Oracle AutoVue, Desktop Deployment

User's Manual

ORACLE

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Contents

1 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.

Related Documents

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

- *Installation and Configuration Guide*
- *Viewing Configuration Guide*
- *Release Notes*
- *Supported Formats List*
- *Product Variations - Feature Matrix*

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.
italic	Italic type indicates book titles, emphasis, or placeholder variables for which you supply particular values.
`monospace`	Monospace type indicates commands within a paragraph, URLs, code in examples, text that appears on the screen, or text that you enter.

2 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 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* 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.

3 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.

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).

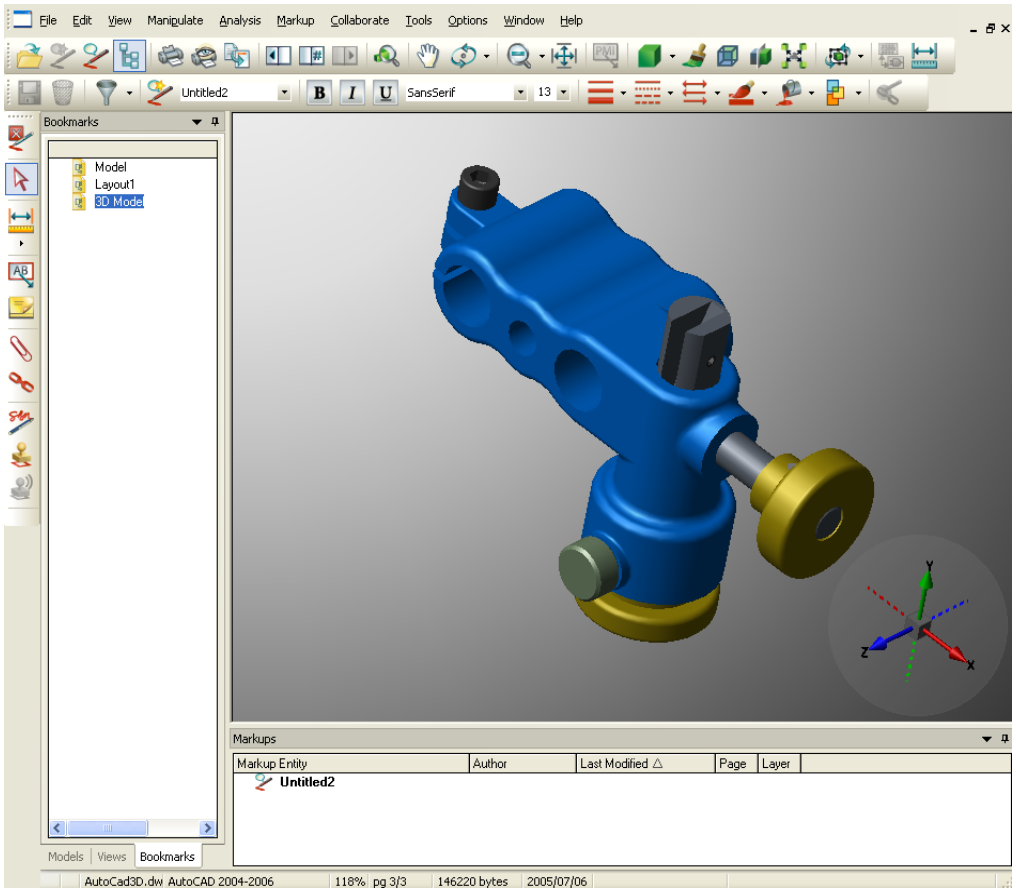
6. Click **Save**.

   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:



NOTE: *This is the default view of AutoVue. If you are using a customized AutoVue GUI file, the menu items and toolbars may vary.*

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.

Toolbars

AutoVue has three toolbars: *AutoVue toolbar, Markup Properties toolbar,* and *Markup Entity toolbar.*

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:

NOTE: *The AutoVue GUI can be customized by the system administrator. The GUI that is displayed is dependent upon whether it is customized and on the type of customization.*

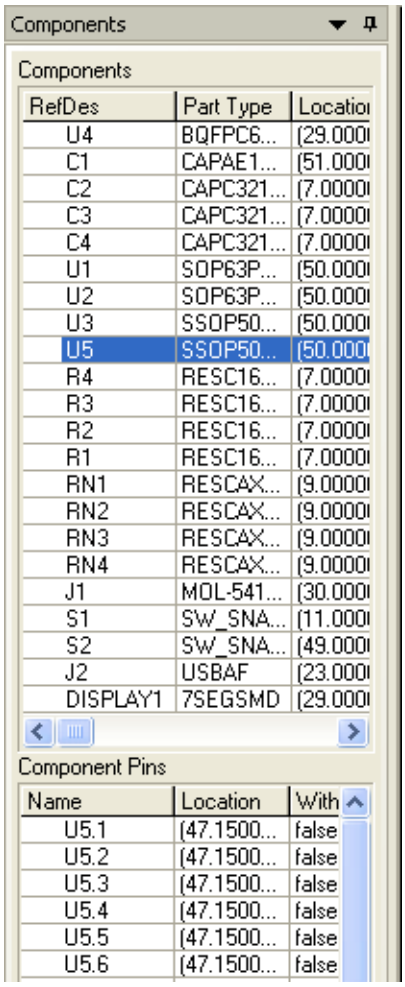
Navigation Panel

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*  .

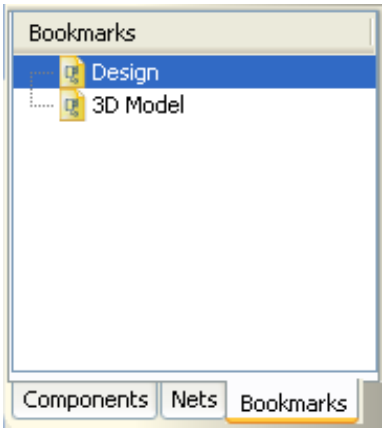
The columns displayed in the Navigation Panel are determined by the profile of the entity types in the current schematic drawing or PCB design. Lists can be sorted in order to group similar component instances.

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.



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.

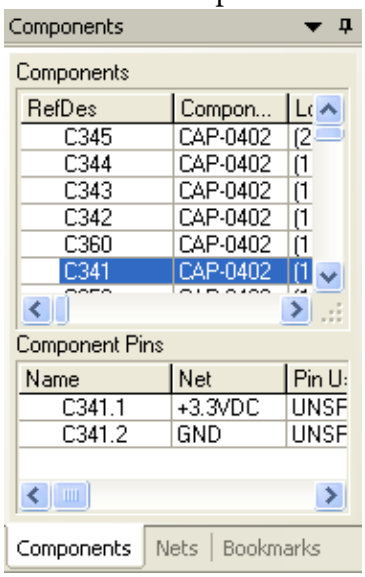
To go to a destination specified by a bookmark, click the bookmark text or the page icon located to the left of the bookmark text.

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.



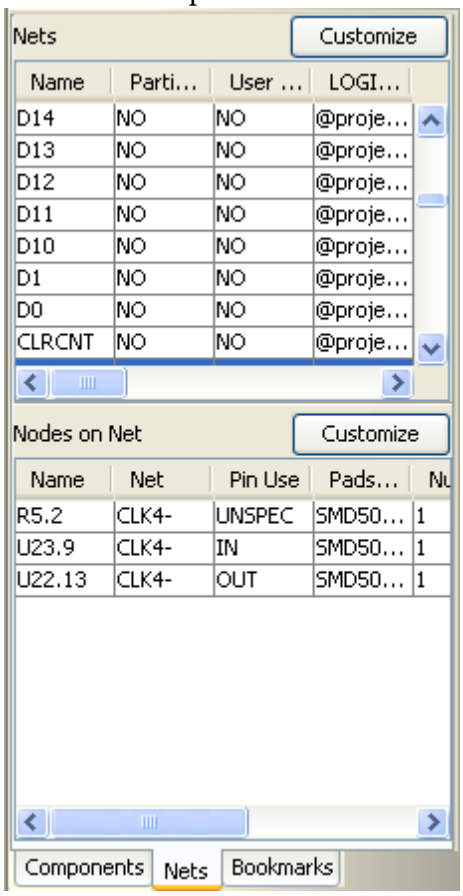
Nets Tab

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.

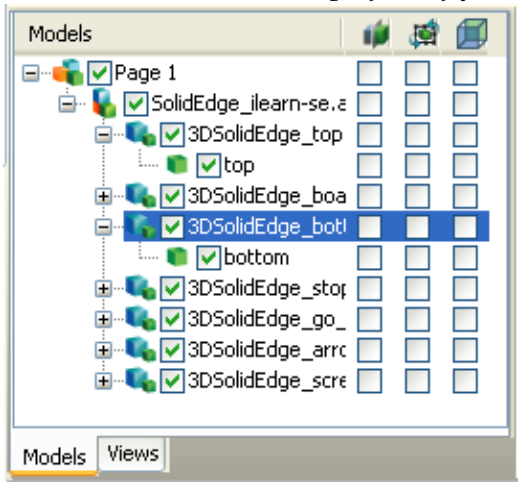


Models Tab and Model Tree

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.

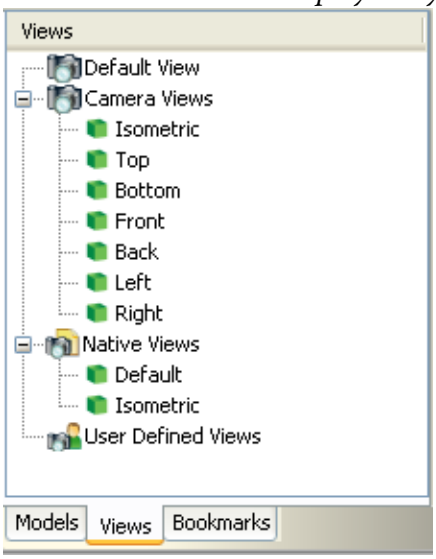
NOTE: The Models tab displays only for 3D drawings.



Views Tab

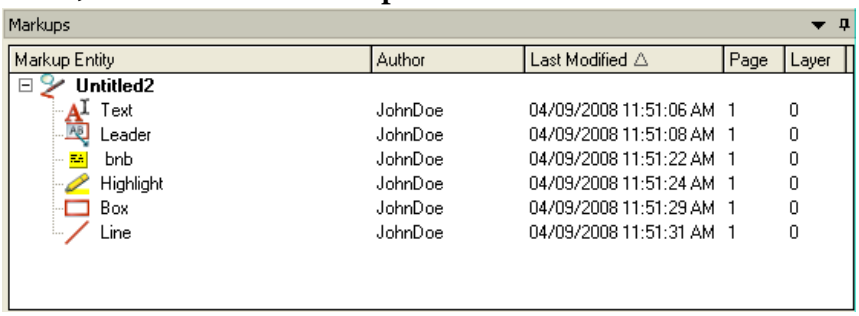
The *Views tab* lists the defaults, standard, native, and user-defined views. You can switch to a standard, native, or user-defined view, as well as add or delete user-defined views.

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.**



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.

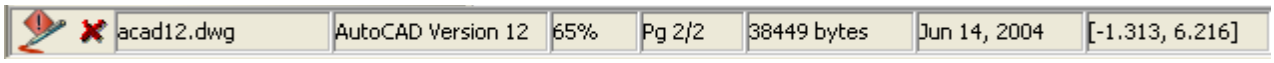
When a markup entity is created, it appears in the tree and the information is recorded and saved in the Markup file.

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 identity 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.

Opening a Local File

TASK

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

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

   STEP RESULT: The File Open dialog appears.

2. Browse to locate the file that you want to open.
3. Click **Open**.

RESULT:

The file opens in the AutoVue workspace.

Opening a URL

You can open a file by specifying the URL to the file.

TASK

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

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

   STEP RESULT: The File Open dialog appears.

2. From the left panel, click **Web URL**  .
3. In the **File name** field, enter the URL of the file to open.

   ADDITIONAL INFORMATION: For example, http://AutoVueServer/files/abc.dwg or ftp://ftpserver1.com/files/abc.doc.

4. Click **Open**.

RESULT:

The file opens in the AutoVue workspace.

Opening a File from Agile PLM

To open a file from Agile PLM, do the following:

TASK

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

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

   STEP RESULT: The File Open dialog appears.

2. From the left panel click **Agile**  .

   ADDITIONAL INFORMATION: Files located in Agile PLM are displayed in the dialog.

3. Select a file and click **Open**.

RESULT:

The file opens in the AutoVue workspace.

Opening a File from a Backend DMS System

When AutoVue is connected to a backend Document Management System (DMS[1]) system, you can open a file located in the system from the File Open dialog. Depending on the backend DMS system you are connected to, you must enter your username and password when opening files stored in the backend system.

Refer to your integration documentation for more information on logging onto your DMS system.

TASK

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

   STEP RESULT: The File Open dialog appears.

2. To open a file from the backend DMS system, from the left panel, click **DMS**[2]  .

   ADDITIONAL INFORMATION: If AutoVue is configured with one DMS system, the name of the DMS would appear instead of DMS. If AutoVue is connected to multiple backend DMS systems, a list of the DMS system names appear when you click **DMS**  . Select the DMS from which you want to open files.

   STEP RESULT: An authentication dialog appears.

1. A DMS/PLM/ERP/UCM system is referred to as DMS in this document.
2. Depending on the type of backend system AutoVue is connected to, the button name will change.

3. Enter the login information and then click **OK**.

   STEP RESULT: The backend DMS system files appear in the File Open dialog.

4. Select a file and then click **Open**.

RESULT:

The file opens in the AutoVue workspace.

Searching for Files in a Backend DMS System

When AutoVue is connected to backend DMS system, you can search for files in the system by entering specific search criteria in the File Open dialog.

Refer to your integration documentation for more information on logging onto your DMS system.

TASK

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

   STEP RESULT: The File Open dialog appears.

2. To search for a file in the backend DMS system, from the left panel, click **Search DMS**  .

   ADDITIONAL INFORMATION: If AutoVue is configured with a single DMS, the name of the DMS appears instead of DMS. If AutoVue is connected to multiple backend DMS systems, a list of the DMS system names appear when you click **Search DMS**  . Select the DMS in which you want to perform the search.

   STEP RESULT: An authentication dialog appears.

3. Enter the login information and then click **OK**.

   STEP RESULT: The search criteria options load in the File Open dialog.

4. Enter the search criteria and then click **Search**.

   ADDITIONAL INFORMATION: Alternately, to view all files in the backend DMS system, leave the search criteria fields empty and click **Search**.

   STEP RESULT: The file results appear in the dialog. For each AutoVue session, all search results are saved and can be referenced from the File Open dialog as Search Results 1, Search Results 2, and so on.

5. Select a file and then click **Open**.

RESULT:

The file opens in the AutoVue workspace.

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.

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

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

   STEP RESULT: The Properties dialog appears.

2. 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.

3. 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.

Navigating a Document with Multiple Pages

AutoVue provides navigation buttons in the AutoVue toolbar to allow you to flip through a drawing (for example, Office, 2D, EDA, or 3D files) that contains multiple pages.

The navigation buttons appear by default. The toolbar contains frequently used tools: **Previous Page**  , **Next Page**  , and **Page Number**  .

You can also access the navigation buttons from the **View** menu by selecting **Page**, and then the required navigation button.

4 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.

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. For PDF files, AutoVue highlights the text but maintains the current zoom level.

