AutoVue
Web Edition
User's Manual

Support Information

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 side and cannot be implemented on the client machine. If the administrator is unable to resolve the issue, please contact Cimmetry Systems, Corp.

Contact Information

If at any time you have questions or concerns regarding AutoVue, call or e-mail us.

General inquiries

Telephone: +1 514 735-3219
Fax: +1 514 735-6440
E-mail: info@cimmetry.com
Web Site: http://www.cimmetry.com

Sales inquiries

Telephone: +1 514 735-3219 or 1 800 361-1904
Fax: +1 514 735-6440
E-mail: sales@cimmetry.com

Customer Support

Telephone: +1 514 735-9941
Web Site: http://www.cimmetry.com/support

CONTENTS

Introduction

Cimmetry Systems Corp. has developed a family of products designed to meet all levels of business and engineering needs. Collectively, our products provide viewing and Markup capabilities, and support the visualization of 3D models, 2D engineering and office formats.

AutoVue

AutoVue is a viewing and Markup application that has been developed for both business and technical users.

AutoVue is capable of displaying over 450 different file formats without the authoring application. Supported file types include text, database, graphic and spreadsheet. Even contents of archive files can viewed in AutoVue. Moreover, when you view a file you do not need to specify the file type. AutoVue automatically detects the type of file you are requesting to view — even files with false extensions!

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**. However, when you display a file with its Markups, the Markups appear to be part of it.

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 or add a symbol 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.

Viewing and Exporting File Versions Information

From the **Help About** dialog, you can view the version, build number and date of the decoder dll files, as well as export the files into a text file.

Viewing File Versions

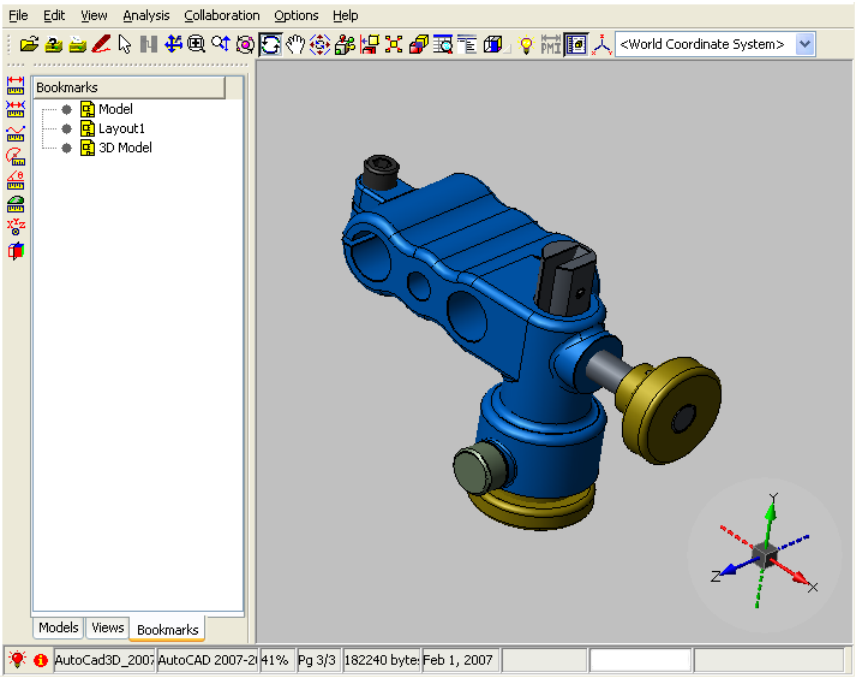
1 Select **Help > About** from the AutoVue main menu.
The **About** dialog appears.
2 Click **Version Info...**.
The **File Versions** dialog appears.
3 When you are finished viewing, click **Close**.

Exporting File Versions

1 Select **Help > About** from the AutoVue main menu.
The **About** dialog appears.
2 Click **Version Info...**.
The **File Versions** dialog appears.
3 Click **Export...**.
The **Export** dialog appears.
4 Navigate and select the directory you want to export the file to.
5 Enter a **File name** with the extension **.txt**.
Note: The file can only be exported to text file.
6 Click **Save**.
The markup file is exported to the selected directory.

AutoVue Basics

This section introduces you to the basics of working with AutoVue's graphical user interface.



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

The **Toolbar** buttons offer easy access to the options presented in the drop-down menus found in the **Menu** bar.

Status Bar

In both **View** mode and **Markup** mode, there is a **Status** bar 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.

In addition to the fields, there are two markers that may be present on the Status bar.

The red light bulb  indicates that the current active file has associated Markups. While in **View** mode, click the red light bulb to view the **Markup Files** dialog, then select a Markup file or group of Markup files to open. By opening a Markup file, Markup mode is automatically launched.

A red circle with an "i"  indicates that some main resources required to properly read the current active file are not available. To identify the missing resources, click the red circle to display the **File 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. The **Quick Menu** options available depend on the mode you are in and what is selected prior to right-clicking.

Opening Files with AutoVue

How to open a file depends on the current active mode in AutoVue. In **View** mode, you can open base files. In **Markup** mode, you can open a Markup file for a base file.

With AutoVue, you can open a file from the current active directory or from another directory, or manually specify the file to open by entering the URL.

Opening a Local File

1 Select **File > Open Local File** from the AutoVue main menu.
The **File Open** dialog appears.
2 Browse to locate the file that you want to open.
3 Click **Open**.
The file appears in the AutoVue workspace.

Opening a URL

You can open a file by specifying the URL.

1 Select **File > Open URL** from the AutoVue main menu.
The **File Open** dialog appears.
2 In the **File Name** type the **URL http** or **ftp** path or click **Browse** to locate the file that you want to open.
3 Click **OK**.
The file appears in the AutoVue workspace.

Opening a File from the Server

1 Select **File > Open from Server** from the AutoVue main menu.
The **File Open** dialog appears.
Note The **Open from Server** option is available in the **File** menu only if you have access to files on the server. Refer to the ***Installation and Administration Manual*** for information on setting up server directories using VueServer.ini.
2 Browse the directory to locate the file that you want to open.
3 Click **Open**.
The file appears in the AutoVue workspace.

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 archive file, the file must be accessible for AutoVue in a format other than archive.

Viewing File Properties

You can access the file properties from the **File** menu. 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	Description
File	Information specific to the current active file, such as filename, file size, date of creation, file type and x, y & z dimensions.
Resource Information	Resources specific to the current active file, such as text font, shape file, line style and external reference files.
Native	Custom properties for file types, such as last person who saved the file, signature verification and author comments.

Note The file properties can vary depending on the file format viewed.

1 Select **File > Properties** from the AutoVue main menu.
The **Properties** dialog appears.
2 To view file properties, click the **File Properties** tab.
To view resource information, click the **Resource Information** tab.
To view native properties, click the **Native Properties** tab.
3 Click **OK** to close the dialog.

Working With 2D Files

AutoVue provides easy access to entity information and displays color-coded comparative data for 2D drawings. You can instantly manipulate how the current active file is displayed. You can also 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.

Note You can not perform a text search on raster files.

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

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.

1 Select **Edit > Search** from the AutoVue main menu.
The **Find** dialog appears.
2 In the **Find What** text box, enter the word or phrase that you want to find.
Note If you are searching in a vector file, select a text string from the list.
3 Customize your search.
4 Click **Find Next**.
AutoVue highlights the text and zooms into the text area.
5 Click **Close** to close the **Find** dialog.
Note To repeat the last search criteria, select **Edit > Repeat Search** from the AutoVue main menu.

Manipulating 2D Views

With the **View** options you can instantly manipulate how the current active file is displayed. You can rotate a file's orientation counterclockwise by 90, 180 or 270 degrees, flip a file's orientation horizontally, vertically or both simultaneously.

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.

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

Menu	Option	Description
Zoom	In	Zoom in by a factor of 2. From the toobar, click the  button.
	Out	Zoom out by a factor of 2. From the toobar, click the  button.
	Full Resolution	Display the file at full resolution. From the toobar, click the  button.
	Fit	Adjust the contents of a file along the horizontal or vertical axis to fully display within the current window. **Fit Both** - AutoVue finds the best fit for the current file with respect to both its vertical and horizontal dimensions. **Note** This option is also known as **Zoom Fit** when right-clicking in the workspace. **Fit 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.

Menu	Option	Description
		Fit Horizontal - Fit the image horizontally in the active window. The vertical dimensions of the image are zoomed proportionally but may be too large or small for the window. **Note** This option is also know as **Zoom Page Width** when right-clicking in the workspace.
	Previous	Revert to the previous selected zoom option.
Note The **Rotate** option is disabled for text-based documents and spreadsheets.		
Rotate	0	Set the file to its original position. From the toobar, click the  button.
	90	Rotate the file 90 degrees anti-clockwise. From the toobar, click the  button.
	180	Rotate the file 180 degrees anti-clockwise. From the toobar, click the  button.
	270	Rotate the file 270 degrees anti-clockwise. From the toobar, click the  button.
Note When in **Markup** mode, you can specify if the rotate is to be applied to the Markups only or to the base file and Markups.		
Note The **Flip** option is disabled for text-based documents and spreadsheets.		
Flip	None	Reset the drawing to its original position.
	Horizontal Axis	Flip the drawing on its horizontal axis.
	Vertical Axis	Flip the drawing on its vertical axis.
	Both Axes	Flip the drawing on its vertical and horizontal axes.
Note When in **Markup** mode, you can specify if the flip is to be applied to the Markups only or to the base file and Markups.		

Menu	Option	Description
Note The **Image** option is used for black-and-white (monochrome) raster files.		
Image	Contrast	Light, normal, dark and darkest.
	Anti-alias	Enhance the details of monochrome raster files. **Note** Also known as “Scale to Grey”.
	Invert	Reverse the background and foreground colors.
Page	Next Page	Go to the next page of a multi-page file. From the toobar, click the  button.
	Previous Page	Go to the previous page of a multi-page file. From the toobar, click the  button.
	Page Number	Go to the specified page of a multi-page file. From the toobar, click the  button.
Special View Modes	Birds Eye	Display a close-up view of a particular area of a file while maintaining a view of the entire file. From the toobar, click the  button. **Note** The **Birds Eye** option is disabled for text-based documents and spreadsheets. ***See Using Bird’s Eye View***
	Magnify Glass	Magnify an area of the file that is specified by the cursor location. From the toobar, click the  button. ***See Using the Magnifying Glass***
	Magnify Window	Magnify a selected area of a file and displays it in the Magnify Window. From the toobar, click the  button. ***See Using the Magnifying Window***
	Pan	Click and drag to move the drawing. To exit right-click.

Menu	Option	Description
Show Model Tree		Show/hide the **Model** tree of the current active file.

Using Birds Eye View

The **Birds Eye** view option allows a close-up view of a particular area of a file while maintaining a full view of the file. The Birds Eye window displays a miniature version of the file. A movable box frame on top of the miniature indicates the area of the file displayed in the AutoVue workspace.

Note The **Birds Eye** option is disabled for text-based documents and spreadsheets.

1 Select **View > Special View Modes > Birds Eye** from the AutoVue main menu.
The **Bird's Eye** window appears displaying a full view of the file.
2 To view the changes in "real time", select **Options > Dynamic** from the **Birds Eye** menu.
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.
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.
Note If you perform a zoom function in the AutoVue workspace, the area displayed in the workspace is highlighted by the frame box in the **Birds Eye** window.
4 Select **Bird's Eye > Exit** to close the **Bird's Eye** dialog.
The last view performed remains in the AutoVue workspace.

Using the Magnify Glass

The **Magnify Glass** view option zooms an area specified by the position of the Magnify Glass. You can view the details of a selected area of a file while maintaining a full view of the file.

1 Select **View > Special View Modes > Magnify Glass** from the AutoVue main menu.

Note You can also click the Magnify Glass button  on the AutoVue toolbar.

2 Move the cursor to the area that you want to magnify.
3 Click and hold the left mouse button.
The area is magnified in the Magnify Glass.
Note To view different areas of the file, click and drag the mouse.
4 Right-click to exit **Magnify Glass**.

Using the Magnify Window

The **Magnify Window** option zooms an area specified by the position of the cursor. You can view the details of a select portion of the displayed file while maintaining the display of the full file.

1 Select **View > Special View Modes > Magnify Window** from the AutoVue main menu.
The **Magnify Window** appears.

Note You can also click the Magnify Window button  on the AutoVue toolbar.
2 Move the cursor to the area that you want to magnify in the current active file.
3 Click once.
The area appears magnified in the **Magnify Window**.
4 Right-click to exit the **Magnify Window**.

Conversion

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

Conversion Options

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

Option	Description
Color Depth	Select an option from the drop-down 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 drop-down list of all the possible types of output file formats currently available for conversion. The available formats are: • CALS GP4 • Encapsulated Postscript (Raster) • HP Laserjet Printer (HLP) • PCX Bitmap • PDF • Run Length RLC File • TIFF • Windows Bitmap • Compuserve GIF
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. **Note** If writing onto an existing file, a warning message appears, asking if you want to overwrite the existing file.
Sub-Format	The **Sub-Format** drop-down list appears when you selected **HP Laserjet Print, TIFF**, or **Compuserve GIF** from the **Convert to Format** drop-down list. Specify the variety of convert-to-format by selecting a sub-format.

Option	Description
Convert Region	The area of the file to be converted. The available options are: • **Display** - refers to the image to fit on the output page. For example, if you zoomed in on a particular region of the file, the zoomed portion of the file is converted. • **Extents** - refers to the entire extents of the file.
Convert Pages	The number of pages to be converted. The available options are: • **All** - converts all pages. • **Current** - converts the current page • **Range** - converts the pages indicated in the range
X and Y	Choose from pixels, inches and millimeters for the units. **Note** 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.

PDF

With AutoVue it is possible to convert Office, 2D and EDA formats to PDF. When converted from **Markup Mode**, markups are "burned" onto the PDF. 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 converting, 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** drop-down list or you can customize page sizes by configuring the Initialization file. For more information, ***see System Administration 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

1 Select **File > Convert** from the AutoVue main menu.
The **Convert** dialog appears.
Note The **Input** area varies according to the type of file. A word-processing file displays the file format; a raster file displays the file format and size. Both vector and database files display the size, file type and dimensions.

2 In the **Save As** text box, enter the path and filename or click **Browse** to locate the directory where you want to save the converted file.
Note If writing on to an existing file, the contents of the output file will be overwritten.

3 Select the convert options that you want to set for the file.
See Conversion Options

4 Click **OK**.
The file is converted and appears in the specified directory.
Note 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.

Note This option only applies to vector files.

1 Select **File > Convert** from the AutoVue main menu.
The **Convert** dialog appears.
2 Click **Pens.**
The **Pen Settings** dialog appears.
3 In the **From/To**, select the **Color Index** for which you want to modify the thickness.
Note To select more than one **Color Index**, click the **Shift** or **Ctrl** key while selecting.
4 From the **Units** drop-down list, select the unit in which you want to set the thickness.
5 Click **Thickness**.
The **Modify Pen Thickness** dialog appears.
6 Enter a thickness.
7 Click **OK**.
The new **Thickness** appears beside the selected **Color Index**.
8 To save the changes that you made, click **Save As**.
The **Save As** dialog appears.
9 Enter a **Name** for the new pen settings.
10 Click **OK**.
The new pen settings are saved and appear in the **Current Pen Settings** list.
Note To modify an existing **Pen Settings**, select the pen settings from the **Current Pen Settings** drop-down list, make the changes, then click **Save**.
11 Click **OK** to close the **Pen Settings** dialog.

Measuring in 2D non-vector Files

AutoVue provides the ability to perform measurements in 2D files. 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”.

AutoVue provides several measure options that you can choose from.You can access the measure options from the **Analysis > Measure** menu. The options are:

Name	Description
Angle	Measure the angle between selected points. Click the Angle button  in the toolbar.
Arc	Measure an arc. Click the Arc button  in the toolbar.
Area	Measure a selected area. Click the Area button  in the toolbar.
Distance	Measure the distance between two points. Click the Distance button  in the toolbar.

Measuring Distance

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the AutoVue toolbar.

2 Click the **Distance** tab.

3 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure the distance.

Note If you want to measure the distance along a path, select **Cumulative**.

4 Click a point on the drawing to define the starting point.

5 Click another point on the drawing to define the end point.

Note If you selected **Cumulative**, continue clicking points along the path that you want to measure.

6 Right-click to complete the measurement.
The points are joined by a line. The measured distance, Delta-X and Delta-Y appear in the **Measurement** dialog.

Note Click **Reset** to take another measurement.

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

Calibrating Distance

1 Measure distance between two points or measure cumulative distance.
See Measuring Distance

2 In the **Measurement** dialog, click **Calibrate**.
The **Distance Calibration** dialog appears displaying the measured distance.

3 From the **Measured Dist.** drop-down list, select the unit to which you want to calibrate the distance.

4 Click **Calibrate to** and enter a value if you want to calibrate the distance to a value.
Click **Set Factor** and enter a value if you want to calibrate the distance by a factor.

5 Click **OK**.
The calibration result appears in the **Measurement** dialog.

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

Measuring Area

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

1 Select **Analysis > Measure** from the AutoVue main menu.The **Measurement** dialog appears.

Note You can also click the Area button  on the AutoVue toolbar.

2 Click the **Area** tab.

3 From the **Measured Area Units** drop-down list, select the unit in which you want to measure the area.
From the **Perimeter Units** drop-down list, select the unit in which you want to measure the perimeter.
Note To cumulate a **Net Area Result** of different areas, select **Add** in the **Measurement** dialog. To subtract an area from the **Net Area Result**, select **Subtract**. Select **Clear** to clear the **Net Area Result**.

4 Click a point on the drawing to define the starting point.

5 Continue clicking points on the drawing to define the area you want to measure.
Each point is joined by a line. The area and perimeter measurements appear in the **Measurement** dialog.

6 Right-click to complete the measurement.
Note Click **Reset** to take another measurement.

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

Measuring an Angle

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Angle button  on the AutoVue toolbar.

2 Click the **Angle** tab.
3 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.
4 Click points on the drawing to define the angle you want to measure.
Angle arms appear with an arc connecting them. The angle measurement appears in the **Measurement** dialog.
Note Click **Reset** to take another measurement.
5 Click **Close** to close the **Measurement** dialog.

Measuring an Arc

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the AutoVue toolbar.

2 Click the **Arc** tab.
3 From the **Arc Info** drop-down list, select the unit in which you want to measure the arc.
4 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.
5 Click points on the drawing to define the arc.
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.
Note Click **Reset** to take another measurement.
6 Click **Close** to close the **Measurement** dialog.

Calibrating an Arc

1 Measure an arc in the drawing.
See Measuring an Arc

2 In the **Measurement** dialog, click **Calibrate**.
The **Radius Calibration** dialog appears displaying the measured distance.
3 From the **Measured Rad.** drop-down list, select the unit to which you want to calibrate the distance.
4 Click **Calibrate to** and enter a value if you want to calibrate to a value.
Click **Set Factor** and enter a value if you want to calibrate by a factor.
5 Click **OK**.
The calibration result appear in the **Measurement** dialog.
6 Click **Close** to close the **Measurement** dialog.

Working with 2D Vector Files

In addition to all 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 and blocks or external reference files that are located outside the file.

Manipulating 2D Vector Views

With the **View** options you can instantly manipulate how the current active file is displayed. You can rotate a file's orientation counterclock-wise by 90, 180 or 270 degrees, flip a file's orientation horizontally, vertically or both simultaneously.

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. For more information on how to manipulate views, ***see Manipulating 2D Views***.

For 2D vector files, there are additional **View** options:

Option	Description
Views	Select and display different views of a drawing. From the toobar, click the  button. ***See Selecting Views***

Option	Description
Layers	Select and display different layers of a drawing. From the toobar, click the  button. ***See Layers***
Blocks	Select and display a block from a drawing. From the toobar, click the  button. ***See Blocks***

Drawing Information

Note This feature is currently not supported with AutoVue Server running on Unix Platforms.

The **Drawing Information** option is available with AutoCAD and MicroStation drawings, and is accessed from **Analysis** of the AutoVue main menu in both **View** and **Markup** modes. The **Drawing Information** options available are: **Select Single Entity**, **List Tags/Attributes** and **Entity Information**.

Viewing Details of a Single Entity

Note This feature is currently not supported with AutoVue Server running on Unix Platforms.

1 Select **Analysis > Drawing Information > Select Single Entity** from the AutoVue main menu.

2 Click the entity for which you want to view information.
The **Get Entity Info** dialog appears displaying the information for the selected entity.
The **XData** button appears if additional information exists for that entity. Click **XData** to view the information.
Note If you did not select an entity, a message appears informing you that no entities were found and prompts you to select again.

3 Click **OK** to close the **Get Entity Info** dialog.

Viewing information for a Set of Entities

Note This feature is currently not supported with AutoVue Server running on Unix Platforms.

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

1 Select **Analysis > Drawing Information > Entity Information** from the AutoVue main menu.
2 Click and drag to draw a box around an area for which you want to view entity information for the list of entities contained inside the area.
The **List Entities** dialog appears displaying the information of all the selected entities.
Note 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.
3 Click **OK** to close the **List Entities** dialog.

Viewing Tags/Attributes

Note This feature is currently not supported with AutoVue Server running on Unix Platforms.

1 Select **Analysis > Drawing Information > List Tags/Attributes** from the AutoVue main menu.
2 Click in the area of the file for which you want to view information on block attributes and tags.
The **Block Attributes** dialog appears displaying the attributes/tags for the selected entity.
Note 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.
3 Click **OK** to close the **Block Attributes** dialog.

Viewing 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 or Xrefs are located outside the file.

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

Note For AutoVue to locate a resource file, the **XREFPATHS** must point to the directory or directories where the resource files exist. For more information, ***see System Administration Guide***.

1 Select **View > XRefs** from the AutoVue main menu.
The **Select the External References to display** dialog appears listing the external reference files associated with the current active file.
2 Select the checkbox beside the **XRefs** that you want to display.
Clear the checkbox beside the **XRefs** that you want to hide.
3 Click **OK**.
The selected **XRefs** are displayed.

Displaying Details About Missing XRefs

If a file has missing resources, a red circle with an "i" appears to the left of the AutoVue status bar.

1 Click the Resource icon on the AutoVue status bar.
The **Properties** dialog appears.
Note You can also select **File > Properties** from the AutoVue main menu.
2 Click the **Resource Information** tab to display details about missing resource information required to properly display this file.

A green check mark indicates the resources that AutoVue is able to access.

A red indicates the external reference files or components that are not accessible.
3 Click **OK** to close the **Properties** dialog.

Layers

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

1 Select **View > Layers** from the AutoVue main menu.
The **Select the layers to display** dialog appears listing the layers and layer visibility for the current active file.

Note You can also click the Layers button on the AutoVue toolbar.
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 Select the checkbox beside the layers that you want to set visible.
Clear the checkbox beside the layers that you want to hide.
4 Click **OK**.
The selected layers are displayed.

Blocks

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

1 Select **View > Blocks** from the AutoVue main menu.
The **Select a block to display** dialog appears listing the blocks for the current active file.

Note You can also click the Block button  on the AutoVue toolbar.

2 From the drop-down list, select the block that you want to display.
3 Click **OK**.
The selected block is displayed.

Selecting Views

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

1 Select **View > Views** from the AutoVue main menu.
The **Select a named view** dialog appears.

Note You can also click the View button  on the AutoVue toolbar.

2 From the drop-down list, select the view that you want to display.
3 Click **OK**.
The selected view is displayed.

Specifying a View Point

The **View Point** option allow you to render a drawing from a selected viewpoint. The default view point is the one used to create the file.

1 Select **View > View Point** from the AutoVue main menu.
The **View Point** dialog appears.
2 Enter the **X**, **Y** and **Z** coordinates for the viewpoint with which you want to render the drawing.
3 Click **OK**.
The drawing is displayed from the selected viewpoint.

Comparing 2D Files

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

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

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

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

1 View the base file in AutoVue.
2 Select **Analysis > Compare** from the AutoVue main menu.
The **File Open** dialog appears.
3 Enter the **File Name** or click **Browse** to select the file that you want to compare with the current active file.
4 Click **OK**.
AutoVue displays three windows, the first displaying the original file, the second displaying the compare file and the third displaying the comparison results.
Note If you apply a change from the **View** menu, all three windows display the synchronized change.
5 To access the **Compare** options, right-click in any of the windows.
A pop-up menu appears displaying the **Compare** options.
Note 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.
6 To exit **Compare** mode, select **File > Exit Compare Mode** from the AutoVue main menu.
The original file appears in the workspace.
Note You can also exit **Compare** mode by right-clicking in any of the windows and selecting **Exit Compare Mode** from the pop-up menu.

See Also Specifying Scale and Offset for a Compare File

Specifying Scale and Offset for a Compare File

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

Note **XOffset** and **YOffset** are relative to the base drawing and all options are displayed at their current values.

1 In Compare mode, select **View > Scale and Offset** from the AutoVue Main window.
The **Scale and Offset** dialog appears.
2 In the **Scale and Offset** dialog, enter the required values for the **Scale Options: XOffset, YOffset** and **Scale**.
3 Click **OK**.
The scaling/offset modifications are applied to the file in the second window.

Overlays

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.

Adding an Overlay

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

1 View the file that you want to use as the base file for the overlay.
2 Select **Analysis > Overlays > Select** from AutoVue main menu.
The **File Open** dialog appears.
3 Enter the **File Name** or click **Browse** to select the file you want to overlay.
4 Click **OK**.
The base file is displayed with the selected overlay file on top of it.
Note To lay multiple files over one base file, repeat steps 2 to 4 using the same base file. Overlays are added one at a time.

Modifying an Overlay

1 Select **Analysis > Overlays > Modify** from AutoVue main menu.
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.

- 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.
  Note As you select the point to define the position of the upper right corner, you can resize the destination box.
- Click **Scale** if you want to resize the overlay.
  Enter the **XOffset** and **YOffset** coordinates and/or the **Scale** factor.
  Note XOffset and **YOffset** are relative to the base drawing and all options are displayed at their current values.
- 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.
  Note The overlay's size is scaled to accommodate the origin and destination points you defined.

4 Click **OK**.
The changes are applied to the selected overlay.
Note To modify other overlays, repeat steps 2 to 4.

Removing an Overlay

1 Select **Analysis > Overlays > Modify** from AutoVue main menu.
The **Modify Overlay** dialog appears.
2 Select the Overlay that you want to remove.
3 Click **Remove**.
A confirmation dialog appears.
4 Click **Yes**.
The overlay is removed from the list in the **Modify Overlay** dialog and from the display.

Measuring in 2D Vector 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, you can only "free snap".

You can access the measure options from the **Analysis > Measure** menu. These options are:

Name	Description
Angle	Measure the angle between selected points. From the toobar, click the  button.
Arc	Measure an arc entity. From the toobar, click the  button.
Area	Measure selected area. From the toobar, click the  button.
Distance	Measure the distance between two points. From the toobar, click the  button.

2D Vector Snapping Modes

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

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

Button	Snap to	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near a linear component's end point.
	Mid-point	Geometric snap mode where a snap box appears when moving the cursor near the halfway point of a linear component.

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

Measuring Distance

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the AutoVue toolbar.

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 clear all **Snapping Modes** click **All Off.**
See 2D Vector Snapping Modes

4 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure the distance.

Note 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.

Note If you selected **Cumulative**, continue clicking points along the path that you want to measure.

7 Right-click to complete the measurement.
The points are joined by a line. The measured distance, Delta-X and Delta-Y appear in the **Measurement** dialog.

Note Click **Reset** to take another measurement.

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

Calibrating Distance

1 Measure distance between two points or measure cumulative distance.
See Measuring Distance

2 In the **Measurement** dialog, click **Calibrate**.

The **Distance Calibration** dialog appears displaying the measured distance.

3 From the **Measured Dist.** drop-down list, select the unit to which you want to calibrate the distance.

4 Click **Calibrate to** and enter a value if you want to calibrate the distance to a value.
Click **Set Factor** and enter a value if you want to calibrate the distance by a factor.

5 Click **OK**.
The calibration result appears in the **Measurement** dialog.

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

Measuring Area

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

1 Select **Analysis > Measure** from the AutoVue main menu.The **Measurement** dialog appears.

Note You can also click the Area button  on the AutoVue toolbar.

2 Click the **Area** tab.

3 Select **Between Points** if you want to measure the area between points on a drawing. **Snapping Modes** are enabled.

4 Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**
See 2D Vector Snapping Modes

5 Select **Shape** if you want to measure the area of a predefined shape on the drawing. **Snapping Modes** are disabled.

6 From the **Measured Area Units** drop-down list, select the unit in which you want to measure the area.

7 From the **Perimeter Units** drop-down list, select the unit in which you want to measure the perimeter.

8 To cumulate a **Net Area Result** of different areas, click **Add**.
To subtract an area from the **Net Area Result**, click **Subtract**.
Select **Clear** to clear the **Net Area Result**.

9 If you selected **Between Points**, click points on the drawing to define the area.
Each point is joined by a line. The area and perimeter measurements appear in the **Measurement** dialog.

10 Right-click to complete the measurement.

11 If you selected **Shape**, click the edge of the predefined shape.
The shape is highlighted and the area and perimeter measurements appear in the **Measurement** dialog.
Note Click **Reset** to take another measurement.

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

Measuring an Angle

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Angle button  on the AutoVue toolbar.

2 Click the **Angle** tab.

3 Select **From 3 Points** if you want to measure the angle between three points. **Snapping Modes** are enabled.

4 Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**
See 2D Vector Snapping Modes

5 Select **Between 2 lines** if you want to measure the angle between two lines. **Snapping Modes** are disabled.

6 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.

7 If you selected **From 3 Points**, click three points on the drawing to define the angle.

8 If you selected **Between 2 Lines**, click two lines on the drawing to define the angle.
Angle arms appear with an arc connecting them. The angle measurement appears in the **Measurement** dialog.
Note Click **Reset** to take another measurement.

