken Highlighted Entity** check box. This option makes the selected entity appear more pronounced. Deselect the check box to return the entity to its default thickness.

Displaying Tooltips

When you hover the mouse over an entity, a tooltip appears to show you information about the entity. You can disable or enable these tooltips.

TASK

1. From the tree, select **EDA**.
 2. Under the **Mouse Hover** heading, select or deselect the **Show Entity Information Tooltip** option.
 3. Click **OK**.
-

Modifying 3D View

You can configure the board thickness and component height for the 3D PCB view of EDA files. Note that these Board Thickness and Component Height values are used only if the underlying EDA file does not contain board thickness and component height information.

TASK

1. From the Configuration tree, select **EDA**.
 2. Enter a value in the **Default Board Thickness** field to change the board's thickness.
ADDITIONAL INFORMATION: The **Default Board Thickness** option only affects boards that have no board thickness defined in the design. If a board has a defined thickness, this option does not affect that board.
 3. Enter a value in the **Default Component Height** field to change the component height.
ADDITIONAL INFORMATION: The **Default Component Height** option affects components that do not have height defined in the design. If a component has a defined height, this option does not affect that component.
 4. To work with a different unit of measurement, select another unit from the **Default Units** list. This is the unit of measurement used for values set in the **Default Board Thickness** and **Default Component Height** fields.
 5. Reload the file to see your changes.
-

Synchronizing Layers when Comparing Files

When you compare files, you can synchronize all layer settings. When the layer settings are synchronized, changing one setting changes the same setting for both files you are comparing.

NOTE: This option is selected by default and is only applicable for PCB drawings.

TASK

1. From the tree, expand **EDA**, and then select **Analysis**.
 2. Select the **Layer Settings** option to synchronize all layer settings when comparing files.
-

Configuring Zoom Behavior when Cross Probing

You can configure the zoom options when you cross probe files.

TASK

1. From the tree, expand **EDA**, and then select **Analysis**.
 2. Select one of the following **Cross Probe Action** options:
 - a Select **Keep Current Zoom Level**. When this option is active, the view of the target files remains the same while you cross probe.
 - b Select **Zoom Selected** to automatically zoom in on selected entities while you cross probe. This option is enabled by default.
 - c Select **Zoom Fit** to automatically adjust the contents of a file along the horizontal and vertical axis to fully display within the current window.
-

Modifying Colors

You can configure the colors for EDA files.

From the tree, expand **EDA**, and then select **Colors**.

In the Color Options dialog, the options that you can configure are grouped under **Common**, **PCB**, and **3D View** as follows:

Common

Option	Description
Background	Set the background color for all views (except the PCB 3D view) and file types.
Selection	Set the color when selecting an entity.

PCB

Option	Description
Min. Distance Set 1	Set color for the first minimum distance set.
Min. Distance Set 2	Set color for the second minimum distance set.
Plated Drill Hole	Set color for plated drill holes. The default color is black. To override the native color, select Override Native Color . <i>NOTE: Applicable to Cadence Allegro format only.</i>
Non-plated Drill Hole	Set color for non-plated drill holes. The default color is black. To override the native color, select Override Native Color . <i>NOTE: Applicable to Cadence Allegro format only.</i>

3D View

Option	Description
Default Board Color	Configure the color of the board in 3D view.
Default Component Color	Configure the component color in 3D view.

Enhanced Display Options

You can configure the following enhanced display options for EDA files.

Main Menu	Option	Description
Display Options	Plated Hole Visibility	Select whether to use the native settings or turn on /off the plated hole visibility. <i>NOTE: Applicable to Cadence Allegro format only.</i>
	Non-plated Hole Visibility	Select whether to use the native settings or turn on /off the non-plated hole visibility. <i>NOTE: Applicable to Cadence Allegro format only.</i>
	Pad Filling	Select whether to use the native settings or turn on/off pad filling. <i>NOTE: Applicable to Cadence Allegro format only.</i>
	Line Caps/Joins	Select whether to use the native settings or turn on/off line caps/joins. <i>NOTE: Applicable to Cadence Allegro format only.</i>
Global Transparency		Move slider to select global transparency level.

Configuring Background Colors for Graphic Files

Specify background colors for mono raster files and color raster files.

TASK

- From the **Options** menu, select **Configure**.
STEP RESULT: The Configuration dialog that appears.
 - Select **Graphics** in the tree.
 - From their respective Background lists, select colors for monochrome raster files and color raster files.
-

Configuring Background Colors for Desktop Office

Specify background colors for the following types of Desktop Office files: Document, Spreadsheet, Database and Archive.

TASK

1. From the **Options** menu, select **Configure**.
STEP RESULT: The Configuration dialog appears.
 2. Select **Desktop Office** in the tree.
 3. Select a background color for each Desktop Office file type.
-

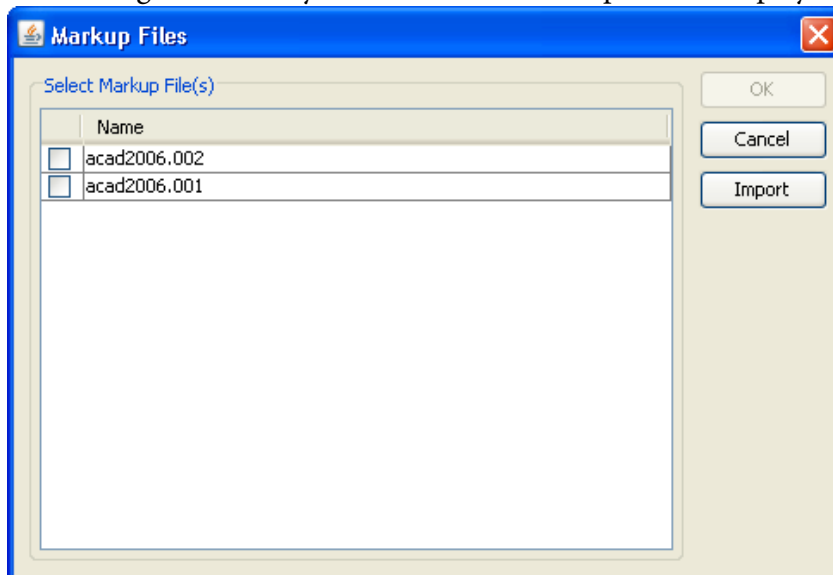
Markups

AutoVue has the ability to view hundreds of different file formats and to create markups for all its readable file formats without the document's authoring application.

Marking up refers to drawing and writing on an electronic document. When you create a markup for a file, the markup is created on top of the original file. Markups are saved in separate files.

***NOTE:** By default, markups are disabled for office formats. Office formats render differently on different machines due to differences in fonts. As a result, markups may appear misplaced with respect to the underlying text in the documents. If you want to re-enable markup creation for office formats, you must ensure that all machines viewing the office formats have the same fonts installed.*

If a file has existing Markup files, the **Markup Indicator** icon  appears in the status bar at the bottom of the AutoVue workspace. Clicking the Markup Indicator opens the Markup Files dialog and allows you to select the Markup files to display.



When you open a Markup file, AutoVue layers the markup over the original file.

In Markup mode you can:

- Create entities such as text, arcs, boxes, circles, clouds, lines, arrows, and polygons.
- Add a stamp or information to an entity by adding a text or a note.

- Create, name, and color layers to organize your work.
- Create a new Markup file that combines copies of selected layers of different Markup files.
- Create measurement markup entities that can be moved or resized.
- Navigate markups through a hierarchy tree, view markup properties, and sort the tree according to each property.
- View markup layers individually or in combination.
- Press **Esc** at any point to cancel the markup creation.

NOTE: By default, markups are disabled for Office documents. For information on how to enable markups for Office documents, refer to the “INI Options” chapter in the *Installation and Administration Manual*.

Markup Navigation Tree

When you are in Markup mode, a **Markup Navigation Tree** appears below the workspace. If the tree does not appear, from the **Options** menu, select **Show Panel**, and then select **Markup Panel**.

Markups					
Markup Entity	Author	Last Modified ▲	Page	Layer	
[-] Untitled2					
[-] Text	JohnDoe	04/09/2008 11:51:06 AM	1	0	
[-] Leader	JohnDoe	04/09/2008 11:51:08 AM	1	0	
[-] bnb	JohnDoe	04/09/2008 11:51:22 AM	1	0	
[-] Highlight	JohnDoe	04/09/2008 11:51:24 AM	1	0	
[-] Box	JohnDoe	04/09/2008 11:51:29 AM	1	0	
[-] Line	JohnDoe	04/09/2008 11:51:31 AM	1	0	

The tree displays a hierarchy of markups or comments created by users. You can navigate through the markups. A set of properties is generated for each markup. You can sort the markups in the tree according to each property by clicking the column headers. These properties are:

Property	Description
Markup Entity	Type of markup entity created.
Author	The name of the user who created the markup entity.
Last Modified	The date and time the markup entity was last modified.

Property	Description
Page	Page number of the original document on which the markup entity is created.
Layer	Markup layer on which the markup entity is created.


When a markup entity is created, it appears in the tree and the information is recorded and saved in the Markup file.

NOTE: Hover your mouse over an entity to display its author and date properties.

Filtering Markups

When viewing markups, you have the option to filter the Markup files or entities that are displayed based on their metadata information.

To do so, from the **Markup** menu, select **Filter** and then select one of the following options: By Author, By Entity Types, By Last Modified, By Page, and By Layer. The Filter Markup Visibility dialog appears.

*NOTE: From the Markup Properties toolbar, you can also click **Markup Filter** .*

- To display markups created by a specific author, click the **Author** tab and select the check box next to the author's name. To remove an author's markup from the filter, deselect the associated check box. In the Markup Navigation Tree, a filter icon appears in the Author column header.
- To display markups by entity type, click the **Markup Entity** tab and select the check box next to the markup entity or entities. To remove a markup entity from the filter, deselect the associated check box. In the Markup Navigation Tree, a filter icon appears in the Markup Entity column header.
- To display markups based on when they were last modified, click the **Last Modified** tab. From the list, select one of the following options:

Option	Description
Anytime	Displays all markup entities.
Before	Displays all markup entities modified before the selected date.
After	Displays all markup entities modified after the selected date.

Option	Description
On	Displays all markup entities modified on the selected date.
Between	Displays all markup entities modified between the selected dates.

In the Markup Navigation Tree, a filter icon appears in the Last Modified column header.

- To display markups by page location, click the **Page** tab and select one of the following:

Option	Description
All pages	Displays markup entities on all the pages.
Current page	Displays markup entities on the currently selected page.
Page range	Displays the markup entities on the selected page range.

- To display markups by layers, click the **Layer** tab and select the check box next to the layer or layers. To remove a layer from the filter, deselect the associated check box. In the Markup Navigation Tree, a filter icon appears in the Layer column header.

NOTE: To remove the filters, from the **Markup** menu, select **Filter**, and then select **Show All**.

Working with Markup Files

Markups can be saved as separate **Markup files** while working with the same document. You have the option to add user information to the Markup file, save and create separate Markup files with different markup IDs, import and export Markup files, or change the active Markup file.

Saved States

When you create and save a Markup file, the view state of the file is also saved. View states include zoom level (extents), rotation and flip settings, transformation, section plane, and visibility. When creating markups for 3D files containing imported models, the imported

design becomes part of the view state. For EDA files, you can save user-defined layers sets with markups.

See “Creating a User-Defined View”


State information is also saved with each markup entity. For instance, if you were at a certain zoom level when you created a markup entity, AutoVue saves the information with the markup entity. To “Go To” the state you were at when you created or modified the markup entity, from the Markup tree, right-click the markup entity and then select **Go To**.

Creating a Markup File

To create a Markup file:

TASK

1. View a file that you want to markup.
2. From the **Markup** menu, select **New**.

ADDITIONAL INFORMATION: From the Markup Properties toolbar, you can also click **New Markup** .

RESULT:

AutoVue enters Markup mode and a new Markup file appears in the Markup Navigation Tree.

See Also “Creating a Markup Layer”

Entering Markup Information

When creating a markup you can provide user information that you can save with the markup.

From the **Markup** menu, select **Properties**. The Markup Information dialog appears.

Enter a user name, department, company name, company location, and telephone number. Click **OK** to save your markup information and to close the dialog.

Saving a New Markup File

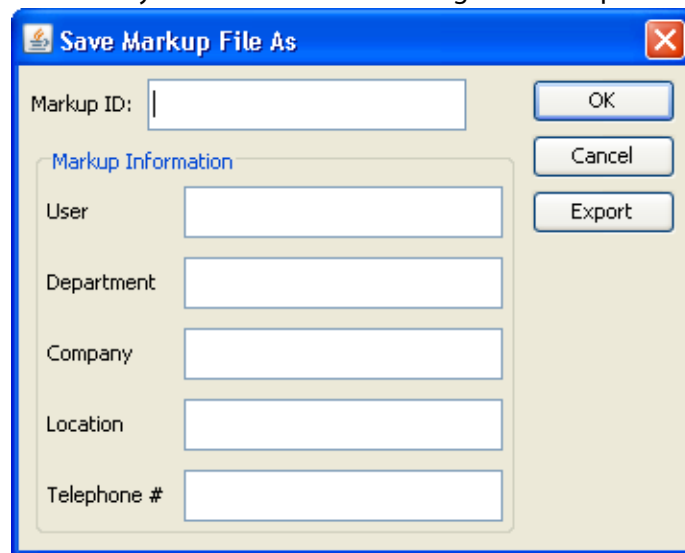
To save a new Markup file:

TASK

1. From the **Markup** menu, select **Save**.

ADDITIONAL INFORMATION: From the Markup Properties toolbar, you can also click **Save Markup** .

STEP RESULT: The Save Markup File As dialog appears displaying the Markup Information that you entered when creating the markup.



The image shows a 'Save Markup File As' dialog box. It has a title bar with a blue gradient and a close button (X) in the top right corner. Inside the dialog, there is a 'Markup ID:' label followed by a text input field. To the right of this field are three buttons: 'OK', 'Cancel', and 'Export'. Below the 'Markup ID' field is a section titled 'Markup Information' in blue text. This section contains five labels with corresponding text input fields: 'User', 'Department', 'Company', 'Location', and 'Telephone #'. The dialog box has a light beige background and a blue border.

2. In the **Markup ID** field, enter an ID consisting of any combination of characters or numbers.
3. Click **OK**.

RESULT: The Markup file is saved. The markups and the Markup file remain displayed in the workspace and in the Markup Navigation tree.



Opening Markup Files

To open a Markup file:

TASK

1. View a file that has existing markups.

2. From the **Markup** menu, select **Open**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Open Markup(s)** , or from the bottom-left corner of the status bar, you can click **Markup Indicator** .

ADDITIONAL INFORMATION: If the Markup Indicator icon does not display, then the file you opened has no existing markups associated to it.

STEP RESULT: The Markup Files dialog appears.

3. From the Markup list, select the check box next to Markup file you want to open.

ADDITIONAL INFORMATION: If you select more than one Markup file, the Active Markup menu lets you select which one of them is initially active. See “Setting the Active Markup File” for more information.


4. Click **OK**.

RESULT: The selected markup or markups appear in the workspace on top of the original file.

NOTE: When you select more than one Markup file, the markups display simultaneously.

Saving an Existing Markup File

To save a modified existing Markup file, from the **Markup** menu, select **Save**.

NOTE: From the Markup Properties toolbar, you can also click **Save Markup** .

If you have multiple markups open that you have modified and want to save them all, from the **Markup** menu, select **Save All**.


To save an existing markup as a new markup, from the **Markup** menu, select **Save As**.

Importing a Markup File

To import a Markup file on top of the original file:

TASK

1. From the **Markup** menu, select **Open**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Open Markup(s)** .

STEP RESULT: The Markup Files dialog appears.

2. Click **Import**.

STEP RESULT: The Select Markup File to Import dialog appears.

3. Navigate to the markup file that you want to import and select it.
 4. Click **Open**.
-


RESULT: The imported Markup file appears in the workspace on top of the original file.

Exporting a Markup File

To export a Markup file:

TASK

1. From the **Markup** menu, select **Save As**.

ADDITIONAL INFORMATION: For a new Markup file, you can also click **Save Markup** .

STEP RESULT: The Save Markup File As dialog appears.

2. Click **Export**.
STEP RESULT: The Save As dialog appears.
 3. Navigate to the location where you want to export the Markup file.
 4. Enter file name.
 5. Click **Save**.
-

RESULT:

The Markup file is exported to the selected directory.

The default format saved is *Markup Files (*.*)*, but you can select another one. In the Save as Type list, there are six formats to choose from:



- Markup Files (*.*)
- DXF Output (*.dxf)
- AutoCAD DWG (*.dwg)
- Microstation DGN Output (*.dgn)

Setting the Active Markup File

A file can have several Markup files. When you open several Markup files simultaneously, you can set one as the active Markup file. Any changes you make is applied to the current active markup.

TASK

1. From the **Markup** menu, select **Open**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Open Markup(s)** , or from the bottom-left corner of the status bar, you can click **Markup Indicator** . If the Markup Indicator icon does not display, the file you opened has no existing markups associated to it.

STEP RESULT: The Markup Files dialog appears.

2. From the Markup list, select the markup or markups you want to open.
3. From the Active Markup list, select the markup you want to make active.

ADDITIONAL INFORMATION: An active markup displays in bold in the Markup Navigation Tree.

4. Click **OK**.
-

RESULT:

The selected markups appear in the AutoVue workspace.

Changing the Active Markup File

When you have multiple Markup files open, you can change the active markup.

NOTE: An active markup displays in bold in the Markup Navigation Tree.

Take one of the following steps:

- In the tree, right-click the name of the Markup file you want to make active, then select **Set Active**.
- From the Markup toolbar, you can set the active markup by selecting a markup from the list on the Markup Properties toolbar.
- Take the following steps:

From the **Markup** menu, select **Set Active**. The Set Active Markup dialog appears. From the Select Active Markup list, select the markup you want to make active. Click **OK** to make the selected Markup file active.

Working with Markup Layers

Markup files can be divided into layers with each layer having its own unique name. You can create, name, and color layers to organize your work. For example, different colors can indicate time priorities and each layer can contain markups with a common purpose.

When working with markup layers, you can view layers individually or in combination, add, rename, or delete layers. You can also specify a different color for each layer.

NOTE: The default layer color is red.

See Also “Moving a Markup Entity to Another Layer”

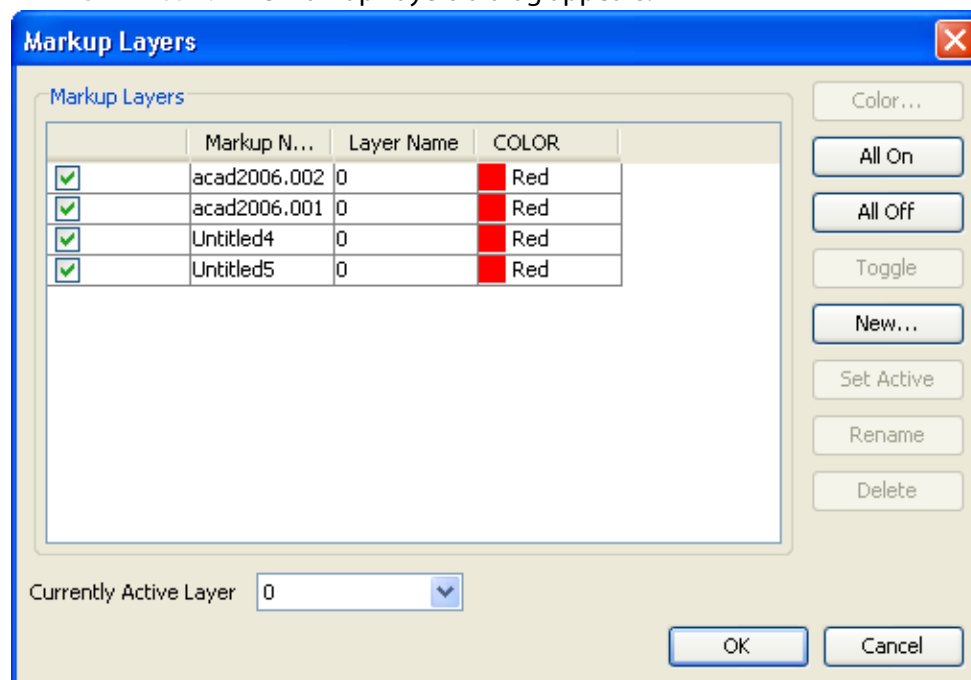
Creating a Markup Layer

From the Markup Layers dialog, you can create a markup layer.

TASK

1. From the **Markup** menu, select **Markup Layers**.

STEP RESULT: The Markup Layers dialog appears.



2. Click **New**.

STEP RESULT: The New Markup Layer dialog appears.

3. Enter a name for the markup layer.

4. Click **OK**.
STEP RESULT: The new markup layer is added to the list in the Markup Layers dialog.
5. To change the color of the new layer, click **Color**.
STEP RESULT: The Layer Color dialog appears. The default layer color is red.
6. Select a color and click **OK**.
7. Click **OK** to close the Markup Layers dialog.

Setting the Active Markup Layer

A markup can have several layers and you can set a layer as the active markup layer. When a markup layer is active, all modifications you make are applied to that markup layer. There are two ways in which to set a layer as active:

TASK

1. From the **Markup** menu, select **Markup Layers**.
STEP RESULT: The Markup Layers dialog appears.
 2. You can perform one of the following:
 - From the Markup Layers list, select the layer you want to make active, and click **Set Active**.
 - From the Currently Active Layer list, select the layer you want to make active.
 3. Click **OK**.
-



Changing the Color of a Markup Layer

After you create a markup layer, you have the option to change the color of its markup entities.

TASK

1. From the **Markup** menu, select **Markup Layers**. The Markup Layers dialog appears.
2. From the Markup Layers list, select the markup layer whose color you want to change.
3. Click **Color**.
STEP RESULT: The Layer Color dialog appears.
4. Select a **Color**.

5. Click **OK**.

ADDITIONAL INFORMATION: Only the entities that were created with the *Bylayer*  color defined in the Markup Layers dialog will change color. Any entities that were created using the color options from the Markup Properties toolbar, will override the Bylayer  color and will not change.

6. Click **OK** to close the Markup Layers dialog.
-

Renaming a Markup Layer

After you create a markup layer, you have the option to rename the layer.

TASK

1. From the **Markup** menu, select **Markup Layers**. The Markup Layers dialog appears.
 2. From the Markup Layers list, select the markup layer you want to rename.
 3. Click **Rename**.
STEP RESULT: The New Markup Layer dialog appears.
 4. Enter the new layer name.
 5. Click **OK**.
STEP RESULT: The markup layer is assigned the new name.
 6. Click **OK** to close the Markup Layers dialog.
-

Toggling between Markup Layers

From the Markup Layers dialog, you can choose to show or hide a layer and its associated markup entities.