5. Click **Close** to close the Find dialog.

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.

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  . *You can also right-click the workspace and select Zoom - Previous.*
	Full Resolution	Display the file at full resolution. From the AutoVue toolbar, you can also click  .

Menu	Sub-Menu	Description
Fit ***NOTE:*** *Resizing the AutoVue window when an image in the workspace has been zoom fitted (Horizontal, Vertical or Both) results in the proportional resizing of the image.*	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. *You can also right-click the workspace and select Zoom - Page Width.*
	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  . *You can also right-click the workspace and select Zoom - Fit.*
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  . *The Pan and Zoom Window option is disabled for text-based documents and spreadsheets.* See "Using the Pan and Zoom Window" for more information.
Pan		Click and drag to move the drawing. To exit right-click. From the AutoVue toolbar, you can also click  .
NOTE: *The Rotate option is disabled for archived files, text-based documents and spreadsheets.*		

Menu	Sub-Menu	Description
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  .

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.

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.

   ADDITIONAL INFORMATION: Raster XRefs cannot be toggled on/off for AutoCAD and MicroStation drawings.
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 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.

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 File**.

   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.

   ADDITIONAL INFORMATION: You can hide/display certain overlays by selecting/deselecting corresponding check boxes in the Visibility column.
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.

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 view-point.

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.

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.
Compare		Compare two files. See "Comparing 2D Files" for more information.
Show Drawing Information	Select Single Entity	Displays information for the selected entity. 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 Entitiesfor more information.

Comparing 2D Files

AutoVue provides the ability to visually *compare* two files and display color-coded comparative data. Generally when comparing two files, you should first open the newer version of the document and then compare it with the older version.

When comparing files, AutoVue displays three windows:

- The right window displays the newer version of the document.
- The left window displays the older version of the document.
- The bottom window displays the comparison results.

By default, the result windows display added, deleted, and unchanged entities.

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. By default, the Comparison Result window displays the unchanged, deleted, and added information.

NOTE: *The Compare feature performs a graphical comparison not a geometrical comparison.*

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	Displays data that is present in the newer file but not in the older file.
View Deletions	Red	Displays data that is missing in the newer file but present in the older file.
View Unchanged	Blue	Indicates that there is no change between the newer file and old file.

TASK

1. View the newer file in AutoVue.
2. From the **Analysis** menu, select **Compare**.

   STEP RESULT: The File Open dialog appears.
3. Enter the file name or browse to select the file that you want to compare with the newer 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 newer file, the second displaying the older 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 any of the windows, click the ☐ button on the title bar of the window that you want to maximize. To minimize, click the – button. To restore the window, click 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 newer file appears in the workspace.

Align and Scale

You can scale or translate a file in order to compare files accurately. With the *Align and Scale* option you can select to automatically align the files, select points to align and scale, modify the coordinates (XOffset and YOffset) or enter a scaling factor for the second file.

TASK

1. While in Compare mode, from the **View** menu, select **Align and Scale**.

   STEP RESULT: The Align and Scale dialog appears.
2. Select one of the following options:

Option	Description
Automatic	AutoVue selects the best fit for the two files.
Select Points to Align	Select snapping points to align to.
Select Points to Align and Scale	Select snapping points to align and scale to.
Custom	You can enter a scale factor as well as X and Y offset values; XOffset and YOffset are relative to the base drawing and all options are displayed at their current values.

3. Click **Apply**.

   STEP RESULT: The align and scale modifications are applied to the file in the second window.
4. Click **OK** to the close the Align and Scale dialog.

Drawing Information

The *Drawing Information* option is only 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.

Working with PDF Files

This section lists AutoVue features that are specific to Adobe PDF files.

Navigating Through Markups

When navigating through markups, AutoVue preserves the current zoom level. For example, when the Go To feature is selected from the Markup Navigation tree, AutoVue highlights the markup entity in the workspace (AutoVue navigates to markup entity if on a different page) and maintains the current zoom level.

Measuring in PDF Files

When measuring vector-based PDFs, AutoVue allows you to snap to certain entities to take measurements. You can snap to the end-points or mid-points of an entity. AutoVue also allows you to snap to shapes (closed entities) for area measurements. This option also applies when adding a Markup Measure entity to the PDF file.

NOTE: *For Angle and Arc entities, only 3-point snapping is available.*

See "Measuring in 2D Files"

See "Creating 2D Vector Markup Measure Entities"

Text Search

You can perform a text search in vector-based PDF files where the scanner performs Optical Character Recognition (OCR). AutoVue highlights the text search result in the workspace and maintains the current document zoom level. If the text is very small, AutoVue zooms in to the highlighted text. If the text if outside the visible workspace area, AutoVue scrolls/pans to display the highlighted text.

See "Searching Text"

Text Selection/Copy

You can select and copy text from a vector-based PDF file where the scanner performs Optical Character Recognition (OCR).

To do so, highlight the text and then select **Copy** from the **Edit** menu (alternately, you can use the shortcut key **Ctrl+C**). You can then paste the text to the Note markup entity or a third-party application (for example, MS Word).

NOTE: *Note the following behaviors when copying text from a PDF file in AutoVue:*

- It may not be possible to copy text if the PDF file has a security restriction.
- When copying text from PDF files, AutoVue displays the text as it appears in the PDF. That is, AutoVue inserts a line break if text continues to a next line (word wrapping) in the PDF.
- In some PDF files, when copying text that includes line spaces, AutoVue may not insert the spaces. This is because the line spaces are not saved in the PDF file.

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"
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.
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

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 **Distance** tab.

3. 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.

4. Click a point on the drawing to define the starting point.
5. 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.
6. 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.
7. Click **Close** to close the Measurement dialog.

Distance in Vector Files

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 displaying the measurement options.
2. 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.

5 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, 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.

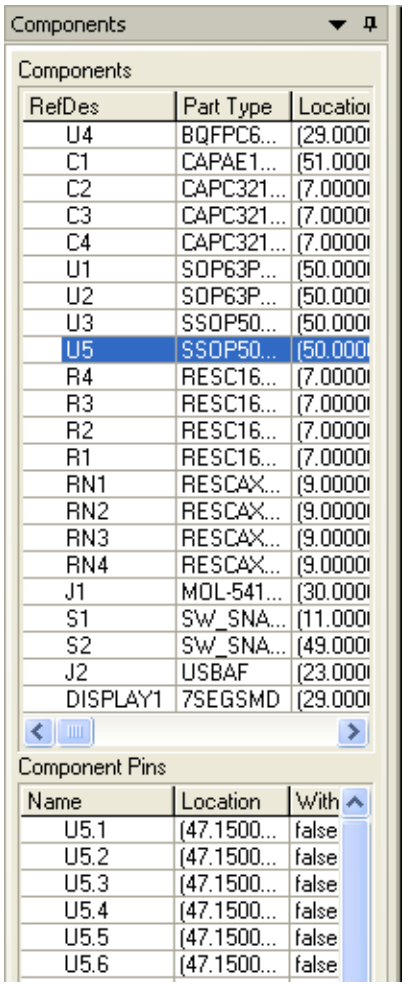
Navigation Panel

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.



Customizing Columns

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

TASK

1. To change the column order or to show/hide a column, click **Customize**.

   STEP RESULT: The Customize Columns dialog appears.

2. Select the check box beside the column or columns you want to display. 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.

3. 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.

4. To save the changes you made to the columns, select **Save column settings**.

   STEP RESULT: The new column settings will be retained the next time the file is opened.

5. Click **OK**.

RESULT:

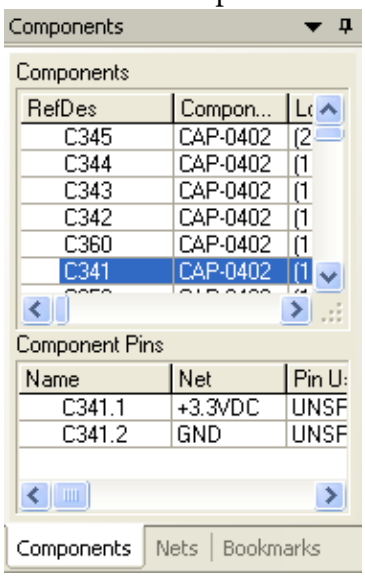
The changes appear in the Navigation Panel.

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.



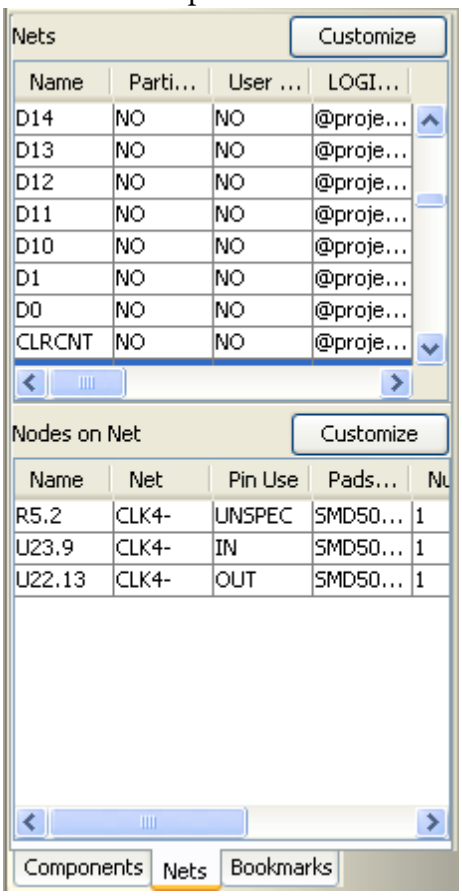
Nets Tab

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.

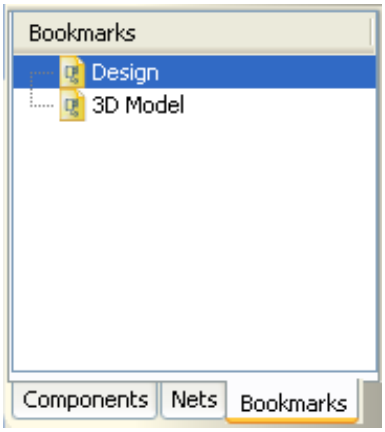
All selected nets are highlighted on the drawing. To select more than one net, press the **Shift** or **Control** key while selecting. Alternately, in the Nets panel,

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.



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 and 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"

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.

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:

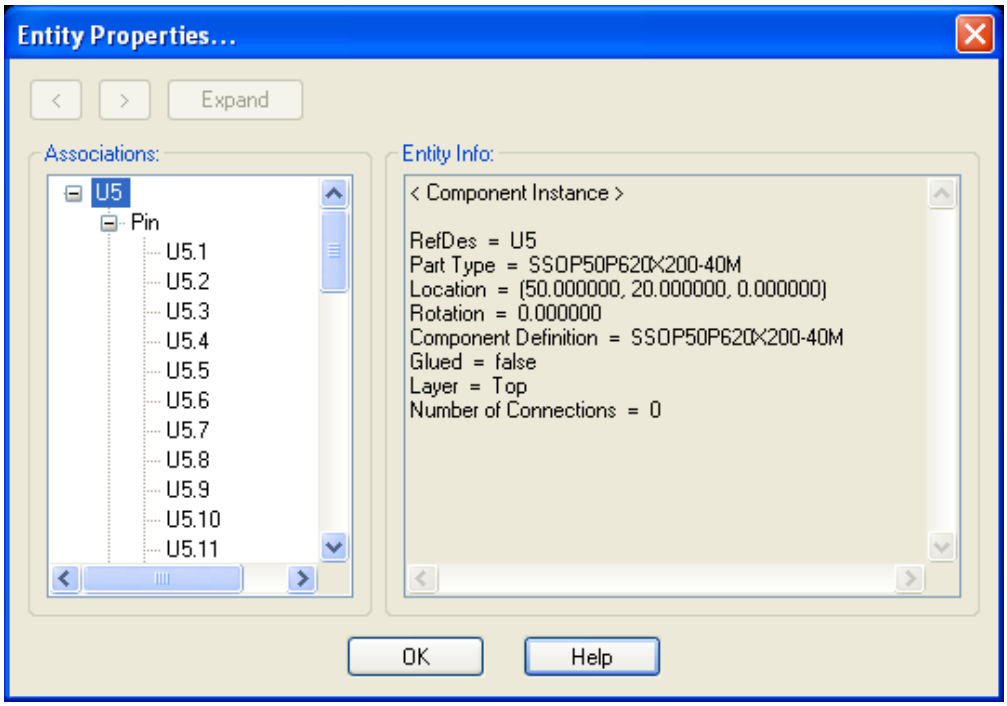
Only the selected entity types remain displayed in the workspace.

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

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.

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.

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.

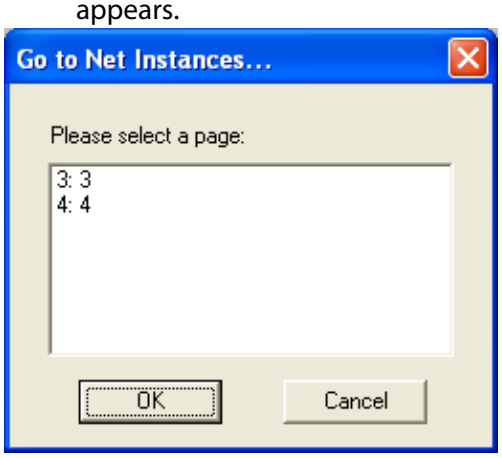
TASK

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.

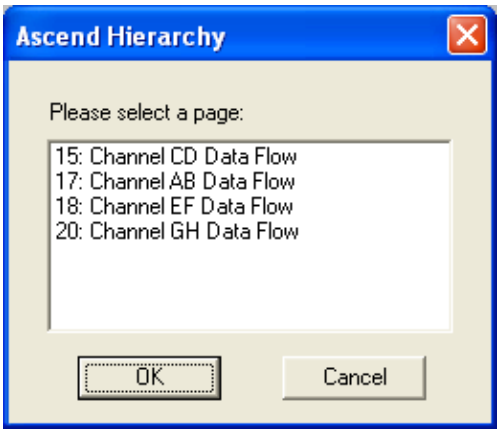
Navigating using Ascend Hierarchy

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.***

Navigating using Descend Hierarchy

On the parent page of your schematic's design hierarchy, select any hierarchical block in the workspace or in the Navigation Panel, then from the **View** menu select **Descend Hierarchy**. AutoVue opens the page with the selected child schematic.

You can also right-click the hierarchical block in the workspace and then select ***Descend 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.
One entity type for one physical layer	Click a specific check box.

3. Click **Apply** to view the changes in the workspace.
4. You can now save your changes as a user-defined layer set.
5. 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

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. 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**  .

4. Click **Apply** to view the changes in the workspace.

5. You can now save your changes as a user-defined layer set.

6. 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.

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.

*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.

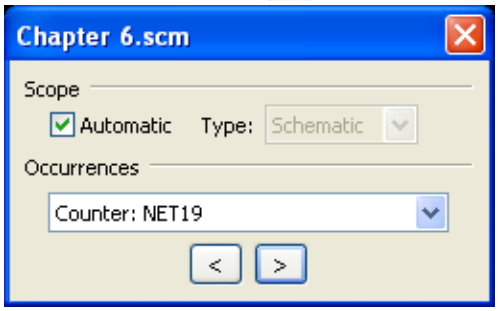
9. To change the view of a file in a window, do the following:

   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 enti-

ties 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.