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

Measuring an Arc

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the AutoVue toolbar.

2 Click the **Arc** tab.

3 Select **From 3 Points** if you want to measure the arc between three points. **Snapping Modes** are enabled.
Click the **Snapping Modes** that you want to select as the points for the measurement.
See 2D Vector Snapping Modes
Note To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**

4 Select **Arc Entity** if you want to measure a predefined arc. **Snapping Modes** are disabled.

5 From the **Arc Info** drop-down list, select the unit in which you want to measure the distance.

6 From the **Measure Angle Units** drop-down list, select the unit in which you want to measure the angle.

7 If you selected **From 3 Points**, click three points on the drawing to define the arc.
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.
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.
Note Click **Reset** to take another measurement.

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

Calibrating an Arc

1 Measure an arc in the drawing.
See Measuring an Arc

2 In the **Measurement** dialog, click **Calibrate**.
The **Radius Calibration** dialog appears displaying the measured distance.

3 From the **Measured Rad.** drop-down list, select the unit to which you want to calibrate.

4 Click **Calibrate to** and enter a value if you want to calibrate to a value.
Click **Set Factor** and enter a value if you want to calibrate by a factor.

5 Click **OK**.

The calibration result appears in the **Measurement** dialog.

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

Configuring AutoVue for 2D Files

You can configure background color and snap settings for 2D files. To access the 2D configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **2D** in the tree.

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.

Access this option from the **Configuration** dialog. Select **Options > Configuration** from the main menu, then click **2D** in the tree.

Configuring Colors

The **Colors** settings let you modify colors for 2D files. In the **Configuration** dialog, select **2D > Colors** in the tree 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.

Working with EDA Files

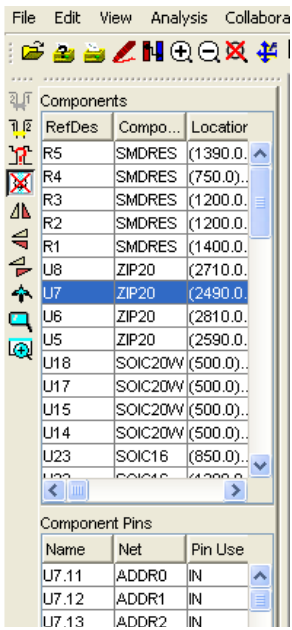
In addition to all features that are available for generic 2D files, AutoVue provides intelligent querying for EDA files. You can create/modify layer sets, you can query entity information, generate BOM, verify EDA designs, cross-probe and perform intelligent measurements.

Navigation Panel

The **Navigation Panel** is displayed on the left-hand side of the AutoVue workspace when you view an EDA drawing. The **Navigation Panel** 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 PCB design.

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) a component or entity; zoom to a component or entity and query entity information.



Components Tab

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

When you select a component, it appears highlighted on the drawing. To select more than one component, press **Shift** or **Ctrl** and then select the desired components. All the components you selected are highlighted. The associated pins for the selected component(s) are displayed.

Nets Tab

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

When you select a net, it appears highlighted on the drawing. To select more than one net, press **Shift** or **Ctrl** and then select the desired nets. You can also click and drag in the **Navigation Panel** list to select multiple nets. All the nets you selected are highlighted. The associated net node list(s) for all the nets you selected are displayed.

Bookmarks Tab

The **Bookmarks** tab allows you to navigate between PCB and schematic pages or between the 2D and 3D view of a PCB design.

Customizing the Navigation Panel

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

1 To sort a column, click the column heading.
2 To change the column order or to show or hide a column, right-click a column heading and select **Customize** from the pop-up menu.
The **Customize Columns** dialog appears.

3 To show or hide a column, select the checkbox beside the column(s) you want to show.
Clear the checkbox beside the column(s) you want to hide.
Note To show all columns, click **Show All**. To hide all columns, click **Hide All**.

4 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.

5 Click **OK**.
The changes appear in the **Navigation Panel**.

Selecting Entities

Selecting an entity or entities is often the first step to many of the operations that you will perform with EDA files. You can select an entity or entities in an EDA file from the **Navigation Panel** or the workspace. You can also select entities from the **Entity Browser** when you search for entities. To specify which types of entities you can or cannot select, use the **Entity Type Filter** dialog.

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

Selecting Entities from the Navigation Panel

Click any components, nets, associated pins or net nodes in the **Navigation Panel** to select them. To select more than one entity, **Shift**-click or **Ctrl**-click.

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 Also Zooming to a Selected Entity

Selecting Entities in the Workspace

Click an entity in the workspace to select it. To select more than one entity, **Shift**-click or **Ctrl**-click the entities.

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. For more information, ***see Filtering Entity Types***.

Note 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.

See Also Zooming to a Selected Entity

Selecting Entities from the Entity Browser

1 Perform an attribute-based or entity type-based search using the **Entity Browser**.

See Entity Browser

2 Select an entity or entities from the **Entity Types** list or the **Attributes** list of the **Entity Browser** dialog.

Note To select more than one entity, press the **Shift** or **Ctrl** key while selecting.

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

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

See Also Zooming to a Selected Entity

Filtering Entity Types

With the **Entity Type Filter** option, you can display certain entity types while hiding others, without having to hide an entire layer. For example, you might want to turn off all entities and display only the component instances.

You can also limit the types of entities you can select in the workspace.

1 Select **View > Entity Type Filter** from the AutoVue menu.
The **Entity Type Filter** dialog appears.

Note You can also click the Entity Type Filter button  on the AutoVue toolbar.

2 Under the **Visibility** column, select the checkbox beside the entity types you want to display in the workspace.
Clear the checkbox to hide the entity types.

3 Under the **Selection** column, select the checkbox beside the entity types that you want to be able to select in the workspace.
Clear the checkbox beside the entity types that you do not want to select.

Note To select all entity types, select the checkbox in the column header. Clear the checkbox to deselect all entity types.

4 Click **Apply** to apply the changes.

5 Close **OK** to close the **Entity Type Filter** dialog.
Only the selected entities types remain displayed in the workspace. When you click in the workspace, only the entities types checked in the **Entity Type Filter** dialog will be highlighted.

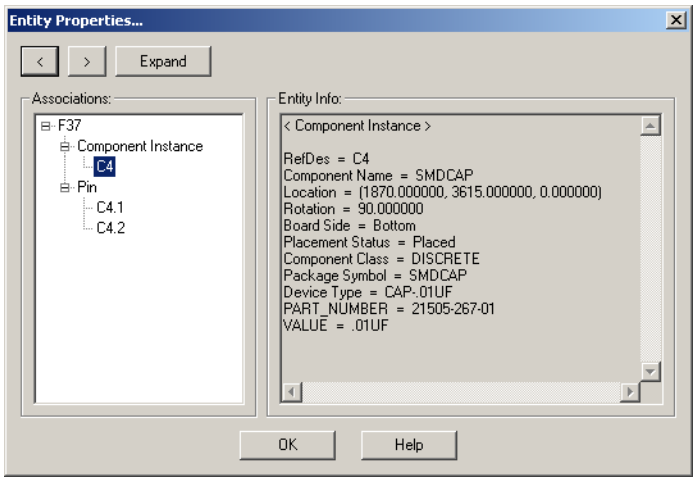
Zooming to a Selected Entity

1 In the workspace or **Navigation Panel** select the entity, then right-click and select **Zoom selected**.

2 In the **Entity Browser** dialog, select the entity from the **Entity Types** list or the **Attributes** list, then right-click and select **Zoom selected**.
AutoVue zooms to the selected entity in the workspace.

Entity Properties

The **Entity Properties** dialog displays detailed information about any selected entity in the current schematic drawing or PCB design. To open the **Entity Properties** dialog, you can double-click an entity in the workspace, or right-click an entity in the workspace, **Navigation Panel** or **Entity Browser** dialog, then select **Entity Properties** from the menu.



On the left side of the **Entity Properties** dialog, a tree control 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.

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.

1 Select the entity in the workspace or from the **Navigation Panel** or **Entity Browser**.

2 Right-click and select **Entity Properties** from the pop-up menu.
The **Entity Properties** dialog appears displaying the selected entity and its associated entities under **Associations**, and its attributes under **Entity Information**.
Note You can also double-click the entity in the workspace for which you want to view its entity information.

3 Select any associated entity in the tree to display its attributes under **Entity Information**.

4 To view all associated entities for any given entity in the tree, select the entity and click **Expand**.
Note 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

1 From the **Navigation Panel**, right-click an entity such as a pin, via or trace for which you want to display the net connectivity and select **Show Net Connectivity** from the pop-up menu.

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

The graphical entities belonging to the connected nets are highlighted.

Displaying the Entity Properties of a Net

It is possible to view properties of a net via **Entity Properties**.

Note **Entity Properties** is disabled when more than one net entity is selected.

1 Select an entity such as a pin, via or trace from the workspace.
The corresponding net is highlighted in the workspace.
2 Right-click the net and select **Entity Properties** from the pop-up menu.
The **Entity Properties** dialog appears displaying the properties of the selected net.
3 Click **OK** to close the **Entity Properties** dialog.

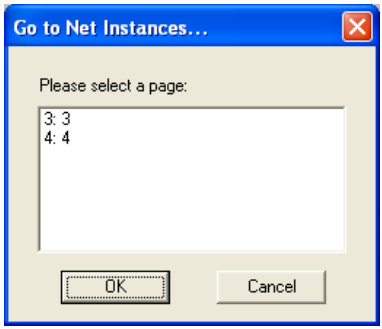
Note You can also select a net from the **Navigation Panel**, right-click and select **Entity Properties**.

Showing Net Instances

In AutoVue, you can display the instances of a net in a multi-page file.

1 From the **Navigation Panel**, click the **Nets** tab and select the net you want to view.
The net is highlighted in the **Navigation Panel** and in the workspace.
2 Select **View > Go To Net Instances** from the AutoVue main menu or right-click the highlighted net from the **Navigation Panel** or workspace and select **Go To Net Instances**.
AutoVue highlights the instance(s) 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**. The selected page is displayed and the instance of the selected net is highlighted.

Note If there are no instances for the selected net the option is disabled.

Design Hierarchy Navigation

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

Navigating using Descend Hierarchy

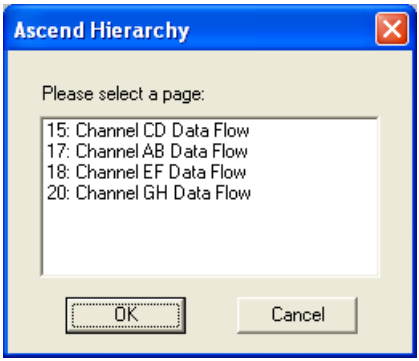
1 On the parent page, select any hierarchical block in the workspace, then right-click and select **Descend Hierarchy** from the pop-up menu. AutoVue opens the page with the selected child schematic.

Note You can also select the hierarchical block in the **Navigation Panel** or in the **Entity Browser** dialog, then right-click and select **Descend Hierarchy**.

Navigating using Ascend Hierarchy

1 On the child page of your schematic's design hierarchy, right click an entity in the workspace and select **Ascend Hierarchy** from the pop-up menu.

AutoVue returns to the parent page.
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 **Navigation Panel** or in the **Entity Browser** dialog, then right-click and select **Ascend Hierarchy**.

Layers

When working with EDA files, AutoVue lets you view all the physical layers and associated logical layers of the EDA drawing. You can view all layer sets for that file, or create your own. You can manipulate certain layer attributes, such as visibility, color, ordering.

You can access the layer settings using the **Layers** dialog. To open the **Layers** dialog, select **View > Layers** from the AutoVue main menu. You can also click the **Layers** button in the toolbar.

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

The **Layers** dialog has a Physical Layers pane and a Logical Layers pane. You can display and hide these panes using the **Expand** and **Collapse** buttons. The Logical Layers pane displays the layers in the order they display in the workspace, whereas the Physical Layers pane displays the layers in the order they appear in the layer stackup when manufactured.

The Physical Layers pane 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, trace, and so on. The entity types that display depend on the entities available in the open file. Select a physical layer to change its visibility, printability, layer order, and color. Select or clear a physical layer's entity type to affect its visibility separately from the layer's other entities.

When you select a physical layer from the Physical Layers pane, associated logical layers are also selected in the Logical Layers pane. You can also select individual logical layers from the Logical Layers pane, or **Shift-click** or **Ctrl-click** to select multiple logical layers.

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

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

When you select physical or logical layers and modify settings from the Layers dialog, a message displays at the bottom of the Layers 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 in AutoVue, it displays all layer sets for that file. You can choose the layer set you need, or create your own.

See Changing the Order of Layers
Modifying Layer Visibility
Setting Printability for Layers
Changing Layer Color
Sorting Logical Layers
Creating User-Defined Layer Sets

Changing the Order of Layers

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

1 Open the **Layers** dialog by taking one of the following steps:

- Click the **Layers** button  in the AutoVue toolbar.
- Select **View > Layers** from the AutoVue main menu.

2 If necessary, click the **Expand** button  to open the **Logical Layers** pane.

3 Select the layer or layers that you want to move. You can:
- Select a physical layer from the Physical Layers pane. Logical layers associated with the selected physical layer are selected. When you make changes to a selected physical layer, logical layers associated with it are also affected.
- Select one or multiple logical layers from the Logical Layers pane. **Shift-click** or **Ctrl-click** to select multiple logical layers.

4 Click one of the following buttons:
- Click the **Bring to Front** button  to move all selected layers to the front in the workspace. In the Logical Layers pane, these layers move to the top of the list.
- Click the **Up** button  to move selected layers up one layer.
- Click the **Down** button  to move selected layers down one layer.

Note You can also drag and drop selected logical layers in the Logical Layer pane.

5 Click **Apply** to see the changes in the workspace.
The file now displays according to the layer scheme you arranged. Also, logical layers are renumbered in the Logical Layers pane's Order column.

6 To save your changes as a user-defined layer set, see **Creating User-Defined Layer Sets**.

7 Click **Close** to close the **Layers** dialog.

Modifying Layer Visibility

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

1 Open the **Layers** dialog by taking one of the following steps:
- Click the **Layers** button  in the AutoVue toolbar.
- Select **View > Layers** from the AutoVue main menu.

2 Modify the visibility of physical layers and their entities from the **Physical Layers** pane.

3 Modify the visibility of individual logical layers from the **Logical Layers** pane.

See Modifying Visibility for Physical Layers

Modifying Visibility for Physical Layers

In the Physical Layers pane, you can show or hide the following:

- **All physical layers:** Click the **All** column header.
- **One physical layer:** Click the checkbox to the left of the layer name.

  You can also select a layer and click the **Visibility** button  .
- **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 checkbox.

Click **Apply** to view the changes in the workspace. To save your changes as a user-defined layer set, see **Creating User-Defined Layer Sets**.

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

Modifying Visibility for Logical Layers

If the **Logical Layers** pane is not open, click the **Expand** button  . The Logical Layers pane lets you show or hide the following:

- **One logical layer:** Click the layer's checkbox in the **Visibility** column  .
- **Multiple logical layers:** To select more than one layer, **Shift-click** or **Ctrl-click**, then click the **Visibility** button  .

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

5 To save your changes as a user-defined layer set, see **Creating User-Defined Layer Sets**.

Setting Printability for Layers

You can select which physical or logical layers to print using the Layers dialog.

Note You can only print layers that are visible in the workspace. See **Modifying Layer Visibility**.

1 Open the **Layers** dialog by taking one of the following steps:
   - Click the **Layers** button  in the AutoVue toolbar.
   - Select **View > Layers** from the AutoVue main menu.

2 If necessary, click the **Expand** button  to open the **Logical Layers** pane. In the Logical Layers pane, there is a **Printability** column that lets you see which layers are set to print.

3 Select the layer or layers that you want to be able to print. You can:
   - Select a physical layer from the **Physical Layers** pane. Logical layers associated with the selected physical layer are also selected.
   - Select one or multiple logical layers from the Logical Layers pane. **Shift-click** or **Ctrl-click** to select multiple logical layers.

4 You can only print layers that are visible, so make sure the layers and entities you want print are visible in the workspace. See **Modifying Layer Visibility**.

5 Click the **Printability** button  .

   Note In the Logical Layers pane, you can also click the checkboxes in the **Printability** column  for each logical layer you want to be able to print.

6 Click **Apply**.
   Only the layers selected in the **Printability** column can be printed.

7 To save your changes as a user-defined layer set, see **Creating User-Defined Layer Sets**.

8 Click **Close** to close the **Layers** dialog.

Changing Layer Color

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

1 Open the **Layers** dialog by taking one of the following steps:
   - Click the **Layers** button  in the AutoVue toolbar.
   - Select **View > Layers** from the AutoVue main menu.

2 If necessary, click the **Expand** button  to open the **Logical Layers** pane.

3 Select the layer or layers for which you want to change the color. You can:
   - Select a physical layer from the Physical Layers pane. Logical layers associated with the selected physical layer are also selected.

- Select one or multiple logical layers from the Logical Layers pane. **Shift-click** or **Ctrl-click** to select multiple logical layers.

4 Once you have selected a layer, you can modify its attributes.

5 Take one of the following steps:

- Click the **Color** button  . 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 **colored square** in the Color column  of the Logical Layers pane. Use the colored square to change the color of individual logical layers.

Note Do not click the Color column heading unless you want to sort the layers in the Logical Layers pane according to color.

6 In the menu that appears, select the color that you want.

7 Click **OK**.
The color of all selected layers changes in the Color column.

8 Click **Apply** to save the changes.
The display updates in the workspace based on the changes you made to the layer color.

9 To save your changes as a user-defined layer set, see **Creating User-Defined Layer Sets**.

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

Sorting Logical Layers

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

1 Open the **Layers** dialog by taking one of the following steps:

- Click the **Layers** button  in the AutoVue toolbar.
- Select **View > Layers** from the AutoVue main menu.

2 If the **Logical Layers** pane is closed, click the **Expand** button  to open it.

3 Click the column header of an attribute. Layers are sorted according to the attribute header you click.

For example, clicking the Color column header  sorts all logical layers in the pane 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 layer(s): z-order, visibility, printability, color, and so on. 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:

1 Take one of the following steps:

- Click the **Layers** button  in the AutoVue toolbar. In the Layers dialog that appears, select a layer set from the **Layer Set** drop-down list, then click **Apply**.
- In the AutoVue main menu, select **View > Layer Sets > x**, where **x** is the layer set you want to see.
- In the AutoVue toolbar, select a layer set from the drop-down list beside the Layers button  .

The selected layer set displays in the workspace.

2 To restore the default layer set, select **View > Layer Sets > (Default)** or **(Default)** from the toolbar or in the **Layers** dialog.

See ***Creating User-Defined Layer Sets***
Deleting User-Defined Layer Sets
Saving User-Defined Layer Sets with Markups

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 Set** list in the **Layers** dialog and to the **Layers** drop-down 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 Set** drop-down list.

1 Open the **Layers** dialog by taking one of the following steps:

- Click the **Layers** button  in the AutoVue toolbar.

- Select **View > Layers** from the AutoVue main menu.

2 Click **Add**.
The **Add Layer Set** dialog appears.
3 Enter the **Layer Set Name**.
4 Click **OK**.
The new layer set appears in the **Layer Set** drop-down list.
5 To modify the attributes of the new layer set, select one or more layers and change the **Visibility**, **Printability**, **Order** or **Color**.

See ***Changing the Order of Layers***
Modifying Layer Visibility
Setting Printability for Layers
Changing Layer 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.
The new layer set appears in the drop-down list beside the Layers button  on the AutoVue toolbar and is also displayed in the workspace.

See ***Viewing Layer Sets***
Deleting User-Defined Layer Sets
Saving User-Defined Layer Sets with Markups

Deleting User-Defined Layer Sets

1 Open the **Layers** dialog by taking one of the following steps:

- Click the **Layers** button  in the AutoVue toolbar.
- Select **View > Layers** from the AutoVue main menu.

2 From the **Layer Set** drop-down list, select the user-defined layer set you want to delete.
Note: You can only delete user-defined layer sets.
3 Click **Delete**.
Note: 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.

See ***Creating User-Defined Layer Sets***
Viewing Layer Sets

Saving User-Defined Layer Sets with Markups

In AutoVue, you can save custom layer sets using markup files.

1 Create custom layer sets.
See Creating User-Defined Layer Sets

2 Enter **Markup** mode by taking one of the following steps:

- Click the Markup button  in the toolbar.
- Select **File > Markup** from the AutoVue main menu.

3 Create any markup entities that you need.

4 Select **File > Save As** from the AutoVue main menu and use the dialog that appears to save the markup file.
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 Creating User-Defined Layer Sets
Viewing Layer Sets
Deleting User-Defined Layer Sets
Saving User-Defined Layer Sets with Markups
Markups

Manipulating EDA Views

With the **View** options you can instantly manipulate how the current active file is displayed. You can rotate a file's orientation counterclock-wise by 90, 180 or 270 degrees, flip a file's orientation horizontally, vertically or both simultaneously.

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.

For more information on how to manipulate views, ***see Manipulating 2D Views***.

3D View

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

1 In the **Navigation Panel**, click the **Bookmarks** tab.
2 Select **3D View** from the **Bookmark Tree**.
The 3D view of the PCB board is displayed in the workspace.
Note You can also display a 3D view by selecting **View > Page > Next Page** or by clicking the Next Page button  on the toolbar.

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. You can also cross probe between the 2D and 3D view of the same file.

See Cross Probing Between Two or More EDA Files
Showing the Net Connectivity when Cross Probing
Cross Probing Between 2D and 3D Views of the Same File
Zooming when Cross Probing

Cross Probing Between Two or More EDA Files

The **Cross Probe** option lets you select entities in the schematic drawing or the PCB and AutoVue highlights what you selected in all the open files.

1 Open a file you want to cross probe.
2 Select **Analysis > Cross Probe** from the AutoVue main menu.
The **Select files for cross-probing** dialog appears.

Note: You can also click the **Cross Probe** button  in the AutoVue toolbar.

3 Click **Add File**.
4 In the **File Open** dialog that appears, enter the **path** and **file name** of the file you want to cross probe, or click **Browse** to locate the file, then click **OK**.
5 Repeat steps 3 and 4 for each additional file that you want to open.
6 In the **Select files for cross-probing** dialog, click **OK**.
Each file you selected 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 is set to **Automatic** by default. When the **Automatic** option is selected for each dialog, you see different behavior depending on the files you cross probe, as follows:

- 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. For example, 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.
- If you cross probe two PCBs, one window automatically displays a 2D PCB and the other window displays a 3D PCB.
- If you cross probe two schematics, both are set to schematic.

7 If necessary, resize the windows so that you can see all the open files.

8 To change the view of a file in a window, take the following steps:

a. Clear the **Automatic** option in the dialog for the window you want to change.

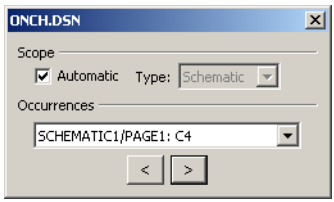
b. Select **Schematic**, **PCB**, or **PCB 3D View** from the **Type** menu. These options are available in the Type menu only when the views are present in the file.

The view of the file changes in the window.

9 To switch between multiple occurrences of a selected component, take the following steps:

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 **Occurrence** menu or click the **Next** button > or **Previous** button < to switch between component occurrences.



10 Select any entity in the schematic drawing.
The same entity is highlighted in the PCB design(s).

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

1 Select **Analysis > Cross Probe** from the AutoVue main menu.
The **Cross Probe** dialog appears.

Note: You can also click the Cross Probe button  in the AutoVue toolbar.

2 Click **Add File**.
3 In the **File Open** dialog that appears, select the same file, then click **OK**.
4 In the **Select files for cross-probing** dialog, click **OK**.
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.
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.
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, clear the **Automatic** option.
   b. Select a different view from the **Type** menu. For example, you can switch between **Schematic** and **PCB 3D View**.
5 Resize the windows so that you can see both views of the file.
6 Select an entity in the 3D view.
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

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

To zoom in on a selected entity:

1 Select an entity or entities in either one of the open files or views.
Note If you are cross probing between a schematic drawing or a PCB design, you can select entities in either file. You can also select entities in any open file when cross probing the 2D view and 3D view of the same file.

2 Right-click and select **Zoom Selected** from the pop-up menu.
AutoVue zooms to the same entities in all the open files.
See Configuring Zoom Behavior when Cross Probing

Comparing a PCB with Artwork

In AutoVue, you can visually compare PCB designs or schematic drawings. In **Compare** mode for EDA files, EDA options such as **Select Entity**, **Entity Browser** and **Entity Type Filter** are available.

For more information on how to compare files, ***see Comparing 2D Files***.

To compare a PCB with artwork:

1 Open the PCB file that you want to compare with artwork.
2 Select **Analysis > Compare** from the AutoVue main menu.
The **File Open** dialog appears.
3 Enter the path and **File Name** or click **Browse** to select the artwork file that you want to compare with the PCB file.
4 Click **OK**.
The **PCB-Artwork Comparison** dialog is displayed.
5 From the drop-down list, select the PCB physical layer that represents the artwork, then click **OK**.
AutoVue displays the selected layer in the first window, compares it with the Artwork in the second window and displays the results in the **Comparison Result** window.
6 To exit Compare mode, select **File > Exit Compare Mode** from the AutoVue main menu.
The original file appears in the AutoVue workspace.
Note You can right-click in any window and select **Exit Compare Mode**.

See Also Specifying Scale and Offset for a Compare File

Generating a Bill of Material

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.

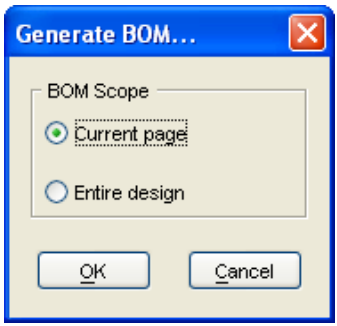
The **Bill of Material** (BOM) report produces a count of the unique components or parts needed for manufacturing. It lists the quantity required, component name, reference designators, value (for resistors and capacitors), size and descriptions. When determining a component's uniqueness, the count considers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

To generate a Bill of Material:

1 Select **Analysis > BOM** from the AutoVue main menu.

Note You can also click the **BOM** button  in the toolbar.

If you are generating a BOM for a schematic with multiple pages, a dialog appears.



Select if you want to generate a **BOM** for the **Current page** or the **Entire design**, then click **OK**.

A dialog appears listing the attributes of the file.

2 Select the attributes that you want to appear in the report, then click **Generate**.

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.

Note To sort a column by alphanumerical or alphabetical order, click on the column heading.

3 To save the Bill of Material, click **Export**. The **Save As** dialog appears. Specify the directory where you want to store the file, the file name and extension, then click **Save**.

Note You can specify either **xml** (Product Data Exchange) or **csv** (Common Separated Values) for the extension.

AutoVue automatically saves the results in a file with the format you specified.

4 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 limitations. Performing these checks will prevent short circuits and process errors. The types of design rules are:

- **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 change the unit of measure and change the value of each check. The available design rule checks are:

Type	#	Design Rule Check	Description
Clearance	1	Min.Clearance (Pad > Pad)	Specify the minimum pad to pad clearance on the same physical layer. **Note** Only the pad outline is considered.
	2	Min.Clearance (Pad > Trace)	Specify the minimum pad to trace clearance and pads on the same physical layer. **Note** Only the pad outline is considered. Does not take into account traces connected to a pin (pad) on the same net.

Type	#	Design Rule Check	Description
	3	Min.Clearance (Trace > Trace)	Specify the minimum trace to trace clearance on the same physical layer. **Note** Does not take into account directly connected traces.
	4	Min. Clearance (Comp > Comp)	Specify the minimum component to component clearance on the same physical layer (component top side or component bottom side). **Note** Only the logical layer that defines the actual outline of the component is considered. **Note** A component entity is usually made up of geometrics on different logical layers (silkscreen, keepout, etc.), but only the logical layer that defines the actual outline of the component is considered.
Manufacturing	5	Min. Annular Ring	Specify the distance between the pad outline and the outline of its drill hole on all physical layers. **Note** The two outlines (geometries) are part of the pin (pad) entity but on different logical layers.
	6	Min. Pad Diameter	Specify the minimum diameter of a pad. **Note** Only applicable to rounded pads.
	8	Acute Angle (in deg.)	Specify the minimum acute angle between connected traces on the same physical layer. **Note** Only two traces belonging to the same net should be considered.
	9	Min. Drill Hole Size	Specify the minimum diameter of the drill hole size allowed. **Note** Only applicable to rounded pads.
	10	Max. Drill Hole Size	Specify the maximum diameter of the drill hole size allowed. **Note** Only applicable to rounded pads.

Type	#	Design Rule Check	Description
Electrical	7	Max. Via Count	Specify the maximum via count allowed for a net.
	11	Min. Route Width	Specify the minimum route (trace) width allowed in the design.
	12	Max. Route Width	Specify the maximum route (trace) width allowed in the design.
	13	Min. Route Length	Specify the minimum route (trace) length allowed in the design.
	14	Max. Route Length	Specify the maximum route (trace) length allowed in the design.
	15	Empty Nets	Select this checkbox 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 checkbox 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 checkbox if you want to check for a net connected to only one pin. **Exception:** power and ground nets.
	18	Short Circuit	Select this checkbox 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 checkbox 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

1 Select **Analysis > Verify Design** from the AutoVue main menu.
A design dialog appears.

Note You can also click the Verify Design button  on the AutoVue toolbar

2 To enable a **Design Rule,** select the checkbox beside the **Design Rule**.
To disable a **Design Rule**, clear the checkmark.
Note You can sort the **Design Rules** by **Enabled**, **Description** or **Value** by clicking the column heading.

3 To add a value to the selected **Design Rules**, double-click in the value box and enter a value.
Note The **Design Rule** must be enabled to be able to add a value.

4 In the **Maximum Violation** text box, enter the maximum number of results you want to display in the **Results** text box.

