Version 3.2 User Manual

Copyright 2000-2004 Holbrook Enterprises, Inc.

24421 Chestnut Street Suite 101

Newhall, CA 91321

(661) 253-4370

WireCAD is a trademark of Holbrook Enterprises, Inc.

All other trademarks are the property of their respective owners.

Installing WireCAD Version 3

Insert the CDROM. The WireCAD installer should start automatically. If not, open the CDROM drive in explorer, or run the set-up utility setup.exe from the command line.

If you are installing WireCAD for the first time, WireCAD will install to the location of your choosing.

The WireCAD installer will also repair damaged installations. Simply re-launch the setup program. Note that this can have undesirable results, as it will replace the databases that you may have modified with clean versions.

Operating System Specific Issues

Operating System	
Windows 95	Unsupported
Windows 98	Unsupported
Windows 98SE	Unsupported
Windows NT 4	Unsupported
Windows ME	Unsupported
Windows 2000	Supported.
Windows XP All Flavors	Supported.

WireCAD Program Levels Authorization

WireCAD Personal	Authorization Code Required. Available from http://www.WireCAD.com.
WireCAD Professional	Authorization Code Required. Available from http://www.WireCAD.com.
WireCAD Professional Trial	Enter 0656-0464-3889-2491-3627-4434-1030 in the authorization code field and click Authorize to activate the professional mode for 15 days.
WireCAD Enterprise	Authorization Code Required. Available from http://www.WireCAD.com.

How to Obtain Technical Support

Please visit the WireCAD Support Forum at www.wirecad.com

Support contacts: support@wirecad.com or phone at: (661) 253-4370.

When you first purchase WireCAD you will receive free support for 30 days from the time of your purchase. The User Support forum on the WireCAD website is available to all users, and provides a peer-to-peer moderated support forum.

WireCAD Assurance Programs

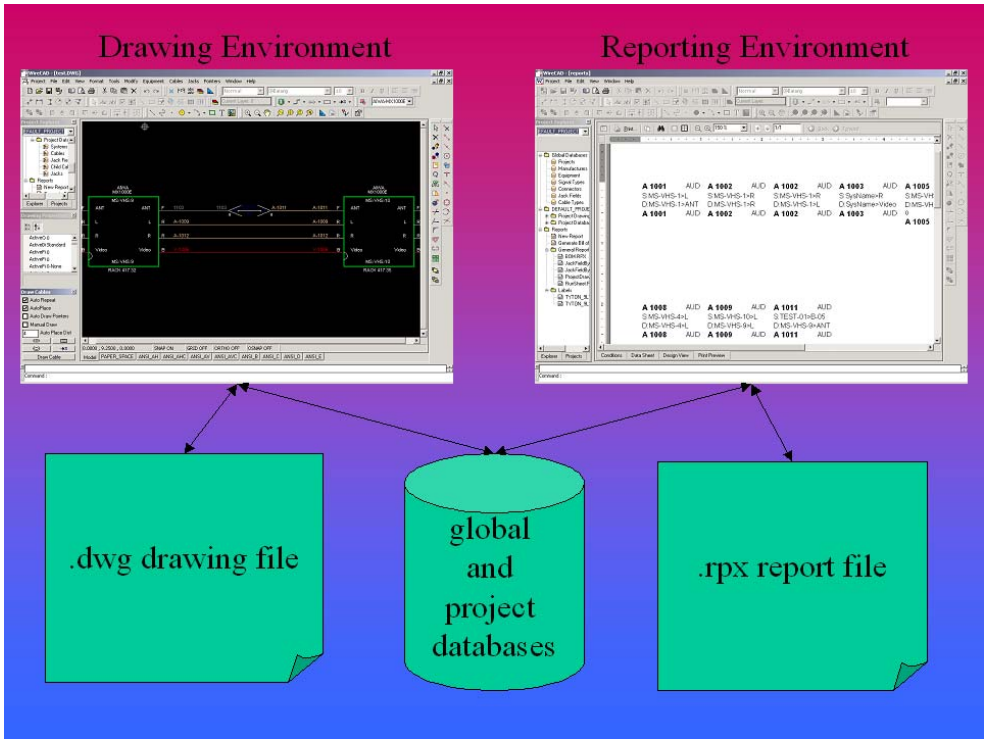
Level	Description	Monthly Cost
Silver	Receive all minor updates for free, 2 phone support occurrences* per anum, unlimited email support.	$9.99/seat
Gold	Receive all minor updates for free, unlimited phone support occurrences, unlimited email support, access to all advanced tutorials.	$24.99/seat
Platinum	Receive all minor updates for free, unlimited phone support occurrences, unlimited email support, access to all advanced tutorials, free webex support. All **Major** updates for free!	$49.99/seat

When calling for technical support, please have your Customer Assurance Identification Number. Call us at the number listed above.

* Support Occurrence is defined as any number of phone or email contacts required to reach a resolution for the user's question or concern. A Support Occurrences will further be defined as any question or concern that does not result in the discovery of a deficiency in the software. You will not be penalized for software bugs.

Please note that we will not be able to respond to your support requests unless your assurance account is current and in good standing.

WireCAD Concept



WireCAD 3 integrates drawing, reporting, and data environments.

What is WireCAD?

WireCAD is a cable management and facility design tool that allows you to easily create AutoCAD™ drawings. WireCAD maintains a database of equipment, from which you can create equipment blocks for your drawings. Equipment blocks are created dynamically from information stored in the equipment database. Rather than maintaining a large library of equipment blocks or symbols, WireCAD stores this information in a database and then creates blocks from the equipment definitions contained therein. Equipment definitions are easily added to the database. In addition to equipment databases, WireCAD also provides drawing tools to rapidly create documentation, and database management tools to track:

- Projects.
- Drawings.
- Revisions.
- Cable Types.
- Signal Types.
- Connectors.
- Jack Fields.
- Jacks.

The biggest time saver comes when it is time to assign System Names and Cable Numbers to the equipment in your drawing. No longer do you have to extract all of the attributes of your drawing to a file, and then import that file into a spreadsheet for ID assignment, and then get out your highlighter pen and your drawing as you manually modify your drawing using the information from your spreadsheet. Now, all you do is double-click on the equipment pieces in the drawing to assign them a system name. Then double-click on the cable and assign it a cable number. All of the information regarding the selected cable is extracted from the drawing and placed in the project cables database and the drawing is updated with a new cable number.

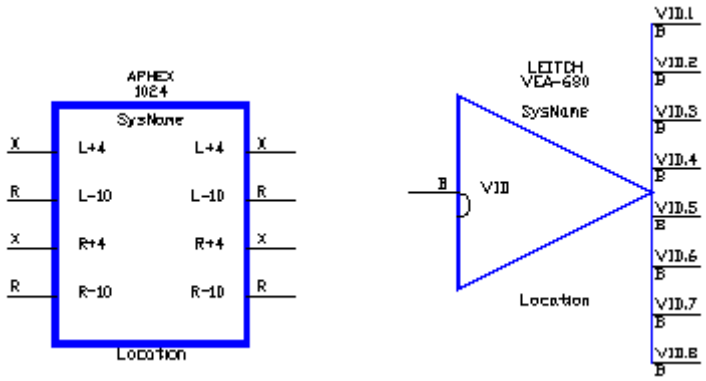
Extensive reporting is available for the project databases including:

- Project drawings.
- Project revisions.
- Cable run sheets.
- Cable kill sheets.
- Cable labels.
- Equipment lists.
- Bill of Materials.

In addition, a powerful report designer is included with WireCAD for creating your own reports and labels, or modifying existing report definition files.

Terminology

WireCAD: A fully integrated drawing, reporting, and cable management tool that aids in the production of single line drawings and database management.



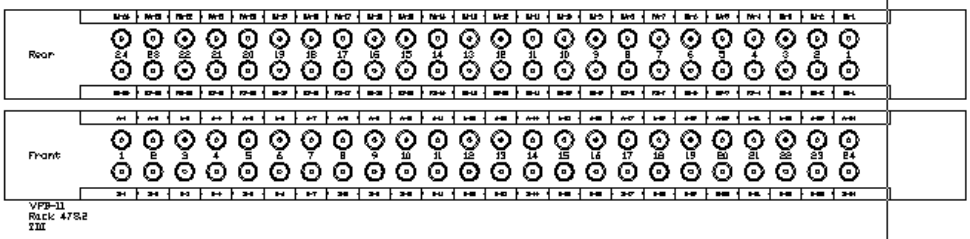
Equipment Block: A representation of a piece of technical equipment to be connected to another piece of equipment. These are created dynamically from the Equipment Library in WireCAD. The basic characteristics of the block such as width and pin spacing can be edited in the Preferences dialog.

AutoCAD Block: A named group of drawing entities that can be inserted into the drawings any number of times. As an example, group of entities that represents a door could be inserted into the drawing multiple times.

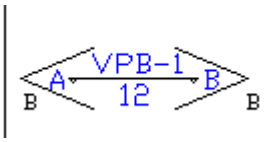
Insert: An instance of a Block.

Assignment or Cable Assignment or System Assignment: The process of retrieving information from the drawing, checking it against existing database entries, and finally appending the database with the data from the drawing. This data is then available for reporting.

Current Project: This is the currently active project in WireCAD. This directs WireCAD to the location for the project drawings and database.

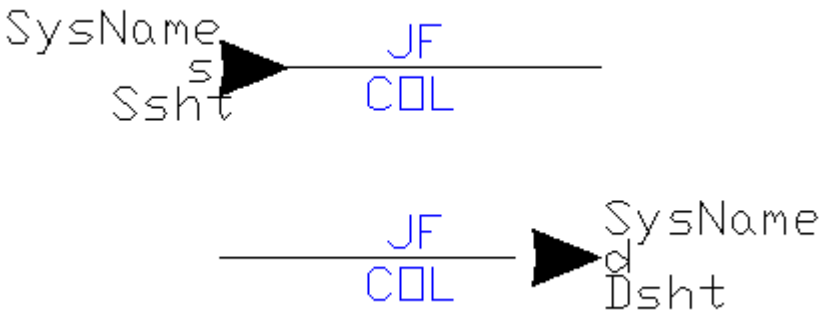


Jack Field: An array of Jacks mounted to a single panel. Jack Fields must be assigned to Projects. This is different from adding a Jack Field to the library. Not until a Jack Field is assigned to the project are all of the Jacks within that field available for assignment.

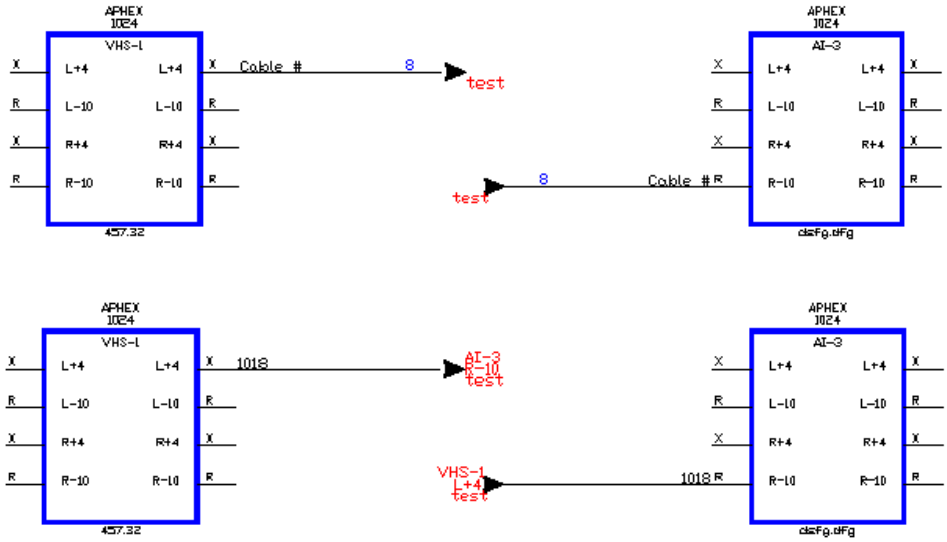


Jack: The electrical representation of a Jack Field point. WireCAD provides three types of jack for you to choose from. The above Jack actually represents two Jacks in a full normal configuration. The A and B Represent the top and bottom rows, VPB-1 is the system name and 12 represents the 12th column in the array.

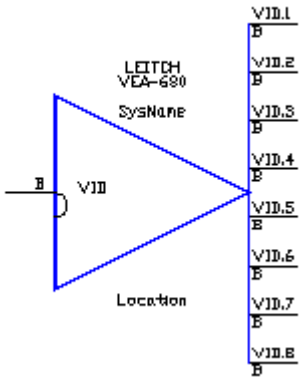
Terminal: Any junction or splice in a cable. This could be a single bulkhead connector on a service panel, or a set of points in a high density connector.



Unlinked Pointers: A Pointer to another system in the project. Pointers can be Linked across drawings, or in the same drawing. A cable cannot be assigned to the database until any Pointers on the cable are Linked together.



Linked Pointers: A Linked Pointer is represented by red text and a number in the JF attribute. The Pointers in the top drawing are Linked together. The blue 8 is there for your convenience; the red test indicates that the opposite Pointer is contained on the sheet 'test.' With the Pointers Linked, WireCAD will assign the cable even if the Source and Destination are on separate sheets. You must open the drawings where the Source and Destination reside. Once the cable is assigned a Cable Number, WireCAD will automatically fill in the rest of data shown in the Pointers.



Distribution Amplifier (DA**)**:

Equipment Type: Used in the Add Equipment Dialog. The Equipment Type is used as a prefix to the system name. As an example, if you enter a piece of gear with the Equipment Type VTR, then WireCAD uses VTR as the prefix for the system name. So the first instance of the piece of gear would be VTR-01. Try to use friendly names for the Equipment Types. For example an Avid non linear editor is easily identified as an Avid-1. Instead of non-linear editor, or NLE-1.

Note:
WireCAD imposes a string length of 32 characters on the length of System Names or I/O labels. Keep in mind, however, that if you are planning to print cable labels, and if your names are too long, the report may truncate them, thereby rendering them unreadable or ambiguous.

Application Path: The path where WireCAD is installed (alternately referred to as InstallPath). See the section on Support Paths.

Project Path: The path for the Currently active Project. See the section on Support Paths.

Report Definition File: A file that contains the design for a report. This file also contains the initial data Source for the report. The format for the Report Definition File is XML.

Typical Work Flow

A typical workflow might include:

- Concept level drawings with just enough information to complete an approval process.
- Bid level drawings with all of the necessary information to go to bid.
- Construction documentation with associated cable and system databases. This is the most tedious process and is made easy by WireCAD.

The Process

The user starts by creating a project in WireCAD and then adding drawings to the project. Once a drawing is created in WireCAD it has all of the necessary layers, block, and text styles to accept input from WireCAD . Next the user opens the “Equipment Library” from WireCAD and selects a piece of equipment for placement in the drawing. The user has the ability to dynamically choose the configuration in which the piece of equipment is represented. i.e. if you are creating a drawing that only displays the video connections of a system, then all of the equipment in the drawing can be represented with only the video connections displayed. Equipment that is not currently in the database can be added very quickly using the “Add Equipment” and “Edit I/O” functions. Once the user has placed equipment in the drawing, they then connect the equipment through any terminal gear that is needed, such as Jacks, J-Boxes and Routers. WireCAD helps to automate this process by providing a set of tools to speed the process. At this point you should have a pretty drawing of your system, but WireCAD has only just begun. Next you assign system names or SysNames to all of the equipment in the drawing. During this process, you will also be prompted for the location of the piece of equipment. WireCAD also supports system aliases or function names. Once you have assigned system names to all of the equipment in the drawing it is time to assign cable numbers to the cables. WireCAD automates this process as well. All you have to do is double-click the cables in the drawing. WireCAD then gets the information from the drawing and checks it against the information already stored in the database. If no duplicates are found then the cable information is sent to the Project database. WireCAD then assigns a cable number and modifies the drawing. Now you have a pretty drawing with all of the system info and all of the cable numbers. Now it is time to build the thing. WireCAD will produce cable run sheets and cable labels, as well as, and equipment lists.

Typical Drawings

The type of and quantity of drawings required for a given project is highly subjective based on the size of the project, the size of the printed drawing, and the density of the drawings. Other considerations will be the requirements imposed by the client. That said, most integration documentation includes the following:

- Connector pin-outs.
- Connector wiring specifications.
- Floor plans.
- Cable chase plans and elevations.
- Single line drawings for all cables.
- Jack Field layouts and designations.
- Patch panel and J-Box mechanical specifications.
- Drawing Conventions.
- Title page.
- Drawing table of contents.
- Cable riser diagrams.
- Rack and furniture elevations.

In addition to the above drawings, you will need to provide documentation to the wiring crews regarding cable runs, and labels. Power consumption, weight, and heat load data should be compiled for the mechanical and electrical engineering staff

.

Databases

WireCAD uses the Microsoft JET engine 4.0 (Access 2000 databases) to perform all of the data storage. There are two databases that reside in the WireCAD folder in the Installation path. The first one is named dbWirePM.mdb. This database maintains pointers to, and meta-data on, all of the projects. The second is named dbEquip.mdb, and contains all of the equipment libraries. A third database is the project database that is contained in every project directory in a sub folder named Project Databases. This database is duplicated for all new Projects, and contains all of the project specific System Names, and Cables, Drawing pointers, etc.

Enterprise Client Edition Note:

Depending on your system configuration, the dbPM.mdb and dbEquip.mdb must be located on a server or network share.

Microsoft Access 2000

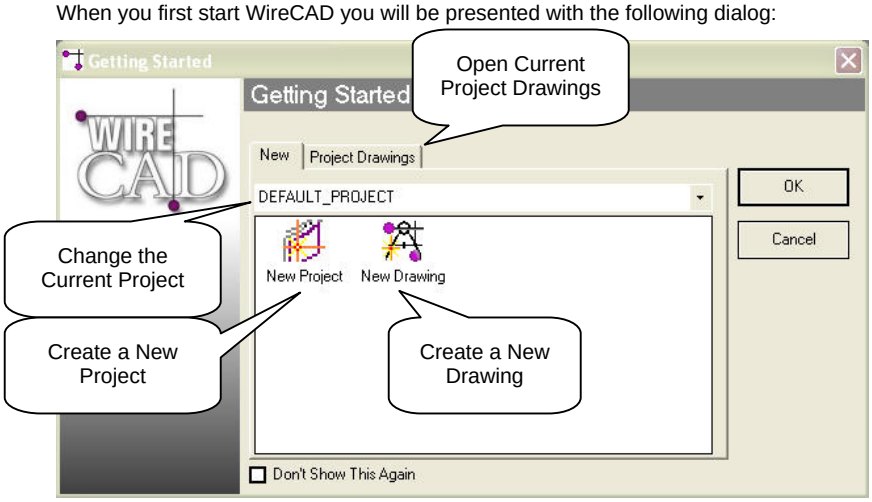
If you own a copy of Microsoft Access 2000 then you will have the ability to add custom reports and labels, as well as custom queries.

Note: if you are using Access 2000 to create a report in a Project Database, the report will be specific to that database unless you copy it, and paste it to another Project Database using the clipboard.

Note: If you use Access 2000 instead of the Report Designer provided in WireCAD, then you will not be able to print these reports from within WireCAD.

The Getting Started Dialog

When you first start WireCAD you will be presented with the following dialog:

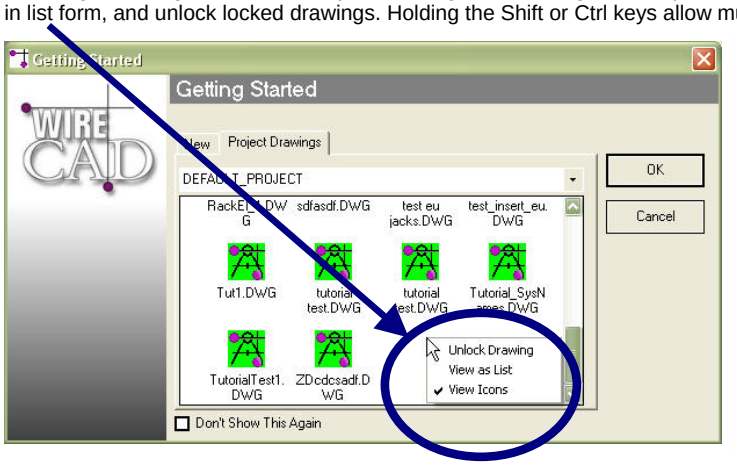


Use the Getting Started dialog as a shortcut when getting started. You can create a New Project, or New Drawing.

The drop-down at the top will change the Current Project.

Switch to the Project Drawings tab for a list drawing associated with the Current Project.

Note: right-clicking while in the Project Drawings tab drawings list will provide a context menu that allows you to display the drawings in list form, and unlock locked drawings. Holding the Shift or Ctrl keys allow multiple selection of Project Drawings.



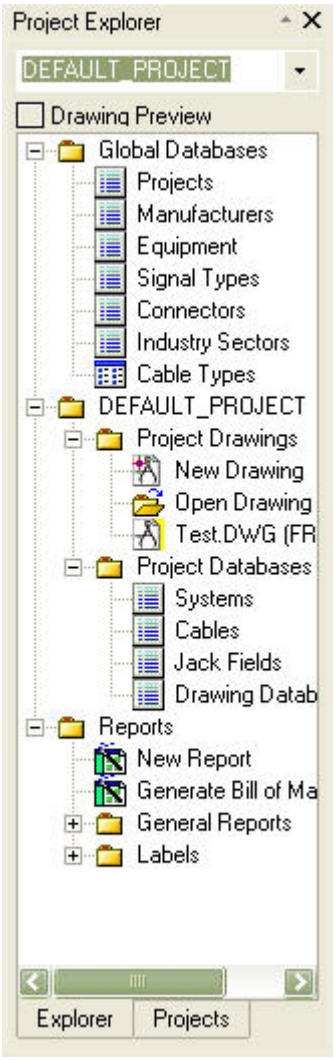
Drawing Icon Color Legend

Green: Drawing found at specified path and available for use.

Yellow: Drawing found at specified path and in use by another user.

Red: Drawing not found at specified path. Possibly renamed or moved.

The Project Explorer



This dockable form provides access to many functions allowing you to open and edit Global and Project Databases, Open and create New Drawings. Open and create New Reports, and most importantly select the current Project Database

Start here to create new drawings

View Drawings associated with this project

Green icons indicate available drawings

Red icons indicate missing drawings

Yellow icons indicate drawings in use

Create new reports

Open existing reports

Selecting the Current Project Database

Use the pull down combo box at the top to set the current project database.

Note: the Personal version only has access to the Default Project.

Global Databases

Global Databases are those that are common to all projects. As follows:

- Manufacturers
- Equipment
- Signal Types
- Connectors
- Cable Types
- Jack Fields

Project Drawings

Create New Drawing

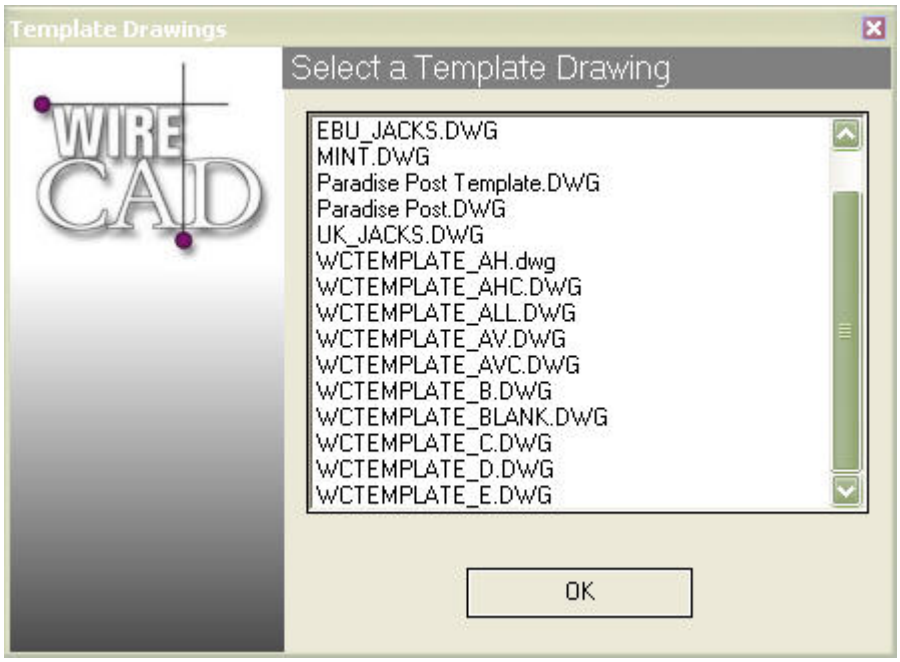
Double-click this icon 

on the Project Explorer or  on the Standard toolbar. You will be prompted to select a WireCAD template drawing.

Template Drawings

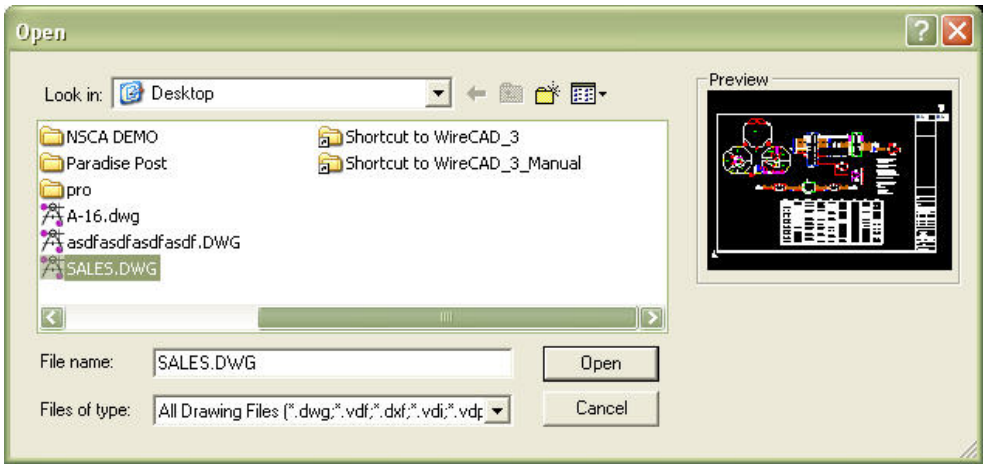
When you create a new drawing in WireCAD you are prompted to select from a number of template drawings. These drawings are prepared to receive input from WireCAD. The template drawings also provide a default layout for the ANSI drawing size indicated. To create your own templates, modify an existing template and save the drawing in the WireCAD3\TemplateDrawings folder.

NOTE: You can customize a template drawing with all of your default company and project information. Then save the drawing into the \WireCAD3\TemplateDrawings folder and it will be available from this view. Now you will not have to enter redundant information and can insure symmetry between drawings.



Open Existing

Click on the Open Drawing folder in the Project Explorer or the button on the Standard toolbar. This will launch an Open dialog:



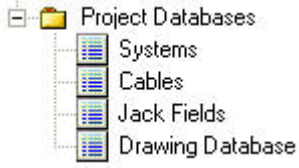
Browse to the location of the drawing and click Open.

Any drawing that you open in WireCAD is automatically modified to include the WireCAD specific Layers, Blocks, and Text Styles.

Associated Drawings

Note: Any drawing opened in WireCAD is automatically associated with the current project in the drawings database. Additional fields are available for editing and reporting.

Project Databases



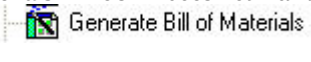
Open project databases in datasheet view including:

- **Project Systems:** Listing of all systems associated with the Current Project. Systems are added to this database by double-clicking on equipment in a drawing.
- **Project Cables:** Listing of all cables associated with the Current Project. cables are added to this database by double-clicking on cables in a drawing.
- **Project Jack Fields/Jacks:** Not Used in Version 3. Retained for backward compatibility.
- **Drawing Database:** Pointers and meta-data to all drawings associated with the Current Project. Drawings are added to this database when they are opened in the Current Project.

Reporting

New Report: 

Create a new report definition file using the new Report Wizard.

Generate Bill of Materials: WireCAD does not maintain a running bill of materials but generates the Bill of Materials on demand. Clicking this icon 

launches the Bill of Materials Wizard to create a new bill of materials or edit an existing one.

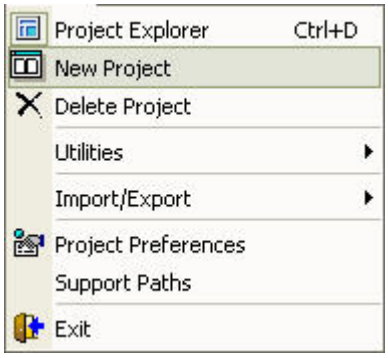
General Reports: Open an existing report definition file. To add a file to the General Reports list, place the .RPX file in the WireCAD Path\Reports\General\

Labels: Open an existing Label definition file. To add a file to the Labels list, place the .RPX file in the WireCAD Path\Reports\Labels\

WireCAD 3 Levels Defined

Level	Multiple Projects	Multiple Users	Report Designer	Unlimited Cables	Unlimited Drawings
Personal			X	X	X
Professional	X		X	X	X
Enterprise	X	X	X	X	X

WireCAD Projects

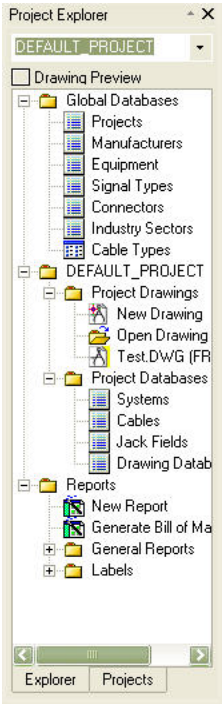


Defined

WireCAD organizes drawings and cable information on a project basis. This allows you to have multiple projects or installations. Each project has a separate set of cable numbers and system ID's. Each project is organized into a series of folders on your local hard drive or on a network share.

Project Explorer Show/Hide

Click this button to show or hide the Project Explorer:



New Project

Creating a new project in WireCAD creates a new folder with the project name in the directory of your choice and places two additional folders in that folder. As follows:

This process simultaneously creates an entry in the projects database.

Link to Existing Project

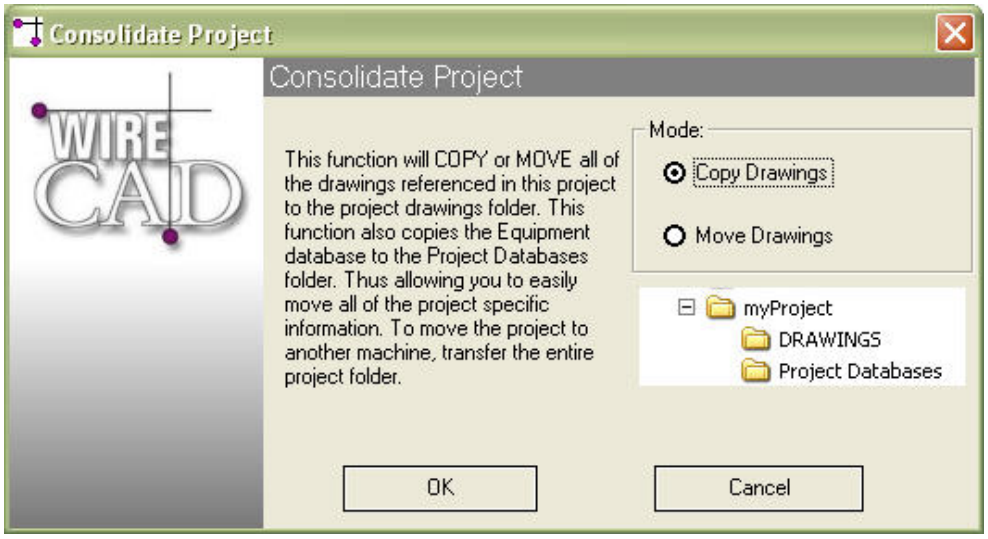
In the event that you need to move a project, use the Link to Existing function to gain access to the data. This function assumes that you have moved the entire project folder to the new location. Browse to, and select the ProjectDB.mdb in the Project Databases folder to link to the project.

Note: as part of the Link function, a copy of the global databases will be place in the WireCAD3\DBUpdates\ folder. When you re-launch WireCAD, the program will synchronize with the new database and make available all of the equipment, signal types, connectors, cable types, etc. that were part of the incoming project.

Consolidate and Move Project

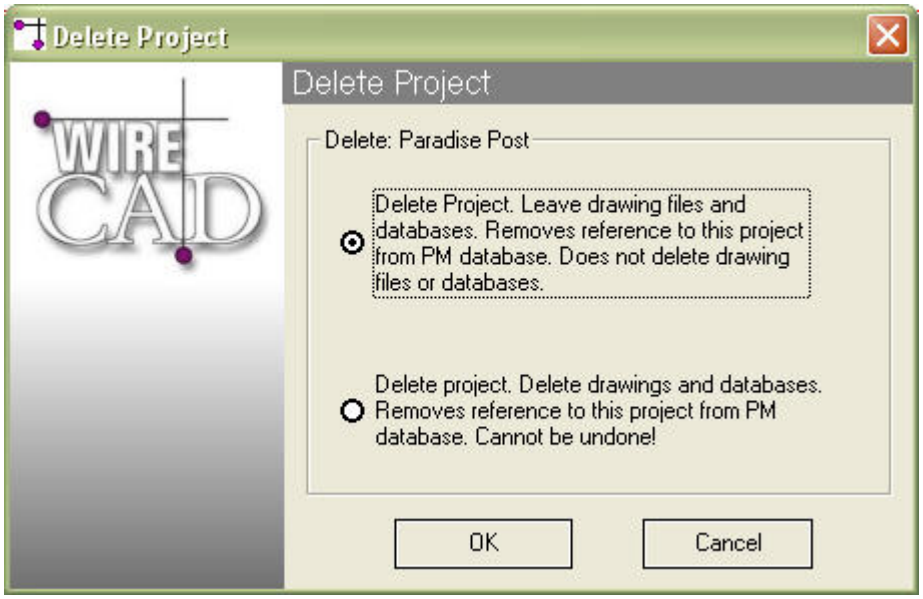
Use this function to consolidate all of the drawings associated with the project into the ProjectFolder\Drawings folder making it easier to move the drawings, and databases associated with a project. You will be given the chance to either Copy or Move the project drawings. Select Copy if you have drawings that are referenced by more than one project.

Note: the Consolidate function also copies the current dbEquip.MDB database into the Project Databases folder. This allows you to easily move the entire project and to have all of the equipment that you created move with it.