TASK

1. From the **Markup** menu, select **Markup Layers**. The Markup Layers dialog appears.

2. To show/hide a layer or layers, perform one of the following:
 - a Select a layer and click **Toggle**. The layer's check box will switch between selected and deselected. To view all the markup layers, click **All On**. To hide all markup layers click **All Off**.
 - b From the Markup Layers list, select the check box next to the layer or layers you want visible, and deselect the check box next to the layer or layers you want to hide.
3. Click **OK**.

RESULT:

The markup entities belonging to the selected layer or layers appear in the workspace on top of the original file.

Deleting a Markup Layer

From the Markup Layers dialog, you can choose to delete a selected layer and its associated markup entities.

TASK

1. From the **Markup** menu, select **Markup Layers**.
STEP RESULT: The Markup Layers dialog appears.
2. From the Markup Layers list, select the markup layer you want to delete.
3. Click **Delete**.
4. Click **OK**.

RESULT:

The layer or layers are deleted along their associated entities.

Moving a Markup Entity to Another Layer

Once you create a markup entity, it is possible to assign it to a pre-existing markup layer.

TASK

1. Select the markup entity or entities that you want to move.
2. From the **Markup** menu, select **Format**, and then select **Move to Layer**.
ADDITIONAL INFORMATION: You can also right-click the markup entity, select **Format**, and then select **Move to Layer**.
STEP RESULT: The Move to Layer dialog appears.

3. From the Layers list, select the destination layer.
 4. Click **OK**.
-

RESULT:

The selected markup entity or entities are moved to the selected layer. Only markup entities using the ByLayer color will change to the new layer's color.

See Also “Working with Markup Entities”

Consolidating Markup Files

The **Consolidate** option allows you to create a new Markup file that combines copies of selected layers of different Markup files. During the review cycle, consolidation simplifies document revisions by providing the author with one combined Markup file instead of several Markup files. Note that the Consolidate option is only active when more than one Markup file is opened.

TASK

1. Open the Markup files that you want to consolidate.
2. From the **Markup** menu, select **Consolidate**.

STEP RESULT: The Consolidate Markups dialog appears.

3. Select the markup layers that you want to consolidate into one file. To select multiple layers, press the **Shift** or **Control** key while selecting.

ADDITIONAL INFORMATION: To select all the layers click **Select All**, and to deselect all the layers click **UnSelect All**.

4. In the **Markup ID** field, enter an ID for the new Markup file.
 5. If you want to open the newly consolidated markup as the active markup, select the **Open as Active Markup** check box.
 6. Click **OK**.
-

RESULT:

The consolidated Markup file is saved. If you selected **Open as Active Markup**, the consolidated markup opens and is set as the active markup.

Marking up 2D and 3D Files






AutoVue provides a number of markup options that display the same behavior when marking up 2D or 3D files. You can add markup entities such as attachments, hyperlinks, signoffs, and stamps.

See Also “2D-Specific Markups” for information on markup entities that are specific to 2D files.

See Also “3D-Specific Markups” for information on markup entities that are specific to 3D files.

NOTE: When you are creating a markup entity, you can press the **Escape** key to cancel.

The following table lists common markup entities for 2D and 3D files.

Option	Description
Attachment	Add an attachment entity to the markup. From the Markup Entity toolbar, you can also click Attachment  . See “Adding an Attachment”
Hyperlink	Click the workspace to attach a hyperlink. From the Markup Entity toolbar, you can also click Hyperlink  . See “Adding a Hyperlink”
Intellistamp	Add a stamp on a document in a connected and disconnected environment. Includes specific document and user information (metadata) pulled directly from the DMS/ERP/PLM/UCM system. From the Markup Entity toolbar, you can also click Intellistamp  . See “AutoVue Mobile” “Adding an Intellistamp”
Signoff	Create an approval stamp containing information about the markup author, date and time of creation. From the Markup Entity toolbar, you can also click Signoff  . See “Adding Signoff Entities”
Stamp	Click and drag a stamp from the dialog box to the workspace. From the Markup Entity toolbar, you can also click Stamp  . See “Adding a Stamp”

Adding an Attachment

You can attach a file of any type (for example, text, audio, or video) as a markup entity. The attachment is embedded into the markup entity and displays as an icon the display.

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Attachment**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Attachment**



2. Click on the drawing where you want to add an attachment.

STEP RESULT: The Attach File dialog box appears.

3. In the **Attachment Name** field, enter a name for the attachment. Optionally, you can also provide a brief description of the attachment in the **Description** box.

4. In the **Data File** field, enter the path to the file or click **Browse**. The Open dialog box appears.

5. Select the file and click **Open**.

ADDITIONAL INFORMATION: You can attach any local file.

6. Select **Open with AutoVue** to open the attachment. Deselect **Open with AutoVue** to open the file in its native program.

7. To display the attachment as an specific image, click **Set Icon**. The Open dialog box appears.

8. Select the image and click **Open**.

ADDITIONAL INFORMATION: You can only select bitmap images (.bmp).

9. Click **OK** to close the Attach File dialog box.

STEP RESULT: The attachment appears on the markup workspace and in the Markup Navigation Tree. If you selected **Set Icon**, the selected icon appears as the attachment on the markup.

10. Right-click in the workspace to complete the attachment.

ADDITIONAL INFORMATION: To open the attachment, double-click on it. It will open in the application selected in step 6.

Opening an Attachment

In the Markup Navigation tree, right-click the attachment entity, select **Attachment**, and then select one of the following:

- **Open:** If **Open With AutoVue** was selected while attaching the file, it will open in a new AutoVue window. Otherwise, it will in its native application.

- **Open With:** You have the option to open the attachment with **AutoVue, Desktop Version** or **Associated Application**.

Editing an Attachment

TASK

1. In the Markup Navigation Tree, right-click the attachment and select **Edit**, or double-click the markup entity.

STEP RESULT: The Attach File dialog appears.

2. Make your changes and click to **OK**.

STEP RESULT: The dialog closes and modifications are implemented.

Adding a Hyperlink

A hyperlink is a link between the current file and a new file or application. You can create hyperlinks in your current file so that your files and applications outside AutoVue are only a click away. The main benefit of adding hyperlinks is that the files are accessible from one location but the information is referenced, not duplicated. This ensures a manageable file size when loading. If changes need to be made to a linked file, they need to be done in one location—the linked file itself.

Creating a Hyperlink

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Hyperlink**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Hyperlink** .

2. Click a point on the document where you want to place the hyperlink.

STEP RESULT: The Establish Hyperlink dialog box appears.

3. Enter a **Link Name**.

4. Optionally, enter a **Link Description**.

ADDITIONAL INFORMATION: From the Link To list, select the type of link.

Link To	Description
Data File	Opens a document file.
Application	Opens an application other than AutoVue.
AutoVue Script	Runs the script command entered in the linked file.
DDE Server	Connects to a DDE server and executes command entered in linked file.
DLL Function	Evokes a function defined in the DLL.

STEP RESULT: The link changes according to the selected link.

- Click **Browse** to locate the data file or application and its path that you want to link to, or enter the information manually.

ADDITIONAL INFORMATION: For **Application**, ensure that the executable file is selected. For example, *mspaint.exe*.

- To change the hyperlink icon, click **Set Icon**. The Open dialog box appears.
- Enter the file name or browse to locate the bitmap file that you want to use as the icon, then click **Open**.
- Click **OK** to close the Establish Hyperlink dialog box.

ADDITIONAL INFORMATION: If you changed the icon, the selected icon appears as the hyperlink on the markup.

STEP RESULT: The hyperlink appears on the markup.

Opening a Hyperlink

To open a hyperlink, double-click on it from the workspace.

The hyperlink file opens in the window you selected in the Establish Hyperlink dialog.

Editing a Hyperlink

TASK

- In the Markup Navigation Tree or from the workspace, select the hyperlink that you want to edit.

2. From the **Markup** menu, select **Format**, and then select **Edit Hyperlink**.

ADDITIONAL INFORMATION: You can also right-click the hyperlink and select **Format**, and then select **Edit Hyperlink**.

STEP RESULT: The Establish Hyperlink dialog appears.

3. Edit the information that you want.
4. Click **OK** to close the Establish Hyperlink dialog.

RESULT:

The changes are saved.

Deleting a Hyperlink

TASK

1. In the Markup Navigation Tree or from the workspace, select the hyperlink that you want to delete.
2. From the **Markup** menu, select **Format**, and then select **Remove Hyperlink**.

ADDITIONAL INFORMATION: You can also right-click the hyperlink and select **Format**, and then select **Remove Hyperlink**. You can also delete the hyperlink by selecting the hyperlink and pressing the Delete key on your keyboard.

RESULT:

The hyperlink is deleted.


Adding Signoff Entities

The **Signoff** entity is an approval stamp containing information about the markup author, date, and time of creation. You create a signoff entity when the Markup file is finalized.

NOTE: If a markup is modified after a signoff is created, the signoff disappears (is rescinded) from the workspace but remains in the Markup Navigation tree. Double-click on the signoff entity in the tree to view the signoff history—the person who rescinded the signoff, the creation date, and the drop date.

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Signoff**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Signoff** 

2. Click and drag to create a box on the drawing where you want the signoff.
STEP RESULT: The Signoff dialog appears displaying details of the signoff.
 3. Click **OK**.
-

RESULT:


The signoff entity appears on the drawing and in the tree, and displays the author, creation date, and approval date.

Rescinding the Signoff

You can rescind a signoff on a drawing.

TASK

1. In the Markup Navigation tree or in the workspace, double-click the signoff markup entity.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Signoff** .

STEP RESULT: The Signoff dialog appears.

2. Click **Rescind**.
-

RESULT:

The signoff disappears from the drawing but remains in the Markup Navigation Tree.

Re-Approving a Signoff

To re-approve the signoff that has been rescinded:

TASK

1. In the tree, double-click the signoff markup entity. The Signoff dialog appears
 2. Click **Approve**.
-

RESULT:

The signoff entity re-appears on the drawing and the tree.

Viewing the History of a Signoff

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Signoff**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Signoff** .

2. Click **History**.

STEP RESULT: The Signoff History dialog appears displaying the author, date and time the signoff was created.

3. Click **Cancel** to close the Signoff dialog.

Adding an Intellistamp


The Intellistamp markup entity lets you create a stamp that includes document and user information (metadata) pulled directly from the backend DMS/ERP/PLM/UCM system. To add an Intellistamp markup entity to a document, do the following:

NOTE: With the Desktop Version, you can add an Intellistamp only when you are marking up a Mobile Pack, and only if the Mobile Pack was created from a backend system that contains an Intellistamp definition.

See “AutoVue Mobile”

TASK

1. Enter Markup mode.
2. From the **Markup** menu, select **Add Entity**, and then select **Intellistamp**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Intellistamp** .

3. In the workspace, click and drag a box to the desired Intellistamp size.

STEP RESULT: The Intellistamp dialog appears.

4. From the Choose Stamp box, select the desired Intellistamp.

ADDITIONAL INFORMATION: To resize the Intellistamp, select **Enable Resizing**.

5. Click **OK**.

STEP RESULT: The Intellistamp dialog closes and the Intellistamp appears in workspace and in the Markup Entity Tree.

6. Right-click the workspace to complete the Intellistamp placement.

Viewing/Modifying Intellistamp Attributes

You can view the attributes of an Intellistamp by double-clicking the Intellistamp markup entity. The DMS Attributes dialog appears listing all attributes of the Intellistamp.

NOTE: All attributes tagged hidden will also display in the Attributes dialog box.

The dialog box contains three columns that include attribute values defined in the design stage:

- The *Name* column displays the attribute name as defined in the design stage. It will either display the backend DMS/ERP/PLM/UCM system-defined attribute name or the user-defined name.
- The *Value* column displays the assigned value. If the attribute is not marked as *read-only* in the design stage, you can modify the value.
- The *Value from DMS* column displays the value assigned from the backend system.

The DMS dialog contains two buttons: **Commit Changes** and **Reset**:

- Click the **Commit Changes** button to upload the changes made to the DMS Attributes dialog to the Mobile Pack.
- Click the **Reset** button to reset the Intellistamp attribute values to those defined in the Mobile Pack.

When you are done modifying the attribute, click **OK** to finalize your changes and to close the DMS Attributes dialog.


Adding a Stamp

A Stamp is a graphical entity, such as a company logo. Before a graphic entity can be used as a stamp, it must be added to a Stamp Library. You can create a Stamp Library and add stamps to it. You can also add or remove stamps from existing libraries.

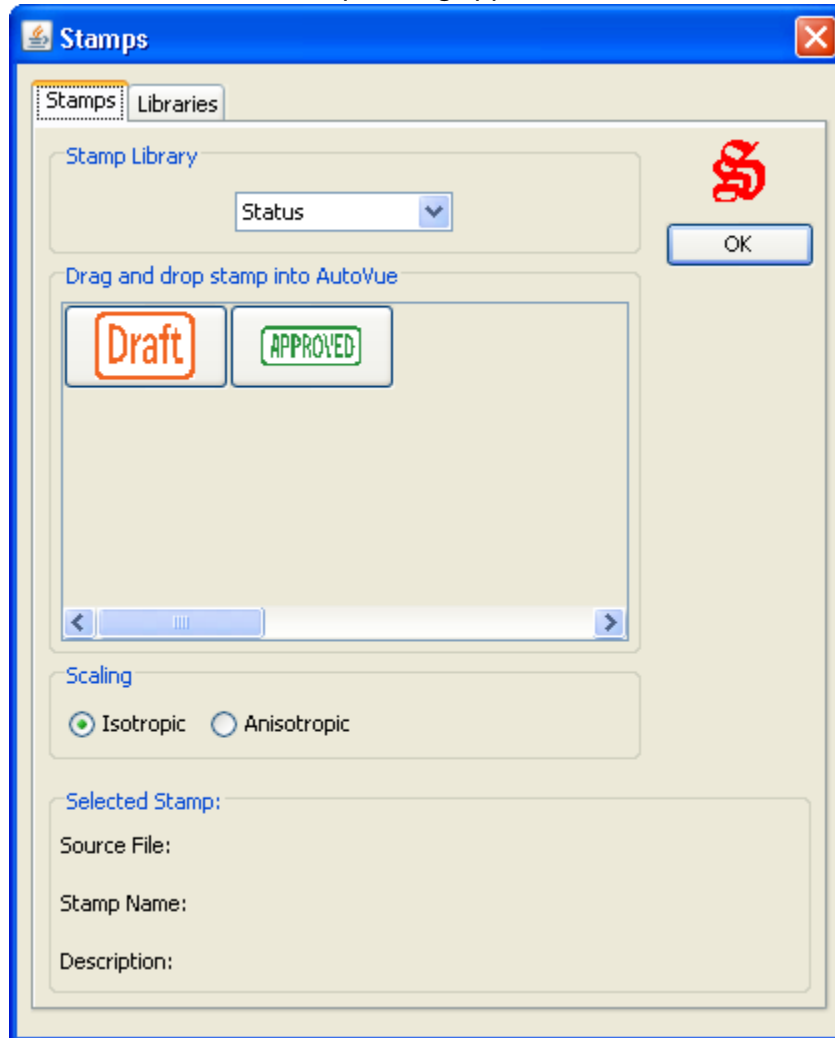
NOTE: If you need to add a stamp that includes any document or user information, you must use the *Intellistamp* markup.

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Stamp**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Stamp** .

STEP RESULT: The Stamps dialog appears.



2. Click the **Stamps** tab.
3. From the Stamp Library list, select the library from which you want to select a stamp.
STEP RESULT: The stamps appear for the selected library.
4. Click **Isotropic** if you want to scale the stamp proportionately.
5. Click **Anisotropic** if you want to scale the stamp disproportionately.

6. Drag the stamp onto the workspace.

ADDITIONAL INFORMATION: To add more stamps, repeat steps 3 to 5.

STEP RESULT: The stamp appears on the drawing and in the Markup Navigation Tree.

7. Close the Stamps dialog.
-

Creating a New Stamp Library

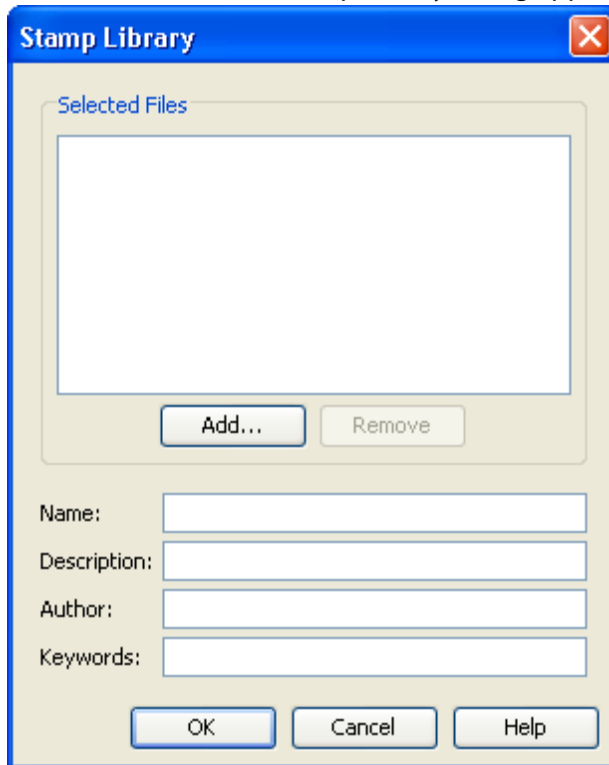
TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Stamp**. The Stamps dialog appears.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Stamp** .

2. Click the **Libraries** tab.
3. Click **Create**.

STEP RESULT: The Stamp Library dialog appears.



The image shows the 'Stamp Library' dialog box. It has a blue title bar with the text 'Stamp Library' and a red close button. Inside the dialog, there is a section titled 'Selected Files' with a large empty rectangular area below it. Below this area are two buttons: 'Add...' and 'Remove'. At the bottom of the dialog, there are four text input fields labeled 'Name:', 'Description:', 'Author:', and 'Keywords:'. Below these fields are three buttons: 'OK', 'Cancel', and 'Help'.


4. Enter the library name, description, author, and keywords in their respective fields.
5. To add a stamp to the library, click **Add**.

STEP RESULT: The Open dialog appears.

6. Browse to locate the stamp you want to add, and then click **Open**.
ADDITIONAL INFORMATION: To add more stamps, repeat steps 4 to 6. To remove a stamp, select the stamp and click **Remove**.
STEP RESULT: The file appears in the Stamp Files list.
7. Close the Stamps dialog.


Adding a Stamp to a Library

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Stamp**. The Stamps dialog appears.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Stamp** .
2. Click the **Stamps** tab.
3. From the Stamp Library list, select the library that you want to add a stamp.
4. Click **Add Stamp**.
STEP RESULT: The Stamp Definition dialog appears.
5. Enter the source file, stamp Name, and description in their respective fields, or click **Browse** to locate the source file.
6. Click **OK**.
ADDITIONAL INFORMATION: To add more than one stamp, repeat steps 4 and 5.
STEP RESULT: The stamp appears under the Drag and Drop a Stamp onto AutoVue section in the Stamps dialog.
7. Close the Stamps dialog.

Deleting a Stamp from a Library


TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Stamp**. The Stamps dialog appears.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Stamp** .
2. Click the **Stamps** tab.
3. From the Stamp Library list, select the library which contains the stamp you want to delete.
4. The stamps appear in the Drag and Drop a Stamp onto AutoVue section.

5. Select the stamp you want to delete, then click **Delete**.
STEP RESULT: The stamp disappears from the library.
 6. Close the Stamps dialog.
-


Editing Stamp Library Information

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Stamp**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Stamp** .
STEP RESULT: The Stamps dialog appears.
 2. Click the **Libraries** tab.
 3. From the Stamp Libraries list, select the library that you want edit.
 4. Click **Edit**.
STEP RESULT: The Stamp Library dialog appears.
 5. Edit the information.
 6. Click **OK**.
 7. Close the Stamps dialog.
-

Deleting a Stamp Library

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Stamp**. The Stamps dialog appears.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Stamp** .
 2. Click the **Libraries** tab.
 3. From the Stamp Libraries list, select the library that you want to delete.
 4. Click **Delete**.
STEP RESULT: The library disappears from the list and from the Stamp Library list under the **Stamps** tab.
 5. Close the Stamps dialog.
-







2D-Specific Markups













AutoVue provides a variety of user-friendly markup options that you can use when marking up 2D files. You can create entities such as arcs, boxes, circles, clouds, lines, and polygons. You can draw a leader with multi-line segments and add text to it.





*NOTE: When you are creating a markup entity, you can press the **Escape** key to cancel.*

2D Markup Entities

You can create many different types of markup entities. To access the markup entities, from the **Markup** menu, select **Add Entity**. The markup entities are:

Option	Description
	To complete a markup entity (for example, to finish creating a box), simply right-click in the AutoVue workspace.
Arc	Click and drag the mouse to draw an Arc. From the Markup Entity toolbar, you can also click Arc  .
Attachment	Add an attachment entity to the markup. From the Markup Entity toolbar, you can also click Attachment  . See “Adding an Attachment”
Box	Click and drag to draw a rectangle. From the Markup Entity toolbar, you can also click Box  . To draw a square instead of a rectangle, press and hold the Shift key while you click and drag.
Cloud	Click and drag to draw a Cloud. From the Markup Entity toolbar, you can also click Cloud  .
Circle	Click and drag to draw a circle. From the Markup Entity toolbar, you can also click Circle  . To draw a circle instead of an ellipse, press and hold the Shift key while you click and drag.
Freestyle	Click, release mouse button, then drag to draw an entity. From the Markup Entity toolbar, you can also click Freestyle  . You can create a contiguous or non-contiguous freestyle entity. See “Adding a Freestyle Entity”

Option	Description
Highlight	Click and drag to highlight a boxed area. From the Markup Entity toolbar, you can also click Highlight  . The highlighted box will be filled with a transparent color.
Hyperlink	Click the workspace to attach a hyperlink. From the Markup Entity toolbar, you can also click Hyperlink  . See “Adding a Hyperlink”
Intellistamp	Add a stamp on a document in a connected and disconnected environment. Includes specific document and user information (metadata) pulled directly from the DMS/ERP/PLM/UCM system. From the Markup Entity toolbar, you can also click Intellistamp  .
Leader	Click and drag to draw a leader. From the Markup Entity toolbar, you can also click Leader  . To force a line segment in a Leader entity to be aligned to the closer axis, horizontal or vertical axis, hold the Shift key while you click and drag for that line segment. See “Adding a Leader”
Line	Click and drag to draw a line. From the Markup Entity toolbar, you can also click Line  . To draw a line and force it to be aligned to the closer axis, horizontal or vertical, press and hold the Shift key while you click and drag. See “Aligning a Line Segment to the Vertical or Horizontal Axes”
Measurement	Create markup measure entities. From the Markup Entity toolbar, you can also click Distance  , Area  , Angle  , Arc  , or Minimum Distance  . See “Creating 2D non-Vector Markup Measure Entities” “Creating 2D Vector Markup Measure Entities” “Creating EDA Markup Measure Entities”
Note	Add a note to the markup. From the Markup Entity toolbar, you can also click Note  . See “Adding a Note”
Polygon	Click and drag to draw a polygon. From the Markup Entity toolbar, you can also click Polygon  .