5 From the **Distance Units** drop down list, select the unit you want to use as the unit of measure.

6 Click **Verify**.
Verify button changes to **Stop**.
Note 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 Violation** text box, appear in the **Results** list.

7 To view the description of a violation result, select the violation from the **Results** list.
The description appears in the **Description** text box. 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.
Note When you select a violation result, AutoVue zooms to the entity or set of entities that were affected and highlights them on the drawing.

8 Continue to select violation **Results** to view its description.
Note Click **Reset** to restart a new check.

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

Exporting the Design Verification Results

You can export the Design Verification results into a text file.

1 Click **Export** in the design dialog.

An export dialog appears.

2 Navigate and select the directory that you want to export the results to.

3 Enter a **File name**.
Click **Save**.
AutoVue saves the Design Verification results in a **txt** (text) or **csv** (Common Separated Values) file listing each violation result and its description.

4 Click **Close** to close the design dialog.

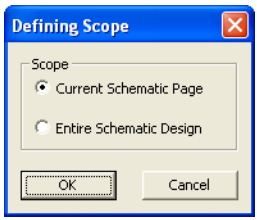
Searching Using Entity Browser

You can search for an entity in a schematic drawing or a PCB design using the **Entity Browser**. 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 .txt file.

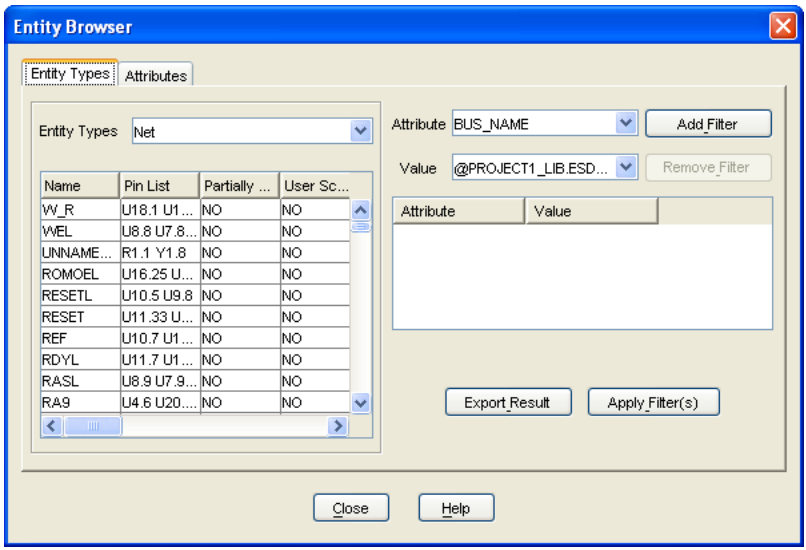
Note If the selected entity is small, a flash box appears indicating the location of the highlighted entity in the workspace.

To open the Entity Browser, select **Analysis > Entity Browser** from the AutoVue main menu.

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.



Performing an Entity Type-based Search

1 Select **Analysis > Entity Browser** from the AutoVue menu.
The **Entity Browser** dialog appears.

Note You can also click the Entity Browser button on the AutoVue toolbar.

2 Click the **Entity Types** tab.

3 From the **Entity Types** drop-down list, select an entity type.
A list of attributes for the selected entity type appear.

Note To sort the list by attribute, click the attribute column heading that you want to sort by.

4 If you want to search by attribute, select an attribute from the **Attribute** drop-down list.
If you want to search by value**,** select a value from the **Value** drop-down list.

5 Click **Add Filter**.
The **Attribute** and its **Value** appear in the list.

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

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

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

Note 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**.

8 To save the results, click **Export Result**.
The **Save As** dialog appears.

9 Browse to locate the directory where you want to store the file and enter a **File name**.

10 Click **Save**.
AutoVue saves the results in a **csv** (Comma Separated Values) file.

11 To view the properties of an entity, select the entity from the**Types** list or the **Attributes** list, then right-click and select **Entity Properties**.
The **Entities Properties** dialog appears displaying 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**.
The net connectivity for the selected entity is highlighted.

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

Performing an Attribute-based Search

1 Select **Analysis > Entity Browser** from the AutoVue menu.
The **Entity Browser** dialog appears.

Note You can also click the Entity Browser button  on the AutoVue toolbar.

2 Click the **Attributes** tab.

3 From the **Attributes** drop-down list, select an attribute.
The **Owner, Type** and **Value** of the selected attribute appear in the list.
Note To sort the list by **Owner**, **Type** or **Value**, click the column heading that you want to sort by.

4 If you want to search by owner, select an owner from the **Owner filter** drop-down list.
If you want to search by value**,** select a value from the **Value Filter** drop-down list.
Note You can also enter an **Owner filter** or **Value Filter** in the text box.

5 Click **Apply Filter(s)**.
A list of entities matching the search criteria appear displaying the **Owner**, **Type** and **Value** columns.

Note Select an item in the result list and right-click to access options like **Zoom Selected** and **Entity Properties**.

6 To save the results, click **Export Result**.
The **Save As** dialog appears.

7 Specify the file name and the directory where you want to store the file, then click **Save**.
AutoVue saves the results in a **csv** (Comma Separated Values) file.

8 Click **Close** to close the **Entity Browser** dialog.

Measuring in EDA Files

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

Click **Analysis > Measure** to access the Measurment options. You can also click the measurement buttons in the toolbar, as shown in the following table:

Button	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 entities.

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 mid, center and end-points of an entity, as well as a pin, via, and symbol:

Button	Snap to	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near the 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.
	Pin	Electrical snap mode where a snap box appears when the cursor touches a pin.
	Via origin	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 the entire component.
	Free snap	Allows snapping at any point on the drawing.

Measuring Distance

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the AutoVue toolbar.

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 clear all **Snapping Modes** click **All Off.**

See EDA Snapping Modes

4 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure the distance.

Note 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.

Note If you selected **Cumulative**, continue clicking points along the path that you want to measure.

7 Right-click to complete the measurement.
The points are joined by a line. The measured distance, Delta-X, Delta-Y and the "Manhattan Distance" appear in the **Measurement** dialog.

Note Click **Reset** to take another measurement.

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

Calibrating Distance

1 Measure the distance between two points or measure cumulative distance.

See Measuring Distance

2 In the **Measurement** dialog, click **Calibrate**.

3 The **Distance Calibration** dialog appears displaying the measured distance.

4 From the **Measured Dist.** drop-down list, select the unit to which you want to calibrate the distance.

5 Click **Calibrate to** and enter a value if you want to calibrate the distance to a value.
Click **Set Factor** and enter a value if you want to calibrate the distance by a factor.

6 Click **OK**.
The calibration results appear in the **Measurement** dialog.

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.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Minimum Distance button  on the AutoVue toolbar.

2 Click the **Min.Distance** tab.

3 Select  **First Set** to select the entities that you want to measure from.

4 Select the **Snapping Modes** that you want to select as the entities for the measurement.
See ***EDA Snapping Modes***
Note If you select **Net**, you cannot select any other type of entity.
5 Click the first set of entities on the drawing.
The entities are highlighted.
Note To clear the last set of entities you selected, click **Clear Set**.
6 Select **Second Set** to select the entities that you want to measure to.
7 Click the second set of entities on the drawing.
The entities are highlighted in a different color.
8 From the **Measured Min Distance** drop-down list, select the unit in which you want to measure the distance.
9 Click **Zoom to Result**, if you want to zoom in to the measurement on the drawing.
10 Click **Compute**.
The minimum distance from the first set of entities to the second set is highlighted by a line. The minimum measured distance, Delta-X, Delta-Y and the Manhattan Distance appear in the **Measure Minimum Distance** dialog.
11 Click **Close** to close the **Measurement** dialog.

Measuring Area

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

1 Select **Analysis > Measure** from the AutoVue main menu.The **Measurement** dialog appears.
Note You can also click the Area button on the AutoVue toolbar.
2 Click the **Area** tab.
3 Select **Between Points** if you want to measure the area between points on a drawing. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**
See ***EDA Snapping Modes***
4 Select **Shape** if you want to measure the area of a predefined shape on the drawing. **Snapping Modes** are disabled.
5 From the **Measured Area Units** drop-down list, select the unit in which you want to measure the area.

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

Measuring an Angle

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

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Angle button  on the AutoVue toolbar.

2 Click the **Angle** tab.
3 Select **From 3 Points** if you want to measure the angle between three points. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**
See EDA Snapping Modes
4 Select **Between 2 lines** if you want to measure the angle between two lines. **Snapping Modes** are disabled.
5 From the **Measured Angle Units** drop-down 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.
Angle arms appear with an arc connecting them. The angle measurement appears in the **Measurement** dialog.
Note Click **Reset** to take another measurement.

8 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 radius, center and diameter.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the AutoVue toolbar.

2 Click the **Arc** tab.
3 Select **From 3 Points** if you want to measure the arc between three points. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**
See EDA Snapping Modes
4 Select **Arc Entity** if you want to measure a predefined arc. **Snapping Modes** are disabled.
5 From the **Arc Info** drop-down list, select the unit in which you want to measure the distance of the arc.
6 From the **Measured Angle Units** drop-down 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.
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.
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.
Note Click **Reset** to take another measurement.
9 Click **Close** to close the **Measurement** dialog.

Calibrating an Arc

1 Measure an arc in the drawing.
See Measuring an Arc

2 In the **Measurement** dialog, click **Calibrate**.
3 The **Radius Calibration** dialog appears displaying the measured distance.
4 From the **Measured Rad.** drop-down list, select the unit to which you want to calibrate the distance.
5 Click **Calibrate to** and enter a value if you want to calibrate to a value.
Click **Set Factor** and enter a value if you want to calibrate by a factor.
6 Click **OK**.
The calibration results appear in the **Measurement** dialog.
7 Click **Close** to close the **Measurement** dialog.

EDA Terms and Definitions

Annular ring

A circular strip of conductive material that remains after a hole has been drilled through the pad of a printed circuit board.

Anti-copper

An area within a fill zone in which copper cannot be placed.

Aperture

An opening, similar to the aperture of a camera, that is used for photo-plotting. Apertures are available in various sizes and shapes.

Aperture list

A text file containing the dimensions for each of the apertures used to photo-plot PCB artwork.

Clusters

Components that are grouped according to their interrelationships and placed in close proximity on the board. This keeps the connections on the PCB short so that the board is easier to work with.

Component

An element or a part of a PCB.

Component density

The quantity of components on a unit area of a PCB.

Component hole

A hole in the printed circuit board that corresponds to a pin or wire of a component. This hole serves the dual function of attaching the component to the board and establishing the electrical connection between the pin or wire and the remainder of the board circuitry.

Component library

A computer data file that contains the footprint patterns for a number of components.

Component side

The uppermost or top layer of a board on which most components are placed.

Component silkscreen

The silkscreen markings of the printed circuit board that appear on the component side. The silkscreen is applied over the solder mask.

Component solder mask

The colored, usually translucent, coating applied to the board over the etched copper. It protects the selected areas from the soldering process.

Connection

An unrouted, partially routed, or completely routed path between two pads. In a net with *n* pads, there are exactly *n–1* connections.

Copper pour

A method by which a copper zone is filled with a specified pattern, with objects that cross the zone or lie within the zone being avoided.

Copper zone

An area on a board designed to be covered by a layer of copper when manufactured. Also known as a "metal zone."

Cross hatching

The breaking up of large conductive areas by the use of a pattern of lines and spaces in the conductive material.

Datum

A specific location (a point) that serves as a reference to locate a PCB pattern or layer for manufacture.

Density

On a PCB, the degree to which components are packed on the board. Generally, the density is given as the number of square inches per equivalent, i.e., a lower number indicates a more dense board.

Discrete components

Components with three or fewer electrical connections (for example, resistors or capacitors).

Electrical check

The process of checking the PCB to ensure that the connections they are on match those specified in the net list.

Fill zone

A zone that defines an area to be filled with copper.

Fine pitch

A class of surface-mount components that is characterized by pins measuring 0.025 inches or less from pad center to pad center.

Footprint

The physical description of a component. It consists of three elements: **padstacks**, representing the pads of the component; **obstacles**, representing among other things, the physical outline of the component, silkscreens, keepouts/keepins, and assembly drawing data; and **text** documenting the footprint information (for example, the component name). You may want each project to have its own footprint library containing all the footprints used in that project.

Ground plane

A large area on the PCB, usually an entire layer, that provides a common ground connection for all component ground pins and other ground connections.

Heatsink

A mechanical device made of a high thermal conductivity material that dissipates heat generated by a component or assembly.

Heuristics

A method of routing that consists of repeated attempts to apply very simple routing patterns to unrouted connections in order to complete the routing quickly and cleanly. Typically, heuristics are used for memory and short point-to-point routing.

Hole

The area where board material must be removed by drilling or milling.

Isolation

The clearance around a pad, track, zone, or via that defines the nearest approach allowed by conductors of another signal set.

Jumper wire

A discrete electrical component or wire used to make electrical connections between points that have no copper etch due to board density or some other factor.

Keepout

An area fill within which no routing is allowed.

Land

The copper pad needed for a surface mount pin.

Layer

One in a series of planes in a PCB design on which tracks are arranged to connect components. Vias connect tracks and zones between layers.

Manual routing

Individual connections, in the form of traces, vertices, arcs, etc., 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.

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.

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.

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.

Via stringer

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.

Working with 3D Files

From AutoVue's 3D mode, you can select model parts to transform independent of the rest of the model. You can import 3D models into the current window and export files to other formats. 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.

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

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

Models Tab

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

Views Tab

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

Bookmarks Tab

The **Bookmarks** tab lists links to specific views (Draft views, 2D plans) or other files with information related to the model. Navigate between these files and views by clicking the appropriate link. Bookmarks lead to various views of CAD files such as CATIA 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

In AutoVue, you can select model parts from the **Model Tree** or from the workspace and have them appear highlighted 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**.

Note In AutoVue, you can configure the **Selection Highlight**, ***see Configuring AutoVue for 3D Files***.

Selecting Model Parts from the Model Tree

1 Click the **Models** tab.

2 Select the part or parts from the **Model Tree**.

Note To select more than one model part, press the **Shift** or **Ctrl** key while selecting.

The selected part(s) appear highlighted on the model and in the **Model Tree**.

Select Model Parts from the Workspace

1 Select a part or parts on the model in the workspace.

Note To select more than one model part, press the **Shift** or **Ctrl** key while selecting.

2 To select the parent entities of a selected part, press the **Shift** key and select the part again.

A pop-up appears listing the parent entities of the selected part. Select an entity from the pop-up list.
The selected part(s) appear highlighted on the model and in the **Model Tree**.

3 To select a group of parts, select **Edit > Select,** then click and drag the mouse around the parts.
The entities within the box are selected and highlighted in the **Model Tree**.
Note 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

1 Select a model part in the workspace or from the **Model Tree**, then right-click and select **Select Identical Parts** from the pop-up menu.
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**.
Note You can also select sub-assemblies and **Select Identical Parts** to display all identical sub-assemblies.

Hiding Model Parts

In AutoVue, you can hide specific parts of a model or display specific parts and hide the rest of the model.

1 Select the part or parts on the model or from the **Model Tree**.
Note To select more than one model part, press the **Shift** or **Ctrl** key while selecting.
The selected part(s) appear highlighted on the model and in the **Model Tree**.

2 To hide the selected part(s), right-click a selected part on the model or from the **Model Tree** and select **Hide** from the pop-up menu.
The selected part(s) are hidden on the model.
To display the selected part(s) and hide the rest of the model, right-click a selected part on the model or from the **Model Tree** and select **Hide Rest** from the pop-up menu.
The selected parts are displayed in the workspace and the rest of the model is hidden.

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

Note The **Re-Center > Selected** option is only available when one or more model parts are selected.

1 Select the model part or parts that you want to use.
2 Select **View > Re-Center > Selected** from the AutoVue main menu. The model is repositioned using the selected model part or parts as the central reference point.

Re-center All

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

1 Select **View > Re-Center > All** from the AutoVue main menu. The model is repositioned to the center of the workspace.

Re-centering a Model to an Entity

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

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

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

1 Select **View > Re-Center > Entity** from the AutoVue main menu.
The **Re-Center** dialog appears.
2 Select the entity that you want to use as the central reference point.
All instances of the entity are highlighted.
3 Click an instance of the entity.
The model is repositioned using the selected entity as the central reference point.
4 Close the **Re-Center** dialog.

Expanding/Collapsing the Model Tree

In AutoVue, you can expand the **Model Tree** to display child entities of a selected node or nodes.

Note In AutoVue, you can also can configure the level you want to expand the **Model Tree**. To configure the **Model Tree Level**, ***see Configuring AutoVue for 3D Files***.

5 Click the **Models** tab.
6 From the **Model Tree**, select the node or nodes that you want to expand, then right-click and select **Expand All Children** from the pop-up menu.
Note To select more than one node, press the **Shift** or **Ctrl** key while selecting.
The selected entities are highlighted. The **Model Tree** expands displaying the child entities of the selected node(s).

7 To collapse a node(s), select the node(s), then right-click and select **Collapse All Children** from the pop-up menu.
The **Model Tree** collapses the selected node(s).

Note You can also expand a node by clicking  . To collapse a node click  .

Creating 3D Mockups

In AutoVue you can import other 3D Models into the current active file.

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

1 Select **Analysis > DMU** from the AutoVue main menu.
The **Import Design** dialog appears.
2 Click **Add**.
The **File Open** dialog appears.
3 Enter path and **File Name** or click **Browse** to locate the file you want to import.
4 Click **OK**.
The file appears in the **Import Design** dialog.
Note To import more than one file, repeat steps 2 to 4.
5 Click **OK**.
The file(s) you imported appear in the workspace and in the **Model Tree**.
Note Once open, you can position these models in the desired manner using the **Transformation** tool, or align the models using **Part Alignment**, or run interference checks.

See Also ***Tranformation***
Part Alignment
Performing Interference Checks

Deleting Models from a Mockup

1 Select **Analysis > DMU** from the AutoVue main menu.
The **Import Design** dialog appears.
2 Select the file(s) that you want to remove.
3 Click **Remove**.
The file(s) are removed from the list.
4 Click **OK**.
The file(s) disappear from the workspace and from the **Model Tree**.

Converting 3D Models to Other Formats

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

Conversion Options

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

Option	Description
Save As	Specify the name and path of the file in which the conversion is to be stored. This file is also known as the output file. You can use **Browse** to provide AutoVue with the file's path.
Convert to Format	A drop-down list of all the possible types of output file formats currently available for conversion. The available formats are: • CALS GP4 • Encapsulated Postscript (Raster) • HP Laserjet Printer (PCL) • Run Length RLC File • PCX Bitmap • PDF • Stereolithography (STL) • TIFF • Virtual Reality Modeling Language (VRML) • Windows Bitmap
Sub-Format	The **Sub-Format** drop-down list appears when you select **TIFF** or **STL** from the **Convert to Format** drop-down list. Select the **Sub-Format** from the drop-down list.

Option	Description
Color Depth	Select an option from the drop-down 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. **Note** Option is not available for **STL** and **VRML** formats.
Postive Triangle Values	When selected, the model is translated so that all the vertex coordinate values are positive. **Note** Option is only available when **Convert to Format** is set to **STL**.
Convert Region	The area of the file to be converted. The available options are: • **All** - converts the entire file • **Display** - converts the image to fit on the output page For example, if you zoomed in on a particular region of the file, the zoomed portion of the file is converted. • **Extent** - converts the entire extents of the file • **Selected** - converts the selected model parts in the file **Note** Options vary depending on the selected format.
Convert Pages	The number of pages to be converted. The available options are: • **All** - converts all pages of the file • **Current** - converts the current page of the file • **Range** - converts the pages of the file indicated in the range **Note** Option is only enabled when converting to **TIFF**.

Option	Description
X and Y	Specify **X** and **Y** to define the resolution for the converted file. Choose from pixels, inches and millimeters for the units. • **X** indicates the number of horizontal pixels • **Y** is the number of vertical pixels for the current active file contents **Note** At times, AutoVue presets **X** and **Y** to match the specifications of the selected conversion file format.

PDF

With AutoVue it is possible to convert Office, 2D and EDA formats to PDF. When converted from **Markup Mode**, markups are "burned" onto the PDF. When you open the PDF, you will see the base file along with all markups.

Converting a 3D Model

1 Select **File > Convert** from the AutoVue main menu.
The **Convert** dialog appears.
2 In the **Save As** text box, 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** drop-down list, select the format you want to convert the file to.
Note The **Output** and **Convert** area options change according to the type of format you choose.
4 Select the output and convert options that you want to set for the file.
See Conversion Options
5 Click **OK**.
The **Conversion in progress** dialog appears. The dialog disappears when the file is converted successfully.

Manipulating Views

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 of the AutoVue main menu. The options are:

Select		Description
Zoom	In	Zoom in by a factor of 2.
	Out	Zoom out by a factor of 2.
	Fit	Resize object to fit window. From the toobar, click the  button or right-click in workspace and select from pop-up menu.
	Previous	Return to the previous zoom level. Right-click in workspace and select from pop-up menu.
	Selected	Resize the object so that the objects **selected** fill the window.
	Box Mode	Click and drag to draw a box around an object that you want to enlarge to fill the window. From the toobar, click the  button or right-click in workspace and select from pop-up menu.
	Dynamic Mode	Click and drag the cursor up to zoom in or down to zoom out. From the toobar, click the  button or right-click in workspace and select from pop-up menu.
Pan		Click and drag to reposition the model, then release the mouse button. From the toobar, click the  button or right-click in the workspace and select from the pop-up menu

Select	Description
Rotate Mode	Click and drag to rotate the model on all three axes, then release the mouse button. From the toobar, click the  button or right-click in workspace and select from pop-up menu.
Spin Mode	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 work-space. **Note** The spin velocity is determined by the speed at which you drag the mouse. From the toobar, click the  button.
Re-center	Repositions the model. **Re-center All** - reposition the entire model back to the center of the AutoVue workspace. **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. **Entity** - select a model part or entity as a central reference point to reposition a model. ***See Re-centering***
Next Page	Go to the next page of a multi-page file.
Previous Page	Go to the previous page of a multi-page file.
Page Number	Go to the specified page of a multi-page file.
Perspective	Show objects in three dimensions with the distances, planes and curved surfaces adjusted to give a sense of depth that reflects the desired perspective to the eye.

Select	Description
Views	Display different views of 3D models or create your own. Right-click in workspace and select the view from the pop-up menu. ***See 3D Views***
Display Attributes	Display a 3D model in different render modes. From the toobar, click the  button or right-click in workspace and select from pop-up menu. ***See Displaying Attributes***
Show Tree	Show the **Model Tree** of the current active file. From the toobar, click the  button.

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:

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.

Method	Description
Silhouette	A wireframe constructed with all internal lines visible but including additional silhouette edges. These are not "true" edges but help to visualize the model.
Wire Polygons	A skeletal model constructed of unfilled polygons.
Shaded Wire	An outlined solid model constructed of planes and surfaces. The outline is a solid line and the model's surfaces are shaded to increase the illusion of three dimensions.

Changing the Render Mode

Note A checkmark beside the attribute indicates the current render mode.

1 From the AutoVue main menu, select **View > Display Attributes** and the render mode that you want to render the model in.

Note You can also change the render mode for selected model parts.

Note You can also click the Render Mode button  on the AutoVue toolbar or right-click in the workspace and select **Display Attributes**.

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

Changing the Visibility

In AutoVue, you can show or hide parts of a model.

1 From the **Model Tree**, clear the checkbox beside the model part(s) that you want to hide.

Select the checkbox beside the model part(s) that you want to set visible.

The selected model part(s) appear or disappear from the model.

Note You can also hide model part(s) by selecting them on the model or from the **Model Tree**, then right-clicking and selecting **Hide** from the pop-up menu. Selecting **Hide Rest** displays the selected model part(s) and hides the rest of the model.

Changing Model Color

In AutoVue, you can change the color of a specific model part or selected model parts.

1 Select the model or model parts whose color you want to change.

Note If no part is selected, the change is applied to the entire model.

2 Select **View > Display Attributes > Color** from the AutoVue main menu.
The **Color** dialog appears.

3 Select the color that you want to change to.

4 Click **OK**.
The selected part(s) 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.

Note You can also right-click in the workspace and select **Display Attributes > Color**. In the **Model Tree**, you can select the model part(s), then right-click and select **Display Attributes > Color** from the pop-up menu.

Adjusting the Transparency

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

1 Select the model or model parts whose transparency you want to adjust.

2 Select **View > Display Attributes > Transparency** from the AutoVue main menu.
The **Set Transparency** dialog appears.

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

Note You can also adjust the transparency by entering a **Value** from 0 to 1:
0 = opaque
1 = transparent

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

Note You can also right-click in the workspace and select **Display Attributes > Transparency**. In the **Model Tree**, you can select the model part(s), then right-click and select **Display Attributes > Transparency** from the pop-up menu.

Light Settings

The default light setting consists of a white light positioned directly in front of the object as it is displayed. In the **Lighting** dialog it appears as a ball of white light in the center 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.

In AutoVue, you can:

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

Setting Ambient Lighting

1 Select **View > Set Light** from the AutoVue main menu.
The **Lighting** dialog appears.

Note You can also click the light bulb button  on the AutoVue main toolbar.

2 Click and drag the **Ambient Light** sliding bar until you achieve the desired lighting.
The lighting automatically changes with the movement of the sliding bar.
Note To set the ambient lighting to its default setting, click **Reset**.

3 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.

1 Select **View > Set Light** from the AutoVue main menu.
The **Lighting** dialog appears.

Note You can also click the light bulb button  on the AutoVue main toolbar.

2 To change the direction of the light, click and drag the small ball until you achieve the desired lighting.
Note You can also drag the white ball outside of the large ball.

The direction of the light reflected on the 3D model automatically changes with the movement of the white ball.

Note To set the light direction to its default setting, click **Reset**.

3 Click **Close** to close the **Lighting** dialog.

Adding a New Light Source

1 Select **View > Set Light** from the AutoVue main menu.
The **Lighting** dialog appears.

Note You can also click the light bulb button  on the AutoVue main toolbar.

2 Right-click inside the square surrounding the ball and select **Create New Light source** from the pop-up menu.
A second small ball appears.

3 Click and drag the small ball until you achieve the desired lighting.

Note To restore the light source to its default setting, right-click inside the square and select **Restore Default** from the pop-up menu.

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

Note You can add up to eight light sources.

Changing the Light Color

1 Select **View > Set Light** from the AutoVue main menu.
The **Lighting** dialog appears.

Note You can also click the light bulb button  on the AutoVue main toolbar.

2 To change the color of the light source, right-click directly on the small ball and select **Set Light Color.**
The **Color** dialog appears.

3 Select a color.

4 Click **OK**.
The small ball changes color and the model reflects the selected light color.

Note To set the light color to its default setting, click **Reset** in the **Lighting** dialog.

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

Removing a Light Source

1 Select **View > Set Light** from the AutoVue main menu.
The **Lighting** dialog appears.

Note You can also click the light bulb button  on the AutoVue main toolbar.

2 Right-click directly on the small ball that you want to remove and select **Remove Light**.
The small ball disappears and the light change is reflected.

3 Click **Close** to close the **Lighting** dialog.

3D Views

In AutoVue, you can display different views of 3D Models or create your own views.

Default View

The **Default View** is the isometric view for three-dimensional CAD models. This viewpoint has the viewing camera positioned at an equal distance from all three axes and pointing directly at the origin.

Setting Standard or Predefined Views

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

Note A checkmark beside the view option indicates the current pre-defined view you want to display.

1 From the AutoVue main menu, select **View > Views** and the pre-defined view you want to display.
The model changes to the selected view.
The view options that you can select from are Isometric, Top, Bottom, Front, Back, Left and Right.

Note You can also click the **Views** tab, then click the view from the **Standard Views** tree, or right-click in the workspace and select **Views**.

Setting Native Views

AutoVue displays views for a 3D file as saved in its native application. The views are listed under **Native Views** from the **Views** tab.

Note Native views are present only if the file contains saved views.

1 From the AutoVue main menu, select **View > Views > Native Views** and the view you want to display.
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.
Note You can also right-click in the workspace and select **Views > 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**.

Note You can define a user view in **View** or **Markup** mode. Any views you apply to displayed models during **Markup** mode are saved as part of the Markup.

1 Apply your own views or transformation to the displayed model.
Note 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 Select **View > User Defined Views > Add** from the AutoVue menu.
The **Add User Defined View** dialog appears.
Note You can also click the **Views** tab, then right-click **User Defined Views** and select **Add** from the pop-up menu or right-click in the workspace and select **Views > User Defined Views**.
3 Enter a **View Name** for the view that you want to define.
4 Click **OK**.
The view you defined is saved and appears in the **Views** tree under **User Defined Views**.
Note To see the view you defined, click the **Views** tab and select it from the tree or select **View > Views > User Defined Views**. You can also alternate between **User Defined Views** and **Standard Views** without affecting your personalized views.

Deleting a User Defined View

1 Under the **User Defined Views**, select the view that you want to delete, then right-click and select **Delete**.
A **Confirmation** dialog appears.
2 Click **OK**.
The view disappears from the **User Defined Views**.

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 viewpoint.

1 Select **View > Perspective** from the AutoVue main menu.
The depth of the model changes.
Note A checkmark beside **Perspective** indicates the view is selected.

Viewing a Model from a Particular Viewpoint

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

1 Select **View > Views > View Point** from the AutoVue main menu.
2 The **View Point** dialog appears.
Note You can also right-click in the workspace and select **Views > View Point**.
3 Enter the coordinates for **X, Y** and **Z**.
For **Camera Position**, enter the coordinates to determine the position of the camera lens.
For **Target Position**, enter the coordinates to determine the position of the 3D model viewed through the camera lens.
For **Up Direction**, change the coordinates to values between 0 and 1 to determine the upward orientation.
4 Click **OK**.
The entity moves to the defined viewpoint. The Global axes and the User Coordinates System change position to reflect the viewpoint.