Delete Project

Selecting Delete presents two choices:



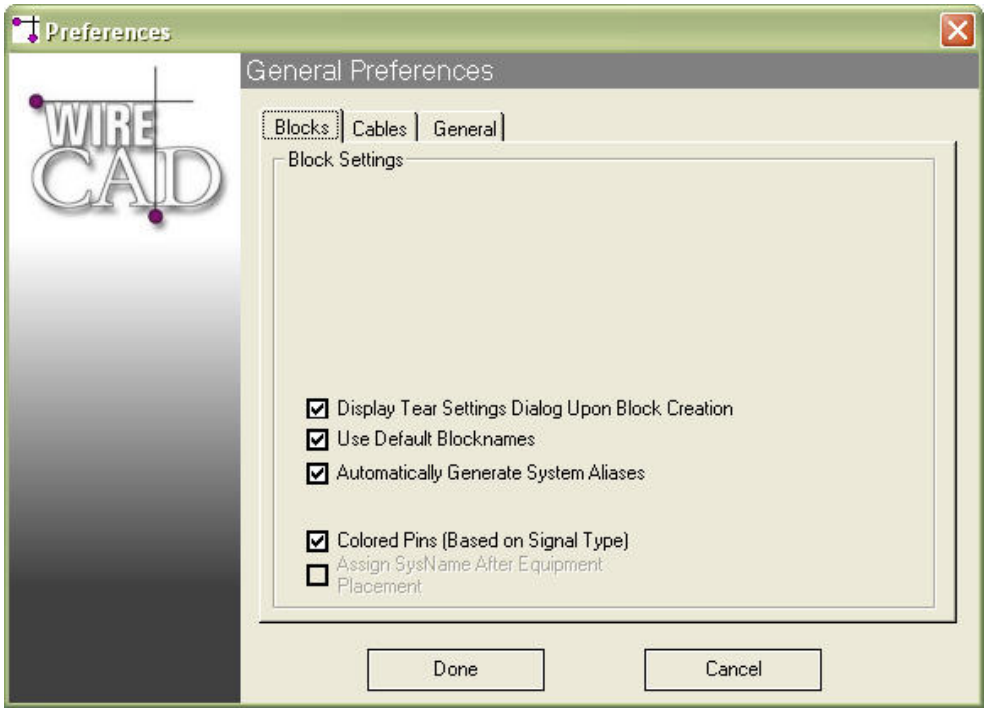
Selecting Leave Drawing Files and Databases will remove the reference in the projects database and leave all drawings and databases in tact. **This is the Default Mode.**

Selecting Delete Drawings and Databases will physically remove (DELETE) all drawings and databases associated with the current project.

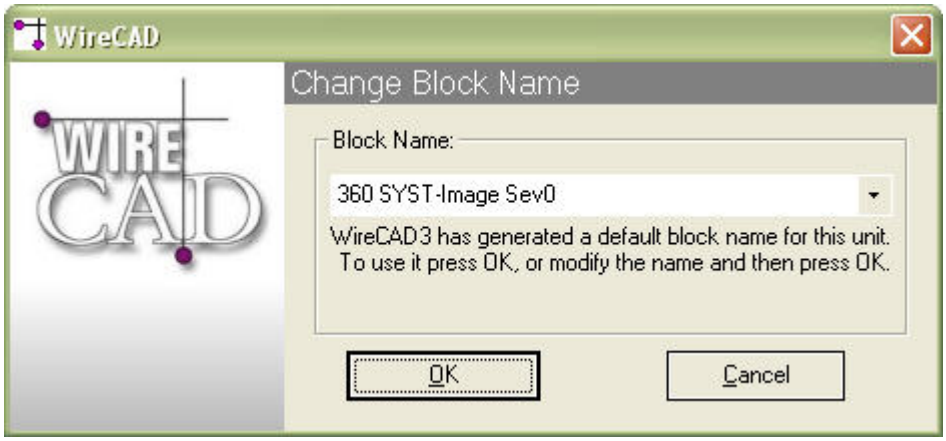
These operations cannot be undone.

Preferences

Blocks Tab



Use Default Block Names: Unchecked allows user editing of the block name.

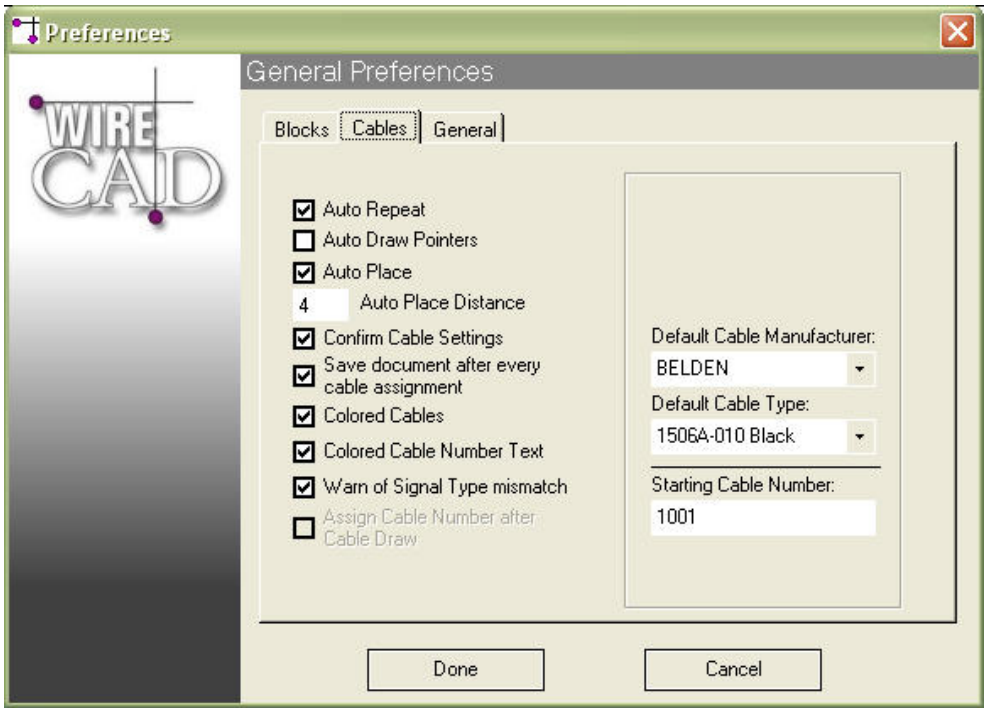


Automatically Generate System Aliases: Checked will use the system name as an alias if no alias is found.

Display Block Preferences …: Checked displays this dialog when creating and equipment block.

Colored Pins: Checked uses information from the Signal Types database to determine the color on block pins.

Cables Tab



The following checkboxes preload the Draw Cables dock-able form:

- **Auto Repeat:** Restarts the Draw Cables Function.
- **Auto Draw Pointers:** Places linked pointers at source and destination pins based on Auto Place Distance. Use only for "On Sheet" pointers.
- **Auto Place:** Places terminal gear (Jacks, Terminals, and Pointers) to the right or left of the source or destination based on the distance defined in the Auto Place Distance Field.
- **Auto Place Distance:** See above.

Confirm Cable Setting: Display the Verify Settings dialog for every cable assignment. Unchecked sends the default information directly to the database.

Save Document After …:Saves the document after every assignment.

Colored Cables: Uses information for the Signal Types database to set the color of the cable.

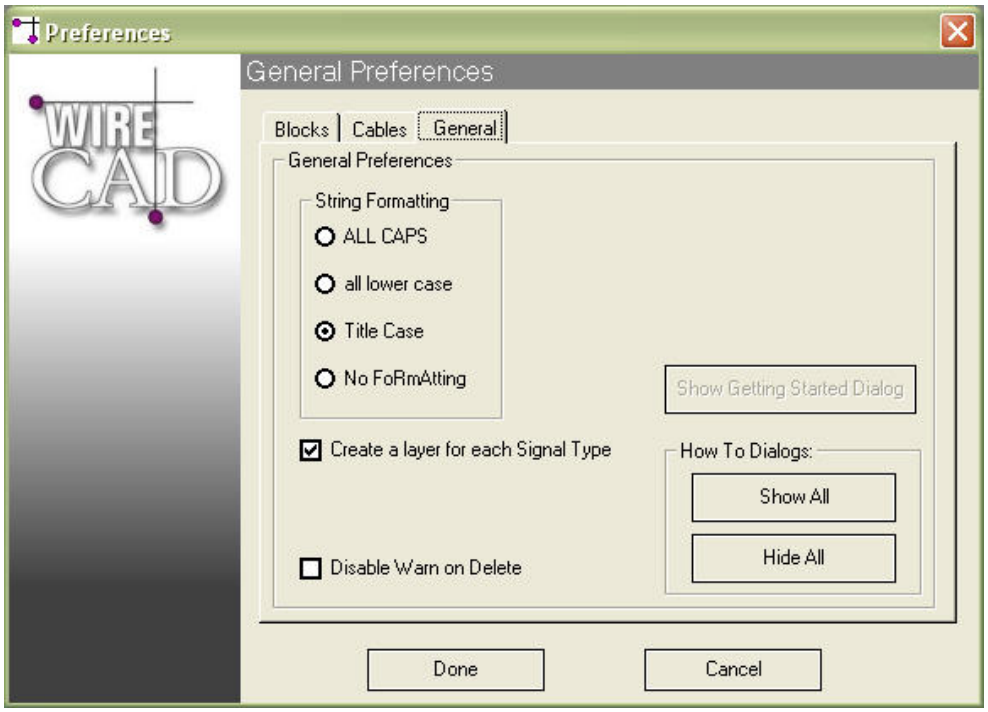
Colored Cable Number …: Uses information for the Signal Types database to set the color of the cable number text.

Warn of Signal Type Mismatch: Warns user every time the source and destination signal types don't match.

Default Cable Type: If the Signal Types database does not contain Cable Type information then this value will be assigned to a cable.

Starting Cable Number: The starting cable number for every Cable Number Prefix.

General Tab



String Formatting: Determines how text is formatted for certain fields in WireCAD.

Create Layer…: Creates a new layer for each Signal Type following this form:

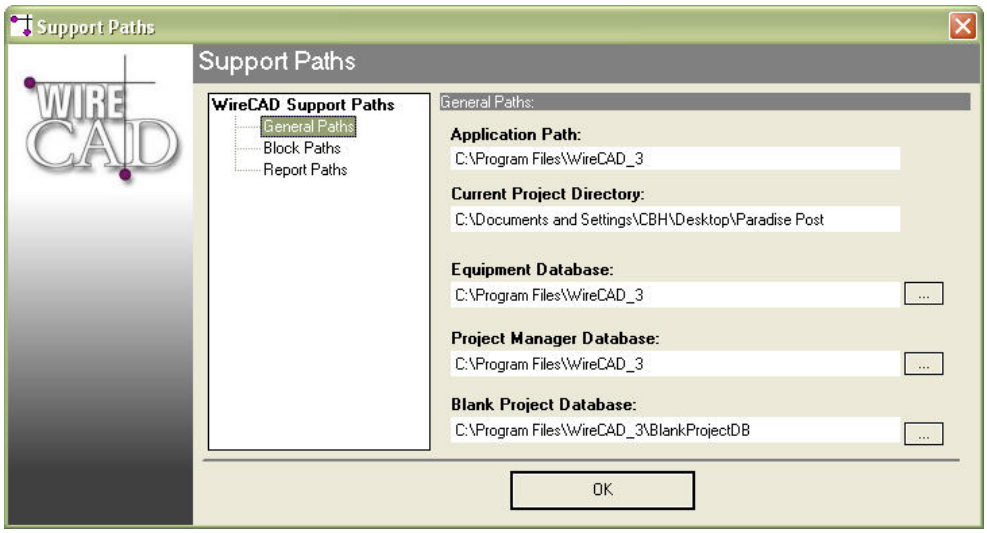
Drawing Entity	Layer Name
Cable Polyline	= SignalType
Cable Number Text	= SignalType_No
Block Pin	= SignalType_Pin

Disable Warn on Delete: Checked disables the warning dialog when deleting an entity that has a reference to the database.

How To Dialogs: Shows or hides the How To dialogs.

Support Paths

This dialog exposes as series of support paths utilized by WireCAD:



General Paths

Application Path (read only)

The current location of the WireCAD v3 application and folder.

Current Project Directory (read only)

The location of the Current Project as selected from the Current Project drop-down

Equipment Database

The location of the dbEquip.mdb file.

Click on the ellipsis button (…) to edit the path to the file.

Project Manager Database

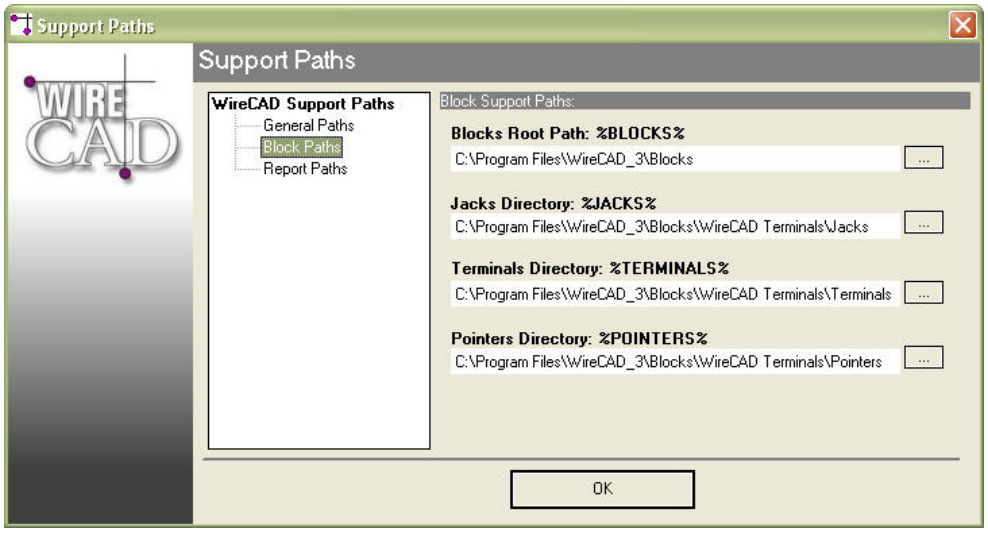
The location of the dbPM.MDB file.

Click on the ellipsis button (…) to edit the path to the file.

Blank Project Database

The location of the blank project database. This database is copied to the new project location, and into the Project Databases folder whereupon it is renamed from BlankDB.MDB to ProjectDB.MDB.

Click on the ellipsis button (…) to edit the path to the file.



Blocks Paths

Blocks Root Path: %BLOCKS%

Defines the %BLOCKS% variable used by several of the WireCAD functions, including the BlockRef path function of the Equipment Library. This function provides for a relative path that is defined by %BLOCKS% variable.

Click on the ellipsis button (…) to edit the path.

Jacks Directory: %JACKS%

Defines the %JACKS% variable used by several of the WireCAD functions, including the Draw Cables function.

Click on the ellipsis button (…) to edit the path.

Terminals Directory: %TERMINALS%

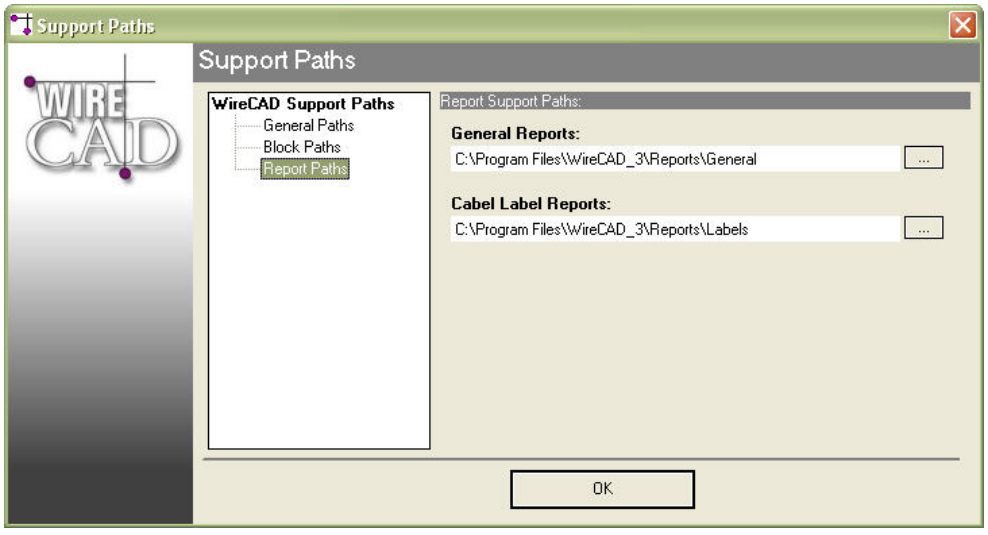
Defines the %TERMINALS% variable used by several of the WireCAD functions, including the Draw Cables function.

Click on the ellipsis button (…) to edit the path.

Pointers Directory: %POINTERS%

Defines the %POINTERS% variable used by several of the WireCAD functions, including the Draw Cables function.

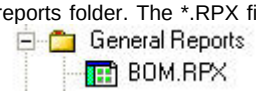
Click on the ellipsis button (…) to edit the path.



Report Paths

General Reports

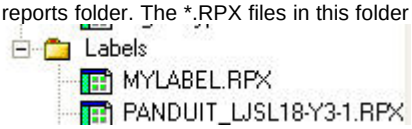
The location of the General reports folder. The *.RPX files in this folder are enumerated in the Reports\General folder in the Project Explorer.



Click on the ellipsis button (…) to edit the path to the folder.

Cable Label Reports

The location of the Label reports folder. The *.RPX files in this folder are enumerated in the Reports\Labels folder in the Project Explorer.



Click on the ellipsis button (…) to edit the path to the folder.

The Drawing Environment (CAD)

WireCAD provides a rich set of drafting tools including many standard functions like:

- Lines.
- Polylines.
- Circles.
- Arcs.
- Points.
- Dimensions.
- Text.
- Images.
- Blocks.
- Viewports.
- Attributes.
- Extended Properties.

DWG File Limitations

While the WireCAD is able to open and create .DWG files, there are some limitations:

Unsupported objects :

- ACIS Objects.
- 3D Solids.
- Region.
- Body.
- MLine (MultiLines) WireCAD converts to single text objects.
- Leader.
- OleFrame.
- Ray.
- Rtext.
- Shape.
- Xline.
- Dimension (we might have some differences from ACAD because we don't support Overwrite properties of ACAD dim style).
- SHX and SHP files. We convert these to the closest TrueType Fonts.
- We convert Hatch to PolyHatch Objects.
- Dictionary - XRecord Objects (You can use XProperties).
- Group of objects.

Drawing Spaces, Layouts, and Viewports

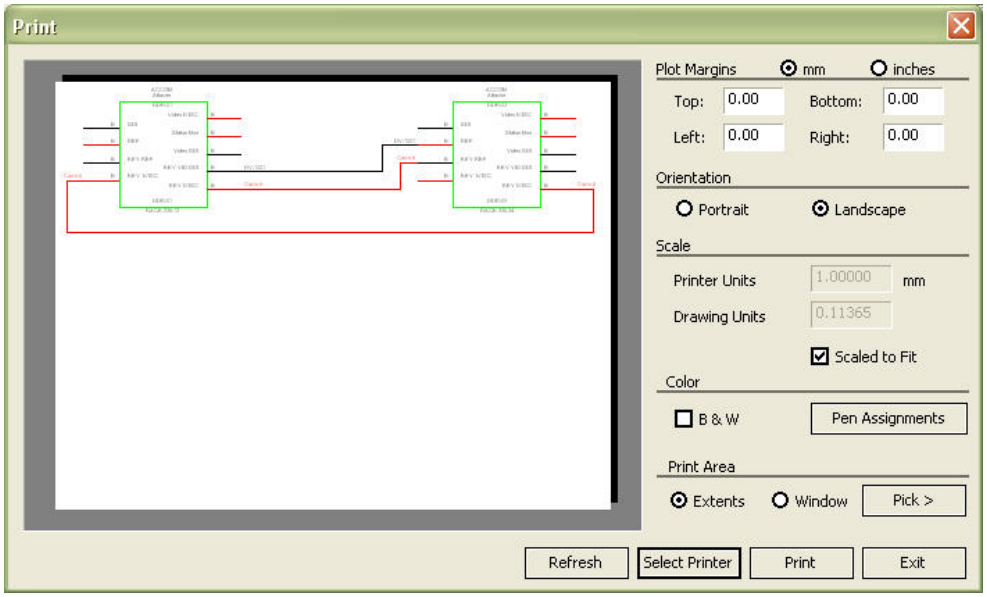
The DWG file format has two drawing spaces – Model space – where you draw your model one to one scale. Paper Space – where you layout the drawing for print. The Paper Space can be explained as follows: imagine a sheet of paper inserted between you and your model. In order for you to see your model you will need to cut a Viewport through you paper to see the model. This is done using the Create Viewport function from the View menu or toolbar  WireCAD supports the ACAD 2000 convention of multiple Paper Spaces or Layouts. Use the Layout Manager  on the View Toolbar.

Note:

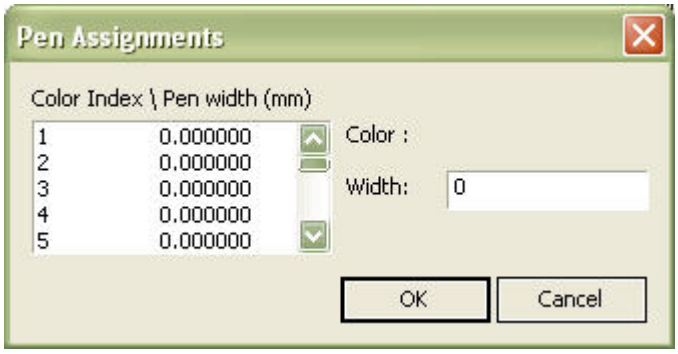
Saving the document down to R-14 or lower will remove any layouts from the drawing file—leaving a single Paper Space.

Printing

Printing: The document size will be based on the current system printer paper size.



Pen Assignments



Assign different pen widths to different colors based on color number. Select a Color number and enter a value in the Width textbox.

Pick >

Pick window to print.

Window/Extents

Print the extents or pick a window to print.

Orientation

Change the orientation between Portrait and Landscape.

Plot Margins

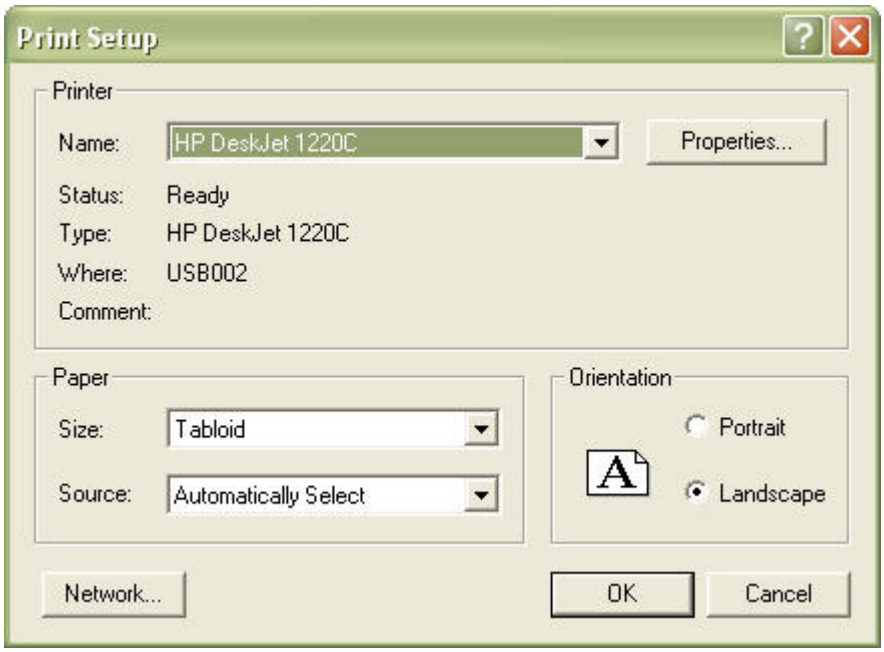
Enter the margins in MM or Inches.

Scale and Color

Scale the window to the paper size, and set print in black and white.

Select Printer

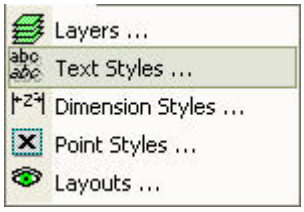
Select a printer and change its properties.



Refresh

Updates the print preview window.

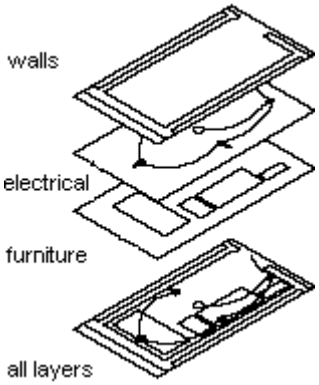
Drawing Format Dialogs



Layers

Layer is the equivalent of the overlay used in paper-based drafting. It is the primary organizational tool in WireCAD, and you can use it to group information by function and to enforce linetype, color, and other standards.

Organizing Layers and the objects on Layers make it easier to manage the information in your drawings. When you put one layer over another then the result is the complete drawing.



There is no limit to the number of layers you can place in a drawing.

When you begin a new drawing, WireCAD creates a special layer named 0. By default, layer 0 is assigned color number 7 (white or black depending upon your background color), the CONTINUOUS linetype and a lineweight of Default (the default setting is .01 inch or .25 mm). Layer 0 cannot be deleted or renamed.

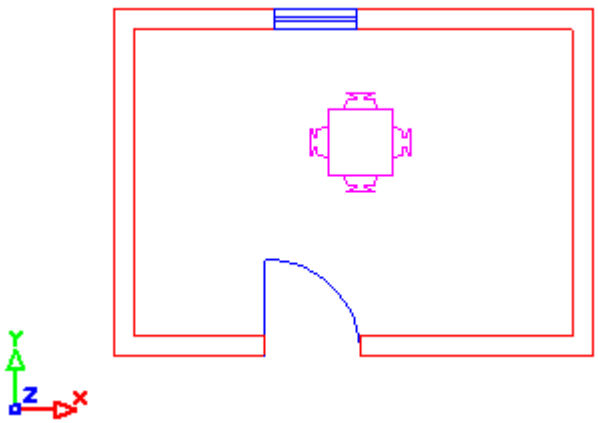
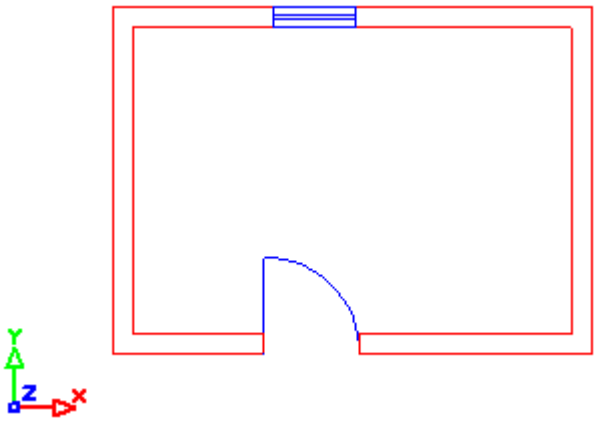
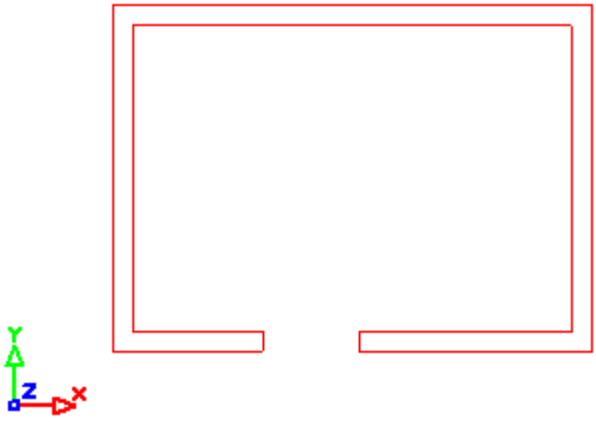
All new objects are added to the active layer if no layer is specified.

Using WireCAD you can Freeze (Hide), Thaw (Show) and Lock layers.

By controlling whether a Layer's state is Thaw or Frozen you can change the appearance of your drawing to display only the information on the Layers that are visible. Freezing unused Layers will help the performance of WireCAD

In the drawing below (Picture 1) there are 3 types of items: **walls** (the lines and Polylines with red color), **doors&windows** (Blue color) and **furniture** (Magenta).

These objects are teamed and drawn in different layers. Walls placed on layer "WALLS", Doors&windows are placed on layer "WIN_DOORS" and furniture are placed on layer "FURNITURE".

Picture 1 All layers are Thawed. All information is shown. In **big** drawings this might confuse the user.	
Picture 2 Layer "FURNITURE" is Frozen. The furniture is not shown. A civil engineer would like to view the drawing this way.	
Picture 3 All layers except the "WALLS" are Frozen. This is the basic information.	

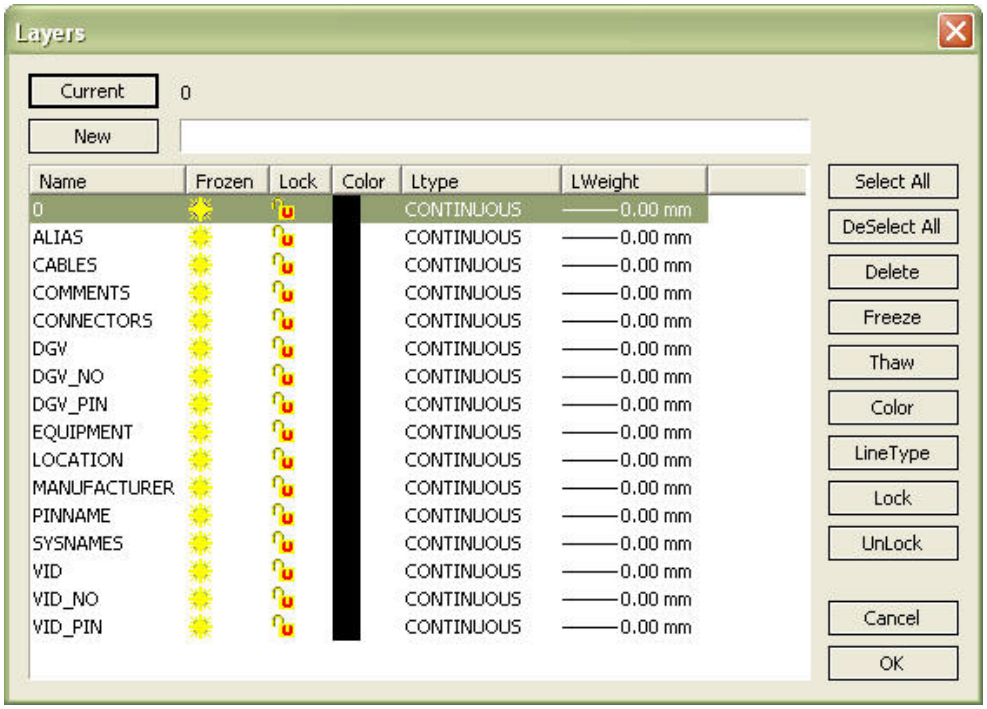
WireCAD Specific Layers

WireCAD requires a specific set of layers in order to operate properly. These layers are added automatically if you open a drawing file in WireCAD. A list of the necessary layers is available in the appendix. WireCAD will also create a series of layers for each signal type that is added to the drawing. As Follows:

Drawing Entity	Layer Name
Cable Polyline	= SignalType
Cable Number Text	= SignalType_No
Block Pin	= SignalType_Pin

The Layers Dialog  allows you to Freeze(hide) and Thaw(show) various layers.

Note: WireCAD will not allow the current layer to be frozen.



Freeze
Hide the selected layer. Frozen layers will not be included in selection sets for deletion.

Thaw
Show the selected layer.

Color
Set the color for entities on the layer with their color property set to Same as Layer.

Line Type
Set the Line Type for entities on the layer with their Line Type property set to Same as Layer.

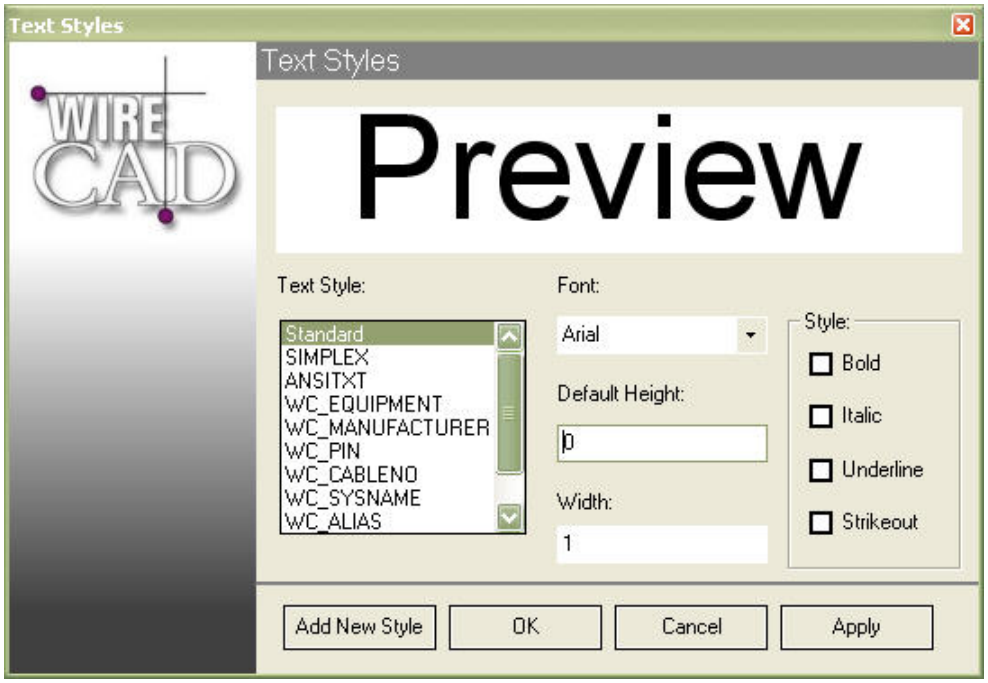
Lock
Lock the layer so that it cannot be edited.

Unlock
Unlock the layer allowing editing.

Text Styles

Text Styles are a way of organizing groups of text together and defining a font and default characteristics for the style.

WireCAD requires several Text Styles be in the drawing document in order to operate properly. These Text Styles are added to a document if it is opened in WireCAD. These Text Styles are enumerated in the appendix.



The text styles associated with WireCAD allow the user to customize the appearance of equipment blocks and cables within a drawing document.