NOTE: *The Compare feature performs a graphical comparison not a geometrical comparison.*

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** EXIT.

RESULT:

The original file appears in the AutoVue workspace.

Generating Bill of Material (BOM)

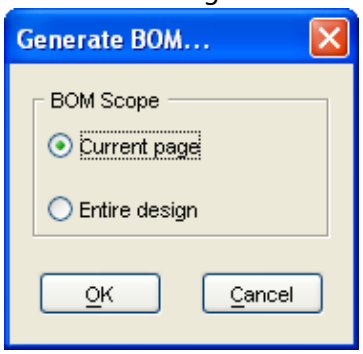
For EDA files, you can obtain a list of the components and parts required for manufacturing the item featured in the schematic drawing or PCB design.

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

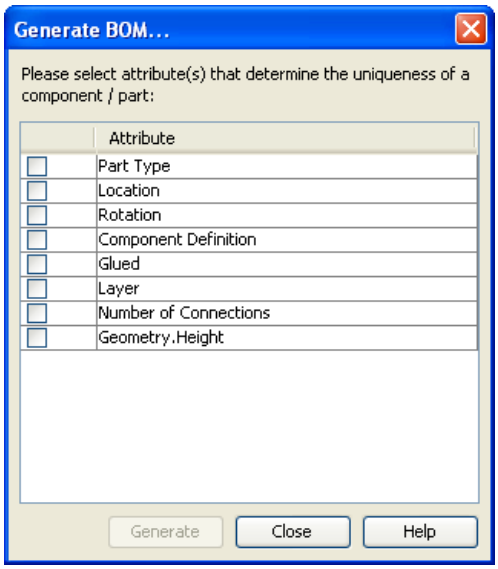
1. From the **Analysis** menu, select **Generate Bill of Material**.

   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**.

   STEP RESULT: The Save dialog appears.

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. *The two outlines (geometries) are part of the pin (pad) entity but on different logical layers.*

Type	#	Design Rule Check	Description
	2	Minimum Clearance (Pad > Trace)	Specify the minimum pad to trace clearance and pads on the same physical layer. *Only the pad outline is considered. Does not take into account traces connected to a pin (pad) on the same net.*
	3	Minimum Clearance (Trace > Trace)	Specify the minimum trace to trace clearance on the same physical layer. *Does not take into account directly connected traces.*
	4	Minimum Clearance (Component > Component)	Specify the minimum component to component clearance on the same physical layer (component top side or component bottom side). *Only the logical layer that defines the actual outline of the component is considered.* *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.*

Type	#	Design Rule Check	Description
Manufacturing	5	Minimum Annular Ring	Specify the minimum distance between the pad outline and the outline of its drill hole on all physical layers. *The two outlines (geometries) are part of the pin (pad) entity but on different logical layers.*
	6	Minimum Pad Diameter	Specify the minimum diameter of a pad. *Only applicable to rounded pads.*
	8	Acute Angle (in degrees)	Specify the minimum acute angle between connected traces on the same physical layer. *Only two traces belonging to the same net are considered.*
	9	Minimum Drill Hole Size	Specify the minimum diameter of the drill hole size allowed. *Only applicable to rounded pads.*
	10	Maximum Drill Hole Size	Specify the maximum diameter of the drill hole size allowed. *Only applicable to rounded pads.*
Electrical	7	Maximum Via Count	Specify the maximum via count allowed for a net.

Type	#	Design Rule Check	Description
	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.
	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. *Power and ground nets are the exception.*

Type	#	Design Rule Check	Description
	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. *NOTE: Trace entities will be divided into n sets where n is the number of nets in the design.*
	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

1. From the **Analysis** menu, select **Verify Design**.

   STEP RESULT: The Verify Design dialog appears.

2. 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.

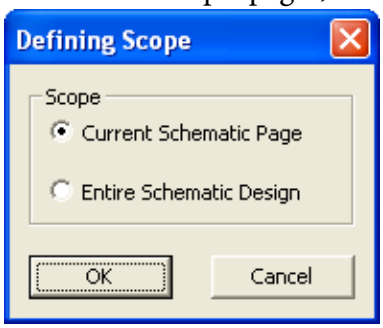
6. Click **Close** to close the Verify Design dialog.

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.

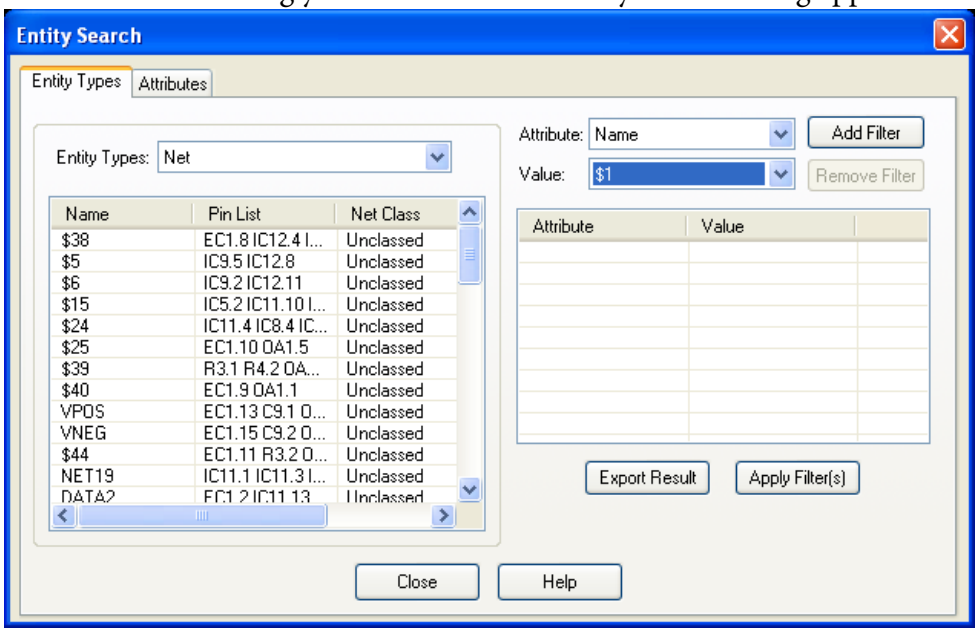
If the selected entity is small, a flash box appears indicating the location of the highlighted entity in the workspace.

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. ***NOTE:*** *This option is only available for PCB files.*

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.

6 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.

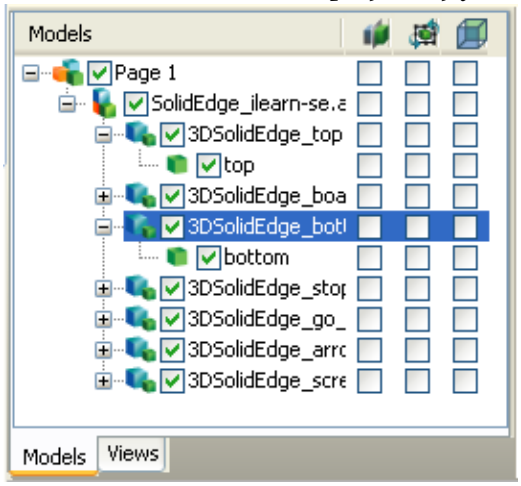
There are also features like Bill of Material (BOM) and Product and Manufacturing Information (PMI) Filtering.

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.

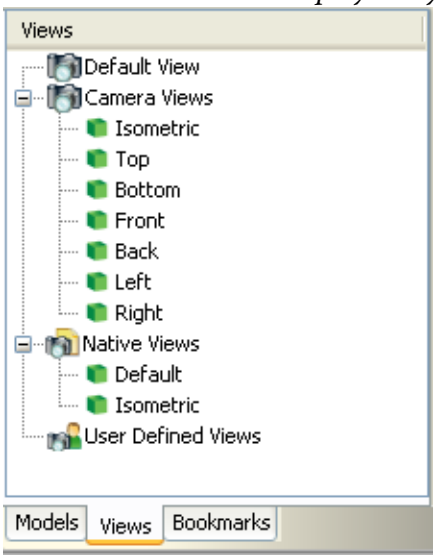
NOTE: *The Models tab displays only for 3D drawings.*



Views Tab

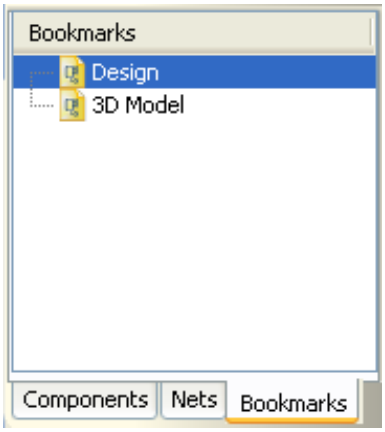
The *Views tab* lists the defaults, standard, native, and user-defined views. You can switch to a standard, native, or user-defined view, as well as add or delete user-defined views.

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 Part.**

   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 appears.

2. Click **Add**.

   STEP RESULT: The Open dialog 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. To import more than one file, repeat steps 2 to 4.

5. The **Enter Markup Mode** check box is selected by default. If you deselect it, any modifications you make to the mockup are not saved.

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

   ADDITIONAL INFORMATION: Once open, you can position these models in the desired manner using the Transformation tool.

RESULT:

The file or files you imported appear in the workspace.

See "Transforming a 3D Model"

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.

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.

Menu	Sub-Menu	Description
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.
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"

Menu	Sub-Menu	Description
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.

Method	Description
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 Part**. 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 applies to shaded and shaded wire models as well as reflective and reflective wire.

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.

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 specularity.

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:

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-orients 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.**

   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**.
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 PMIattributes 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 theEntity 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.

File Properties

The File Properties tab provides file-related properties of the selected entity, such as author, date of creation, keywords, title, and so on.

To view the file properties, select a part (or the whole model) and then select **Analysis** and then **Show Entity Properties**. From the Entity Properties dialog appears, click the File Properties tab.

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, double-click the PMI hyperlink entity.

   STEP RESULT: The 3D Hyperlink dialog appears if there are multiple hyperlinks associated with the selected PMI entity.
2. 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:

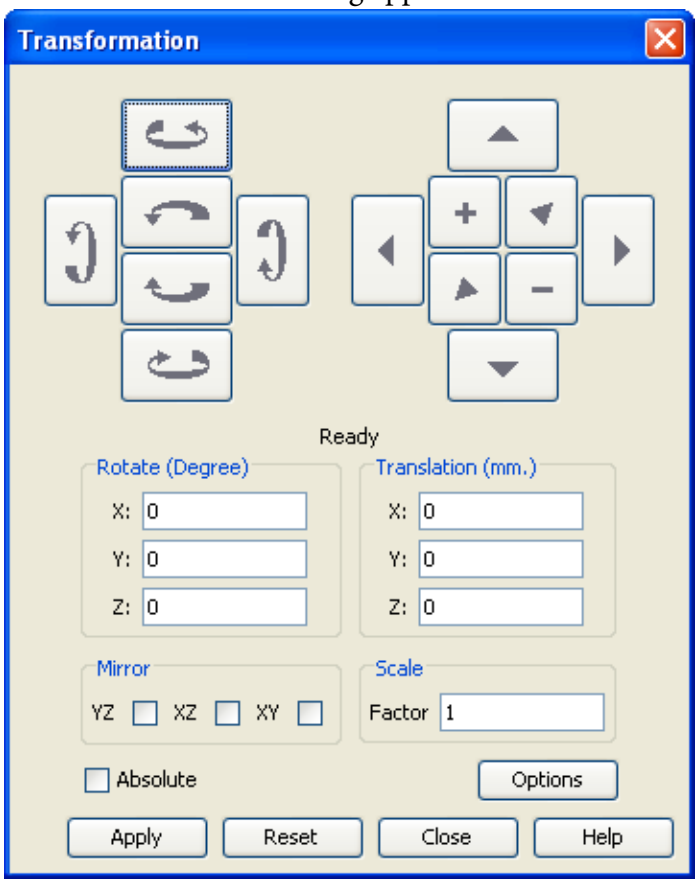
To restore the default state of a part of the model, from the **Manipulate** menu, select **Transform,** and then select **Reset Selected.** To restore the default state for the whole model, select **Reset All.**

Transforming a 3D Model

Using the illustrated transformation buttons or by entering X, Y or Z values, you can rotate, scale, or translate a model or any selection of model parts.

From the Manipulate menu, select **Transform,** and then select **Define.**

The **Transformation** dialog appears as follows:



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

In the Transformation dialog, the buttons on the left correspond to a rotational movement along the three axes, and the buttons on the right correspond to a translational movement 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 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**.

   ADDITIONAL INFORMATION: When you click a rotation, translation or scale button, the model transforms in increments of the values entered in the Options dialog.

   STEP RESULT: The Options dialog appears.

5. Enter the Translate Increment value in inches.

6. Enter the Rotate Increment value in degrees.

7. Enter the Scale Increment value.

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

9. Click **Reset** to set the model to its original transformation.

10. Click **Close** to close the Transformation dialog.

    STEP RESULT: The transformation state remains displayed.

11. 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  .

RESULT:

The model or selected parts are reverted back 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 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, select a check box.
6. To scale the model by a factor, enter a scale factor in the Factor field.
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.

    STEP RESULT: The transformation state remains displayed.

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.**

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.

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 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 >>.

   STEP RESULT: The whole model or the select model parts explode.

7. To implode the model, click <<.

   ADDITIONAL INFORMATION: To manually change the explosion state, click and drag the scroll bar.

8. Click **OK** to close the Explode dialog.

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, you can save the exploded view of the 3D model.

TASK

1. Explode a 3D Model.

2. Click **Save View** in the Explode dialog.

   STEP RESULT: The Add User Defined View dialog 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.

RESULT:

The exploded state is saved.

See "Resetting the Transformation of a 3D Model"

Comparing 3D Files

When comparing 3D files in a non-integrated environment we recommend that you use Universal Naming Convention (UNC) file names or the server protocol so as to ensure that all required sub-assemblies and parts are retrieved from correct paths. For information on UNC file names and the server protocol, refer to the *Installation and Configuration Guide*.

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. Generally when comparing two files, you should first open the newer version of the document and then compare it with the older version.

When comparing files, AutoVue displays three windows:

- The right window displays the newer version of the document.
- The left window displays the older version of the document.
- The bottom window displays the comparison results.

By default, the result windows display added, deleted, and unchanged entities.

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 , or deleted .

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. By default, the Comparison Result window displays the unchanged, deleted, and added information.

NOTE: *The Compare feature performs a graphical comparison not a geometrical comparison.*

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	Displays parts that are present in the newer file but not in the older file.
View Deletions	Red	Displays parts that are missing in the newer file but present in the older file.
View Unchanged	Blue	Indicates that there is no change between the newer file and older file.

Comparing 3D Files

The following steps describe how to compare files:

TASK

1. View the newer 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 newer file.
4. Click **Open.**

   STEP RESULT: AutoVue displays the Compare Tree and three windows. The first window displays the newer file, the second displays the older file, and the third displays the comparison results.
5. To view the properties of a modified or moved entity in the newer file and in the older 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 newer file to the older 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 newer file appears in the workspace.