Option	Description
Polyline	<p>Click and drag to draw a polyline.</p> <p>From the Markup Entity toolbar, you can also click Polyline .</p> <p>To force a line segment in a Polyline entity to be aligned to the closer axis, horizontal or vertical, press and hold the Shift key while you click and drag for that line segment.</p> <p>See “Aligning a Line Segment to the Vertical or Horizontal Axes”</p>
Signoff	<p>Create an approval stamp containing information about the markup author, date and time of creation.</p> <p>From the Markup Entity toolbar, you can also click Signoff .</p> <p>See “Adding Signoff Entities”</p>
Stamp	<p>Click and drag a stamp to the markup.</p> <p>From the Markup Entity toolbar, you can also click Stamp .</p> <p>See “Adding a Stamp”</p>
Text	<p>Add text to the markup.</p> <p>From the Markup Entity toolbar, you can also click Text .</p> <p>See “Adding Text”</p>

Adding a Freestyle Entity

You can create a freestyle markup entity. You have the option of making the entity non-contiguous or contiguous.

Creating a Non-Contiguous Freestyle Entity

TASK

- From the **Markup** menu, select **Add Entity**, and then select **Freestyle**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Freestyle** .
- Click a point on the drawing where you want to start the Freestyle entity.
- Move the cursor to create the Freestyle entity.
- Click a point on the drawing where you want to interrupt the Freestyle entity.
- Click another point on the drawing where you want to restart the Freestyle entity.
ADDITIONAL INFORMATION: You can interrupt the Freestyle entity as many times as you like by repeating steps 4 and 5.
- Right-click to end the Freestyle entity.

Creating a Contiguous Freestyle Entity

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Freestyle**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Freestyle** .

2. Click a point on the drawing where you want to start the Freestyle entity.
 3. Move the cursor to create the Freestyle entity.
 4. Right-click to end the Freestyle entity.
-

Adding a Leader

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Leader**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Leader** .

2. Click a point on the document where define the Leader anchor point.
3. Move the cursor to draw the Leader.

ADDITIONAL INFORMATION: To draw a Leader segment and force it to be aligned to the closest axis, vertical or horizontal, hold the **Shift** key while moving the cursor.

4. To draw a Leader with multiple line segments, repeat steps 2 and 3 as often as you like. You can click, then drag as often as you like.
5. Right-click to end the Leader. A text box appears at the end of the Leader.
6. In the text box, enter the text you want to attach to the Leader.

STEP RESULT: The height of the text box will resize to fit the entered text.

7. To change text box font properties, from the **Markup** menu, select **Format**, and then select **Font**.

ADDITIONAL INFORMATION: From the respective font property lists on the Markup Properties toolbar, you can also modify the font type, style, and size

STEP RESULT: The Font dialog appears allowing you to change the font type, style, and size.

8. Click **OK** to implement the font changes and to close the Font dialog.
9. To change the line properties or fill color of the Leader, select the Leader, then from the **Markup** menu, select **Format**, and then select the property or properties you want to change.

ADDITIONAL INFORMATION: From the respective line property lists on the Markup Properties toolbar, you can also modify the line style, line thickness, fill types, and fill colors.

10. Right-click outside the text area to complete the modification.

ADDITIONAL INFORMATION: To edit the Leader text, double-click the Leader in the tree or in the workspace.

STEP RESULT: The text appears on the drawing and in the Markup Navigation Tree.

Aligning a Line Segment to the Vertical or Horizontal Axes

A line segment drawn with the Free Snap snapping mod can be forced to align with the closer horizontal or vertical axis. You can also take an existing line segment and have it align with the closer axis. The types of line segments that you can align are lines, line segments of leaders and polylines, and measure entities.

See “2D Vector Snapping Modes” for more information Free Snap.

TASK

1. To draw and force a line segment, press and hold the **Shift** key while you click and drag for that line segment.
 2. To force an existing line segment, click and hold the left mouse button on the line segment, then press and hold the **Shift** key.
 3. When you see that the line segment is horizontal or vertical, release the left mouse button, then release the **Shift** key.
-

Creating 2D non-Vector Markup Measure Entities

When marking up 2D non-vector files, in addition to all the markup options available for 2D files, you can create markup measure entities. The measure options in Markup mode work a bit differently than in View mode.





See “2D-Specific Markups”

When measuring in Markup mode, the specified measurement lines and values are displayed on the current active markup layer as entities. These entities can be moved, resized, or deleted. You can also modify the font of a measure entity, align a “free snap” measure entity to the horizontal or vertical axis, as well as add units of measure and symbols to a measurement and have them appear on the drawing.

NOTE: When you are creating a markup entity, you can press the **Escape** key to cancel.

Measure options vary between vector and non-vector files. For vector files, AutoVue provides the option to “snap” to fixed points on the drawing. For non-vector files, you can only “free snap”.

In Markup mode, you can choose from several measure options to create markup measure entities. From the **Markup** menu, select **Add Entity**, and then select **Measurement** to access the following measure options:

Option	Description
Angle	Measure the angle between selected points. From the Markup Entity toolbar, you can also click Angle  .
Arc	Measure an arc entity. From the Markup Entity toolbar, you can also click Arc  .
Area	Measure a selected area. From the Markup Entity toolbar, you can also click Area  .
Distance	Measure the distance between two points. From the Markup Entity toolbar, you can also click Distance  .

Measuring Distance


Use the **Distance** option to measure the distance between two specific points.

See “Changing Measurement Units and Symbols”

“Changing Font”

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Distance**.

ADDITIONAL INFORMATION: The Measurement Entities dialog appears. From the Markup Entity toolbar, you can also click **Distance** .

2. From the Measured Distance Units list, select the unit in which you want to measure the distance.
3. Click a point on the drawing to define the starting point.
4. Click another point on the drawing to define the end point.

STEP RESULT: The measured line path appears as an entity on the current active markup.

5. Move the cursor and click a location on the drawing to display the measure distance.

STEP RESULT: The measurement and unit appear in a value box entity on the current active markup layer. The measured distance, Delta-X, and Delta-Y appear in the Measurement Entities dialog.


6. To resize the value box, select it and then click and drag the frame handles.

7. You can click and drag the text box anywhere on the drawing.
ADDITIONAL INFORMATION: To make another measurement, click **Reset**.
8. Click **Close** to close the Measurement Entities dialog.

Measuring Cumulative Distance

Use the **Cumulative Distance** option to measure the distance along a path of multi-faceted (adjoining) points.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Distance**.
ADDITIONAL INFORMATION: The Measurement Entities dialog appears. From the Markup Entity toolbar, you can also click **Distance** .
2. From the Measured Distance Units list, select the unit in which you want to measure the distance.
3. Select **Cumulative**.
4. Click a point on the drawing to define the starting point.
5. Continue clicking points along the path that you want to measure.
STEP RESULT: Each point is joined by a line.
6. Right-click to complete the measurement.
ADDITIONAL INFORMATION: To make another measurement, click **Reset**.
STEP RESULT: The measured line path, cumulated measurement and unit appear in a value box entity on the current active markup layer. The cumulated measured distance, Delta-X, and Delta-Y appear in the Measurement Entities dialog.
7. Click **Close** to close the Measurement Entities dialog.

Calibrating Distance

Calibrate the distance measurement.

TASK


1. Measure distance between two points or measure cumulative distance.
2. From the **Distance** tab, click **Calibrate**.
STEP RESULT: The Distance Calibration dialog appears displaying the measured distance
3. From the Units list, select the unit to which you want to calibrate the distance.

4. Select **Calibrate to** and enter a value if you want to calibrate the distance to a value.
 5. Select **Set Factor** and enter a value if you want to calibrate the distance by a factor.
 6. Click **OK**.
STEP RESULT: The calibration result appears in the Measurement dialog.
 7. Click **Close** to close the Distance tab.
-

Measuring Area

Use the **Area** option to measure the area and perimeter of a region.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Area**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Area** .
STEP RESULT: The Measurement Entities dialog appears.
 2. From the Measured Area Units list, select the unit in which you want to measure the area.
 3. From the Perimeter Units list, select the unit in which you want to measure the perimeter.
 4. To cumulate a Net Area Result of different areas, select **Add**.
 5. To subtract an area from the Net Area Result, select **Subtract**.
 6. Select **Clear** to clear the Net Area Result field.
 7. Click a point on the drawing to define the starting point.
 8. Continue clicking points on the drawing to define the area you want to measure.
STEP RESULT: Each point is joined by a line.
 9. Right click to complete the measurement.
ADDITIONAL INFORMATION: To make another measurement, click **Reset**.
STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The area and perimeter measurements appear in the Measurement Entities dialog.
 10. Click **Close** to close the Measurement Entities dialog.
-

Measuring an Angle

Use the **Angle** option to measure the angle between points on a drawing.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Angle**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Angle** .

STEP RESULT: The Measurement Entities dialog appears.

2. From the Measured Angle Units list, select the unit in which you want to measure the angle.
3. Click points on the drawing to define the angle you want to measure:
 - The first click defines the starting point of the angle measurement.
 - The second click defines the vertex of the angle measurement.
 - The third click defines the endpoint of the angle measurement.
 - The points are joined by angle arms with an arc connecting them.
4. Move the cursor to increase or decrease the radial and arc length.
5. Click again to complete the measurement.

STEP RESULT: The measured line path, angle measurement and unit appear in a value box entity on the current active markup layer. The measured angle also appears in the Measurement Entities dialog.

6. To resize the value box, select it and then click and drag the frame handles.
7. You can click and drag the value box anywhere on the drawing.

ADDITIONAL INFORMATION: To make another measurement, click **Reset**.

8. Click **Close** to close the Measurement Entities dialog.

Measuring an Arc

Use the **Arc** option to define an arc in the drawing and measure its radius, center and diameter.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Arc**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Arc** .

STEP RESULT: The Measurement Entities dialog appears.

2. From the Arc Info Length list, select the unit in which you want to measure the arc.

3. From the Measured Angle Units list, select the unit in which you want to measure the angle. The options are degrees or radians.
 4. Select **Add Radius** if you want to measure the radius of the arc.
 5. Select **Add Diameter** if you want to measure the diameter of the arc.
 6. Click three points on the drawing to define the arc you want to measure.
STEP RESULT: The points are joined by an arc.
 7. Click again to complete the measurement.
STEP RESULT: The measured line path, arc measurement and unit appear in a value box entity on the current active markup layer and in the Measurement Entities dialog.
 8. To resize the value box, select it and then click and drag the frame handles.
 9. You can click and drag the text box anywhere on the drawing.
ADDITIONAL INFORMATION: To make another measurement, click **Reset**.
 10. Click **Close** to close the Measurement Entities dialog.
-

Calibrating an Arc

TASK

1. Measure an arc in the drawing.
 2. From the **Arc** tab, click **Calibrate**.
STEP RESULT: The Radius Calibration dialog appears displaying the measured distance.
 3. From the Units list, select a unit of measurement to which you want to calibrate the distance.
 4. Click **Calibrate to** and enter a value if you want to calibrate to a value.
 5. Click **Set Factor** and enter a value if you want to calibrate by a factor.
 6. Click **OK**.
STEP RESULT: The calibration results appear in the Arc tab.
 7. Click **Close** to close the Measurement dialog.
-

Creating 2D Vector Markup Measure Entities





When marking up 2D vector files, in addition to all the markup options available for 2D files, you can create markup measure entities. The measure options in Markup mode work slightly different than in View mode.

Measure options vary between vector and non-vector files. For vector files, AutoVue provides the option to “snap” to fixed points on the drawing. For non-vector files, you can only “free snap”.

When measuring in Markup mode, the specified measurement lines and values are displayed on the current active markup layer as entities. These entities can be moved, resized, hidden or deleted. You can also modify the font of a measure entity, align a “free snap” measure entity to the horizontal or vertical axis, as well as add units of measure and symbols to a measurement and have them appear on the drawing.

NOTE: When you are creating a markup entity, you can press the **Escape** key to cancel.



In Markup mode, you can choose from several measure options to create markup measure entities. From the **Markup** menu, select **Add Entity**, and then select **Measurement** to access the following measure options:



Option	Description
Angle	Measure the angle between selected points. From the Markup Entity toolbar, you can also click Angle  .
Arc	Measure an arc entity. From the Markup Entity toolbar, you can also click Arc  .
Area	Measure a selected area. From the Markup Entity toolbar, you can also click Area  .
Distance	Measure the distance between two points. From the Markup Entity toolbar, you can also click Distance  .

2D Vector Snapping Modes

The available **snapping modes** allow you to click to precise geometrical points on a drawing. For example, if you select **Snap to End-Point** and you move the cursor over an end-point of a line, the end-point will be highlighted by a snap box.

The snapping modes allow you to snap to the mid, center, and end-points of an entity:

Button	Snap to	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near a linear component's end point.
	Mid-point	Geometric snap mode where a snap box appears when moving the cursor near the halfway point of a linear component.

Button	Snap to	Description
	Center-point	Geometric snap mode where a snap box appears when moving the cursor near the center of an elliptical component.
	Free snap	Allow snapping at any point on the drawing.

NOTE: When selecting a snapping mode you also have the option to select all snapping modes (**All On**) or to turn off snapping modes (**All Off**)


The below table lists snapping locations for certain measurements:

Measurement	Snapping Location
Area	Snaps to a shape on the drawing.
Arc	Snaps to an arc on the drawing.
Angle	Snap to two non-parallel lines.

Measuring Distance

Use the **Distance** option to measure the distance between two specific points.

TASK

- From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Distance**.
ADDITIONAL INFORMATION: The Measurement Entities dialog appears. From the Markup Entity toolbar, you can also click **Distance** .
- Select the snapping modes that you want to use for measuring.
- To select all snapping modes click **All On**. To clear all snapping modes click **All Off**.
- From the Measured Distance Units list, select the unit in which you want to measure the distance.
- Click a point on the drawing to define the starting point.
- Click another point on the drawing to define the end point.

ADDITIONAL INFORMATION: If you selected **Cumulative**, continue clicking points along the path that you want to measure.

STEP RESULT: The measured line path appears as an entity on the current active markup.

7. Move the cursor and click a location on the drawing to display the measure distance.

STEP RESULT: The measurement and unit appear in a value box entity on the current active markup layer. The measured distance, Delta-X, and Delta-Y appear in the Measurement Entities dialog.

8. To resize the value box, select it and then click and drag the frame handles.
9. You can click and drag the text box anywhere on the drawing.

ADDITIONAL INFORMATION: To make another measurement, click **Reset**.

10. Click **Close** to close the Measurement Entities dialog.

Measuring Cumulative Distance

Use the **Cumulative Distance** option to measure the distance along a path of multi-faceted (adjoining) points.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Distance**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Distance** .

STEP RESULT: The Measurement Entities dialog appears.

2. Select the snapping modes that you want to use for measuring.
3. To select all snapping modes, click **All On**. To deselect all snapping modes, click **All Off**.
4. From the Measured Distance Units list, select the unit in which you want to measure the distance.
5. Select **Cumulative**.
6. Click the first entity to define the starting point.
7. Continue clicking points along the path that you want to measure.

STEP RESULT: Each point is joined by a line.

8. Right-click to complete the measurement.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measured distance, Delta-X, Delta-Y and the "Manhattan Distance" appear in the Measurement Entities dialog.

9. Click **Close** to close the Measurement Entities dialog.

Calibrating Distance

Calibrate the distance measurement.


TASK

1. Measure distance between two points or measure cumulative distance.
 2. From the **Distance** tab, click **Calibrate**.
STEP RESULT: The Distance Calibration dialog appears displaying the measured distance
 3. From the Units list, select the unit to which you want to calibrate the distance.
 4. Select **Calibrate to** and enter a value if you want to calibrate the distance to a value.
 5. Select **Set Factor** and enter a value if you want to calibrate the distance by a factor.
 6. Click **OK**.
STEP RESULT: The calibration result appears in the Measurement dialog.
 7. Click **Close** to close the Distance tab.
-

Measuring Area

Use the Area option to measure the area and perimeter of a region.

TASK


1. From the **Analysis** menu, select **Measure**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Measure** .
STEP RESULT: The Measurement dialog appears.
2. Click the **Area** tab.
3. Select **Between Points** if you want to measure the area between points on a drawing. The snapping modes are displayed.
4. Select the snapping modes that you want to use for measuring.
5. Select **Shape** if you want to measure the area of a predefined shape on the drawing; the snapping modes are disabled.
6. From the Measured Area Units list, select the unit in which you want to measure the area.
7. From the Perimeter Units list, select the unit in which you want to measure the perimeter.
8. In the Net Area Result section of the dialog, select **Add** to cumulate a net area result of different areas.
9. To subtract an area from the Net Area Result, select **Subtract**.

10. Select **Clear** to clear the Net Area Result.
11. If you selected **Between Points**, click points on the drawing to define the area.
STEP RESULT: Each point is joined by a line. The area and perimeter measurements appear in the Measurement dialog.
12. If you selected **Shape**, click the edge of the predefined shape that you want to measure.
ADDITIONAL INFORMATION: Click Reset to take another measurement.
STEP RESULT: The shape is highlighted. The area and perimeter measurements appear in the Measurement dialog.
13. Click **Close** to close the Measurement dialog.

Measuring an Angle

Use the **Angle** option to measure the precise angle between points on a drawing.

TASK


1. From the **Markup** menu, select **Add Entity**, and then select **Angle**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Angle** .
STEP RESULT: The Measurement Entities dialog appears.
2. Select **From 3 Points** if you want to measure the angle between three points. The snapping modes are displayed.
STEP RESULT: The snapping modes are displayed.
3. Select the snapping modes that you want to use for measuring.
4. Select **Between 2 lines** if you want to measure the angle between two lines.
5. From the Measured Angle Units list, select the unit in which you want to measure the angle.
6. If you selected **From 3 Points**, click three points on the drawing to define the angle. If you selected **Between 2 lines** click two lines on the drawing to define the angle.
STEP RESULT: Angle arms appear with an arc connecting them.
7. Click again to complete the measurement.
STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measurement appears in the Measurement Entities dialog.
8. To change the size of the arc, click and drag it to the desired size.
9. To move the value box, click and drag it to anywhere on the markup.

10. To resize the value box, select it and then click and drag the frame handles.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
 11. Click **Close** to close the Measurement Entities dialog.
-

Measuring an Arc

Use the **Arc** option to measure the precise radius, length and angles of any arc on the model. It also calculates the center point location.

TASK

1. From the **Markup** menu, select **Add Entity, Measurement**, and then **Arc**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Arc** .
STEP RESULT: The Measurement Entities dialog appears.
 2. Select **Arc Entity** if you want to measure a predefined arc.
STEP RESULT: All arc and circles are highlighted on the model. Snapping modes are disabled.
 3. Select **From 3 Points** if you want to measure the arc between three points. Snapping modes are displayed.
 4. Select the snapping modes that you want to use for measuring.
STEP RESULT: All entities of the selected entity types are highlighted on the model.
 5. From the Dist. Units list, select the unit in which you want to measure the arc distance.
 6. From the Angles Units list, select the unit in which you want to measure the angle.
 7. If you selected **From 3 Points**, click three points to define the arc.
STEP RESULT: The points are joined by an arc.
 8. If you selected **Arc Entity**, click the edge of an arc.
STEP RESULT: The arc is highlighted.
 9. Click to complete the measurement.
STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measurements for arc center coordinates, radius, diameter, ratio, arc length, start and end of angle and sweep appear in the Measurement Entities dialog.
 10. To move the value box, click and drag it to anywhere on the markup.
 11. To resize the value box, select it and then click and drag the frame handles.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
 12. Click **Close** to close the Measurement Entities dialog.
-

Calibrating an Arc

TASK

1. Measure an arc in the drawing.
 2. From the **Arc** tab, click **Calibrate**.
STEP RESULT: The Radius Calibration dialog appears displaying the measured distance.
 3. From the Units list, select a unit of measurement to which you want to calibrate the distance.
 4. Click **Calibrate to** and enter a value if you want to calibrate to a value.
 5. Click **Set Factor** and enter a value if you want to calibrate by a factor.
 6. Click **OK**.
STEP RESULT: The calibration results appear in the Arc tab.
 7. Click **Close** to close the Measurement dialog.
-

Creating EDA Markup Measure Entities

When marking up EDA files, in addition to all the markup options available for 2D files, you can create markup measure entities. The measure options in Markup mode work slightly different than in View mode.






See “2D-Specific Markups”

NOTE: When you are creating a markup entity, you can press the **Escape** key to cancel.

When measuring in Markup mode, the specified measurement lines and values are displayed on the current active markup layer as entities. These entities can be moved, resized, or deleted. You can also modify the font of a measure entity, align a “free snap” measure entity to the horizontal or vertical axis, as well as add units of measure and symbols to a measurement and have them appear on the drawing.

AutoVue provides the option to “snap” to geometrical or electrical points on the drawing.




In Markup mode, you can choose from several measure options to create markup measure entities. From the **Markup** menu, select **Add Entity**, and then select **Measurement** to access the following measure options:







Option	Description
Angle	Measure the angle between selected points. From the Markup Entity toolbar, you can also click Angle  .
Arc	Measure an arc entity. From the Markup Entity toolbar, you can also click Arc  .
Area	Measure a selected area. From the Markup Entity toolbar, you can also click Area  .
Distance	Measure the distance between two points. From the Markup Entity toolbar, you can also click Distance  .
Minimum Distance	Measure the minimum distance between entities. From the Markup Entity toolbar, click Minimum Distance  .

EDA Snapping Modes

The *snapping modes* allow you to click to precise geometrical or electrical points. For example, when you select **Snap to Pin**, move the cursor over the pin you want to select until the pin is highlighted, then click. Highlight and click a second pin to measure the distance between them.

The snapping modes allow you to snap to the middle, center, and end-points of an entity, as well as a pin, via, and symbol. The following table outlines the available snapping modes:

Button	Snap To	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near the entity's end point.
	Mid-point	Geometric snap mode where a snap box appears when moving the cursor near the halfway point of a linear entity.
	Center-point	Geometric snap mode where a snap box appears when moving the cursor near the center of an entity.

Button	Snap To	Description
	Pin	Electrical snap mode where a snap box appears when the cursor touches a pin.
	Via	Electrical snap mode where a snap box appears when the cursor touches a via.
	Symbol origin	Electrical snap mode where a snap box appears when the cursor touches a component.
	Free snap	Allows snapping at any point on the drawing.
	Net	Allows snapping to a net.
	Trace	Allows snapping to a trace.

Measuring Distance

Use the **Distance** option to measure the distance between two specific points.

TASK

- From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Distance**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Distance** .