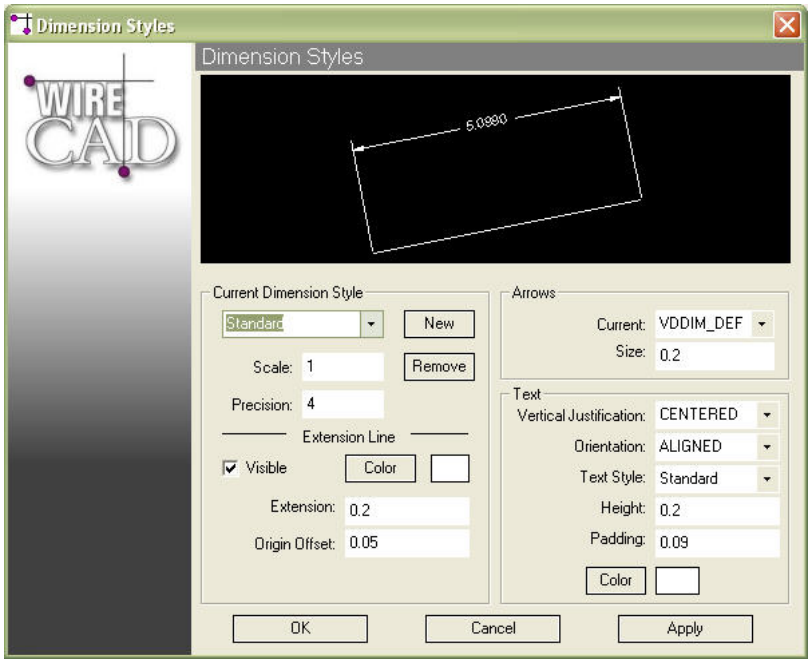
Dimension Styles

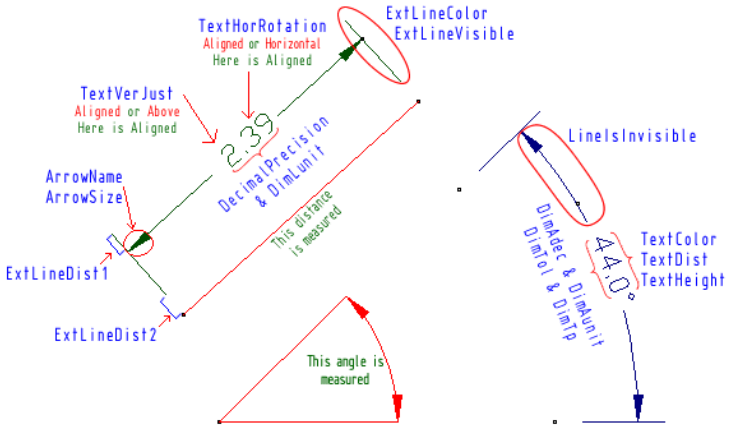
The active dimension style determines the appearance of new dimensions created in the drawing. To change the style of an existing dimension, use the StyleName property found on the dimension.

When you create a dimension, the current dimension style is associated with that dimension. The dimension retains this dimension style unless you apply a new dimension style to it or set up dimension style overrides.

Customize dimension styles.

Add new, and edit existing dimension styles. Arrow blocks can be added to the drawing using the Insert Block functions.

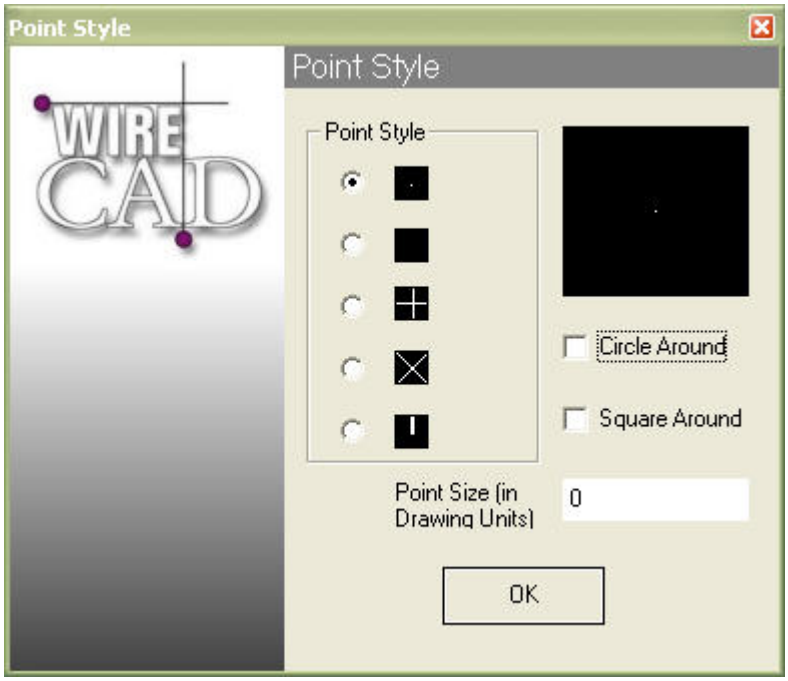




Point Styles

Set the Point Styles for the drawing.

These settings will apply to all points in the drawing.



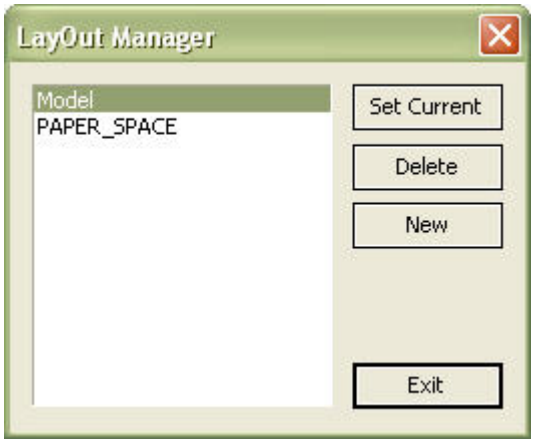
Layouts Dialog

Default Keyboard Shortcut: SO

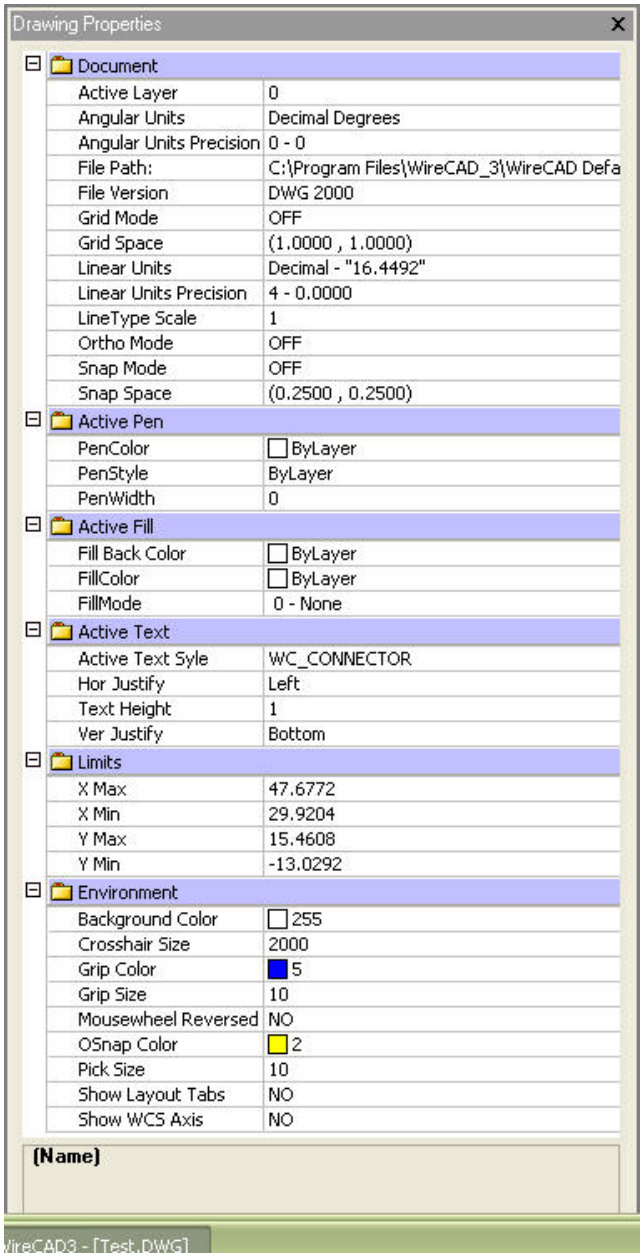
Display the layouts dialog. Here you can add, edit and delete layouts.

The DWG 2000 file format provides from multiple paper spaces call Layouts. This dialog provides access to these functions.

Note: Creating more than one Paper space and then saving the DWG file back to R14 or earlier will cause the additional layouts to be deleted from the saved file.



The Properties Window



Clicking on a property in this window displays a description in the window below the properties.

Enter values directly or use the pull down buttons to open enumerations or dialogs depending on the property. This window applies to the following:

- Drawing properties.
- Drawing entity properties.

- Report designer properties (See the Report Designer Section).

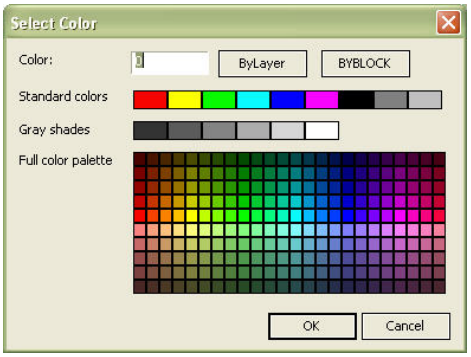
Depending on the property, the property window will accept input in one of the following formats (data types):

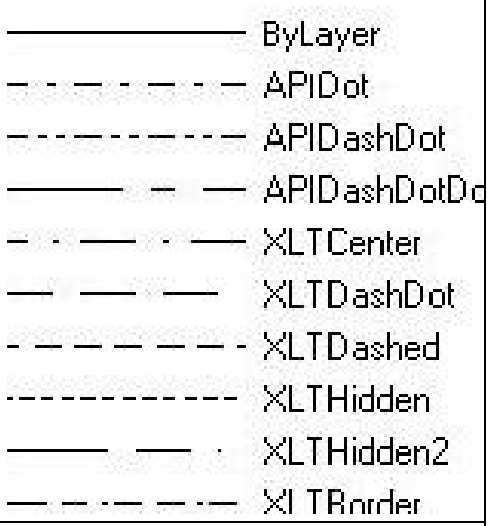
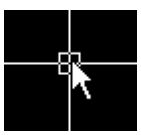
Format	Range	Example
String	Alphanumeric	Hello World 1234
Coordinate	HorizontalDouble, VerticalDouble	1,1 will locate a vertex at 1 drawing unit away from the horizontal and vertical origins.
Double	-1.79769313486231E308 to -4.94065645841247E-324 for negative values and from 4.94065645841247E-324 to 1.79769313486232E308 for positive values	1.234567
Integer	Depends on the property	Colors range from 0-255.
Path	Windows long path	C:\Program files\WireCAD2.

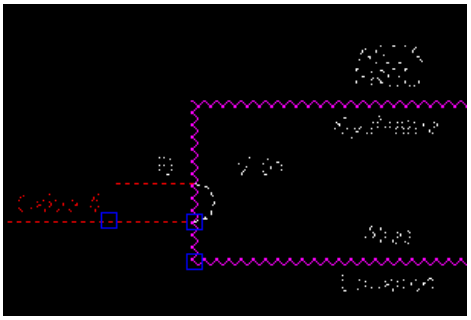
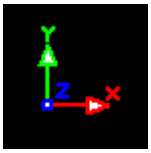
General Drawing Properties

The WireCAD drawing environment supports many different environment variables. These are available by pressing the button on the View toolbar. In order to expose the general properties, be sure that no entities in the drawing are selected before pressing the properties button.

The following is a list of the general drawing properties:

Property	Style/Value Range	Description
Document Settings		
Active Layer	Dropdown / Enum	Sets the current layer. Use the Layers Dialog to define layers .
Angular Units	Dropdown / Enum	Sets the Angular units for the drawing.
Angular Units Precision	Dropdown / Enum	Sets the number of decimal places for Angular units.
FileName	Read Only	Returns the name and path of the drawing file.
FileVersion	Read Only	Returns the current file version.
Grid Mode	Boolean	Toggles the *display* grid.
Grid Space	Direct Entry / Double (x,y)	Sets the *display* grid spacing.
Linear Units	Dropdown / Enum	Sets the Linear units for the drawing.
Linear Units Precision	Dropdown / Enum	Sets the number of decimal places for Linear units.
Line Type Scale	Direct Entry / Double	Sets the scale for all line types in the drawing.
Ortho Mode	Boolean	Toggles the orthogonal (right angle) snap mode.
Snap Mode	Boolean	Toggle the Snap to Grid mode.
Snap Space	Direct Entry / Double. (x,y)	Sets the *snap* spacing. Note: the snap spacing can differ from the *Display* grid spacing
Active Pen		
Active Pen Color	Dialog / 0-255	Sets the current color to draw entities: 
Active Pen Style	Dropdown / Enum	Sets the current line type for entities to be drawn. Penstyle is scaleable. The scale is set from the LineTypeScale property. **TIP**: Instead of changing line type every time you want a different line type, you can create different layers with the desired line

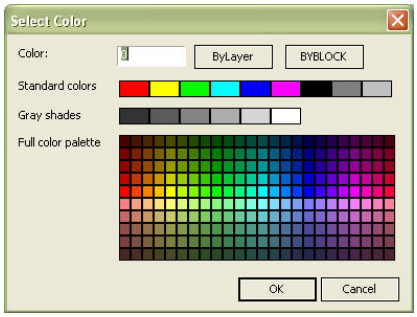
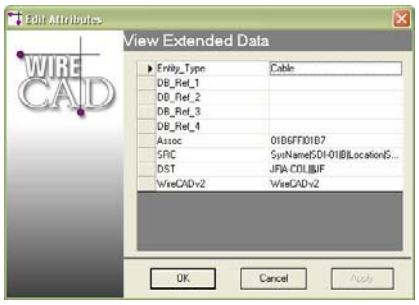
		type. 
Active Pen Width	Direct Entry / Double	Sets the line width for entities to be drawn.
Active Fill		
Active Fill Background Color	Dialog / 0-255	Sets the background color for filled entities.
Active Fill Color	Dialog / 0-255	Sets the color for filled entities.
Active Fill Mode	Dropdown / Enum	Sets the hatch type for filled entities.
Active Text		
Text Style	Dropdown / Enum	Sets the active Text Style.
Text Height	Direct Entry / Double	Sets the default text height.
Hor Justify	Dropdown / Enum	Sets the horizontal text justification.
Ver Justify	Dropdown / Enum	Sets the vertical text justification.
Drawing Limits		
X Min	Direct Entry / Coord	Sets the lower left X corner of the grid.
X Max	Direct Entry / Coord	Sets the upper right X corner of the grid.
Y Min	Direct Entry / Coord	Sets the lower left Y corner of the grid.
Y Max	Direct Entry / Coord	Sets the upper right Y corner of the grid.
Environment Variables		
Background Color	Dialog / 0-255	Sets the background color of the display.
Crosshair Size	Direct Entry / Double	Sets the size of the crosshair. 

		Set to 2000 or so for a full-field crosshair.
GripColor	Dialog / 0-255	Sets the grip color. 
Grip Size	Direct Entry / Double	Sets the grip size.
Mouse Wheel Reversed	Boolean	Reverses the default behaviour of the Mouse-Wheel.
Osnap Color	Dialog / 0-255	Sets the color of Object Snap grips
Pick Size	Direct Entry / Double	Sets the size of the square at the center of the cursor.
Show Layouts Tabs	Boolean	Toggles the display of the layout tabs at the bottom of the drawing space.
ShowWCSAxis	Boolean	Toggles the display of the WCS Axis: 

Entity Specific Properties

General Properties

Every WireCAD drawing entity shares some general properties they are as follows:

Property	Style/Value Range	Description
Area	Read Only	Displays the area of the entity.
Handle	Read Only	Displays the entity's handle. This identifier will last for the life of the entity.
Label	Direct Entry / String	User definable field associated with entity.
Layer	Dropdown / Enum	Sets the layer that the entity is associated with.
Line Type Scale	Direct Entry / Double	Sets the scale of the LineType associated with the entity.
Pen Color	Dialog / 0-255	Color dialog to set the color of the entity: 
Pen Style	Dropdown / Enum	Set the line type used for the entity.
Pen Width	Direct Entry / Double	Sets the pen width for the entity.
Has Extended Data	Read Only / Dialog	Displays a count of the extended data items associated with the entity. Note: This dialog is read only. 

Line Properties:

Property	Style/Value Range	Description
End Point	Direct Entry / Coord	The end point vertex of the line.
Start Point	Direct Entry / Coord	The start point vertex of the line.

Polyline Properties:

Property	Style/Value Range	Description
Hatch Block	Direct Entry / Path	Path to block file to use as hatch for filled

		entities.
Fill Block Scale	Direct Entry / Double	Scale value. Used when the Fill Mode is set to 11-HatchBlock and the Hatch Block property is not null.
Fill Background Color	Dialog / 0-255	Sets the background color for filled entities.
Fill Color	Dialog / 0-255	Sets the color for filled entities.
Fill Mode	Dropdown / Enum	Sets the hatch type for filled entities.

Point Properties:

Property	Style/Value Range	Description
None		

Circle Properties:

Property	Style/Value Range	Description
CenterPoint	Direct Entry / Coord	Sets the center point vertex of the circle.
Radius	Direct Entry / Double	Sets the radius of the circle.
Hatch Block	Direct Entry / Path	Path to block file to use as hatch for filled entities.
Fill Block Scale	Direct Entry / Double	Scale value. Used when the Fill Mode is set to 11-HatchBlock and the Hatch Block property is not null.
Fill Background Color	Dialog / 0-255	Sets the background color for filled entities.
Fill Color	Dialog / 0-255	Sets the color for filled entities.
Fill Mode	Dropdown / Enum	Sets the hatch type for filled entities.

Arc Properties:

Property	Style/Value Range	Description
CenterPoint	Direct Entry / Coord	Sets the center point vertex of the arc
Radius	Direct Entry / Double	Sets the radius of the arc.
Hatch Block	Direct Entry / Path	Path to block file to use as hatch for filled entities.
Fill Block Scale	Direct Entry / Double	Scale value. Used when the Fill Mode is set to 11-HatchBlock and the Hatch Block property is not null.
Fill Background Color	Dialog / 0-255	Sets the background color for filled entities.
Fill Color	Dialog / 0-255	Sets the color for filled entities.
Fill Mode	Dropdown / Enum	Sets the hatch type for filled entities.

Rectangle Properties:

Property	Style/Value Range	Description
Area	Read Only	Displays the area of the rectangle.
InsertionPoint	Direct Entry / Coord	The insertion point vertex of the rectangle.
Hatch Block	Direct Entry / Path	Path to block file to use as hatch for filled entities.

Fill Block Scale	Direct Entry / Double	Scale value. Used when the Fill Mode is set to 11-HatchBlock and the Hatch Block property is not null.
Fill Background Color	Dialog / 0-255	Sets the background color for filled entities.
Fill Color	Dialog / 0-255	Sets the color for filled entities.
Fill Mode	Dropdown / Enum	Sets the hatch type for filled entities.

Text Properties:

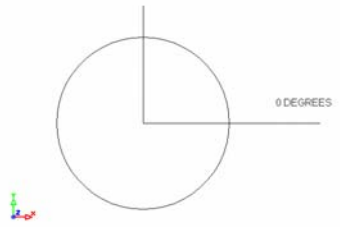
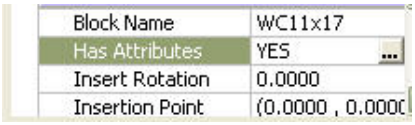
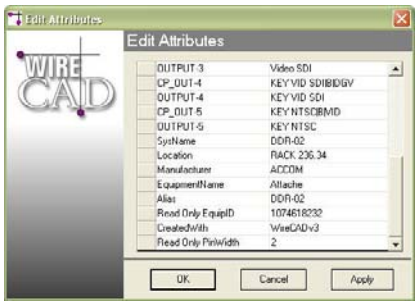
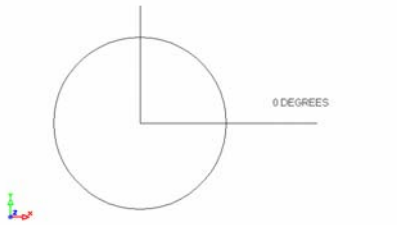
Property	Style/Value Range	Description
Height	Direct Entry / Double	Sets the text height.
HorJustify	Dropdown / Enum	Sets the horizontal justification
InsertionPoint	Direct Entry / Coord	Sets the insertion point for the text.
Rotation	Direct Entry / Degrees	Sets the rotation in degrees for the text. 

Image Properties:

Property	Style/Value Range	Description
Display	Dropdown / Enum	Sets the display properties of the image.
Height	Direct Entry / Double	Sets the height in drawing units of the image.
InsertionPoint	Direct Entry / Coord	Sets the insertion point vertex of the image.
Width	Direct Entry / Double	Sets the width in drawing units of the image.

Insert Properties:

Property	Style/Value Range	Description
Area	Read Only	Displays the area of the entity.
Has Attributes	Boolean/Dialog	Displays the number of attributes associated with the block. Clicking the … button activates the Edit Attributes dialog:  
Block Name	Read Only	Displays the name of the block.
Insertion Point	Direct Entry / Coord	Sets the insertion point vertex of the insert.
Insert Rotation	Direct Entry / Degrees	Sets the rotation in degrees for the insert: 
Scale	Direct Entry / Double	Sets the scale of the insert.

WireCAD Specific Entities Properties:

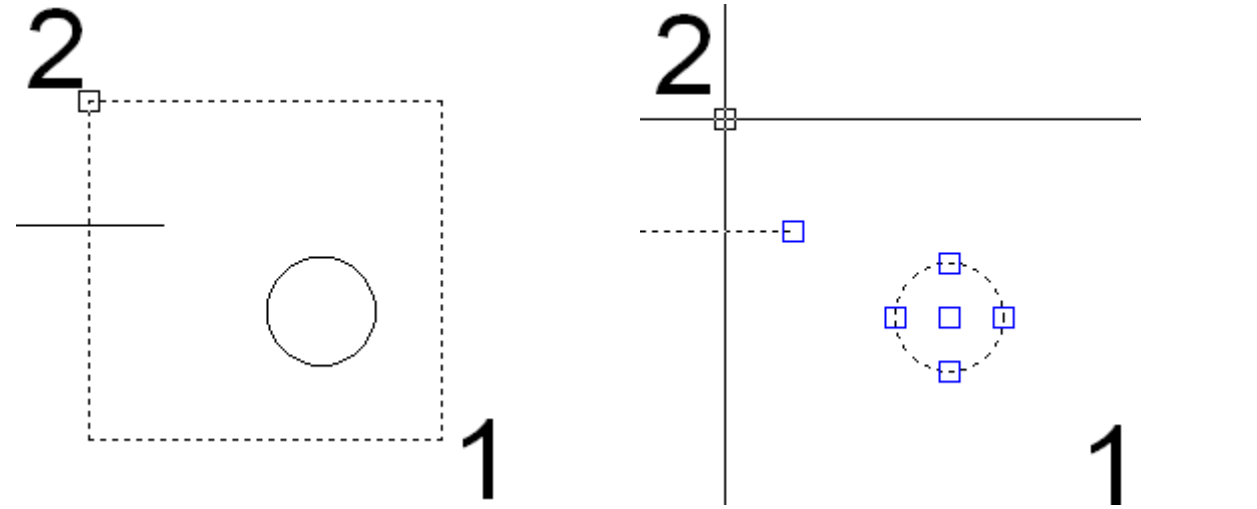
Property	Style/Value Range	Description
XProperties	Read Only	Displays the extended properties for an entity. All WireCAD specific entities (equipment blocks, cables, terminals, etc.) have a specific set of extended properties. These help define the functions, associations, and database links of WireCAD specific entities.

Entity Selection

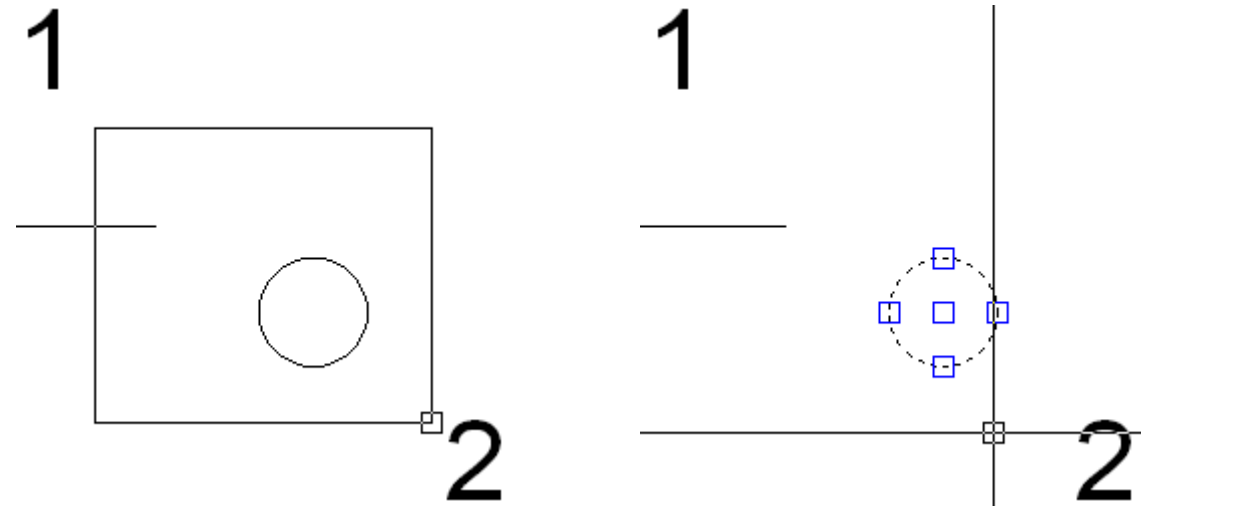
When you run an edit command (which allows multiple object selection) you have to select the objects (or object) that you want to edit.

There are three ways to select objects:

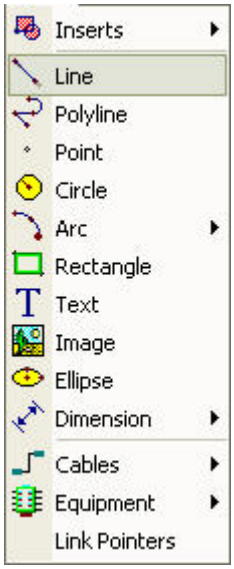
- **Click on the entity:** You can add or remove entities to your selection set by holding the Shift key while clicking on an entity. Clicking on a entity that has already been added to a selection set will remove it (if the Shift Key is pressed).
- **Crossing Window:** Selection windows that are drawn from **Right-to-Left** will appear dashed indicating a Crossing window. Any entity that intersects with, or is wholly contained within, the selection window will be included in the selection set. In the example below, both the line and the circle will be added to the selection set. The circle because it is wholly contained in the Crossing window and the line because it intersects with the Crossing window.



- **Containing Window:** Selection windows that are drawn from **Left-to-Right** will appear solid indicating a Containing window. Any entity that is wholly contained within the window will be included in the selection set. Entities that merely intersect with, but are not wholly contained within the window will by excluded. In the example below the only entity that is wholly contained within the window is the circle. Therefore it is the only entity to be added.



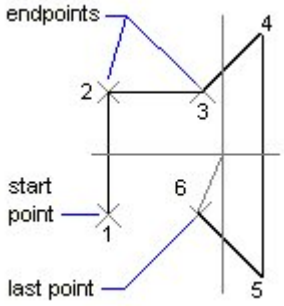
Tools



Draw Line

A single Line segment.

Lines can be one segment or a series of connected segments, but each segment is a separate line object.



Default Keyboard Shortcut: DL

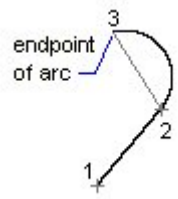
After you start the command, follow the command line prompts for picking each point that defines the object. As you draw, you can:

Left-click to finish one line segment and begin another, connected segment.

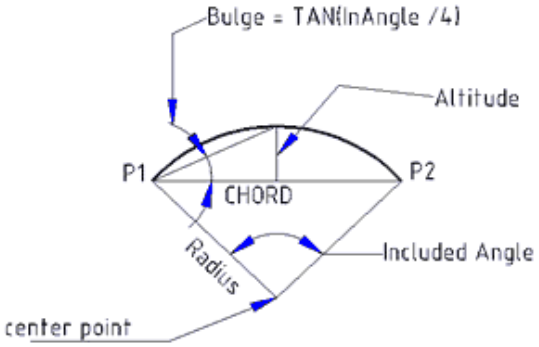
Finish the line: Right-click or press enter.

Draw Polyline

This object is a 2D line composed of line and arc (bulges) segments. Polyline is specified by an array of Vertexes (points).

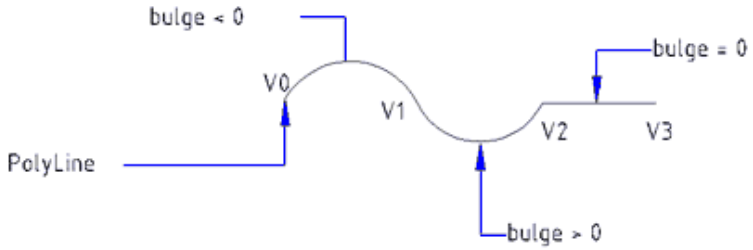


Bulge Geometry:

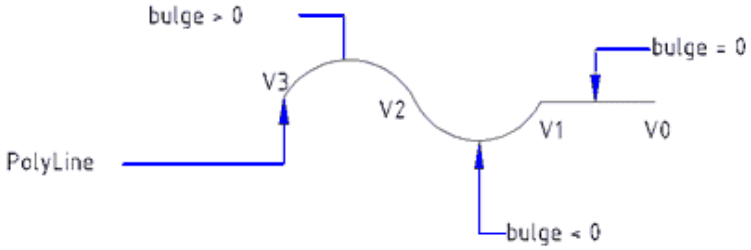


chord= SQRT((P2x - P1x)^2 + (P2y - P2x)^2)
Bulge = 2 × Altitude / chord = (TAN(IncludedAngle / 4))
Altitude = radius - (radius × cos(IncludedAngle / 2.0))
radius = ((chord / 2.0) × (chord / 2.0) + (altitude ×altitude)) / (2.0 × altitude)
IncludedAngle = atan(chord / 2.0 / sqrt(radius^2 - (chord / 2.0)^2)) × 2.0
Arc_Lenght = IncludedAngle × radius

Bulge sign in PolyLines



Opposite direction...



Default Keyboard Shortcut: DP

Draw a two-dimensional polyline, consisting of one or more straight line segments or arc segments connected at vertices.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Left-click to finish one polyline segment and begin another, connected segment.

Finish the line: Right-click or press enter.

Draw Point

Create a point entity based on the global point settings.

Default Keyboard Shortcut: DP

Draw a single point object.

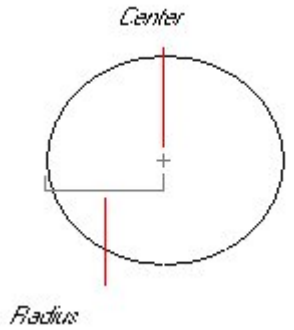
After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Left-click to finish the point.

Exit: Right-click or press enter.

Draw Circle

A full circle is defined by its CenterPoint and its Radius.



Default Keyboard Shortcut: DCC

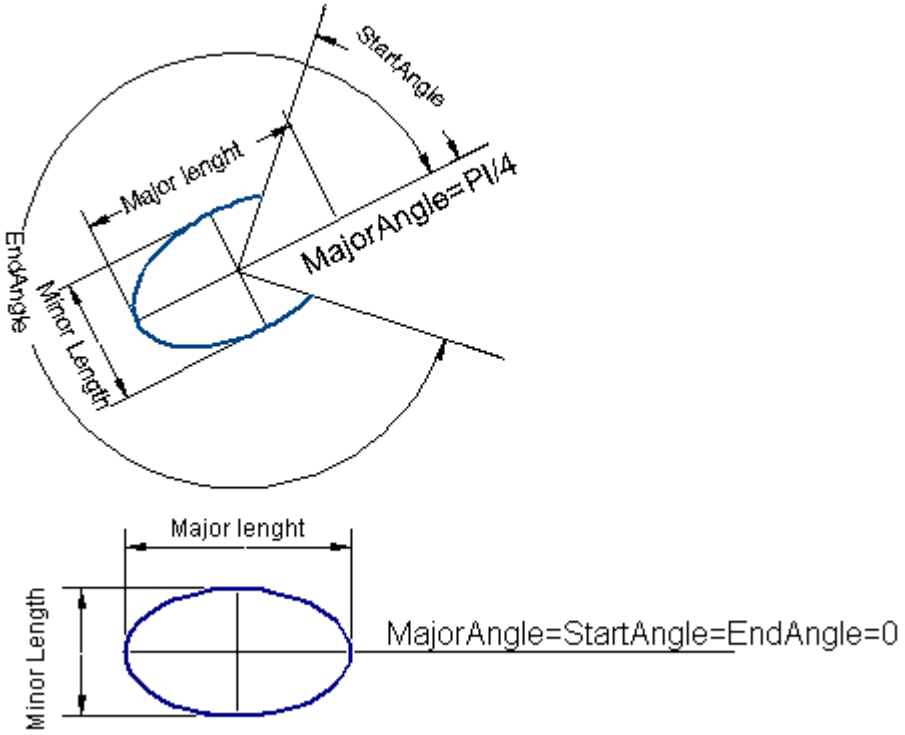
Draw a circle defined by center point, then radius.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Finish the circle: Right-click or press enter.

Draw Ellipse

Draw an Ellipse by defining its Center, Major Angle (Axis 1), and Minor Length (Endpoint).



Default Keyboard Shortcut: NONE

After you start the command, follow the Command Line prompts for picking each point that defines the object.

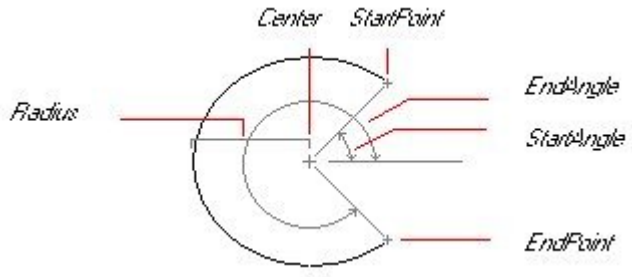
Finish the Ellipse: Right-click or press enter.

Draw Arc

A circular arc is defined by the center point, the radius, the start angle and the end angle.

An arc is always drawn anti-clockwise from the StartAngle to the EndAngle.

The StartPoint and EndPoint properties of an arc are calculated through the StartAngle, EndAngle and Radius properties.



Default Keyboard Shortcut: DCC

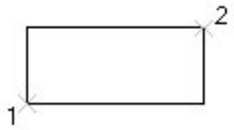
Draw an arc defined by center point, then radius, then angle.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Finish the arc: Right-click or press enter.