Layers

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

1 Select **View > Layers** from the AutoVue main menu.
The **Select the layers to display** dialog appears listing the layers and layer visibility for the current active file.

Note You can also click the Layers button  on the AutoVue toolbar.

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 Select the checkbox beside the layers that you want to set visible.
Clear the checkbox beside the layers that you want to hide.
4 Click **OK**.
The selected layers are displayed.

Entity Properties

In AutoVue, you can view properties such as visibility, colors, transparency, mass properties and extents of a model or model parts.

Viewing Attributes

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

Attribute	Description
Color	Color of the selected model part.
Density	The density of the model or selected model part(s).
Mesh Resolution	The number of polygons drawn when displaying a model.
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 **Wire-frame**.

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

If you are displaying the 3D view of an EDA, AutoVue displays the general attributes mentioned above. In addition, 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 and attributes specific to the PCB board. Some of the viewable **PMI** attributes are X-Axis, Y-Axis, font color, text font name and tolerance type.

1 To view the attributes of specific model part(s), select the parts from the model.
To view the attributes of the entire 3D model, make sure that no parts are selected.
2 Select **Analysis > Entity Properties** form the AutoVue main menu.
The **Entity Properties** dialog appears.

Note You can also click the Entity Properties button  on the AutoVue toolbar or right-click in the workspace and select **Entity Properties**. In the **Model Tree**, select the model part(s), then right-click and select **Entity Properties** from the pop-up menu.

3 Click the **Attributes** tab.
The attributes of the selected model part(s) are displayed in a hierarchical tree.
4 Click **Close** to close the **Entity Properties** dialog.

Viewing Mass Properties

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

1 Select the part(s) for which you want to calculate the **Mass Properties**.
Note To view the **Mass Properties** of the entire 3D model, make sure no parts are selected.
2 Open the **Entity Properties** dialog by taking one of the following steps:
- Click the **Entity Properties** button  in the AutoVue toolbar.
- Double-click a model part or parts in the workspace.
- Right-click model part or parts in either the **Model Tree** or the workspace, then select **Entity Properties** from the pop-up menu.
- Select **Analysis > Entity Properties** from the AutoVue main menu.

3 Select a model part or parts.
4 Click the **Mass Properties** tab to view the Mass properties.
Note When a Mass property cannot be calculated, "N/A" displays in red for that property. When this is the case, you can also click the **Error Report** button for a list of Mass properties that could not be calculated.
5 Click **Options** to change the density, change measurement units, or configure computation of inertia tensor.
The **Options** dialog appears.
See Configuring Mass Properties
6 Click **Close** to close the **Entity Properties** dialog.

Configuring Mass Properties

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

1 From the **Mass Properties** pane, click **Options**.
The **Options** dialog appears.
2 To change the **Density**, enter a value.
To change the **Density** units, select the unit for **Mass** and **Length** from their respective drop-down lists.
3 To apply the density to model parts with unknown density, select **Use Only for Parts with Unknown Density**.
To apply the density to all model parts, select **Use for All Parts**.
4 To change the **Display** units, select the unit for **Mass** and **Length** from their respective drop-down lists.
5 To compute the inertia tensor based on output coordinate system, select **Output Coordinate System Origin**.
To compute the inertia tensor based on the center of gravity, select **Center of Gravity**.
6 Click **OK**.

Mass properties are instantly calculated and displayed in the **Mass Properties** panel.

Note When a Mass property cannot be calculated, "N/A" displays for that property. When this is the case, you can also click the **Error Report** button for a list of Mass properties that could not be calculated.

7 To save the changes, click **Save As**. The **Save As** dialog appears. Specify the path where you want to store the file and enter the **File name**, then click **Save**.
AutoVue saves the results in a **csv** (Comma Separated Values) file.

8 Click **Close** 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 for any model or selection, as well as the width, height and depth measurements.

1 Select the part(s) for which you want view **Extents**.
Note To view the **Extents** of the entire 3D model, make sure no parts are selected.

2 Select **Analysis > Entity Properties** form the AutoVue main menu.
The **Entity Properties** dialog appears.

Note You can also click the Entity Properties button  on the AutoVue toolbar or right-click in the workspace and select **Entity Properties**. In the **Model Tree**, select the model part or parts, then right-click and select **Entity Properties** from the pop-up menu.

3 Click the **Extents** tab.

4 Click **Transformed** if you want to view the extents after you have transformed the model.
Click **UnTransformed** if you want to view the extents of the model without transformation.
Click **Oriented** if you want to view the X, Y and Z coordinates of the reoriented model or selection.

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

PMI Filtering

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 (construction lines, surfaces, 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 & GD&T)
- Functional Tolerance Annotation (FTA or FTA&A)

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

1 Select **View > PMI Filtering** from the AutoVue main menu.
The **PMI Filtering** dialog appears displaying all PMI types.

Note You can also click the PMI Filtering button 

on the AutoVue toolbar.

2 In the **Tree** column, select the checkbox beside the items that you want to show in the **Model Tree**.
In the **View** column, select the checkbox beside the items that you want to show on the model.
Clear the checkbox beside the items that you want to hide.
Note To show or hide all items, click **All** to show all or **None** to hide all.

3 Click **OK**.
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.
Note Visibility settings are saved and will be restored the next time you open the **PMI filtering** dialog.

4 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** from the pop-up menu.

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** from the pop-up menu.

AutoVue zooms to the selected PMI item.

Manipulators

In AutoVue, 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.

Using the **Manipulators**, 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(s) of the 3D model.

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

1 Select **Analysis > Manipulators** from the AutoVue main menu.
You are now in Manipulators mode.

Note You can also click the Manipulators button  on the AutoVue toolbar or right-click in the workspace and select **Manipulators**. In the **Model Tree**, you can select the model part(s), then right-click and select **Manipulators** from the pop-up menu.

2 Select the model part or parts that you want to move.
Note To select more than one part, press the **Shift** or **Ctrl** key when selecting.
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.
Note To remove the Manipulators, click the Manipulators button.

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

1 Select **Analysis > Manipulators** from the AutoVue main menu.

You are now in Manipulators mode.

Note You can also click the Manipulators button  on the AutoVue toolbar or right-click in the workspace and select **Manipulators**. In the **Model Tree**, you can select the model part(s), then right-click and select **Manipulators** from the pop-up menu.

2 Select the model part or parts that you want to manipulate.
Note To select more than one part, press the **Shift** or **Ctrl** key when selecting.
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.
Clicking on an axis sphere allows rotation around one of the other two axes.
Note The initial mouse movement determines which of the two axes will mark the site of rotation. If the axis you chose 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 Move the mouse to rotate the model by the selected axis.
Note If you want to rotate the selection freely, as done in Rotate Mode, press the **Ctrl** key during rotation.

5 To remove the Manipulators, click the Manipulators button  .
Note To remove Manipulators, you can also select **Analysis > Manipulators**.
Mouse actions are restored to what they were prior to entering Manipulators mode.

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

1 Select **Analysis > Manipulators** from the AutoVue main menu.
You are now in Manipulators mode.

Note You can also click the Manipulators button  on the AutoVue toolbar or right-click in the workspace and select **Manipulators**. In the **Model Tree**, you can select the model part(s), then right-click and select **Manipulators** from the pop-up menu.

2 Select the model part or parts that you want to scale.
Note To select more than one part, press the **Shift** or **Ctrl** key when selecting.

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.

Note To remove the Manipulators, click the Manipulators button.

Part Alignment

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

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

With the **Show Preview** option, you can visualize the part alignment before applying the change.

When aligning models parts, you can choose from one of the following **Constraint Types**.

Option	Type	Description
Constraint	Coincident	The **Mobile** part is positioned so that the selected point on the mobile part coincides with the selected part on the **Fixed** part.
	Parallel	Align the **Mobile** point so that it is parallel to the **Fixed** point.
	Perpendicular	Align the **Mobile** point so that it is perpendicular to the **Fixed** point.
	Concentric	Align the **Mobile** point so that it has the same center as the **Fixed** point

Part Alignment Constraints

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

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

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

Where:

Coin - **Coin**cident

Par - **Par**allel

Per - **Per**pendicular

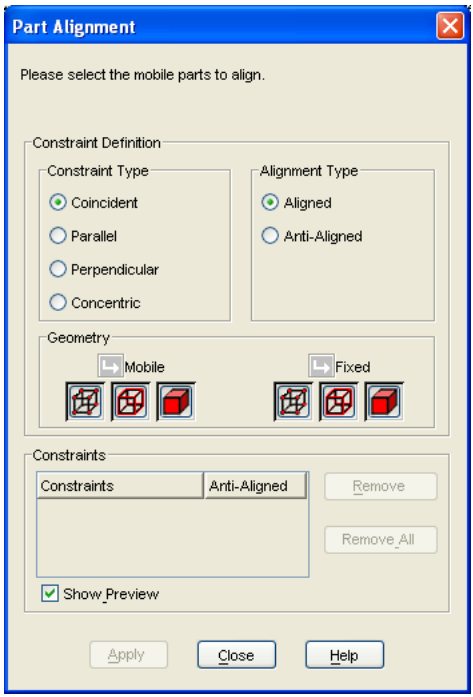
Con - **Con**centric

Aligning Model Parts

Note To see a preview of the alignment before applying the change, click **Show Preview**.

1 Select **Analysis > Part Alignment** from the AutoVue main menu.
The **Part Alignment** dialog appears.

Note You can also click the Part Alignment button on the AutoVue toolbar or right-click in the workspace and select **Part Alignment**. In the **Model Tree**, you can select the model part(s), then right-click and select **Part Alignment** from the pop-up menu.



2 Select the **Constraint Type** that you want to perform.
See Part Alignment Constraints

3 Select the **Alignment Type**.
Select **Aligned** if you want to align the **Mobile** point on the model part to the **Fixed** point on the model.
Select **Anti-Aligned** if you want to align the **Mobile** point on the model part so that it is opposite to the **Fixed** point on the model.

4 Select a model part. The **Mobile** arrow is enabled.

5 Click **Vertex** , **Edge** or **Face** to select the type of geometrical point you want to click on the model part to be aligned.

All instances of the selected geometrical point appear highlighted.

6 Click the geometrical point on the selected model part.
The geometrical point is highlighted.

The **Fixed** arrow  is enabled.

7 Click **Vertex**  , **Edge**  or **Face**  to select the type of geometrical point you want to click on the model part to remain fixed.

8 Click the geometrical point on the model.

9 Click **Apply**.
The model part is aligned according to the chosen options. The **Constraint Type** and the types of geometrical points aligned appear under **Constraints.**

Note To remove a constraint, select the **Constraint** that you want to remove, then click **Remove**. To remove all part alignments, click **Remove All**.

10 Click **Close** to close the **Part Alignment** dialog.

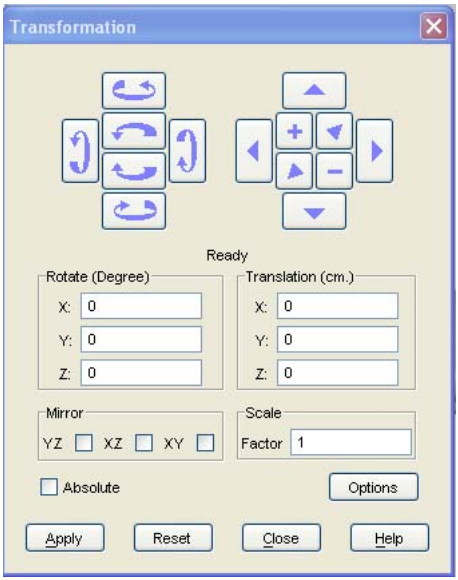
Note If you did not apply the changes before clicking **Close**, a message appears indicating **Do you wish to apply the constraint(s)?**

Transformation

With the **Transformation** option, you can rotate, scale or translate a model or any selection of model parts, by using the illustrated buttons or by entering X, Y or Z values.

To access the **Transformation** dialog, you can select **Analysis > Transform > Apply** from the AutoVue menu, or click the Transform button  on the AutoVue toolbar. From the **Model Tree**, you can select the model part(s), then right-click and select **Transform > Apply** from the pop-up menu.

The buttons on the left correspond to the rotational moves along the three axes. The buttons on the right correspond to the translational moves along the three axes and the scaling up and down.



Transforming a Model using Illustration Buttons

1 Select **Analysis > Transform > Apply** from the AutoVue main menu. The **Transformation** dialog appears.

Note You can also click the Transform button on the AutoVue toolbar.

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

Note To select more than one model part, press the **Shift** or **Ctrl** key while selecting.

Note If no part is selected, transformation is applied to the entire model.

3 Click the direction buttons on the left-hand side to rotate the model in the direction that you want.

Click the translate buttons on the right-hand side to move the model in the direction that you want.

To scale the model, click  to scale up or click  to scale down.

Note To repeat a transformation, continue clicking the illustrated button.

4 To set the translate, rotate and scale increments, click **Options**. The **Options** dialog appears.
Enter the **Translate Increment** value in inches.
Enter the **Rotate Increment** value in degrees.
Enter the **Scale Increment** value, then click **OK**.
When you click a rotation, translation or scale button, the model transforms in increments of the values entered in the **Options** dialog.

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

5 Click **Close** to close the **Transformation** dialog.
The transformation state remains displayed.

See Also ***Resetting the Transformation of a 3D Model***

Transforming a Model by Setting Values

1 Select **Analysis > Transform > Apply** from the AutoVue main menu.
The **Transformation** dialog appears.

Note You can also click the Transform button  on the AutoVue toolbar.

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

Note To select more than one model part, press the **Shift** or **Ctrl** key while selecting.

3 To rotate the model by set values, enter the **X**, **Y** and **Z** value.
To translate the model by set values, enter the **X**, **Y** and **Z** value.

4 To view a mirror image, select the checkbox beside the **Mirror** plane.

5 To scale the model by a factor, enter a **Scale Factor**.

6 Click **OK**.
To apply absolute transformation instead of incremental, select **Absolute**.

7 Click **Apply** in the **Transformation** dialog.

8 To repeat a transformation, ensure that **Absolute** is cleared, then continue clicking **Apply**.

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

9 Click **Close** to close the **Transformation** dialog.
The transformation state remains displayed.

See Also *Resetting the Transformation of a 3D Model*

Resetting the Transformation of a 3D Model

In AutoVue, you can reset a model or selected parts to their original transformation.

1 View the transformed 3D Model in AutoVue.
2 To reset the entire model to its original transformation, select **Analysis > Transform > Reset All**.
To reset selected model part(s) to their original transformation, select the part(s) on the model, then select **Analysis Transform > Reset Selected**.
The model or selected parts changes to their original transformation.
Note You can also reset selected model part(s), by clearing the checkmark beside the part(s) in the Transformation column  of the **Model Tree**.

Cross Section/Cut-through

With **Cross Section/Cut-through** 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 below 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.

Option	Description
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 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

1 Select **Analysis > Section > Define** from the AutoVue main menu. The **Define Section** dialog appears.

Note You can also click the Define Section button  on the AutoVue toolbar. In the **Model Tree**, you can select model part(s), then right-click and select **Section > Define** from the pop-up menu.

2 From the **Section Plane** drop-down list, select the orientation for the section plane.

See Section Plane Options

3 From the **Cut Options** drop-down list, select the cut option that you want.
See Cut Options

4 To define the plane position, click and drag the **Plane Position** slider to the position you want.
Click **Dynamic**, if you want the plane to move simultaneously with the slider.

5 Select **Show Plane**, if you want the section plane visible.

6 Select **Fill**, if you want the section plane filled.
The section plane is displayed as filled and the **Section Area** is calculated.
Note **Fill** color can be set from the **Configuration** dialog.

7 From the **Measured Units** drop-down list, select the unit in which you want to measure the section surface.

8 Click **Close** to close the **Define Section** dialog.

9 Once you have defined the section plane and closed the **Define Section** dialog, you can access the cut options from the **Section** menu.

10 To remove a previously defined section, select **Section > Remove** from the AutoVue main menu.
The section plane and cut section disappears instantly.

Exploding

In AutoVue, users can use the **Explode** option to better understand the structure of an assembly and to analyze the dismount capability. The user 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 drop-down list. • **Selected parts** - the number of available levels to be added to the drop-down 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 the >> button brings the explosion level to the (n-1) level if current explosion level is between n-1 (inclusive) and n level (exclusive). • **Implode -** pressing the << button 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

1 Select **Analysis > Explode** from the AutoVue main menu.
2 Select the parts that you want to explode.

Note To select more than one model part, press the **Shift** or **Ctrl** key while selecting.

To explode the entire model, ensure that no parts are selected.

3 From the **Maximum Depth** drop-down 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. Set by default.

5 Select **Show Arrows** if you want to display the arrows.

6 To explode the model, click the >> button.
The whole model or the select model parts explode.
See Saving an Exploded View of a 3D Model

Note To implode the model, click the << button. To manually change the explosion state, click and drag the scroll bar.

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

The explode state remains displayed and theTransformation column  in the **Model Tree** is updated indicating the exploded parts.

See Also 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.

1 Explode a 3D Model.
See Exploding a 3D Model

2 Click **Save View** in the **Explode** dialog.
The **Add User Defined View** dialog appears.

3 Enter a **View Name**.

4 Click **OK**.
The view is saved and appears in the **Views** tree under **User Defined Views**.

5 Click **OK** to close the **Explode** dialog.
The exploded state is saved.

See Also Resetting the Transformation of a 3D Model

User-defined Coordinate Systems

In AutoVue you have the option of creating customized three-axis coordinate systems. As with the **Global Axes** and the **World Coordinate System**, the user-defined axes are represented by a red X-axis, a green Y- axis and a yellow Z-axis.

When you set a User-defined Coordinate System (UCS) as the active axes, operations such as measuring, transforming a model, or defining a viewpoint are applied with respect to these axes.

You can create multiple UCS's for a 3D file. For multiple-page 3D files, you can even set active a different UCS for each page.

Position Options

When defining a UCS, you can define the point where you want to place the user coordinate system. The options are:

Position	Description
(x, y, z) Coordinates	If selected, enter a value to specify where the three axes are positioned.
Vertex	If selected, all vertices are highlighted on the model. A snap box appears when you move the cursor on a vertex. Click the vertex point where you want to place the user coordinate system. The **x**, **y** and **z** coordinate values for the vertex are displayed. The UCS axes move to the new position.
Edge	If selected, all edges are highlighted on the model. A snap box appears when you move the cursor near an edge. Click at a point on an edge where you want to place the user coordinate system. The **x**, **y** and **z** coordinate values for the edge point are displayed. The UCS axes move to the new position.
Midedge	If selected, all edges are highlighted on the model. A snap box appears when you move the cursor near the halfway point of an edge. Click at the mid-edge point where you want to place the user coordinate system. The **x**, **y** and **z** coordinate values for the mid-edge point are displayed. The UCS axes move to the new position.

Position	Description
Arc Center	If selected, all arcs and circles are highlighted on the model. A snap box appears when you move the cursor near the center of a circle or arc. Click on the arc center where you want to place the user coordinate system. The **x**, **y** and **z** coordinate values for the arc center are displayed. The UCS axes move to the new position.
Face	If selected, when you move the cursor along a model face, the face is highlighted. Click at a point on the face where you want to place the user coordinate system. The **x**, **y** and **z** coordinate values for the arc center are displayed. The UCS axes move to the new position.

Orientation Options

When defining a UCS, you can define the orientation for the coordinate system. The options are:

Position	Description
(x, y, z) Coordinates	If selected, enter a value to specify the UCS orientation according to one or more of the three axes.
Edge Tangent	If selected, all edges are highlighted on the model. A snap box appears when you move the cursor near an edge. First select the axis that will determine the UCS orientation. Click at a point on an edge to define the orientation. The **x**, **y** and **z** values for the edge point are displayed. The UCS axes are reoriented to the selected edge.
Face Normal	If selected, when you move the cursor along a model face, the face is highlighted. First select the axis that will determine the UCS orientation. Click at a point on an face to define the orientation. The **x**, **y** and **z** values for the face point are displayed. The UCS axes are reoriented to the selected face.

Position	Description
2 Vertices	If selected, all vertices are highlighted on the model. A snap box appears when you move the cursor on a vertex. First select the axis that will determine the UCS orientation. Click two vertex points to define the orientation. The **X**, **Y** and **Z** values for the vertices are displayed. The UCS axes are reoriented to the selected vertices.
3 Vertices	If selected, all vertices are highlighted on the model. A snap box appears when you move the cursor on a vertex. First select the axis that will determine the UCS orientation. Click three vertex points to define the orientation. The **X**, **Y** and **Z** values for the vertices are displayed. The UCS axes are reoriented to the selected vertices.

Defining a User Coordinate System

1 Select **Analysis > User Coordinate Systems** from the AutoVue main menu.
The **User Coordinate Systems** dialog appears.

Note You can also click the User Coordinate Systems button  on the AutoVue toolbar.

2 Click **New**.
The name of the User-defined Coordinate System (**UCS1** for the first one) appears in the **Coordinate Systems** box.

3 To change the name, type the new **Name** that you want for the UCS.

4 Click the **Position** tab to define the origin of the UCS.

5 From the **Define from** drop-down list, select the point where you want the UCS axes to appear.
See Position Options

6 Click the **Orientation** tab to define the orientation of the UCS.

7 From the **Define from** drop-down list, select the point where you want to orient the UCS.
See Orientation Options

8 To hide the UCS axes on the model, clear the checkbox beside **Show Trihedron**. Enabled by default.

9 Click **OK**.

The newly active trihedral coordinate system appears bolder and larger and its name appears in the User Coordinate Systems drop-down list on the AutoVue toolbar.

Note If there is more than one coordinate system for this file, select which coordinate system the new UCS will be **Relative To**. Otherwise, the new UCS will be calculated with respect to the World Coordinate System.

10 Click **OK** to close the **User Coordinate Systems** dialog.

Modifying a User-defined Coordinate System

1 Select **Analysis > User Coordinate Systems** from the AutoVue main menu.
The **User Coordinate Systems** dialog appears.

Note You can also click the User Coordinate Systems button  on the AutoVue toolbar.

2 From the **Coordinate Systems** list, select the UCS that you want to modify.
The settings you configured for the UCS are displayed.

3 To change the **Name**, highlight the name and enter the new name.

4 To change the **Position**, click the **Position** tab, then select the point where you want the UCS axes to appear from the **Define From** drop-down list.
See Position Options

5 To change the **Orientation**, click the **Orientation** tab, then select the point where you want to orient the UCS from the **Define From** drop-down list.
See Orientation Options

6 To show the UCS axes, select **Show Trihedron**. To hide the UCS axes, clear the checkmark.

7 To change the UCS that it is relative to, select the UCS from the **Relative To** drop-down list.

8 Click **OK** to close the **User Coordinate Systems** dialog.
If you modified the **Position** or **Orientation**, the UCS axes dynamically move to the new position. A change in **Show Trihedron**, will make the UCS axes appear or disappear accordingly. A **Name** change, will display the new name in the **Coordinate Systems** list and in the drop-down list next to the UCS button  on the toolbar.

Activating a Coordinate System

When you set a user-defined coordinate system (UCS) as the active axes, operations such as measuring, transforming a model or defining a viewpoint are applied with respect to these axes.

1 From the **User Coordinate Systems** dialog, select the **Coordinate System** that you want to use.
2 Click **Set Active**.
3 Click **OK** to close the **User Coordinate Systems** dialog.
The newly active trihedral coordinate system appears bolder and larger on the model and its name is displayed in the drop-down list next to the UCS button  on the AutoVue toolbar.

Note You can also set a different coordinate system from the **User Coordinate Systems** drop-down list next to the UCS button  on the toolbar.

Deleting a User Coordinate System

1 From the **User Coordinate Systems** dialog, select the **Coordinate System** that you want to delete.
2 Click **Delete**.
Note The UCS disappears from the list and from the **User Coordinate Systems** drop-down list next to the UCS button  on the toolbar.
3 Click **OK** to close the **User Coordinate Systems** dialog.

Interference Checks

In AutoVue, you can check for spatial interference of any two model parts.

Interference Check Options

With the **Interference Checking** option, you can perform different interference checks. These checks are:

Check	Description
All Against All	Check for interference of any of the parts in the model.

Check	Description
Set Against the Rest	Check where a model part or set of model parts interfere with other model part(s).
Set Against Itself	Check where a model part or set of model parts interfere with itself.
Set 1 Against Set 2	Check where a model part or set of model parts interfere with another model part or set of model parts.

Performing an Interference Check

1 Select **Analysis > Interference Checking** from the AutoVue main menu.
The **Interference Checking** dialog appears.

Note You can also click the Interference Checking button on the AutoVue toolbar.

2 Click the type of interference check that you want to perform.
See Interference Check Options

3 If you selected **Set Against the Rest** or **Set Against Itself**, select one or more model parts on the model or from the **Model Tree**.
Note To select more than one model part, press the **Shift** or **Ctrl** key while selecting.
The model part(s) appear in the **Selected Set** list and are highlighted on the model and in the **Model Tree**.
Note To clear the last set of selected model part(s), click **Clear**.

4 If you selected **Set 1 Against Set 2**, click the **Set 1** arrow , then select one or more model parts on the model or from the **Model Tree**.
The model part(s) appear in the **Set 1** list and are highlighted on the model and in the **Model Tree**.
Note To clear a selected part, press the **Ctrl** key and click on the part on the model. To clear the last set of selected model part(s), click **Clear**.

Click the **Set 2** arrow then select one or more model parts on the model or from the **Model Tree**.
The model part(s) appear in the **Set 2** list and are highlighted in a different color on the model and in the **Model Tree**.

5 Enter a **Minimum Clearance Distance**. Default value is 0.

Note Adding a **Minimum Clearance Distance** not only checks if the two model parts interfere, but also checks if they are located within a set minimum distance.

6 From the **Units** drop-down list, select the unit for the clearance distance.

7 To configure the interference check results, click **Options**.
The **Options** dialog appears.
See Interference Check Results Options

8 Click **Compute** in the **Interference Checking** dialog.
Compute button changes to **Stop**.
Note Section plane is not taken into consideration when computing an interface check.
Note To stop the Interference Checking process at any point, click **Stop**.
When the process is complete, all the instances of interfering pairs appear in the **Results** list. The number of interfering pairs found appears in the **Description** box.
Note To view a pair's interference information, click the corresponding **Results** line. The information appears in the **Description** box and AutoVue zooms into the interfering area.

9 To save the results, click **Export**.
The **Export Results** dialog appears.

10 Specify the file name and the directory where you want to save the file.

11 Click **Save**.
AutoVue saves the results in a **csv** (Comma Separated Values) file.
Note To perform another interference check click **Reset**, then click the type of interference check.

12 Click **Close** to close the **Interference Checking** dialog.

Interference Check Results Options

With AutoVue, you can configure the results of an Interference Check. The configuration options are:

- **Actions** - provides a close-up look at how model parts interfere
- **Selected Pair** - configure if the selected pair from the **Results** list should display opaque, transparent or hidden

- **The Rest** - configure if the other parts of the model should display transparent or hidden while the selected pair is displayed

Option	Select	Description
Actions	Zoom To	Once the interference check is computed and a result set is selected from the list of results, AutoVue zooms to the result.
	Show Intersection	Once the interference check is computed and a result set is selected from the list of results, AutoVue shows where the intersection of model parts occur.
Selected Pair	Opaque	The selected pair from the list of results appear opaque.
	Transparent	The selected pair from the list of results appear transparent.
	Hide	The selected pair from the list of results appear hidden.
The Rest	Transparent	The rest of the model appears transparent while the selected pair is displayed.
	Hide	The rest of the model is hidden while the selected pair is displayed.

Comparing 3D Files

Note When comparing 3D files in a non-integrated environment we recommend that you use 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 regarding UNC file names and the server protocol, ***see Installation and Administration Manual***.

AutoVue provides the ability to compare two 3D files or two sets of entities from the same file or different files and display color-coded comparative data. When you compare two files, AutoVue displays a **Compare Tree** and three windows, the first containing the base file, the second containing the file you compared the base against, and the third containing the comparison results.

The **Compare Tree** displays the model's hierarchy with a **State** column displaying different icons representing the results of the file comparison. The icons indicate if an model part has been added , modified , moved deleted .

In the **Comparison Result** window, results are displayed in different colors to differentiate the results of the file comparison. The comparison options and corresponding colors are:

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

To compare files:

1 View the base file in AutoVue.
2 Select **Analysis > Compare** from the AutoVue main menu.
The **File Open** dialog appears.
3 Enter the **File Name** or click **Browse** to locate the file that you want to compare with the current active file.
4 Click **OK**.
AutoVue displays the **Compare Tree** and three windows. The first window displays the base file, the second displays the compare file and the third displays the comparison results.
5 To view the properties of a modified or moved entity in the base file and in the compare file, select the modified or moved entity from the **Compare Tree**, then right-click and select **Entity 1 Properties**.
The **Entity Properties** dialog appears displaying the properties of the entity in the base file.
See Entity Properties
Select the entity again, then right-click and select **Entity 2 Properties**.
The **Entity Properties** dialog display the properties of the entity in the compare file.
6 To compare the results of the entity in the base file to the compare file, select the entity from **Compare Tree**, then right-click and select **Compare results**.

The **Compare results** dialog appears.
To view the attribute differences, click **Attributes**.
To view the transformation difference, click **Transformation**.
Note If there are not attribute or transformation differences, the buttons are disabled.

7 Click **OK** to close the **Compare results** dialog.
Note 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.

8 To exit **Compare** mode, select **File > Exit Compare Mode** from the AutoVue main menu.
The base file appears in the workspace.
Note You can also exit **Compare** mode by right-clicking in any of the windows and selecting **Exit Compare Mode** from the pop-up menu.