Comparing Entity Sets

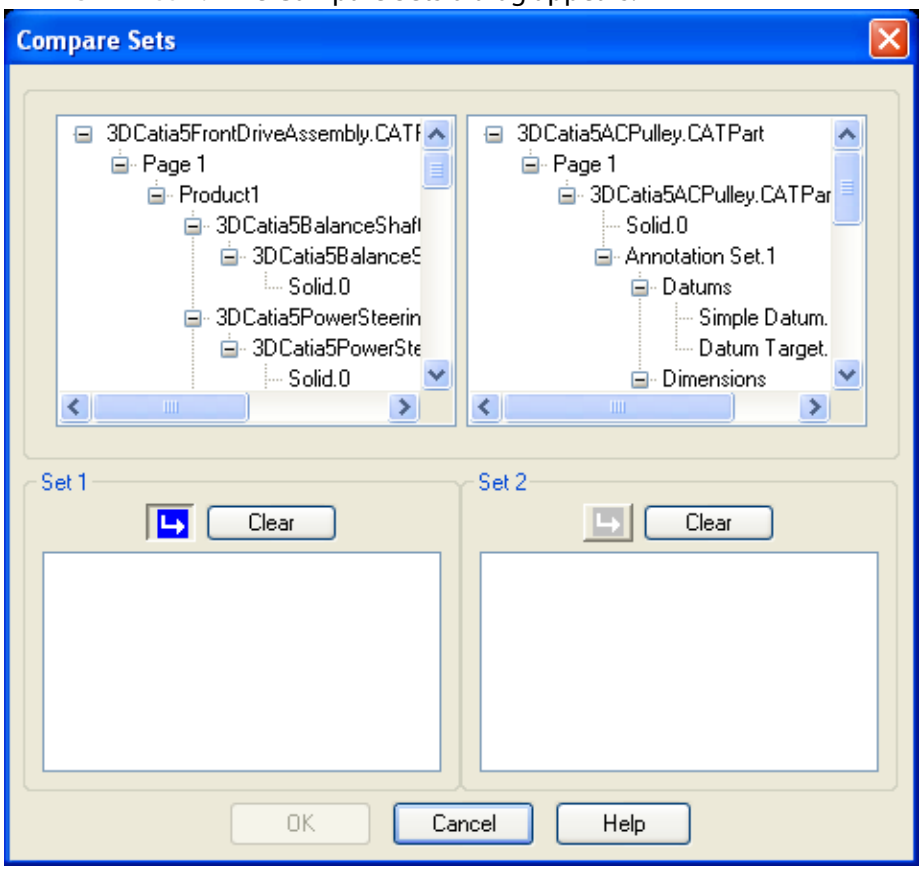
From the file comparison, you can compare a set of entities from one file against a set of entities from the another file. You can also compare sets of entities from the same file.

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.

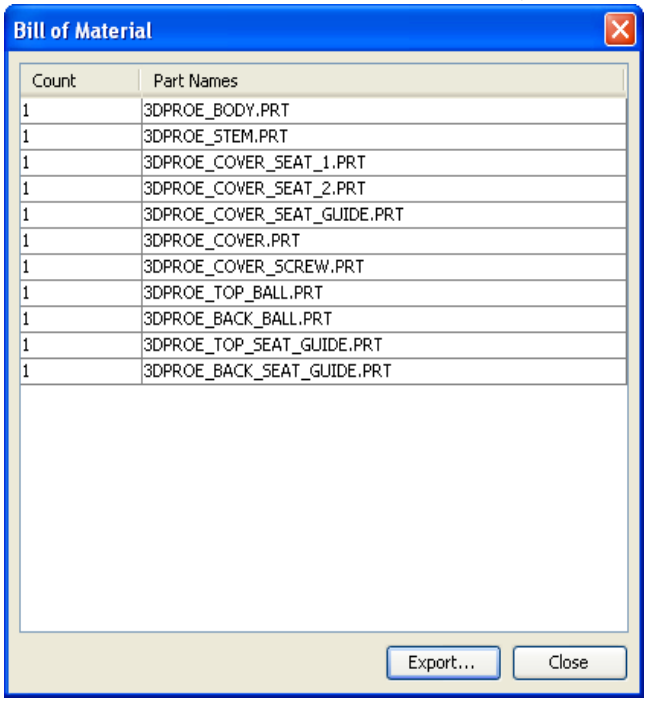
Generating a Bill of Material

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 Save As dialog appears.

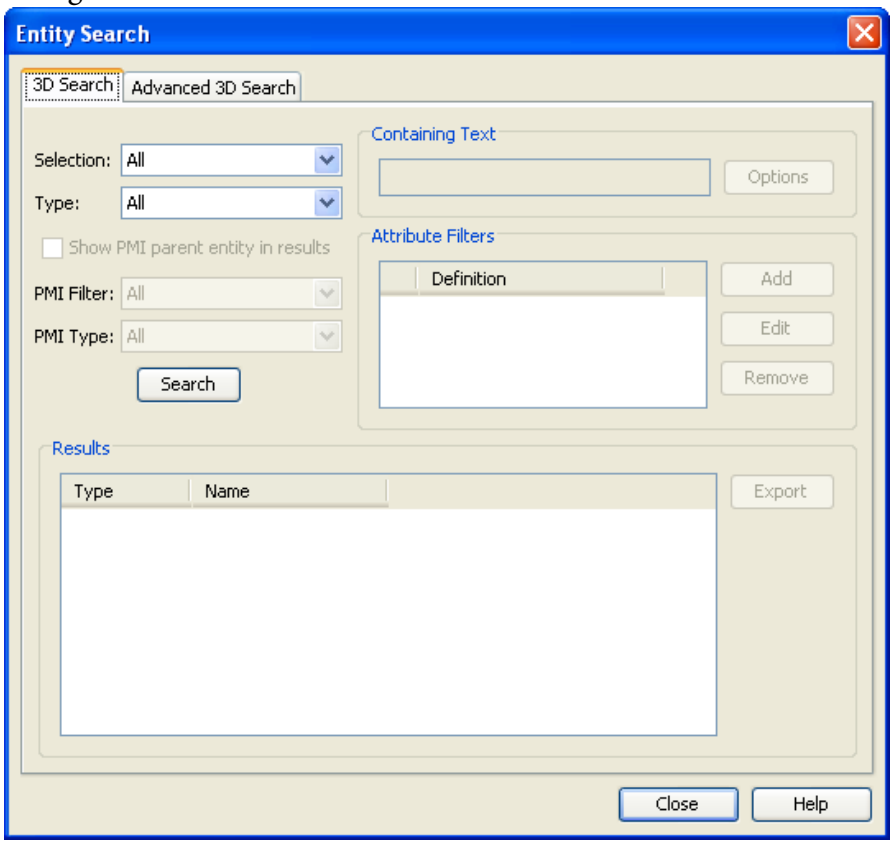
6. Enter a file name or browse to locate the directory where you want to store the file, then click **Save.**

   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 ***NOTE:*** *The attributes vary depending on the file being viewed.* 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

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. 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**.

4. 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.

5. 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.

6. 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 a 3D Text Search

From the **Containing Text** field of the Entity Search dialog you can search for a text string in the 3D model.

By default, AutoVue searches the attribute name and attribute value for the entered text.

Enter the text you want to search for in the **Containing Text** field and then click **Search**.

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

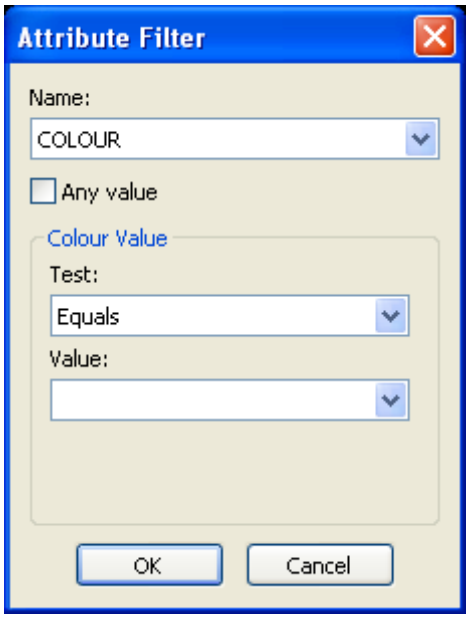
1. From the **Edit** menu, select **Entity Search.**

   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.

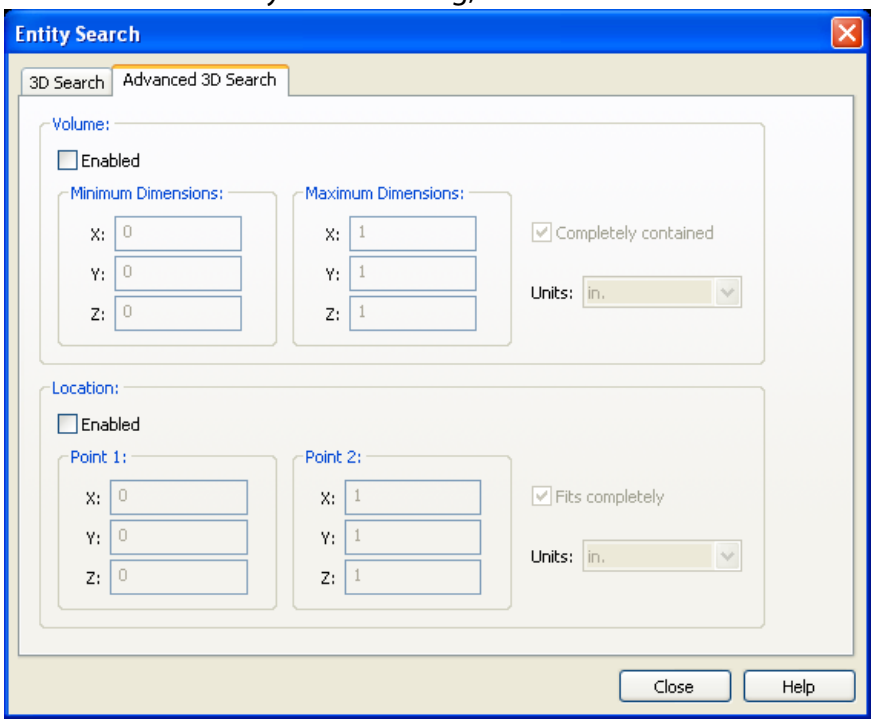
14. Click **Close** to close the Entity Search dialog.

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.



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.
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

S*napping modes* allow you to select or snap to different entity types on a model. For example, if you select **Vertex**, all vertices are highlighted and a snap box appears when you move the cursor over a vertex.

Snapping modes allow you to snap to the following entities.

NOTE: *The behavior of snapping modes change in regards to the selected measurement option. For example, the behavior of arc center is not the same for distance and minimum distance.*

Button	Description	Behavior
Vertex 	Highlights vertices on the model.	A snap box appears when you hover the mouse pointer over a vertex.
Edge (Line) 	Highlights edges on the model.	The edge is highlighted when you hover the mouse pointer. ***NOTE:*** *When measuring Minimum Distance, the finite edge is selected. However, when measuring Distance, the infinite edge is selected.*
Edge Middle 	Highlights edges on the model.	The edge is highlighted and a tooltip appears indicating the middle edge coordinate when you hover the mouse pointer.
Arc 	Highlights arcs on the model.	The arc is highlighted when you hover the mouse pointer.
Arc Center 	Highlights arcs and circles on the model.	The arc is highlighted and a tooltip appears indicating the arc center coordinate of the arc when you hover the mouse pointer. ***NOTE:*** *When measuring Minimum Distance, the arc center is selected. However, when measuring Distance, the infinite arc axis is selected.*
Face (Plane) 	Highlights faces when you move the cursor over a face.	The face is highlighted when you hover the mouse pointer.

Measuring Distance

Use the *Distance* option to measure the distance between two vertices, edges, arc axes, faces or any combination of these entity types.

The following table details how the distance between two entities are measured using snapping modes.

Snapping Mode Entities	Vertex	Line	Arc Center	Plane
Vertex	Distance between two points.	The shortest segment connecting the point and the line.	The shortest segment connecting the point and the arc.	The shortest segment connecting the point and the plane.
Line		Distance between two lines. ***NOTE:*** *The lines must be parallel.*	The line and axis must be parallel. Distance between the line and arc axis.	The distance between the line and the plane. ***NOTE:*** *The line must be parallel to the plane.*
Arc Center			Distance between the axes of the arcs. ***NOTE:*** *Arc planes must be parallel.*	Distance between the arc axis and the plane. ***NOTE:*** *The plane and the arc plane must be perpendicular.*
Plane				Distance between the two planes. ***NOTE:*** *The planes must be parallel.*

TASK

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

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

   STEP RESULT: The Measurementdialog appears.

2. Click the **Distance** tab.

3. Select the snapping modes that you want to use for measuring.

   ADDITIONAL INFORMATION: See "3D Snapping Modes" for more information.

   STEP RESULT: All entities of the selected snapping modes are highlighted on the model.

4. From the Units list, select a measurement unit.

5. On the model, select an entity as the starting point for the measurement.

   ADDITIONAL INFORMATION: If you want to take more than one measurement from the same starting point, select **Fix Position**.

6. Select the snapping modes 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.

RESULT:

The distance from the first set of entities to the second set is highlighted by a line. The measured distance, Delta X, Delta Y and Delta Z appear in the Measurement dialog. Additionally, the coordinates for the center point of each entity displays in the **From** and **To** fields.

To take another measurement click **Reset**.

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** an **Z** coordinates appear in the Measurement dialog.

6. Click **Close** the 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.

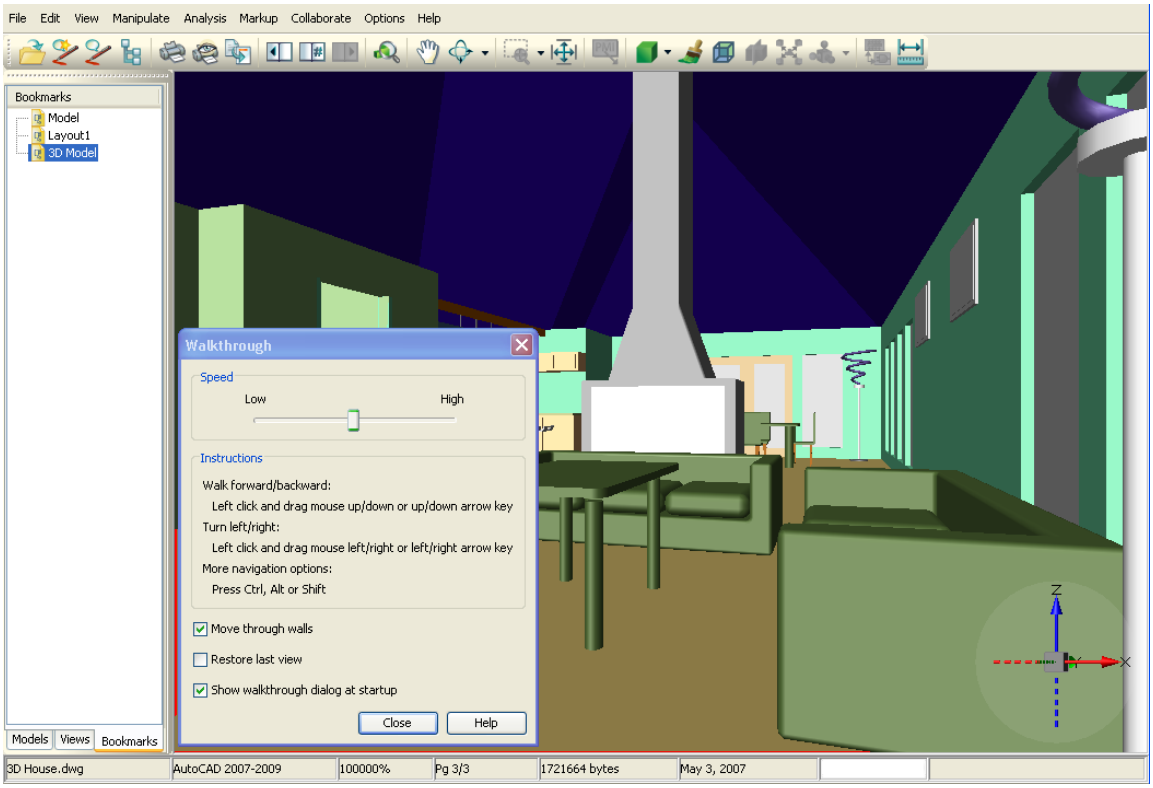
   STEP RESULT: The selected entity is highlighted and the measured surface appears in the Measurement dialog.