Draw Rectangle

The rectangle is defined by InsertionPoint, Rotation, Width and Height.



Default Keyboard Shortcut: DR

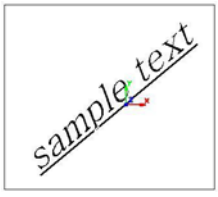
Draw a rectangle defined by upper left, then lower right coordinates.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Finish the rectangle: Right-click or press enter.

Draw Text

Text is defined by start point where the text will begin, the rotation angle of the text, and the text string. Notice that the text will be drawn with the current Text Style of the document.



Default Keyboard Shortcut: DT

Draw text defined by basepoint coordinates, then rotation, then text string.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

Finish the text: Right-click or press enter.

Draw Image 

A basic object for inserting images in the drawing.

Inserted Images can be BMP, GIF, JPG, PNG and TIFF

When the inserted image is an 1-bit Image (B&W) then the background color of the image (usually the white) is shown transparent and the other color is shown black or white, depending on the background color. This is useful for drawings to be scanned and then inserted as images in other drawings.

Image has a Scale property that is used to resize keeping the aspect ratio of the image.

The image is defined by InsertionPoint, Rotation, Width and Height.

Default Keyboard Shortcut: DI

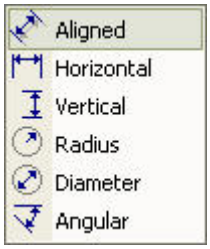
Draw an image defined by basepoint coordinates, then selecting an image file to link to.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

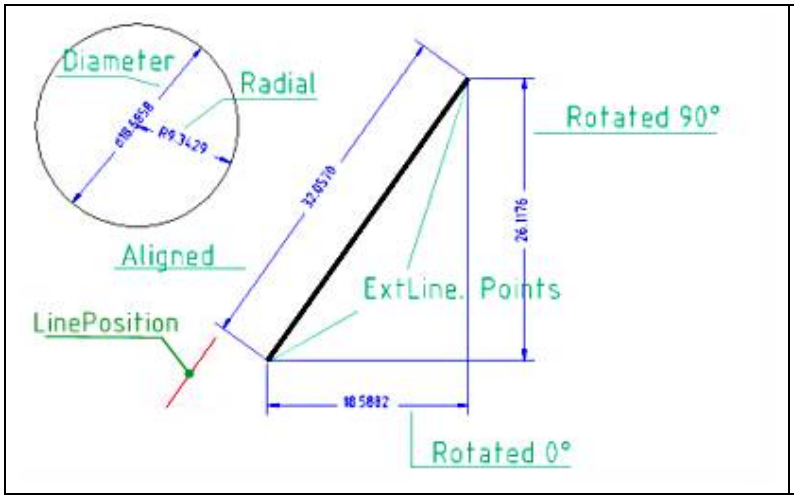
Finish the image: Right-click or press enter.

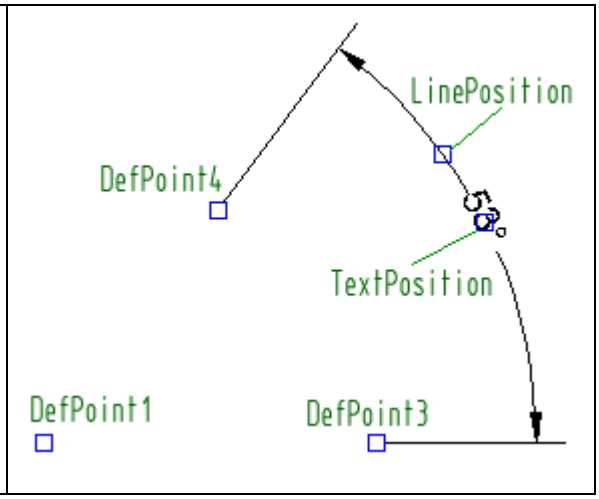
Dimensions



Dimensioning is the process of adding measurement annotation to a drawing. User has many ways to dimension objects and many ways to format dimensions. You can create dimensions for a wide variety of object shapes in many different orientations. You can create dimension styles to format dimensions quickly and ensure that dimensions in your drawing conform to industry or project standards.

Dimensions show the measurements of objects, the distances or angles between objects, or the distance of a feature from an origin you specify. User has three basic types of dimensioning: linear, radial, and angular. Dimensions can be horizontal, vertical, aligned, rotated, angular. A linear dimension measuring the distance between two points which is displayed parallel to the points being measured. In aligned dimensions, the dimension line is parallel to the extension line origins. The extension line origins are specified using the DefPoint1 and DefPoint2 properties.





Draw Aligned Dimension

Default Keyboard Shortcut: DDA

Draw an aligned dimension.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

Finish the aligned dimension: Right-click or press enter.

Draw Horizontal Dimension 

Default Keyboard Shortcut: DDA

Draw a horizontal dimension.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

Finish the horizontal dimension: Right-click or press enter.

Draw Vertical Dimension 

Default Keyboard Shortcut: DDA

Draw a vertical dimension.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

Finish the vertical dimension: Right-click or press enter.

Draw Radius Dimension 

Default Keyboard Shortcut: DDA

Draw a radius dimension.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

Finish the radius dimension: Right-click or press enter.

Draw Diameter Dimension 

Default Keyboard Shortcut: DDA

Draw a diameter dimension.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

Finish the diameter dimension: Right-click or press enter.

Draw Angular Dimension 

Default Keyboard Shortcut: DDA

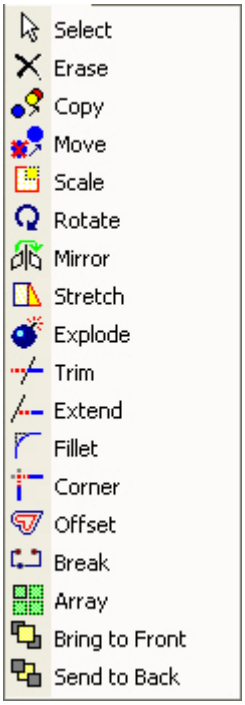
Draw a diameter dimension.

After you start the command, follow the Command Line prompts for picking each point that defines the object. As you draw, you can:

Enter data directly in the command line.

Finish the diameter dimension: Right-click or press enter.

Modifying Drawing Entities



Delete

With the delete command you can delete one or more objects of the drawing.

If you want to erase multiple objects you have to execute the select method.

After the delete command, the objects no more "exist" in the document and in the collection they belonged to, however the objects still exist as "deleted" objects in memory. So with undo command you can get them back to the drawing.

Default Keyboard Shortcut: The DEL Key

Delete Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Copy

With copy command you can copy one or more objects of the drawing.

When copy command starts, you will be prompted to select the objects to copy. Next you will be prompted to select two points. These two points define the "copy vector" and can either belong to the selected objects or not. The first point specifies the begining of the "copy vector" and the second point the end of it.

Default Keyboard Shortcut: MC

Copy Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Move

With move command you can move one or more drawing objects.

Select the objects to move and define two points that define the distance and the direction of the movement.

The first point defines the beginning of the "movement vector" and the second the end of the vector.

Default Keyboard Shortcut: MM

Move Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Scale

With scale command you can increase or decrease the size of one or more objects.

First you have to select one or more objects. Then you have to pick one point, which is going to be the base point. Next step is to specify the scale factor. You can either type or pick this value from the drawing.

Default Keyboard Shortcut: MS

Scale Selected Entities.

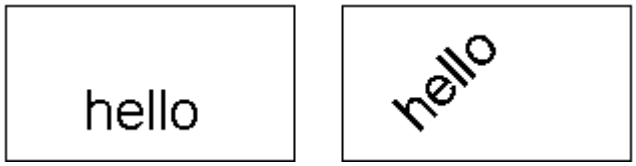
After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Rotate

With rotate command you can rotate one or more objects around a base point.

First you have to select the object or objects and then specify the base point. Then select an angle in degrees to rotate the object.



Default Keyboard Shortcut: MR

Rotate Selected Entities.

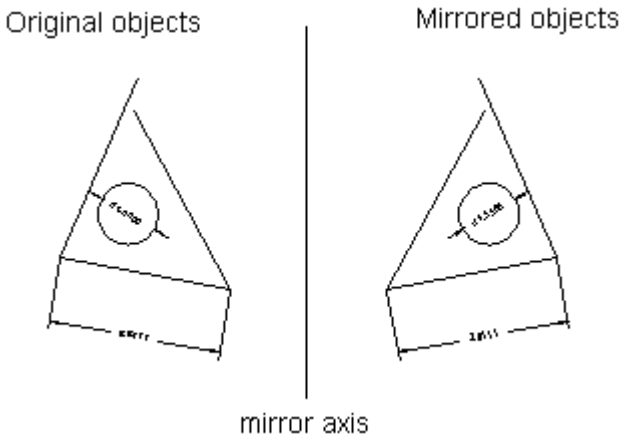
After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Mirror

With mirror command you can create the symmetrical of one or more objects reflected by an axis defined by the user.

First, select the objects you want to mirror. Next, set the axis by setting the first point of mirror line and then the second. At this point you have to choose if the source objects will be deleted or not.



Default Keyboard Shortcut: MS

Mirror Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Stretch

Default Keyboard Shortcut: MS

Stretch Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Explode

With explode command you can break compound objects such as: inserts, dimensions, and polylines into their sub entities.

Default Keyboard Shortcut: MDA

Explode Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Trim

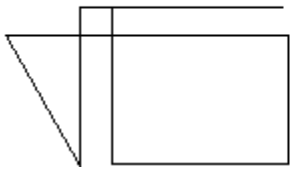
With trim command you can trim objects at a cutting edge defined by other objects.

First select the objects that define the cutting edges at which you want to trim an object and then the object. Objects that can be trimmed include arcs, circles, elliptical arcs, lines.

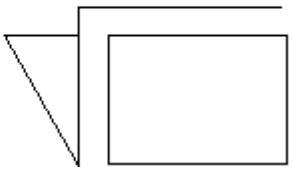
Notice that the trim command does not function if the objects do not intersect.

At the example below there are some lines that were trimmed.

before trim



after trim



Default Keyboard Shortcut: MT

Trim Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

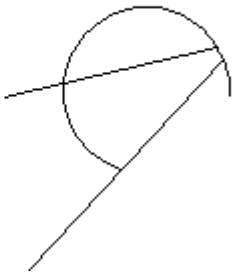
Extend

With extend command you can extend lines, arcs, polylines until they intersect with some other object which is used as the limit of the extension.

First, select the objects that define the limits of the extension. Then you have to choose a point on an object that you want to extend. If the object you want to extend does not intersect with above objects then nothing will happen.

Before extend

After extending the objects



Default Keyboard Shortcut: ME

Extend Selected Entities.

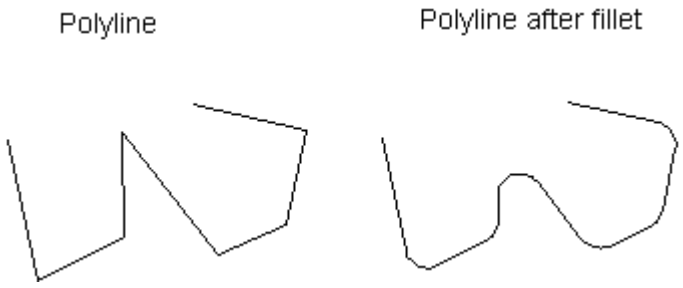
After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Fillet

Insert fillet arcs at vertexes of specified index of a polyline where two line segments meet if the specified radius is enough small to fit into lines.

If the radius is bigger then it is ignored for the specific vertex.



Default Keyboard Shortcut: ME

Fillet Selected Polyline Entities.

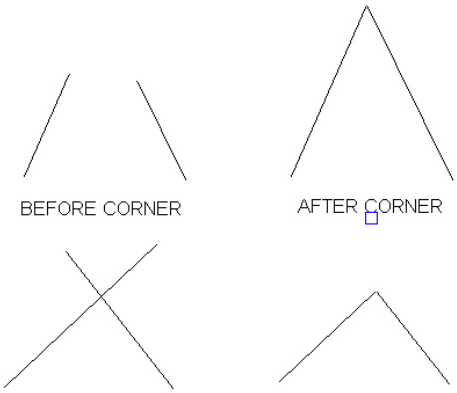
After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Corner

Use the corner function to either extend two lines to their intersection or to erase lines that extend past their intersection.

Note: only applies to Line entities.



Default Keyboard Shortcut: MN

Corner Selected Line Entities.

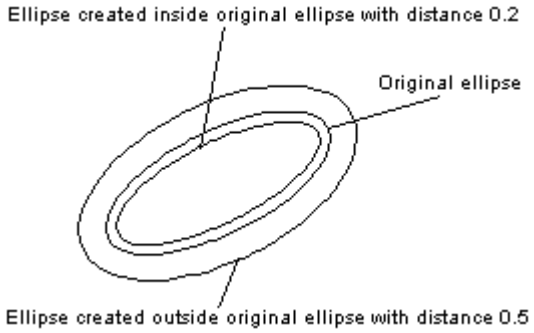
After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Offset

With Offset command you can create a new object in parallel direction and at a specified distance from the original object that is used as pattern for the new object.

When you execute offset command, you are prompted to select an object. Then you have to specify the offset distance. This is the distance that the new object will be draw from the original object. Then you have to set the side that the object will be draw because there are two sides.



Default Keyboard Shortcut: ME

Fillet Selected Polyline Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Break

With break command you can divide one object into two objects, or you can remove one part of it (this part will be defined by two points that you have to pick).

First you have to select the object. Next, define the first break point and the second break point. When you pick these points the part defined by the two points will be deleted from the object.

The two points can be the same point. In this case you only explode the object into two other objects (for example you can cut a line into two lines).

If break command involves a circle, you must define two different points in order to see a result because a circle cannot break in one point. After break command the circle becomes arcs.

Default Keyboard Shortcut: MB

Break Selected Entities.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Array

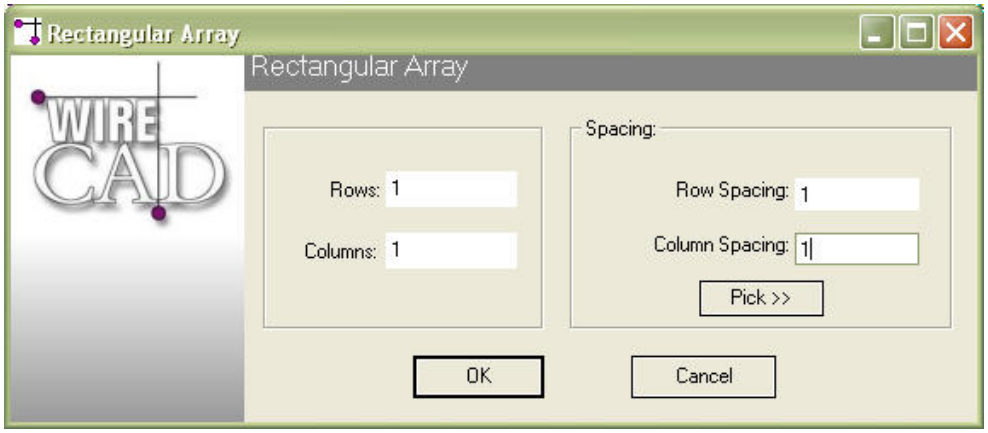
Creates multiple copies of objects in a rectangular pattern.

With the rectangular array you can create an array defined by a number of rows and columns of copies of the selected object. First, select the objects. Then define number of rows and number of columns of the rectangle, the distance between rows and the distance between columns.

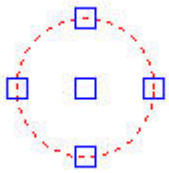
Default Keyboard Shortcut: MA

Array Selected Entities.

Use this form to determine the number of rows and columns in the array, and set the distance between the rows and columns. Clicking on the Pick >> button allows you to determine the distance by selecting points in the drawing. See example below.



To create an array of entities, select the entities to include. Here we will create a 2 x 2 array of the selected circle:

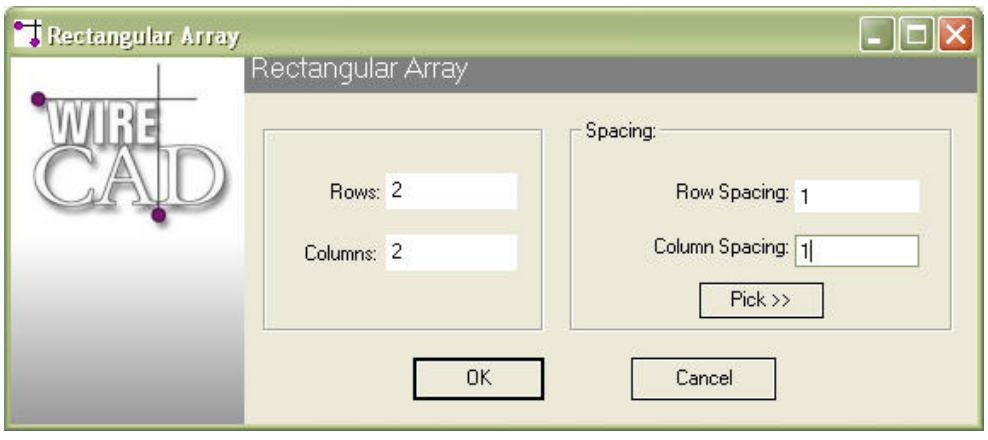


Next click on the button.

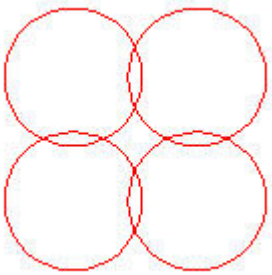
Here we enter the number of rows and columns, and determine the distance between each row and column.

Note:

When creating an array, always include the starting row and column. So the number entered in the rows and columns field is the total number of rows and columns.



Clicking OK yields the following:



Bring to Front

Default Keyboard Shortcut: MB

Bring Selected Entities to the front.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Send to Back

Default Keyboard Shortcut: MB

Send Selected Entities to the back.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Viewing, Panning, Zooming

Zoom In

Zoom In command allows the user to increase the apparent size of objects , so the user can control the part of the drawing that is included in the screen.

Note: Zoom In command is a transparent command.

Transparent commands are commands that can be invoked when another command is active.

Default Keyboard Shortcut: ZI

Zoom into the drawing by 5 percent.

Alternately use the mouse-wheel.

Zoom Out

Zoom Out command allows the user to decrease the apparent size of objects , so the user can control the part of the drawing that is included in the screen.

Note: Zoom Out command is a transparent command.

Transparent commands are commands that can be invoked when another command is active.

Default Keyboard Shortcut: ZO

Zoom into the drawing by 5 percent.

Alternately use the mouse-wheel.

Pan

You can shift the location of your view by using pan or by using the window scroll bars. Like panning with a camera, pan does not change the location or magnification of objects on your drawing; it changes only the view.

Note: Pan command is a transparent command.

Transparent commands are commands that can be invoked when another command is active.

Default Keyboard Shortcut: VP

Pan the drawing.

Alternately, hold the mouse-wheel down and move the mouse to pan the drawing view.

Zoom Window 

Zoom Window command allows the user to increase the apparent size of objects using a selection window, so the user can control the part of the drawing that is included in the screen.

Note: Zoom Window command is a transparent command.

Transparent commands are commands that can be invoked when another command is active.

Default Keyboard Shortcut: ZW

Select a window to zoom to.

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

Zoom Extents 

Zoom Extents command Zooms to display the drawing extents.

Note: Zoom Extents command is a transparent command.

Transparent commands are commands that can be invoked when another command is active.

Default Keyboard Shortcut: ZE

Zoom to the extents of all drawing entities. This will zoom to show all entities in the current drawing space.

Zoom All 

Zoom All command zooms to the drawing limits or current extents, whichever is greater.

Note: Zoom All command is a transparent command.

Transparent commands are commands that can be invoked when another command is active.

Default Keyboard Shortcut: ZA

Zoom Last View 

Zoom Last View command Zooms to display the previous view.

Note: Zoom Last View command is a transparent command.

Transparent commands are commands that can be invoked when another command is active.

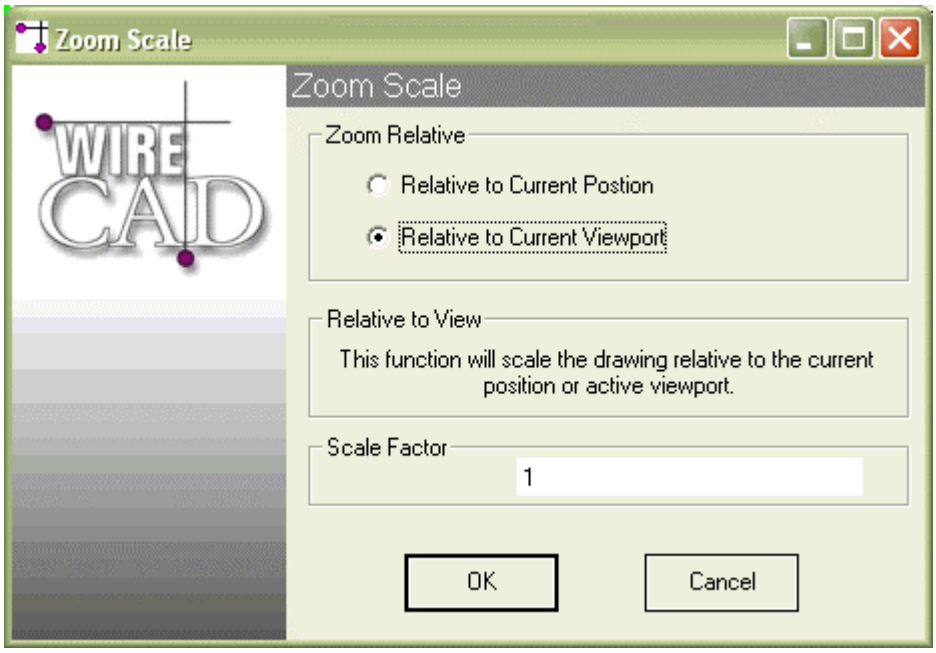
Default Keyboard Shortcut: ZL

Switch to the last view in the current drawing space.

Zoom Scale

Opens the Zoom Scale dialog. Values entered here will cause the entities in the selected Viewport to display relative the the height of the selected Viewport. This function is useful for producing a final scaled output of an architecthural or mechanical drawing. For example, entering 2 doubles the apparent display size of any objects from what it would be if you were zoomed to the limits of the drawing. Entering 0.5 causes each object to be displayed at half its current size on the screen, or relative to the height of the current Viewport, depending on the option selected.

Default Keyboard Shortcut: NONE



Regenerate Drawing

Default Keyboard Shortcut: ZE

Regenerate the entire drawing.

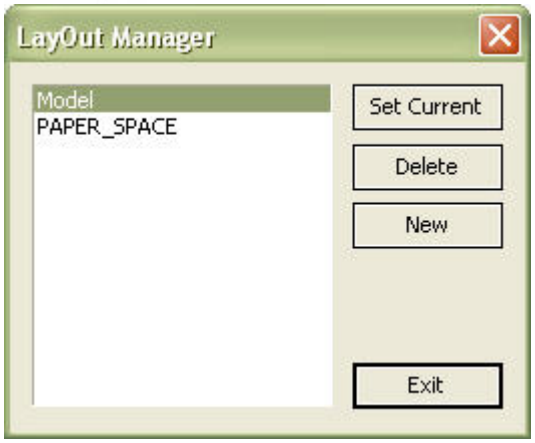
Layouts Dialog

Default Keyboard Shortcut: SO

Display the layouts dialog. Here you can add, edit and delete layouts.

The DWG 2000 file format provides from multiple paper spaces call Layouts. This dialog provides access to these functions.

Note: Creating more than one Paper space and then saving the DWG file back to R14 or earlier will cause the additional layouts to be deleted from the saved file.



Create Viewport

Create new Viewport entity in the current Paper Space or Layout. Viewports allow you to view through the Paper Space to the Model Space.

Default Keyboard Shortcut: None

Note: Only active when the current layout is not Model Space.

Activate Viewport

Activate the selected Viewport for Panning and Zooming. Once active, you can use any of the panning, zooming, or zooming scale functions.

Default Keyboard Shortcut: None

Double-click a Viewport to activate / deactivate.

Note: The cursor will be constrained to the Viewport when the Viewport is Active.

Blocks and Inserts

Defined

Blocks represent logical groupings of drawing entities stored together within a drawing. Blocks may contain any drawing entity or entities. Blocks are created by selecting a group of entities and defining a base-point for re-insertion.

Blocks may be written out of the drawing (creating a new drawing file) using the Write Block function

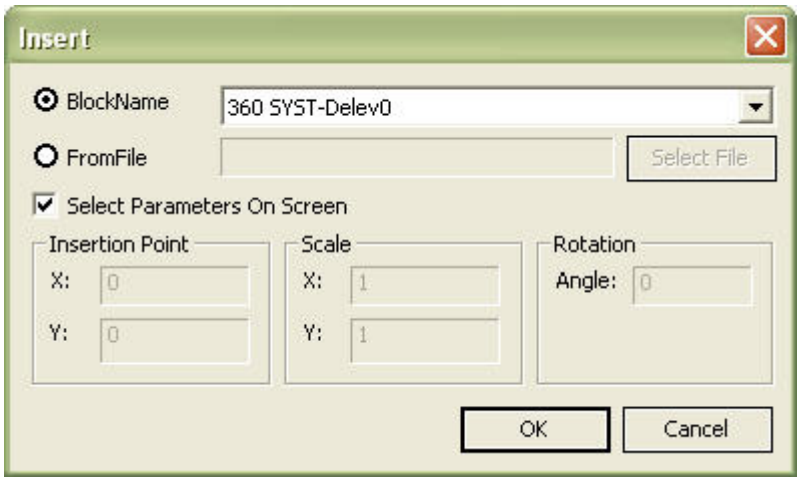
Blocks can also be inserted multiple times in the drawing This function displays the Inserts dialog (see below).

Other drawings may be inserted into the current drawing.

Other drawings may be externally referenced (Xref) or linked to the current drawing. This allows the drawing to be maintained elsewhere and updated remotely. The changes can then be updated in the current drawing.

An insert is a single instance of a block. A single block definition may have multiple inserts within the drawing. Take, for example, the case of a door block. The block is comprised of a straight line and an arc. These entities are grouped together and a base point is defined at the hinge point using the Tools>Insert>Create Block. Once the door block is defined, inserts of the block will be placed throughout the drawing as needed. This saves the user the tedious task of recreating the door entities every time a door is needed.

Use the Blocks Dialog to place inserts in the current drawing space.



Creating New Blocks

Tools>Inserts>Create Block

After you start the command, follow the Command Line prompts to complete the command.

Escape or right click to exit.

You will be prompted to select the entities to include in the block. Select the entities. Next, right-click to continue. You will be prompted to select the base point or the insertion point of the block. This is the point from which an insert of this block with be placed

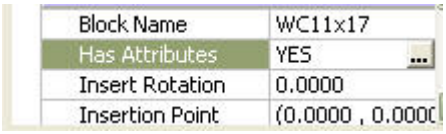
Text Versus Attributes

A block may contain Text entities, as well as a special type of text entity called an Attribute. Attributes are unique in that they may contain insert specific information. Attributes maintain data in TAG/VALUE pairs. The tag is used to identify the data contained in the value.

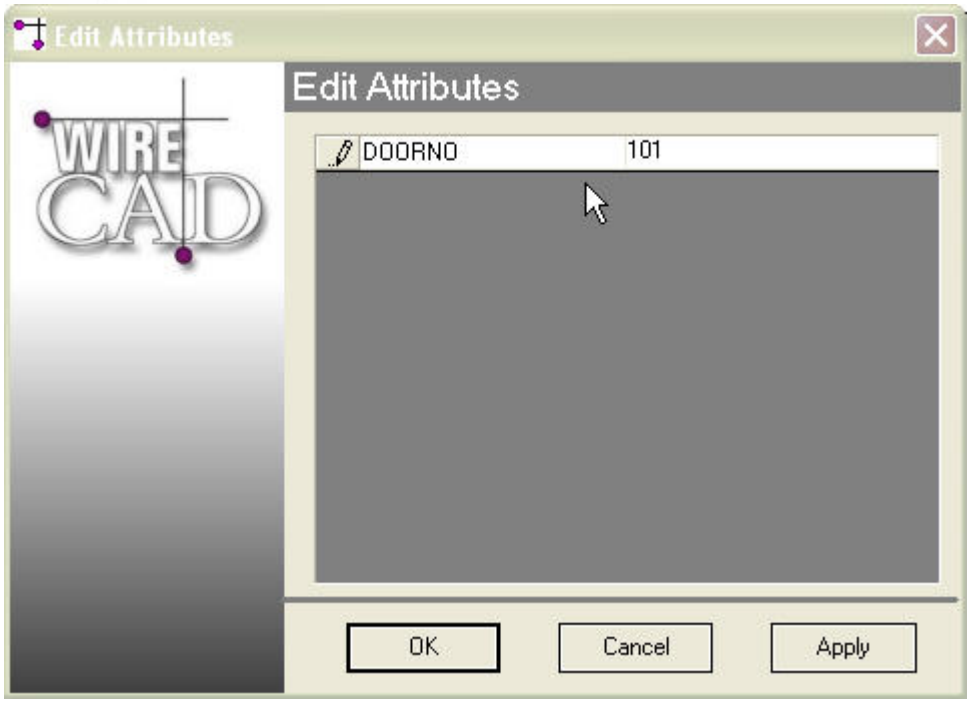
Attributes may be defined as visible or invisible.

To use our door block example from above. Say we define an Attribute to group along with the line and arc. We designate the TAG as DoorNo and we enter DOORNO in the value field, and we set the visible value to true in order to display the attribute value upon block insertion. We then create our new door block and insert an instance of it in our drawing.

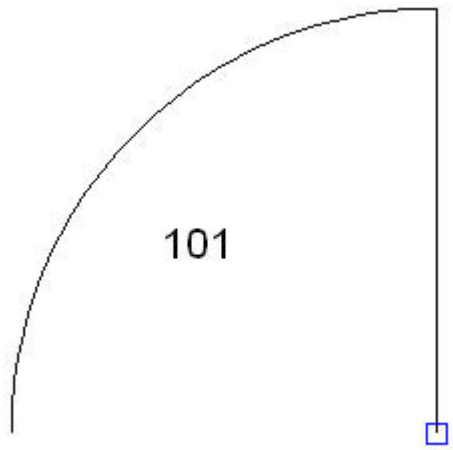
Next we edit the attributes of the door insert by highlighting the door insert and clicking Tools>Inserts>Edit Attributes or by pressing the ellipsis (…) button



on the Has Attributes line of the properties dialog. We change the value to 101 to indicate that this is door number 101 and click OK

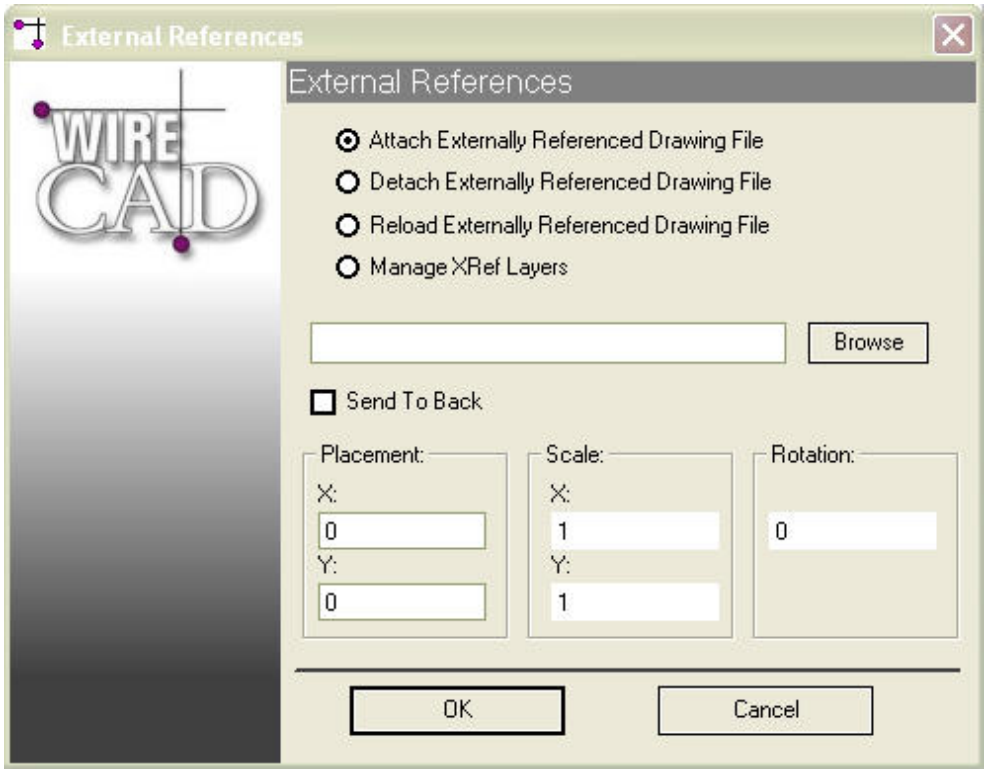


Our insert now displays the insert specific data as follows:



Externally Referenced Drawings (Xref)

WireCAD v3 provides the ability to externally reference another drawing or drawings within the current drawing. Launching this function will display the Xref Manager dialog. From this dialog you can attach, detach, reload, and manage the layers of externally referenced drawings.



Functions:

Attach Xref

Browse  to the drawing file. Determine the Placement, Scale and Rotation. Click OK to attach to the specified file.

Detach Xref

A list of currently attached Xrefs will be displayed. Select the one to detach. Click OK to detach the specified file.

Reload Xref

A list of currently attached Xrefs will be displayed. Select the one to reload. Click OK to reload the specified file.

Manage Xref Layers

A list of currently attached Xrefs will be displayed. Select the one to manage. Click OK to display a Layers dialog for the specified file.