STEP RESULT: The Measurement Entities dialog appears.

- Select the snapping modes that you want to use for measuring.
- To select all snapping modes, click **All On**. To deselect all snapping modes, click **All Off**.
- From the Measured Distance Units list, select the unit in which you want to measure the distance.
- Click a point on the drawing to define the starting point.
- Click another point on the drawing to define the end point.

STEP RESULT: The points are joined by a line. The measured line path appears as an entity on the current active markup.

- Drag to move the measured line path.

8. Click on the measured line path.

STEP RESULT: The measurement and unit appear in a value box entity on the current active markup layer. The measured distance, Delta-X, Delta-Y and the “Manhattan Distance” appear in the Measurement Entities dialog.

9. To resize the value box, select it and then click and drag the frame handles.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

10. Click **Close** to close the Measurement Entities dialog.
-

Measuring Cumulative Distance

Use the **Cumulative Distance** option to measure the distance along a path of multi-faceted (adjoining) points.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Distance**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Distance** .

STEP RESULT: The Measurement Entities dialog appears.

2. Select the snapping modes that you want to use for measuring.
3. To select all snapping modes, click **All On**. To deselect all snapping modes, click **All Off**.

4. From the Measured Distance Units list, select the unit in which you want to measure the distance.

5. Select **Cumulative**.

6. Click the first entity to define the starting point.

7. Continue clicking points along the path that you want to measure.

STEP RESULT: Each point is joined by a line.

8. Right-click to complete the measurement.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measured distance, Delta-X, Delta-Y and the “Manhattan Distance” appear in the Measurement Entities dialog.

9. Click **Close** to close the Measurement Entities dialog.
-

Calibrating Distance

Calibrate the distance measurement.


TASK

1. Measure distance between two points or measure cumulative distance.
 2. From the **Distance** tab, click **Calibrate**.
STEP RESULT: The Distance Calibration dialog appears displaying the measured distance
 3. From the Units list, select the unit to which you want to calibrate the distance.
 4. Select **Calibrate to** and enter a value if you want to calibrate the distance to a value.
 5. Select **Set Factor** and enter a value if you want to calibrate the distance by a factor.
 6. Click **OK**.
STEP RESULT: The calibration result appears in the Measurement dialog.
 7. Click **Close** to close the Distance tab.
-

Measuring Area

Use the **Area** option to measure the area and perimeter of a region.

TASK


1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Area**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Area** .
STEP RESULT: The Measurement Entities dialog appears.
2. Select **Between Points** if you want to measure the area between points on a drawing. Snapping modes are displayed.
3. Select the snapping modes that you want to use for measuring.
ADDITIONAL INFORMATION: To select all snapping modes, click **All On**. To deselect all snapping modes, click **All Off**.
4. Select **Shape** if you want to measure the area of a predefined shape on the drawing.
5. From the Measured Area Units list, select the unit in which you want to measure the area.
6. From the Perimeter Units list, select the unit in which you want to measure the perimeter.

7. To cumulate a Net Area Result of different areas, select **Add** in the Measurement Entities dialog.
 8. To subtract an area from the Net Area Result, select **Subtract**.
 9. Select **Clear** to clear the Net Area Result field.
 10. If you selected **Between Points**, click points on the drawing to define the area.
STEP RESULT: Each point is joined by a line and the measurement appears in the Measurement Entities dialog.
 11. If you selected **Shape**, click the edge of a predefined shape on the drawing.
STEP RESULT: The shape is highlighted.
 12. Right-click to complete the measurement.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The area and perimeter measurements appear in the Measurement Entities dialog.
 13. Click **Close** to close the Measurement Entities dialog.
-

Measuring an Angle

Use the **Angle** option to measure the angle between points on a drawing.

TASK


1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Angle**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Angle** .
STEP RESULT: The Measurement Entities dialog appears.
2. Select **From 3 Points** if you want to measure the angle between three points. Snapping modes are displayed.
3. Select the snapping modes that you want to use for measuring.
ADDITIONAL INFORMATION: To select all snapping modes, click **All On**. To deselect all snapping modes, click **All Off**.
4. Select **Between 2 lines** if you want to measure the angle between two lines.
5. From the Measured Angle Units list, select the unit in which you want to measure the angle.
6. If you selected **From 3 Points**, click three points to define the angle.
7. If you selected **Between 2 Lines**, click two lines to define the angle.
STEP RESULT: Angle arms appear with an arc connecting them.

8. Right-click to complete the measurement.
STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measured angle appears in the Measurement Entities dialog.
9. To change the size of the arc, click and drag the it to the desired size.
10. To move the value box, click and drag it to anywhere on the drawing.
11. To resize the value box, select it and then click and drag the frame handles.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
12. Click **Close** to close the Measurement Entities dialog.

Measuring an Arc

Use the **Arc** option to define an arc in the drawing and measure its radius, center and diameter.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Arc**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Arc** .
STEP RESULT: The Measurement Entities dialog appears.
2. Select **From 3 Points** if you want to measure the arc between three points. Snapping modes are displayed.
3. Select the snapping modes that you want to use for measuring.
ADDITIONAL INFORMATION: To select all snapping modes, click **All On**. To deselect all snapping modes, click **All Off**.
4. Select **Arc Entity** if you want to measure a predefined arc.
5. From the Arc Info list, select the unit in which you want to measure the distance of the arc.
6. From the Measured Angle Units list, select the unit in which you want to measure the angle.
7. Select **Add Radius** if you want measure the radius.
8. Select **Add Diameter** if you want to measure the diameter.
9. If you selected **From 3 Points**, click three points to define the arc.
STEP RESULT: The points are joined by an arc.
10. If you selected an **Arc Entity**, click an edge of the arc.
STEP RESULT: The arc is highlighted.

11. Click to complete the measurement.

STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measurements for center point coordinates, radius, diameter, arc length, start and end of angle and sweep appear in the Measurement Entities dialog.

12. To move the value box, click and drag it to anywhere on the drawing.
13. To resize the value box, select it and then click and drag the frame handles.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

14. Click **Close** to close the Measurement Entities dialog.
-

Calibrating an Arc

TASK

1. Measure an arc in the drawing.
2. From the **Arc** tab, click **Calibrate**.

STEP RESULT: The Radius Calibration dialog appears displaying the measured distance.

3. From the Units list, select a unit of measurement to which you want to calibrate the distance.
4. Click **Calibrate to** and enter a value if you want to calibrate to a value.
5. Click **Set Factor** and enter a value if you want to calibrate by a factor.
6. Click **OK**.

STEP RESULT: The calibration results appear in the Arc tab.

7. Click **Close** to close the Measurement dialog.
-


Measuring Minimum Distance

Use the **Minimum Distance** option to measure the minimum distance between entities. The available entities for snapping are nets, pins, vias and traces.


*NOTE: To measure the minimum distance between entities on separate layers, select **Across Layers**. If a physical layer does not exist, this option determines whether or not the minimum distance should be calculated across logical layers.*

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Minimum Distance**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Minimum Distance** .

STEP RESULT: The Measurement Entities dialog appears.


2. Click  **First Set** to select the entities that you want to measure from.
3. Select the snapping modes that you want to use for measuring.

ADDITIONAL INFORMATION: If you select **Net**, you cannot select any other type of entity.

4. Click the first set of entities on the drawing.

ADDITIONAL INFORMATION: To clear the last set of entities you selected, click **Clear Set**.

STEP RESULT: The entities are highlighted.

5. Click  **Second Set** to select the entities that you want to measure to.
6. Click the second set of entities on the drawing.

STEP RESULT: The entities are highlighted in a different color.

7. From the Measured Min. Distance list, select the unit in which you want to measure the distance.
8. Select **Zoom to Result**, if you want to zoom in on the measured value on the drawing.
9. Click **Compute**.

STEP RESULT: The minimum distance from the first set of entities to the second set is highlighted by a line. The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measurement, Delta-X, Delta-Y, and the Manhattan Distance appear in the Measurement Entities dialog.

10. To move the value box, click and drag it to anywhere on the drawing.
11. To resize the value box, select it and then click and drag the frame handles.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

12. Click **Close** to close the Measurement Entities dialog.
-

Adding Text

With AutoVue, you can add a text box entity to a markup.

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Text**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Text** .

2. Click and drag on the drawing to define the dimensions of the text box.

3. Click in the text box and enter the text you want to add.

ADDITIONAL INFORMATION: The height of the text box will adjust to accommodate the text.

4. To change text box font properties, from the **Markup** menu, select **Format**, and then select **Font**.

ADDITIONAL INFORMATION: From the respective font property lists on the Markup Properties toolbar, you can also modify the font type, style, and size.

STEP RESULT: The Font dialog appears allowing you to change the font type, style, and size.

5. Click **OK** to implement the font changes and to close the Font dialog.

6. To change the line properties or fill color of the text box, select the text box, then from the **Markup** menu, select **Format**, and then select the property or properties you want to change.

ADDITIONAL INFORMATION: From the respective line property lists on the Markup Properties toolbar, you can also modify the line style, line thickness, fill types, and fill colors.

7. Right-click outside the text area to complete the modification.

STEP RESULT: The text appears on the drawing and in the Markup Navigation Tree.

8. To move the text box click and drag it.

9. Click and drag the frame handles of the text box to enlarge it.

ADDITIONAL INFORMATION: To edit the text, double-click the text box in the tree or in the workspace.

Hiding the box surrounding the text

TASK

1. Select the text box entity.

2. From the **Markup** menu, select **Format**, and then select **Markup Entity Attributes**.


STEP RESULT: The Markup Entity Properties dialog appears.

3. From the Text Box Visibility list, select **OFF**.
4. Click **OK**.

ADDITIONAL INFORMATION: To undo, repeat steps 1 through 4, except select **ON**.

STEP RESULT: The dialog closes and the text box is hidden.

Adding a Note

You can attach longer comments as a markup note with AutoVue's Markup Control Note Editor. A note displays in the workspace as a standard size graphical symbol . Each note is labelled as *Note<n>*—where *n* represents the numerical order of occurrence of the note (for example, the first note is labelled as *Note1*). To read the note, double-click the entity to open it or move the mouse over the entity to display the tooltip.

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Note**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Note** .

2. Click a point on the document where you want to insert the note.

STEP RESULT: The Markup Control Note Editor dialog box appears.

3. Enter the text that you want in the note.
4. To modify the text, select **Formatting** and the format you want to apply.
5. From the **File** menu, select **Information**.


STEP RESULT: The Note Information dialog box appears.

6. In **Name**, **Author** and **Key Words** fields, enter the required text for the note.
7. Click **OK**.

8. Close the Markup Control Note Editor dialog box.

STEP RESULT: The note appears on the drawing and in the Markup Navigation Tree.

9. Right-click outside the note area to complete the modification.
10. To move the note, click and drag it.

ADDITIONAL INFORMATION: To edit a note, double-click the note symbol  in the workspace. You can also select the note in the tree, right-click, and select **Edit**, or double-click the note.

Viewing and Printing the List of Notes

You can view a list of the notes you added to a file. You can also print the notes or find a note in the workspace.

TASK

1. From the **Markup** menu, select **Print Notes**.
STEP RESULT: The List Notes dialog box appears.
 2. To find a note, select the note from the list and click **Go To**.
ADDITIONAL INFORMATION: You can also select the note from the Markup Navigation Tree, right-click, and select **Go To**.
STEP RESULT: AutoVue zooms into the selected note in the workspace.
 3. To print a note, select the note from the list.
ADDITIONAL INFORMATION: To print more than one note, press the **Shift** or **Control** key while selecting. To print all the notes in the list, click **Select All**.
 4. Click **Print**.
ADDITIONAL INFORMATION: To print each selected note on a separate page, select **One Note Per Page**.
STEP RESULT: The Note Print Options dialog box appears.
 5. Configure the print options, then click **OK**.
STEP RESULT: The selected notes are printed with their unique headers (for example, Note1, Note2, and so on.)
 6. Click **OK** to close the List Notes dialog box.
-

Nesting Markup Entities

You can add the Attachment or Hyperlink markup entities as a nested child to any markup entity.

TASK

1. Add a markup entity to a 3D file (for example, a Text entity).
2. From the workspace or Markup Tree, select the Text entity.

- From the **Markup** menu, select **Attachment** or **Hyperlink**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Attachment** or **Hyperlink**.

RESULT:

In the Markup Tree, the selected markup entity appears as a child of the Text (parent) entity.

3D-Specific Markups










When marking up 3D files, you can attach text or a note, as well as create markup measure entities. The measure options in Markup mode work slightly different than in View mode.

NOTE: When you are creating a markup entity, you can press the **Escape** key to cancel.

3D Markup Entities

You can create many different types of markup entities. To access the markup entities, from the **Markup** menu, select **Add Entity**. The markup entities are:

Option	Description
	To complete a markup entity (for example, to finish creating a box), simply right-click in the AutoVue workspace.
Text	<p>Add text to the markup.</p> <p>From the Markup Entity toolbar, you can also click 3D Text.</p> <p>See “Adding Text”</p>
Attachment	<p>Add an attachment entity to the markup.</p> <p>From the Markup Entity toolbar, you can also click Attachment.</p> <p>See “Adding an Attachment”</p>
Hyperlink	<p>Click the workspace to attach a hyperlink.</p> <p>From the Markup Entity toolbar, you can also click Hyperlink.</p> <p>See “Adding a Hyperlink”</p>

Option	Description
Intellistamp	Add a stamp on a document in a connected and disconnected environment. Includes specific document and user information (metadata) pulled directly from the DMS/ERP/PLM/UCM system. From the Markup Entity toolbar, you can also click Intellistamp  .
Measurement	Create markup measure entities. From the Markup Entity toolbar, you can also click Distance  , Area  , Angle  , Arc  , or Minimum Distance  . See “Creating 3D Markup Measure Entities”
Note	Add a note to the markup. From the Markup Entity toolbar, you can also click Note  . See “Adding a Note”
Signoff	Create an approval stamp containing information about the markup author, date and time of creation. From the Markup Entity toolbar, you can also click Signoff  . See “Adding Signoff Entities”
Stamp	Click and drag a stamp to the markup. From the Markup Entity toolbar, you can also click Stamp  . See “Adding a Stamp”

Creating 3D Markup Measure Entities






When marking up 3D files, you can create markup measure entities. The measure options in Markup mode work slightly different than in View mode.

When measuring in Markup mode, the specified measurement lines and values are displayed on the current active markup layer as entities. These entities can be moved, resized, or deleted.

NOTE: *If you manipulate a part of a model, the measure entity values do not update accordingly.*

AutoVue provides the option to “snap” to different entity types on the model.




In Markup mode, you can choose from several measurement options to create markup measure entities. From the **Markup** menu, select **Add Entity**, and then select **Measure-**
ment to access the following measure options:

Name	Description
Angle	Measure the precise angle between three vertices or any two edges, planes or faces. From the Markup Entity toolbar, you can also click Angle  .
Arc	Measure the precise radius, length and angle of any arc and calculate the center point location. From the Markup Entity toolbar, you can also click Arc  .
Distance	Measure the precise distance between any two Vertex, Edge, Midedge, Arc Center or Face. From the Markup Entity toolbar, you can also click Distance  .
Minimum Distance	Measure minimum distance between any two Vertex, Edge, Midedge, Arc Center or Face. From the Markup Entity toolbar, you can also click Minimum Distance  . You cannot create a markup measure entity when measuring minimum distance.
Vertex Coordinates	Provide the coordinates of each vertex. From the Markup Entity toolbar, you can also click Vertex Coordinates  .

3D Snapping Modes

The *snapping modes* available allow you to select or snap to different entity types on a model. For example, if you select **Vertex**, all vertices are highlighted and when you move the cursor over a vertex, a snap box appears.

The snapping modes allow you to snap to vertices, edges, faces, planes and arcs.





Button	Snap to	Description
	Edge	Edges are highlighted on the model. When moving the cursor over an edge, a snap circle appears.
	Face	Faces are highlighted when you move the cursor over a face and a snap triangle appears.
	Vertex	Vertices are highlighted on the model. When moving the cursor over a vertex, a snap box appears.

Measuring Distance

Use the **Distance** option to measure the precise distance between two vertices, edges, mid-edges, arc centers, faces, or any combination of entity types.

NOTE: When measuring the distance between faces, if the faces are parallel, AutoVue compares the shortest distance between the parallel faces.

TASK

1. From the **Markup** menu, select **Add Entity**, and then select **Distance**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Distance** .
STEP RESULT: The Measurement Entities dialog appears.
 2. From the Measured Distance Units list, select the unit in which you want to measure the distance.
 3. Select the snapping modes that you want to use for measuring.
STEP RESULT: All entities of the selected entity types are highlighted on the model.
 4. On the model, select the entity you want to measure from.
ADDITIONAL INFORMATION: If you want to take more than one measurement from the same starting point, select the **Fix Position** check box.
STEP RESULT: The location of the entity appears in the **From**  field.
 5. Click inside the **To**  field to select the entity type that you want to measure to.
 6. Select the snapping mode that you want to select as the end point for the measurement.
STEP RESULT: All entities of the selected entity types are highlighted on the model.
 7. On the model, select the entity you want to measure to.
STEP RESULT: The location of the entity appears in the **To**  field.
 8. Click again to complete the measurement.
STEP RESULT: The measured line path appears in a value box entity on the current active markup. The measured distance, Delta-X, Delta-Y and Delta-Z appear in the Measurement Entities dialog.
 9. Drag to move the measured line path.
 10. Click on the measured line path.
 11. To move the value box, click and drag it to anywhere on the drawing.
 12. To resize the value box, select it and then click and drag the frame handles.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
 13. Click **Close** to close the Measurement Entities dialog.
-

Calibrating Distance

Calibrate the distance measurement.


TASK

1. Measure distance between two points or measure cumulative distance.
 2. From the **Distance** tab, click **Calibrate**.
STEP RESULT: The Distance Calibration dialog appears displaying the measured distance
 3. From the Units list, select the unit to which you want to calibrate the distance.
 4. Select **Calibrate to** and enter a value if you want to calibrate the distance to a value.
 5. Select **Set Factor** and enter a value if you want to calibrate the distance by a factor.
 6. Click **OK**.
STEP RESULT: The calibration result appears in the Measurement dialog.
 7. Click **Close** to close the Distance tab.
-

Measuring an Angle

Use the **Angle** option to measure the precise angle between points on a drawing.

TASK

1. From the **Markup** menu, select **Add entity**, select **Measurement**, and then select **Angle**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Angle** .
STEP RESULT: The Measurement Entities dialog appears.
2. Click the **Angle** tab.
3. Select the snapping modes that you want to use for measuring.
STEP RESULT: Entities of the selected types are highlighted on the model.
4. To measure the angle between an entity type and a plane, select the **With Plane** check box, and then select the plane from the list.
5. From the Measured Angle Units list, select the unit in which you want to measure the angle.
6. Click two points on the model to define the angle.

7. If you selected a Plane, click the vertex, edge or face whose angle you want to measure between the plane.

ADDITIONAL INFORMATION: To take another measurement click **Reset**.

STEP RESULT: Angle arms appears indicating the angle. The measurement appears in the Measurement dialog.


8. Click **Close** to close the Measurement dialog.
-

Measuring an Arc

Use the **Arc** option to measure the precise radius, length and angles of any arc on the model. It also calculates the center point location.

TASK

1. From the **Markup** menu, select **Add Entity, Measurement**, and then **Arc**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Arc** .

STEP RESULT: The Measurement Entities dialog appears.

2. Select **Arc Entity** if you want to measure a predefined arc.

STEP RESULT: All arc and circles are highlighted on the model. Snapping modes are disabled.

3. Select **From 3 Points** if you want to measure the arc between three points. Snapping modes are displayed.

4. Select the snapping modes that you want to use for measuring.

STEP RESULT: All entities of the selected entity types are highlighted on the model.

5. From the Dist. Units list, select the unit in which you want to measure the arc distance.

6. From the Angles Units list, select the unit in which you want to measure the angle.

7. If you selected **From 3 Points**, click three points to define the arc.

STEP RESULT: The points are joined by an arc.

8. If you selected **Arc Entity**, click the edge of an arc.

STEP RESULT: The arc is highlighted.

9. Click to complete the measurement.

STEP RESULT: The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The measurements for arc center coordinates, radius, diameter, ratio, arc length, start and end of angle and sweep appear in the Measurement Entities dialog.

10. To move the value box, click and drag it to anywhere on the markup.

11. To resize the value box, select it and then click and drag the frame handles.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

12. Click **Close** to close the Measurement Entities dialog.

Calibrating an Arc



TASK


1. Measure an arc in the drawing.
2. From the **Arc** tab, click **Calibrate**.
STEP RESULT: The Radius Calibration dialog appears displaying the measured distance.
3. From the Units list, select a unit of measurement to which you want to calibrate the distance.
4. Click **Calibrate to** and enter a value if you want to calibrate to a value.
5. Click **Set Factor** and enter a value if you want to calibrate by a factor.
6. Click **OK**.
STEP RESULT: The calibration results appear in the Arc tab.
7. Click **Close** to close the Measurement dialog.

Measuring Minimum Distance

With the **Minimum Distance** option you can measure the minimum distance between model parts, as well as any two points from the selection sets: vertices, edges, mid-edges, arc centers, faces or any combination of entity types.

TASK


1. From the **Markup** menu, select **Add Entity**, and then select **Minimum Distance**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Minimum Distance** .
STEP RESULT: The Measurement Entities dialog box appears.
2. Click  **Set 1**.
3. Select **Entity** if you want to measure the distance between model parts. Snapping modes are displayed.
4. Select **Geometry** if you want to measure the distance between entity types.
5. If you selected **Entity**, select a part or parts on the model.
STEP RESULT: The model part or parts appear in the list under **Set 1** and are highlighted on the model and in the Model Tree.

6. If you selected **Geometry**, select the snapping modes that you want to use for measuring.
STEP RESULT: All entities of the selected entity type are highlighted on the model.
 7. Select the entity or entities on the model.
ADDITIONAL INFORMATION: To reset a **Set**, click **Clear**. To clear items from a **Set**, select the items and press the **Delete** key. To deselect a part or entity type on the model, press the **Control** key and left-click the part or entity type.
STEP RESULT: The selected entities appears in the list under **Set 1** and are highlighted on the model.
 8. Click  **Set 2**.
 9. Repeat steps 4 through 6.
STEP RESULT: The model part appears in the list under **Set 2**.
 10. From the Measured Min. Distance Units list, select the unit in which you want to measure the distance.
 11. Click **Compute**.
STEP RESULT: The minimum distance from the first set of entities to the second set is highlighted by a line. The measured line path, measurement and unit appear in a value box entity on the current active markup layer. The X, Y and Z coordinates for Position 1 and X, Y and Z coordinates for Position 2 appear in the Measurement Entities dialog box.
 12. Click and drag the value box anywhere on the markup.
 13. To resize the value box, select it and then click and drag the frame handles.
ADDITIONAL INFORMATION: Click **Reset** to take another measurement.
 14. Click **Close** to close the Measurement Entities dialog box.
-

Measuring Vertex Coordinates

The **Vertex Coordinates** option provides the coordinates of vertices on the model.