10. To take another measurement click **Reset**.

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

Walkthrough

The Walkthrough feature in AutoVue provides an interactive visualization of 3D models.

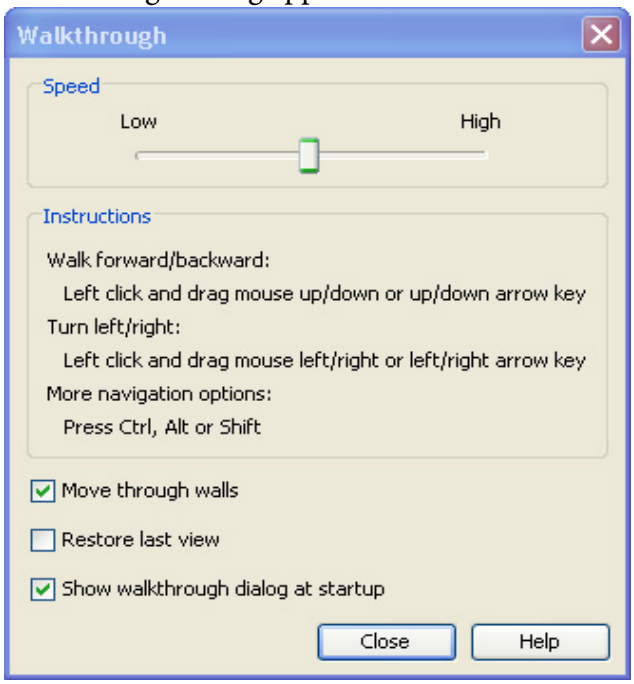


In Walkthrough mode you can manipulate the view of a 3D model as if you are walking or flying through the model, take measurements of the model, and change the camera position and orientation. For example, when viewing a 3D model of house, you can enter the front door and move between rooms and

floors. All the while, you have a 360 degree view of your surroundings and the ability to add markups.

Walkthrough Dialog

To enter Walkthrough mode, from the **View** menu, select **Walkthrough**. The Walkthrough dialog appears.



NOTE: *Closing the Walkthrough dialog does not exit Walkthrough mode. To exit, you must deselect* ***Walkthrough*** *from the* ***View*** *menu.*

The Instructions panel explains the possible model manipulation options while you perform the walkthrough. See "Walking Through a 3D Model" for more information.

From the Speed panel of the Walkthrough dialog, you can adjust the increment speed of the camera for all walkthrough functions.

By default, the **Move through walls** option is selected. This feature allows you to walk through walls when viewing a model. You can deselect the option to enable the collision detection feature.

The **Restore last view** option saves the last view of the model before exiting walkthrough mode. As a result, when you re-enter walkthrough mode, the last view displays.

Select the **Show walkthrough dialog at startup** option to display the Walkthrough dialog when you enter Walkthrough mode. Deselect the option to disable the Walkthrough dialog on startup.

*NOTE: If you want to re-enable the Walkthrough dialog on startup, from the **Options** menu select **Configure**. From the Configuration dialog, select **Miscellaneous** and then select the **Show walkthrough dialog at startup** checkbox. Refer to Configuring AutoVue for 3D Files for more information.*

Walking Through a 3D Model

If you want to position yourself on a surface of the 3D model, press the **Alt** key and double-click a point on the surface. For example, to walk through a 3D model of a house, you can Alt + double-click the landing in front of the door.

Once you are at your desired view point of the 3D model, you can use the arrow keys or mouse to navigate through the model.

The following table lists the available keyboard/mouse actions and their function:

Keyboard Action	Mouse Action	Function
Up arrow, Down arrow	Click and drag up, down, or roll the mouse wheel forward and backward	Move forward, backward
Left arrow, Right arrow	Click and drag left, right	Turn left, right
Alt + Up arrow, Alt + Down arrow	Alt + drag up, down or Mouse middle button click and drag up, down	Move up, down (Elevation)
Alt + Left arrow, Alt + Right arrow	Alt + drag left, right or Mouse middle button click and drag left, right	Side step left, right
Ctrl + Up arrow, Ctrl + Down arrow	Ctrl + click and drag up, down	Look up, down
Ctrl + Left arrow, Ctrl + Right arrow	Ctrl + click and drag left, right	Rotate
	Alt + double click	Places camera perpendicular to selected plane

Adding Markups in Walkthrough Mode

When walking through a 3D model you can add a Note markup entity.

NOTE: *The Note markup entity is the only markup supported in Walkthrough mode.*

TASK

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

   ADDITIONAL INFORMATION: From the Markup Properties toolbar, you can also click **New Markup**.

   STEP RESULT: AutoVue enters Markup mode.

2. Create a Note markup entity. See "Adding a Note" for more information on how to create Note markup entities.

   ADDITIONAL INFORMATION: You can continue walking through the model while creating new Note markup entities.

3. Save your new markups. See "Saving a New Markup File" for more information.

   STEP RESULT: The markups are saved.

4. To exit Markup mode, from the **Markup** menu, select **Close All**.

5. To view the 3D model's associated Markup files, select **Open** from the **Markup** menu. See "Opening Markup Files" for more information.

   ADDITIONAL INFORMATION: The Note markup entities appear in the Markup Entity tree, but not in the workspace. To view the markups, double-click the Note entities in the Markup Entity tree.

   STEP RESULT: The markups open and the Note markup icon appears in the workspace.

7 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.**

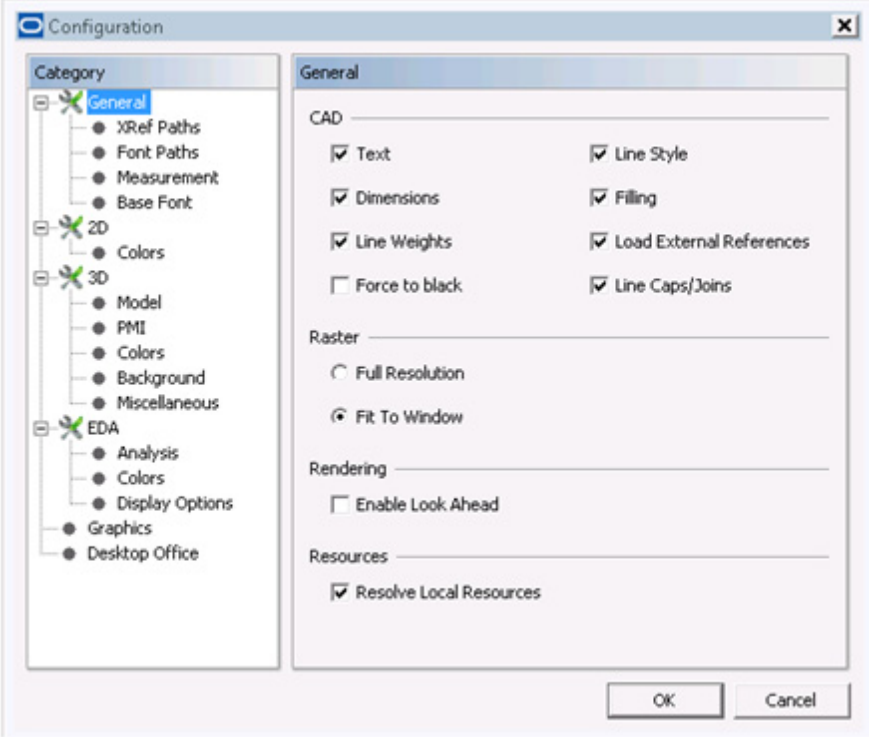
General Options

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.

Rendering

When **Enable Look Ahead** is selected and you zoom into a part of a file, AutoVue renders adjacent tiles. The advantage to this is improvement in performance when zooming in on parts of a file. The disadvantage is that if the client machine is idle for at least one second, **Enable Look Ahead** is triggered which could slow down current operations. However, once all tiles are rendered, all zoom operations speed up again.

If **Enable Look Ahead** is deselected, AutoVue renders the tiles when requested, that is, only when you zoom in on parts of the file.

Resources

If **Resolve Local Resources** is selected, and AutoVue tries to locate XREFs on the client-side using client path settings. If not found on the client-side,

AutoVue will attempt to retrieve the XREFs on the server. If Resolve Local Resources is unchecked, AutoVue attempts to resolve the XRefs on the server-side only.

Configuring Paths

Configure the paths for XRefs and fonts. These path settings are read only if Resolve Local Resources is checked.

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.

Path	Description
XRefs	The directory paths for any external reference files associated with 2D, 3D or EDA files.
Font	The directory paths for fonts required by AutoVue's vector files.

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 Category tree, expand **General,** and then select **XRef Paths**.
2. Click **Add**.

   STEP RESULT: The Add Path dialog 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**** results 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*** results 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**.
9. Click **OK** to close the Configuration dialog.

Configuring Font Paths

Font Paths are the directory paths for fonts required by 2D, 3D, EDA, or PDF files.

TASK

1. From the Category tree, expand **General,** and then select **Font Paths.**
2. Click **Add**.

   STEP RESULT: The Add Path dialog appears.
3. Enter the directory path or scroll to locate the directory where the external font files are located.

   ADDITIONAL INFORMATION: 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". 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".
4. Click **OK**.

   ADDITIONAL INFORMATION: To add more than one path to the list, repeat steps 3 to 5.

   STEP RESULT: The directory path appears.
5. 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.
6. To remove a path, select the path and click **Remove**.

   STEP RESULT: The selected path disappears from the list.
7. Click **OK** to close the Configuration dialog.

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
Decimal Digits	Lets you specify the number of decimal digits displayed for each measurement. Enter a number from 1 to 18.
Default File Units	Lets you specify the units to use for drawings when the drawings do not contain units.
Measurement Units	Lets you specify the default unit to use for measurements.

Configuring the Base Font for Archive and Text Files

The **Base Font** option lets you define the font properties for archive, text, and spreadsheet files.

TASK

1. From the Category tree, expand **General,** and then select **Base Font.**
2. Select a font from the Font list.
3. Select a font size from the Size list.
4. Select the **Bold** option, **Italic** option, or both optionsto change the font style.

   ADDITIONAL INFORMATION: You can see a preview of the text in the Sample area.
5. Click **OK** to apply the font change and close the Configuration dialog.

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.

NOTE: *You must restart the AutoVue client for the configuration to take effect.*

Overlays Extents Settings

When you add overlays, AutoVue automatically tries to scale the overlay extents to match the base file extents. If you wish to disable this behavior, clear the Match Extents checkbox in the Overlays section of the 2D Configure dialog.

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

1. From the **Options** menu, select **Configure.**

   STEP RESULT: The Configuration dialog appears.

2. From the Category tree, select **3D.**

   STEP RESULT: The **Rendering, Dynamic Rendering,** and **Frame Rate** 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.
Tristrip	If selected, enable/disable tristripping of mesh data for display.

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.

Frame Rate

The **Frame Rate** slider lets you define the frame rate for rotating and dynamic zooming on 3D files. Drag the slider to specify the frame rate. Drag the slider to the left for a lower frame rate and to the right for a higher frame rate.

A higher frame rate results in a lower resolution.

Optimized Rendering

Select the **Enable Optimized Rendering** option to accelerate the rendering time of drawings. Note that

NOTE: *You must restart your AutoVue client after selecting or deselecting this option to apply the setting.*

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, Dynamic Load Mesh Resolution, Initial Visibility,** and **PMI Initial Visibility** options.

Loading

Control the streaming method with the **Dynamic Loader** option.

When the option is selected, AutoVue loads enough detail to display an accurate visual representation of the model. The data is loaded based on the level of detail that is necessary at that point in time.

When the option is deselected, AutoVue requests data in 10% chunks until it reaches full resolution for the file. You would see the model initially at coarse resolution, but continuously refining.

Dynamic Load Mesh Resolution

Control the initial resolution with the **Dynamic Load Mesh Resolution** slider.

When you select **Dynamic Loader**, you can control the initial resolution by setting the **Dynamic Load Mesh Resolution**. When you set the Dynamic Load Mesh Resolution to High, the file displays at a higher resolution when you zoom, resulting in a smoother look.

For more information, refer to the *Installation and Configuration Guide.*

Initial Visibility

With the **Initial Visibility** options, you can specify the visibility of model parts when first opening a 3D file.

Option	Description
Default Visibility	Load model with default visibility options.
All Visible	Force all parts ON in the display.
All Invisible	Force all parts OFF in the display. To display model parts, select the part(s) from the Model Tree.

PMI Initial Visibility

With the **PMI Initial Visibility** option, you have the option to set a threshold for the number of PMIs to display for large models. To do, select the **Don't display PMI for large models** check box, and then enter the number of PMIs to display in the **PMI Threshold** field.

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 3D view.
Selection	Set the color when selecting a model or model parts.
Min. Distance Set 1	Set the color of the first set-point when measuring minimum distance.
Min. Distance Set 2	Set the color of the second set-point when measuring minimum distance.

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 the collapse the whole tree.

Manipulator

Enable the **Align with current UCS** option to automatically align the manipulators with the current User-defined Coordinate System.

Selection

The following **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.

Walkthrough

The following **Walkthrough** options let you configure the view to use when you enter walkthrough mode and to control whether or not you want to see the Walkthrough dialog at startup.

Option	Description
Restore Last View	Select this option to save the last view of the model before exiting walkthrough mode. As a result, when you re-enter walkthrough mode, the last view displays.
Show Walkthrough dialog on startup	Select this option if you want the Walkthrough dialog to display when you start walkthrough mode. Clear this option if you do not want to see the Walkthrough dialog on startup.

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 theConfiguration 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. When the option is selected, AutoVue automatically retrieves the entity information from the server. Clear this option if you do not want this behavior every time your mouse hovers over an entity.