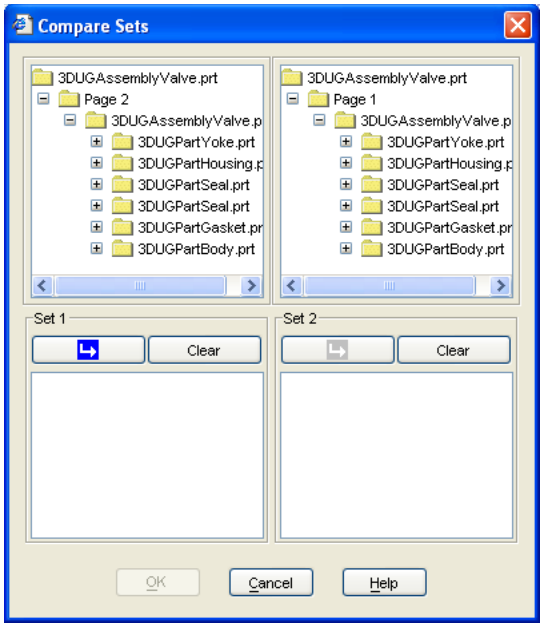
Comparing Entity Sets

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.

1 Select the files you want to compare.
See Comparing 3D Files

2 Select **File > Compare Sets**.

The **Compare Sets** dialog appears.



3 Click **Set 1**. Set by default.

4 Select the entities from either the tree on the left or the tree on the right.

Note The tree on the left is the base file and the tree on the right is the compare file.

The selected entities appear in the **Set 1** list.

5 Click **Set 2** point.

6 Select the entities from the other tree that you did not select for **Set 1**.

The selected entities appear in the **Set 2** list.

Note 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.

7 Click **OK**.

The **Compare Tree** and the three windows are updated with the set comparison results.

8 To restore the comparison files, select **File > Compare Files**.

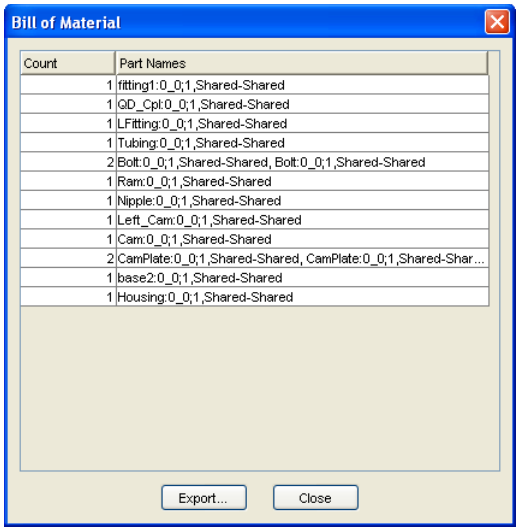
The files appear in the three windows.

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:

1 Select **Analysis > Bill of Material** from the AutoVue main menu.
The **Bill of Material** dialog appears.



2 To sort the list numerically, click the **Count** column header.
To sort the list alphabetically, click the **Part Names** column header.

3 To view a part or parts on the model, select the part(s) from the **Part Names** list.

Note To select more than one part, press the **Shift** or **Ctrl** key while selecting.

The parts are highlighted on the model and in the **Model Tree**.

4 To save the Bill of Material, click **Export**. The **Save As** dialog appears. Enter a **File name** or browse to locate the directory where you want to store the file, then click **Save**.
AutoVue saves the results in a **csv** (Comma Separated Values) or **xml** file.

5 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, 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. **Note** 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. **Note** Option is enabled if **Type** is set to **PMI** or **All**.
	PMI Type	Specify the specific PMI attribute you want to search. **Note** Option is disabled if **PMI Filter** is set to **All**.

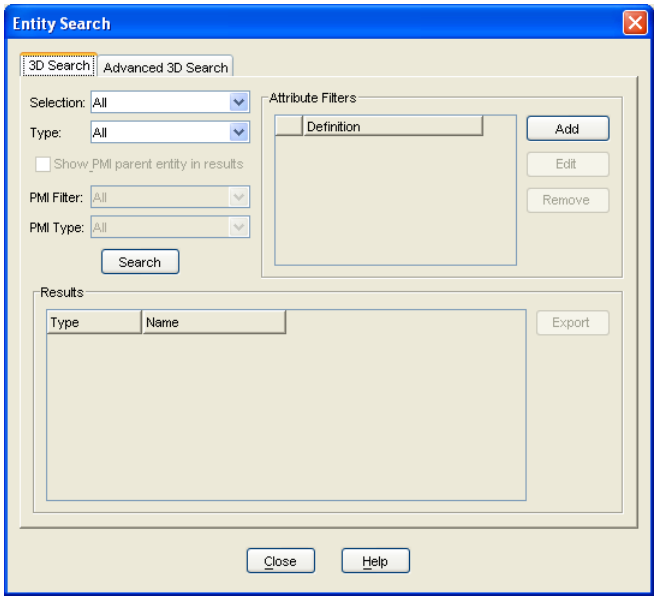
Where	Option	Description
Attribute Filter dialog	Name	Specify the specific attribute you want to search. The available attributes are: • Colour • Density • Display_Mode • Filepath • Layer_ID • Mesh_Resolution • Name • Shading_Style • Translucency • Visibility **Note** Additional attributes may be available depending on the file selected.
	Any Value	When selected, AutoVue searches for any value of the selected attribute. **Note** The attribute value options are disabled. If cleared, specify the values for the selected attribute you want to search. **Note** The attribute value options correspond to the selected attribute.
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

1 Select **Analysis > Entity Search** from the AutoVue menu.
The **Entity Search** dialog appears.

Note You can also click the Entity Search button  on the AutoVue toolbar.

2 Click the **3D Search** tab.



3 From the **Selection** drop-down 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** drop-down list, select the type of entity that you want to search.
Note For 3D Models that have PMI information, the PMI search features are enabled.

5 Select the checkbox beside **Show PMI parent entity in results** if you want only the parent entity of the selected PMI entity or entities to appear in the **Results** list.
Clear the checkmark if you want all PMI entities to appear in the **Results** list.
Note Option is enabled if **Type** is set to **PMI** or **All** and files has PMI entities.

6 From the **PMI filter** drop-down 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**.

7 From the **PMI type** drop-down list, select the type of entity that you want to search.

Note **PMI type** is disabled if **PMI Filter** is set to **All**.

See ***Saving Search Results***

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

Performing an Attribute-based Search

From the **3D Search** dialog, you can search for entities by their attributes, such as color, density, shading, etc.

1 Select **Analysis > Entity Search** from the AutoVue menu.
The **Entity Search** dialog appears.

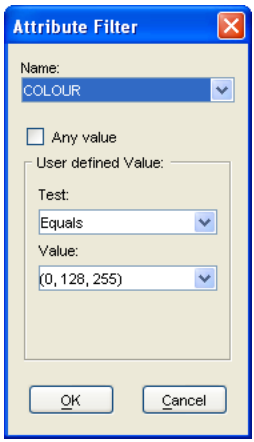
Note You can also click the 3D Entity Search button on the AutoVue toolbar.

2 Click the **3D Search** tab.

3 Select the search criteria that you want to use for your search.

See ***Performing a Search***

4 Click **Add**. The **Attribute Filter** dialog appears.



5 From the **Name** drop-down list, select the attribute that you want to search.
The **Value** group box appears corresponding to the selected attribute.

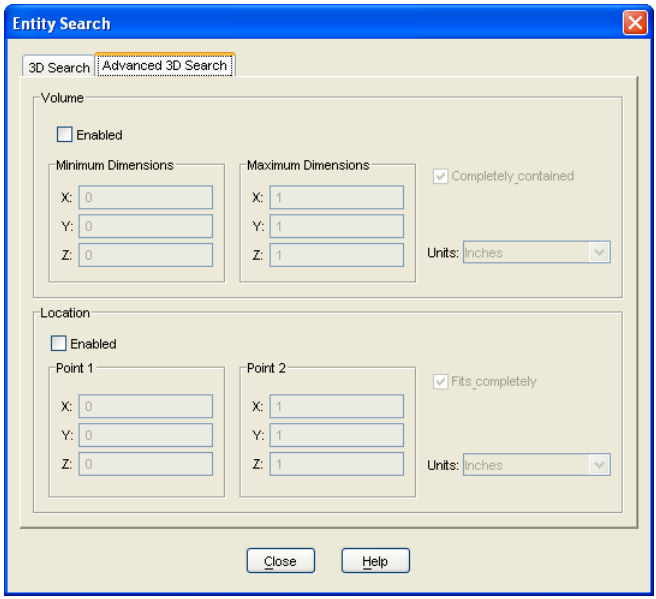
6 Select the checkbox beside the **Any Value** if you want to search by any value. The **Value** options are disabled.

7 Clear the checkbox if you want to search by a specific value, then from the **Value** drop-down list(s), select the values that you want to search.
8 Click **OK**.
Note To add more than one **Attribute Filter**, repeat steps 4 to 5.
9 To edit an **Attribute Filter**, select the filter and click **Edit**.
The **Attribute Filter** dialog box appears allowing you to edit the filter.
10 To disable an **Attribute Filter**, clear the checkbox beside the filter(s) that you want to disable.
To enable an **Attribute Filter**, select the checkbox beside the filter(s).
11 To remove an **Attribute Filter**, select the filter and click **Remove** in the **Entity Search** dialog.
The **Attribute Filter** disappears from the list.
12 In the **Entity Browser**, click **Search**.
A list of entities matching the search criteria appear in the **Results** list displaying the **Type** and **Name**.
See Saving Search Results
13 Click **Close** to close the **Entity Search** dialog.

Performing an Advanced 3D Search

From the **3D 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.

1 From the **Entity Search** dialog, click the **Advanced 3D Search** tab.



2 To search for entities by **Volume**, select the checkbox beside **Enabled**.

Note If you want to search for entities that match all three dimensions, select the checkbox beside **Completely contained**. If the checkmark is cleared, then an entity only has to match one dimension in order to appear in the **Results** list.

3 Enter the **Minimum** and **Maximum Dimensions**.

4 From the **Units** drop-down list, select the unit you want to set for the **Volume** dimensions.

5 To search for entities by **Location**, select the checkbox beside **Enabled**.

Note If you want to search for entities that match all three dimensions, select the checkbox beside **Fits completely**. If the checkmark is cleared, then an entity only has to match one dimension to appear in the **Results** list.

6 From the **Units** drop-down 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**.

A list of entities matching the search criteria appears in the **Results** list.

See Saving Search Results

10 Click **Close** to close the **3D Search** dialog.

Saving Search Results

1 Perform an 3D entity search.
See ***Performing a Search***
Performing an Attribute-based Search
Performing an Advanced Search
2 Click **Search**.
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**.
The **Save As** dialog appears.
4 Specify the path where you want to store the file and enter the **File name**, then click **Save**.
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.

You can access the measure options from the **Analysis > Measure** menu. The options are:

Button	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.

Button	Name	Description
	Minimum Distance	Measure minimum distance between any two vertices, edges, mid-edges, arc centers, faces or any combination of these entity types.
	Edge Length	Measure the precise length of a edge.
	Face Surface	Measure the precise surface area.
	Vertex Coordinates	Provide the coordinates of each vertex.

3D Snapping Modes

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

The **Snapping Modes** allow you to snap to vertices, edges, faces, planes and arcs.

Button	Snap to	Description
	Edge	Edges are highlighted on the model. When moving the cursor over an edge, a snap circle appears.
	Face	Faces are highlighted when you move the cursor over a face and a snap triangle appears.
	Vertex	Vertices are highlighted on the model. When moving the cursor over a vertex, a snap box appears.

Measuring Distance

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

Note When measuring the distance between faces, if the faces are parallel, AutoVue compares the shortest distance between the faces.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the AutoVue toolbar.

2 Click the **Distance** tab.
3 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure.
4 Click the **From** arrow  to select the entity type that you want to measure from.
5 Select the **Snapping Modes** that you want to use for measuring.
See 3D Snapping Modes
All entities of the selected entity type are highlighted on the model.
6 On the model, select the entity you want to measure from.
Note If you want to take more than one measurement from the same starting point, click **Fix Position**.
7 Click the **To** arrow  to select the entity type that you want to measure to.
8 Select the **Snapping Modes** that you want to select as the end point for the measurement.
All entities of the selected entity type are highlighted on the model.
9 On the model, select the entity you want to measure to.
The points are joined by a line. The measured distance, Delta X, Delta Y and Delta Z appear in the **Measurement** dialog.
Note Click **Reset** to take another measurement.
10 Click **Close** to close the **Measurement** dialog.

Calibrating Distance

1 Measure a distance.
See Measuring Distance
2 In the **Measurement** dialog, click **Calibrate**.
3 The **Distance Calibration** dialog appears displaying the measured distance.
4 In the **Measured Dist** drop-down list, select the unit to which you want to calibrate the distance.
5 Click **Calibrate to** and enter a value if you want to calibrate the distance to a value.
Click **Set Factor** and enter a value if you want to calibrate the distance by a factor.

6 Click **OK**.
The calibration result appears in the **Measurement** dialog.
7 Click **Close** to close the **Measurement** dialog.

Measuring Minimum Distance

With the **Minimum Distance** option you can measure the minimum distance between model parts, as well as any two points from the selection sets: vertices, edges, mid-edges, arc centers, faces or any combination of entity types.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Minimum Distance button  on the AutoVue toolbar.

2 Click the **Min. Distance** tab.
3 Click  **Set 1** point.
4 Click **Entity** if you want to measure the distance between model parts. **Snapping Modes** are disabled.
Click **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.
The model part(s) appears in the list under **Set 1** and are highlighted on the model and in the **Model Tree**.
If you selected **Geometry**, select the **Snapping Modes** that you want to use for measuring.

See 3D Snapping Modes

All entities of the selected entity type are highlighted on the model.
Select the entity or entities on the model.
The selected entities appears in the list under **Set 1** and are highlighted on the model.

Note 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 **Ctrl** key and left-click the part or entity type.

6 Click  **Set 2** point.
7 Repeat step 5.
The model part appears in the list under **Set 2**.
8 From the **Measured Min Distance** drop-down list, select the unit in which you want to measure the distance.
9 Click **Compute**.

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 **Position1** and X, Y and Z coordinates for **Position2** appear in the **Measurement** dialog.

Note Click **Reset** to take another measurement.

10 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.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Angle button  on the AutoVue toolbar.

2 Click the **Angle** tab.

3 Select the **Snapping Modes** that you want to use for measuring.
Entities of the selected types are highlighted on the model.

See 3D Snapping Modes

4 To measure the angle between an entity type and a plane, select the checkbox beside **Plane**, then from the drop-down list select the plane.

5 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.

6 Click two points on the model to define the angle.
If you selected a **Plane**, click the vertex, edge or face whose angle you want to measure between the plane.
Angle arms appears indicating the angle.The measurement appears in the **Measurement** dialog.

Note Click **Reset** to take another measurement.

7 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. You can also calculates the center point location.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the AutoVue toolbar.

2 Click the **Arc** tab.
3 Select **Arc Entity** if you want to measure a predefined arc.
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.
Snapping Modes are enabled.
Select the **Snapping Modes** that you want to use for measuring.
See 3D Snapping Modes
5 From the **Arc Info Dist. Units** drop-down list, select the unit in which you want to measure the arc distance.
6 From the **Angles Units** drop-down 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.
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.
If you selected **Arc Entity**, click the edge of the arc that you want to measure.
The arc is highlighted. The measurements for center point coordinates, radius, ratio, arc length, angles start, end and sweep appear in the **Measurement** dialog.
Note Click **Reset** to take another measurement.
8 Click **Close** to close the **Measurement** dialog.

Calibrating an Arc

1 Measure an arc.
See Measuring an Arc
2 In the **Measurement** dialog, click **Calibrate**.
3 The **Radius Calibration** dialog appears displaying the measured distance.
4 From the **Measured Rad.** drop-down list, select the unit to which you want to calibrate the distance.
5 Click **Calibrate to** and enter a value if you want to calibrate to a value.
Click **Set Factor** and enter a value if you want to calibrate by a factor.
6 Click **OK**.
The calibration results appear in the **Measurement** dialog.
7 Click **Close** to close the **Measurement** dialog.

Measuring Vertex Coordinates

The **Vertex Coordinates** option provides the coordinates of vertices on the model.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Vertex Coordinates button  on the AutoVue toolbar.

2 Click the **Vertex** tab.
All vertices on the model are highlighted.

3 From the **Coordinates Units** drop-down list, select the unit in which you want to measure.

4 Move the cursor over the highlighted vertex that you want to measure.
The **X**, **Y** and **Z** coordinates appear in a tooltip.

5 Select the vertex.
The **X**, **Y** an **Z** coordinates appear in the **Measurement** dialog.
Note Click **Reset** to take another measurement.

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.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Edge Length button  on the AutoVue toolbar.

2 Click the **Edge Length** tab.
All edges on the model are highlighted.

3 From the **Units** drop-down list, select the unit in which you want to measure the edge length.
Note If you want to cumulate the measurement of more than one edge, click **Cumulative**.

4 Click the edge that you want to measure.
The edge length measurement appears in the **Measurement** dialog.
Note Click **Reset** to take another measurement.

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

Measuring Face Surface

Use the **Surface** option to measure the surface area of a entity face or an entire entity on the model.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Face Surface button  on the AutoVue toolbar.

2 Click the **Surface** tab.
Select **Face Surface** if you want to measure the surface area of a face on an entity.
Select **Entity Surface** if you want to measure the surface area of an entire entity.

3 From the **Measure Surface Units** drop-down list, select the unit in which you want to measure the surface.

Note If you want to cumulate the measurement of more than one surface, click **Cumulative**.

4 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.
The face surface is highlighted. The measured surface appears in the **Measurement** dialog.

5 If you selected **Entity Surface**, click the entity that you want to measure.
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.

6 To measure the face surface of the entity, click on the highlighted body.
A pop-up list appears displaying the entity and its parents.
Select the entity or a parent from the pop-up list.
The selected entity is highlighted and the measured surface appears in the **Measurement** dialog.

Note To measure the face surface of a parent, select the parent from the pop-up list.

Note Click **Reset** to take another measurement.

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

Configuring AutoVue

Use the configuration options to configure the AutoVue workspace for different groups of file formats or for all files in general. For example, you can set different background colors for EDA, 2D, 3D, or Office files. You can also set paths to locate external resources such as fonts, symbols, or XREFS, or configure measurement options.

You can access the configuration options using the Configuration dialog. To open the Configuration dialog, select **Options > Configuration** from the AutoVue main menu.

For details on each configuration option, see:

General Options
Configuring AutoVue for 2D Files
Configuring AutoVue for 3D Files
Configuring AutoVue for EDA Files
Configuring Background Colors for Graphic Files
Configuring Background Colors for Desktop Office

General Options

To access the **General** configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **General** in the tree.

Configuring Options for CAD Files

Configure how you want to display w.r.t. text, dimensions, line styles, and so on for CAD files:

Option	Description
Text	• Select to display text entities. • Clear to hide text entities.
Dimensions	• Select to display all dimensional entities. • Clear to hide simple dimension entities.

Option	Description
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 load Xrefs automatically. • Clear to keep Xrefs from loading automatically.

Raster Files

Select **Full Resolution** to display raster files at full resolution, or 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 cleared, AutoVue renders the tiles when requested, that is, only when you zoom in on parts of the file).

Configuring XRef Paths

Xref Paths are the directory paths for any external reference files associated with 2D, 3D, or EDA files.

1 Select **Options > Configuration** from the main menu.
The **Configuration** dialog appears.

2 Click **General > XRef Paths** in the tree.
3 Click the **Add** button
The **Add Path** dialog appears.
4 Enter the directory path, or scroll to locate the directory where the external reference files are located.
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".
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".
5 Click **OK**.
The directory path appears.
Note To add more than one path to the list, repeat steps 3 to 5.
6 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.
7 To remove a path, select the path and click **Remove**.
8 Click **OK** to close the **Configuration** dialog.

Configuring Font Paths

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

1 Select **Options > Configuration** from the AutoVue main menu.
The **Configuration** dialog appears.
2 Click the **General > Font Paths** in the tree.
3 Click the **Add** button.
The **Add Path** dialog appears.
4 Enter the directory path or scroll to locate the directory where the external font files are located.
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".

5 Click **OK**.
The directory path appears.
Note To add more than one path to the list, repeat steps 3 to 5.
6 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.
7 To remove a path, select the path and click **Remove**.
The selected path disappears from the list.
8 Click **OK** to close the **Configuration** dialog.

Measurement

To access the common **Measurement** configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **General > Measurement** in the tree.

The **Measurement** options let you define the default measurement units and the number of decimal places, 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 Units	Lets you specify the default unit for measuring. Select a unit from the drop-down list.

Configuring the Base Font for Text Files

1 Select **Options > Configuration** from the main menu.
The **Configuration** dialog appears.
2 Select **General > Base Font** from the tree.
3 Select a font from the **Font** list.
4 Select a font size from the **Size** list.
5 Select the **Bold** option, **Italic** option, or both options to change the font style.
Note You can see a preview of the text in the **Sample** area.
6 Click **OK** to apply the font change and close the **Configuration** dialog.

Note The font changes will only apply to text files.

Configuring AutoVue for 2D Files

You can configure background color and snap settings for 2D files. To access the 2D configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **2D** in the tree.

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.

Access this option from the **Configuration** dialog. Select **Options > Configuration** from the main menu, then click **2D** in the tree.

Configuring Colors

The **Colors** settings let you modify colors for 2D files. In the **Configuration** dialog, select **2D > Colors** in the tree 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

In AutoVue, there are configuration options that you can set to customize the work environment when working with 3D files. To access the 3D configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **3D** in the tree.

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. **Note** 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. **Note** 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.

Menu Option	Description
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.

Note A higher frame rate results in a lower resolution.

Model

The **Model** configuration options let you control the streaming method, the dynamic load mesh resolution, and the visibility of model parts.

To access the Model configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **3D > Model** in the tree.

Dynamic Loader

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

When the option is active, 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 not active, 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.

Note For more information, ***see System Administration Guide***.

Dynamic Load Mesh Resolution

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

When you enable **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.

Note For more information, ***see the System Administration 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. **Note** To display model parts, select the part(s) from the **Model Tree**.

PMI Filtering

In the **Configuration** dialog, select **3D > PMI** in the tree to display the **PMI** options.

The **PMI Filtering** options let you configure which types of product and manufacturing information to display. The checkboxes in the **Tree** column let you select which PMI entity types display in the 3D model tree. The checkboxes in the **View** column let you select which PMI entity types display in the workspace. For more information on PMI Filtering, see **PMI Filtering**.

The following **PMI Text Rendering Style** options let you define the style for the PMI text:

Option	Description
Native Setting (from file)	PMI text is displayed with the default setting
3D	PMI text is displayed in 3 dimension

Option	Description
Flat-to-screen	PMI text is displayed in 2 dimension

Configuring Color

With the **Color** options, you can configure the color to allow easier viewing of 3D file details. The Color options are grouped under **Common**, **Section Highlight**, and **Geometry Highlight**, as follows:

Common

Option	Description
Background	Set the background color for 3D files.
Selection	Set the color when selecting a model or model parts.
Entity Default	Set the default color to use when the model color is not readable.
Measurement	Set the color for measurements.
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.

Miscellaneous

In the Configuration dialog, select **3D > Miscellaneous** in the tree to display the Miscellaneous options.

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.

Model Tree

Enter a different value in the **Tree Level** field to define the level at which you want the **Model Tree** to collapse when opening a 3D file. The default is 3 levels.

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.

Configuring AutoVue for EDA Files

In AutoVue, there are configuration options that allow you to customize your work environment when working with EDA files. To access the EDA configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **EDA** in the tree.

See Customizing Selections
Displaying Tooltips
Modifying 3D View
Synchronizing Layers when Comparing Files
Configuring Zoom Behavior when Cross Probing
Modifying Colors

Customizing Selections

Configure how you want selected components to be highlighted. Click **Options > Configuration** to open the Configuration dialog, then click **EDA** in the tree. 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. To change the highlight color, see **Modifying Colors**.
- **Dim Unselected:** Activate this option to dim all the entities that are not selected. Selected entities retain their original entity color.

Note: You can also select the **Dim Unselected** button  in the main toolbar.

When the Dim Unselected option is enabled, you can set the dimness level for entities that are not selected. Drag the slider to the right to increase the dimness level and to the left to decrease the dimness level. The icon to the right of the dimness settings in the Configuration dialog lets you preview the dimness level.

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.

1 Click **Options > Configuration** to open the Configuration dialog.
2 Click **EDA** in the tree.
3 Under the Mouse Hover heading, select or clear the **Show Entity Information Tooltip** option.
4 Click **OK**.

Modifying 3D View

You can configure the 3D PCB view of EDA files.

Note The Default Board Thickness and Component Height values are read-only if the EDA 3D file does not contain board and component information.

1 Click **Options > Configuration** to open the Configuration dialog.
2 Click **EDA** in the tree.
3 Enter a value in the **Default Board Thickness** field to change the board's thickness.

   Note 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.
4 Enter a value in the **Default Component Height** field to change the component height.

   Note 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.

5 To work with a different unit of measurement, select another unit from the **Default Units** menu. This is the unit of measurement used for values set in the Default Board Thickness and Default Component Height fields.
6 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.

1 Click **Options > Configuration** to open the Configuration dialog.
2 Click **EDA > Analysis** in the tree.
3 Activate the **Layer Settings** option to synchronize all layer settings when comparing files.

Note: This option is only applicable for PCB drawings.

Configuring Zoom Behavior when Cross Probing

You can configure the zoom options when you cross probe files.

1 Click **Options > Configuration** to open the Configuration dialog.
2 Select **EDA > Analysis** in the tree.
3 Select one of the following Cross Probe options:
   - Click **Keep Current Zoom Level**. When this option is active, the view of the target files remains the same while you cross probe.
   - Click **Zoom Selected** to automatically zoom in on selected entities while you cross probe. This option is enabled by default.
   - Click **Zoom Fit** to automatically adjust the contents of a file along the horizontal or vertical axis to fully display within the current window.
4 Click **OK**.

Modifying Colors

You can configure the colors for EDA files. To access these options, click **Options > Configuration** to open the Configuration dialog, then click **EDA > Colors** in the tree.

In the **Color Options** dialog, the options that you can configure are grouped under Common and 3D View as follows:

Common

Option	Description
Background	Change the background color for EDA files.
Selection	Configure the color used to highlight entities.
Min. Distance Set 1	Configure the color of the first selection set point when measuring minimum distance.
Min. Distance Set 2	Configure the color of the second selection set point when measuring minimum distance.

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.

Configuring Background Colors for Graphic Files

Specify background colors for mono raster files and color raster files.

To access these configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **Graphics** in the tree.

Configuring Background Colors for Desktop Office

Specify background colors for the following types of Desktop Office files:

- Document
- Spreadsheet
- Database
- Archive

To access these configuration options, select **Options > Configuration** from the main menu. In the **Configuration** dialog that appears, click **Desktop Office** in the tree.

Markups

AutoVue has the ability to view over 450 different file formats and to create markups for all its readable file formats without the document's authoring application.

Marking up refers to drawing and writing on an electronic document. AutoVue provides the ability to markup different formats without modifying the original file. When you create a markup for a file, the markup is created on top of the original file. Markups are saved in separate files.

If a file has existing markup files, a red light bulb  appears on the status bar at the bottom of the AutoVue workspace. Clicking this button opens the **Markup Files** dialog allowing you to select markup files to open or create a new markup file. When you open a markup file, AutoVue lays the markup over the original file.

In Markup mode you can:

- create entities such as arcs, boxes, circles, clouds, lines, arrows and polygons
- add a symbol, add information to an entity by adding text or a note
- create, name and color layers to organize your work
- create a new markup file that combines copies of selected layers of different markup files
- create measurement markup entities that can be moved or resized
- navigate markups through a hierarchy tree, view markup properties and sort the tree according to each property
- view markup layers individually or in combination

Note Markup capability is only available in certain versions of the AutoVue product. See the ***Product Variations Checklist*** for more information.

Markup Navigation Tree

When opening a file in **Markup** mode, a **Markup Navigation Tree** appears to the left of the document. If the **Markup Navigation Tree** does not appear, select **View > Show Model Tree**, then click the **Markups** tab.

The **Markup Navigation Tree** displays a hierarchy tree of markups or comments created by users. You can navigate through the markups. A set of

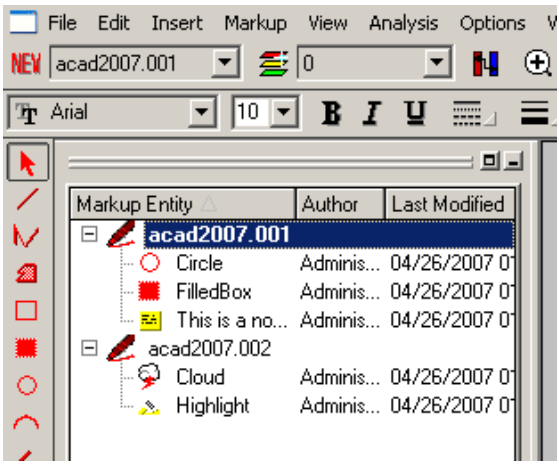
properties are generated for each markup. You can sort the markups in the tree according to each property by clicking the column headers. These properties are:

Property	Description
Entity	Type of markup entity created.
Author	The name of the user who created the markup.
Last Modified	The date and time the markup was 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 **Markup Navigation Tree** and the information is recorded and saved in the markup file.

Note Hover your mouse over an entity to see its Author and Date properties appear in a tooltip.

To see more markup properties or to expand the tree, click and drag the **Markup Navigation Tree** window. You can also drag the column headers left or right to resize the columns. Click the column headers to sort the markups in the tree.



Working with Markup Files

Markup files are divided into uniquely named layers. Create, name and color layers to organize your work. For example, different colors can indicate time priorities and each layer can contain Markups related to a common purpose.

Saved States

When you create and save a Markup file, you also save the view state, such as the zoom level at which you are working. If you save the Markup file while you are working at your preferred zoom level, the next time you open the Markup it displays at the same zoom level. Some examples of view states include zoom level (extents), rotation and flip settings, transformation, section plane and visibility. When creating Markups for 3D files containing imported models, the imported design becomes part of the view state. For more information on 3D view states, ***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, select the markup entity and select **Go To**.