TASK

1. From the **Markup** menu, select **Add Entity, Measurement**, and then **Vertex Coordinate**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Vertex Coordinate** .
STEP RESULT: The Measurement Entities dialog appears. All vertices on the model are highlighted.
2. Scroll over the vertex whose coordinates you want to add to the markup.
STEP RESULT: The X, Y, and Z coordinates appear in a tooltip.

- Click the vertex.

STEP RESULT: The X, Y, and Z coordinates and unit appear in a value box entity on the current active markup layer and in the Measurement Entities dialog.

- Click and drag the value box anywhere on the markup.

ADDITIONAL INFORMATION: Click **Reset** to take another measurement.

STEP RESULT: Click and drag the frame handles to enlarge the value box.

- To remove the highlighted vertices on the model, click **Close** in the Measurement Entities dialog.

Adding Text

With AutoVue, you can add a 3D text box entity to a markup.

TASK

- From the **Markup** menu, select **Add Entity**, and then select **Text**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **3D Text** .

- The Attach To dialog appears and provides six snapping options:

ADDITIONAL INFORMATION:


Option	Description
None	Does not snap to the model.
Vertex	Snaps to a vertex on the model.
Edge	Snaps to an edge on the model.
Face	Snaps to a face on the model.
Mid Edge	Snaps to the middle of an edge on the model.
Ar Center	Snaps to the center point of an arc on the model.

- To insert a text box that snaps to a part of the drawing with a leader line, select one of the snapping modes.
- Click and drag on the drawing to define the dimensions of the text box.
- Click in the text box and enter the text you want to add.


ADDITIONAL INFORMATION: The height of the text box will change to accommodate the text.

6. To change text box font properties, from the **Markup** menu, select **Format**, and then select **Font**.
ADDITIONAL INFORMATION: From the respective font property lists on the Markup Properties toolbar, you can also modify the font type, style, and size.
STEP RESULT: The Font dialog appears allowing you to change the font type, style, and size.
 7. Click **OK** to implement the font changes and to close the Font dialog.
 8. To change the line properties or fill color of the text box, select the text box, then from the **Markup** menu, select **Format**, and then select the property or properties you want to change.
ADDITIONAL INFORMATION: From the respective line property lists on the Markup Properties toolbar, you can also modify the line style, line thickness, fill types, and fill colors.
 9. Right-click outside the text area to complete the modification.
STEP RESULT: The text appears on the drawing and in the Markup Navigation Tree.
 10. To move the text box, click and drag it to anywhere on the drawing.
 11. To resize the text box, select it and then click and drag the frame handles.
ADDITIONAL INFORMATION: To edit the text, double-click the text box in the tree or in the workspace.
-

Adding a Note

You can attach longer comments as a markup note with AutoVue's Markup Control Note Editor. A note displays in the workspace as a standard size graphical symbol . Each note is labelled as *Note<n>*—where *n* represents the numerical order of occurrence of the note (for example, the first note is labelled as *Note1*). To read the note, double-click the entity to open it or move the mouse over the entity to display the tooltip.

TASK


1. From the **Markup** menu, select **Add Entity**, and then select **Note**.
ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Note** .
2. Click a point on the document where you want to insert the note.
STEP RESULT: The Markup Control Note Editor dialog box appears.
3. Enter the text that you want in the note.
4. To modify the text, select **Formatting** and the format you want to apply.
5. From the **File** menu, select **Information**.
STEP RESULT: The Note Information dialog box appears.
6. In **Name**, **Author** and **Key Words** fields, enter the required text for the note.
7. Click **OK**.

8. Close the Markup Control Note Editor dialog box.

STEP RESULT: The note appears on the drawing and in the Markup Navigation Tree.

9. Right-click outside the note area to complete the modification.

10. To move the note, click and drag it.

ADDITIONAL INFORMATION: To edit a note, double-click the note symbol  in the workspace. You can also select the note in the tree, right-click, and select **Edit**, or double-click the note.

Viewing and Printing the List of Notes

You can view a list of the notes you added to a file. You can also print the notes or find a note in the workspace.

TASK

1. From the **Markup** menu, select **Print Notes**.

STEP RESULT: The List Notes dialog box appears.

2. To find a note, select the note from the list and click **Go To**.

ADDITIONAL INFORMATION: You can also select the note from the Markup Navigation Tree, right-click, and select **Go To**.

STEP RESULT: AutoVue zooms into the selected note in the workspace.

3. To print a note, select the note from the list.

ADDITIONAL INFORMATION: To print more than one note, press the **Shift** or **Control** key while selecting. To print all the notes in the list, click **Select All**.

4. Click **Print**.

ADDITIONAL INFORMATION: To print each selected note on a separate page, select **One Note Per Page**.

STEP RESULT: The Note Print Options dialog box appears.

5. Configure the print options, then click **OK**.

STEP RESULT: The selected notes are printed with their unique headers (for example, Note1, Note2, and so on.)



6. Click **OK** to close the List Notes dialog box.

Nesting Markup Entities

You can add the Attachment or Hyperlink markup entities as a nested child to any markup entity.

TASK

1. Add a markup entity to a 3D file (for example, a Text entity).
2. From the workspace or Markup Tree, select the Text entity.
3. From the **Markup** menu, select **Attachment** or **Hyperlink**.

ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Attachment**  or **Hyperlink** .

RESULT:

In the Markup Tree, the selected markup entity appears as a child of the Text (parent) entity.

Working with Markup Entities

With AutoVue, you can assign a markup entity its own color, the same color as the current active layer, or a custom color. You also have the option of grouping markup entities. When you group markup entities, you can manage the group as you would a single entity.

In Markup mode, there are several options for modifying an entity. You can apply these options to selected pre-existing entities or to new entities that you add.

NOTE: You can edit certain markup entities after they have been created. To do so, in the Markup Navigation Tree, right-click the markup entity and then select **Edit**.

Go To a Markup Entity

The **Go To** feature restores the view state to when an entity was first created. Opening an existing Markup file also restores the last saved view state.

From the Markup Navigation Tree, right-click the markup entity that you want to view and select **Go To**.

AutoVue displays the markup page containing the entity.

If you selected a markup entity that is on another page of the Markup file, the page containing that entity will be displayed.

Selecting Markup Entities

To select a markup entity, click the markup entity's outer edge.

To select multiple entities, press the **Shift** or **Control** key while selecting. The markup entity or entities are selected. You can also select the markup entities from the Markup Navigation Tree. To select multiple entities, press the **Shift** or **Control** key while selecting.

Moving a Markup Entity

In the Markup Navigation Tree or in the workspace, select the markup entity or entities that you want to move.

To select multiple entities, press the **Shift** or **Control** key while selecting.

In the workspace, click and drag the selected markup entity or entities to anywhere in the workspace.

Transforming Markup Entities

NOTE: This menu option is only available for 2D and EDA files.

AutoVue provides the option to flip or rotate markup entities.

Rotating all Markup Entities

From the **Markup** menu, select **Transform**, and then select **Rotate**. From the menu that appears, you have one of two options to select from:

- **Rotate Clockwise:** Rotates the markup entity 90 degrees in the clockwise direction.
- **Rotate Counter Clockwise:** Rotates the markup entity 90 degrees in the counter-clockwise direction.

NOTE: The markup entities rotate with respect to the center point of the drawing.

Rotating a selected Markup Entity

This feature is only supported for the Text and Stamp markup entities.

TASK

1. From the workspace or the Markup Navigation Tree, select the markup entity to rotate.
ADDITIONAL INFORMATION: It is not possible to select multiple markup entities to rotate.
 2. From the **Markup** menu, select **Object**, and then select **Rotate**.
 3. Rotate the markup entity by clicking and dragging its control points. Pressing the Shift key while dragging will snap the rotation to 45 degree intervals.
-

Flipping all Markup Entities

From the **Markup** menu, select **Transform**, and then select **Flip**. From the menu that appears, you have one of two options to select from:

- **Flip Horizontal Axis:** Flips the direction of the horizontal axis of the markup entity.
- **Flip Vertical Axis:** Flips the direction of the vertical axis of the markup entity.

NOTE: The horizontal and vertical axes are aligned with the center of the drawing.

Hiding All Markup Entities

To hide all markup entities, do the following:

TASK

1. From the **Markup** menu, select **Hide Markups**.
ADDITIONAL INFORMATION: A check mark appears next to the Hide Markups option and the markup entities are hidden on the Markup file.
 2. To undo the hide option, from the **Markup** menu, deselect **Hide Markups**.
-

Grouping and Ungrouping Markup Entities

When you group markup entities, you can move, delete, copy and paste, transform, or perform any modification on the group of entities that you would on a single markup entity.

NOTE: You can only group markup entities created on the same page.

Grouping Markup Entities

TASK

1. In the Markup Navigation Tree or from the workspace, select the markup entities that you want to group.
2. From the **Markup** menu, select **Object**, and then select **Group**.

ADDITIONAL INFORMATION: You can also right-click the selected markup entities and then select **Group**.

STEP RESULT: The group of entities appear in the tree under *Group*.

3. Perform any modifications.

RESULT: The modifications are applied to all the entities in the group.

Ungrouping Markup Entities


TASK

1. In the Markup Navigation Tree or the workspace, select the group that you want to ungroup.
2. From the **Markup** menu, select **Object**, and then select **UnGroup**.

RESULT: The group of entities appear on the markup as individual entities.

Deleting Markup Entities

TASK

1. Select the markup entity that you want to delete. To select multiple markup entities, press the **Shift** or **Control** keys while selecting.
2. From the Markup Properties toolbar, click **Delete Markups** .

RESULT:

The selected entity or entities are deleted from the current active file.

*NOTE: You can also press the **Delete** key to delete the markup entities. Alternatively, right-click the entity in the workspace or Markup Navigation Tree, and select **Delete**.*

Formatting Markup Entity Properties

With AutoVue, you can modify the formatting of a markup entity via the **Format** option from the **Markup** menu, or via the Markup Entity Properties dialog.


See “Using the Markup Entity Properties Dialog”

When creating a markup entity, you have the option to change the line color, line style, line thickness, arrow style, fill color, and fill type. Additionally, you can assign the markup entity the same color as the layer.

Changing Line Color

TASK


1. Select the markup entity or entities for which you want to change the line color.
2. From the **Markup** menu, select **Format**, and then select **Line Color**.

ADDITIONAL INFORMATION: From the Markup Properties toolbar, you can also click **Line Color** . Alternatively, right-click the entity in the workspace or Markup Navigation Tree, select **Format**, and then select **Line Color**.

STEP RESULT: The Line Color dialog appears.

3. From the Line Color list, select the color that you want for the entity.

ADDITIONAL INFORMATION: Selecting **Bylayer**  changes the markup entity color to the color of the layer.

4. To define a custom line color, from the Line Color list, select **Custom Color** .
 5. From the Color dialog that appears, select a color and click **OK**.
 6. Click **OK** to close the Line Color dialog.
-

RESULT:

The line color changes for the selected markup entity or entities.

NOTE: Any new entities that you create will have the new line color.

Changing Line Style


The current line style is highlighted in the **Line Style** option.

TASK

1. Select the markup entity or entities for which you want to change the line style.
 2. From the **Markup** menu, select **Format**, and then select **Line Style**. Select the new line style from the options provided.
-

RESULT:

The line style changes for the selected markup entity or entities.

NOTE: From the Markup Properties toolbar, you can also click **Line Style** . Alternatively, right-click the entity in the workspace or Markup Navigation Tree, select **Format**, and then select **Line Style**.


NOTE: Any new entities that you create will have the new line style.

Changing Line Thickness

The current line thickness is highlighted in the **Line Thickness** option.

TASK

1. Select the markup entity or entities for which you want to change the line thickness.
2. From the **Markup** menu, select **Format**, and then select **Line Thickness**. Select the new line thickness from the options provided. The line thickness changes for the selected markup entity or entities.

ADDITIONAL INFORMATION: From the Markup Properties toolbar, you can also click **Line Thickness** . Alternatively, right-click the entity in the workspace or Markup Navigation Tree, select **Format**, and then select **Line Thickness**.

3. To define a custom line thickness, from the **Markup** menu, select **Format**, select **Line Thickness**, and then select **Customize**.

STEP RESULT: The Custom Thickness dialog appears.

4. In the **Line Thickness** field, enter an integer value in pixels.
5. Click **OK** to close the Custom Thickness dialog.

ADDITIONAL INFORMATION: Any new markup entities that you create will have the new line thickness.

Changing Arrow Style


AutoVue gives you the option to add an arrow head to certain markup entities. For example, you can change the arrow style of the line, polyline, arc, freestyle, and polygon markup entities.

TASK

1. Select the markup line entity or entities for which you want to change the arrow style.
 2. From the **Markup** menu, select **Format**, and then select **Arrow Style**. Select the new arrow style from the options provided.
-

RESULT:

The arrow style changes for the selected markup line entity or entities.


NOTE: From the Markup Properties toolbar, you can also click **Arrow Style** . Alternatively, right-click the entity in the workspace or Markup Navigation Tree, select **Format**, and then select **Arrow Style**.

NOTE: Any new entities that you create will have the new arrow style.

Changing Fill Type

TASK

1. Select the markup entity or entities for which you want to change the fill type.
2. From the **Markup** menu, select **Format**, and then select **Fill Type**.

ADDITIONAL INFORMATION: From the Markup Properties toolbar, you can also click **Fill Type** . Alternatively, right-click the entity in the workspace or Markup Navigation Tree, select **Format**, and then select **Fill Type**.

STEP RESULT: The Fill Type dialog appears.


3. From the Fill Type list, select the fill type that you want for the markup entity:
 - Select **Solid Fill**, if you want the fill color to be solid.
 - Select **Transparent Fill**, if you want the fill color to be transparent.
 - Select **No Fill**, if you do not want any fill color.
 4. Click **OK**.
-

RESULT: The fill type changes for the selected markup entity or entities. Any new markup entities that you create will have the new fill type.

Changing Fill Color

TASK


1. Select the markup entity or entities for which you want to change the fill color.
2. From the **Markup** menu, select **Format**, and then select **Fill Color**.

ADDITIONAL INFORMATION: From the Markup Properties toolbar, you can also click **Fill Color** . Alternatively, right-click the entity in the workspace or Markup Navigation Tree, select **Format**, and then select **Fill Color**.

STEP RESULT: The Fill Color dialog appears.

3. From the Fill Color list, select the color that you want for the markup entity.

ADDITIONAL INFORMATION: Selecting **Bylayer**  changes the markup entity color to the color of the layer it is on and selecting **Line Color**  assigns the same color as the markup entity's line.

4. To define a custom line color, from the Line Color list, select **Custom Color** .
5. From the Color dialog that appears, select a color and click **OK**.

STEP RESULT: The fill color changes for the selected markup entity or entities.

6. Click **OK** to close the Fill Color dialog.


ADDITIONAL INFORMATION: Any new markup entities that you create will have the new fill color.


Assigning a Markup Entity the Same Color as the Layer


TASK


1. Select the markup entity or entities for which you want to assign the color of the layer that they belong to.

2. To assign the line color:
 - From the **Markup** menu, select **Format**, and then select **Line Color**. The Line Color dialog appears.

*NOTE: From the Markup Properties toolbar, you can also click **Line Color** . Alternatively, right-click the entity in the workspace or Markup Navigation Tree, select **Format**, and then select **Line Color**.*

 - From the Line Color list, select **Bylayer** .
 3. To assign the fill color:
 - From the **Markup** menu, select **Format**, and then select **Fill Color**. The Fill Color dialog appears.

*NOTE: From the Markup Properties toolbar, you can also click **Fill Color** .*

 - From the Fill Color list, select **Bylayer** .
 4. Click **OK**.
-

RESULT: The selected markup entity or entities change to the color of the layer that the entity belongs to.

Changing Font

You can change the font of the text box, leader, and measurement entities.

TASK

1. Create the markup entity that you want.
2. From the workspace or the Markup Navigation Tree, select the markup entity for which you want to modify the font then do one of the following:
 - From **Markup** menu, select **Format**, and then select **Font**. The Font dialog appears. From the lists select the font, font style, and font size. You also can

select to add a strikethrough and underline by selecting their check boxes. Click **OK** to close the dialog and implement the changes.

- From the Markup Entity toolbar, select the font, font size, and font style (bold, italic, and underline) from their respective lists and buttons. The font changes are implemented.

Changing Measurement Units and Symbols

You can change the unit of measure and add a symbol to a measurement and have it appear on the drawing.

TASK

1. Create the markup measure entity that you want.
2. Double-click the measurement that you want to change the unit of measure or add a symbol to.
STEP RESULT: The appropriate Measure dialog appears.
3. Deselect **Display Unit** check box if you want to hide the unit on the drawing. It is selected by default.
4. From the Units list, select the unit that you want to change the measurement to.
5. From the Symbol list, select the symbol that you want to add to the measurement.
6. Click **OK**.

RESULT: The unit of measure changes and the selected symbol is added to the measurement and appears in the workspace.

Using the Markup Entity Properties Dialog

You can use the Markup Entity Properties dialog to modify markup entities.

TASK

1. Select the markup entity you want to modify. To select multiple markup entities, press the **Shift** or **Control** key while selecting.
2. From the **Markup** menu, select **Format**, and then select **Markup Entity Attributes**.

ADDITIONAL INFORMATION: You can also right-click a markup entity in the Markup Navigation Tree or the workspace, select **Format**, and then select **Markup Entity Attributes**.


STEP RESULT: The Markup Entity Properties dialog appears.

Line Color

Change the line color the selected markup entity or entities.

TASK

1. Select the markup entity or entities for which you want to change the line color.
2. From the Line Color list, select the color that you want for the entity.

ADDITIONAL INFORMATION: Selecting **Bylayer**  changes the entity color to the color of the layer.

3. Click **OK** to close the Markup Entity Properties dialog.
-

Defining a Custom Line Color

To define a custom line color, take the following steps:

TASK

1. From the Line Color list, select **Custom Color** .

STEP RESULT: The Color dialog appears.

2. Select a color and click **OK**.
3. Click **OK** to close the Markup Entity Properties dialog.

ADDITIONAL INFORMATION: If you selected markup entities prior to specifying a custom color, only the selected entities will have the new line color. To apply the new line color to any new markup entity you create, make sure no markup entity is selected before opening the Markup Entity Properties dialog.

RESULT:

The line color changes for the selected markup entity or entities.

Line Style

Change the line style for a selected markup entity or entities.

TASK

1. Select the markup entity you want to change the line style. To select multiple markup entities, press the **Shift** or **Control** key while selecting.
2. From the Line Style list, select the required line style.

- Click **OK** to close the Markup Entity Properties dialog.

ADDITIONAL INFORMATION: Any new markup entities that you create will have the new line style.

RESULT:

The line style changes for the selected markup entity or entities.

Line Thickness

Change the line thickness for a selected markup entity or entities.

TASK

- Select the markup entity for which you want to change the line thickness. To select multiple markup entities, press the **Shift** or **Control** key while selecting.
- From the Line Thickness list, select the required line thickness.

STEP RESULT: The line thickness changes for the selected markup entity or entities.

Defining a Custom Line Thickness

To define a custom line thickness, take the following steps:

TASK

- From the Line Thickness list, select **Customize**.
- In the **Width (Pixels)** field, enter the desired width.
- Click **OK** to close the Markup Entity Properties dialog.

RESULT:

The line thickness changes for the selected markup entity or entities.

NOTE: Any new markup entities that you create will have the new line thickness.

Width (Pixels)

Customize the line thickness when **Customize** is selected in the Line Thickness list. When other line thicknesses are selected, this field displays its value in pixels but cannot be edited.

Arrow Style

Add an arrow head at one or both ends of a markup line entity.

TASK

1. Select the markup line entity you want to change the arrow style. To select multiple markup line entities, press the **Shift** or **Control** key while selecting.
 2. From the Arrow Style list, select the style of arrow that you want for the markup line entity.
 3. Click **OK** to close the Markup Entity Properties dialog.
-

RESULT:

The arrow style changes for the selected markup line entity or entities.

NOTE: Any new markup line entities that you create will have the new arrow style.

Fill Type

Change the transparency for selected markup entity or entities.

TASK

1. Select the markup entity you want to change the fill type. To select multiple markup entities, press the **Shift** or **Control** key while selecting.
 2. From the Fill Type list, select the fill type that you want for the markup entity:
 - Select **Solid Fill**, if you want the fill color to be solid.
 - Select **Transparent Fill**, if you want the fill color to be transparent.
 - Select **No Fill**, if you do not want any fill color.
 3. Click **OK** to close the Markup Entity Properties dialog.
-

RESULT:

The fill type changes for the selected markup entity or entities.

NOTE: Any new markup entities that you create will have the new fill type.

Fill Color

Change the line color, fill color, and fill type for a selected markup entity or entities.

TASK

1. Select the markup entity you want to change the fill color. To select multiple markup entities, press the **Shift** or **Control** key while selecting.
2. From the Fill Color list, select the color that you want for the markup entity.

ADDITIONAL INFORMATION: Selecting **Bylayer**  changes the markup entity color to the color of the layer.

Defining a Custom Color

To define your own color, take the following steps:

TASK

1. From the Fill Color list, select **Custom Color** .

STEP RESULT: The Color dialog appears.

2. Select a color and click **OK**.

STEP RESULT: The fill color changes for the selected markup entity or entities.

3. Click **OK** to close the Markup Entity Properties dialog.
-

RESULT:

The fill type changes for the selected markup entity or entities.

NOTE: Any new markup entities that you create will have the new fill color.

Markup Layers

You can move selected markup entities to a specific layer—the markup entities will inherit the properties of the layer.

TASK

1. From the Markup Layer list, select the layer you want to make active.
 2. Click **OK** to close the Markup Entity Properties dialog.
-

RESULT: The selected layer appears in the workspace.

See “Setting the Active Markup Layer”

Leader Alignment

With the Leader markup entity, you can adjust the location of the leader line's anchor point to the text box. The Leader Alignment list has the following option for anchor point location with respect to the text box: Top Left, Top Center, Top Right, Center Left, Center, Center Right, Bottom Left, Bottom Center, and Bottom Right.

Text Box Visibility

You can choose to hide the surrounding box for the Leader, Text, and 3D Text markup entities.

Printing

With AutoVue, you can print and preview files. Original files can be printed on their own or with associated markups and overlays. You can also choose which markup layers to make visible so that they can print.

With the **Batch Print** option, you can send a list of files to print at the same time.