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**. ***NOTE:*** *Applicable to Cadence Allegro format only.*

Option	Description
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**. ***NOTE:*** *Applicable to Cadence Allegro format only.*

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. ***NOTE:*** *Applicable to Cadence Allegro format only.*
	Non-plated Hole Visibility	Select whether to use the native settings or turn on /off the non-plated hole visibility. ***NOTE:*** *Applicable to Cadence Allegro format only.*
	Pad Filling	Select whether to use the native settings or turn on/off pad filling. ***NOTE:*** *Applicable to Cadence Allegro format only.*
	Line Caps/Joins	Select whether to use the native settings or turn on/off line caps/joins. ***NOTE:*** *Applicable to Cadence Allegro format only.*
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

1. From the **Options** menu, select **Configure**.
   STEP RESULT: The Configuration dialog that appears.

2. Select **Graphics** in the tree.
3. 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

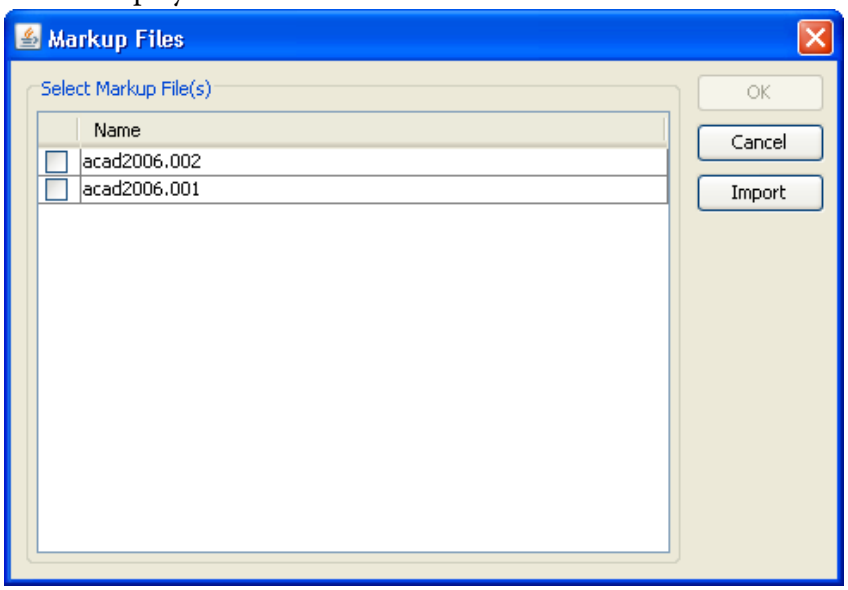
1. From the **Options**menu, select**Configure.**
   STEP RESULT: The Configuration dialog appears.
2. Select **Desktop Office** in the tree.
3. Select a background color for each Desktop Office file type.

8 Markups

AutoVue has the ability to view many 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 entities are saved in Markup files.

If a document 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 Filesdialog and allows you to select the Markup files to display.



When you open a Markup file, AutoVue layers the markup over the original file.

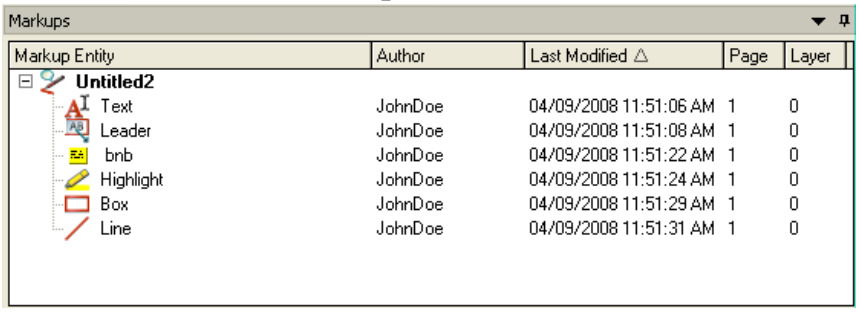
In Markup mode you can:

- Create entities such as text, arcs, boxes, circles, clouds, lines, arrows, and polygons. Note that AutoVue exits markup creation mode after you create a markup entity. In previous releases of AutoVue, you were able to continue adding markups until you right-click the workspace.
- 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. Note that measurements made in Markup mode have up to a 1% margin of error.
- Navigate markups through a hierarchy tree, view markup properties, and sort the tree according to each property.
- View markup layers individually or in combination.
- When in Markup mode, you can press Shift+Left Mouse button to select multiple markups entities in order to drag or copy and paste to a new location.
- Press **Esc** at any point to cancel the markup creation.

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.**



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.

Property	Description
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.*

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.

Option	Description
Before	Displays all markup entities modified before the selected date.
After	Displays all markup entities modified after the selected date.
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, with the exception of PDF files, the view state of the file is also saved. View states include zoom level (extents), rotation 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.**

NOTE: *When selecting Go To feature when viewing PDF files, AutoVue highlights the markup entity and preserves the current zoom level.*

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

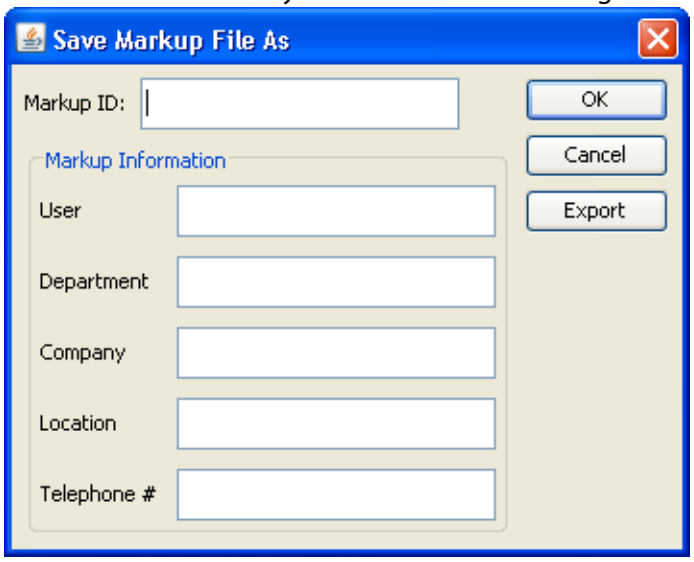
To save a new Markup file:

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.



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 (*.dfx)
- 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

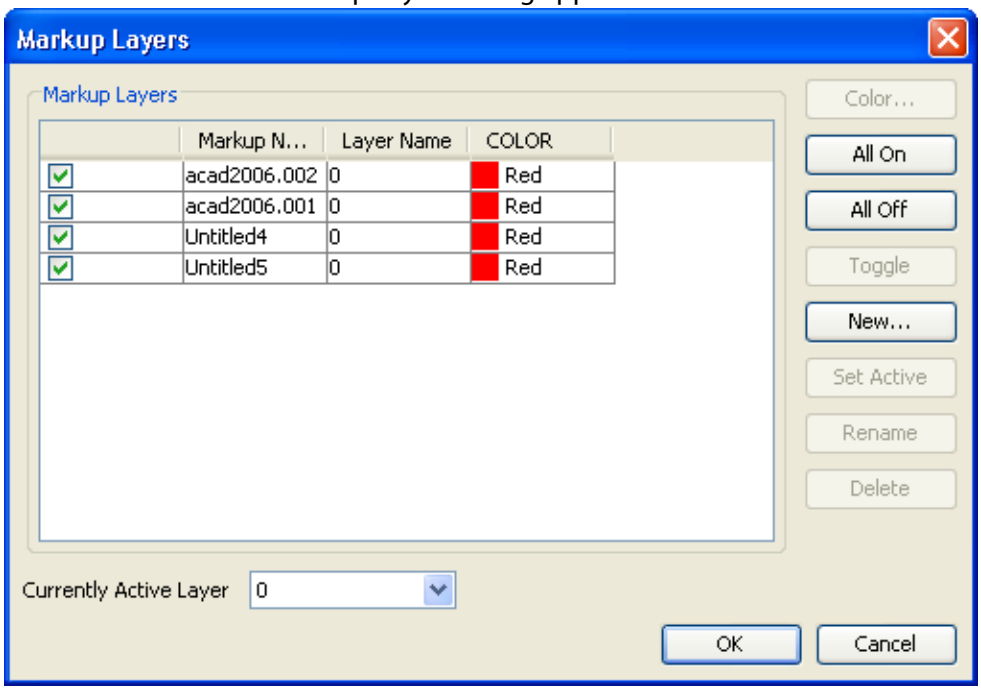
Creating a Markup Layer

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 andclick **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 IDfor 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 "2D-Specific Markups" for information on markup entities that are specific to 2D files.

See "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 action.*

The following tables 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	Attaches a hyperlink as a markup. From the Markup Entity toolbar, you can also click **Hyperlink**. See "Adding a Hyperlink"
Signoff	Creates 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"
Symbol	Adds a symbol markup to the document. From the Markup Entity toolbar, you can also click **Symbol**. See "Adding a Symbol"

Adding an Attachment

In AutoVue, 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 appears.

3. In the **Link 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 **URL** field, enter the path to the file or click **Browse.** The **File Open** dialog appears.

5. Select the file or click **Browse** and click **OK.**

   ADDITIONAL INFORMATION: You can attach any local file or any DMS file.

6. From the Open In list, select one the following:
   - **A new applet window:** Opens the attachment in a new AutoVue window.
   - **Current applet window:** Opens the attachment in the current AutoVue window.
   - **Associated application:** Opens the attachment in its associated application.

7. Click **OK** to close the Attach File dialog.

   STEP RESULT: The attachment appears on the markup workspace and in the Markup Navigation Tree.

8. Right-click in the workspace to complete the attachment.

Opening an Attachment

To open an attachment, do one of the following:

- Double-click or the attachment icon in the workspace or Markup Navigation Tree.
  - If **A new applet window** was selected while attaching the file, it will open the attachment in a new AutoVue window.
  - If **Current applet window** was selected while attaching the file, it will open in the current AutoVue window.
  - If **Associated application** was selected while attaching the file, it will open in its associated application.
- In the tree, right-click the attachment entity in the tree, select **Attachment**, and then select one of the following:
  - **Open:** The attachment opens in AutoVue.
  - If **A new applet window** was selected while attaching the file, it will open in a new AutoVue window.
  - If **Current applet window** was selected while attaching the file, it will open in the current AutoVue window.
  - **Open With:** You have the option to open the attachment with **AutoVue** or its **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, a third-party software applicaiton, or a Web page URL. You can create hyperlinks in your current file so that your outside files, software applications, and Web pages 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

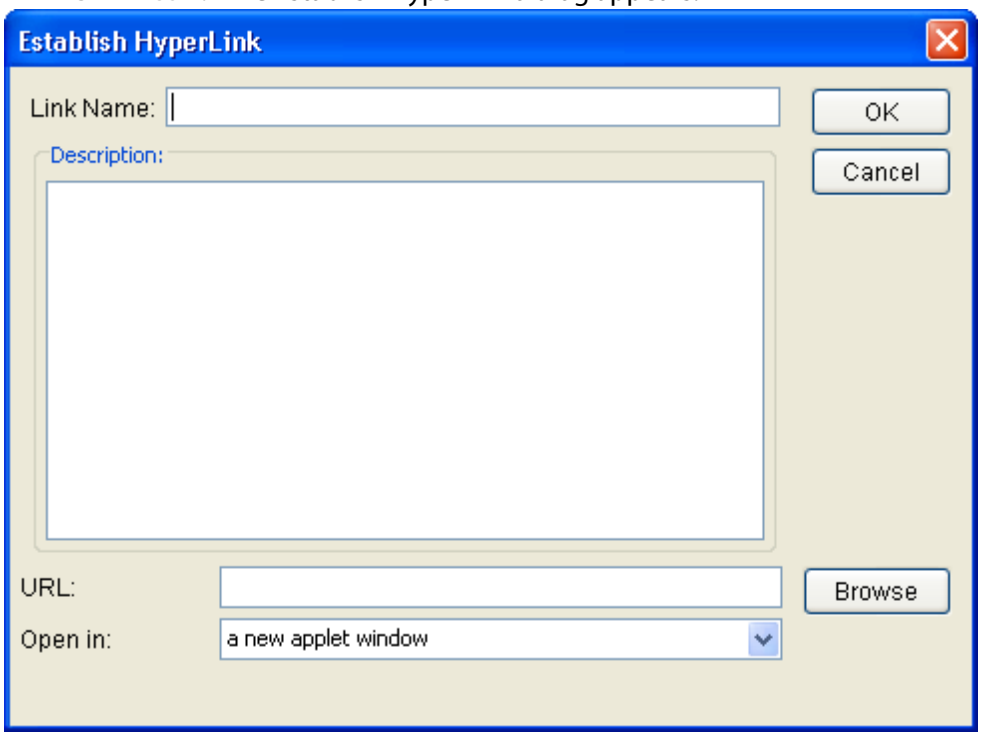
TASK

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 appears.



3. Enter a **Link Name**.
4. Optionally, enter a **Description**.
5. Type the **URL** or click **Browse** to locate the file that you want to link to.

   ADDITIONAL INFORMATION: Make sure to include the http:// protocol in the URL. If the protocol is not included, or if no value is entered in the URL field, then an error message is thrown when the Hyperlink markup entity is opened.
6. From the Open In list, select where you want the hyperlink to open.

Open in	Description
A new applet window	Opens the file in another AutoVue window.
Current applet window	Opens the file in the current AutoVue window.
A new browser window	Opens the file in the default browser window.
A current browser window	Opens the file in the current browser window.

ADDITIONAL INFORMATION: If you are creating a hyperlink to a Web page URL, then you must select **A new browser window**.

7. Click **OK**.
8. Right-click to exit hyperlink creation mode.

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

1. 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 a Symbol

A Symbol[1] is a graphical entity, such as a company logo. Before a graphic entity can be used as a Symbol, it must be added to a Symbol Library.

You can create a Symbol Library and add Symbols to it. You can also add or remove Symbols from existing libraries.

NOTE: *When a Symbol is created, the graphic entity is converted to a Windows Metafile Format (WMF). As a result, the resulting graphic in the Symbol may differ from the original image as it has been converted to another format. In order to avoid these discrepancies in the output, it is recommended to use AutoCAD formats as the graphical entity. However, you may still use any 2D/raster image*

1. In previous releases of AutoVue, a Symbol markup entity was called a Stamp.

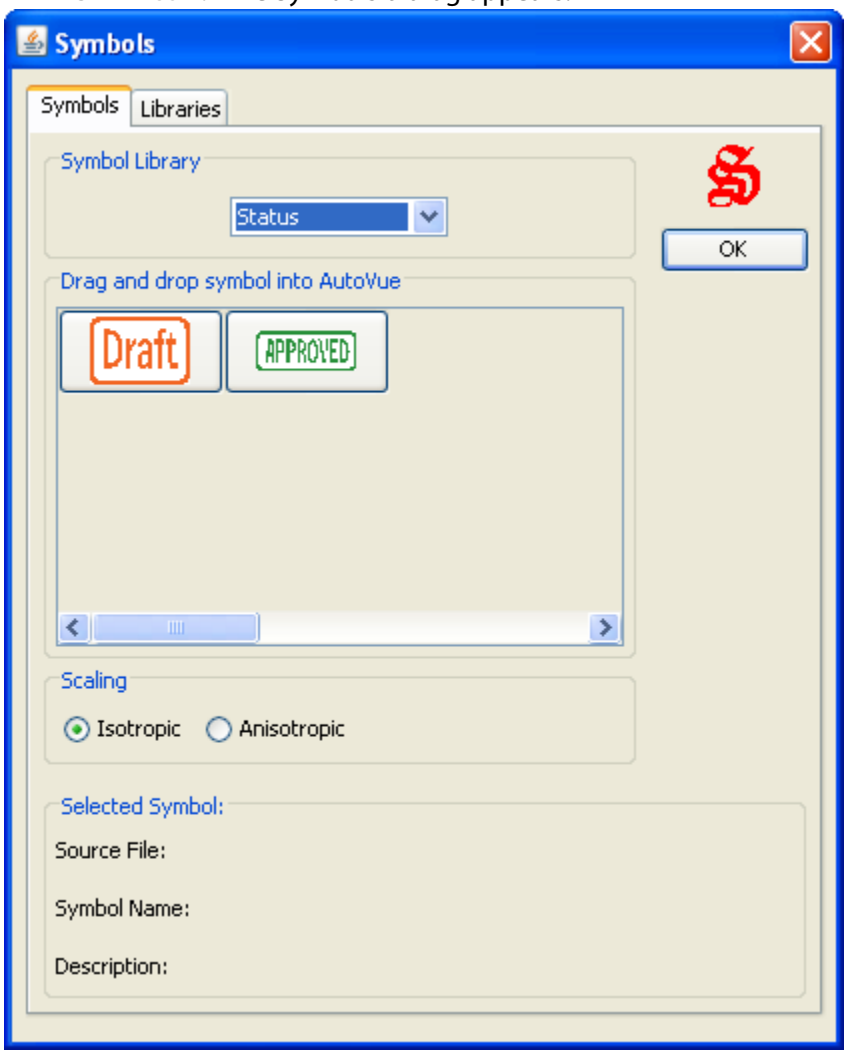
as the graphical entity, but differences between the Sybmol and native file should be expected.