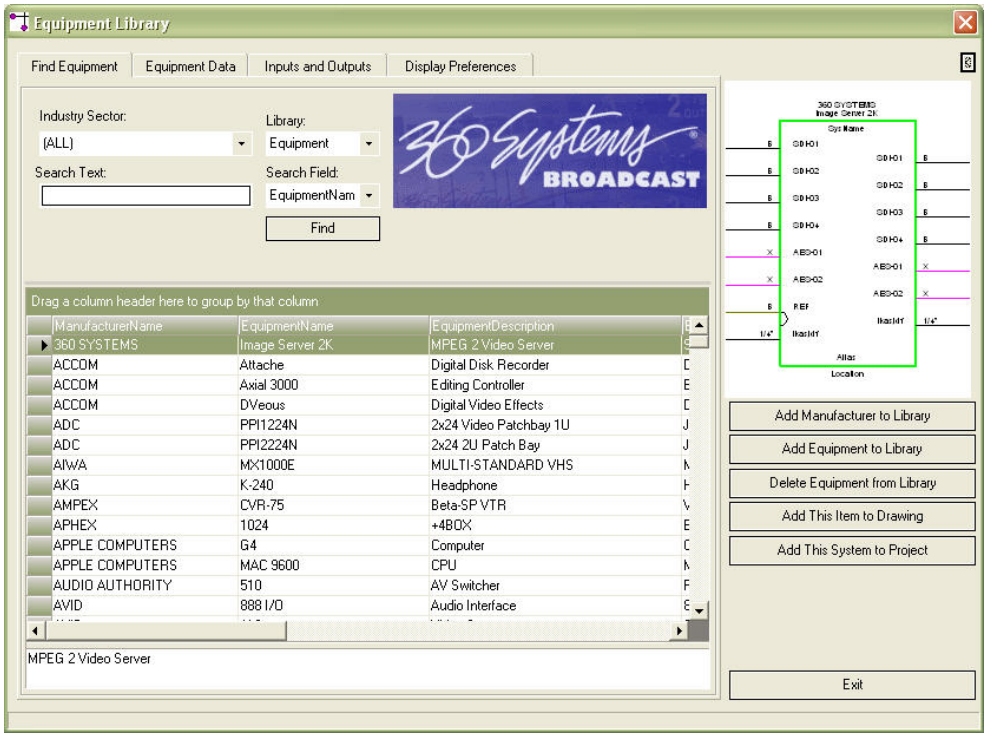
The WireCAD Equipment Library

Adding Equipment to Drawings

Open the Equipment Library and use the Create Block function.

Equipment

The Find Equipment Tab



The Find tab allows you to search the Equipment database, narrow your search parameters using the Industry Sector, and Search Field drop-downs. The default search will be conducted on all Industry Sectors using the EquipmentName field.

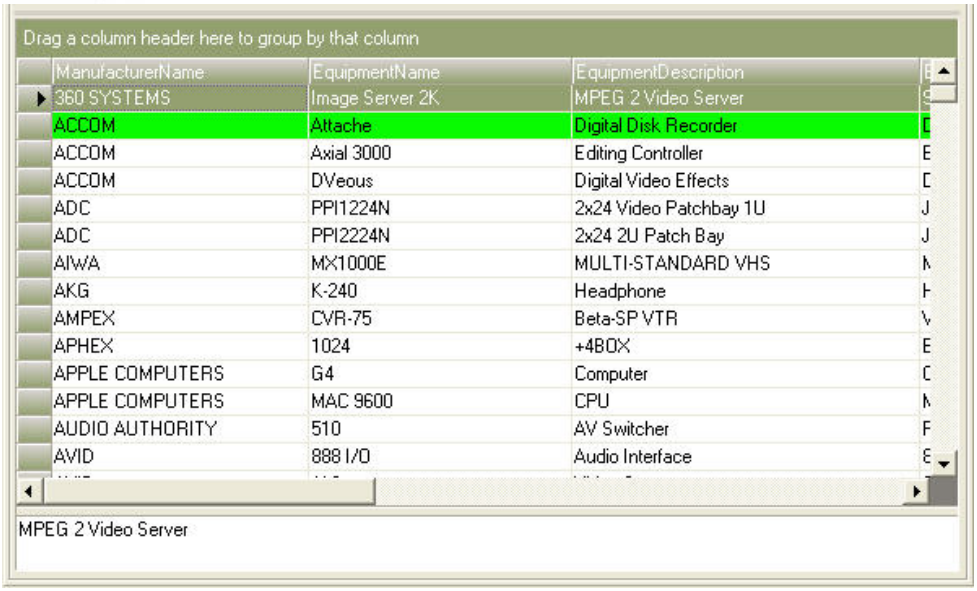
Note that unlike WireCAD v2, Manufacturer names will not appear in this list until you have associated equipment with them.

Copy Equipment Definition function

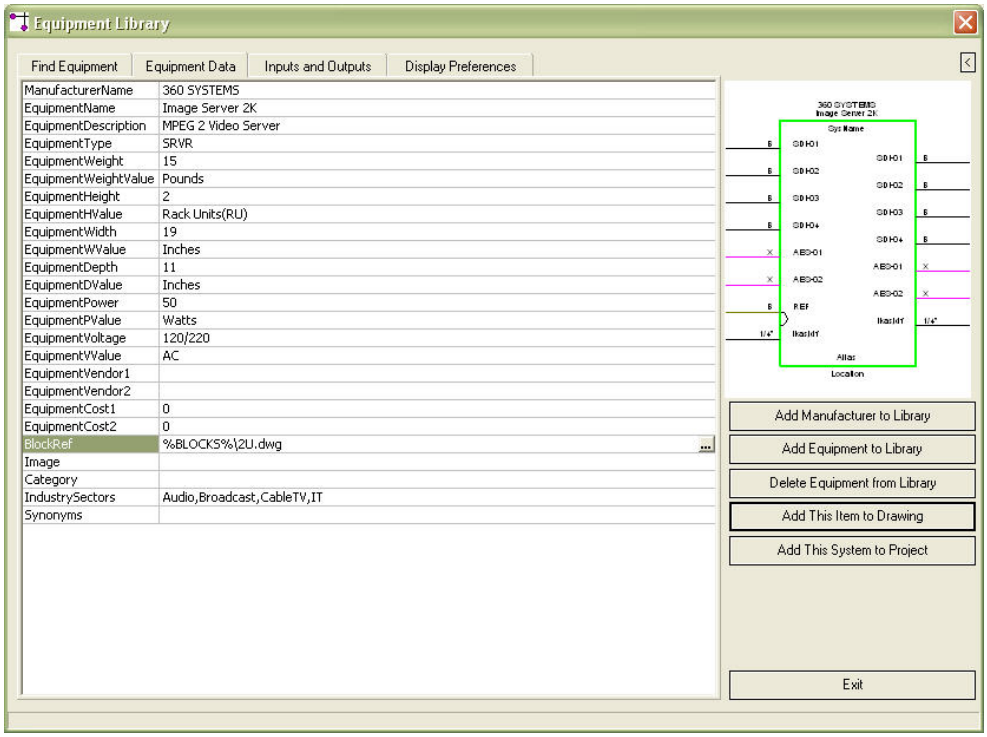
This function assumes that you have a source equipment definition and that you have added the destination equipment to the library.

To copy the entire definition from an existing piece of equipment:

1. Select the equipment to copy from (source) and while holding the Alt key drag the selection to the destination equipment.
2. During the drag operation the current destination will be highlighted in green. See below.
3. Drop the source equipment on the destination equipment.
4. You will be prompted to select whether to copy the Inputs and Outputs. Selecting Yes will copy the Inputs and Outputs from the source equipment into the destination equipment **appending** them to the table.
5. You will then be prompted to whether to copy the Equipment Data. This is the physical data for the source equipment. Selecting Yes will **overwrite** any existing equipment data for the destination equipment. This function cannot be undone.


Drag a column header here to group by that column
ManufacturerName
EquipmentName
EquipmentDescription
360 SYSTEMS
Image Server 2K
MPEG 2 Video Server
ACCOM
Attache
Digital Disk Recorder
ACCOM
Axial 3000
Editing Controller
ACCOM
DVeous
Digital Video Effects
ADC
PPI1224N
2x24 Video Patchbay 1U
ADC
PPI2224N
2x24 2U Patch Bay
AIWA
MX1000E
MULTI-STANDARD VHS
AKG
K-240
Headphone
AMPEX
CVR-75
Beta-SP VTR
APHEX
1024
+4BOX
APPLE COMPUTERS
G4
Computer
APPLE COMPUTERS
MAC 9600
CPU
AUDIO AUTHORITY
510
AV Switcher
AVID
888 I/O
Audio Interface
MPEG 2 Video Server

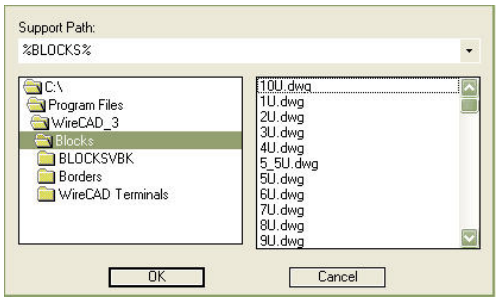
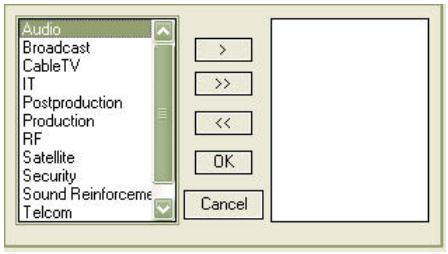
Equipment Data Tab



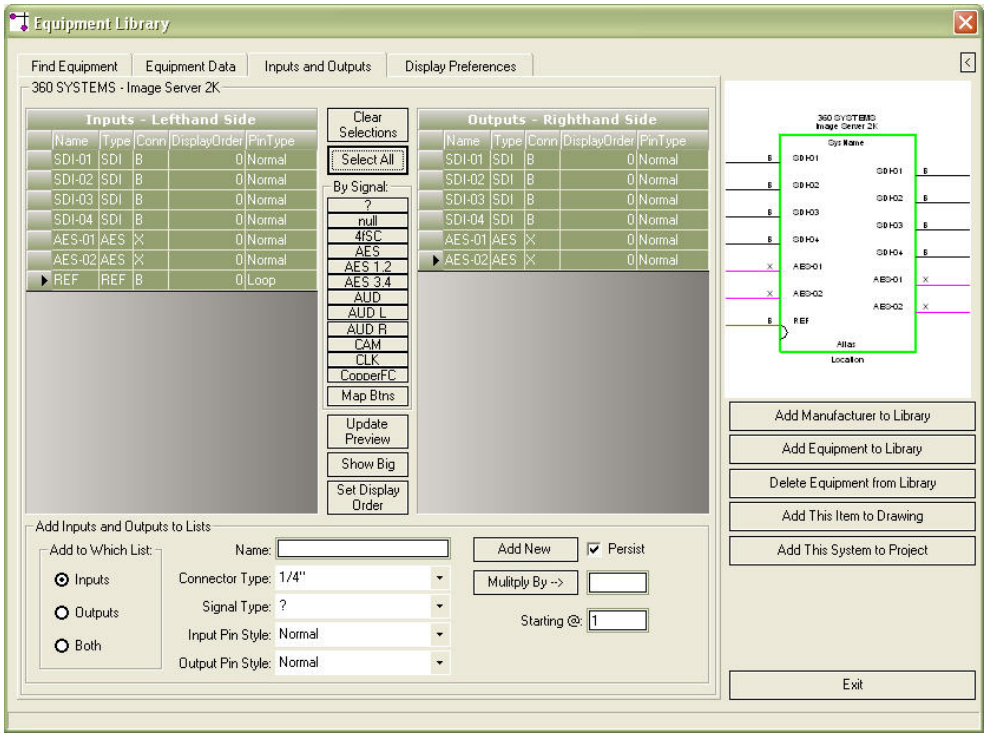
This is where you enter the physical data for the piece of equipment.

The following table illustrates each field and its associated properties:

Field Name	Description	Attributes
ManufacturerName	The Manufacturer Name	Read Only
EquipmentName	The Model Name or Number	Read Only
EquipmentDescription	Description	Editing Allowed
EquipmentType	SysName Prefix Field	Editing Allowed
EquipmentWeight	Weight Quantity	Editing Allowed
EquipmentWeightValue	Weight Value. i.e. pounds, kilos, etc.	Editing Allowed (Pull Down List)
EquipmentHeight	Height Quantity	Editing Allowed
EquipmentHValue	Height Value. i.e. Rack Units, Inches, Centimetres.	Editing Allowed (Pull Down List)
EquipmentWidth	Width Quantity	Editing Allowed
EquipmentWValue	Width Value. i.e. Inches, Centimetres.	Editing Allowed (Pull Down List)
EquipmentDepth	Depth Quantity	Editing Allowed

EquipmentDValue	Depth Value. i.e. Inches, Centimetres.	Editing Allowed (Pull Down List)
EquipmentPower	Power Consumption Quantity	Editing Allowed
EquipmentPValue	Power Consumption Value. i.e. Watts, VA, KVA	Editing Allowed (Pull Down List)
EquipmentVotlage	Voltage Quantity	Editing Allowed
EquipmentVValue	Voltage Value. i.e. AC, DC	Editing Allowed (Pull Down List)
Vendor1, 2	Vendor Name	Editing Allowed
Cost1,2	Cost associated with vendor 1 and 2	Currency, Editing Allowed
BlockRef	External CAD Block Reference	Path to an external block that presents the physical parameters of this device. 
Image	External Image File Reference	Not yet Implemented
Category	Categorical search field	Editing Allowed. Use commas to separate categories.
IndustrySectors	Industry sectors that this equipment touches	Drop down form populated from the IndustrySectors Global Database. 
Synomyns	Synonym search field	Editing Allowed. Use commas to separate categories.

Inputs and Outputs Tab



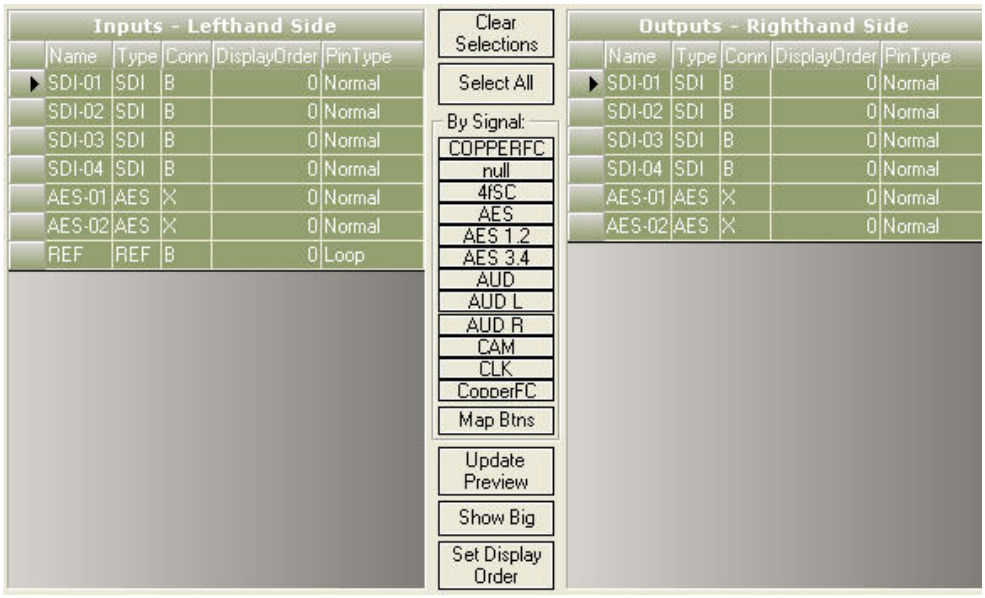
The Input/Output (I/O) section is comprised of:

- Inputs and Outputs lists.
- Search Buttons.
- Select All.
- Clear Selections.
- I/O Adder.
- Update Preview.

The Input/Output tab is where the bulk of the work takes place; use the Equipment Library Input/Outputs tab to:

- Add Inputs and Outputs.
- Add multiple Inputs, Outputs or both to the I/O lists with a single operation.
- Copy Inputs to Outputs and vice versa.
- Edit or Delete Inputs or Outputs.
- Search the lists to select specific Signal Types.
- Select the Inputs, Outputs, or both to use when creating Equipment Blocks.

Input/Output lists



The Input and Output lists provide several functions. Not only are the inputs and outputs displayed but you can also edit any of the information displayed.

To select an Input for display in an equipment block, simply select the box to the left of the Name. To select multiple I/O, hold the Ctrl or Shift key while selecting additional I/O.

Press the Select All 

button to select all of the I/O.

Press the Clear Selections 

button to de-select all I/O.

Press the Update Preview 

button to send changes to the preview display.

Copying Inputs to the Outputs Table

To copy selected inputs to the outputs table:

1. Select the desired Inputs.
2. While holding the Alt key, drag the selection to the Outputs table.
3. Drop the selection on the Outputs table.

The selected inputs will be copied to the Outputs table.

Copying Outputs to the Inputs Table

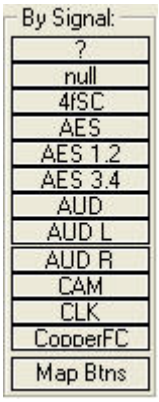
To copy selected inputs to the outputs table:

4. Select the desired Inputs.
5. While holding the Alt key, drag the selection to the Outputs table.
6. Drop the selection on the Outputs table.

The selected inputs will be copied to the Outputs table.

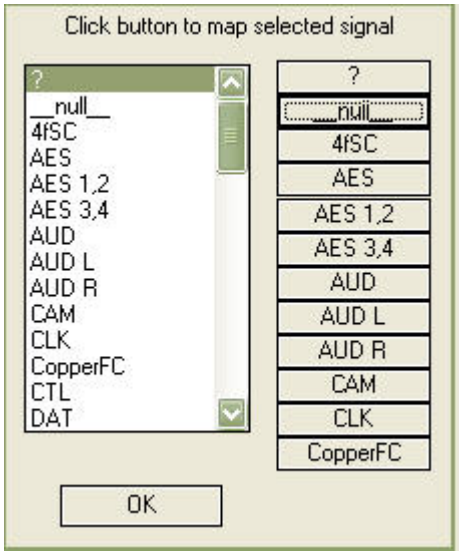
Note: You must use the I/O Adder to add Inputs and Outputs to the lists.

Search Buttons



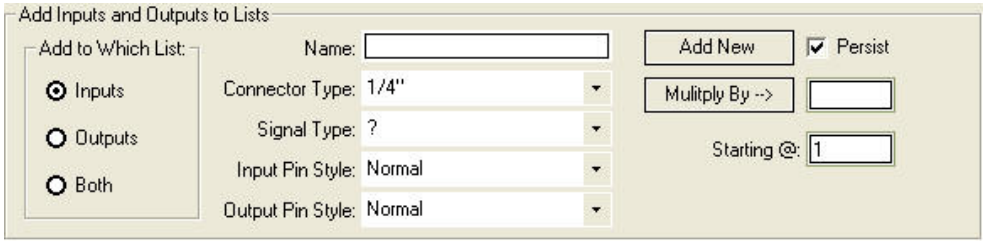
Mapping Signals to the Search Buttons

This function selects all of the inputs and outputs of the selected Signal Type. To map an existing signal type to one of the twelve buttons, click “Map Btns” to display the following dialog:



Select one of the Signal Types in the left window and then click on the right-hand button that you wish to map the Signal Type to. The update will occur immediately. Click OK to close the dialog.

I/O Adder



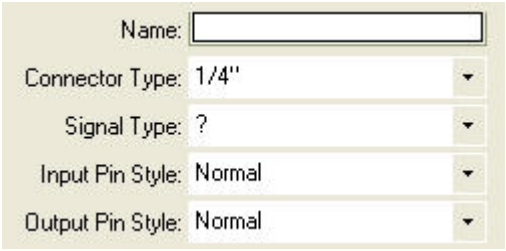
This is where you add inputs or outputs or both inputs and outputs simultaneously.

How it works

Determine whether you are going to add data to the Inputs list, the Outputs list, or both by selecting the appropriate button in the Add to Which List: frame:

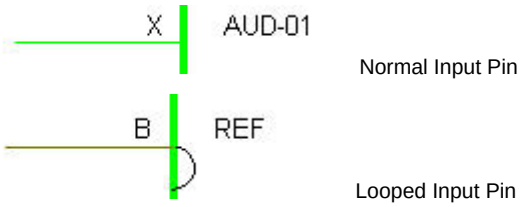
Type the name of the I/O in the “Name” field and enter the appropriate data in the “Connector,” “Signal Type,” and “Input Pin Style” and “Output Pin Style” fields.

The Connector and Signal Type drop-downs are populated from the Connectors, and Signal Types Global Databases.



Input Pin Styles

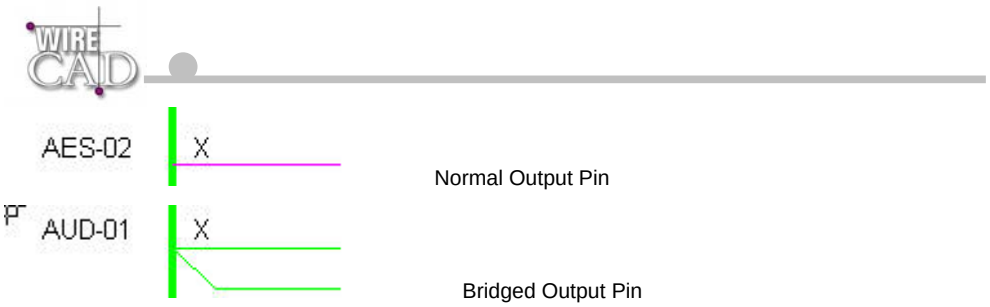
Input pins can be displayed in one of two forms: Normal or Looped. See illustration below.



A looped input pin passes the signal on to another input and thus acts as both an input and an output.

Output Pin Styles

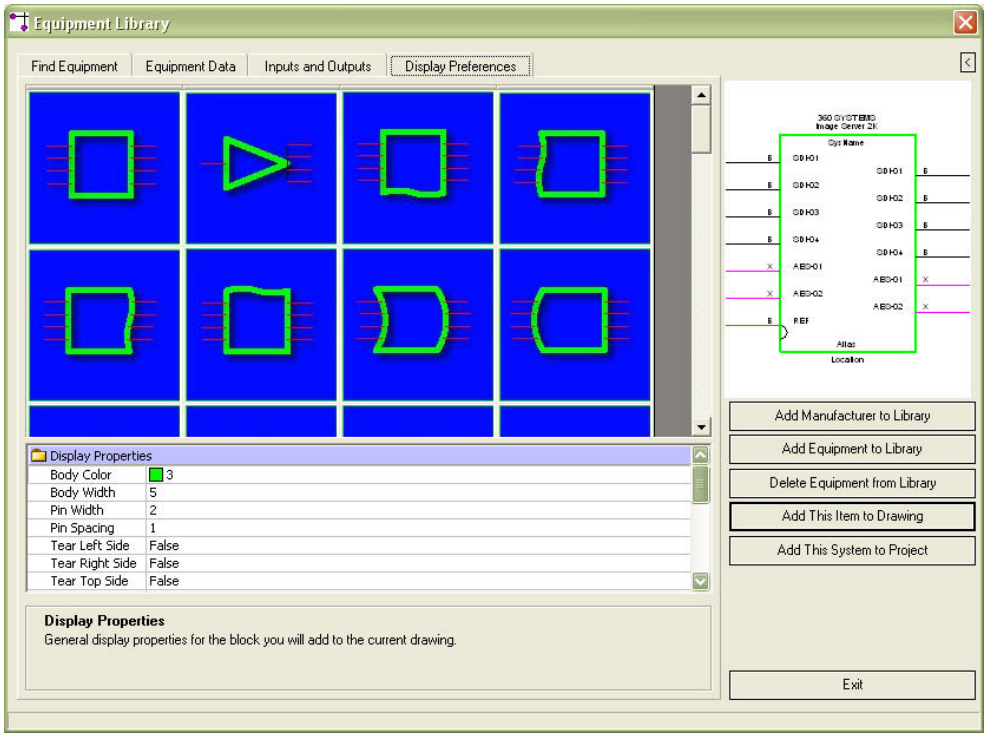
Output pins can be displayed in one of two forms: Normal or Bridged. See illustration below.



A bridged output provides a parallel connection from a single output.

If you are adding a single input, output, or both then select Add New. If you wish to add multiple I/O, then enter the number to multiply by in the Multiply By field and the appropriate start number in the Starting @ field. Then click on the Multiply By button (to the left of the Multiply By field) and WireCAD will create a new I/O for every multiple and append the number to the Name field. For example if you have a device that has 16 inputs all named “Line In”, rather than typing an entry for each of them, you would type “Line In-” in the name field. Then enter 16 in the Multiply By field and 1 in the Starting @ field. Selecting the Multiply By button will produce 16 entries in the input list starting at Line In-01 and ending at Line In-16. Selecting the persist info check box will leave the data in the fields after creating a new I/O.

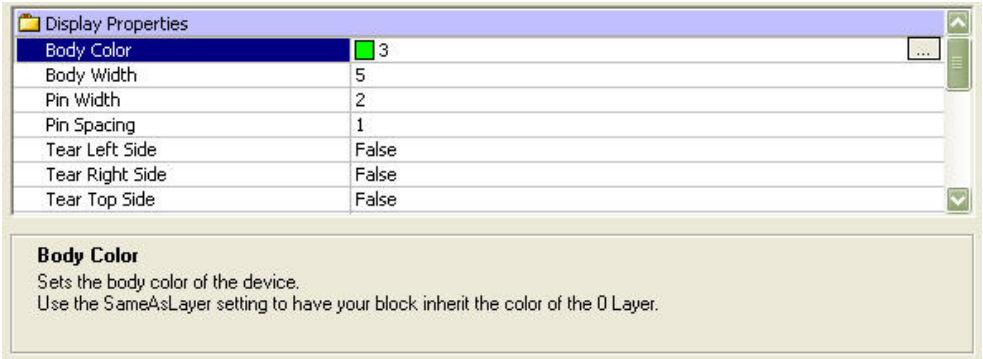
Display Preferences Tab



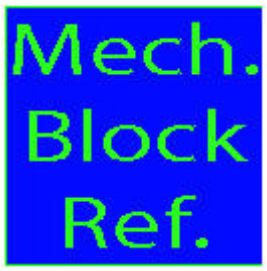
The Display Preferences Tab allows you to determine a number of different ways to display the equipment block that you are about to create. You have the ability to display the block as any number of the following permutations:

Display Properties

On the Display Preferences tab are a number of Display Properties that determine the outcome of the displayed device.

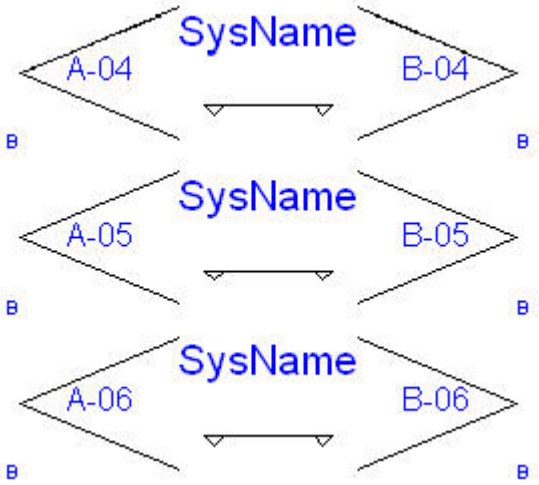


Property	Description	Value	Applies To:
Body Color	Sets the Body Color of the block.	0-255	Normal equipment blocks. Not terminals.
Body Width	Sets the width of the block in drawing units.	1-~	Normal equipment blocks. Not terminals.
Pin Width	Sets the width of the pin. In drawing units.	Enum / Drop-down 1,1.5,2	Normal equipment blocks. Not terminals.
Pin Spacing	Sets the vertical pin spacing. In drawing units	>1	Normal equipment blocks. Not terminals.
Tear Left Side	Tears the left side of the block. Works with the	Boolean	Normal equipment blocks. Not terminals.

	Bulge Left Value		
Tear Right Side	Tears the right side of the block. Works with the Bulge Right Value	Boolean	Normal equipment blocks. Not terminals.
Tear Top Side	Tears the top side of the block. Works with the Bulge Top Value	Boolean	Normal equipment blocks. Not terminals.
Tear Bottom Side	Tears the bottom side of the block. Works with the Bulge Bottom Value	Boolean	Normal equipment blocks. Not terminals.
Bulge Left Side	Polyline Bulge Distance. This value is set by the display style. Modify it to suit.	Signed Double	Normal equipment blocks. Not terminals.
Bulge Right Side	Polyline Bulge Distance. This value is set by the display style. Modify it to suit.	Signed Double	Normal equipment blocks. Not terminals.
Bulge Top Side	Polyline Bulge Distance. This value is set by the display style. Modify it to suit.	Signed Double	Normal equipment blocks. Not terminals.
Bulge Bottom Side	Polyline Bulge Distance. This value is set by the display style. Modify it to suit.	Signed Double	Normal equipment blocks. Not terminals.
Visible Manufacturer	Sets the visibility of the ManufacturerName Attribute	Boolean	Normal equipment blocks. Not terminals.
Visible Equipment Name	Sets the visibility of the EquipmentName Attribute	Boolean	Normal equipment blocks. Not terminals.
Visible SysName	Sets the visibility of the SysName Attribute	Boolean	Normal equipment blocks. Not terminals.
Visible Alias	Sets the visibility of the Alias Attribute	Boolean	Normal equipment blocks. Not terminals.
Visible Location	Sets the visibility of the Location Attribute	Boolean	Normal equipment blocks. Not terminals.
Visible Connectors	Sets the visibility of the Connectors text.	Boolean	Normal equipment blocks. Not terminals.
Text Height	Sets the Height of all visible text associated with a block	Double	Normal equipment blocks. Not terminals.
Block Reference	Path to an external drawing that physically represents the equipment.	This value is set in the BlockRef field of the Equipment Data tab	Used by the  button. Ignores all other settings and inserts the

			external reference into the drawing.
Terminal Merge Mode	Map merge modes are used to combine information from an input and an output into one terminal that displays both	Enum / Drop-down None By Position Matching Text	Used when mapping definitions to terminals. Not used by Normal equipment blocks.

In addition, WireCAD v3 provides the ability to map an equipment definition onto a series of Terminals. This allows you the flexibility to display a jack-field, router, punch-block, or bulk-head panel as a series of terminals, as follows:



Above demonstrates mapping an equipment definition of a jack-field onto a jack terminal.

To map a selection to the terminal:

1. Sort the input and output grids so that they line up appropriately. Remember that there are three map merge modes. Map merge modes are used to combine information from an input and an output into one terminal that has both; as in above.
2. Select the inputs and outputs to display.
3. Switch to the Display Preferences tab and select the terminal from the display grid. See Below.
4. Select the Merge Map Mode.
5. Check the preview display to insure a proper display.
6. Click the 

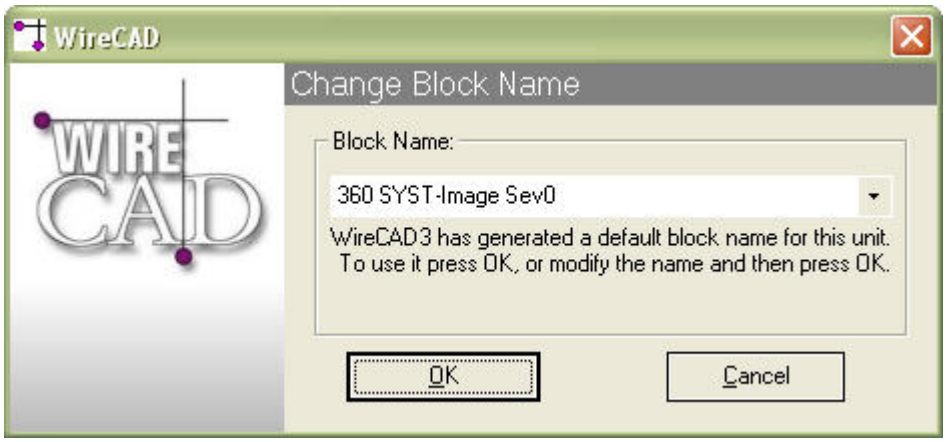
to send the terminal(s) to the drawing.

Create Equipment Block

After you have found the piece of equipment that you wish to represent in your drawing, click on the Inputs/Outputs tab and select the appropriate I/O, either by clicking on the individual I/O or using the buttons. Then click Add This Item to Drawing.

If you have not checked the Use Default Block Names check box in the preferences dialog,

then you will see a dialog asking you to confirm the name of the block to be created. You may either select the default, or enter a name of your own.



Anatomy of an Equipment Block

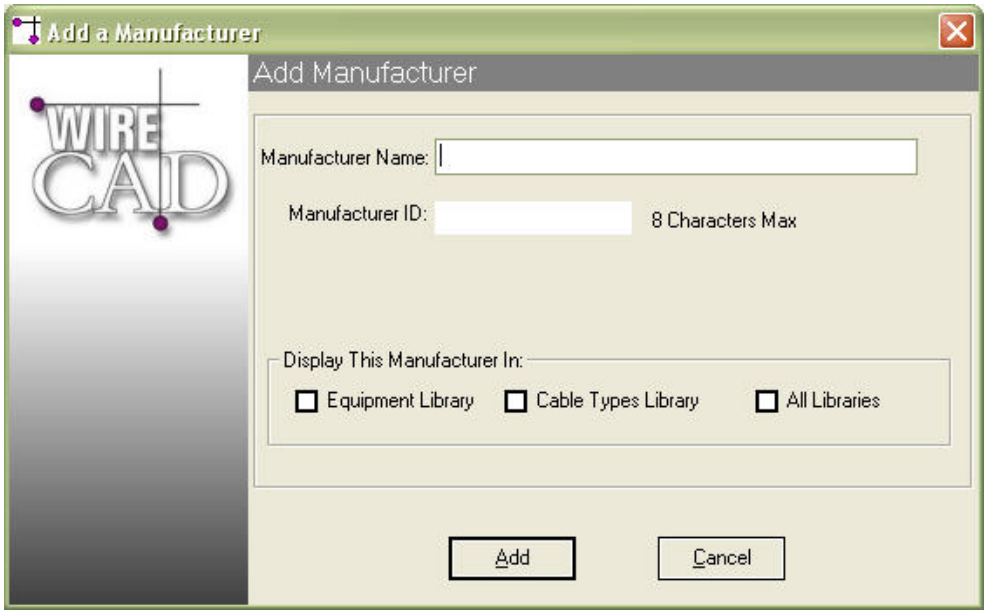
Equipment Blocks are comprised of the following attributes:

The following table provides an explanation for each of the attributes:

Attribute	Description
INPUT##	Input. The inputs are labelled from the top of the block to the bottom
OUTPUT##	Output. The outputs are labelled from the top of the block to the bottom
SysName	The SysName. Concatenation of the SysNamePrefix and SysID
Location	Concatenation of the Location and Elevation fields of the SysName database
Manufacturer	Manufacturer
EquipmentName	Equipment Model
EquipID	Reference to the Equipment Library
PinWidth	A reference to the pin width
Alias	System Alias Field

Add Manufacturer to Library

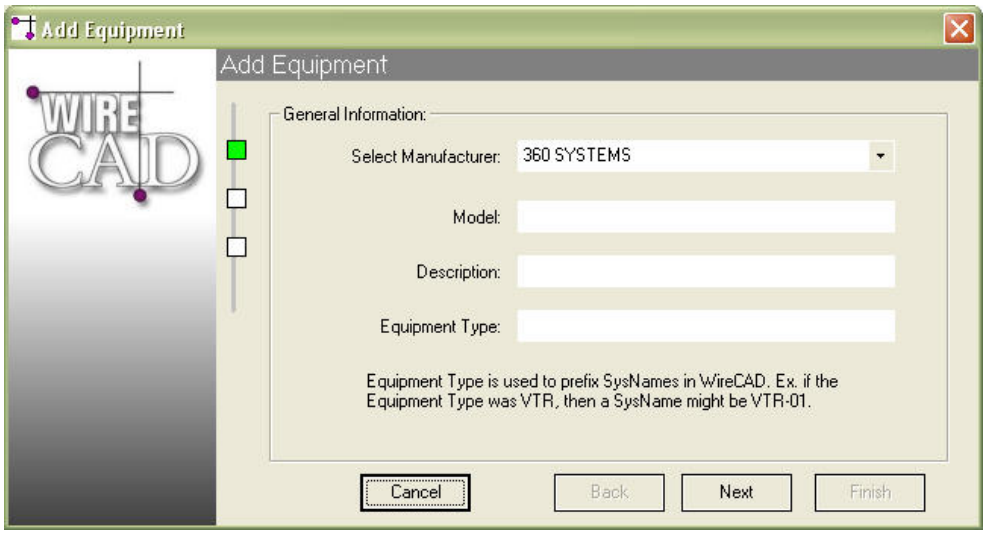
To add a manufacturer to the library click on the Add Manufacturer to Library button. 