Creating a Markup File

1 View a file that you want to markup.
2 Select **File > Markup** from the AutoVue main menu.

Note You can also click the Markup button  in the AutoVue main toolbar.

AutoVue enters **Markup** mode and a new Markup file appears in the Markup Navigation tree.

See Also Creating a Markup Layer

Entering Markup Information

In AutoVue, when creating a Markup you can provide user information that you can save with the Markup.

1 Select **Markup > Information** from the main menu.
The **Markup Information** dialog appears.
2 Enter a **User** name.

3 Enter a **Department**.
4 Enter a **Company** name.
5 Enter the **Location** of the Company.
6 Enter a **Telephone** number.
7 Click **OK**.
The Markup Information is saved.

Saving a New Markup File

1 Select **File > Save As** from the AutoVue main menu.
The **Save Markup File As** dialog appears.
The **Markup Information** that you entered when creating the Markup displays in the dialog.
2 Enter a **Markup ID** consisting of any combination of characters or numbers.
3 Click **OK**.
The Markup file is saved and remains displayed in the workspace.

Opening Markup Files

1 View a file that has existing markups.
2 Click the red lightbulb button  in the bottom-left corner of the status bar.
Note: If this button does not display, the file you opened has no existing markups associated to it.
The **Markup Files** dialog appears.
3 From the **Markup list**, select the checkbox beside the markup(s) that you want to open.
Note: If you select more than one Markup, the **Active Markup** menu lets you select which one of them is initially active.
4 Click **OK**.
The selected markup(s) appear in the workspace on top of the original file.
Note: When you select more than one markup file, the markups display simultaneously.

See Also ***Setting the Active Markup File***

Saving an Existing Markup File

1 Modify the markup file.
2 Select **File > Save** from the AutoVue main menu.

Note If you have multiple markups open that you have modified and want to save them all, select **File > Save All**.

Note To save an existing Markup as a new Markup, select **File > Save As**.

Importing a Markup File

1 Select **File > Open** from **Markup** mode.
2 In the **Markup Files** dialog that appears, click **Import**.
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**.
The imported Markup file appears in the workspace on top of the original file.

Exporting a Markup File

1 In Markup mode, select **File > Save As**.
Note For a new Markup file, select **File > Save**.
2 Click **Export**.
The **Select markup file export to** dialog appears.
3 Navigate and select the directory you want to export the markup file to.
4 Enter **File name**.
5 Click **Save**.
The markup file is exported to the selected directory.
Note The default format saved is **Markup Files (*.*)**, but you can select another one. In the **Save as type** drop-down list, there are six formats to choose from: Markup Files (*.*), DXF Output (*.dfx), AutoCAD DWG R12 (.dwg), AutoCAD DWG R13 (.dwg), AutoCAD DWG 14 (.dwg) and 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 of the markups as the active markup. Any changes you make is applied to the current active markup.

1 In Markup Mode, select **File > Open**.
The **Markup Files** dialog appears.
2 Click the checkbox beside the markups you want to open.

The **Active Markup** drop-down list appears when you select more than one markup.

3 From the **Active Markup** drop-down list, select the markup you want to make active.

Note: An active markup displays in bold in the **Markup Navigation Tree**.

4 Click **OK**.

The selected markup appears in the AutoVue workspace.

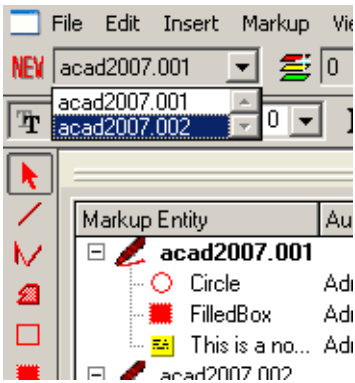
Changing the Active Markup File

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 Markup Navigation Tree, right-click the name of the markup file you want to make active, then select **Set Active** from the menu.
- You can set the active markup by selecting a markup from the drop-down list on the Markup too1bar.



- Take the following steps:
  a. Select **File > Set Active** from the Markup menu.
  b. In the **Set Active Markup** dialog appears, select the markup you want to make active from the **Select Active Markup** drop-down list.
  c. Click **OK** to make the selected markup file active.

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 relating to 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; default layer color is red.

Creating a Markup Layer

Note Ensure that no markup entity is selected when creating a markup layer.

1 Select **Markup > Markup Layers** from the Markup menu.
The **Markup Layers** dialog appears.

Note You can also click the Markup Layers button  on the Markup toolbar.

2 In the **Markup Layers** dialog, click **New**.
The **New Markup Layer** dialog appears.
3 Enter the name that you want for the markup layer.
4 Click **OK**.
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** in the **Markup Layer** dialog.
The **Layer Color** dialog appears. Select a **Color,** then click **OK**.
6 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.

Note Ensure that no markup entity is selected when setting the active markup layer.

1 Select **Markup > Markup Layers** from the Markup menu.
The **Markup Layers** dialog appears.

Note You can also click the Markup Layers button on the Markup toolbar.

2 From the **Markup Layers** list, select the checkbox beside the layer you want to make active.

3 Click **OK**.
The selected Markup layer appears in the workspace.

Changing the Active Markup Layer

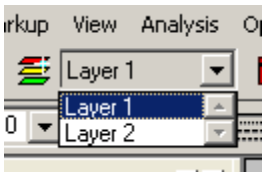
When you have multiple markup layers, you can change the active layer to another layer.

1 Select **Markup > Markup Layers** from the Markup menu.
The **Markup Layers** dialog box appears.

2 From the **Currently Active Layer** drop-down list, select the layer you want to make active.

3 Click **OK**.
The selected layer appears in the workspace.

You can also change the active layer by selecting the layer from the drop-down list on the Markup too1bar.



Changing the Color of a Markup Layer

Note Ensure that no markup entity is selected when changing the color of a markup layer.

1 Select **Markup > Markup Layers** from the Markup menu.
The **Markup Layers** dialog appears.

Note You can also click the Markup Layers button from the Markup toolbar.

2 Select the markup layer that you want to change the color.

3 Click **Color**.
The **Layer Color** dialog appears.

4 Select a **Color**.

5 Click **OK**.

Note Only the entities that were created with the by-layer color defined in the **Markup Layers** dialog will change color. Any entities that were created using the color options from the Markup toolbar, will override the by-layer color and will not change color.

6 Click **OK** to close the **Markup Layers** dialog.

Renaming a Markup Layer

1 Select **Markup > Markup Layers** from the Markup menu.
The **Markup Layers** dialog appears.

Note You can also click the Markup Layers button from the Markup toolbar.

2 Under **Markup Layers**, select the checkbox beside the markup layer you want to rename.
3 Click **Rename**.
The **New Markup Layer** dialog appears.
4 Enter the new layer name.
5 Click **OK**.
The markup layer is assigned the new name.
6 Click **OK** to close the **Markup Layers** dialog.

Toggling Markup Layers

From the **Markup Layers** dialog, you can turn a layer's visibility on and off, even if there is only one layer.Turning a layer's visibility off, hides all markups belonging to that layer.

1 Select **Markup > Markup Layers** from the Markup menu.
The **Markup Layers** dialog appears.

Note You can also click the Markup Layers button from the Markup toolbar.

2 Under **Markup Layers**, select the layer(s) you want visible.
3 Click **Toggle**.
A checkmark appears beside the selected layers.
4 To hide a layer, select the layer(s) with a checkmark, then click **Toggle** again.

Note You can also view a layer by selecting the checkbox beside it. To hide a layer, clear the checkmark. To view all the Markup layers, click **All On**. To hide all Markup layers click **All Off**.

5 Click **OK**.
The markup entities belonging to the selected layer(s) appear in the workspace on top of the original file.

Deleting a Markup Layer

1 Select **Markup > Markup Layers** from the Markup menu.
The **Markup Layers** dialog appears.

Note You can also click the Markup Layers button  from the Markup toolbar.

2 From the **Markup Layers** list, select the checkbox beside the markup layer you want to delete.
Note You can select more than one layer to delete at the same time.

3 Click **Delete**.

4 Click **OK**.
The layer is deleted along with all entities belonging to the layer.

Moving a Markup Entity to Another Layer

1 Select the entity or entities that you want to move.

2 Select **Markup > Markup Layers** from the Markup menu.
The **Move to Layer** dialog appears.

Note You can also click the Layer button  on the Markup toolbar.

3 Select the **Layer** you want to move the entity or entities to.

4 Click **OK**.
The selected entity or entities are moved to the selected layer.

Note Only the entities that were created with the  **By Layer** color defined in the **Markup Layers** dialog will change color. Any entities that were created using the color options from the Markup toolbar, will override the by-layer color and will not change color.

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 The **Consolidate** option is only active when more than one markup file is opened.

1 Open the markup files that you want to consolidate.
2 Select **File > Consolidate** from the Markup menu.
The **Consolidate Markups** dialog appears.
3 Select the markup layers that you want to consolidate into one file.
4 Enter a **Markup ID** for the new markup file.
5 If you want to open the newly consolidated markup as the active markup, select the **Open as active markup** option.
6 Click **OK**.
The consolidated markup file is saved. If you selected **Open as active markup**, the consolidated markup opens and is set as the active markup.

Modifying Markup Entities

With AutoVue, you can assign an entity its own color, the same color as the current active layer or create 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 entities while you specify the change or new entities that you add.

The modify options can be accessed from the **Markup** menu in Markup mode. The options are:

Option	Description
Delete	Delete a selected markup entity or entities.
Delete All	Delete all markup entities.
Group or Ungroup	Group or ungroup markup entities. ***See Grouping Markup Entities***

Option	Description
Line Style	Change the line style for a selected markup entity or entities. Click the Line Style button  in the toolbar. ***See Changing the Line Style***
Line Thickness	Change the line thickness for a selected markup entity or entities. Click the Line Thickness button  in the toolbar. ***See Changing the Line Thickness***
Arrow Style	Add an arrow head at one or both ends of a markup line entity. Click the Arrow Style button  in the toolbar. ***See Changing the Arrow Style***
Fill Type	Change the transparency for selected markup entity or entities. Click the Fill Type button  in the toolbar. ***See Changing the Fill Type of An Entity***
Entity Color	Change the line color, fill color and fill type for a selected markup entity or entities. Click the Line Color button  in the toolbar to change the line color. ***See Changing the Line Color of An Entity*** Click the Fill Color button  in the toolbar to change the fill color. ***See Changing the Fill Color of An Entity***
Entity Rotate	Rotate a selected Symbol markup entity.
Markup Entity Properties	Open the Markup Entity Properties dialog, which lets you see and modify markup entity properties from one dialog. ***See Modifying Markup Entity Properties***
Hide Markups	Hide all markup entities. ***See Hiding Markup Entities***

“Go To” a Markup Entity

Go To restores the view state when a created entity is saved. Opening an existing markup for a file also restores the last saved view state.

1 From the **Markup Navigation Tree**, select the markup entity that you want to view.

2 Right-click and select **Go To** from the drop-down menu.
AutoVue displays the markup page containing the entity.
Note If you selected a Markup entity that is on another page of the Markup, the page containing that entity will be displayed.

Selecting Markup Entities

1 Click the Select button  from the Markup toolbar, then from the workspace click the entity’s outer edge.
Note You can also select the entity from the **Markup Navigation Tree**.
Note To select multiple entities, press the **Shift** or **Ctrl** key while selecting.

Moving a Markup Entity

1 In the **Markup Navigation Tree** or in the workspace, select the entity or entities that you want to move.
See Selecting Markup Entities

2 In the workspace, click and drag the selected entity or entities to anywhere in the workspace.

Editing a Markup Entity

The entities that you can edit are notes, text, dimensions, leaders and hyperlinks.

1 In the **Markup Navigation Tree** or from the workspace, double-click the entity that you want to edit.
AutoVue zooms in on the entity and the appropriate dialog appears allowing you to make changes to the entity.
Note You can also edit an entity by right-clicking it in the **Markup Navigation Tree** and selecting **Edit** from the pop-up menu.

Grouping Markup Entities

When you group markup entities, you can move, delete, copy and paste, resize, or perform any modification on the group of entities that you would on a single markup entity.

1 In the **Markup Navigation Tree** or from the workspace, select the entities that you want to group.
See Selecting Markup Entities
2 Select **Markup > Group** from the Markup menu.
The group of entities appear on the **Markup Navigation Tree** as a **Group**.
3 Perform any modifications.
The modifications are applied to all the entities in the group.

Ungrouping Markup Entities

1 In the **Markup Navigation Tree** or the workspace, select the **Group** that you want to ungroup.
2 Select **Markup > Ungroup** from the Markup menu.
The group of entities appear on the markup as individual entities.

Hiding Markup Entities

To hide the Markup entities of a markup file, take one of the following steps:

- In the Markup Navigation Tree, right-click the markup file, then select **Hide**.
Note To show hidden Markups, right-click the file name again and select **Show**.
- Select **Markup > Hide Markups** from the Markup menu.
The markup entities are hidden on the markup file.
Note To “unhide” markups, select **Markup > Hide Markups** again from the Markup menu.

Changing Line Thickness of an Entity

Note A checkmark beside the thickness option indicates the current line thickness.

1 Select the entity or entities for which you want to change the line thickness.
See Selecting Markup Entities

2 From the Markup menu, select **Markup > Line Thickness** and the line thickness that you want.
The line thickness changes for the selected entity or entities.

Note You can also click the Line Thickness button  on the Markup toolbar.

3 To define a custom line thickness, select **Modify > Custom Thickness** from the Markup menu.
The **Custom Thickness** dialog appears.
4 Enter an integer value in pixels.
5 Click **OK** to close the **Custom Thickness** dialog.

Note Any new entities that you create will have the new line thickness.

See Also Modifying Markup Entity Properties

Changing Line Style of an Entity

Note A checkmark beside the style option indicates the current line style.

1 Select the entity or entities for which you want to change the line style.
See Selecting Markup Entities
2 From the Markup menu, select **Markup > Line Style** and the line style that you want.
The line style changes for the selected entity or entities.

Note You can also click the Line Style button  on the Markup toolbar.

Note Any new entities that you create will have the new line style.

See Also Modifying Markup Entity Properties

Changing the Arrow Style

1 Select the line entity or entities for which you want to change the arrow style.
See Selecting Markup Entities
2 Take one of the following steps:
- Right-click the entity and select **Arrow Style** from the menu.
- Click the Arrow Style button  in the toolbar.
- Select **Markup > Arrow Style** from the main menu.

3 From the **Arrow Style** drop-down list, select the style of arrow that you want for the entity.

You have the choice between no arrow heads, an arrow head at either one or the other end of the line entity, or arrow heads at both ends of the line entity. The arrow style changes for the selected line entity or entities.

Note Any new entities that you create will have the new arrow style.

See Also Modifying Markup Entity Properties

Changing Line Color of an Entity

1 Select the entity or entities for which you want to change the line color.
See Selecting Markup Entities

2 Select **Markup > Entity Color** from the Markup menu.
The **Entity Color** dialog appears.

Note You can also click the **Line Color** button  on the Markup toolbar.

3 From the **Line Color** drop-down list, select the color that you want for the entity.

Note Selecting **By Layer**  changes the entity color to the color of the layer.

4 To define a custom line color, select **Custom Color**  from the **Line Color** drop-down list.

5 From the **Color** dialog that appears, select a color and click **OK**.

6 Click **OK** to close the **Entity Color** dialog.
The line color changes for the selected entity or entities.

Note Any new entities that you create will have the new line color.

See Also Modifying Markup Entity Properties

Changing Fill Color of an Entity

1 Select the entity or entities for which you want to change the fill color.
See Selecting Markup Entities

2 Select **Markup > Entity Color** from the Markup menu.
The **Entity Color** dialog appears.

Note You can also click the Fill Color button  on the Markup toolbar.

3 From the **Fill Color** drop-down list, select the color that you want for the entity.

Note Selecting  **By Layer** changes the entity color to the color of the layer.

4 To define you own color, select **Custom Colr**  from the **Fill Color** drop-down list.

5 From the **Color** dialog that appears, select a color and click **OK**.
The fill color changes for the selected entity or entities.

6 Click **OK** to close the **Entity Color** dialog.

Note Any new entities that you create will have the new fill color.

See Also Modifying Markup Entity Properties

Changing Fill Type of an Entity

1 Select the entity or entities for which you want to change the fill type.
See Selecting Markup Entities

2 Select **Markup > Entity Color** from the Markup menu.
The **Entity Color** dialog appears.

Note You can also click the Fill Type button  on the Markup toolbar.

3 From the **Fill Type** drop-down list, select the fill type that you want for the 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**.
The fill type changes for the selected entity or entities.

Note Any new entities that you create will have the new fill type.

See Also Modifying Markup Entity Properties

Assigning an Entity the Same Color as the Layer

1 Select the entity or entities for which you want to assign the color of the layer that they belong to.
See Selecting Markup Entities

2 Select **Markup > Entity Color** from the Markup menu.
The **Entity Color** dialog appears.

3 To assign the line color, select the  **BYLAYER** from the **Line Color** drop-down list.

Note You can also click the Line Color button  from the Markup toolbar.

4 To assign the fill color, select the  **BYLAYER** from the **Fill Color** drop-down list.

Note You can also click the Fill Color button  from the Markup toolbar.

5 Click **OK**.
The selected entity or entities change to the color of the layer that the entity belongs to.

Deleting Markup Entities

1 Select the entity or entities that you want to delete.
See Selecting Markup Entities

2 Select **Markup > Delete** from the Markup menu.
The selected entity or entities are deleted from the current active file.
Note To delete all entities, select **Markup > Delete All** from the Markup menu. **Delete All** will delete all existing entities from the active markup, whether they are visible or not.

Modifying Measurement Markup Entities

In AutoVue, you can modify the font of a measurement.

1 Create the markup measure entity that you want.
See Creating 2D non-vector Markup Measure Entities
Creating 2D Vector Markup Measure Entities
Creating EDA Markup Measure Entities
Creating 3D Markup Measure Entities

2 Double-click the measurement for which you want to modify the font.
The appropriate **Measure** dialog appears.

3 Click **Font**. The **Font** dialog appears.
From the **Font** drop-down list, select the type of font.
From the **Size** drop-down list, select the size of font.

Select the font **Style(s)**, then click **OK**.

4 Click **OK** to close the appropriate **Measure** dialog.
The measurement font is modified.

Changing Measurement Units and Symbols

In AutoVue, you can change the unit of measure and add a symbol to a measurement and have it appear on the drawing.

1 Create the markup measure entity that you want.
See ***Creating 2D non-vector Markup Measure Entities***
Creating 2D Vector Markup Measure Entities
Creating EDA Markup Measure Entities
Creating 3D Markup Measure Entities

2 Double-click the measurement that you want to change the unit of measure or add a symbol to.
The appropriate **Measure** dialog appears.

3 Clear the checkmark beside **Display Unit** if you want to hide the unit on the drawing. Unit appears by default.

4 From the **Units** drop-down list, select the unit that you want to change the measurement to.

5 From the **Symbol** drop-down list, select the symbol that you want to add to the measurement.

6 Click **OK**.
The unit of measure changes and the selected symbol is added to the measurement and appears in the workspace.

Modifying Markup Entity Properties

In AutoVue, you can use the Markup Entity Properties dialog to modify markup entities.

To open the Markup Entity Properties dialog, select **Markup > Markup Entity Properties** from the Markup menu. You can also right-click a markup entity in the Markup Navigation Tree or the workspace and select **Markup Entity Properties** from the contextual menu.

Line Color

Change the line color the selected markup entity or entities.

1 Select the entity or entities for which you want to change the line color.
See Selecting Markup Entities
2 From the **Line Color** drop-down list, select the color that you want for the entity.

Note: Selecting **By Layer** changes the entity color to the color of the layer.
3 To define a custom line color, take the following steps:

a. Select **Custom Color** from the **Line Color** drop-down list.
b. From the **Color** dialog that appears, select a color and click **OK**.
4 Make other changes you want with the **Markup Entity Properties** dialog, then click **OK** to close it.
The line color changes for the selected entity or entities.
Note: Only selected entities that you create will have the new line color. To apply the new line color to any new entity you create, make sure no entity is selected before opening the **Markup Entity Properties** dialog.
See Also Changing the Line Color of An Entity

Line Style

Change the line style for a selected markup entity or entities.

1 Select the entity or entities for which you want to change the line style.
See Selecting Markup Entities
2 Select the line style that you want from the **Line Style** menu.
The line style changes for the selected entity or entities.
Note: Any new entities that you create will have the new line style.
See Also Changing the Line Style

Line Thickness

Change the line thickness for a selected markup entity or entities.

1 Select the entity or entities for which you want to change the line thickness.
See Selecting Markup Entities
2 Select the line thickness that you want from the **Line Thickness** menu.
The line thickness changes for the selected entity or entities.

To define a custom line thickness, take the following steps:

1 Select **Customize** from the **Line Thickness** menu.
2 In the **Width (Pixels)** menu, enter the desired width.

3 Make other changes you want with the **Markup Entity Properties** dialog, then click **OK** to close it.
The line thickness changes for the selected entity or entities.
Note: Any new entities that you create will have the new line thickness.

See Also Changing the Line Thickness

Width (Pixels)

Customize the line thickness when **Customize** is selected in the Line Thickness menu. When other line thicknesses are selected, this field displays its value in pixels but cannot be edited.

See Line Thickness

Arrow Style

Add an arrow head at one or both ends of a markup line entity.

1 Select the line entity or entities for which you want to change the arrow style.
See Selecting Markup Entities
2 From the **Arrow Style** drop-down list, select the style of arrow that you want for the entity.
3 Make other changes you want with the **Markup Entity Properties** dialog, then click **OK** to close it.
The arrow style changes for the selected entity or entities.
Note: Any new entities that you create will have the new arrow style.

See Also Changing the Arrow Style

Fill Type

Change the transparency for selected markup entity or entities.

1 Select the entity or entities for which you want to change the fill type.
See Selecting Markup Entities
2 From the **Fill Type** drop-down list, select the fill type that you want for the 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 Make other changes you want with the **Markup Entity Properties** dialog, then click **OK** to close it.
The fill type changes for the selected entity or entities.
Note: Any new entities that you create will have the new fill type.

See Also Changing the Fill Type of An Entity

Fill Color

Change the line color, fill color and fill type for a selected markup entity or entities.

1 Select the entity or entities for which you want to change the fill color.
See Selecting Markup Entities
2 From the **Fill Color** drop-down list, select the color that you want for the entity.

Note: Selecting  **By Layer** changes the entity color to the color of the layer.
3 To define you own color, take the following steps:

a. Select **Custom Color**  from the **Fill Color** drop-down list.
b. From the **Color** dialog that appears, select a color and click **OK**.
The fill color changes for the selected entity or entities.
4 Make other changes you want with the **Markup Entity Properties** dialog, then click **OK** to close it.
The fill type changes for the selected entity or entities.
Note: Any new entities that you create will have the new fill color.

See Also Changing the Fill Color of An Entity

Markup Layers

When you have multiple markup layers, you can change the active layer to another layer.

1 From the **Markup Layers** drop-down list, select the layer you want to make active.
2 Make other changes you want with the **Markup Entity Properties** dialog, then click **OK** to close it.
The selected layer appears in the workspace.

See Also Changing the Active Markup Layer

Marking Up 2D Files

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. You can also add a symbol, add text to an entity or add information by adding a note.

Note When you are creating a markup entity, you can press **Escape** to cancel.

2D Markup Entities

In AutoVue, you can drawing many different types of entities. You can access the entities from the **Entities** menu in Markup mode. The entities are:

Option	Description
Note Right click to complete an entity.	
Arc	Click and drag the mouse to draw an arc. From the toolbar, click the Arc button  .
Box	Click and drag to draw a rectangle. From the toolbar, click the Box button  . **Note** To draw a square instead of a rectangle, press and hold the **Shift** key while you click and drag.
Closed Polyline	Click and drag to draw a closed Polyline. Right-click to define the last point. From the toolbar, click the Closed Polyline button  .
Cloud	Click and drag to draw a cloud. From the toolbar, click the cloud button  .
Circle	Click and drag to draw an ellipse. From the toolbar, click the Circle button  . **Note** To draw a circle instead of an ellipse, press and hold the **Shift** key while you click and drag.

Option	Description
Filled Box	Click and drag to draw a filled box. The box is filled with a solid color. From the toolbar, click the Filled Box button  . **Note** To draw a square Filled Box instead of a rectangular Filled Box, press and hold the **Shift** key while you click and drag.
Freestyle	Click, release mouse button, then drag to draw an entity. From the toolbar, click the Freestyle button  . **Note** You can create a contiguous or non-contiguous freestyle entity. ***See Creating a Non-contiguous Freestyle Entity*** ***See Creating a Contiguous Freestyle Entity***
Highlight	Click and drag to highlight a boxed area. From the toolbar, click the Hightlight button  . **Note** The highlighted box will be filled with a transparent color.
Leader	Click and drag to draw a leader. From the toolbar, click the Leader button  . **Note** To force a line segment in a Leader entity to be aligned to the closer axis, horizontal or vertical axis, hold the **Shift** key while you click and drag for that line segment. ***See Adding a Leader*** ***See Forcing a Line Segment***
Line	Click and drag to draw a line. From the toolbar, click the Line button  . **Note** 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 Forcing a Line Segment***
Note	Add a note to the Markup. From the toolbar, click the Note button  . ***See Adding a Note***

Option	Description
Polyline	Click and drag to draw a polyline. Right-click to define the last point. From the toolbar, click the Polyline button  . **Note** 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 Forcing a Line Segment***
Symbol	Click and drag a symbol to the Markup. From the toolbar, click the Symbol button  . ***See Adding a Symbol***
Text	Add text to the Markup. From the toolbar, click the Text button  . ***See Adding Text***

Creating a Non-contiguous Freestyle Entity

1 Select **Entities > FreeStyle** from the Markup menu.

Note You can also click the Freestyle button  from the Markup toolbar.

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.

Note 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.

Creating a Contiguous Freestyle Entity

1 Select **Entities > FreeStyle** from the Markup menu.

Note You can also click the Freestyle button  on the Markup toolbar.

2 Click a point on the drawing where you want to start the freestyle entity.
3 Move the cursor to create the freestyle entity.
4 Right-click to end the freestyle entity.

Adding a Leader

1 Select **Entities > Leader** from the Markup menu.

Note You can also click the Leader button  from the Markup toolbar.

2 Click a point on the document where you want the leader to start.
3 Move the cursor to draw the leader.

Note To draw a leader 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.
The **Text** dialog appears.
6 To change the font, click **Font**. The **Font** dialog appears.
From the **Font** drop-down list, select the type of font.
From the **Size** drop-down list, select the size of font.
Select the checkbox beside the font style(s).
Type the text that you want to attach to the leader, then click **OK**.
7 Right-click outside the text area to complete the modification.
The leader appears on the drawing and in the **Markup Navigation Tree**.

Note To edit the leader text, double-click the leader in the **Markup Navigation Tree** or in the workspace to open the **Text** dialog.

See Also ***Forcing a Line Segment***

Forcing a Line Segment to Become Horizontal or Vertical

You can draw a line segment and force it to be aligned with the closer axis, horizontal or vertical. 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.

Note This procedure only works for measure entities that are drawn using **Free snap**.

1 To draw and force a line segment, press and hold the **Shift** key while you click and drag for that line segment.

To force an existing line segment, click and hold the left mouse button on the line segment, then press and hold the **Shift** key.

2 When you see that the line segment is horizontal or vertical, release the left mouse button, then release the **Shift** key.

Adding Text

With AutoVue, you can add text entities to a markup.

1 Select **Entities > Text** from the Markup menu.

Note You can also click the Text button  from the Markup toolbar.

2 Click a point on the drawing where you want to add text.
The **Text** dialog appears.

3 To change the font, click **Font**. The **Font** dialog appears.
From the **Font** drop-down list, select the type of font.
From the **Size** drop-down list, select the size of font.
Select the checkbox beside the font style(s).
Type the text that you want to add, then click **OK**.

4 Right-click outside the text area to complete the modification.
The text appears on the drawing and in the **Markup Navigation Tree**.

5 To move the text box click and drag it.

6 Click and drag the frame handles to enlarge the text box.

Note To edit text, double-click the text in the **Markup Navigation Tree** or in the workspace to open the **Text** dialog.

Adding a Note

You can attach longer comments and notes to a markup with AutoVue's Markup Control Note Editor. A note is represented by a standard size graphical symbol labelled **Note**. To read the text inside, double-click the entity to open it or move the mouse over the entity to display the tooltip.

1 Select **Entities > Note** from the Markup menu.

Note You can also click the Note button  from the Markup toolbar.

2 Click a point on the document where you want to insert the note.
The **Note** dialog appears.

3 Enter the text that you want in the note.

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

5 Close the **Note** dialog.
The note appears on the drawing and in the **Markup Navigation Tree**.

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

7 To move the note, click and drag it.

Note To edit a note, double-click the note symbol  in the **Markup Navigation Tree** or in the workspace to open the **Note** dialog.

Symbols

A **Symbol** 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**.

In AutoVue, you can create a Symbol Library and add symbols to it.

Adding a Symbol

A symbol is a graphical entity, such as a company logo. With AutoVue, you can add a symbol to a markup.

1 Select **Entities > Symbol** from the Markup menu.
The **Symbols** dialog appears.

Note You can also click the Symbol button  in the Markup toolbar.

2 Click the **Symbols** tab.
3 From the **Symbol Library** drop-down list, select the library from which you want to select a symbol.
The symbols appear for the selected library.
4 Click **Isotropic** if you want to scale the symbol proportionately.
Click **Anisotropic** if you want to scale the symbol disproportionately.
5 Drag the symbol onto the workspace.
The symbol appears on the drawing and in the **Markup Navigation Tree**
Note To add more symbols, repeat steps 3 to 5.
6 Click **OK**.