TASK

1. From the **Markup** menu, select **Add Entity,** and then select **Symbol.**

   ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Symbol** .

   STEP RESULT: The Symbols dialog appears.



2. Click the **Symbols** tab.
3. From the Symbol Library list, select the library from which you want to select a Symbol.

   STEP RESULT: The Symbols appear for the selected library.
4. Click **Isotropic** if you want to scale the Symbol proportionately.
5. Click **Anisotropic** if you want to scale the Symbol disproportionately.

6. Drag the Symbol onto the workspace.

   ADDITIONAL INFORMATION: To add more Symbols, repeat steps 3 to 5.

   STEP RESULT: The Symbol appears on the drawing and in the Markup Navigation Tree.

7. Close the Symbols dialog.

Creating a New Symbol Library

Once a Symbol library is created, the existing library should not be modified. If it is modified, it will not be possible to add or remove existing symbols.

TASK

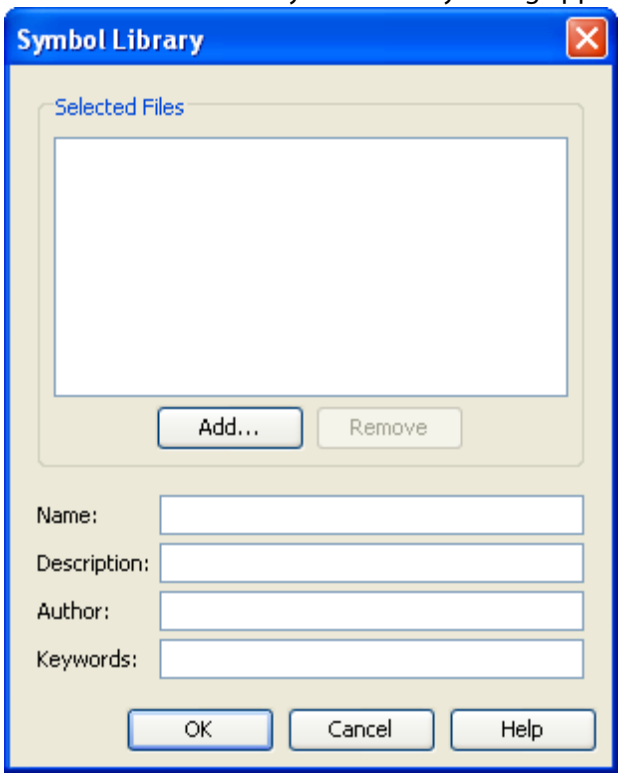
1. From the **Markup** menu, select **Add Entity,** and then select **Symbol.**The Symbols dialog appears.

   ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Symbol**  .

2. Click the **Libraries** tab.

3. Click **Create.**

   STEP RESULT: The Symbol Library dialog appears.



4. Enter the library name, description, author, and keywords in their respective fields.

5. To add a symbol to the library, click **Add.**

   STEP RESULT: The Open dialog appears.

6. Browse to locate the symbol you want to add, and then click **Open**.

   ADDITIONAL INFORMATION: To add more symbol, repeat steps 4 to 6. To remove a symbol, select the symbol and click **Remove**.

   STEP RESULT: The file appears in the Symbol Files list.

7. Close the Symbol dialog.

Deleting a Symbol Library

TASK

1. From the **Markup** menu, select **Add Entity,** and then select **Symbol.** The Symbols dialog appears.

   ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Symbol**  .

2. Click the **Libraries** tab.

3. From the Symbol Libraries list, select the library that you want to delete.

4. Click **Delete**.

   STEP RESULT: The library disappears from the list and from the Symbol Library list under the **Symbols** tab.

5. Close the Symbols 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 2D markup entities. To access the markup entities, from the **Markup** menu, select **Add Entity.** The available 2D markup entities are listed in the following table.

Markup Entity	Entity Information
Arc	Click and drag the mouse to draw an Arc. From the Markup Entity toolbar, you can also click **Arc**  .
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	Create freestyle entities. From the Markup Entity toolbar, you can also click **Freestyle**  . See "Adding a Freestyle Entity"
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.
Leader	Create leader entities. From the Markup Entity toolbar, you can also click **Leader**  . 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"

Markup Entity	Entity Information
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**.
Polyline	Click and drag to draw a polyline. From the Markup Entity toolbar, you can also click **Polyline**. 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. See "Aligning a Line Segment to the Vertical or Horizontal Axes"
Stamp	Adds a Stamp to a document that 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 **Stamp**. See "Creating a Stamp"
Text	Add text to the markup. From the Markup Entity toolbar, you can also click **Text**. See "Adding Text"

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 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.

Creating a Non-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. Click a point on the drawing where you want to interrupt the Freestyle entity.
5. 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.
6. Right-click to end the Freestyle entity.

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.

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.

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**  .

Option	Description
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 theMeasurement 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 a Stamp

NOTE: In previous releases of AutoVue, Stamp markup entities were called Intellistamp.

The Stamp markup entity lets you create a stamp that includes document and user information (metadata) pulled directly from the backend DMS/ERP/PLM/UCM system.

NOTE: The Stamp is only available if a system administrator has defined and configured the Stamp.

A Stamp can retrieve a document's attributes directly from the backend system. They can also update attributes from the Stamp to the document in the backend system.

Backend system attributes that the Stamp can read and write to can be simple values or a list of values. When lists are constrained, you can only select from the pre-defined list. When lists are non-constrained, you can enter a value that is not in the drop-down list. When an attribute is a multi-valued attribute, AutoVue lets you select multiple values from the drop-down list. In this case, values are separated by a semi-colon (;).

As of AutoVue 20.2, Stamps are disabled for 3D files. Existing stamps that were defined in previous releases of AutoVue can still be viewed but cannot be modified.

Adding a Stamp

To add a Stamp markup entity to a document, do the following:

TASK

1. Enter Markup mode.
2. 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 Stamp dialog appears.
3. From the Choose Stamp box, select the desired Stamp and then click OK.

   ADDITIONAL INFORMATION: If the Enable Resizing check box is selected then the Stamp can be resized in the workspace. If the check box is grayed out, then the Stamp is not resizable and has a set size.

4. In the workspace, click and drag a box to the desired Stamp size.

   STEP RESULT: The Stamp dialog closes and the Stamp appears in workspace and in the Markup Entity Tree.

RESULT:

Depending on how the Stamp is configured, it may appear as a fized size in the workspace or it may be resizable. If it is resizable, you can select the stamp outline and drag to the desired size. If the isotropic option is selected on the stamp, it resizes proportionally in all directions. To resize the stamp non-proportionally, your system administrator must updated the Stamp definition.

NOTE: *Depending the format of the background image, when resizing the Stamp, attributes and/or text may not align properly with the background image. To avoid this issue, Enhanced Metafiles (EMF) should be used as the background image. Refer to the Oracle AutoVue Client/Server Deployment Installation and Configuration Guide for more information.*

If the Stamp is configured to retrieve backend system attributes, then the values of these attributes are displayed on the Stamp in the workspace.

To modify the text font on the Stamp, select the required font and style from the Markup toolbar.

Viewing/Modifying Stamp Attributes

The section describes how to view and modify Stamp attributes.

If your Stamp is configured to read backend attritbutes, then the Stamp entity displays these backend attributes. Reading values from the backend can be done once during Stamp creation (the attribute is configured with the ReadOnce permission by the system administrator) or the backend values can be read when requested (the attribute is configured with the Read permission by the system administrator). The following describes each attribute.

- If an attribute is configured with the Edit permission, then you can modify the value of the attribute on the Stamp.
- If an attribute is configured with the Write permission, then you can save the value on the Stamp to backend system.

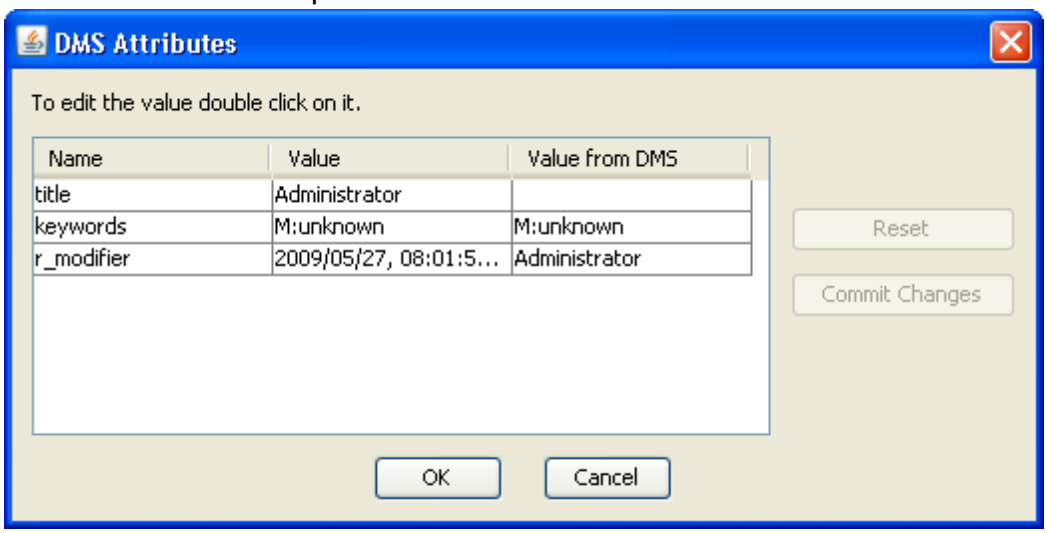
  NOTE: *Provided that you have permissions to update the attribute on the backend system.*

- If an attribute is configured with the Hidden permission, then you cannot see the attribute on the Stamp. However, you can view the attribute in the Edit dialog and when editing the Stamp.

- If an attribute is configured with the current date or user name default values, then these values are displayed on the Stamp (only if no value is retrieved from the backend system)

TASK

1. You can view the attributes of a Stamp by double-clicking the Stamp markup entity. The DMS Attributes dialog appears and lists all attributes of the Stamp.



ADDITIONAL INFORMATION: All attributes tagged as **Hidden** during Stamp design also display in the DMS Attributes dialog.

The dialog 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 either displays the backend DMS/ERP/PLM/UCM system-defined attribute name or the user-defined name.
- The *Value* column displays the value that is currently on the Stamp.
- The *Value from DMS* column displays the value assigned from the backend system.

2. To modify any value of the Stamp, double-click the Value cell corresponding to the attribute.
   - If the attribute does not have Edit permissions, the value cannot be modified.
   - If the attribute is defined as Hidden, then the attribute is displayed in the DMS Attributes dialog. You may edit the value of a Hidden attribute, but it does not display in the workspace.
   - If the attribute is a non-constrained list, then you may either select a value from the list or enter your own value. If the attribute is a constrained list, you may only select from the list.
   - If the attribute is a multi-valued list, then you may select a value from the list. AutoVue appends this value to any pre-existing values using a semi-colon (;) as a separator.

- If an attribute is defined with Hidden permissions, then the value from the DMS column reflects the value read during Stamp creation. It does not reflect the current value in the DMS.

ADDITIONAL INFORMATION: After a change is made to a value cell, you must press **Tab** or **Enter** to finalize the changes.

3. To commit changes to values for backend system attributes, click **Commit Changes**.
4. To reset changes and revert to backend system values, click **Reset**.
5. When you are done modifying the attributes, click **OK** to finalize your changes and to close the DMS Attributes dialog.

RESULT:

The Stamp is updated to reflect any changes you made.

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.
	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

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. Select the snapping modes that you want to use for measuring.
3. To select all snapping modes click **All On**. To clear 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. 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.

   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 theMeasurement 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 theMeasurement 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.

Button	Snap To	Description
	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 **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. Click a point on the drawing to define the starting point.
6. 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.
7. 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 unitin 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 unitin 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.

*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 sticky note on the drawing. 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.

NOTE: *When creating multiple Note entities, it is possible to group the entities. Once the Note entities are grouped, clicking on the group opens the last Note created.*

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: A new markup note entity is created and the Note editor appears.

3. Enter the text that you want in the Note editor.

4. From the **File** menu, select **Note Information.**

   STEP RESULT: The Note Information dialog appears.

5. To change the default font, select **Font** and the type of font.

6. Close the Note editor.

   STEP RESULT: The note symbol appears on the drawing and in the Markup Navigation Tree.

7. Right-click outside the note area to complete the modification.

   ADDITIONAL INFORMATION: To edit a note, double-click on the note.

Nesting Markup Entities

You can add a Note or Hyperlink markup entity as a nested child to other markup entites.

TASK

1. Add a markup entity to a file (for example, a Box entity).
2. From the workspace or Markup Tree, select the Box entity.
3. From the **Markup** menu, select **Note** or **Hyperlink.**

   ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Note**  or **Hyperlink**  .

RESULT:

In the Markup Tree, the selected markup entity appears as a child of the Box (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
Text	Add text to the markup. From the Markup Entity toolbar, you can also click **3D Text**  . See "Adding Text"

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"
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"

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 **Measurement** 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

Snapping modes allow you to select or snap to different entity types on a model. For example, if you select **Vertex**, all vertices are highlighted and a snap box appears when you move the cursor over a vertex.

Snapping modes allow you to snap to the following entities.

NOTE: *The behavior of snapping modes change in regards to the selected measurement option. For example, the behavior of arc center is not the same for distance and minimum distance.*

Button	Description	Behavior
Vertex 	Highlights vertices on the model.	A snap box appears when you hover the mouse pointer over a vertex.

Button	Description	Behavior
Edge (Line) 	Highlights edges on the model.	The edge is highlighted when you hover the mouse pointer. ***NOTE:*** *When measuring Minimum Distance, the finite edge is selected. However, when measuring Distance, the infinite edge is selected.*
Edge Middle 	Highlights edges on the model.	The edge is highlighted and a tooltip appears indicating the middle edge coordinate when you hover the mouse pointer.
Arc 	Highlights arcs on the model.	The arc is highlighted when you hover the mouse pointer.
Arc Center 	Highlights arcs and circles on the model.	The arc is highlighted and a tooltip appears indicating the arc center coordinate of the arc when you hover the mouse pointer. ***NOTE:*** *When measuring Minimum Distance, the arc center is selected. However, when measuring Distance, the infinite arc axis is selected.*
Face (Plane) 	Highlights faces when you move the cursor over a face.	The face is highlighted when you hover the mouse pointer.