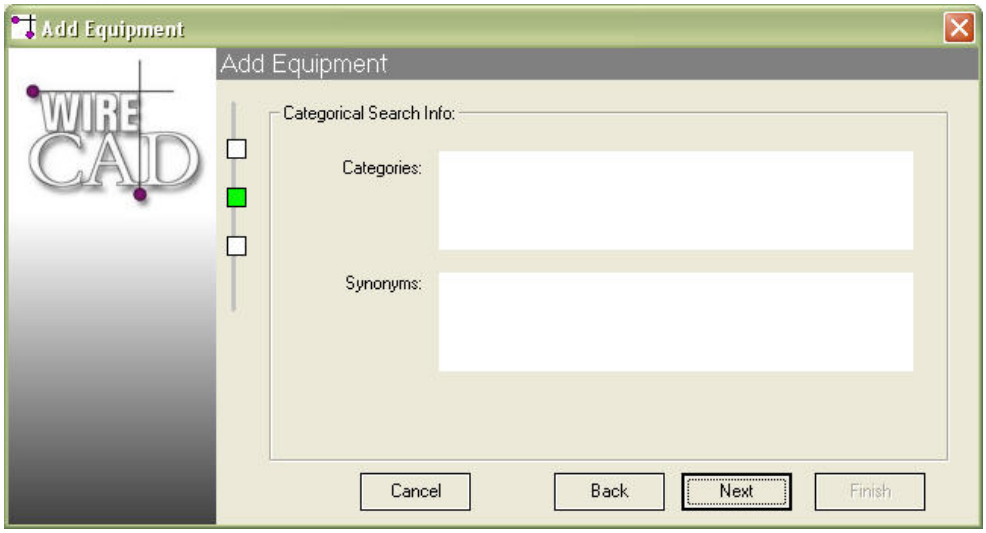


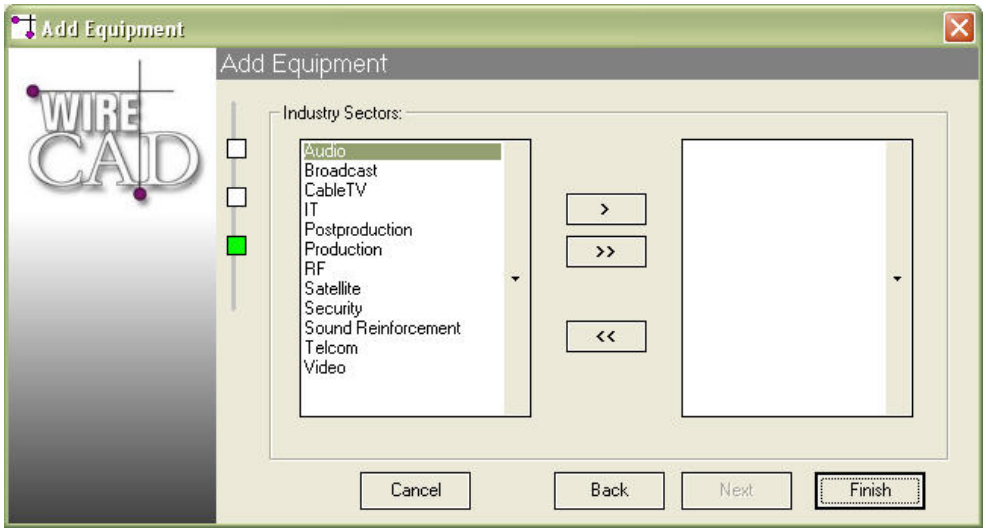
Type the Manufacturer Name in the field. The ManufacturerID will be filled in automatically for you. Select the library where the name is to be displayed. Click Add. If the ManfacturerID already exists, you will be warn and given a chance to modify it.

Add Equipment to Library

To add a new piece of equipment to the library, click on the Add Equipment to Library button.







Select Manufacturer: Select an existing manufacturer from the drop-down list.

Equipment Model: Enter the Equipment Model Number.

Equipment Description: A short description of the equipment.

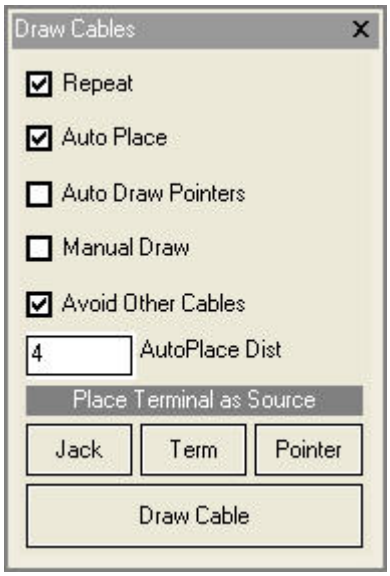
Equipment Type: This field is used to prefix system names. Note: use types that are easily recognizable to the engineering staff. This will aid in keeping your documentation readable. Example: if you are using an AVID non-linear editor (NLE) in your system it may be more readable to use AVID as the prefix instead of NLE; however, this assumes that the system will always by occupied by and AVID and may not be the best way to future proof your system. It's a "salt to taste" issue.

Drawing Cables, Terminals, Pointers etc.

Important!! Draw cables in your drawing from output to input. WireCAD v3 now has a cable auto-router that will draw your cable for you and try to avoid other drawing entities and cables.

Note: if you check the Avoid Other Cables box on the Draw Cables toolbar, WireCAD v3 will lay cables on top of other cables.

The Draw Cables Toolbar



Repeat

Restarts the Draw Cables function after successfully drawing a cable.

Auto Place

Automatically places Jacks, Terminals (Term) and Pointers in the drawing to the right or left of the selected input or output (based in the value in the AutoPlace Dist field).

Auto Draw Pointers

Checking this box with place linked pointer between the selected input and output.

Note: this function will only work when the output and input are on the same sheet.

Manual Draw

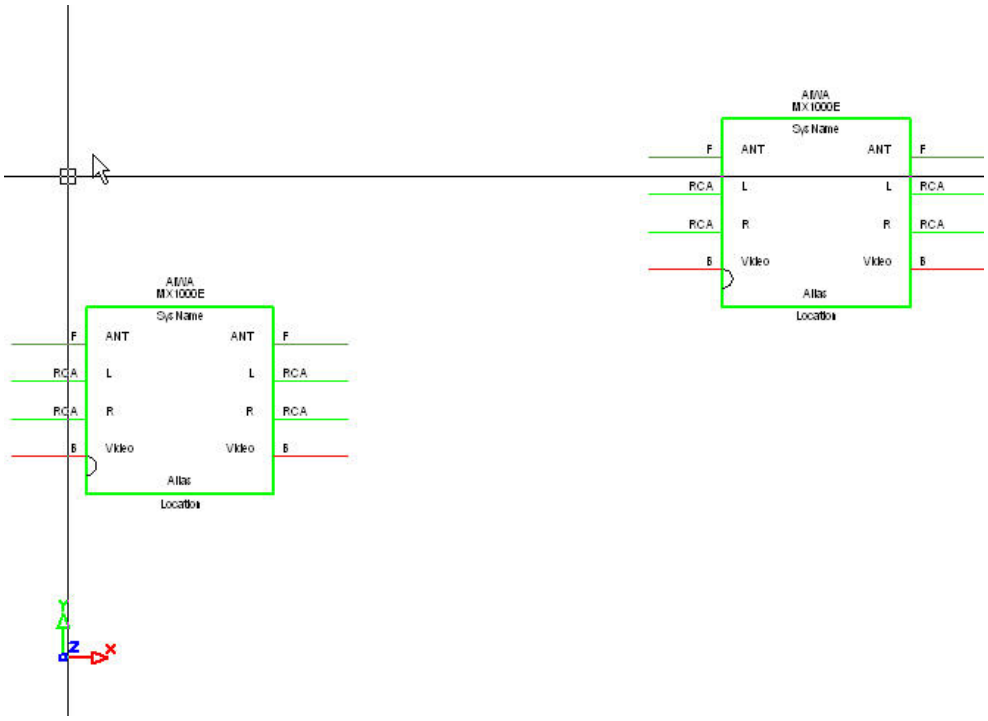
Manually draw the cable.

Note: When this function is selected, you will be prompted to draw a standard 2D polyline. Start by selecting the output and the left-click for every vertex in the line (cable). Right-click to finish the command and draw the cable-text entities.

Drawing Cables Tutorial

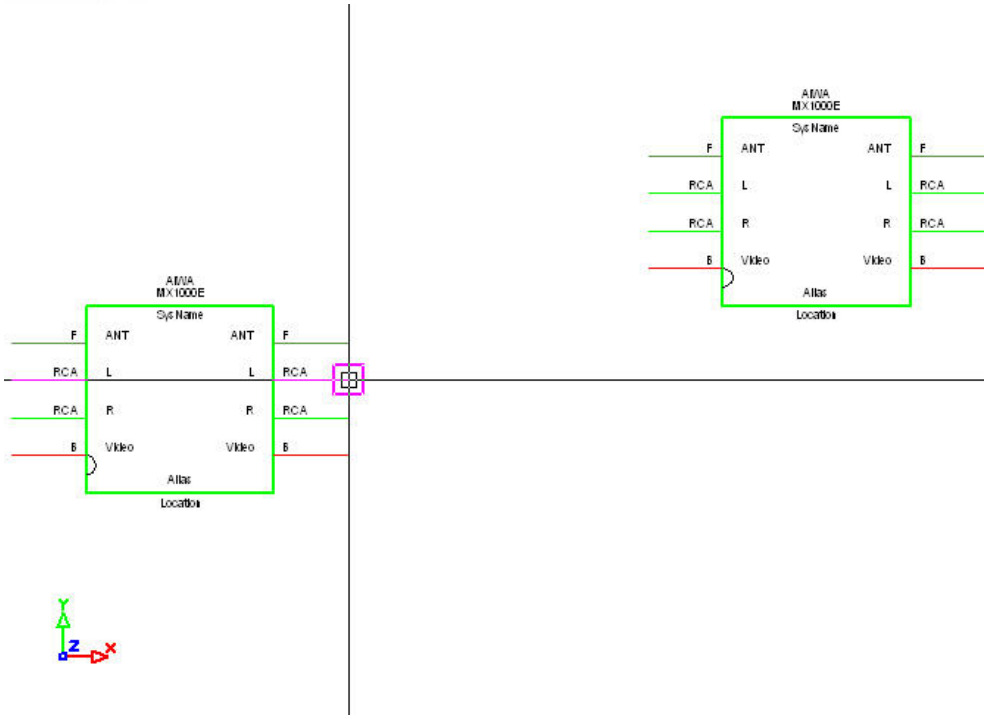
WireCAD version 3.2.1355 allows the drawing are cable from input to input and from output to output. Note, however, that the pin that is selected first will be assigned to the SrcPin and the pin that follows will be assigned to the DstPin field. Note further that this practice may cause erroneous connector counts. What follows is a detailed explanation of drawing cables.

Start with a piece of equipment in the drawing:



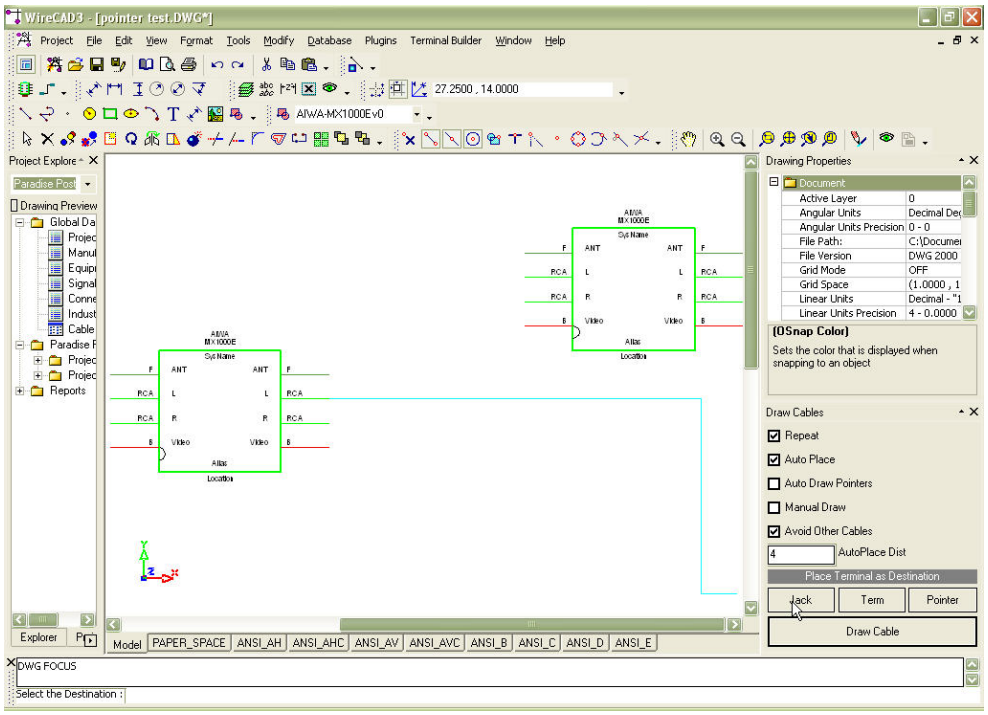
Next, select the Draw Cables button:

Next, select an output on the equipment block:

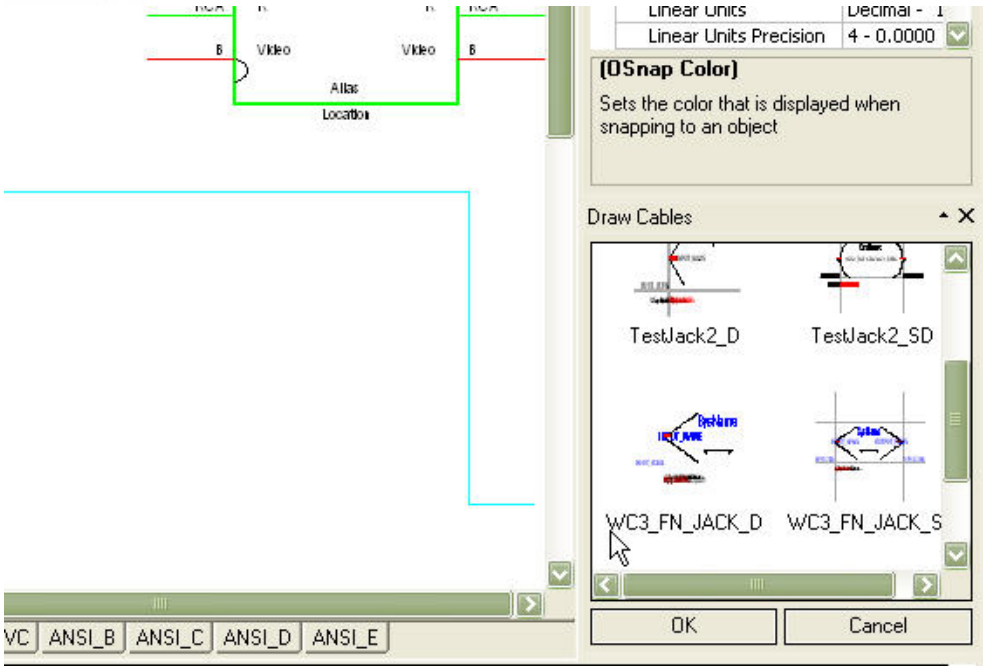


We will now automatically draw a Jack to the right of the selected output. Select the Jack button:

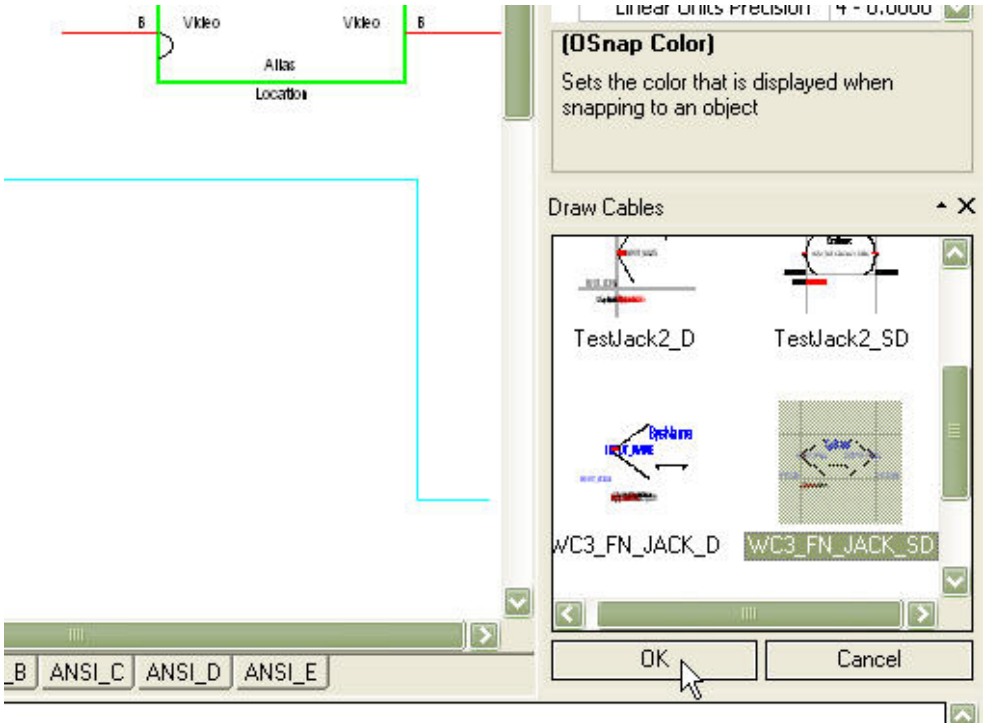
As follows:



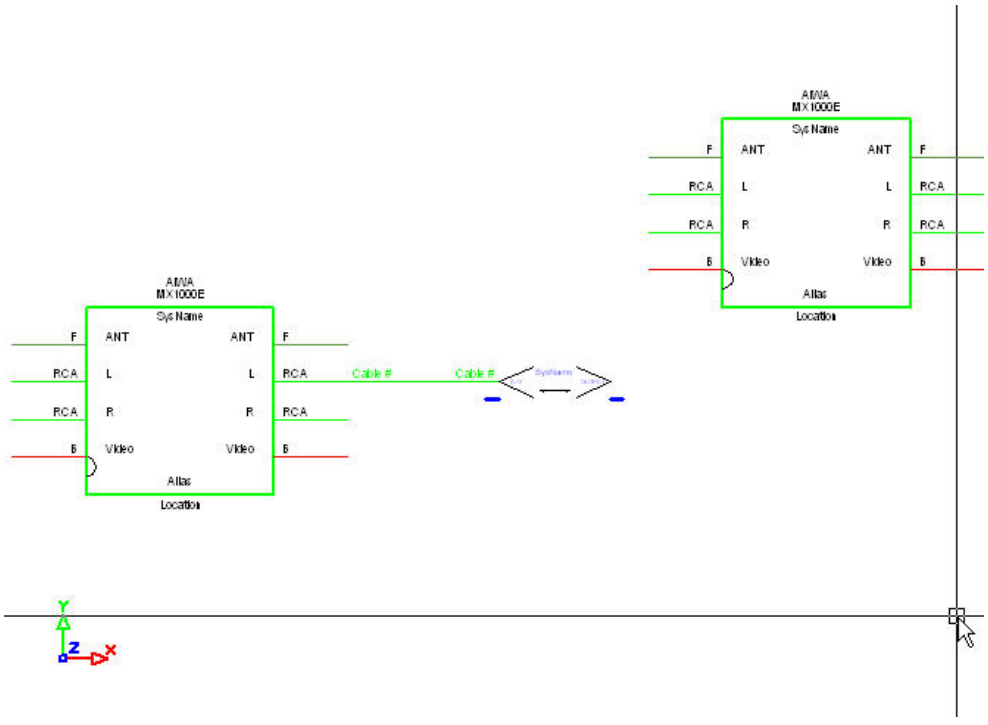
When selecting a Jack, Terminal, or Pointer you are given a choice of styles. Shown here are the Jack types:


Linear Units
Decimal - 1
Linear Units Precision
4 - 0.0000
(OSnap Color)
Sets the color that is displayed when snapping to an object
Draw Cables
TestJack2_D
TestJack2_SD
WC3_FN_JACK_D
WC3_FN_JACK_S
OK
Cancel
Video
Video
Alias
Location
ANSI_B
ANSI_C
ANSI_D
ANSI_E

Select the one called WC3_FN_JACK_SD:

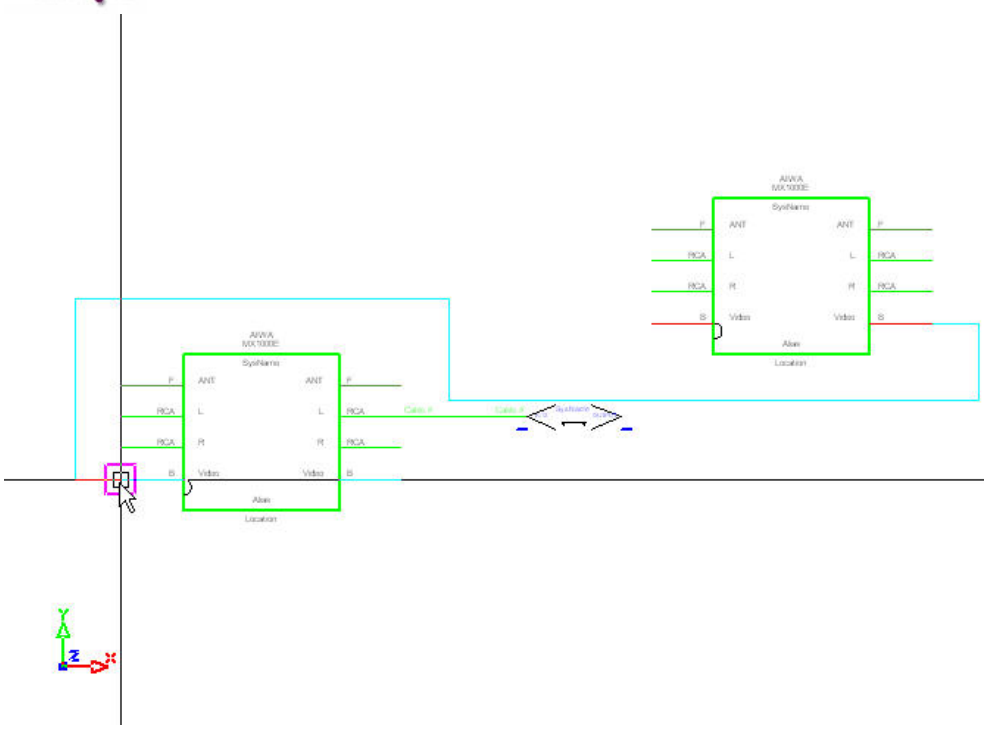


Then click OK. This will insert a fully normalled jack pair:



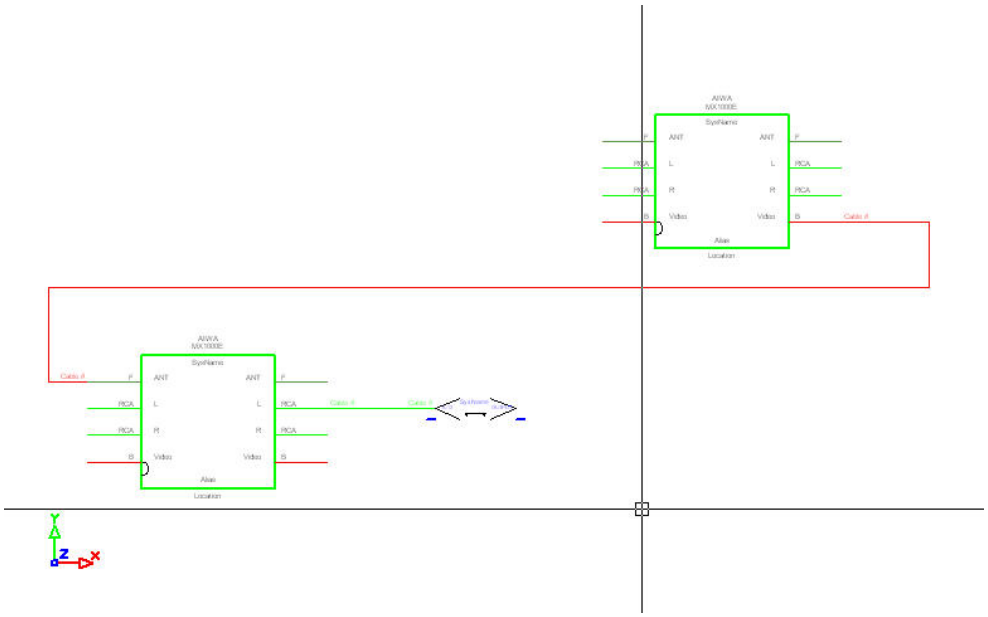
Note that the jack, cable, and cable number text are automatically placed in the drawing 4 drawing (number in the AutoPlace Dist field) units to the right of the output.

If the objective is to draw from output to input on different blocks, then select the output first:



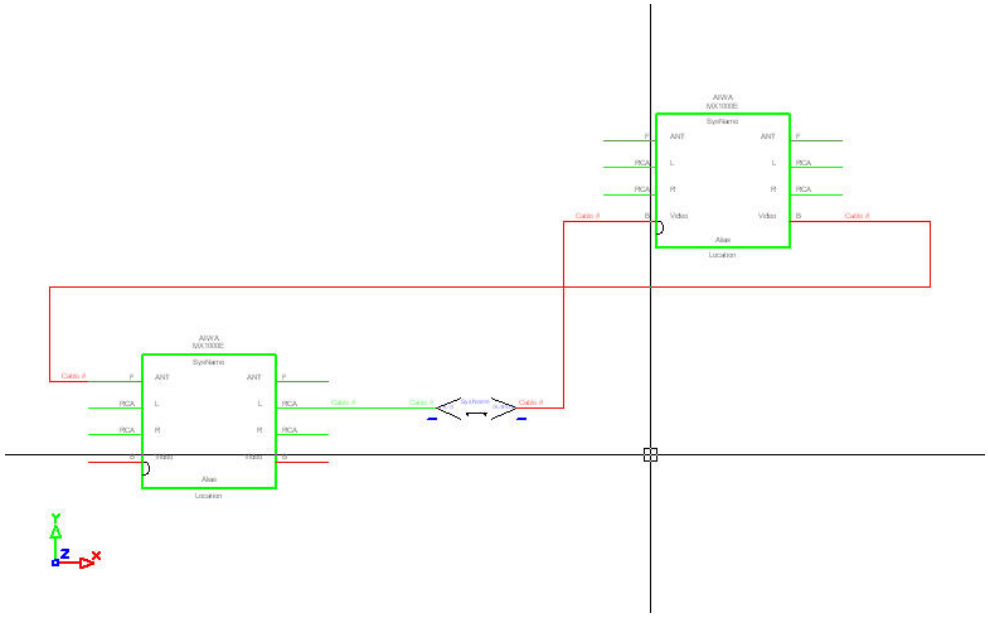
Next, select the input of the device. A cable and cable number text will be placed automatically.

Note: WireCAD will try to avoid equipment and other cables (based on the Avoid Other Cables checkbox). If you need to move a piece of equipment, select it and then grab its grip and drag it to its new location. The wires will remain attached to it and relocate based on the location.



If the objective is to draw from our existing jack to the block, then start the Draw Cable function

Draw Cable and click on the jack.

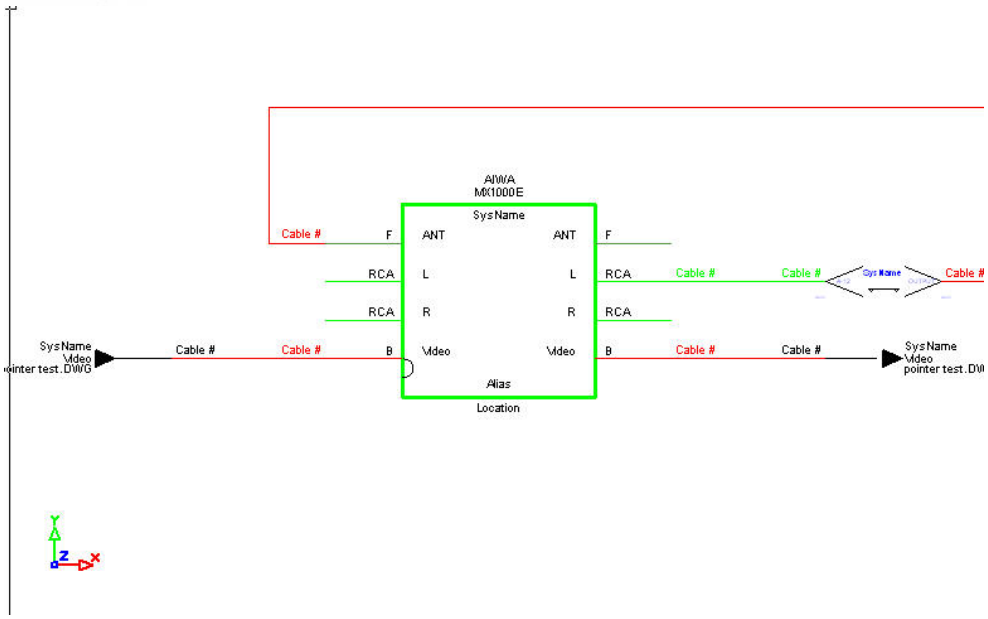


Drawing Pointers Automatically

Check the Auto Draw Pointers checkbox. Start the Draw Cables function.

Draw Cable

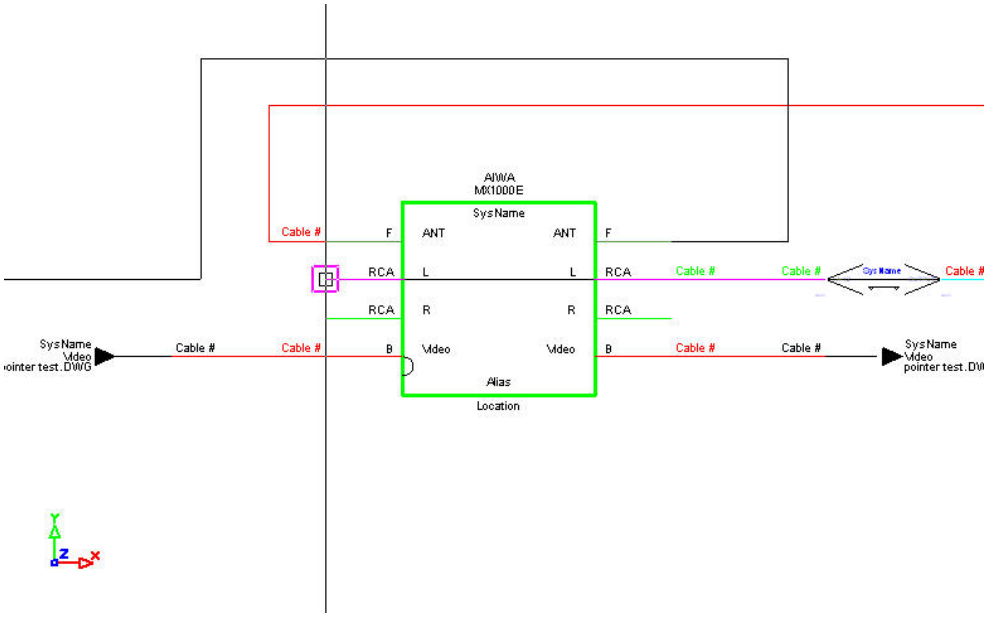
Next, Select the Output followed by the Input.



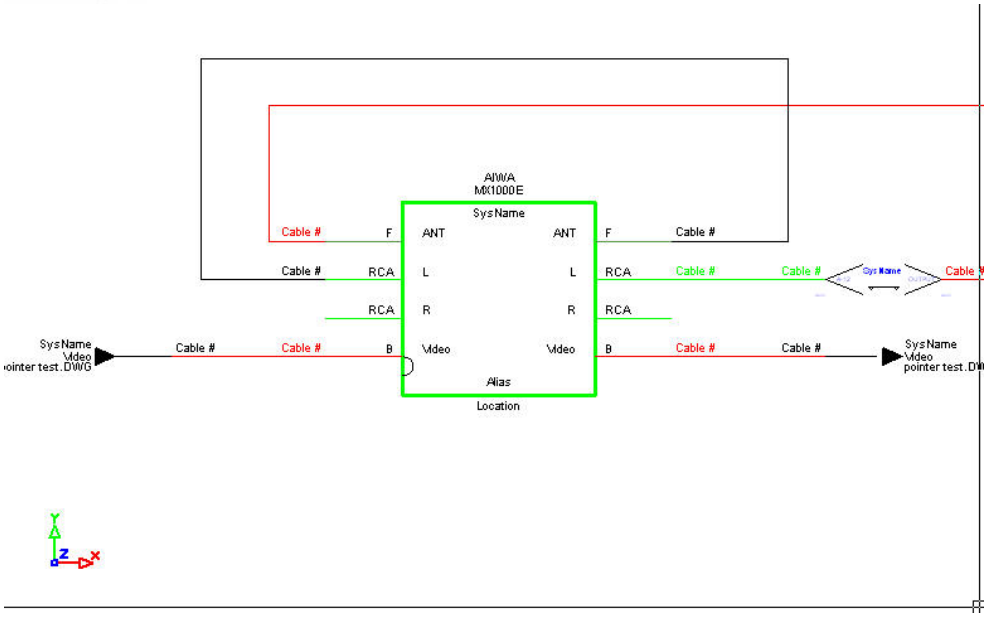
Manually Drawing Cables

Check the Manual Draw checkbox. Start the Draw Cables function.