Creating a New Symbol Library

1 Select **Entities > Symbol** from the Markup menu.
The **Symbols** dialog appears.

Note You can also click the Symbol button  in the Markup toolbar.

2 Click the **Libraries** tab.
3 Click **Create**.
The **Symbol Library** dialog appears.
4 Enter the **Library Name**, **Description**, **Author** and **Keywords**.

5 To add a symbol to the library, click **Add**.
The **Open** dialog appears.
6 Browse to locate the symbol you want to add, then click **Open**.
The file appears in the **Selected Files** list.
Note To add more symbols, repeat steps 4 and 5. To remove a symbol, select the symbol and click **Remove**.
7 Click **OK** to close the **Symbols** dialog.

Editing Symbol Library Information

1 Select **Entities > Symbol** from the Markup menu.
The **Symbols** dialog appears.

Note You can also click the Symbol button  in the Markup toolbar.
2 Click the **Libraries** tab.
3 From the **Symbol Libraries** list, select the library that you want edit.
4 Click **Edit**.
The **Symbol Library** dialog appears.
5 Edit the information.
6 Click **OK**.
7 Close the **Symbols** dialog.

Deleting a Symbol Library

1 In the **Symbols** dialog, click the **Libraries** tab.
2 From the **Symbol Libraries** list, select the library that you want to delete.
3 Click **Delete**.
The library disappears from the list and from the **Symbol Library** drop-down list under the **Symbols** tab.
4 Close the **Symbols** dialog.

Working with Hyperlinks

A hyperlink is a link between the current file and the new file or application. You can create hyperlinks within your current file so that your files and applications outside AutoVue are only a click away. The main benefit of adding hyperlinks to a file is that you can gather all files of related information into one file with the files kept separate. In other words, the files may be 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 — to the linked file itself.

Creating a Hyperlink

When you create a hyperlink, you can choose where to open the hyperlinks.

Open in	Description
Applet	Opens the file in another AutoVue window.
Current Applet	Opens the file in the current AutoVue window.
Browser	Opens the file in the default browser window.
Current Browser	Opens the file in the current browser window.

1 Select **Markup > Hyperlink > Establish** from the Markup menu.
2 Click a point on the document where you want to place the hyperlink.
The **Establish Hyperlink** dialog appears.
3 Enter a **Link Name**.
4 Enter a **Description** (optional).
5 Type the **URL** or click **Browse** to locate the file that you want to link to.
6 From the **Link to** drop-down list, select the type of link.
For example, **Data File** opens a document file, whereas **Application** open an application other than AutoVue.
7 From the **Open in** drop-down list, select where you want the hyperlink to open.
8 Click **OK**.
The hyperlink appears on the Markup.

Firing a Hyperlink

1 To fire a hyperlink, double-click on it from the workspace.
The hyperlink file opens in the window you selected in the **Establish Hyperlink** dialog.
Note You can also fire a hyperlink by selecting it, then right-clicking and selecting **Hyperlinks > Fire** from the pop-up menu or selecting **Hyperlink > Fire** from the Markup menu.

Editing a Hyperlink

1 In the **Markup Navigation Tree** or from the workspace, select the hyperlink that you want to edit.
2 Select **Markup > Hyperlink > Edit** from the Markup menu.
The **Edit Hyperlink** dialog appears.
Note You can right-click a hyperlink in the **Markup Navigation Tree** and select **Edit** from the pop-up menu.
3 Edit the information that you want.
4 Click **OK**.
The changes are saved.

Deleting a Hyperlink

1 In the **Markup Navigation Tree** or from the workspace, select the hyperlink that you want to delete.
2 Select **Markup > Hyperlink > Break** or **Markup > Delete** from the Markup menu.
The hyperlink is deleted from the file.
Note You can also delete a hyperlink by right-clicking it in the **Markup Navigation Tree** and selecting **Delete** from the pop-up menu.

Viewing List of Hyperlinks

1 Select **Markup > Hyperlink > List** from the Markup menu.
The **List Hyperlinks** dialog appears listing the hyperlinks.
Note You can right-click in the workspace and select **Hyperlinks > List** from the pop-up menu.
Note You can fire a hyperlink by selecting the hyperlink and clicking **Fire**.
2 Click **OK**.

Viewing History of a Hyperlink

AutoVue lets you view information about a hyperlink's path. From a file that is launched from a hyperlink, you can see what document invoked the hyperlink.

1 From the hyperlink file, select **File > Markup** from the AutoVue main menu.
2 Select **Markup > Hyperlink > History** from the Markup menu.

The **Hyperlink History** dialog appears listing the path linking the hyperlink file to the file that invoked the hyperlink.

3 To jump to the file where you fired the hyperlink from, select the path in the **Hyperlinks History** dialog.

4 Click **Jump to**.

Note You can also go back by right-clicking in the hyperlink file and selecting **Go Back** from the pop-up menu.

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. For information on marking up 2D files, ***see Marking Up 2D Files***.

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 **Escape** 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. You can access the measure options from the **Analysis > Measure** menu. The options are:

Option	Description
Angle	Measure the angle between selected points. From the toolbar, click the Angle button  .
Arc	Measure an arc entity. From the toolbar, click the Arc button  .

Option	Description
Area	Measure a selected area. From the toolbar, click the Area button  .
Distance	Measure the distance between two points. From the toolbar, click the Distance button  .

Measuring Distance

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

Note If you want a markup entity for the measurement, you must ensure that **Add Distance** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the Markup toolbar.

2 Click the **Distance** tab.
3 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure the distance.
4 Click a point on the drawing to define the starting point.
5 Click another point on the drawing to define the end point.
The measured line path appears as an entity on the current active markup.
6 Drag to move the measured line path.
7 Click on the measured line path.
The measurement and unit appear as an entity on the current active markup layer. The measured distance, Delta-X and Delta-Y appear in the **Measurement** dialog.
8 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
9 Click **Close** to close the **Measurement** dialog.

See Also ***Modifying Measurement Markup Entities***

Measuring Cumulative Distance

Use the **Cumulative Distance** option to measure the distance along a path of multi-faceted (adjoining) points.

Note If you want a markup entity for the measurement, you must ensure that **Add Distance** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the AutoVue toolbar.

2 Click the **Distance** tab.
3 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure the distance.
4 Select **Cumulative**.
5 Click a point on the drawing to define the starting point.
6 Continue clicking points along the path that you want to measure.
Each point is joined by a line.
7 Right-click to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The cumulated measured distance, Delta-X and Delta-Y appear in the **Measurement** dialog.
8 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
9 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Calibrating Distance

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

Measuring Area

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

Note If you want a markup entity for the measurement, you must ensure that **Add Area** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.The **Measurement** dialog appears.

Note You can also click the Area button  on the Markup toolbar.

2 Click the **Area** tab.

3 From the **Measured Area Units** drop-down list, select the unit in which you want to measure the area.
From the **Perimeter Units** drop-down list, select the unit in which you want to measure the perimeter.

4 To cumulate a **Net Area Result** of different areas, select **Add** in the **Measurement** dialog.
To subtract an area from the **Net Area Result**, select **Subtract**.
Select **Clear** to clear the **Net Area Result**.

5 Click a point on the drawing to define the starting point.

6 Continue clicking points on the drawing to define the area you want to measure.
Each point is joined by a line.

7 Right click to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The area and perimeter measurements appear in the **Measurement** dialog.

8 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.

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

See Also ***Modifying Measurement Markup Entities***

Measuring an Angle

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

Note If you want a markup entity for the measurement, you must ensure that **Add dimensions** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Angle button  on the Markup toolbar.

2 Click the **Angle** tab.
3 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.
4 Click points on the drawing to define the angle you want to measure.
The points are joined by angle arms with an arc connecting them.
5 Click again to complete the measurement.
The measured line path, angle measurement and unit appear as an entity on the current active markup layer and in the **Measurement** dialog.
6 Click and drag to change the size of the arc.
7 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
8 Click **Close** to close the **Measurement** dialog.

See Also ***Modifying Measurement Markup Entities***

Measuring an Arc

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

Note If you want a markup entity for the measurement, you must ensure that **Add Radius** or **Add Diameter** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the Markup toolbar.

2 Click the **Arc** tab.
3 From the **Arc Info** drop-down list, select the unit in which you want to measure the arc.
4 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.
5 Select **Add Radius** if you want to measure the radius of the arc.
Select **Add Diameter** if you want to measure the diameter of the arc.
6 Click points on the drawing to define the arc you want to measure.
The points are joined by an arc.
7 Click again to complete the measurement.
The measured line path, arc measurement and unit appear as an entity on the current active markup layer and in the **Measurement** dialog.
8 Click and drag the box anywhere on the drawing.
Click and drag the frame handles to change the size of the box.

Note Click **Reset** to take another measurement.

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

See Also Modifying Measurement Markup Entities

Calibrating an Arc

1 Measure an arc in the drawing.
See Measuring an Arc

2 In the **Measurement** dialog, click **Calibrate**.
The **Distance Calibration** dialog appears displaying the measured distance.

3 From the **Distance** drop-down list, select the unit to which you want to calibrate the distance.

4 Click **Calibrate to** and enter a value if you want to calibrate to a value.
Click **Set Factor** and enter a value if you want to calibrate by a factor.

5 Click **OK**.
The calibration result appears in the **Measurement** dialog.

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

Creating 2D Vector Markup Measure Entities

When marking up 2D vector files, in addition to all the markup options available for 2D files, you can create markup measure entities. The measure options in Markup mode work slightly different than in View mode. For information on marking up 2D files, ***see Marking Up 2D Files***.

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 **Escape** to cancel.

In Markup mode, you can choose from several measure options to create markup measure entities. You can access the measure options from the **Analysis > Measure** menu. The options are:

Option	Description
Angle	Measure the angle between selected points. From the toolbar, click the Angle button  .
Arc	Measure an arc entity. From the toolbar, click the Arc button  .
Area	Measure selected area. From the toolbar, click the Area button  .
Distance	Measure the distance between two points. From the toolbar, click the Distance button  .

2D Vector Snapping Modes

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

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

Button	Snap to	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near a linear component's end point.
	Mid-point	Geometric snap mode where a snap box appears when moving the cursor near the halfway point of a linear component.

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

Measuring Distance

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

Note If you want a markup entity for the measurement, you must ensure that **Add Distance** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the Markup toolbar.

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 clear all **Snapping Modes** click **All Off.**

See 2D Vector Snapping Modes

4 From the **Measure Distance Units** drop-down 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.
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.
The measurement and unit appear as an entity on the current active markup layer. The measured distance, Delta-X and Delta-Y appear in the **Measurement** dialog.
9 Click and drag the frame handles to change the size of the box.

Note Click **Reset** to take another measurement.

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

See Also Modifying Measurement Markup Entities

Measuring Cumulative Distance

Use the **Cumulative Distance** option to measure the distance along a path of multi-faceted (adjoining) points.

Note If you want a markup entity for the measurement, you must ensure that **Add Distance** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the Markup toolbar.

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 clear all **Snapping Modes** click **All Off.**
See 2D Vector Snapping Modes
4 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure the distance.
5 Select **Cumulative**.
6 Click a point on the drawing to define the starting point.
7 Continue clicking points along the path that you want to measure.
Each point is joined by a line.
8 Right-click to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The cumulated measured distance, Delta-X and Delta-Y appear in the **Measurement** dialog.
9 Click and drag the frame handle to change the size of the box.
Note Click **Reset** to take another measurement.
10 Click **Close** to close the **Measure Distance** dialog.
See Also Modifying Measurement Markup Entities

Calibrating Distance

1 Measure the distance between two points or measure cumulative distance.
See Measuring Distance
2 In the **Measurement** dialog, click **Calibrate**.
The **Distance Calibration** dialog appears displaying the measured distance.
3 In the **Measured Distance** drop-down list, select the unit to which you want to calibrate the distance.

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

Measuring Area

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

Note If you want a markup entity for the measurement, you must ensure that **Add Area** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Area button  on the Markup toolbar.

2 Click the **Area** tab.
3 Select **Between Points** if you want to measure the area between points on a drawing. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**
See 2D Vector Snapping Modes
4 Select **Shape** if you want to measure the area of a predefined shape on the drawing. **Snapping Modes** are disabled.
5 From the **Measured Area Units** drop-down list, select the unit in which you want to measure the area.
6 From the **Perimeter Units** drop-down list, select the unit in which you want to measure the perimeter.
7 To cumulate a **Net Area Result** of different areas, select **Add** in the **Measurement** dialog.
To subtract an area from the **Net Area Result**, select **Subtract**.
Select **Clear** to clear the **Net Area Result**.
8 If you selected **Between Points**, click points on the drawing to define the area, then right click to complete the measurement.
Each point is joined by a line. The measured line path, measurement and unit appear as an entity on the current active markup layer. The area and perimeter measurements appear in the **Measurement** dialog.
9 If you selected **Shape**, click the edge of a predefined shape on the drawing.

The shape is highlighted. The measured line path, measurement and unit appear as an entity on the current active markup layer. The area and perimeter measurements appear in the **Measurement** dialog.

10 Click and drag the frame handles to change the size of the box.

Note Click **Reset** to take another measurement.

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

See Also Modifying Measurement Markup Entities

Measuring an Angle

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

Note If you want a markup entity for the measurement, you must ensure that **Add dimensions** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Angle button  on the Markup toolbar.

2 Click the **Angle** tab.
Select **From 3 Points** if you want to measure the angle between three points. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**

See 2D Vector Snapping Modes

3 Select **Between 2 lines** if you want to measure the angle between two lines. **Snapping Modes** are disabled.

4 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.

5 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.
Angle arms appear with an arc connecting them.

6 Click to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The measured angle appears in the **Measurement** dialog.

7 Click and drag to change the size of the arc.

8 Click and drag the box anywhere on the drawing.

9 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
10 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Measuring an Arc

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

Note To create a markup entity for the measurement, you must ensure that **Add Radius** or **Add Diameter** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the Markup toolbar.

2 Click the **Arc** tab.
3 Select **From 3 Points** if you want to measure the arc between three points. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**

See 2D Vector Snapping Modes

4 Select **Arc Entity** if you want to measure a predefined arc. **Snapping Modes** are disabled.
5 From the **Arc Info** drop-down list, select the unit in which you want to measure the distance.
6 From the **Measure Angle Units** drop-down list, select the unit in which you want to measure the angle.
7 Select **Add Radius** if you want to measure the radius.
Select **Add Diameter** if you want to measure the diameter
8 If you selected **From 3 Points**, click three points to define the arc, then click to complete the measurement
The points are joined by an arc.
If you selected **Arc Entity**, click the edge of the arc that you want to measure. The arc is highlighted.
9 Click to complete the measurement.
The measured line path, measurement and unit appear as an 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** dialog.

10 Click and drag the box anywhere on the drawing.

11 Click and drag the frame handles to change the size of the box.

Note Click **Reset** to take another measurement.

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

See Also Modifying Measurement Markup Entities

Calibrating an Arc

1 Measure an arc in the drawing.

See Measuring an Arc

2 In the **Measurement** dialog, click **Calibrate**.

The **Radius Calibration** dialog appears displaying the measured distance.

3 From the **Measured Rad.** drop-down list, select the unit to which you want to calibrate the distance.

4 Click **Calibrate to** and enter a value if you want to calibrate to a value.

Click **Set Factor** and enter a value if you want to calibrate by a factor.

5 Click **OK**.

The calibration result appears in the **Measurement** dialog.

6 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. For information on marking up 2D files, ***see Marking Up 2D Files***.

Note When you are creating a markup entity, you can press **Escape** 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, 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.

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. You can access the measure options from the **Analysis > Measure** menu. The options are:

Option	Description
Angle	Measure the angle between selected points. From the toolbar, click the Angle button  .
Arc	Measure arc entity. From the toolbar, click the Arc button  .
Area	Measure a selected area. From the toolbar, click the Area button  .
Distance	Measures the distance between two points. From the toolbar, click the Distance button  .
Minimum Distance	Measure the minimum distance between entities. From the toolbar, click the Minimum Distance button  .

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 mid, center and end-points of an entity, as well as a pin, via, and symbol:

Button	Snap to	Description
	End-point	Geometric snap mode where a snap box appears when moving the cursor near the component’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 component.
	Center-point	Geometric snap mode where a snap box appears when moving the cursor near the center of an elliptical component.
	Pin	Electrical snap mode where a snap box appears when the cursor touches a pin.
	Via origin	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 the entire component.
	Free snap	Allows snapping at any point on the drawing.

Measuring Distance

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

Note If you want a markup entity for the measurement, you must ensure that **Add Distance** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the Markup toolbar.

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 clear all **Snapping Modes** click **All Off.**

See EDA Snapping Modes

4 From the **Measured Distance Units** drop-down 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.
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.
The measurement and unit appear as an entity on the current active markup layer. The measured distance, Delta-X, Delta-Y and the "Manhattan Distance" appear in the **Measurement** dialog.
9 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
10 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Measuring Cumulative Distance

Use the **Cumulative Distance** option to measure the distance along a path of multi-faceted (adjoining) points.

Note If you want a markup entity for the measurement, you must ensure that **Add Distance** is selected in the **Measurement** dialog.

11 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the Markup toolbar.

12 Click the **Distance** tab.
13 Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**
See EDA Snapping Modes
14 From the **Measured Distance Units** drop-down list, select the unit that you want to measure the distance.
15 Select **Cumulative**.
16 Click the first entity to define the starting point.
17 Continue clicking points along the path that you want to measure.
Each point is joined by a line.
18 Right-click to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The measured distance, Delta-X, Delta-Y and the "Manhattan Distance" appear in the **Measurement** dialog.
19 Click and drag the frame handles to change the size of the box.

Note Click **Reset** to take another measurement.

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

See Also Modifying Measurement Markup Entities

Calibrating Distance

1 Measure the distance between two points or measure cumulative distance.
See Measuring Distance

2 In the **Measurement** dialog, click **Calibrate**.

3 The **Distance Calibration** dialog appears displaying the measured distance.

4 In the **Measured Distance** drop-down list, select the unit to which you want to calibrate the distance.

5 Click **Calibrate to** and enter a value if you want to calibrate the distance to a value.
Click **Set Factor** and enter a value if you want to calibrate the distance by a factor.

6 Click **OK**.
The calibration result appears in the **Measurement** dialog.

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

Measuring Minimum Distance

Use the **Minimum Distance** option to measure the minimum distance between entities. The available entities for snapping are nets, pins, vias and traces.

Note If you want a markup entity for the measurement, you must ensure that **Add Distance** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Minimum Distance button  on the Markup toolbar.

2 Click the **Min. Distance** tab.

3 Click  **First Set** to select the entities that you want to measure from.

4 Select the **Snapping Modes** that you want to use for measuring.
See EDA Snapping Modes
Note If you select **Net**, you cannot select any other type of entity.

5 Click the first set of entities on the drawing.

The entities are highlighted.

Note To clear the last set of entities you selected, click **Clear Set**.

6 Click **Second Set** to select the entities that you want to measure to.

7 Click the second set of entities on the drawing.
The entities are highlighted in a different color.

8 From the **Measured Min Distance** drop-down list, select the unit in which you want to measure the distance.

9 Click **Zoom to Result**, if you want to zoom in on the measured value on the drawing.

10 Click **Compute**.
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 as an entity on the current active markup layer. The measurement, delta-X, delta-Y and the "Manhattan Distance" appear in the **Measurement** dialog.

Note Click **Reset** to take another measurement.

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

See Also ***Modifying Measurement Markup Entities***

Measuring Area

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

Note If you want a markup entity for the measurement, you must ensure that **Add Area** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Area button on the Markup toolbar.

2 Click the **Area** tab.

3 Select **Between Points** if you want to measure the area between points on a drawing. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**

See ***EDA Snapping Modes***

4 Select **Shape** if you want to measure the area of a predefined shape on the drawing. **Snapping Modes** are disabled.

5 From the **Measured Area Units** drop-down list, select the unit in which you want to measure the area.

6 From the **Perimeter Units** drop-down list, select the unit in which you want to measure the perimeter.
7 To cumulate a **Net Area Result** of different areas, select **Add** in the **Measurement** dialog.
To subtract an area from the **Net Area Result**, select **Subtract**.
Select **Clear** to clear the **Net Area Result**.
8 If you selected **Between Points**, click points on the drawing to define the area.
Each point is joined by a line and the measurement appears in the **Measurement** dialog.
9 If you selected **Shape**, click the edge of a predefined shape on the drawing. The shape is highlighted.
10 Right-click to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The area and perimeter measurements appear in the **Measurement** dialog.
Note Click **Reset** to take another measurement.
11 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Measuring an Angle

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

Note If you want a markup entity for the measurement, you must ensure that **Add dimensions** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Angle button  on the Markup toolbar.

2 Click the **Angle** tab.
3 Select **From 3 Points** if you want to measure the angle between three points. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**

See EDA Snapping Modes

4 Select **Between 2 lines** if you want to measure the angle between two lines. **Snapping Modes** are disabled.

5 From the **Measured Angle Units** drop-down 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.
If you selected **Between 2 Lines**, click two lines to define the angle.
Angle arms appear with an arc connecting them.
7 Right-click to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The measured angle appears in the **Measurement** dialog.
8 Click and drag to change the size of the arc.
9 Click and drag the box anywhere on the drawing.
10 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
11 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Measuring an Arc

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

Note To create a markup entity for the measurement, you must ensure that **Add Radius** or **Add Diameter** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the Markup toolbar.

2 Click the **Arc** tab.
3 Select **From 3 Points** if you want to measure the arc between three points. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.
To select all **Snapping Modes** click **All On**. To clear all **Snapping Modes** click **All Off.**

See EDA Snapping Modes

4 Select **Arc Entity** if you want to measure a predefined arc. **Snapping Modes** are disabled.
5 From the **Arc Info** drop-down list, select the unit in which you want to measure the distance of the arc.
6 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.

7 Select **Add Radius** if you want measure the radius.
Select **Add Diameter** if you want to measure the diameter.
8 If you selected **From 3 Points**, click three points to define the arc.
The points are joined by an arc.
If you selected an **Arc Entity**, click an edge of the arc.
The arc is highlighted.
9 Click to complete the measurement.
The measured line path, measurement and unit appear as an 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** dialog.
10 Click and drag the box anywhere on the drawing.
11 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
12 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Calibrating an Arc

1 Measure an arc in the drawing.
See Measuring an Arc
2 In the **Measurement** dialog, click **Calibrate**.
3 The **Distance Calibration** dialog appears displaying the measured distance.
4 From the **Measured Rad.** drop-down list, select the unit to which you want to calibrate the distance.
5 Click **Calibrate to** and enter a value if you want to calibrate to a value.
Click **Set Factor** and enter a value if you want to calibrate by a factor.
6 Click **OK**.
The calibration result appears in the **Measurement** dialog.
7 Click **Close** to close the **Measurement** dialog.

Marking up 3D Files

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 **Escape** to cancel.

3D Snapping Modes

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

The Snapping Modes allow you to snap to vertices, edges, faces, planes and arcs.

Button	Snap to	Description
	Vertex	Vertices are highlighted on the model. When moving the cursor over a vertex, a snap box appears.
	Edge	Edges are highlighted on the model. When moving the cursor over an edge, a snap circle appears.
	Face	Faces are highlighted when you move the cursor over a face and a snap triangle appears.

Attaching a Note

1 Select **Entities > Note** from the Markup menu.
The **Attach to** dialog appears.

Note You can also click the Note button  from the Markup toolbar.

2 In the **Attach to** dialog, click the entity type that you want to attach the note to.

See 3D Snapping Modes

3 Select the entity on the model that you want to attach a note to.
The **Note** dialog appears.

4 Type the text that you want in the note.

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

6 Close the **Note** dialog.
The note symbol appears on the entity and in the **Markup Navigation Tree**.

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

Note To edit a note, double-click on the note to open the **Note** dialog.

Note 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.

Attaching Text

1 Select **Entities > Text** from the Markup menu.
The **Attach to** dialog appears.

Note You can also click the Text button  from the Markup toolbar.

2 In the **Attach to** dialog, click the entity type that you want to attach the text.
See 3D Snapping Modes

3 Click the entity on the drawing that you want to attach text.
The **Text** dialog appears.

4 To change the font, click **Font**. The **Font** dialog appears.
From the **Font** drop-down list, select the type of font.
From the **Size** drop-down list, select the size of font.
Select the font style and type the text that you want to attach, then click **OK**.
The text appears on the entity and in the **Markup Navigation Tree**.

5 Right-click outside the text area to complete the modification.

Note To edit text, double-click on the text to open the **Text** dialog.

Note 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.

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, hidden or deleted.

Note When you are creating a markup entity, you can press **Escape** to cancel.

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. You can access the measurement options from the **Analysis > Measure** menu. The options are:

Name	Description
Angle	Measure the precise angle between three vertices or any two edges, planes or faces. From the toolbar, click the Angle button  .
Arc	Measure the precise radius, length and angle of any arc and calculate the center point location. From the toolbar, click the Arc button  .
Distance	Measure the precise distance between any two Vertex, Edge, Midedge, Arc Center or Face. From the toolbar, click the Distance button  .
Minimum Distance	Measure minimum distance between any two Vertex, Edge, Midedge, Arc Center or Face. From the toolbar, click the Minimum Distance button  . **Note** You cannot create a Markup measure entity when measuring minimum distance.
Edge Length	Measure the precise length of a edge. From the toolbar, click the Edge Length button  . **Note** You cannot create a Markup measure entity when measuring the length of an edge.
Face Surface	Measure the precise surface area. From the toolbar, click the Face Surface button  . **Note** You cannot create a Markup measure entity when measuring surface area.
Vertex Coordinates	Provide the coordinates of each vertex. From the toolbar, click the Vertex Coordinates button  .

Measuring Distance

Use the **Distance** option to measure the precise distance between two vertices, edges, mid-edges, arc centers, faces or any combination of entity types.

Note When measuring the distance between faces, if the faces are parallel, AutoVue compares the shortest distance between the parallel faces.

Note If you want a markup entity for the measurement, you must ensure that **Add dimensions** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Distance button  on the Markup toolbar.

2 Click the **Distance** tab.

3 From the **Measured Distance Units** drop-down list, select the unit in which you want to measure the distance.

4 Click the **From** arrow  to select the entity type that you want to measure from.

5 Select the **Snapping Modes** that you want to use for measuring.
See 3D Snapping Modes
All entities of the selected entity types are highlighted on the model.

6 On the model, select the entity you want to measure from.
Note If you want to take more than one measurement from the same starting point, click **Fix Position**.

7 Click the **To** arrow  to select the entity type that you want to measure to.

8 Select the **Snapping Modes** that you want to select as the end point for the measurement.
All entities of the selected entity types are highlighted on the model.

9 On the model, select the entity you want to measure to.

10 Click again to complete the measurement.
The measured line path appears as an entity on the current active markup.

11 Drag to move the measured line path.

12 Click on the measured line path.
The measurement and unit appear as an entity on the current active markup layer. The measured distance, Delta-X, Delta-Y and Delta-Z appear in the **Measurement** dialog.

13 Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.

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

See Also ***Modifying Measurement Markup Entities***

Calibrating Distance

1 Measure the distance between to points.
See ***Measuring Distance***
2 In the **Measurement** dialog, click **Calibrate**.
3 The **Distance Calibration** dialog appears displaying the measured distance.
4 In the **Measured Distance** drop-down list, select the unit to which you want to calibrate the distance.
5 Click **Calibrate to** and enter a value if you want to calibrate the distance to a value.
Click **Set Factor** and enter a value if you want to calibrate the distance by a factor.
6 Click **OK**.
The calibration result appears in the **Measurement** dialog.
7 Click **Close** to close the **Measurement** dialog.

Measuring Minimum Distance

With the **Minimum Distance** option you can measure the minimum distance between model parts, as well as any two points from the selection sets: vertices, edges, mid-edges, arc centers, faces or any combination of entity types.

Note If you want a markup entity for the measurement, you must ensure that **Add dimensions** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Minimum Distance button  on the AutoVue toolbar.

2 Click the **Min. Distance** tab.
3 Click  **Set 1** point.
4 Click **Entity** if you want to measure the distance between model parts. **Snapping Modes** are disabled.
Click **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.

The model part(s) appears in the list under **Set 1** and are highlighted on the model and in the **Model Tree**.
If you selected **Geometry**, select the **Snapping Modes** that you want to use for measuring.

See 3D Snapping Modes

All entities of the selected entity type are highlighted on the model.
Select the entity or entities on the model.
The selected entities appears in the list under **Set 1** and are highlighted on the model.

Note 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 **Ctrl** key and left-click the part or entity type.

6 Click **Set 2** point.
7 Repeat step 5.
The model part appears in the list under **Set 2**.
8 From the **Measured Min Distance Units** drop-down list, select the unit in which you want to measure the distance.
9 Click **Compute**.
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 as an entity on the current active markup layer. The X, Y and Z coordinates for **Position1** and X, Y and Z coordinates for **Position2** appear in the **Measurement** dialog.
Note Click **Reset** to take another measurement.
10 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Measuring an Angle

Use the **Angle** option to measure the precise angle between three vertices, any two edges, planes or faces or any combination of entity types.

If you want a markup entity for the measurement, you must ensure that **Add dimensions** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Angle button on the Markup toolbar.

2 Click the **Angle** tab.
3 Select the **Snapping Modes** that you want to use for measuring.

See 3D Snapping Modes

All entities of the selected entity types are highlighted on the model.

4 To measure the angle between an entity type and a plane, select the checkbox beside **Plane**, then from the drop-down list select the plane.

5 From the **Measured Angle Units** drop-down list, select the unit in which you want to measure the angle.

6 On the model, click points to define the angle.
Angle arms appear with an arc connecting them.

7 Click again to complete the measurement.
The measured line path, measurement and unit appear as an entity on the current active markup layer. The measurement appears in the **Measurement** dialog.

8 Drag to change the size of the arc.

9 Click and drag the frame handles to change the size of the box.

Note Click **Reset** to take another measurement.

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

See Also Modifying Measurement Markup Entities

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.