When printing a file, there are print properties that you can define. From the **File** menu, select **Print** to display the following properties:

Option	Description
Print	Define the print options. See “Print Options”
Margins	Define the margin settings. See “Print Margins”
Headers/Footers	Define the headers/footers to be included on every page printed. See “Header/Footer”
Watermark	Define the watermark to be included on every page printed. See “Watermarks”
Stamps	Define the stamp to be included on every page printed. See “Adding a Stamp”
Pen Settings	Change the thickness assigned to a pen. See “Assigning Pen Settings”

Print Options

With the **Options** tab, you can define print options for the printed file. The options are:

Option	Type	Description
Print to File		Click to print to file and not to a printer.
Modify		Sets printer properties, such as printer, paper size, orientation, and so on.
Scaling	Fit to page	Scales the image to fit on the output page.
	Factor	Scales the image according to the scaling factors that you have manually entered in the input fields. Units can be entered as inches, millimeters or pixels.
	Scale	Scales the image according to percentage. You can select a predefined scaling factor or manually enter a customer scaling factor. Decimal places are excepted.
Document Pages		Sets the document pages to print: All , Current or Range .
Alignment/Offset		Specifies where the drawing will appear on the print out.
Page Area	Extents	Prints the extents of the document.
	Displayed	Prints the area displayed in the View window.
	Limits	Limits printing to one page when the scaling options selected causes a single document page to span over multiple printer pages. Option is only enabled when Current is selected for Document Pages.
	Selected	Prints the area selected in the View window. Option will only be enabled when Current is selected for Document Pages.
Force all colors to black		Forces all colors to black.
Output a single page		Prints file on a single page.
Print Row Headers		Prints row headers. Option is only enabled for spreadsheets, archive and database files.
Print Column Headers		Prints column headers. Option is only enabled for spreadsheets, archive and database files.
Partial Preview		Displays a view of the printer page, highlighting the printable area.

Option	Type	Description
Number of Copies		Select the number of copies you want to print of the file.

Configuring the Print Options

TASK

1. From the **File** menu, select **Print**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print** .

STEP RESULT: The Print Properties dialog appears.

2. Click the **Options** tab.
3. Configure the print options.
4. Configure other print options.
5. To view a partial view of the file, click **Partial Preview**.

STEP RESULT: The Partial Print Preview dialog appears.

6. Click **OK** to print.

RESULT: The Print dialog appears indicating the document is printing.

Print Margins

With the **Margin** tab, you can define the top, bottom, right and left margins for the printed file. You can define:

Option	Description
Margins	Set the value for the Left , Top , Right , and Bottom margins.
Minimum	Sets the acceptable minimum margins for the selected printer.
Override printer-minimum margins	If selected, overrides the minimum margins for the selected printer.

Option	Description
Units	Specify the unit for the margins.

Setting the Margins

TASK

1. From the **File** menu, select **Print**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print** .

STEP RESULT: The Print Properties dialog box appears.

2. Click the **Margins** tab.
3. Enter the margin size that you want for **Left**, **Top**, **Right**, and **Bottom** or click **Minimum** if you want to set the margins to the minimum acceptable for the selected printer.

ADDITIONAL INFORMATION: When you click **Minimum**, the minimum margins allowable automatically appear in the **Margins**.

4. Select **Override printer-minimum margins** if you want to override the minimum margins set for the printer.
5. From the **Units** list, select the unit to which you want to set the margins.
6. Configure other print options.
7. To view a partial view of the file, click **Partial Preview**.

STEP RESULT: The Partial Print Preview dialog box appears.

8. Click **OK** to print.

RESULT: The Print dialog box appears indicating the document is printing.

Header/Footer

From the **Headers/Footers** tab of the Print Properties dialog box, you can define the headers and footers that you want to print on every page of the document. You can enter the text manually or choose from a list of **Insert Codes**.

The list of insert codes are:

- %f: Full path of document

- %v: Document Drive
- %d: Document Directory
- %b: Document Base name
- %e: Document file extension
- %n: Total document pages
- %p: Current page number
- %N: Total tiled-pages
- %P: Current tile number
- %Y: Date: Year
- %M: Date: Month
- %D: Date: Day
- %W: Date: Day of week
- %H: Time: Hour
- %U: Time: Minute
- %S: Time: Seconds
- %r: New line
- %F: Native Print Settings (Excel)

NOTE: A literal percentage mark is entered manually as %%.

Adding a Header and Footer

In the Headers/Footers tab, you can manually enter text or choose from a list of Insert Codes to appear in the headers and footers. You can also print system variables.

TASK

1. From the **File** menu, select **Print**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print** .

STEP RESULT: The Print Properties dialog appears.

2. Click the **Headers/Footers** tab.
3. Click in the **Left**, **Center** or **Right** field and enter the text that you want to appear in the header and footer, or choose an **Insert Code** or **Insert Variable**.
4. To **Insert Code** in the header or footer, click **Left**, **Center** or **Right**, then select a code from the list.

5. To **Insert Variable** in the header or footer, click **Left**, **Center** or **Right**, then select a variable from the list.
 6. To change the Font, click **Set Font**. The Font dialog appears.
 7. From the Font list, select the type of font.
 8. From the Size list, select the size of font.
 9. To set the font **Style**, click **Bold** or **Italic** or both, then click **OK**.
 10. In the Print Properties dialog, configure other print properties.
 11. To view a partial view of the file, click **Partial Preview**.
STEP RESULT: The Partial Preview dialog appears.
 12. Click **OK** to print.
-

RESULT: The Print dialog appears indicating the document is printing.

Native Print Settings

You can choose to include predefined headers and footers when printing an Excel file from AutoVue.

TASK

1. From the Print Properties dialog, click the **Headers/Footers** tab.
 2. Under **Headers** and **Footers**, click inside the **Left**, **Center**, and **Right** boxes corresponding to the header or footer position in the original file.
 3. Select **%F: Native Print Settings (Excel)** from the **Insert Code** list.
 4. Click **OK** to print.
-

RESULT: The Excel file's headers and footers are printed.

Watermarks

With the **Watermark** tab, you can specify a watermark that will appear on a printed file. When printing a watermark, it appears transparent on the file contents. You can choose a horizontally or vertically oriented watermark.

In addition to entering text, you can:

- set the type, size and style of font
- insert Watermark information
- print system variables

To insert Watermark information, AutoVue provides a list of codes that you can choose from. These codes are:


- %f: Full path of document
- %v: Document Drive
- %d: Document Directory
- %b: Document Base name
- %e: Document file extension
- %n: Total document pages
- %p: Current page number
- %N: Total tiled-pages
- %P: Current tile number
- %Y: Date: Year
- %M: Date: Month
- %D: Date: Day
- %W: Date: Day of week
- %H: Time: Hour
- %U: Time: Minute
- %S: Time: seconds
- %r: New line

NOTE: A literal percentage mark is entered manually as %%.

Adding a Watermark

TASK

1. From the **File** menu, select **Print**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print** .

STEP RESULT: The Print Properties dialog box appears.

2. Click the **Watermark** tab.

3. Enter the **Watermark Text** that you want to appear on the file.
 4. To insert a code in the watermark, select a code from the **Insert Code** list.
 5. For example, if you select **%n: Total document pages**, the total number of pages for the selected file will appear in the watermark.
ADDITIONAL INFORMATION: You can insert more than one code.
 6. To set the orientation of the watermark, click **Diagonal, Horizontal, or Vertical**.
 7. To change the Font, click **Font**. The Font dialog box appears.
 8. Configure the font, then click **OK**.
 9. In the Print Properties dialog box, configure other print properties.
 10. To view a partial view of the file, click **Partial Preview**.
STEP RESULT: The Partial Print Preview dialog box appears.
 11. Click **OK** to print.
-


RESULT:

The Print dialog box appears indicating the document is printing.

Adding a Stamp

With the **Stamps** tab, you can set the position and size of a stamp to print over the current file. You also have the option to include the stamp on every page of the printed file. When printing, the stamp hides any area of the print file that it covers.

TASK

1. From the **File** menu, select **Print**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print** .
STEP RESULT: The Print Properties dialog box appears.
2. Click the **Stamps** tab.
3. Click **Add**.
STEP RESULT: The Open dialog box appears.
4. Enter a file name or browse to locate the file that you want to use as a stamp.
STEP RESULT: The file appears in the Print Properties dialog box.
5. Select the **Page selections** to specify the pages that you want the stamp to appear on.

6. To set the **Position** of the stamp, enter the width in the first field, then length in the second.
ADDITIONAL INFORMATION: You can also change the position by clicking and dragging the box in the **Preview** window.
7. To set the **Size** of the stamp, enter the width in the first field, then length in the second.
ADDITIONAL INFORMATION: You can also change the size by clicking and dragging the handles on the box in the **Preview** window.
STEP RESULT: The size of the stamp is represented in the **Preview** area.
8. From the **Units** list, select the unit in which you want to set for the width and length of the stamp.
9. To add more than one stamp to a file, repeat steps 3 to 8.
10. To remove a stamp, select the stamp and click **Remove**.
11. In the Print Properties dialog box, configure other print properties.
12. To view a partial view of the file, click **Partial Preview**.
STEP RESULT: The Partial Print Preview dialog box appears.
13. Click **OK** to print.


RESULT: The Print dialog box appears indicating the document is printing.

Assigning Pen Settings

From the **Pen Settings** tab of the Print Properties dialog box, you can set the thickness of the color indices for the print file.

NOTE: AutoVue uses the default pen color indices of the native document for the vector file being viewed. You will not be able to change the color assigned to a pen with AutoVue.

TASK

1. From the **File** menu, select **Print**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print** .
STEP RESULT: The Print Properties dialog box appears.
2. Click the **Pen Settings** tab.

3. Under **From/To**, select the **Color Index** that you want to assign a new pen thickness.

ADDITIONAL INFORMATION: To change more than one color indices, press the **Shift** or **Control** key while selecting.

4. From the **Units** list, select the unit of measure that you want to set the thickness.
5. Enter the new thickness.

STEP RESULT: The new **Thickness** appears beside the **Color Index**.

6. In the Print Properties dialog box, configure other print properties
7. Configure other print options.
8. To view a partial view of the file, click **Partial Preview**.

STEP RESULT: The Partial Print Preview dialog box appears.

9. Click **OK** to print.

STEP RESULT: The Print dialog box appears indicating the document is printing.

RESULT:

NOTE: Pen settings are stored in **.c2t** files. The **Current Pen Settings** list displays all existing **.c2t** files for the current file. The default pen settings are stored in **default.c2t** and alternate pen settings are stored in **alt.c2t**.

Creating a New Pen Setting

TASK

1. Set the thickness for the color indices that you want.
2. Click **Save As**. The Save As dialog appears.
3. Enter a file name with a **.c2t** extension, then click **OK**.

STEP RESULT: The new pen setting appears in the **Current Pen Settings** list.

4. In the Print Properties dialog, click **OK** to print.
-

RESULT: The Print dialog appears indicating the document is printing.

Deleting a Pen Setting

TASK

1. Select the pen setting that you want to delete from the **Current Pen Settings**.

2. Click **Delete**.
STEP RESULT: The pen setting disappears from the list.
3. Click **Cancel** to close the Print Properties dialog.

Partial Preview of a File

The Partial Print Preview dialog displays the printable area on top of the page area so that users can have a clear idea of what will be printed.

*NOTE: Option is only enabled when **Current** is selected from **Document Pages**.*


TASK

1. Configure the print properties.
 2. Click **Partial Preview** in the Print Properties dialog.
STEP RESULT: The Partial Print Preview dialog appears and highlights the area that will be printed. The **Paper size**, **Printable Area**, and **Drawing Area** are displayed.
 3. Click **OK** to close the Partial Print Preview dialog.
-

Previewing a File Before Printing

You can preview a print copy of the current active file on screen according to your printer's capabilities and the print property settings.

TASK

1. View the file in AutoVue.
2. From the **File** menu, select **Print Preview**.
ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print Preview** .
STEP RESULT: The Print Properties dialog box appears.
3. Configure the print properties.

4. Click **OK**.

ADDITIONAL INFORMATION: You can print the file from the **Print Preview** window by clicking **Print**. You can also zoom in and out of a file, as well as navigate from one page to another of a multi-page file.

STEP RESULT: The file appears in Print Preview Mode in the **Print Preview** window.

5. Click **Close** to close the **Print Preview** window.
-

Printing a File

You can print original files along with their Markup files and selected markup layers together so that they appear as one file.

TASK

1. Open the file you want to print.
2. To print the associated markups, open the Markup file or files you want to print.
3. If you are printing Markup files and you want to print the visible layers, from the **Markup** menu, select **Markup Layers**.

STEP RESULT: The Markup Layers dialog appears.

4. Select the markup layers you want visible.
5. Click **OK** to close the Markup Layers dialog.
6. From the **File** menu, select **Print**.

ADDITIONAL INFORMATION: From the AutoVue toolbar, you can also click **Print** .

STEP RESULT: The Print Properties dialog appears.

7. Configure the print properties.
 8. To view a partial view of the file, click **Partial Preview**.
 9. Click **OK** to print.
-

RESULT: The Print dialog appears indicating the document is printing.

Batch Printing

With the **Batch Print** option you can send a list of files to print at the same time. You can also generate a batch by simultaneously opening all the files included in the batch print.

TASK

1. From the **File** menu, select **Batch Print**.

STEP RESULT: The Batch Print dialog appears.

2. To add a file to be printed, click **Add**.

STEP RESULT: The Open dialog appears.

3. Enter the file name or browse to locate the file or files that you want to add, then click **Open**.

ADDITIONAL INFORMATION: To add more files to the list, repeat steps 2 to 4. To remove a file(s), select the file from the **List of files to be printed** and click **Remove**.

STEP RESULT: The files appear in the **List of files to be printed**.

4. Click **OK** in the Batch Print dialog.

STEP RESULT: The Print Properties dialog appears.

5. Configure the print properties.

6. Click **OK** to print.
-

RESULT: The Print dialog appears indicating the document is printing.

Conversion

Sometimes you need to convert a file to be able to use it with an application it was not created from. AutoVue provides several conversion file formats for you.

Conversion Options

Depending on the conversion type being performed, the available options in the Convert dialog box will vary. These options are:

Option	Description
Save As	Specify the name and path of the file in which the conversion is to be stored. This file is also known as the output file. You can use Browse to provide AutoVue with the file's path.
Convert to Format	A list of all the possible types of output file formats currently available for conversion. The available formats are: CALS GP4 Encapsulated Postscript (Raster) HP Laserjet Printer (PCL) Run Length RLC File PCX Bitmap PDF Stereolithography (STL) TIFF Virtual Reality Modeling Language (VRML) Windows Bitmap
Sub-Format	The Sub-Format list appears when you select TIFF or STL from the Convert to Format list. Select the Sub-Format from the list.

Option	Description
Color Depth	Select an option from the list: 1 = Black and white 4 = 16 colors 8 = 256 colors 24 = True color auto = AutoVue selects the color depth that best matches the original file. Option is not available for STL and VRML formats.
Positive Triangle Values	When selected, the model is translated so that all the vertex coordinate values are positive. Option is only available when Convert to Format is set to STL .
Convert Region	The area of the file to be converted. The available options are: All - converts the entire file Display - converts the image to fit on the output page For example, if you zoomed in on a particular region of the file, the zoomed portion of the file is converted. Extent - converts the entire extents of the file Selected - converts the selected model parts in the file Options vary depending on the selected format.
Convert Pages	The number of pages to be converted. The available options are: All - converts all pages of the file Current - converts the current page of the file Range - converts the pages of the file indicated in the range Option is only enabled when converting to TIFF .
X and Y	Specify X and Y to define the resolution for the converted file. X indicates the number of horizontal pixels Y is the number of vertical pixels for the current active file contents At times, AutoVue presets X and Y to match the specifications of the selected conversion file format.

PDF

With AutoVue it is possible to convert Office, 2D and EDA formats to PDF. When converted from Markup Mode, markups are added to the contents of the base PDF file. When you open the PDF, you will see the base file along with all markups.

X and Y

Three factors affect the resolution of an image: the type of image you are scanning, the output device and the acceptable file size. High-resolution scans often require large files, causing longer processing and print time. Note that a high-resolution may not produce a better-quality printed image if your output device does not recognize the higher resolution information stored in the file. To keep file sizes manageable, select the lowest resolution that provides acceptable quality on your output device.

With some file types, the **Size** option appears giving you a choice between millimeters and inches. Page sizes can be selected from the **Size** list or you can customize page sizes by configuring the Initialization file. For more information, refer to the *Installation and Configuration Guide*.

Technical Drawing Page Sizes	ISO Paper Format
A8.5" X 11.0" (216 mm X 279 mm)	A4 285 mm X 198 mm
B11.0" X 17.0" (279 mm X 432 mm)	A3 396 mm X 273 mm
C17.0" X 22.0" (432 mm X 559 mm)	A2 570 mm X 396 mm
D22.0" X 34.0" (559 mm X 864 mm)	A1 817 mm X 570 mm
E34.0" X 44.0" (864 mm X 1118 mm)	A0 1165 mm X 817 mm

Converting a File

TASK

1. From the **File** menu, select **Convert**.

ADDITIONAL INFORMATION: The **Input** area varies according to the type of file. For example, a word-processing file displays the file and format, a raster file displays the file path/name, format and size, and vector and database files display the size, file type and dimensions.

STEP RESULT: The Convert dialog appears.

2. In the **Save As** field, enter the path and filename or click **Browse** to locate the directory where you want to save the converted file.

ADDITIONAL INFORMATION: If writing on to an existing file, the contents of the output file will be overwritten.

3. Specify the conversion options.
4. Click **OK**. The file is converted and appears in the specified directory.

ADDITIONAL INFORMATION: You can convert several selected pages of a multi-page file to a multi-page TIFF.

Changing the Pen Settings

With AutoVue, you can specify a thickness for each pen color when doing a conversion. This option only applies to vector files.

TASK

1. From the **File** menu, select **Convert**.

STEP RESULT: The Convert dialog appears.

2. Click **Pens**.

STEP RESULT: The pen Thickness dialog appears.

3. In the **From/To** section of the Pen Thickness dialog, select the **Color Index** for which you want to modify the thickness.

ADDITIONAL INFORMATION: To select more than one Color Index, press the Shift or Control keys while selecting.

4. From the **Units** list, select the unit in which you want to set the thickness.

5. In the **Thickness** field, enter a thickness value.

6. Press the **Enter** key.

STEP RESULT: The new thickness value appears beside the selected Color Index in the From/To section of the Pen Thickness dialog.

7. To save the changes that you made, click **Save As**.

STEP RESULT: The Save As dialog appears.

8. Enter a file name for the new pen settings.

9. Click **OK**.

ADDITIONAL INFORMATION: To modify an existing pen setting, select the pen setting from the Current Pen Settings list, make the changes, then click Save.

STEP RESULT: The new pen settings are saved and appear in the Current Pen Settings list.

10. Click **OK** to close the Pen Thickness dialog.
-

Batch Conversion

You can save time by simultaneously converting files with the same output file format.

TASK

1. From the **File** menu, select **Convert**.

STEP RESULT: The Convert dialog box appears. The Convert dialog box appears. The Convert dialog box appears.

2. Click **Batch Convert**.

STEP RESULT: The Batch Convert dialog box appears.

3. Click **Add**.

STEP RESULT: The Open dialog box appears.

4. Enter the file name or browse to locate the file that you want to add, then click **Open**.

ADDITIONAL INFORMATION: To add more files, repeat steps 3 to 4. To remove a file, select the file from Input File List and click **Remove**.

STEP RESULT: The file appears in Input File List. The file appears in Input File List.

5. Customize the convert options.

6. Click **OK**.
-

RESULT:

AutoVue stores the newly converted files in the same directory as the first file you entered in Input File List.

Customizing AutoVue

There are configuration options that allow you to customize the AutoVue work environment. You can create tools to automate a variety of functions, including creating a hot key or access key, and also customize toolbars and thumbnails to display according to your preferences. You can access the configuration options from the **Options** menu.

Defining the Clipboard

With the **Clipboard Options** you can define the type of information copied to the clipboard for later transfer to other applications.

The types of information that you can copy for raster, vector, spreadsheet, document and database files are:

Type	Description
DIB	The acronym for Device Independent Bitmap and refers to the hardware information that is used to create a conversion bitmap.
Bitmap	A rectangular array of colors representing a pixel bit map image.
Palette	Refers to color image information.
Metafile	A file storing the commands necessary for a Windows application to recreate a vector format image.
Text	Refers to ASCII formatted text.
RTF	Acronym for Rich Text Format and refers to the formatting information found in documents that allow fonts, margins and tabs to remain in documents.
Include Headings	Refers to displaying the row and column titles for databases and spreadsheets.

To define clipboard options:

TASK

1. From the **Options** menu, select **Configure Clipboard**.
STEP RESULT: The Clipboard Options dialog box appears.
 2. Under the different file types, click the type of information that you want to copy to the clipboard.
 3. Click **OK**.
-

Copying Information

TASK

1. From the **Edit** menu, select **Select**.
 2. Click and drag to select the area you want to copy.
 3. From the **Edit** menu, select **Copy**.
STEP RESULT: The information is copied to the clipboard.
 4. Launch the application that you want to copy the information to.
 5. Paste the information.
-

Tools

Tools are shortcuts for commonly repeated tasks. You can create tools to automate a variety of function such as importing objects, files and applications into the user interface; running a script file; connecting AutoVue to Dynamic Data Exchange (DDE) servers and linking to Dynamic Link Library (DLL) loads.

Adding a Tools Menu Item

TASK

1. From the **Options** menu, select **Customize Tools**.
STEP RESULT: The Add Tool Menu Item dialog box appears.

2. Click **Add**.
STEP RESULT: The Configure Tools Menu Item dialog box appears.
 3. In the **Menu Item Name** field, enter a name.
 4. Select one of the following from the **Menu Item Action** section:
 - a **Application** to launch an outside application.
 - b **Script File** to launch a script file.
 - c **DDE Server** to send DDE commands to a server application.
 - d **DLL** to invoke a DLL.
 5. In the field to right of the **Menu Item Name** options, enter the name and path of the executable file. Otherwise, click **Browse** to locate the specific file or application.
 6. If you select **Application**, click **Configure**. The Link to Application dialog box appears. Enter the command line to launch an application, then click **OK**.
 7. If you select **Script File**, click **OK**.
 8. If you select **DDE Server**, click **Configure**. The Link to DDE dialog box appears. Enter the **Topic** and **Commands**, click **Start Application**, then click **OK**.
 9. If you select **DLL Load**, click **Configure**. The Link to DLL dialog box appears. Enter the **Entry Function** to invoke a DLL, then click **OK**.
 10. In the **Menu Item Description** field, enter a menu item description.
 11. Click **OK**.
 12. From the **Options** menu, click **Customize Tools** to view new menu item.
 13. Click **OK** to close the Add Tool Menu Item dialog box.
-

Creating a Hot Key

TASK

1. From the **Options** menu, select **Customize Tools**.
STEP RESULT: The Add Tool Menu Item dialog box appears.
2. Click **Add**.
STEP RESULT: The Configure Tools Menu Item dialog box appears.
3. Enter an ampersand (&) before the **Menu Item Name**.
ADDITIONAL INFORMATION: For example, if you entered the menu item name **&Application**. The letter **A** after the **&** will be the hotkey.
4. Click **OK**.