Measuring Distance

Use the *Distance* option to measure the distance between two vertices, edges, arc axes, faces or any combination of these entity types.

NOTE: *To revert to the Point to Point distance measurement, set the SHOW_POINTOPOINT_PAGE INI option. Refer to theInstallation and Configuration Guide for more information.*

The following table details how the distance between two entities are measured using snapping modes.

Snapping Mode Entities	Vertex	Line	Arc Center	Plane
Vertex	Distance between two points.	The shortest segment connecting the point and the line.	The shortest segment connecting the point and the arc.	The shortest segment connecting the point and the plane.
Line		Distance between two lines. ***NOTE:*** *The lines must be parallel.*	The line and axis must be parallel. Distance between the line and arc axis.	The distance between the line and the plane. ***NOTE:*** *The line must be parallel to the plane.*
Arc Center			Distance between the axes of the arcs. ***NOTE:*** *Arc planes must be parallel.*	Distance between the arc axis and the plane. ***NOTE:*** *The plane and the arc plane must be perpendicular.*
Plane				Distance between the two planes. ***NOTE:*** *The planes must be parallel.*

TASK

1. From the **Markup** menu, select **Add Entity,Measurement**, and then **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.

   ADDITIONAL INFORMATION: See "3D Snapping Modes" for more information.

   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 modes 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 theMeasurement 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 axes, arc centers, faces or any combination of entity types.

TASK

1. From the **Markup** menu, select **Add Entity**, **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  **Set 1**.
3. 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 displayed.

   ADDITIONAL INFORMATION: See "3D Snapping Modes" for more information.

4. 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.

5. Click **Set 2**.

6. Repeat step 3 and 4.

   STEP RESULT: The model part appears in the list under **Set 2**.

7. From the Measured Min. Distance Units list, select the unit in which you want to measure the distance.

8. 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.

9. Click and drag the value box 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 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.

3. Click the vertex.

   STEP RESULT: The X, Y, an Z coordinates and unit appear a value box entity on the current active markup layer and in the Measurement Entities dialog.

4. 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.

5. 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

1. From the **Markup** menu, select **Add Entity,** and then select **Text**.

   ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **3D Text**  .

2. 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.

3. To insert a text box that snaps to a part of the drawing with a leader line, select one of the snapping modes.

4. Click and drag on the drawing to define the dimensions of the text box.

5. 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 theFont 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 sticky note on the drawing. 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.

While editing a Note entity, you can continue to make selections from the AutoVue toolbar and menubar. Clicking in the workspace saves the last modified state of the Note and closes the Note dialog.

When creating multiple Note entities, it is possible to group the entities. Once the Note entities are grouped, clicking on the group opens the last Note created.

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**  .

   STEP RESULT: The Attach To dialog appears.

2. In the Attach To dialog, click the entity type that you want to attach the note to.
3. Select the entity on the model that you want to attach a note to.

   STEP RESULT: The Note dialog appears.

4. Enter the text that you want in the dialog.
5. From the **File** menu, select **Information.**

   STEP RESULT: The Note Information dialog appears.

6. To change the default font, select **Font** and the type of font.
7. Close the Note dialog.

   STEP RESULT: The note symbol appears on the entity and in the Markup Navigation Tree.

8. Right-click outside the note area to complete the modification.

   ADDITIONAL INFORMATION: To edit a note, double-click on the note.

   ADDITIONAL INFORMATION: Whenever an entity is attached to a 3D model, the anchor point (the point at which the entity is attached) is highlighted by a small square. The square is visible only when the anchor point is visible. This feature allows you to precisely identify the location of the anchor point and whether the associated entities are visible or hidden.

Nesting Markup Entities

You can add a Note or Hyperlink markup entity as a nested child to other markup entites.

TASK

1. Add a markup entity to a file (for example, a Box entity).
2. From the workspace or Markup Tree, select the Box entity.

3. From the **Markup** menu, select **Note** or **Hyperlink.**

   ADDITIONAL INFORMATION: From the Markup Entity toolbar, you can also click **Note**  or **Hyperlink**  .

RESULT:

In the Markup Tree, the selected markup entity appears as a child of the Box (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.**

NOTE: *Double-clicking on a markup entity from the Markup Navigation tree also has the same result as the Go To feature.*

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 rotate markup entities.

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.

Changing Object Order

You can change the order of markup entities in a markup layer. To move an entity forward or backward, do the following

TASK

1. Select an entity from the workspace or Markup Entity tree.
2. From the **Markup** menu, select **Object**, then select **Order**, and then select one of the following options:

Option	Description
To Back	Moves the markup entity behind all the other entities in the markup layer.
To Front	Moves the markup entity in front of all other entities in the markup layer.
Backward	Moves the markup entity backward by one level.
Forward	Moves the markup entity forward by one level.

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**.

Hiding Markup Files

To hide selected a Markup file, do the following:

TASK

1. In the Markup Navigation Tree, right-click the Markup file, then select **Hide.**
2. To undo the hide option, right-click the file name again and select **Show.**

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.

   ADDITIONAL INFORMATION: When selecting markup entities in Markup mode, press Ctrl and drag the box to select all the markup entities within the area.
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 strikeout and underline by selecting their

check boxes. Click **OK** to the 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.
3. 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

1. 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.
2. 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

1. From the Line Thickness list, select **Customize.**
2. In the **Width (Pixels)** field, enter the desired width.
3. 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.

9 Printing

With AutoVue, you can print and preview files. 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"
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:

Print Dialog Section	Option	Description
Printer	Modify	Select a printer and configure print properties, such as paper size, number of copies, and so on.
	Paper size	Displays selected paper size. This option also allows you to modify the paper size.
	Orientation	**Portrait** - Specify if file should be printed portrait. **Landscape** - Specify if the file should be printed landscape. **Auto** - Specify if orientations should be printed as saved in the file.
	Units	From the drop-down list, set the unit for Scaling and Alignment/Offset. The available units are pixels, inches and millimeters.
Scaling	Fit to Page	Scales the image to fit on the output page.
	Factor	Specifies the number of drawing units per specified number of inches.
	Scale	Scales the image according to a percentage. You can select a predefined scaling factor or manually enter a custom scaling factor. Decimal places are accepted.
Alignment/Offset		Specify where the drawing will appear on the printout. You can define custom alignment in inches by entering a X and Y value. ***NOTE:*** *This feature is disabled for spreadsheet formats.*
Document Pages		Specify the document pages to print: **All**, **Current** or **Range**.

Print Dialog Section	Option	Description
Page Area		Select the page area to print.
	Extents	Print the extent of the document.
	Displayed	Print the area displayed in the View window. Option is always enabled.
	Limits	Prints the file limits instead of extents. Option is disabled when **Current** is selected for Document Pages. ***NOTE:*** *This option is only available for AutoCAD drawings that have the limits defined.*
	Selected	This option is grayed out by default. To enable, click **Select** and then select a print range. The **Selected** option is now enabled.
Native settings (from file)		This option only applies to Excel files. When selected, AutoVue uses Excel's print settings that are saved in the excel document. The following print settings are supported: Page Orientation, Scaling, Paper Size, Print Areas, Page Breaks, Row and Column headers, Grid lines, Left/Right/Top/Bottom margins.
Force to Black		Force all colors to grayscale.
Output a single page		Limits output to a single page when the scaling options selected causes a single page to span over several pages. ***NOTE:*** *This option is only enabled when **Scale** of **Factor** is selected in the Scaling section.*
Skip Blank Pages		This option omits blank pages when printing.
Print Notes		When in Markup mode, select this option to print the Note markups of the current file. ***NOTE:*** *This option is only available when in Markup mode.*
Print One Note Per Page		When in Markup mode, select this option to print one Note markup per page of the current file. ***NOTE:*** *This option is only available when in Markup mode.*
Print Row Headers		Select this option to print row headers. Option is only enabled for spreadsheets and archive files.

Print Dialog Section	Option	Description
Print Column Headers		Select this option to print column headers. Option is only enabled for spreadsheets and archive files.
Partial Preview		Display a view of the printer page, highlighting the printable area.

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

From the **Margins** tab of the Print Properties dialog, 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	Set the acceptable minimum margins for the selected printer.
Units	Specify the unit for the margins.
Override Printing Minimum Margins	Select this option to override the predefined minimum print 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 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.
4. To define margins that are less the minimum acceptable for the selected printer, select **Override Printing Minimum Margins**.
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 Previewdialog appears.

8. Click **OK** to print.

RESULT: The Print dialog appears indicating the document is printing.

Header/Footer

From the **Headers/Footers** tab of the Print Properties dialog, 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 %%.*

You can also print system variables in headers and footers. Select a variable from the Insert Variable list. Some **Insert Variable** options are: user.name, browser and java.home.

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

From the **Watermark** tab of the Print Properties dialog, 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 diagonal, horizontal or vertical orientation for the 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 %%.*

You can also print system variables in watermarks. Select a variable from the **Insert Variable** list. Some **Insert Variable** options are: user.name, browser and java.home.

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 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.

   ADDITIONAL INFORMATION: For example, if you select **%n: Total document pages**, the total number of pages for the selected file will appear in the Watermark. You can insert more than one code.

5. To insert a system variable, select a variable from the Insert Variable list.

   ADDITIONAL INFORMATION: For example, if you select **browser.version**, the version of the browser that the file is displayed in will appear in the Watermark.

   ADDITIONAL INFORMATION: You can insert more than one system variable.

6. To set the orientation of the watermark, click **Diagonal, Horizontal or Vertical**.
7. To change the Font, click **Set Font**.The Font dialog appears.
   a From the Font list, select the type of font.
   b From the Size list, select the size of font.
8. In the Print Properties dialog, configure other print properties.

9. To view a partial view of the file, click **Partial Preview**.

   STEP RESULT: The Partial Preview dialog appears.

10. Click **OK** to print.

RESULT: The Print dialog appears indicating the document is printing.

Assigning Pen Settings

From the **Pen Settings** tab of the Print Properties dialog, 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 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, 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 appears.

9. Click **OK** to print.

   STEP RESULT: The Print dialog appears indicating the document is printing.

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.**

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

   STEP RESULT: The Print Properties dialog 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**.

   STEP RESULT: The Partial Print Preview dialog appears.
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 3. 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**.

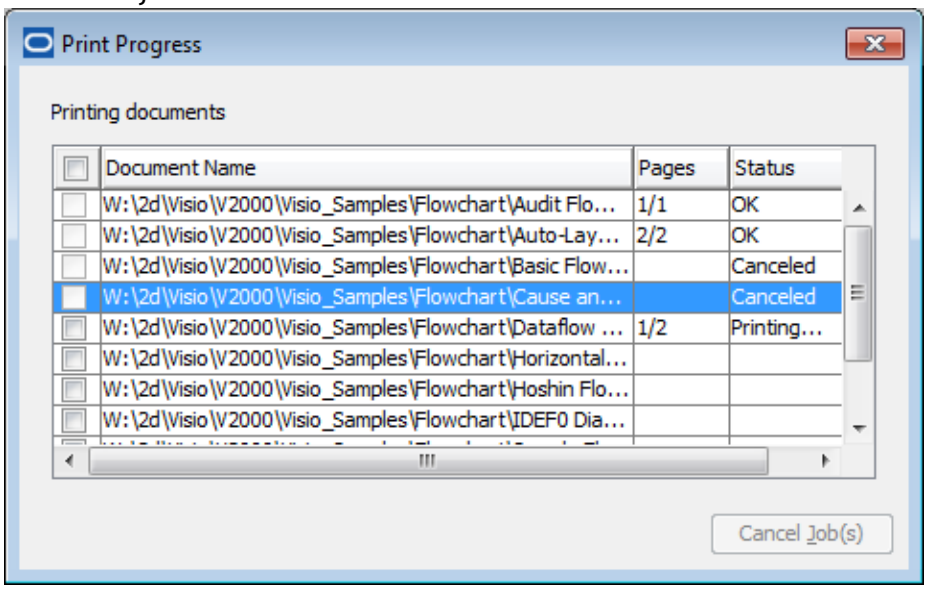
4. Click **Print** in the Batch Print dialog.

   STEP RESULT: The Print Properties dialog appears.

5. Configure the print properties.

6. Click **OK** to print.

   STEP RESULT: The Print Progress dialog appears. It displays document name, number of pages and printing status (OK, Canceled or Printing). When the print job completes, the dialog remains open and displays the print status of each job.



7. To cancel a print job, select a check box to the left of the file and click **Cancel Job(s)**.

   ADDITIONAL INFORMATION: Clicking the dialog close icon minimizes the Batch Pring dialog to the lower left corner of the AutoVue status bar. To restore the Batch Print dialog, click the Batch Print icon on the AutoVue status bar.

10 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

From the **File** menu, select **Convert** to open the Convert dialog. Depending on the conversion type being performed, the available options in the **Convert** dialog may vary. These options are:

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.
Convert to Format	A list of all the possible types of output file formats currently available for conversion. The available formats are: PDF TIFF Windows Bitmap ***NOTE:*** *Conversion to PDF is not supported when the AutoVue server is running on Linux platforms.*
Output	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. If writing onto an existing file, a warning message appears, asking if you want to overwrite the existing file.

Option	Description
Sub-Format	The **Sub-Format**list appears when you select **HP Laserjet Print** or **TIFF** from the Convert to Format list. Select a sub-format from the list.
Convert Region	The area of the file to be converted. The available options are: **Display** - The area displayed in the workspace. For example, if you zoomed in on a particular region of the file, the zoomed portion of the file is converted. **Extent** - Refers to the entire extents of the file. **Selected** - The selected entity is converted.
Convert Pages	The pages to be converted. The available options are: **All** - convert all pages **Current** - convert the current page **Range** - convert the pages indicated in the range
Size	You can set the size and units (pixels, inches and millimeters) of the output file. With raster files the units are preset as pixels: **X** indicates the number of horizontal pixels **Y** is the number of vertical pixels for the current active file contents. At times, AutoVue will preset **X** and **Y** to match the specifications of the selected conversion file format. Other times, **X** and **Y** will be available and may be changed according to your preferences. Your selection here will not affect the current display but will affect the conversion file's resolution.
Scale	Set the scale percentage of the converted page.
DPI	You may set the dots per inch (DPI) for the converted page.

Converting a 3D Model

TASK

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

   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 convert the file to, then enter a file name.

3. 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 formatyou choose.

4. Select the output and convert options that you want to set for the file.
5. Click **OK**.

RESULT:

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

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.

11 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.

EDA Term	Definition
Component library	A computer data file that contains the footprint patterns for a number of components.
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.

EDA Term	Definition
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.
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.

EDA Term	Definition
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.
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.

EDA Term	Definition
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 underlaying 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.
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.

EDA Term	Definition
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.
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.

EDA Term	Definition
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.

12 Feedback

If you have any questions or require support for AutoVue please contact your system administrator. Some customization and maintenance must be done on the server and cannot be implemented on the client machine. If the administrator is unable to resolve the issue, please contact Oracle Corp.

General Inquiries

Web Site:	http://www.oracle.com/us/products/applications/autoVue/index.html
Blog:	https://blogs.oracle.com/enterprisevisualization/

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