Note If you want a markup entity for the measurement, you must ensure that **Add dimensions** is selected in the **Measurement** dialog.

1 Select **Analysis > Measure** from the Markup menu.
The **Measurement** dialog appears.

Note You can also click the Arc button  on the Markup toolbar.

2 Click the **Arc** tab.

3 Select **Arc Entity** if you want to measure a predefined arc.
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. **Snapping Modes** are enabled.
Select the **Snapping Modes** that you want to use for measuring.

See 3D Snapping Modes

All entities of the selected entity types are highlighted on the model.

5 From the **Arc Info Dist. Units** drop-down list, select the unit in which you want to measure the arc distance.

6 From the **Angles Units** drop-down 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.
The points are joined by an arc.
If you selected **Arc Entity**, click the edge of an arc.
The arc is highlighted.
8 Click to complete the measurement.
The measured line path, measurement and unit appear as an 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** dialog.
9 Click and drag the box anywhere on the markup.
Click and drag the frame handles to change the size of the box.
Note Click **Reset** to take another measurement.
10 Click **Close** to close the **Measurement** dialog.

See Also Modifying Measurement Markup Entities

Calibrating an Arc

1 Measure an arc on the model.
See Measuring an Arc
2 In the **Measurement** dialog, click **Calibrate**.
3 The **Calibration** dialog appears displaying the measured distance.
4 From the **Measured Rad.** drop-down list, select the unit to which you want to calibrate the distance.
5 Click **Calibrate to** and enter a value if you want to calibrate to a value.
Click **Set Factor** and enter a value if you want to calibrate by a factor.
6 Click **OK**.
The calibration result appears in the **Measurement** dialog.
7 Click **Close** to close the **Measurement** dialog.

Measuring Vertex Coordinates

The **Vertex Coordinates** option provides the coordinates of vertices on the model.

1 Select **Analysis > Measure** from the AutoVue main menu.
The **Measurement** dialog appears.

Note You can also click the Vertex Coordinates button  on the AutoVue toolbar.

2 Click the **Vertex** tab.
All vertices on the model are highlighted.

3 Double-click the vertex whose coordinates you want to add to the Markup.
The **X**, **Y** and **Z** coordinates appear in a tooltip.

4 Select the vertex.
The **X**, **Y** an **Z** coordinates and unit appear as an entity on the current active markup layer and in the **Measurement** dialog.

5 Click and drag the box anywhere on the markup.
Click and drag the frame handles to enlarge the box.

6 To remove the highlighted vertices on the model, click **Close** in the **Measurement** dialog.
Note You can also remove the highlighted vertices by selecting **Analysis > Measure > Vertex** again.

See Also ***Modifying Measurement Markup Entities***

Printing

With AutoVue, you can print and preview files. Original files can be printed on their own or with associated Markups and Overlays.

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. The properties are:

Option	Description
Print	Define the print options. ***See Configuring the Print Options***
Margins	Define the margin settings. ***See Setting the Print Margins***
Headers/Footers	Define the headers/footers to be included on every page printed. ***See Adding a Header/Footer***
Watermark	Define the watermark to be included on every page printed. ***See Adding a Watermark***
Pen Settings	Change the thickness assigned to a pen. ***See Assigning Pen Settings***

Print Options

From the **Options** tab of the **Print Properties** dialog, you can define options for the printed file.

Option	Type	Description
Printer	Modify	Select a printer and configure print properties, such as paper size, number of copies, etc.
	Paper size	Displays selected paper size.

Option	Type	Description
	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 **Note** Option is available with Java 2 and is only enabled for PDF and word documents.
	Units	Select a unit from the drop-down list to 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	Scales the image according to the scaling factors that you have manually entered in the input fields.
	Scale	Scales the image according to a percentage. You can select a predefined scaling factor or manually enter a custom scaling factor. **Note** Decimal places are accepted.
Alignment/Offset		Specify where the drawing will appear on the printout. Some of the available options are Top-Left, Center-Right, Bottom-Center, etc. **Note** You can define custom alignment by entering a X and Y value.
Document Pages		Specify the document pages to print: **All**, **Current** or **Range**.
Page Area		Select the page area to print.
	Extents	Print the extent of the document.
	Displayed	Print the area displayed in the View window. **Note** Option is only enabled when **Current** is selected for Document Pages.

Option	Type	Description
	Limits	Prints the file limits instead of extents. **Note** Option is only enabled when **Current** is selected for Document Pages.
	Selected	Print selected area. When option is selected, the select button is enabled allowing you to select an area on the drawing. **Note** Option will only be enabled when **Current** is selected for Document Pages.
Force to Black		Force all colors to black.
High Resolution		Prints the file with more definition. **Note** AutoVue Web Edition is based on Java 1.1 to make it compatible with the current browsers. One drawback with Java 1.1 is its limited support for printing -- the print resolution is limited to 72 dpi. AutoVue implements and delivers this limited print capability as part of the package. Cimmetry has identified the need for higher resolution and has implemented a solution for High Resolution and Large Format printers/ plotters output for client machines running on a Windows platform.
Output a single page		Limits output to a single page when the scaling options selected causes a single page to span over several pages.
Print Row Headers		Print row headers. **Note** Option is only enabled for spreadsheets, archive and database files.
Print Column Headers		Print column headers. **Note** Option is only enabled for spreadsheets, archive and database files.
Partial Preview		Display a view of the printer page, highlighting the printable area.

Configuring the Print Options

1 Select **File > Print** from the AutoVue main menu.
The **Print Properties** dialog appears.

Note You can also click the Print button  on the AutoVue toolbar.

2 Click the **Options** tab.

3 Configure the print options.
See ***Configuring Print Options***

4 Configure other print properties.
See ***Setting the Print Margins***
Adding a Header/Footer
Adding a Watermark
Assigning Pen Settings

5 To view a partial view of the file, click **Partial Preview**.
The **Partial Preview** dialog appears.
See ***Partial Preview of a File***

6 Click **OK** to print.
The **Print** dialog appears indicating the document is printing.
See Also ***Previewing a File Before Printing***

Print Margins

From the **Margin** 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.

Setting the Margins

1 Select **File > Print** from the AutoVue main menu.
The **Print Properties** dialog appears.

Note You can also click the Print button  on the AutoVue toolbar.

2 Click the **Margins** tab.

3 Enter the margin 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.

Note When you click **Minimum**, the minimum margins allowable automatically appear in the **Margins**.

4 From the **Units** drop-down list, select the unit in which you want to set the margins at.

5 Configure other print properties.

See ***Configuring Print Options***
Adding a Header/Footer
Adding a Watermark
Assigning Pen Settings

6 To view a partial view of the file, click **Partial Preview**.
The **Partial Preview** dialog appears.

See ***Partial Preview of a File***

7 Click **OK** to print.
The **Print** dialog appears indicating the document is printing.

See Also ***Previewing a File Before Printing***

Headers/Footers

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** drop-down list. Some **Insert Variable** options are: user.name, browser and java.home.

Adding a Header and Footer

1 Select **File > Print** from the AutoVue main menu.
The **Print Properties** dialog appears.

Note You can also click the Print button  on the AutoVue toolbar.

2 Click the **Headers/Footers** tab.
3 Click in the **Left**, **Center** or **Right** box 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 drop-down list.
5 To **Insert Variable** in the header or footer, click **Left**, **Center** or **Right**, then select a variable from the drop-down list.
6 To change the Font, click **Set Font**. The **Font** dialog appears.
From the **Font** drop-down list, select the type of font.
From the **Size** drop-down list, select the size of font.
To set the font **Style**, click **Bold** or **Italic** or both, then click **OK**.
7 In the **Print Properties** dialog, configure other print properties.

See ***Configuring Print Options***
Setting the Print Margins
Adding a Watermark
Assigning Pen Settings

8 To view a partial view of the file, click **Partial Preview**.
The **Partial Preview** dialog appears.

See Partial Preview of a File

9 Click **OK** to print.
The **Print** dialog appears indicating the document is printing.

See Also Previewing a File Before Printing

Native Print Settings

You can choose to included predefined headers and footers when printing an Excel file from AutoVue.

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** drop-down list.
4 Click **OK** to print.
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 horizontally or vertically oriented watermark.

In addition to entering text, you can:

- set the type, size and style of font
- insert Watermark information
- print system variables

To insert Watermark information, AutoVue provides a list of codes that you can choose from. These codes are:

- %f: Full path of document
- %v: Document Drive
- %d: Document Directory
- %b: Document Base name
- %e: Document file extension
- %n: Total document pages
- %p: Current page number
- %N: Total tiled-pages
- %P: Current tile number

- %Y: Date: Year
- %M: Date: Month
- %D: Date: Day
- %W: Date: Day of week
- %H: Time: Hour
- %U: Time: Minute
- %S: Time: seconds
- %r: New line

Note A literal percentage mark is entered manually as %%.

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

Adding a Watermark

1 Select **File > Print** from the AutoVue main menu.
The **Print Properties** dialog appears.

Note You can also click the Print button  on the AutoVue toolbar.

2 Click the **Watermark** tab.

3 Enter the **Watermark Text** that you want to appear on the file.
To insert a code in the watermark, select a code from the **Insert Code** drop-down list.
For example, if you select **%n: Total document pages**, the total number of pages for the selected file will appear in the Watermark.
Note You can insert more than one code.

4 To insert a system variable, select a variable from the **Insert Variable** drop-down list.
For example, if you select **browser.version**, the version of the browser that the file is displayed in will appear in the Watermark.
Note You can insert more than one system variable.

5 To set the orientation of the watermark, click **Diagonal, Horizontal or Vertical**.

6 To change the Font, click **Set Font**.The **Font** dialog appears.
From the **Font** drop-down list, select the type of font.
From the **Size** drop-down list, select the size of font.
To set the font **Style**, click **Bold** or **Italic** or both, then click **OK**.

7 In the **Print Properties** dialog, configure other print properties.
See Configuring Print Options

Setting the Print Margins
Adding a Header/Footer
Assigning Pen Settings

8 To view a partial view of the file, click **Partial Preview**.
The **Partial Preview** dialog appears.
See Partial Preview of a File

9 Click **OK** to print.
The **Print** dialog appears indicating the document is printing.
See Also Previewing a File Before 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.

1 Select **File > Print** from the AutoVue main menu.
The **Print Properties** dialog appears.

Note You can also click the Print button  on the AutoVue toolbar.

2 Click the **Pen Settings** tab.

3 Under **From/To**, select the **Color Index** for which you want to assign a new pen thickness.

Note To change more than one color indices, press the **Shift** or **Ctrl** key while selecting.

4 From the **Units** drop-down list, select the unit that you want to set for the thickness.

5 Click **Thickness**. The **Modify pen thickness** dialog appears.
Enter the new thickness, then click **OK**.
The new **Thickness** appears beside the **Color Index**.

6 In the **Print Properties** dialog, configure other print properties.
See Configuring Print Options
Setting the Print Margins
Adding a Header/Footer
Adding a Watermark

7 To view a partial view of the file, click **Partial Preview**.
The **Partial Preview** dialog appears.

See Partial Preview of a File

8 Click **OK** to print.
The **Print** dialog appears indicating the document is printing.

See Also Previewing a File Before Printing

Creating a New Pen Setting

1 Set the thickness for the color indices that you want.
See Assigning Pen Settings
2 Click **Save As**. The **Save As** dialog appears.
Enter a pen **Name** for your pen setting, then click **OK**.
The new pen setting appears in the **Current Pen Settings** drop-down list.
3 In the **Print Properties** dialog, click **OK** to print.
The **Print** dialog appears indicating the document is printing.

Deleting a Pen Setting

1 Select the pen setting that you want to delete from the **Current Pen Settings**.
2 Click **Delete**.
The pen setting disappears from the list.
3 Click **Cancel** to close the **Print Properties** dialog.

Partial Preview of a File

The **Partial 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**.

1 Configure the print properties.
See Configuring Print Options
Setting the Print Margins
Adding a Header/Footer
Adding a Watermark
Assigning Pen Settings
2 Click **Partial Preview** in the **Print Properties** dialog.

The **Partial Preview** dialog appears.The **Partial Preview** dialog highlights the area that will be printed. The **Paper size**, **Printable Area** and **Drawing Area** are displayed.

3 Click **OK** to close the **Partial 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.

1 View the file in AutoVue.

2 Select **File > Print Preview** from the AutoVue main menu.
The **Print Properties** dialog appears.

Note You can also click the Print Preview button  on the AutoVue toolbar.

3 Configure the print properties.

See ***Configuring Print Options***
Setting the Print Margins
Adding a Header/Footer
Adding a Watermark
Assigning Pen Settings

4 Click **OK**.
The file appears in Print Preview Mode in the **Print Preview** window.
Note 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.

5 Click **Close** to close the **Print Preview** window.

Printing a File

In AutoVue you can print original files along with their Markup files and selected Markup layers together so that they appear as one file.

1 Open the file you want to print in AutoVue.
To print the associated Markups, open the Markup file(s) you want to print. If you are printing Markup file(s) and you want to print the visible layers, select **Modify > Markup Layers** from the Markup menu, then from the **Markup Layers** dialog select the Markup layers you want visible.

2 Select **File > Print** from the AutoVue main menu.

The **Print Properties** dialog appears.

Note You can also click the Print button  on the AutoVue toolbar.

3 Configure the print properties.

See ***Configuring Print Options***
Setting the Print Margins
Adding a Header/Footer
Adding a Watermark
Assigning Pen Settings

4 To view a partial view of the file, click **Partial Preview**.
The **Partial Preview** dialog appears.

See ***Partial Preview of a File***

5 Click **OK** to print.
The **Print** dialog appears indicating the document is printing.

See Also ***Previewing a File Before Printing***

Batch Printing

The **Batch Print** option lets you 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.

1 Select **File > Batch Print** from the AutoVue main menu.
The **Batch Print** dialog appears.

2 To add a file to the **List of files to be printed**, click **Add**. The **File Open** dialog appears.
Enter a **File Name** or click browse to locate the file you want to add, then click **OK**.
The file(s) appears in the **List of files to be printed**.
Note To add more files to the list, repeat steps 2 to 4. To remove a file(s), select the file from the **List of files to be printed** and click **Remove**.

3 Click **Print** in the **Batch Print** dialog.
The **Print Properties** dialog appears.

4 Configure the print properties.

See ***Configuring Print Options***
Setting the Print Margins
Adding a Header/Footer
Adding a Watermark
Assigning Pen Settings

5 Click **OK** to print.
The **Print** dialog appears indicating the document is printing.

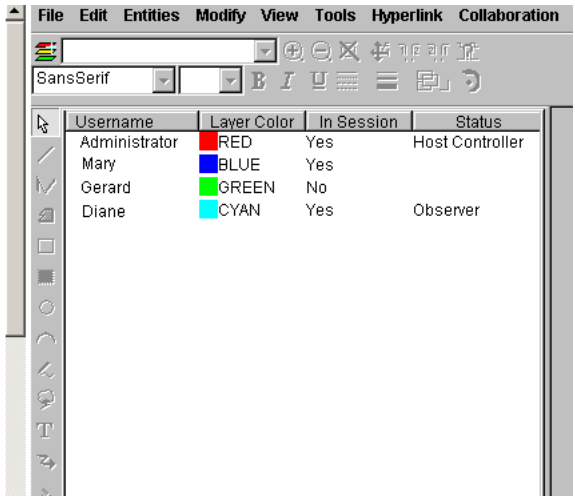
Collaboration

Collaboration option enables multiple users to review files interactively and simultaneously, thereby shortening the review process for documents.

Participants in a Collaboration Session assume different roles. Each Collaboration Session has a Host, a Controller and one or more users. These participants have varying privileges during the Collaboration Session.

Collaboration User Tree

When you are in a Collaboration Session, the bar on the left-hand side of the screen displays the **Users** and **Markup Tree** tabs, in addition to the tabs displayed when you are normally viewing a file. The **Users** tab displays the **Username**, the **Layer Color** representing the user, the **In Session** value and the user **Status**. The **Markup Tree** tab displays the Markup entities added to the file.



The Host

The person who initiates the Collaboration Session is automatically the acting Host and Controller by default. The Host and Controller are not always the same

person. The acting Host can assign another user to be the Host by selecting **Assign Host** from the **Collaboration** menu.

The Host is the only one who can save the Session Markup. When initiating a Collaboration Session, the Host can specify the Markup to open; the host can also open a Markup file during a session. The Host can invite other users to join the Collaboration Session during the session or at initiation of the session.

The Controller

The **Controller** is the person who controls the base file's view during the Collaboration Session. Other participants in the Collaboration Session can choose between displaying or hiding the Controller's View changes.

When a session is initiated, the Controller is also the Host. The Controller can change the base file anytime during the Collaboration Session. However, when the Controller is no longer the Host, opening a new file has to be approved by the acting Host.

At any time during a session, a participant who is not an Observer can request control of a Collaboration Session.

Lock

Any user who is not the Controller of the Collaboration Session can select **Lock View** from the **Collaboration** menu or click the Lock View button  in the lower left part of the screen to display the Controller's View modifications on the base file and the other participants' modifications on the Markup file made during the Collaboration Session.

When you lock your view, the file displayed will have the same Configuration options the Controller has set in the **Configuration** menu.

Note For all users, except for the Controller, view options are disabled.

Observer

Observers can take part in a Collaboration Session but cannot create Markup entities. They can observe and make comments in the Chat window. The Host can designate a user as an Observer when he invites him to the Collaboration Session. A user can also decide to take part as an Observer in the Collaboration

Session by selecting **Observer** in the **Join Session** dialog. There can be more than one Observer during a Collaboration Session.

Open a File

The Controller is the only user who can open a file during a Collaboration Session. To open a URL, select **File > Open URL** from the menu. To open a local file, select **File > Open Local File** from the menu. The **File Open** dialog box appears, letting you specify the file to open.

When AutoVue Web Edition is integrated with a Document Management System (DMS), clicking **Open URL** opens dialog that lets you view a DMS file. You can work on this file opened through **Open URL** during the Collaboration Session. When the Host opens a URL, he or she is prompted to save the Session Markup from the Collaboration Session that was in session. When the Controller opens a URL, the Host still owns the Collaboration Session and is the only one who can save the Session Markup.

Pointer

The **Pointer**  is the cross-hair marker the Controller uses to point to a specific place in the file.

Session Information

The **Session Information** option is available from the **Collaboration** menu during a Collaboration Session. It displays the **Session Subject** (the name assigned to the Collaboration Session), the **Session ID**, the name of the base file viewed, the **Host** and the list of users participating.

Session Markup

This is the Markup file created during the Collaboration Session. All participants can add Markup entities to the Session Markup file.

Note Only the Host can save the Session Markup. **Save** and **Save As** are disabled for all other users.

Show Tracker

The **Show Tracker** option in the **Collaboration** menu is available to any user whose view is set to unlocked. When you select **Show Tracker** from the **Collaboration** menu, the **Collaboration View** window appears displaying a bird's eye view of the base file viewed and the Markup changes done during the Collaboration Session.

When you use **Show Tracker**, you can simultaneously track the Controller's base file view changes and everyone's Markup changes in the **Collaboration View** window while your main window display remains unaffected by anyone's modifications. It is like having a locked view and an unlocked view on hand at the same time.

Collaboration Session

The **Collaboration** menu provides you with a means to initiate a Collaboration Session or join an existing session. When you initiate a Collaboration Session, a session object is created on the server and you are the Host and Controller of the session. The Host owns the Session Markup and is the only one who can save it and open an existing Markup. The Host can also invite other users during a session.

Initiating a Collaboration Session

When you initiate a Collaboration Session, a session object is created on the server. The session object is maintained for the duration of the Collaboration Session. When you initiate a session from View mode, you are the Host and the Controller of the session.

1 Select **Collaboration > Initiate Session** from the AutoVue main menu. The **Initiate Session** dialog appears.
2 Under **Session Subject**, enter a session name.
3 If you want to open a Markup for the Collaboration Session, click **Browse**.
See Opening a Markup While Initiating a Session
4 Click **Public** if you want the Collaboration Session visible to others who may want to join the session.
Click **Private** if you want the Collaboration Session only visible to the participants.
5 If you want to set a password for the Collaboration Session, enter a **Password**, then enter it again to **Confirm**.

Note If you selected **Private** and chose not to set password, when you click **OK**, a confirmation dialog appears asking if you are sure you do not want to set a password.

6 From the **Users** list, select the users that you want to invite.

7 Click **Add**.
The user you selected appears in the **Invited** list.
Note To remove a user, select the user from the **Invited** list and click **Remove**.

8 To designate a user as an **Observer**, select the checkbox under **Invited**.

9 Click **OK**.
AutoVue goes into **Collaboration** mode and the session is started. The user(s) that are invited receive a tooltip notification that he or she has been invited to join the session. The notification message indicates Session Name, File and Initiator.

Changing a User's Layer Color of a Session

1 In the **Initiate Session** dialog, click **Layer Color**.
The **Layer Color** dialog appears.

2 Select a **Layer Color** from the drop-down list or select **Let user choose** if you want the user to choose their own color.

3 Click **OK**.
In the **Initiate Session dialog**, the selected **Layer Color** appears beside the **Username** in the **Invited** list.
If you selected **Let user choose**, a custom color  appears beside the **Username**, indicating that the color can be changed **By User**.

Adding New Users to a Session

1 In the **Initiate Session** dialog, click **Add New**.
The **Add User** dialog appears.

2 Enter **Username**.

3 Select **Observer** if you want to designate the user as an Observer.

4 Select a **Layer Color** from the drop-down list or select **Let user choose** if you want the user to choose their own color.

5 Click **OK**.
The new user is added to the **Invited** list in the **Initiate Session** dialog.
Note To add more than one user, repeat steps 1 to 5.

Opening a Markup While Initiating a Session

1 Select **Collaboration > Initiate Session** from the AutoVue main menu.
The **Initiate Session** dialog appears.
2 Click **Browse**.
The **Open markup** dialog appears.
3 In the **Markup list**, select the checkbox beside the Markup that you want to open.
4 To import a Markup file, click **Import**.
The **Select markup file to import** dialog appears.
5 Locate the file that you want to open.
6 Click **Open**.
7 Click **OK** in the **Open markup** dialog.
8 In the **Initiate Session** dialog, select **Session Markup** if you want to open a Session Markup and keep the layer colors and Markup entities from the Collaboration Session.
9 Click **OK**.
The selected Markup appears in the **Initiate Session** dialog.

Opening a Markup During a Session

The Host can open Markups during a Collaboration Session. These Markups can be session markups created during Collaboration Sessions or any non-session Markup files.

1 During a Collaboration Session, select **File > Open** from the AutoVue main menu.
A confirmation dialog appears, prompting you to save the current Session Markup.
2 Click **Yes** or **No**.
The **Open markup** dialog appears.
3 Click the checkbox beside the Markup that you want to open.
4 To import a Markup file, click **Import**.
The **Select markup file to import** dialog appears.
5 Locate the file that you want to open.
6 Click **Open**.
7 In the **Open markup** dialog, select **Session Markup** if you want to open a Session Markup and keep layer colors and Markup entities from the Collaboration Session.
8 Click **OK**.

The selected Markup appears in the Collaboration window.

Joining a Session

Note If you do not want to receive any notifications to join Collaboration Sessions, select **Collaboration > Do Not Disturb** from the AutoVue main menu.

1 Select **Collaboration > Join** from the AutoVue main menu.
The **Join Session** dialog appears.

Note You can also click the Join Session button  on the status bar at the bottom of the workspace.

2 From the **Session ID** drop-down list, select the active session you want to join.
3 Enter the **Session ID** of the session you want to join.
4 Enter the **Password** if the Host has set one.
5 If you just want to be an **Observer**, select the checkbox.
6 If you want to select a layer color, click **Layer Color**.
The **Layer Color** dialog appears.
Note The **Layer Color** is disabled if the Host did not select **Let user choose** when initiating the Collaboration Session or when you are invited to join the session.
7 Select a color from the drop-down list.
8 Click **OK**.
9 Click **OK** in the **Join Session** dialog.
You have joined the session.

Inviting Users During a Session

1 Select **Collaboration > Invite** from the AutoVue main menu.
The **Invite** dialog appears.
2 From the **Users** list, select the user(s) that you want to invite.
3 Click **Add**.
The user(s) you selected appears in the **Invited** list.
Note You can add user(s) that are going to join the session at a later time by clicking **Add New** and entering the **Username**.
4 Click **OK**.
The user(s) receives a tooltip notification that he or she has been invited to join the session.

Removing Users from a Session

1 Select **Collaboration > Invite** from the AutoVue main menu.
The **Invite** dialog appears.
2 From the **Invited** list, select the user(s) that you want to remove.
3 Click **Remove**.
The user(s) you selected appear in the **Online** list.
4 Click **OK**.
The user(s) receives a tooltip notification that he or she is no longer invited to the session.

Assigning a Host During a Session

1 Select **Collaboration > Assign Host** from the AutoVue main menu.
The **Assign Host** dialog appears.
2 From the **Users** list, select the user that you want to assign as host.
3 Click **OK**.
The user you selected is now the host of the Collaboration Session.

Granting Control of a Session to Another User

When you initiate a Collaboration Session, you are the default Controller of the session until you relinquish control to another user. You may grant control to another user any time during the session. Once you grant control to another user, all other users will automatically lock their views to the new Controller.

1 Select **Collaboration > Grant Control** from the AutoVue main menu.
The **Grant Control** dialog appears.
2 From the **Users** list, select the user that you want to grant control to.
3 Click **OK**.
The user you selected has control of the Collaboration Session.

Saving a Session Markup

The Host owns the Session Markup and is the only one who can save it.

1 At the end of the Collaboration Session, select **File > Save As** from the AutoVue main menu.
The **Save Markup File As** dialog appears.
2 Enter the **Markup ID**.

Note The **Markup Information** is optional.

3 Click **OK**.
The session Markup is saved.

Closing a Session

Only a Host can close a Collaboration Session.

1 Select **Collaboration > Close Collaboration Session** from the AutoVue main menu.
The **Save Markup** dialog appears prompting you to save the Session Markup.
2 Click **Yes**.
All users are notified that the Collaboration Session is closed.

Requesting Control of a Session

1 Select **Collaboration > Request Control** from the AutoVue main menu.

Note You can also click the Wand button  on the status bar at the bottom of the workspace.

The Controller receives a message that you have requested control.
2 The Controller clicks **Yes.**
You become the Controller of the Collaboration Session.

Note If the Controller does not respond within 10 seconds, control is automatically granted to the user who requested it.

Tracking Changes

The **Show Tracker** option allows you to simultaneously track the Controller's base file view changes and everyone's markup changes while your markup file remains unchanged.

Note You must be in an unlocked state before selecting **Show Tracker**.

1 Select **Collaboration > Show Tracker** from the AutoVue main menu.
You can now track changes without affecting your view.

Unlocking a View

If you do not want to see the Controller's view changes, clear the Lock View.

1 Select **Collaboration** and deselect **Lock View** from the AutoVue main menu.
The checkmark disappears beside the **Lock View** option.

Note You can also click the Unlock View button  on the status bar.

2 Select **Collaboration > Show Tracker** if you want to simultaneously track the Controller's base file view changes and everyone's markup changes while your markup file remains unchanged.
You can now track changes without affecting your view.
Note The **Show Tracker** option is only available when a view is set unlock.

Locking a View

The **Lock View** option allows you view changes the Controller is making. When you select the **Lock View** option, you are also propagating your Markup modifications to the other participants in the session.

Note **Lock View** is the default for all participants in a session.

1 Select **Collaboration > Lock View** from the AutoVue main menu. Lock View set by default.

Note You can also click the Lock View button  on the status bar at the bottom of the workspace.
A checkmark appears beside the **Lock View** option and the Lock View button  turns red on the status bar.

Viewing Session Information

1 Select **Collaboration > Session Information** from the AutoVue main menu.
The **Session Information** dialog appears displaying the **Subject**, **Session ID**, **File Name**, **Host** and **Users** of the session.
2 Click **OK**.

Leaving a Session

Only a Host can close a Collaboration Session. All other participants can leave a Collaboration Session and rejoin at a later time if it is still in session.

1 Select **Collaboration > Leave Collaboration Session** from the AutoVue main menu.
The other participants are notified that you left the Collaboration Session and AutoVue returns to **View** mode.

Chat Window

The **Chat Window**, is a tool you can use to communicate with other users. You send each other messages via the **Chat Window**. The **Chat Window** is also useful in giving you Collaboration Session updates such as which users are present and which users have joined or left the session.

At the end of the Collaboration Session, a transcript of the **Chat Window** is saved as a **Note** entity along with the Markup as the Collaboration Session Markup file. The **Note** entity contains session information such who the Host was, who was invited and session time and duration.

The **Chat Window** contains these menu options:

Menu	Option	Description
File	Send Message	Sends text messages to other users.
	Print	Prints the contents of the Chat window.
	Quit	Closes the Chat window.
Edit	Copy	Copy text from the Chat window.
	Clear	Clears all the text in the Chat window.
	Select All	Selects the entire text in the Chat window.
Options	Users	Displays the users in the Collaboration Session.

Sending a Message to Selected Users

1 Select **Collaboration > Chat Window** from the AutoVue main menu.
The **Chat** dialog appears.

Note You can also click the Chat Window button  on the status bar at the bottom of the workspace.

2 Click **Selected users**.
A list of **Users** appear in the **Chat** dialog.

3 Under **Users**, select the user(s) that you want to send the message to.
Note To select more than one user, press the **Shift** or **Ctrl** key while selecting.

4 In the **Send Message to:** type your message.

5 Click the Send Message  button.
The private message is sent to the selected users of the Collaboration Session.

Sending a Message to All Users

1 Select **Collaboration > Chat Window** from the AutoVue main menu.
The **Chat** dialog appears.

Note You can also click the Chat Window button  on the status bar at the bottom of the workspace.

2 Click **All Users**.

3 In the **Send Message to:** type your message.

4 Click the Send Message  button.
The message is sent to all users in the Collaboration Session.

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