5. From the **Options** menu, click **Customize Tools** to view new menu item.
ADDITIONAL INFORMATION: It has the letter **A** underlined.
 6. Click **OK** to close the Add Tool Menu Item dialog box.
ADDITIONAL INFORMATION: To activate the hotkey, hold the **Alt** key while pressing **T** and then **A**.
-

Modifying a Tools Menu Item

TASK

1. From the **Options** menu, select **Customize Tools**.
STEP RESULT: The Add Tool Menu Item dialog box appears.
 2. Under **Menu Contents**, select the item that you want to modify.
 3. Click **Modify**.
STEP RESULT: The Configure Tools Menu Item dialog box appears.
 4. Make the necessary changes.
 5. Click **OK**.
 6. Click **OK** to close the Add Tool Menu Item dialog box.
-

Moving a Tools Menu Item

TASK

1. From the **Options** menu, select **Customize Tools**.
STEP RESULT: The Add Tool Menu Item dialog box appears.
 2. From the **Menu Contents** list, select the item that you want to move.
 3. Click **Up** or **Down** to move the item to where you want in the list.
 4. Click **OK** to close the Add Tool Menu Item dialog box.
-

Deleting a Tools Menu Item

TASK

1. From the **Options** menu, select **Customize Tools**.
STEP RESULT: The Add Tool Menu Item dialog box appears.

2. From the **Menu Contents** list, select the item that you want to delete.
 3. Click **Delete**.
STEP RESULT: The item disappears from the list.
 4. Click **OK** to close the Add Tool Menu Item dialog box.
-

Customizing Toolbars

You have the option to hide or display toolbars and to changed the size of the toolbar buttons.

TASK

1. From the **Options** menu, select **Customize Toolbars**.
STEP RESULT: The Toolbar Settings dialog box appears.
 2. Select the check box beside the toolbars that you want to display or deselect the check box beside the toolbars that you want to hide.
 3. Select the **Toolbar Button Size** that you want the buttons to appear on the toolbars.
 4. Click **OK**.
-

RESULT: The selected toolbars are changed.

Configuring the Mail Utility

At the time of installation, AutoVue integrates with MAPI compliant mail utilities that are installed on your computer, such as Microsoft Outlook. Electronic mail can then be accessed from within AutoVue, with the option to send a message only, send a message with links to Markup files or send a message with copies of Markup files.

The graphical interface and mailing options that are available to you are determined by the MAPI compliant mail utility you are using. AutoVue automatically attaches the current active file to messages when the **Mail** option is activated. However, Markup files associated with the original file are left for you to select and attached to your mail message.

To configure the mail utility:

TASK

1. From the **Options**, select **Configure Email**.
STEP RESULT: The Mail Options dialog box appears.
 2. From the **Mail Using** list, select the type of MAPI compliant mail utility you are using.
 3. Select the **Mailing Preferences** that you want.
 4. Click **OK** to close the Mail Options dialog box.
-

Thumbnails

Thumbnails allow you to build a visual representation of the entire contents of a directory in a thumbnails folder. Miniature visuals of the thumbnails folder file contents is created to help you preview files.

Also available is a generic customized icon that is particularly helpful for identifying files that are more readily acknowledged by their file type. For example, a word-processing document located in a directory where the majority of the files are graphic formats. As files are modified, their thumbnails can be updated quickly and easily. AutoVue stores thumbnails information in the **avwin/folders** directory.

Creating a Thumbnail Folder

If a thumbnails folder already exists for the current directory, the **Thumbnails** window will appear displaying the thumbnails.

TASK

1. From the **File** menu, select **Thumbnails**.

STEP RESULT: The New Folder dialog box appears.

2. Click **Create**.

ADDITIONAL INFORMATION: This may take a few minutes depending on the size and number of files in the folder. If the thumbnails folder is taking to long to create, you can skip it by clicking **Skip** in the New Folder dialog box. To resume creating the folder, from **Thumbnail** menu, select **Update All**.

STEP RESULT: AutoVue creates the thumbnails folder for the current directory and the **Thumbnails** window appears displaying the thumbnails for that folder.

3. To close the **Thumbnails** window, from the **Thumbnail** menu, select **Exit Thumbnails**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Exit Thumbnails**



Customizing Thumbnails

You can customize thumbnails to display according to your preferences.

TASK

1. From the **File** menu, select **Thumbnails**.

STEP RESULT: The **Thumbnails** window appears.

2. From the **Options** menu, select **Configure Thumbnails**.

STEP RESULT: The Configure Thumbnails dialog box appears.

3. Select the **Size** in pixels of the thumbnails.

ADDITIONAL INFORMATION: You can customize the size of the thumbnails by clicking **Custom** and entering a value.

4. Enter the **Space** between thumbnails.

ADDITIONAL INFORMATION: The space between thumbnails must be between 2 and 20.

5. Select the **File Details** that you want to display in the thumbnail.

- Select **Full** if you want to display the filename, size, date and time of creation.
- Select **Partial** if you want to only display the filename.

6. Click **OK**.

7. To close the **Thumbnails** window, from the **Thumbnail** menu, select **Exit Thumbnails**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Exit Thumbnails**



Viewing a Thumbnail Folder


You can display thumbnail folders from **View** or **Markup** mode. If a thumbnails folder does not exist for the current directory, the New Folder dialog box appears with the option to create a new folder.

TASK

1. From the **File** menu, select **Thumbnails**.

STEP RESULT: The **Thumbnails** window appears.

2. From the **Thumbnail** menu, select **Browse for Folder**.


ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Browse for Folder** .

STEP RESULT: The Browse for Folder dialog box appears.

3. Browse to locate the thumbnails folder that you want to view.
4. Click **OK**.

STEP RESULT: The thumbnails for the selected folder appear in the **Thumbnails** window.

5. To close the **Thumbnails** window, from the **Thumbnail** menu, select **Exit Thumbnails**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Exit Thumbnails** .


Sorting a Thumbnail Folder

TASK

1. From the **File** menu, select **Thumbnails**.

STEP RESULT: The **Thumbnails** window appears.

2. From the **Thumbnail** menu, select **Browse for Folder**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Browse for Folder** .

STEP RESULT: The Browse for Folder dialog box appears.


3. Browse to locate the thumbnails folder that you want to sort.
4. Click **OK**.

STEP RESULT: The thumbnails for the selected folder appear in the **Thumbnails** window.

5. From the **Thumbnail** menu, select **Sort** and the option that you want to sort by.

ADDITIONAL INFORMATION: A check mark beside the sort option indicates the current sort order.

6. To sort the thumbnails in ascending or descending order, from the **Thumbnail** menu, select **Sort Order**, and then select **Ascending** or **Descending**.
7. To close the **Thumbnails** window, from the **Thumbnail** menu, select **Exit Thumbnails**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Exit Thumbnails** .

Viewing a Thumbnail


Files are instantly accessible from the Thumbnails window.

TASK

1. From the **File** menu, select **Thumbnails**.

ADDITIONAL INFORMATION: The **Thumbnails** window appears.

2. From the **Thumbnail** menu, select **Browse for Folder**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Browse for Folder** .

STEP RESULT: The Browse for Folder dialog box appears.

3. Browse to locate the thumbnails folder that you want to view.
4. Click **OK**.

STEP RESULT: The thumbnails for the selected folder appear in the **Thumbnails** window.

5. Double-click the thumbnail that you want to view.
-

RESULT: The thumbnail is launched in the AutoVue workspace.

Updating Thumbnails


You can update all the thumbnails in the current folder or update one or more thumbnails in the current folder.

TASK

1. From the **File** menu, select **Thumbnails**.

ADDITIONAL INFORMATION: The Thumbnails window appears.



2. From the **Thumbnail** menu, select **Browse for Folder**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Browse for Folder** .

STEP RESULT: The Browse for Folder dialog box appears.


3. Browse to locate the thumbnails folder that you want to view.
4. Click **OK**.

STEP RESULT: The thumbnails for the selected folder appear in the Thumbnails window.


5. Select the thumbnail or thumbnails that you want update.
ADDITIONAL INFORMATION: To select more than one thumbnail, press the **Shift** or **Control** key while selecting.
6. From the **Thumbnail** menu, select **Updated Selected**.
ADDITIONAL INFORMATION: To update all thumbnails, from the **Thumbnail** menu, select **Update All**, or from the Thumbnail toolbar, click **Update Thumbnail Folder** .
STEP RESULT: The selected thumbnails are updated in the current folder.
7. To close the **Thumbnails** window, from the **Thumbnail** menu, select **Exit Thumbnails**.
ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Exit Thumbnails** .

Converting Thumbnails

TASK

1. From the **File** menu, select **Thumbnails**.
ADDITIONAL INFORMATION: The **Thumbnails** window appears.
2. From the **Thumbnail** menu, select **Browse for Folder**.
ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Browse for Folder** .
STEP RESULT: The Browse for Folder dialog box appears.
3. Scroll to locate the thumbnails folder that you want to convert.
4. Click **OK**.
STEP RESULT: The thumbnails for the selected folder appear in the **Thumbnails** window.
5. Select the thumbnail or thumbnails that you want to convert.
ADDITIONAL INFORMATION: To select more than one thumbnail, press the **Shift** or **Control** key while selecting.
6. From the **Thumbnail** menu, select **Convert**.
STEP RESULT: The Batch Convert dialog box appears.
7. Customize the conversion options.
8. Click **OK**.

9. To close the **Thumbnails** window, from the **Thumbnail** menu, select **Exit Thumbnails**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Exit Thumbnails Mode** .


Printing Thumbnails

TASK

1. From the **File** menu, select **Thumbnails**.

ADDITIONAL INFORMATION: The **Thumbnails** window appears.

2. From the **Thumbnail** menu, select **Browse for Folder**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Browse for Folder** .

STEP RESULT: The Browse for Folder dialog box appears.

3. Scroll to locate the thumbnails folder that you want to print.
4. Click **OK**.

STEP RESULT: The thumbnails for the selected folder appear in the **Thumbnails** window.

5. Select the thumbnail or thumbnails that you want to print.


ADDITIONAL INFORMATION: To select more than one thumbnail, press the **Shift** or **Control** key while selecting.

6. From the Thumbnail toolbar, click **Print** .

STEP RESULT: The Print dialog box appears.

7. Click **OK**.

8. To close the **Thumbnails** window, from the **Thumbnail** menu, select **Exit Thumbnails**.

ADDITIONAL INFORMATION: From the Thumbnail toolbar, you can also click **Exit Thumbnails Mode** .

Full Text Extraction

The Full Text Extraction application provides a facility for obtaining all the textual information from a selected file. This may be used by a search engine to provide document indexing services. It eliminates duplicates and enables the search for all type of documents, including CAD.

NOTE: Auto Text Extraction is not available for raster files.

Using the Full Text Extraction Utility

TASK

1. From the **Tools** menu, select **Auto Text Extraction**.

ADDITIONAL INFORMATION: You can also open the Auto Text Extraction dialog box by opening the file **Outtext.exe** located in the directory **C:\Program Files\av\avwin**.

STEP RESULT: The Automatic Text Extraction dialog box appears.

2. In the **From** field, enter the path and filename of the file from which the text is to be extracted.
3. In the **To** field, enter the path and filename or click **Browse** to locate the file to which the text is to be copied.

ADDITIONAL INFORMATION: File extensions include **.txt** and **.out**. The default is **text.out** located in the **temp** directory. If the file does not exist, the file will be created for you.

4. Select **Display Output Contents** if you want to preview the extracted text under **Contents**.
5. Click **Extract**.

STEP RESULT: The path and filename, and the text appears in the **Output Contents**.

6. Click **Close** to close the Automatic Text Extraction dialog box.
-

CAD Information Extraction

This utility allows users to extract XRef information from a CAD file. This is useful for batch importing AutoCAD, MicroStation and various other types of CAD files into a document management system.

TASK

1. From the **Tools** menu, select **Auto Text Extraction**.

STEP RESULT: The CAD/Doc Text Extraction dialog box appears.

2. In the **From** field, enter the path and file name of the file from which the text is to be extracted.

3. In the **To** field, enter the path and file name or click **Browse** to locate the file to which the text is to be copied.

ADDITIONAL INFORMATION: File extensions include **.txt** and **.out**. The default is **text.out** located in the **temp** directory. If the file does not exist, the file will be created for you.

4. Select **Display Output Contents** if you want to preview the extracted text under **Contents**.

5. Click **Extract**.

STEP RESULT: The path and filename, and the text appears in the **Output Contents**.

6. Click **Close** to close the CAD/Doc Text Extraction dialog box.

ADDITIONAL INFORMATION: Examples of integrations can be found in the **Integrat** directory of the media pack.

AutoVue Mobile

AutoVue Mobile provides the ability to view and add markups to files in a disconnected environment. Whether you are travelling or need to share files with an external partner, you can still view files, markups, and add new markups. Additionally, when connected, you can update your backend DMS/PLM/ERP/UCM system with edits you make offline.

Using the AutoVue Mobile feature in AutoVue, you can create a Mobile Pack (a "packaged" file that contains the base file, all the external resources—fonts, XRefs—needed to fully display the file, and existing markups for the file). The streaming files and renditions can also be included when creating the Mobile Pack.

Once a Mobile Pack is created, you can view your drawing and markups by viewing the Mobile Pack. You can create new markups, consolidate them with existing markups into one markup, or create a master Markup file that loads up automatically when you view the Mobile Pack.

In a disconnected environment, depending on the markup policy defined during Mobile Pack creation, you can add markups using AutoVue Desktop Version or the Desktop deployment of AutoVue. Markup policies define whether or not users can create/save new or existing markups, modify/delete (DMS only) existing markups, or automatically upload markups.

When you have access to your backend system, from AutoVue, you can update all the markups into the backend system. These markups are imported into the system and are associated with the base file from which you originally created the Mobile Pack.

The following sections provide details on creating a Mobile Pack, creating markups in AutoVue Mobile, understanding markup policies, and updating Markup files from Mobile Pack.

Creating a Mobile Pack

The *Mobile Pack* includes the native file or streaming file, metadata, Intellistamp designs, all reference files (XRefs), and associated markups. Optionally, you can include renditions such as TIFF or PDF.

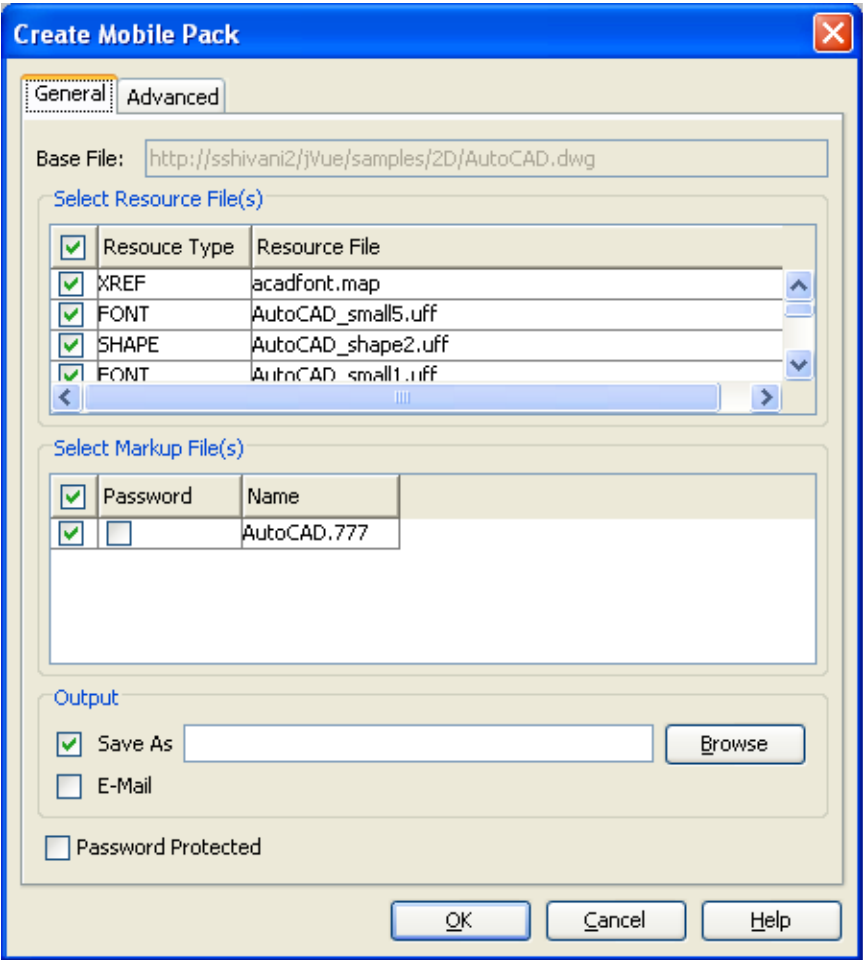
The following steps explain how to create a Mobile Pack for a file with attached Markup files:

TASK

1. Open the base file.
2. From the AutoVue menu bar, select **Collaborate**, and then select **Create Mobile Pack**.

STEP RESULT: The Create Mobile Pack dialog appears. It displays the General and Advanced tabs, which contain the basic and advanced options for creating a Mobile Pack, respectively.

3. Click the **General** tab.



ADDITIONAL INFORMATION: The following table lists the options available in the General tab:

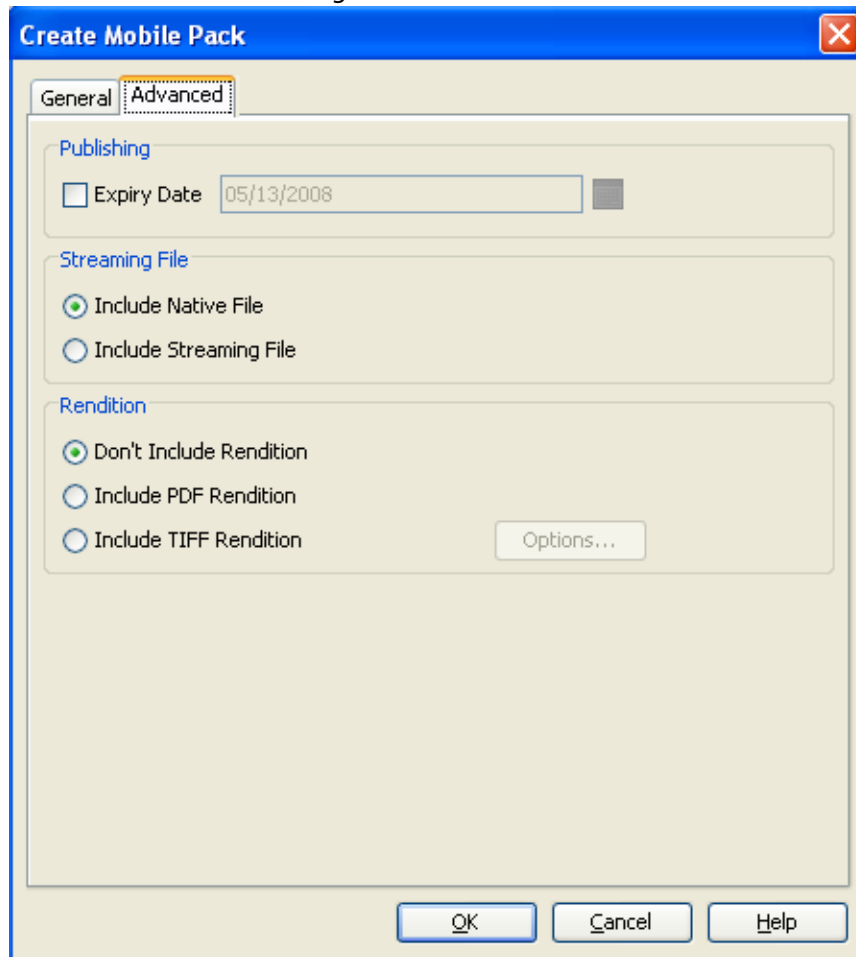
Option	Description
Base File	Displays the file path of the base file.

Option	Description
Select Resource File(s)	Lists all the fonts, XRefs, and other external resources that are used to fully display the base file. As a default, all resource files are selected.
Select Markup File(s)	Lists all Markup files associated with the base file. You can password protect single or multiple Markup files.
Output	You can select the output location for the Mobile Pack, and also send the file using your default e-mail client.
Password Protected	You can password protect the Mobile Pack.

4. From the **Select Resource File(s)** box, select the resource files to include in the Mobile Pack.
5. From the **Select Markup File(s)** box, select the Markup files to include in the Mobile Pack.
6. If you wish to password protect the Markup files, select the **Password** check box next to the Markup file.
 - a In the **Password** field, enter a password of any length and character.
 - b In the **Confirm Password** field, re-enter the password.
 - c Click **OK** to close the Password dialog.
7. To save the Mobile Pack locally, from the Output options, specify the location and name:
 - a Click **Browse** to locate the directory where you want to save, or in the **Save As** field, enter the file path.
 - b Specify a name for the Mobile Pack.
8. If you wish to send the Mobile Pack using your default e-mail client, select the **E-Mail** check box.
9. If you wish to password protect the Mobile Pack, select the **Password Protected** check box.
 - a In the **Password** field, enter a password of any length and character.
 - b In the **Confirm Password** field, re-enter the password.
 - c Click **OK** to close the Password dialog.
10. To select Advanced options such as including streaming file and setting an expiry date on the Mobile Pack, continue to the next step.
11. If you do not wish to modify advanced options, click **Finish** to complete Mobile Pack creation.

12. Click the **Advanced** tab.

ADDITIONAL INFORMATION: Depending on the Company Wide Policy settings, the Advanced tab options might be disabled. For more information on Company Wide Policy, refer to the *Installation and Configuration Guide*.



ADDITIONAL INFORMATION: The following table lists the options available in the Advanced tab:

Option	Description
Publishing	Lets you specify an expiry date for the Mobile Pack. After the expiry date, the content of the Mobile Pack will not be accessible.
Streaming File	Lets you choose to include the native file or the streaming file in the Mobile Pack. There are certain limitations when loading streaming files: Streaming files do not support blocks and views options. Streaming file requirements change with each release of AutoVue. As a result, a streaming file from a previous version of AutoVue cannot be loaded in the latest release. For 3D files, including the streaming file reduces the file size of the Mobile Pack. Also, since native files are not released, including the streaming file increases security. See Also "Streaming Files"

Option	Description
Rendition	Lets you choose to include a PDF or TIFF rendition in the Mobile Pack.

13. If you wish to set an expiry date for the Mobile Pack, from the Publishing option, select the **Expiry Date** check box and then select a date. The Mobile Pack cannot be opened after this expiry date.
14. If you wish to include a streaming file, from the **Streaming File** option, select whether to include a native file or streaming file in the Mobile Pack.
15. If you wish, from the **Rendition** option, you can include a PDF or TIFF rendition in the Mobile Pack.
16. Click **OK** to complete Mobile Pack creation.