You will be prompted to start drawing a polyline. Select the Output. Draw the polyline in any direction needed.



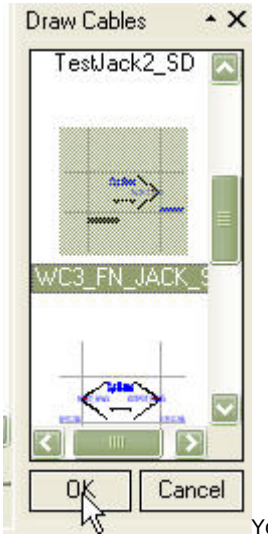
Finish by selecting the desired Input. End the command by right-clicking. This will place the cable# text entities.



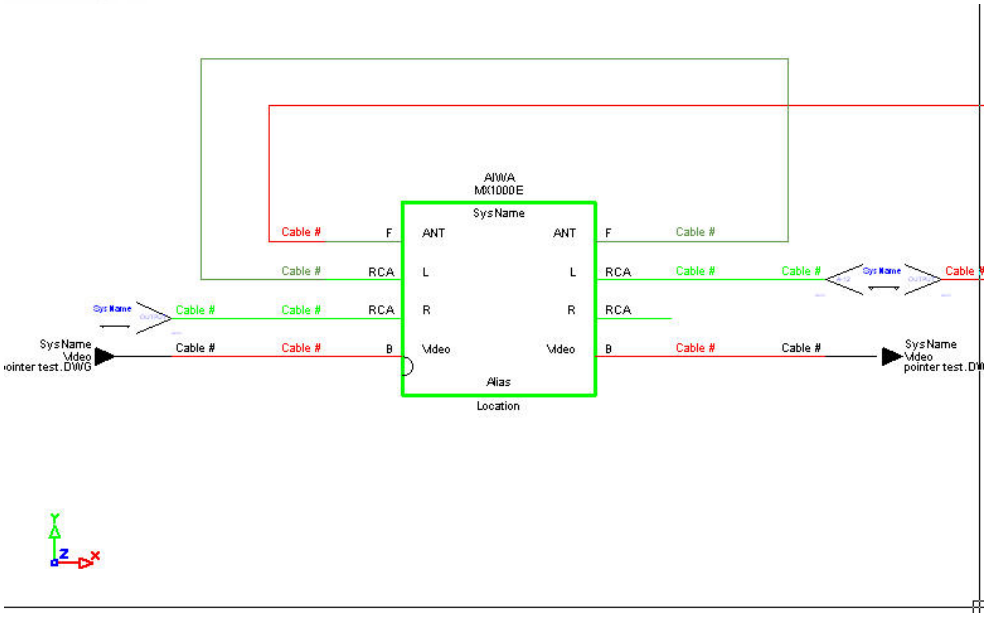
Drawing Jacks, J-Boxes, etc as Sources to the Cable

If you need to have a Jack, Terminal, or Pointer feed an input. Select the terminal button first. 

Note that the Manual Draw function must be unchecked in order to enable the terminal switches. Selecting any of the terminal buttons will start the Draw Cable function.



You are then prompted to select a destination. Upon doing so, if the Auto Place function is checked, the selected Terminal will be placed to the left of the Input based on the number defined in the Auto Place Dist field:



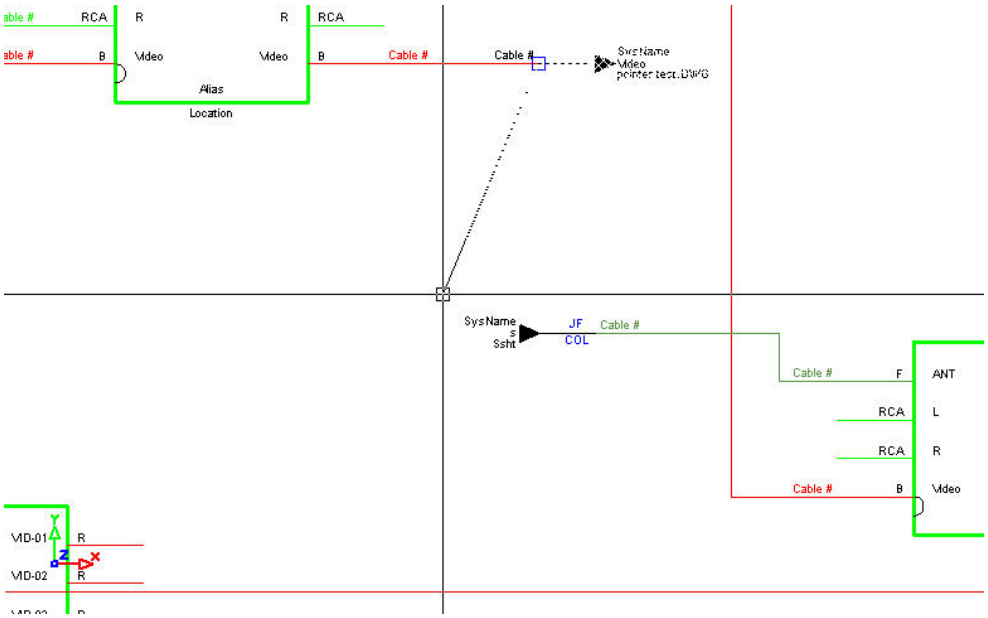
Turning off Auto Place will prompt you for locations to place the terminal devices and then draw cables to the selected location.

On-Sheet and Off-Sheet Pointers

WireCAD supports the use of on-sheet and off-sheet pointers. These are placed in the drawing using the Draw Cables function. Alternately, pointers can be placed using the Insert block function and then drawn to or from them. When using the Auto Draw Pointers function, WireCAD automatically links the pointers together. When using the Pointer Terminal button to place pointers in the drawing, these pointers will require linking.

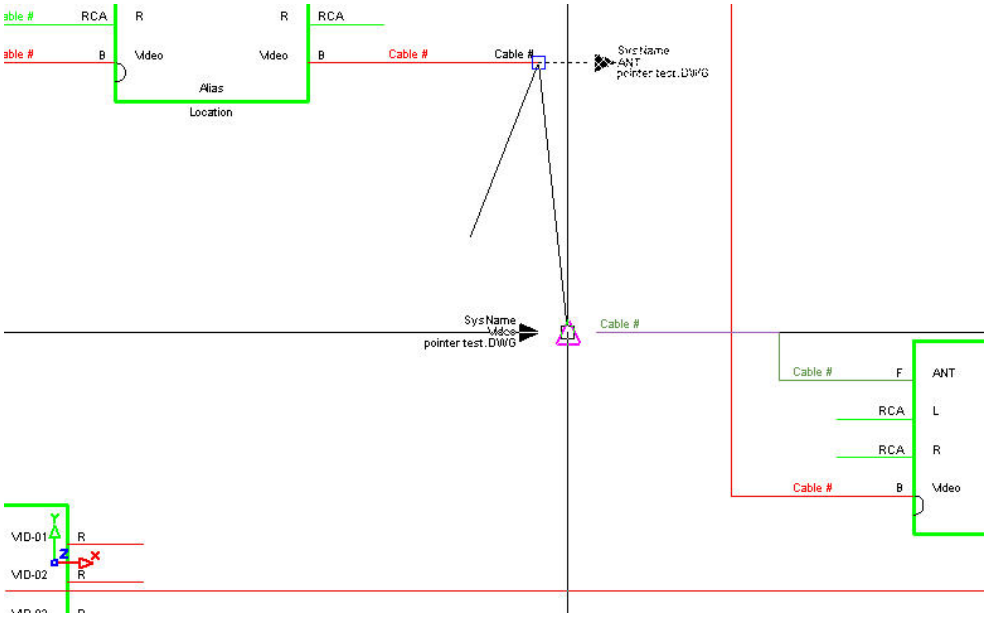
Linking Pointers

Pointers can be linked on sheet or off. Simply double-click the pointer on the output side of the cable:



You will be prompted to select a pointer in the current drawing or in another drawing:

In any case, click on the other pointer to link them together.



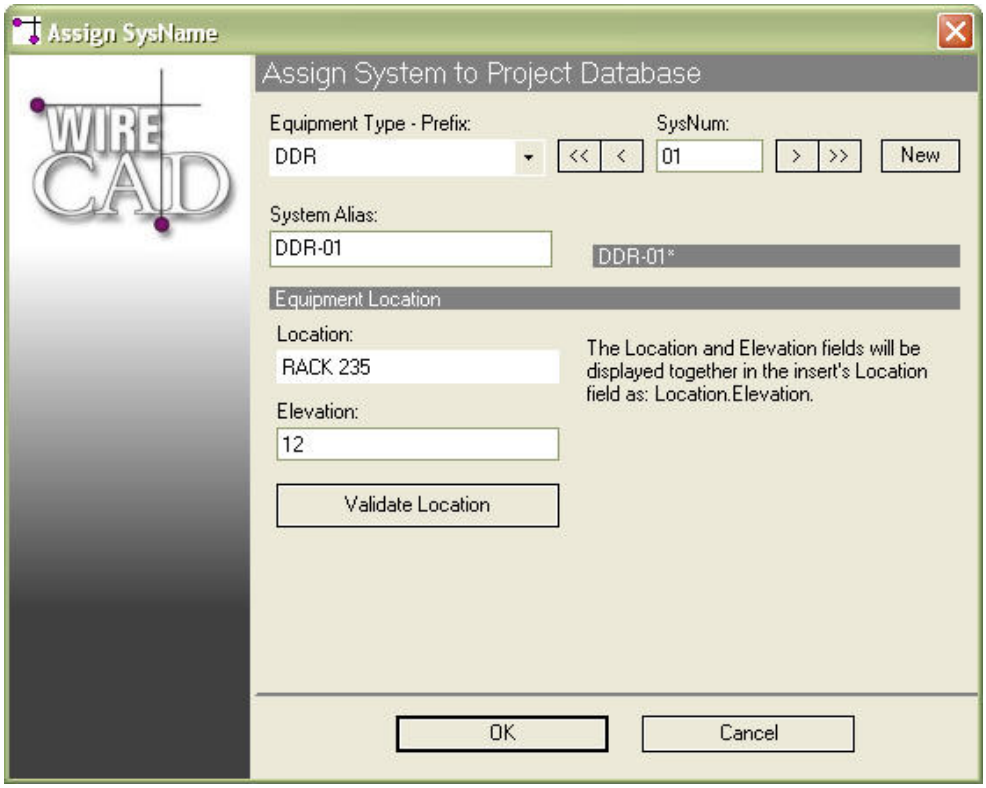
Note that the two pointers are filled in with the reciprocal information. In other words, the pointer on the output side is filled in with the information from the input side and vice versa:

Assigning System Names to Equipment

Assignment is the term used to indicate that a drawing entity has an associated entry in the database. We refer to several types of assignment:

- SysNames, or system specific identifiers. Thus allowing multiple instances of the same equipment type. This process is where you define the physical location of the equipment in the system.
- Terminals, terminals must first be assigned to the project database. This process is where you define the physical location of the equipment in the system.
- Cable Numbers.

Assigning System Names (SysNames), Jacks, Jack Fields, and Cable Numbers is as easy as double-clicking on the entity you wish to assign. Depending on the entity type you will be presented with a series of different dialogs to handle the interface between the drawing environment and the database. If you double-click an equipment block, the following dialog will appear:



Note: A SysName can be assigned to multiple instances of an equipment insert. As an example, assume two drawings – one containing only video, the other containing only audio. Both with a VTR called VTR-01.

Prefix: Sets the prefix for the system Name. Direct entry into this field is possible. This field is referenced to the EquipmentType field in the Equipment Library. The Prefix and the SysNum will be concatenated to form the SysName.

Record Selectors: 

Used to select an existing SysName. Upon selecting an existing SysName, click OK. You will be informed that the System Name already exists, and prompted whether to proceed with

naming the insert. Selecting the New Record button 

will append a record to the database and increment the SysNum counter.

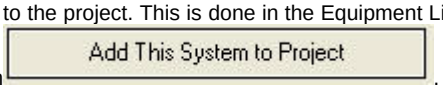
Note that WireCAD v3 automatically assumes a new record as indicated above by the * in the SRVR-01* SysName preview.

Location: User defined location field.

Elevation: User defined elevation field.

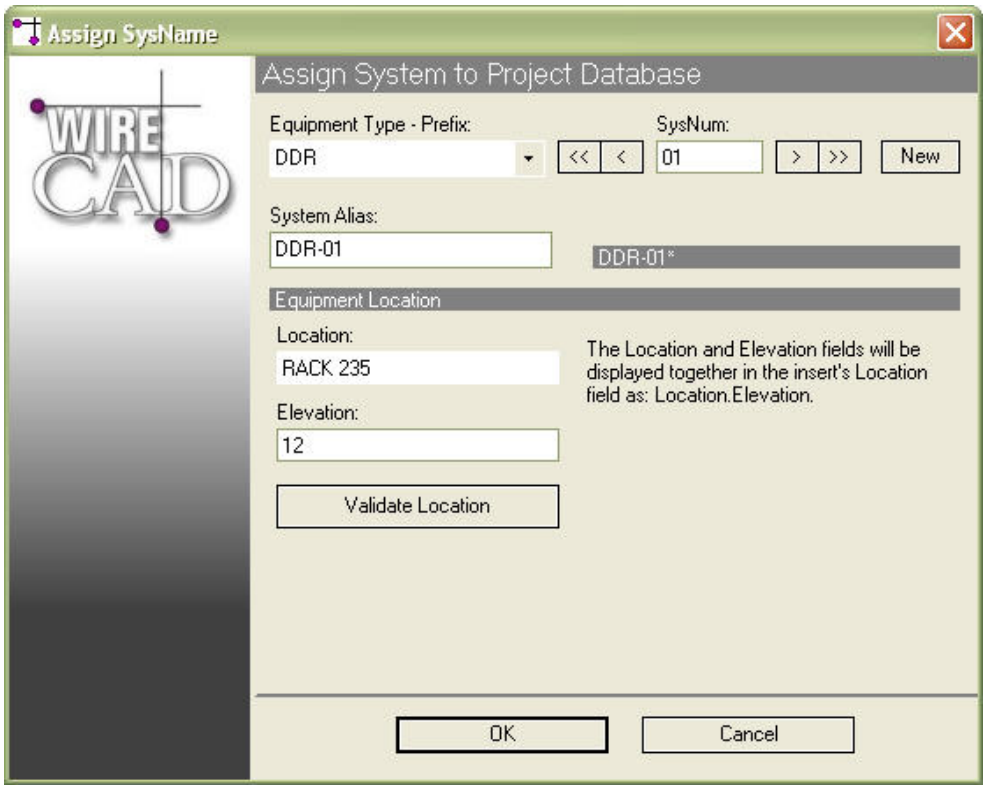
System Alias: Use this field to provide functional or friendly names to systems. Example: suppose you have VTR-450 in your system but the function of this device is Fred's Viewing Deck. Enter "Fred's Viewing Deck" in the alias field to provide an additional description to VTR-450.

Assigning System Names to Individual Terminals

If you have paced WireCAD Terminals in your drawing, these will need to associated with a specific input/output of a specific SysName. For example: suppose we have a jack point in a drawing. That jack point will need to physically occupy a position in a jack-field. The jack-field will have a SysName such as JF-01 and the jack may be the first jack in the top row, say, A-1. The first step in the process is to assign the jack-field to the project. This is done in the Equipment Library  . Find the equipment definition and click the Add This System to Project button 

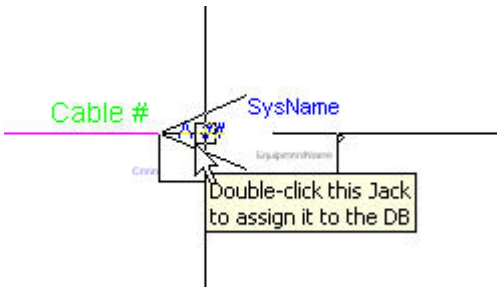
.

This will launch the now familiar Assign Systems dialog mentioned above.

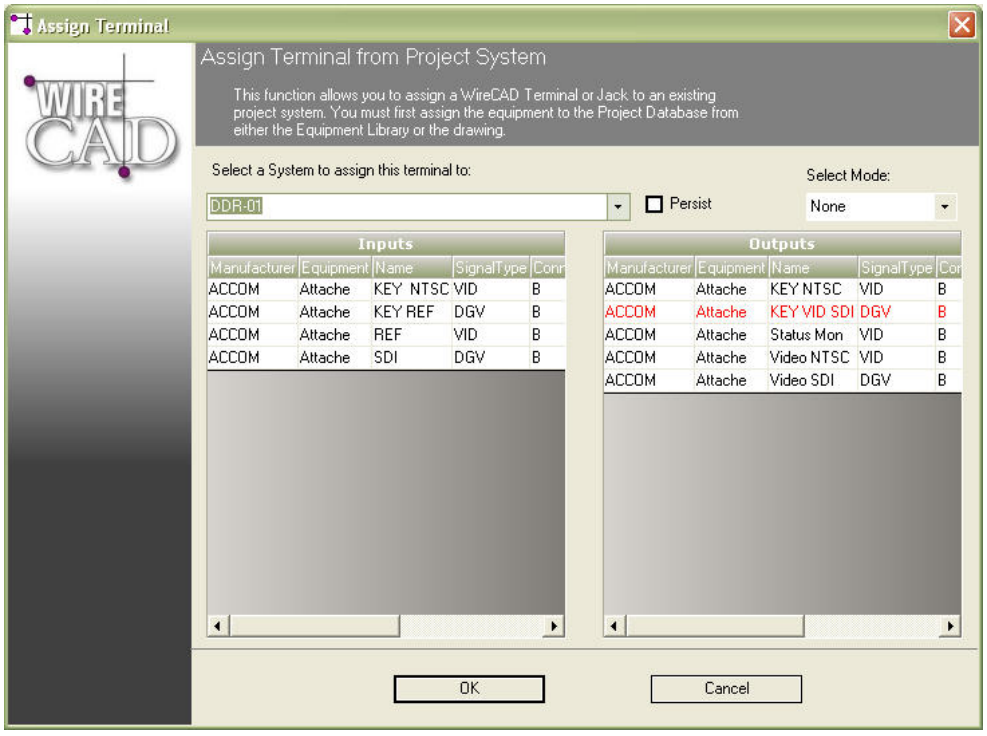


Once you have defined a SysName for your terminal device, you can now assign the individual terminal to the SysName.

To accomplish this, double-click on the terminal in the drawing.:

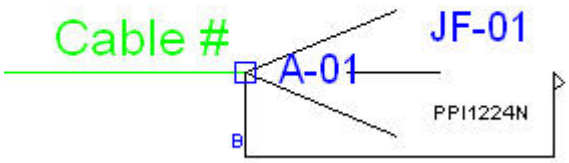


This launches the following dialog where you determine the input/ouput or both that this terminal is to represent:



Select a system from the drop-down. This displays the collection of inputs and outputs for the selected system. Records that are displayed in red are already assigned cable numbers.

Clicking OK will update the information in the drawing:



Assigning Cable Numbers

Once the systems attached to a cable have been given SysNames, Cables can be assigned Cable Numbers.

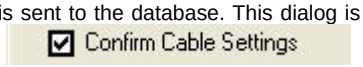
To assign a single cable, double-click on a cable to assign it to the database.

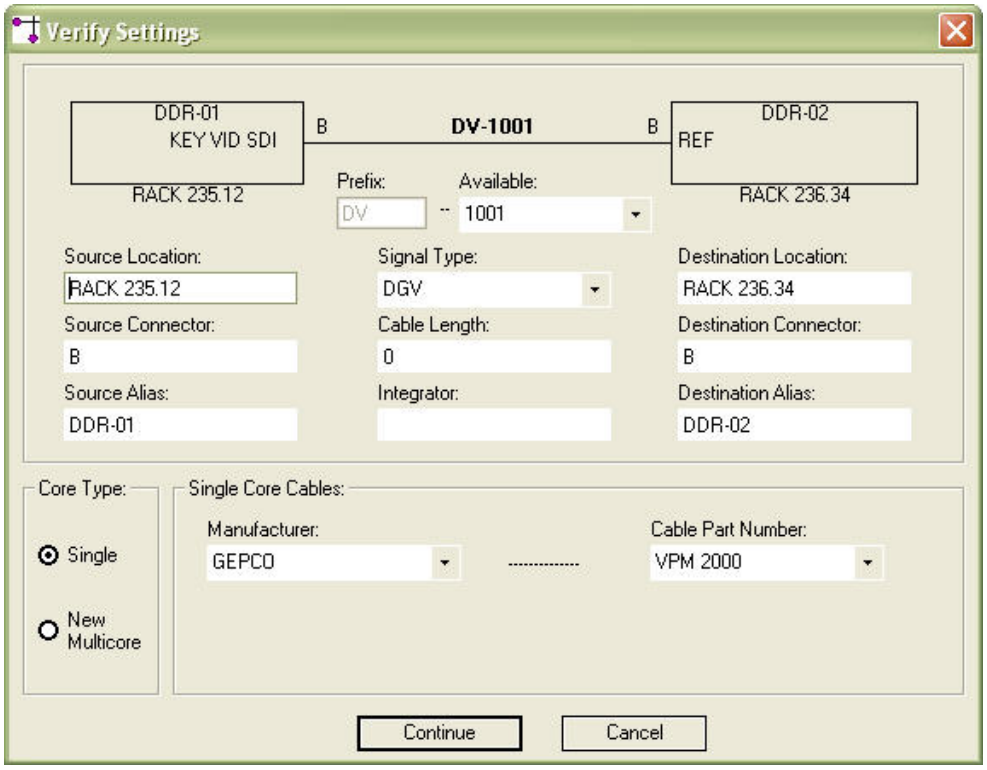
To assign multiple cables at once, use the menu Cables>Assign to Database. Then select the cables to assign. Right click to continue.

The Assign cable numbers function does several things:

- Retrieves cable source and destination information from the drawing.
- Retrieves the Cable Number Prefix to be used from the Signal Types database based on the source signal type.
- Checks the database for existing entries. If found presents a dialog prompting how to proceed.
- If the Verify Settings check box in the Project Preferences>Cables is checked, displays the Verify Settings dialog allowing editing of the information to be sent to the database. This is also where you determine if you want to assign the cable as a multi-core cable.
- Updates the database.
- Updates the drawing.

Verify Cable Settings

The Verify Settings dialog allows you to edit information before it is sent to the database. This dialog is displayed only if the Project Preferences>Cables>Confirm Cable Settings checkbox is checked. 



Prefix: Cable number prefix determined by the Signal Type field.

Available: The next cable number or choose an existing one. This field is determined by the Prefix field, and will retrieve the next number in sequence for the given prefix along with all available cable numbers associated with the Prefix. If Prefix is null, Available provides the next un-prefixed and all available un-prefixed numbers. This field is directly editable in the event that you need to enter cable numbers out of order.

Source/Destination Location: The source and destination locations.

Source/Destination Connector: The source and destination Connectors.

Source/Destination Alias: The source and destination Aliases.

Cable Length: User definable field.

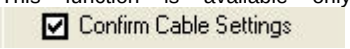
Integrator: User definable field.

Core Type: Single and Multi-core cable types.

Cable Type Manufacturer: The cable type manufacturer.

Cable Type Part Number: The cable type part number. Note that this field displays single core cables when the Core Type Single, and Multi-core cables models when Core Type is New Multi-core.

Multi-core Cable Numbers

This function is available only when Project Preferences>Cables>Confirm Cable Settings checkbox is checked. 

WireCAD supports multi-core cable numbers. WireCAD does this by building a multi-core cable structure based on the cable number prefix, the current number, and the pair data from the selected multi-core cable part number. WireCAD creates an entry in the Project Cables Database for every core in the selected Cable Type following this convention:

CableNoPrefix – Current Number – Pair Data

Example:

Assume a Cable Number Prefix of V the current number is 1001 and a multi-core CableType with three cores. Further, that the Pair Data for the three cores is a simple numeric progression. The resultant Project Cable Database entries would be created:

V-1001-1

V-1001-2

V-1001-3

Note that the CableNoPrefix is actually a separate field and is only concatenated here and in the drawing.

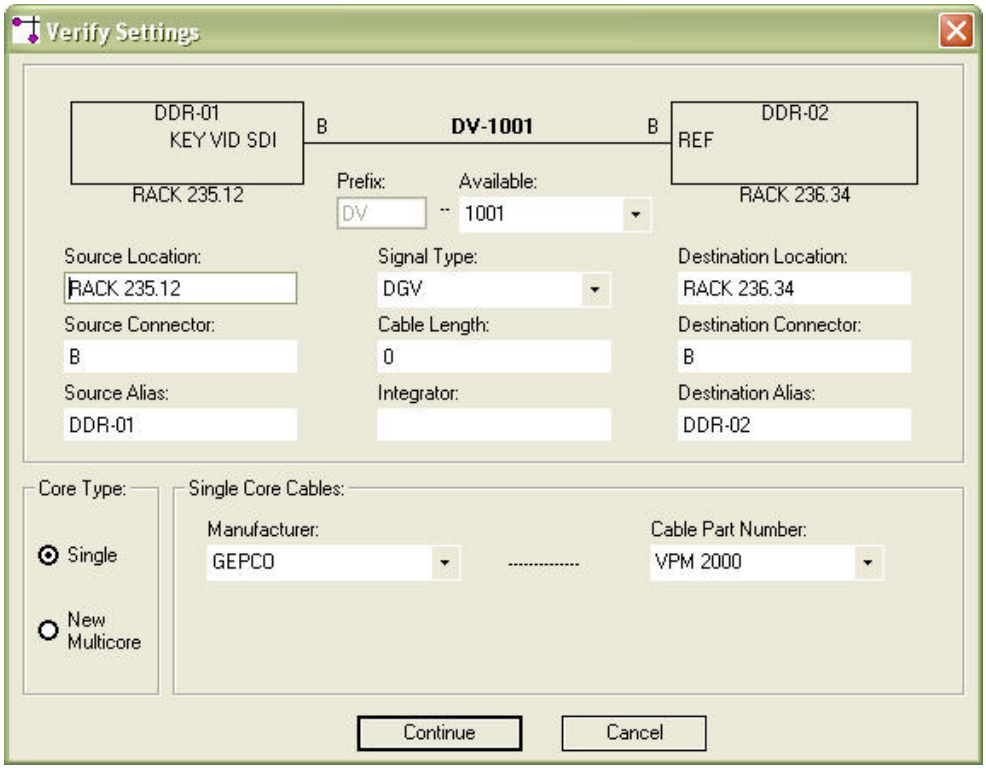
Further, assume that the Pair Data is not a numeric progression but a color code containing the entries Red, Grn, Blu. The resultant Project Cable Database entries would be created:

V-1001-Red

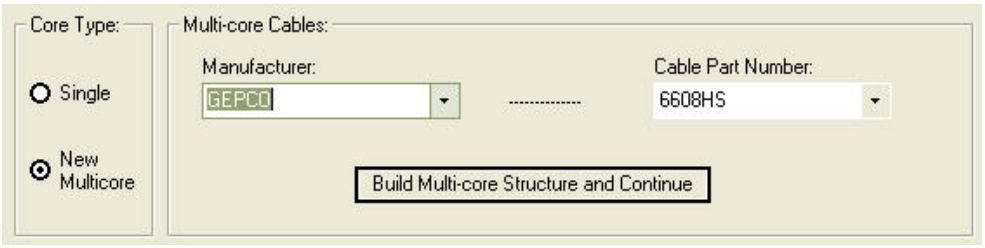
V-1001-Grn

V-1001-Blu

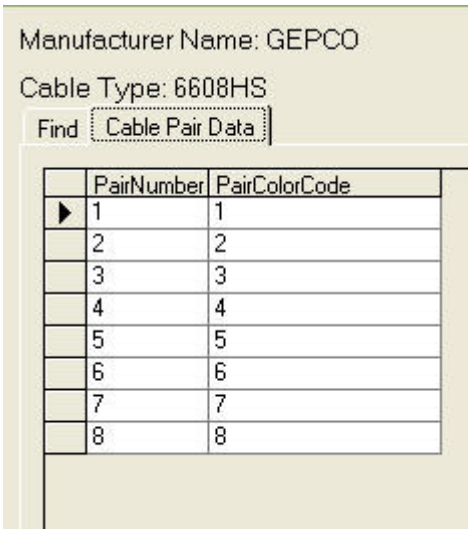
The data from the current assignment is given to the first entry and the remaining entries are marked available and show up in the Available drop down discussed above.



To build a multi-core cable structure, select New Multi-core from the Core Type frame:

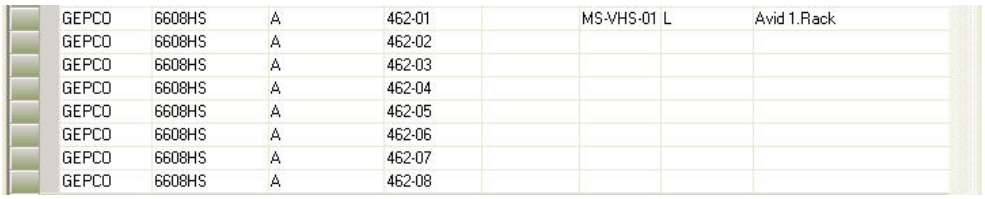


Next, select the Manufacturer and Cable Part Number from which to build the structure. Here we have selected the Gepco 6608 HS that we have defined in our Cable Types library to have the following pair data:



PairNumber	PairColorCode
1	1
2	2
3	3
4	4
5	5
6	6
7	7
8	8

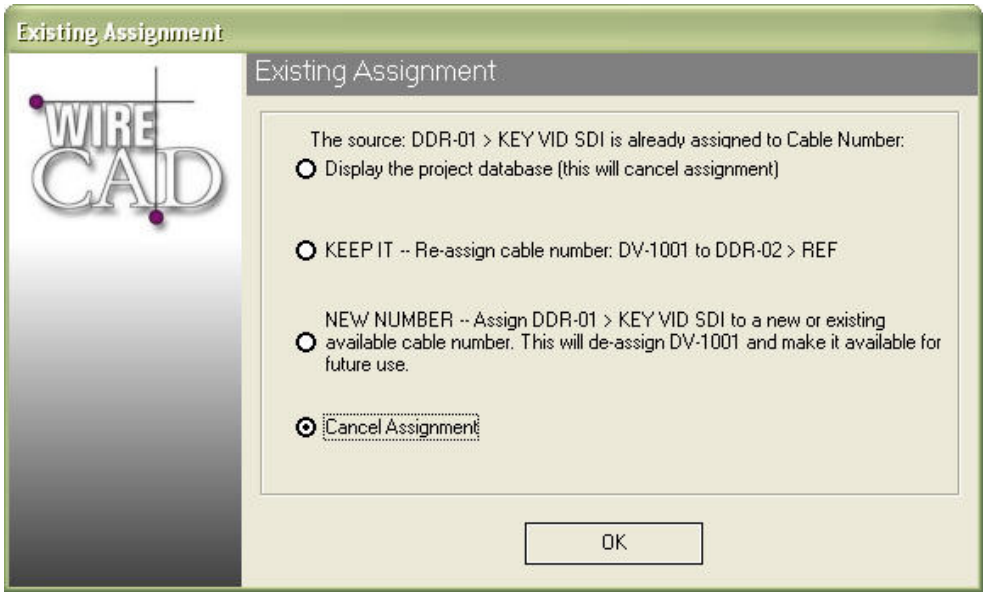
Next click the Build Multi-core Structure and Continue button. The information from the Source and Destination fields is given to the first entry in the structure and the remaining entries are created and marked available.



GEPCO	6608HS	A	462-01		MS-VHS-01 L		Avid 1.Rack
GEPCO	6608HS	A	462-02				
GEPCO	6608HS	A	462-03				
GEPCO	6608HS	A	462-04				
GEPCO	6608HS	A	462-05				
GEPCO	6608HS	A	462-06				
GEPCO	6608HS	A	462-07				
GEPCO	6608HS	A	462-08				

Existing Assignment Dialog

In the event that you are assigning a cable that already exists in the drawing (this can occur for a number of reasons including drawing revisions, etc.). WireCAD will present the following dialog prompting you to take action:



Here you are given a number of options including keeping the same number with the assignment or giving a new number to the assignment. WireCAD then checks the drawing for existing instances of the cable number in question and if found prompts you to keep to delete each instance.

If the Verify Settings checkbox is checked you will then be presented with the Verify Settings dialog. Otherwise, the assignment will proceed.

Global Databases

WireCAD maintains a set of databases that are common to all projects. These databases manage:

- Projects information.
- Manufacturers.
- Equipment.
- Connectors.
- Signal Types.
- Cable Types.
- Jack Fields.
- Industry Sectors.

Enterprise Client Edition Note

The Enterprise Client Edition also exposes a table that displays the permissions of the user.

A Note About WireCAD Datasheets

Some WireCAD datasheets are directly editable while others are presented as read only.

If a table will allow the addition of records it will contain this Add Record field at the top of the sheet:

Some datasheets allow grouping by column. Simply drag the column's header into this box:

The grid will be reconfigured to apply the grouping.