RESULT: If you selected the **E-Mail** check box, your default e-mail client opens with the Mobile Pack (*.avp) as an attachment. Otherwise, the Mobile Pack is output to the specified location.

Viewing Mobile Packs

Mobile Packs are opened in the same way as ordinary files. Users have access to all the external reference files (XRefs), resource files, and markups bundled with the base file. Additionally, users have the option to modify existing markups or create new ones to be associated with the Mobile Pack.

You can open a Mobile Pack as you would open any other file in AutoVue:

TASK

1. From the AutoVue menu bar, select **File**, and then select **Open**.
STEP RESULT: The Open dialog appears.
2. Browse to the location of the Mobile Pack (*.avp), and then select **Open**.

3. If the Mobile Pack is password-protected during creation, the Password dialog appears. In the **Password** field, enter the password and then click **OK**.

ADDITIONAL INFORMATION: The native file or streaming file opens in AutoVue with all associated XRefs, resource files, markups, and renditions.



There are certain limitations when loading a streaming file:


- Streaming files do not support blocks or viewing options.
- Streaming file constraints change with each version of AutoVue. As a result, the streaming file from an earlier version of AutoVue cannot be opened in a later version.
- Streaming files are platform-specific. Therefore, a streaming file created on a Windows platform cannot work on a UNIX platform, and vice versa.

4. If the Rendition option is selected during the creation of the Mobile Pack, from the AutoVue menu bar, select **Collaborate**, and then select **View Rendition from Mobile Pack**.

ADDITIONAL INFORMATION: It is not possible to add markups to a rendition.

STEP RESULT: The rendition appears in a new AutoVue window.

5. If there are associated Markup files, the **Markup Indicator** icon  appears in the status bar. To open the Markup files, click the **Markup Indicator** icon .

ADDITIONAL INFORMATION: From the menu bar, you can also click **Open Markup(s)** .

STEP RESULT: The Choose a New or an Existing Markup dialog appears.

6. If you wish to create a new Markup file to include in the Mobile Pack, click **Create a new Markup file**.
 7. If you wish to open Markup files included in the Mobile Pack, click **Choose an existing Markup file**, and then select the Markup files.
 8. If the Markup file is password-protected during creation, The Password dialog appears. In the **Password** field, enter the password and then click **OK**.
 9. If you wish to import a Markup file from your local machine, click **Import**.
STEP RESULT: The File Open dialog appears.
 10. Select the Markup file and click **Open**.
-


Creating Markup Files

When marking up a Mobile Pack, you can create new markups or consolidate existing markups to a new markup. You cannot modify markups that are bundled during Mobile Pack creation.

When working with a Mobile Pack in the Desktop deployment of AutoVue, you may be restricted from saving new markups, editing markups, deleting markups, opening markups, and filtering as a result of the markup policy. The markup policy, which is defined in Client/Server deployment of AutoVue, includes a set of rules to determine certain restrictions and privileges for users of the Mobile Pack.

To save a new Markup file with a Mobile Pack, do the following:

TASK

1. Create markups.
 2. From the AutoVue toolbar, click **Save Markup** .
STEP RESULT: The Save Markup dialog appears.
 3. Enter a name for the Markup file in the **Name** field.
 4. If you wish to password protect the Markup file, select the **Password Protected** check box.
 5. If you wish to save the Markup file to a location on a local computer, click **Export**.
 6. Click **OK**.
 7. If you wish to send the updated Mobile Pack with your default e-mail client, from the AutoVue menu bar, select **Collaborate**, and then select **Reply to Mobile Pack**.
-

RESULT: Your default e-mail client opens with the Mobile Pack (*.avp) as an attachment.

Updating from Mobile Pack

When you are finished working with the Mobile Pack, you can update your changes to the backend DMS/ERP/PLM/UCM system. You can synchronize all of the markups created in the Mobile Pack to your backend system. These markups will be associated with the file in the backend system from which you originally created the Mobile Pack.

NOTE: This feature is only available in Client/Server deployment of AutoVue.

Appendix A: EDA Terms and Definitions

The following table presents commonly used EDA terms and their corresponding definitions.

EDA Term	Definition
Annular ring	A circular strip of conductive material that remains after a hole has been drilled through the pad of a printed circuit board.
Anti-copper	An area within a fill zone in which copper cannot be placed.
Aperture	An opening, similar to the aperture of a camera, that is used for photo-plotting. Apertures are available in various sizes and shapes.
Aperture list	A text file containing the dimensions for each of the apertures used to photo-plot PCB artwork.
Clusters	Components that are grouped according to their interrelationships and placed in close proximity on the board. This keeps the connections on the PCB short so that the board is easier to work with.
Component	An element or a part of a PCB.
Component density	The quantity of components on a unit area of a PCB.
Component hole	A hole in the printed circuit board that corresponds to a pin or wire of a component. This hole serves the dual function of attaching the component to the board and establishing the electrical connection between the pin or wire and the remainder of the board circuitry.
Component library	A computer data file that contains the footprint patterns for a number of components.

EDA Term	Definition
Component side	The uppermost or top layer of a board on which most components are placed.
Component silkscreen	The silkscreen markings of the printed circuit board that appear on the component side. The silkscreen is applied over the solder mask.
Component solder mask	The colored, usually translucent, coating applied to the board over the etched copper. It protects the selected areas from the soldering process.
Connection	An unrouted, partially routed, or completely routed path between two pads. In a net with n pads, there are exactly $n-1$ connections.
Copper pour	A method by which a copper zone is filled with a specified pattern, with objects that cross the zone or lie within the zone being avoided.
Copper zone	An area on a board designed to be covered by a layer of copper when manufactured. Also known as a "metal zone".
Cross hatching	The breaking up of large conductive areas by the use of a pattern of lines and spaces in the conductive material.
Datum	A specific location (a point) that serves as a reference to locate a PCB pattern or layer for manufacture.
Density	On a PCB, the degree to which components are packed on the board. Generally, the density is given as the number of square inches per equivalent, i.e., a lower number indicates a more dense board.
Discrete components	Components with three or fewer electrical connections (for example, resistors or capacitors).
Electrical check	The process of checking the PCB to ensure that the connections they are on match those specified in the net list.
Fill zone	A zone that defines an area to be filled with copper.
Fine pitch	A class of surface-mount components that is characterized by pins measuring 0.025 inches or less from pad center to pad center.

EDA Term	Definition
Footprint	The physical description of a component. It consists of three elements: padstacks, representing the pads of the component; obstacles, representing among other things, the physical outline of the component, silkscreens, keepouts/keepins, and assembly drawing data; and text documenting the footprint information (for example, the component name). You may want each project to have its own footprint library containing all the footprints used in that project.
Ground plane	A large area on the PCB, usually an entire layer, that provides a common ground connection for all component ground pins and other ground connections.
Heatsink	A mechanical device made of a high thermal conductivity material that dissipates heat generated by a component or assembly.
Heuristics	A method of routing that consists of repeated attempts to apply very simple routing patterns to unrouted connections in order to complete the routing quickly and cleanly. Typically, heuristics are used for memory and short point-to-point routing.
Hole	The area where board material must be removed by drilling or milling.
Isolation	The clearance around a pad, track, zone, or via that defines the nearest approach allowed by conductors of another signal set.
Jumper wire	A discrete electrical component or wire used to make electrical connections between points that have no copper etch due to board density or some other factor.
Keepout	An area fill within which no routing is allowed.
Land	The copper pad needed for a surface mount pin.
Layer	The copper pad needed for a surface mount pin.
Manual routing	Individual connections, in the form of traces, vertices, arcs, and so on, which are entered manually into the PCB design.
Mounting hole	A hole used for the mechanical support of a PCB or for the mechanical attachment of components to a PCB.

EDA Term	Definition
Multi-layer board	A PCB that has multiple layers, separated by dielectric material, with connectivity between layers established by vias or through-holes. This term usually refers to a board with more than two layers.
Net	A logical construct (circuit) that originates in a schematic and is transferred to a board to describe required electrical connections. The connections may be completed by using vias, tracks, or zones.
Net list	List of names of symbols or parts and their connection points which are logically connected in each net of a circuit. A net list can be extracted electronically on a computer from a properly prepared schematic.
Obstacle	An outline representing an object on the board. It must be taken into account during routing, placement, or copper pour.
Pad	On a PCB, a copper shape on one or more layers (there may be a hole and an isolation surrounding the copper) used for connecting a component pin to the PCB. The pad indicates where pins of a component are placed.
Padstack	A numbered list of pad descriptions. Each description contains a pad definition, including layer, style, drill diameter, size, offset, and solder mask guard width.
PCB - Printed Circuit Board	A PCB is a board made up of components affixed to a common surface and connected by copper tracks.
Pin	The portion of a component to which an electrical connection can be made.
Polarity	Positive polarity refers to visible items. Negative polarity refers to items that become transparent to reveal underlying layers or background color.
Ratsnest	A number of unrouted straight-line connections between two or more pads that represent the electrical connections in the netlist. The ratsnest serves as a reminder that the pads must be connected, and that, currently, there is no track on the board to make that connection.

EDA Term	Definition
Reference designator	A character string denoting the type of component and a number that is specific to that component.
Routing	Placing conductive interconnects between components on a PCB layout. The process of turning nets into tracks.
Schematic	A graphical description of an electrical circuit.
Segment	The partial track that exists between two adjacent vertices or between a vertex and a pin. Sometimes the track between two pins is also called a segment, although connection is usually the more appropriate term here.
Signal	An electrical impulse of a predetermined voltage, current, polarity, and pulse width.
Silkscreen	Text or outlines (in ink) on the solder mask, on the top, and sometimes on the bottom of board modules. A silkscreen is used for component and identification placement on a PCB and usually includes component outlines, reference designators, polarity indicators, pin one markings, part numbers, the company name, and copyright info.
SMT - Surface Mount Technology	PCB technology whereby the leads on the chips and components are soldered onto the surface of the board rather than inserted into it. The use of SMT results in smaller and faster printed circuit boards.
Solder mask	A negative plot of pads with a guard band around the pads. Also, a lacquer applied to prevent solder from adhering to unwanted areas on the PCB.
Solder paste	A pattern that serves as a template for solder paste application when the board is manufactured.
Solder side	The PCB surface opposite the one on which most components are mounted (component side). Also, the bottom layer of the board.
Test point	A special point of access to an electrical circuit that is used for electrical testing purposes.
Through-hole via	A via that connects the surface layers on a PCB.

EDA Term	Definition
Trace (Track)	The copper trails (electrical connection between two or more points) on the PCB and the onscreen representation of that copper.
Venting patterns	Patterns etched in the board that allow gases formed during fabrication to escape.
Vertex	A logical point at which a track is ended and restarted. A vertex is located at each change of direction on the track.
Via (feed-through hole)	A hole connecting layers of a PCB. A through-hole via connects the surface layers of a board. On multilayer boards, a via not reaching a surface layer on one side is called a blind via, and a via not reaching a surface layer on either side, thus being externally invisible, is called a buried via.
Viastack	A numbered list of via descriptions. Each description contains a via definition, including layer, style, drill diameter, size, offset, and solder mask guard width.
Via string	The copper etch that exists between a SMT pad and a corresponding fanout via.
Zero-length connection	An unrouted connection between layers where the end points in the connection have the same X- and Y- coordinates.
Zone	An area on a PCB layer designated as copper or anti-copper. Copper zones may have net names, while anti-copper zones may not.

Appendix B: Gerber Layer File Format

The Gerber Layer File (GBL) format allows you to open multiple Gerber files at once in AutoVue. This file format also allows you to analyze multiple Gerber files for net connectivity by creating net entities in AutoVue to connect the different traces and pins.

A GBL is divided into four main sections:

- Header section
- INI configuration section
- Aperture file section (optional)
- Gerber layers section

Header

The header is used to recognize a file as a Gerber Layer File (GBL). All GBLs must begin with the following line:

```
GBL FILE (CSI)
```

If a file does not begin with this line, it will not be treated as a GBL.

INI Configuration

The INI configuration section allows you to configure the following INI options that are used when processing each Gerber layer and Gerber Layer File.

- APERTURE_FORMAT_FILEPATH

- INCREMENTALMODE
- NUMDECIMALS
- NUMDIGITS
- TRAILINGZEROS
- TOOLFILEPATH
- TOOLFILETYPE
- TOOL_UNIT
- UNITS

These INI options are the same as the Gerber-specific options. For more information, refer to the “Gerber Formats” section of the *Oracle AutoVue Installation and Administration Manual*.

Aperture File

The aperture file section is used to specify the aperture file required by Gerber files. If you do not provide an aperture file in the Gerber Layer File, then the aperture file specified in AutoVue’s INI file is used.

The aperture file section is a single line in the GBL and follows this structure:

```
APP <aperture_file_path> <aperture_file_type>
```

Where:

- APP is a keyword to indicate that the current line refers to aperture information.
- <aperture_file_path> is the absolute or relative file system path of the aperture file. This field is mandatory.
- <aperture_file_type> is an optional field that indicates the format used to store the aperture information. The following table provides a list of possible values.

File Type	Typical File Extension	Description
0	.too	AutoVue default aperture file type.
1	.app	OrCAD Layout file aperture file type.

File Type	Typical File Extension	Description
2	.gap	ECAM aperture file type.
3	.rep	Protel/Altium Designer aperture file type.
4	.apt	Artwork aperture file type.
5	.txt	Allegro aperture file type.

For example:

```
APP aperture.txt 5
```

This example specifies an Allegro-type aperture file, `aperture.txt`, in the same folder as the GBL.

Gerber Layers

The Gerber layers section is used to list the Gerber files to be opened as layers in the Gerber Layer File and includes two fields: `OPTIONALATTRIBUTES` and `LAYER`.

`OPTIONALATTRIBUTES` is an optional field used to list the names of the attributes that are provided for each Gerber layer. It is specified as a semicolon (;) separated list.

For example:

```
OPTIONALATTRIBUTES=<attribute1>;<attribute2>;...
```

There are currently seven supported attributes: `COLOR`, `LAYERNAME`, `LAYERTYPE`, `OFFSET`, `VISIBLE`, `POLARITY` and `UNITS`.

- **COLOR:** Used to provide the color of the layer. The color can be given either as a color index (refer to the following table) or as an RGB color.

The following table provides a list of possible color indices:

Index	Color
0	Black
1	Red
2	Yellow
3	Green
4	Cyan
5	Blue
6	Magenta
7	White
8	Dark grey

RGB colors are specified as follows:

RGB(<r_components>,<g_component>,<b_components>)

- **LAYERNAME:** Used to provide the name of the layer that appears in the logical layer dialog.
- **LAYERTYPE:** Used to provide the type of the layer. This is used to construct the net connectivity. There are three possible values:
 - signal
 - nonsignal
 - plane

Only the signal and plane layer types are considered when constructing the net connectivity. Any value other than these three is treated as a nonsignal.

- **OFFSET:** Used to provide the offset of the layer in native units.

Syntax: (x,y)

For example: (10.0, 20.0)

- **VISIBLE:** Used to provide the initial visibility of a layer. To set a layer as invisible, the value of this attribute should be set to **false** (case-sensitive). Any other values are treated as **true**.
- **POLARITY:** Used to provide the initial polarity of a layer. To set a layer's polarity to negative, the value of this attribute must be set to **negative**. Note that the attribute value is case-sensitive.

- **UNITS:** Used to provide the units of a layer. Possible values are **in** for inches, **mm** for millimeters, and **mil** for microns.

NOTE: To include whitespaces, the argument should be enclosed between double quotes ("").

As the following example shows, the order of the attributes are not important and their names are not case sensitive (that is, COLOR is the same as color).

```
OPTIONALATTRIBUTES=COLOR;LAYERTYPE
```

```
OPTIONALATTRIBUTES=layerName;Color;layerTYPE
```

The LAYER field entry represents a Gerber file that makes up a layer in the GBL. It is specified as follows:

```
LAYER <optional_attributes> <Gerber_filename>
```

Where:

- *LAYER* is the keyword for this field.
- *<optional_attributes>* is a list of attributes in the order given by the OPTIONALATTRIBUTES field. All attributes given in the OPTIONALATTRIBUTES field must be provided for each LAYER entry in the same order.
- *<Gerber_filename>* is the absolute or relative file system path of the Gerber file.

Appendix C: AutoVue Product Variations – Feature Matrix

The following tables list the available features of the AutoVue Enterprise Visualization Desktop Version product family on the Windows platform. The tables are organized by the following features: general features, 2-D and EDA common features, 2-D features, EDA features, 3-D features, markup features, and AutoVue Mobile features.

Note that several of the following features are supported only if the file that is being viewed in AutoVue contains relevant information.

General Features					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Edit, Select, Copy & Clipboard	x	x	x	x	x
File Properties	x	x	x	x	x
Page Navigation	x	x	x	x	x
Thumbnails	x	x	x	x	x
Toolbar Customization	x	x	x	x	x
Folder Navigation					
Browse	x	x	x	x	x
Browse Next File	x	x	x	x	x
Browse Previous File	x	x	x	x	x
Print					
Batch Print	x	x	x	x	x
Print	x	x	x	x	x
Print Preview	x	x	x	x	x
Conversion					
Basic Conversion	x	x	x	x	x
Conversion to 3-D			x	x	x
Conversion to PDF ¹	x	x	x	x	x
Convert a batch of files	x	x	x	x	x
Configuration ²	x	x	x	x	x

¹Not supported for 3-D designs.

²AutoVue provides different configuration options for different file types

2-D Common Features (Raster/Graphics, 2-D Vector, Desktop/Office, EDA)					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Overlays	x	x	x	x	x
Text Extraction ³	x	x	x	x	x
Text Search ³	x	x	x	x	x
Basic Manipulation					
Flip	x	x	x	x	x
Rotate	x	x	x	x	x
Zoom	x	x	x	x	x
Special View Modes					
Magnify Glass	x	x	x	x	x
Magnify Window	x	x	x	x	x
Pan	x	x	x	x	x
Pan and Zoom Window	x	x	x	x	x
Compare					
Compare Files	x	x	x	x	x

³Not supported for raster formats

Raster/Graphics Features					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Image Enhancements					
Anti-Alias	x	x	x	x	x
Contrast ⁴	x	x	x	x	x
Invert ⁴	x	x	x	x	x

⁴Applies to raster formats and to raster overlays in other formats

2-D (Vector Features)					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Measurements					
Entity Snapping		x	x	x	x
Angle	x	x	x	x	x
Arc	x	x	x	x	x
Area	x	x	x	x	x
Calibration	x	x	x	x	x
Distance	x	x	x	x	x
2-D CAD Features					
Blocks		x	x		x
CAD/Doc Text Extraction		x	x		x
Drawing Information		x	x		x
Layers		x	x		x
Views		x	x		x
XRefs		x	x		x
Conversion					
Change Pen Settings		x	x		x
Printing					
Create/Assign Pen Settings		x	x		x

EDA Features					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Bill of Material (BOM)				x	x
Blocks				x	x
Cross Probe				x	x
Design Hierarchy Navigation				x	x
Design Verification				x	x
EDA Layers				x	x
Entity Filters				x	x
Entity Properties				x	x
Entity Search				x	x
Entity Selection				x	x
Go to Net Instance				x	x
Layer Sets				x	x
EDA Measurement					
Entity Snapping				x	x
Angle				x	x
Arc				x	x
Area				x	x
Calibration				x	x
Distance				x	x
Minimum Distance				x	x
Compare					
Layer State Propagation				x	x

3-D Features					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Basic 3-D Manipulation					
3-D Render Modes			x	x ⁵	x
Change Part Color			x	x ⁵	x
Change Transparency			x	x ⁵	x
Free Rotate			x	x ⁵	x
Expand/Collapse Tree			x	x ⁵	x
Identical Part Detection			x	x ⁵	x
Light Settings			x	x ⁵	x
Manipulators			x	x ⁵	x
Re-center			x	x ⁵	x
Selection			x	x ⁵	x
Spin			x	x ⁵	x
Advanced 3-D Manipulation					
3-D Digital Mockup			x		x
3-D Search			x	x ⁵	x
3-D Entity Properties			x	x ⁵	x
Bill of Material (BOM)			x	x ⁵	x
Electro-Mechanical Digital Mockup					x
Explode			x	x ⁵	x
Interference Checks			x	x ⁵	x
Part Alignment			x	x ⁵	x
PMI Filtering			x		x
Section			x	x ⁵	x
Transformation			x	x ⁵	x
User-Defined Coordinate Systems			x	x ⁵	x
User-Defined View			x	x ⁵	x
3-D Measurements					
Entity Snapping			x	x ⁵	x
Angle			x	x ⁵	x

⁵The 3-D features in AutoVue EDA Professional are only available when viewing 3-D PCBs.

3-D Features					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Arc			x	x ⁵	x
Calibrate			x	x ⁵	x
Distance			x	x ⁵	x
Edge			x	x ⁵	x
Minimum Distance			x	x ⁵	x
Surface			x	x ⁵	x
Vertex Coordinates			x	x ⁵	x
Compare					
Compare 3-D Designs			x	x ⁵	x
Compare Sets			x	x ⁵	x

⁵The 3-D features in AutoVue EDA Professional are only available when viewing 3-D PCBs.

Markup Features					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Create and Save	x	x	x	x	x
Consolidate	x	x	x	x	x
Import/Export	x	x	x	x	x
Layers	x	x	x	x	x
Markup Filtering	x	x	x	x	x
Markup Information	x	x	x	x	x
Modify	x	x	x	x	x
Saved States	x	x	x	x	x
Markup Entities					
Attachment	x	x	x	x	x
Hyperlink	x	x	x	x	x
Intellistamp	x	x	x	x	x
Leader	x	x	x	x	x
Note	x	x	x	x	x
Signoff	x	x	x	x	x
Stamp	x	x	x	x	x
Stamp Library	x	x	x	x	x

Markup Features					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Text	x	x	x	x	x
2-D Markup Entities					
Arc	x	x	x	x	x
Box	x	x	x	x	x
Circle	x	x	x	x	x
Cloud	x	x	x	x	x
Freestyle	x	x	x	x	x
Highlight	x	x	x	x	x
Line	x	x	x	x	x
Polygon	x	x	x	x	x
Polyline	x	x	x	x	x
Markup Measurement Entities					
2-D Measurement	x	x	x	x	x
3-D Measurement			x	x ⁶	x
EDA Measurement				x	x

⁶The 3-D features in AutoVue EDA Professional are only available when viewing 3-D PCBs.

AutoVue Mobile					
Feature	AutoVue Office	AutoVue 2D Professional	AutoVue 3D Professional Advanced	AutoVue EDA Professional	AutoVue Electro-Mechanical Professional
Markup Mobile Pack	x	x	x	x	x
View Mobile Pack	x	x	x	x	x

Note: For the list of features for ActiveX, refer to ActiveXFeatures.pdf

Feedback

If at any time you have questions or concerns regarding AutoVue, call or e-mail us.

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