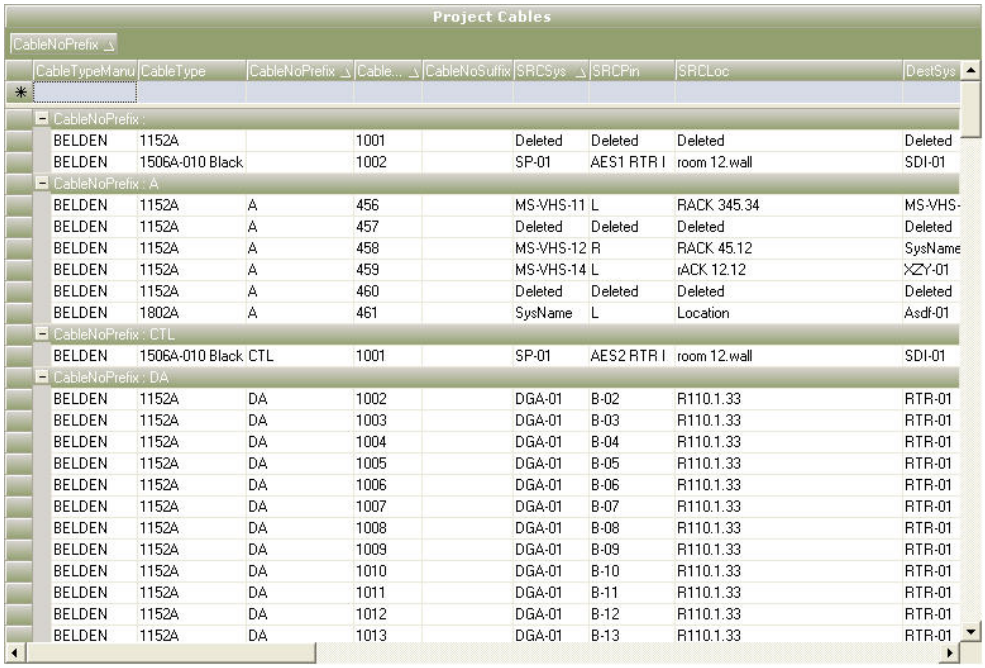
The following grid displays the project cable data without grouping:

Project Cables

Drag a column header here to group by that column

CableTypeManu	CableType	CableNo...	Cable...	CableNoSuffix	SRCSys	SRCPin	SRCLoc	DestSys
BELDEN	1152A		1001		Deleted	Deleted	Deleted	Deleted
BELDEN	1506A-010 Black	CTL	1001		SP-01	AES2 RTR I	room 12.wall	SDI-01
BELDEN	1152A	DV	1001		SDI-01	B-01	R110.1.35	RTR-01
GEPCO	VPM 2000	V	1001		VID-01	B-01	R110.1.29	RTR-01
BELDEN	1506A-010 Black		1002		SP-01	AES1 RTR I	room 12.wall	SDI-01
BELDEN	1152A	DA	1002		DGA-01	B-02	R110.1.33	RTR-01
BELDEN	1152A	DV	1002		SDI-01	B-02	R110.1.35	RTR-01
GEPCO	VPM 2000	V	1002		VID-01	B-02	R110.1.29	RTR-01
BELDEN	1152A	DA	1003		DGA-01	B-03	R110.1.33	RTR-01
BELDEN	1152A	DV	1003		SDI-01	B-03	R110.1.35	RTR-01
GEPCO	VPM 2000	V	1003		VID-01	B-03	R110.1.29	RTR-01
BELDEN	1152A	DA	1004		DGA-01	B-04	R110.1.33	RTR-01
BELDEN	1152A	DV	1004		SDI-01	B-04	R110.1.35	RTR-01
GEPCO	VPM 2000	V	1004		VID-01	B-04	R110.1.29	RTR-01
BELDEN	1152A	DA	1005		DGA-01	B-05	R110.1.33	RTR-01
BELDEN	1152A	DV	1005		SDI-01	B-05	R110.1.35	RTR-01
GEPCO	VPM 2000	V	1005		VID-01	B-05	R110.1.29	RTR-01
BELDEN	1152A	DA	1006		DGA-01	B-06	R110.1.33	RTR-01
BELDEN	1152A	DV	1006		SDI-01	B-06	R110.1.35	RTR-01
GEPCO	VPM 2000	V	1006		VID-01	B-06	R110.1.29	RTR-01
BELDEN	1152A	DA	1007		DGA-01	B-07	R110.1.33	RTR-01
BELDEN	1152A	DV	1007		SDI-01	B-07	R110.1.35	RTR-01
GEPCO	VPM 2000	V	1007		VID-01	B-07	R110.1.29	RTR-01
BELDEN	1152A	DA	1008		DGA-01	B-08	R110.1.33	RTR-01
BELDEN	1152A	DV	1008		SDI-01	B-08	R110.1.35	RTR-01

The following grid displays the project cable data grouped by the CableNoPrefix field:



Project Cables

CableNoPrefix

CableTypeManu	CableType	CableNoPrefix	Cable...	CableNoSuffix	SRCSys	SRCPin	SRCLoc	DestSys
CableNoPrefix :								
BELDEN	1152A		1001		Deleted	Deleted	Deleted	Deleted
BELDEN	1506A-010 Black		1002		SP-01	AES1 RTR I	room 12.wall	SDI-01
CableNoPrefix : A								
BELDEN	1152A	A	456		MS-VHS-11	L	RACK 345.34	MS-VHS-
BELDEN	1152A	A	457		Deleted	Deleted	Deleted	Deleted
BELDEN	1152A	A	458		MS-VHS-12	R	RACK 45.12	SysName
BELDEN	1152A	A	459		MS-VHS-14	L	rACK 12.12	XZY-01
BELDEN	1152A	A	460		Deleted	Deleted	Deleted	Deleted
BELDEN	1802A	A	461		SysName	L	Location	Asdf-01
CableNoPrefix : CTL								
BELDEN	1506A-010 Black	CTL	1001		SP-01	AES2 RTR I	room 12.wall	SDI-01
CableNoPrefix : DA								
BELDEN	1152A	DA	1002		DGA-01	B-02	R110.1.33	RTR-01
BELDEN	1152A	DA	1003		DGA-01	B-03	R110.1.33	RTR-01
BELDEN	1152A	DA	1004		DGA-01	B-04	R110.1.33	RTR-01
BELDEN	1152A	DA	1005		DGA-01	B-05	R110.1.33	RTR-01
BELDEN	1152A	DA	1006		DGA-01	B-06	R110.1.33	RTR-01
BELDEN	1152A	DA	1007		DGA-01	B-07	R110.1.33	RTR-01
BELDEN	1152A	DA	1008		DGA-01	B-08	R110.1.33	RTR-01
BELDEN	1152A	DA	1009		DGA-01	B-09	R110.1.33	RTR-01
BELDEN	1152A	DA	1010		DGA-01	B-10	R110.1.33	RTR-01
BELDEN	1152A	DA	1011		DGA-01	B-11	R110.1.33	RTR-01
BELDEN	1152A	DA	1012		DGA-01	B-12	R110.1.33	RTR-01
BELDEN	1152A	DA	1013		DGA-01	B-13	R110.1.33	RTR-01

Exporting and Printing Grids

WireCAD datasheets can be exported to MS Excel format files using the function Projects>Import/Export>Export to Excel.

Most WireCAD datasheets can be printed using the Projects>Print Grid function. The exception is the Reports Datasheet that displays the data associated with a report that is loaded into the Report generator.

Sorting Data

Clicking the column header with cause the datasheet to sort according to the selected column in ascending order clicking again with sort descending.

The following example displays the column headers for the fields “CableNoPrefix” and “CableNo”. The grid is sorted in ascending order on the “CableNo” field.

A grid may only be sorted on one field unless the grid allows grouping. In which case the grouped field will be sorted as well as an additional field of your choice.

Projects

This database maintains pointers to the existing Project Databases.

A datasheet view of this database is available from the Project Explorer>Global Databases node and from the menu Database>Projects. This database also maintains the “CurrentRevision”. This field can be used to manage cables and data by revision. Anytime a cable number is assigned the value in the “CurrentRevision” field will be placed in the cables database.

Please note that changes to the “CurrentRevision” field will not be reflected in the project until you re-launch WireCAD

Manufacturers

This database maintains Manufacturer names and the Libraries that they are to be displayed in.

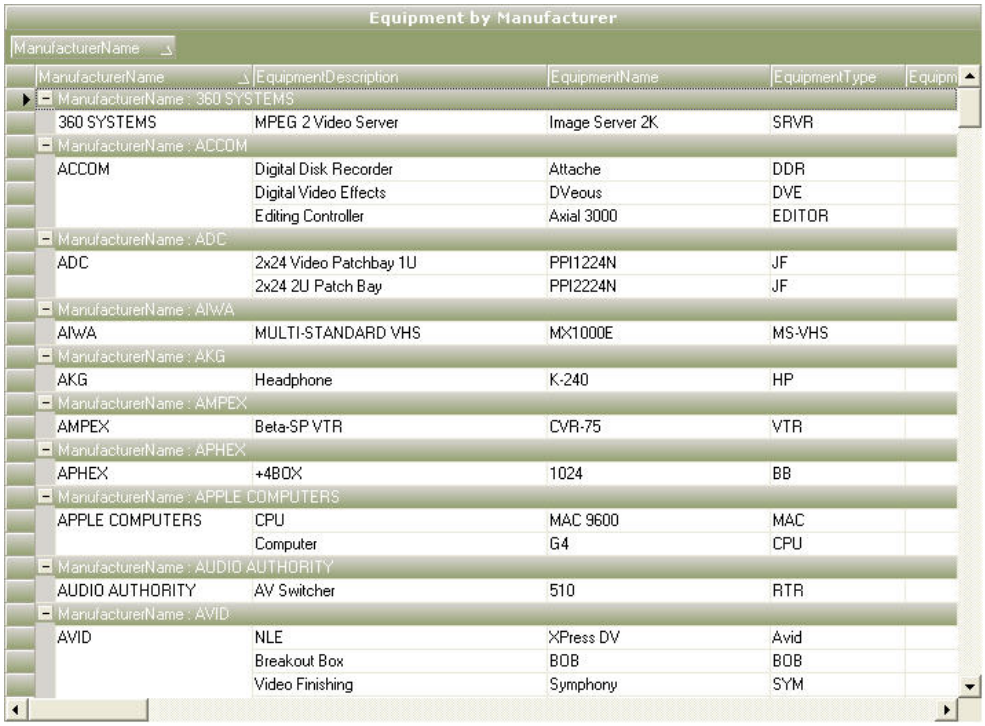
A datasheet view of this database is available from the Project Explorer>Global Databases node and from the menu Databases>Manufacturers.

Equipment Library

The equipment library is comprised of several databases and references the Manufacturers database. Tables included are:

- Equipment
- Manufacturers.

A datasheet view of this database is available from the Project Explorer>Global Databases node and from the menu Databases>Equipment. The grid display of this database is useful for editing equipment data such as the vendor and cost information.



Equipment by Manufacturer

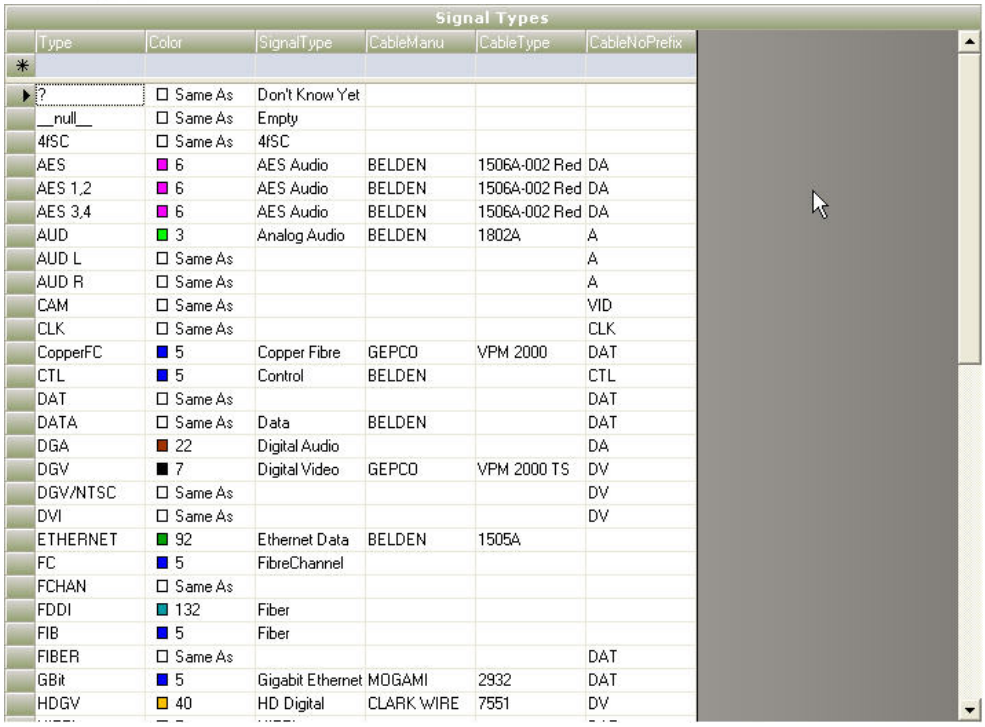
ManufacturerName

ManufacturerName	EquipmentDescription	EquipmentName	EquipmentType	Equipm
ManufacturerName : 360 SYSTEMS				
360 SYSTEMS	MPEG 2 Video Server	Image Server 2K	SRVR	
ManufacturerName : ACCOM				
ACCOM	Digital Disk Recorder	Attache	DDR	
	Digital Video Effects	DVeous	DVE	
	Editing Controller	Axial 3000	EDITOR	
ManufacturerName : ADC				
ADC	2x24 Video Patchbay 1U	PPI1224N	JF	
	2x24 2U Patch Bay	PPI2224N	JF	
ManufacturerName : AIWA				
AIWA	MULTI-STANDARD VHS	MX1000E	MS-VHS	
ManufacturerName : AKG				
AKG	Headphone	K-240	HP	
ManufacturerName : AMPEX				
AMPEX	Beta-SP VTR	CVR-75	VTR	
ManufacturerName : APHEX				
APHEX	+4BOX	1024	BB	
ManufacturerName : APPLE COMPUTERS				
APPLE COMPUTERS	CPU	MAC 9600	MAC	
	Computer	G4	CPU	
ManufacturerName : AUDIO AUTHORITY				
AUDIO AUTHORITY	AV Switcher	510	RTR	
ManufacturerName : AVID				
AVID	NLE	XPress DV	Avid	
	Breakout Box	BOB	BOB	
	Video Finishing	Symphony	SYM	

Note that the global datasheet view will not display Inputs and Outputs.

Signal Types

The Signal Type database determines the behaviour of many WireCAD functions. It maintains information that relates which cable type and cable number prefix to which Signal Type as well as determining how to display a Signal Type in your drawing:



Type	Color	SignalType	CableManu	CableType	CableNoPrefix
?	Same As	Don't Know Yet			
__null__	Same As	Empty			
4fSC	Same As	4fSC			
AES	6	AES Audio	BELDEN	1506A-002 Red	DA
AES 1,2	6	AES Audio	BELDEN	1506A-002 Red	DA
AES 3,4	6	AES Audio	BELDEN	1506A-002 Red	DA
AUD	3	Analog Audio	BELDEN	1802A	A
AUD L	Same As				A
AUD R	Same As				A
CAM	Same As				VID
CLK	Same As				CLK
CopperFC	5	Copper Fibre	GEPCO	VPM 2000	DAT
CTL	5	Control	BELDEN		CTL
DAT	Same As				DAT
DATA	Same As	Data	BELDEN		DAT
DGA	22	Digital Audio			DA
DGV	7	Digital Video	GEPCO	VPM 2000 TS	DV
DGV/NTSC	Same As				DV
DVI	Same As				DV
ETHERNET	92	Ethernet Data	BELDEN	1505A	
FC	5	FibreChannel			
FCHAN	Same As				
FDDI	132	Fiber			
FIB	5	Fiber			
FIBER	Same As				DAT
GBit	5	Gigabit Ethernet	MOGAMI	2932	DAT
HDGV	40	HD Digital	CLARK WIRE	7551	DV

Type: The abbreviated Signal Type used in many Signal Type selection boxes throughout WireCAD.

Color: The Color Number. Range 0-255 and Same as Layer. Same as layer will permit the Signal Layer to determine the color of the entity.

SignalType: Description.

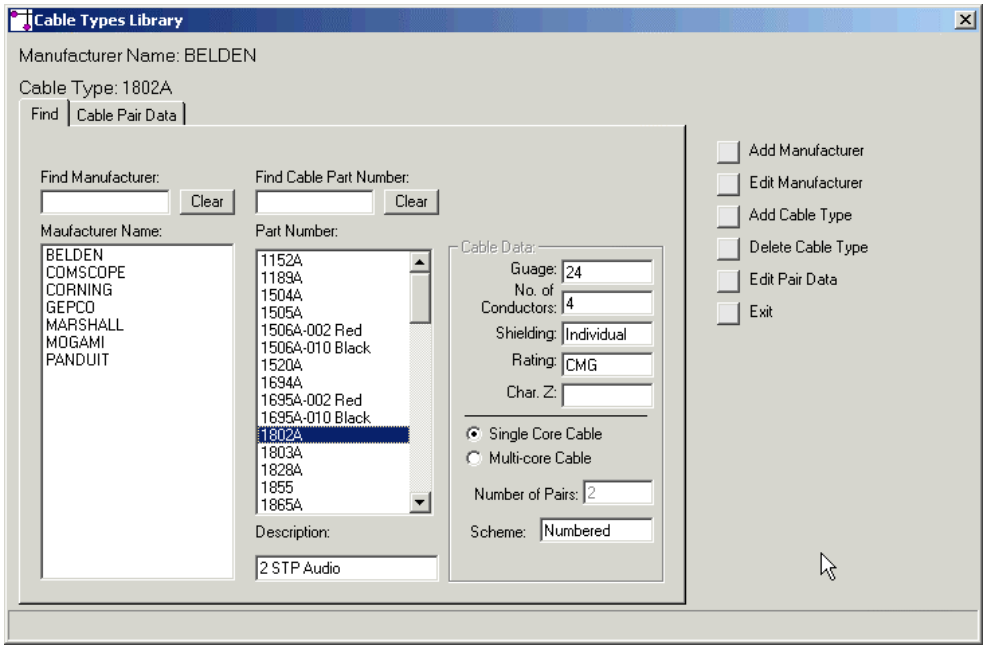
CableManu: Cable Type Manufacturer.

CableType: Cable Type model number.

CableNoPrefix: Cable Number Prefix. Used in the assignment of cable numbers. If this field is blank the prefix will be blank.

Cable Types

Add and edit Cable Types and associated Pair Data. Pair data is used when creating Multi-core cable numbers.



Add Manufacturer: Opens the Add Manufacturers dialog.

Edit Manufacturer: Future.

Add Cable Type: Opens the Add Cable Types dialog.

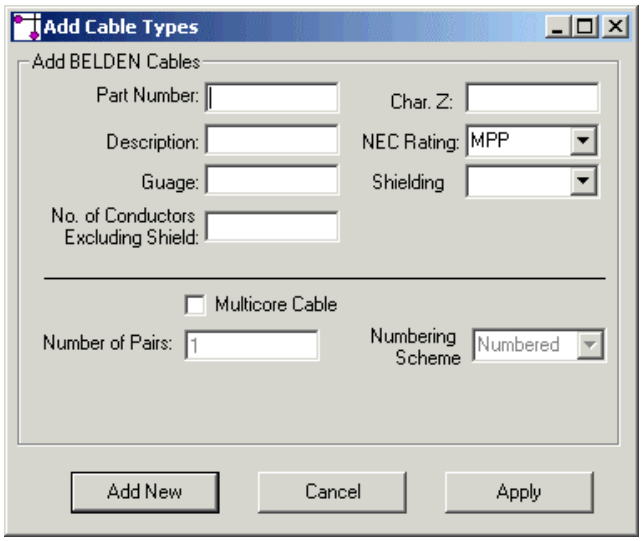
Delete Cable Type: Deletes the highlighted cable type. This cannot be undone.

Edit Pair Data: Same as clicking the Cable Pair Data tab.

Cable Data: Information about the highlighted Cable Type.

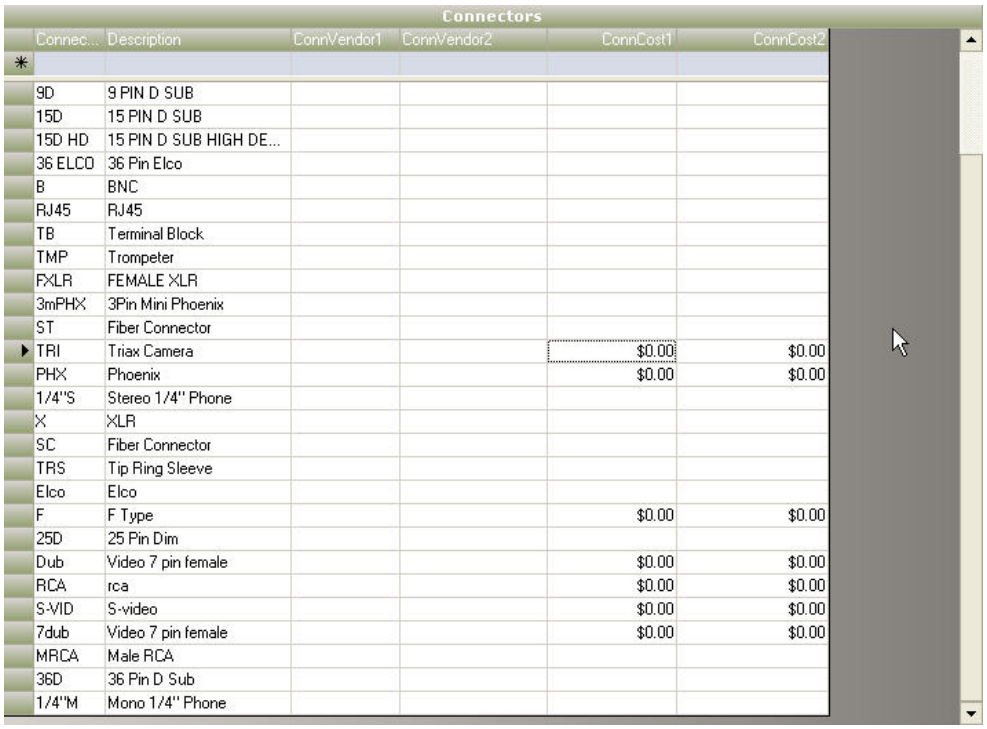
Adding Cable Types

Add a new Cable Type with its associated data. This data is for reference only as it not use elsewhere. The exception is the cable pair data which is used to determine the number of cores to add for a Multi-core cable and determines how they are labelled.



Connectors

The Connectors database maintains information on connectors



Connec...	Description	ConnVendor1	ConnVendor2	ConnCost1	ConnCost2
9D	9 PIN D SUB				
15D	15 PIN D SUB				
15D HD	15 PIN D SUB HIGH DE...				
36 ELCO	36 Pin Elco				
B	BNC				
RJ45	RJ45				
TB	Terminal Block				
TMP	Trompeter				
FXLR	FEMALE XLR				
3mPHX	3Pin Mini Phoenix				
ST	Fiber Connector				
TRI	Triax Camera			$0.00	$0.00
PHX	Phoenix			$0.00	$0.00
1/4"S	Stereo 1/4" Phone				
X	XLR				
SC	Fiber Connector				
TRS	Tip Ring Sleeve				
Elco	Elco				
F	F Type			$0.00	$0.00
25D	25 Pin Dim				
Dub	Video 7 pin female			$0.00	$0.00
RCA	rca			$0.00	$0.00
S-VID	S-video			$0.00	$0.00
7dub	Video 7 pin female			$0.00	$0.00
MRCA	Male RCA				
36D	36 Pin D Sub				
1/4"M	Mono 1/4" Phone				

Connector: The abbreviated connector used in many connector selection boxes throughout WireCAD.

Description: Description.

ConnVendor1: One of two user definable connector vendor fields.

ConnVendor2: The other.

ConnCost1: One of two user definable connector cost fields.

ConnCost2: The other.

Project Databases

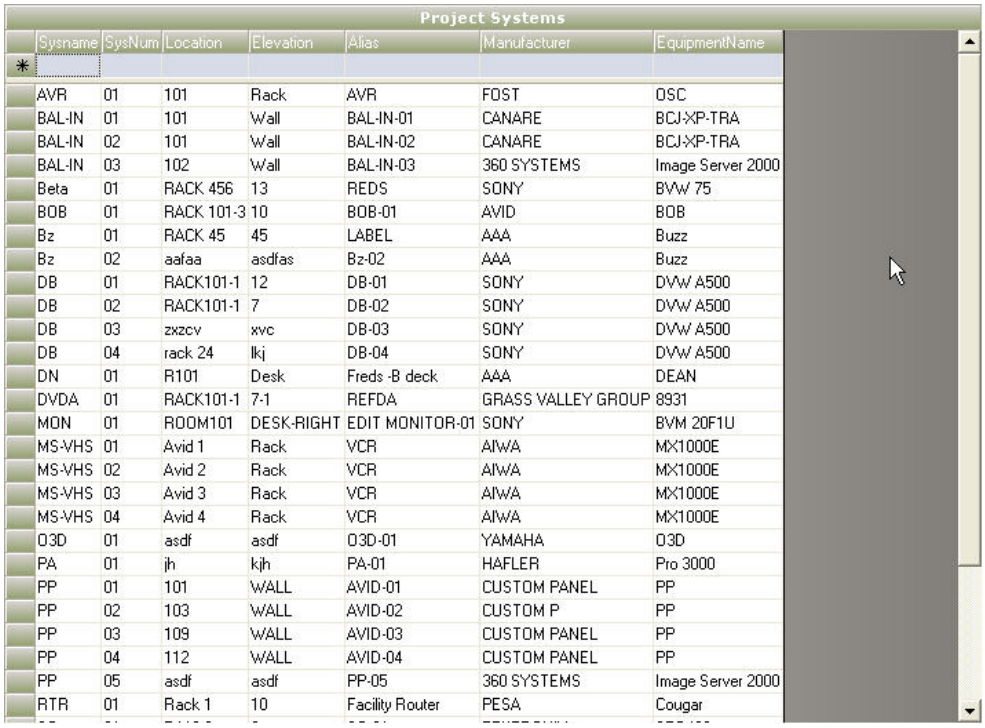
WireCAD maintains a separate project database for each project. This database is located in a folder inside the project folder named “Project Databases”. Project databases contain project specific information like:

- Project Drawings.
- Systems.
- Cables.
- Jack Fields (for v2 compatibility).

Note: WireCAD Personal contains only one project named Default Project contained in the WireCAD directory\WireCAD Default Project\Project Databases\.

Systems

Contains System Name and manufacturer and model information for each System Name in the project.

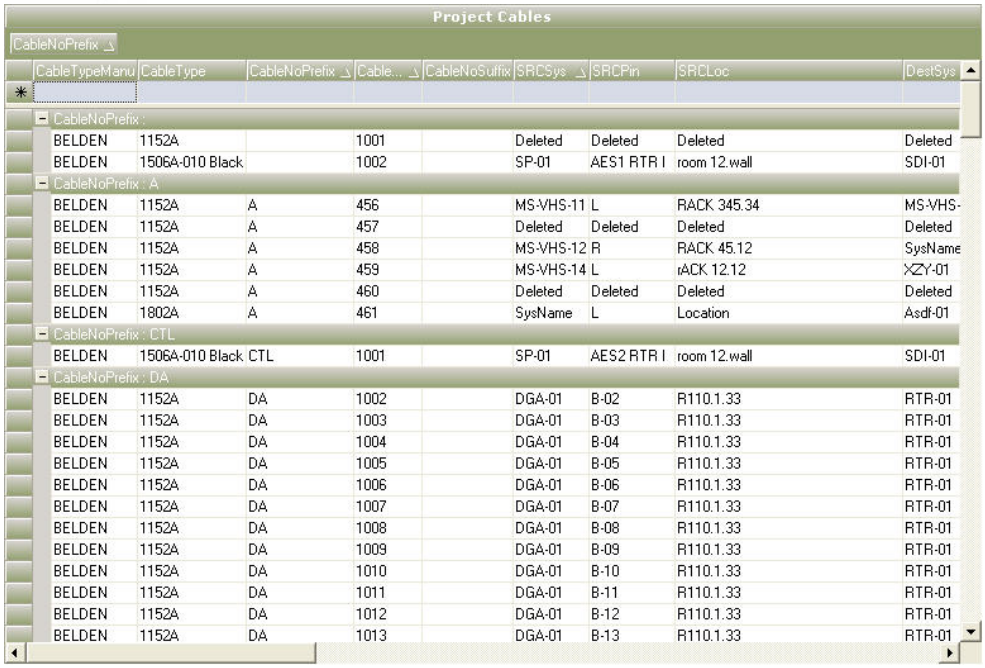


Project Systems

Sysname	SysNum	Location	Elevation	Alias	Manufacturer	EquipmentName
AVR	01	101	Rack	AVR	FOST	OSC
BAL-IN	01	101	Wall	BAL-IN-01	CANARE	BCJ-XP-TRA
BAL-IN	02	101	Wall	BAL-IN-02	CANARE	BCJ-XP-TRA
BAL-IN	03	102	Wall	BAL-IN-03	360 SYSTEMS	Image Server 2000
Beta	01	RACK 456	13	REDS	SONY	BVW 75
BOB	01	RACK 101-3	10	BOB-01	AVID	BOB
Bz	01	RACK 45	45	LABEL	AAA	Buzz
Bz	02	aafaa	asdfas	Bz-02	AAA	Buzz
DB	01	RACK101-1	12	DB-01	SONY	DVW A500
DB	02	RACK101-1	7	DB-02	SONY	DVW A500
DB	03	zxzcv	xvc	DB-03	SONY	DVW A500
DB	04	rack 24	lkj	DB-04	SONY	DVW A500
DN	01	R101	Desk	Freds -B deck	AAA	DEAN
DVDA	01	RACK101-1	7-1	REFDA	GRASS VALLEY GROUP	8931
MON	01	ROOM101	DESK-RIGHT	EDIT MONITOR-01	SONY	BVM 20F1U
MS-VHS	01	Avid 1	Rack	VCR	AIWA	MX1000E
MS-VHS	02	Avid 2	Rack	VCR	AIWA	MX1000E
MS-VHS	03	Avid 3	Rack	VCR	AIWA	MX1000E
MS-VHS	04	Avid 4	Rack	VCR	AIWA	MX1000E
O3D	01	asdf	asdf	O3D-01	YAMAHA	O3D
PA	01	jh	kjh	PA-01	HAFLER	Pro 3000
PP	01	101	WALL	AVID-01	CUSTOM PANEL	PP
PP	02	103	WALL	AVID-02	CUSTOM P	PP
PP	03	109	WALL	AVID-03	CUSTOM PANEL	PP
PP	04	112	WALL	AVID-04	CUSTOM PANEL	PP
PP	05	asdf	asdf	PP-05	360 SYSTEMS	Image Server 2000
RTR	01	Rack 1	10	Facility Router	PESA	Cougar

Cables

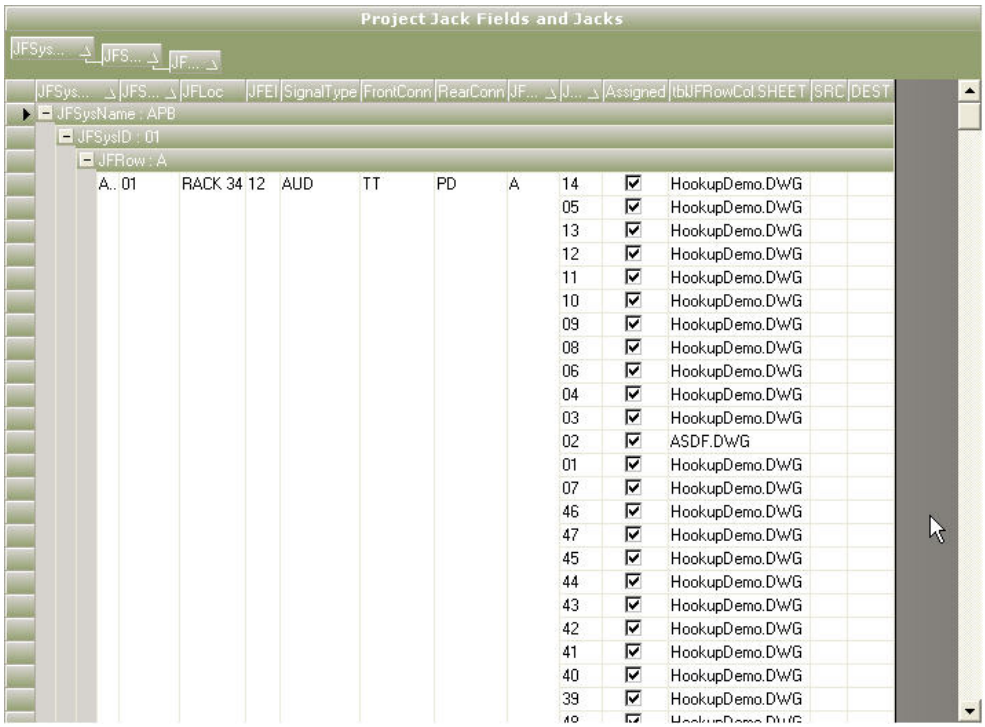
Contains Cable information. Note that WireCAD does not actually delete an existing cable but marks it as available. If cables are deleted from the database (which is possible) you may notice inconsistencies in the new cable number.



Jack Fields

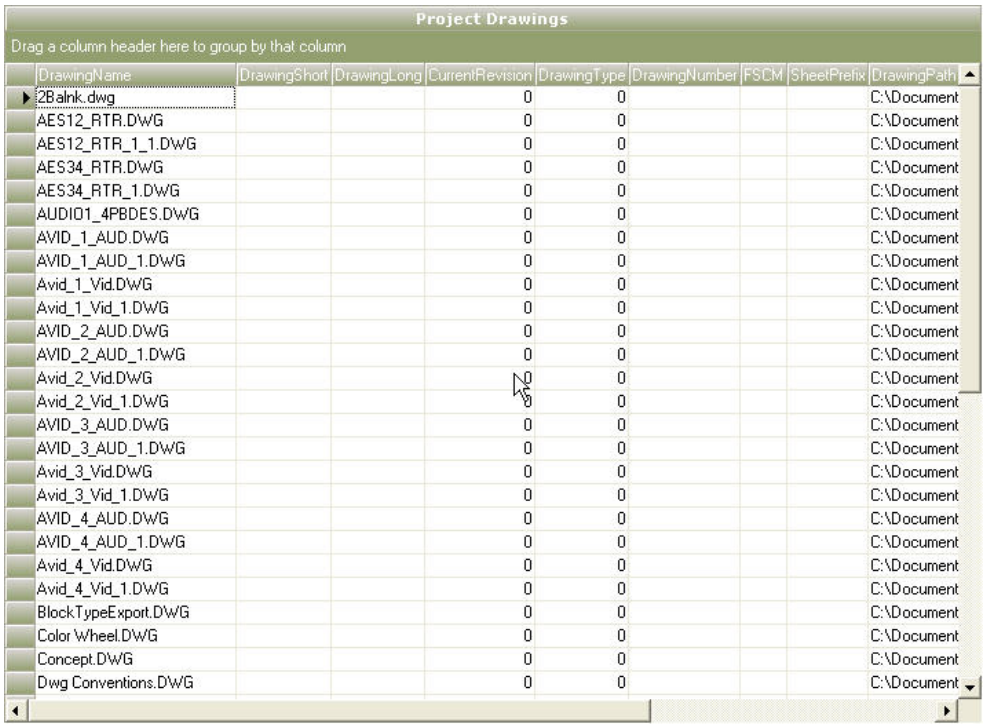
Retained for legacy purposes. WireCAD v3 does not use this table unless you are working with version 2 jacks and jackfields.

Contains Jack Field and Jack information.



Drawings

Displays a list of all drawings that are associated with the current project. Any drawing opened in WireCAD is automatically associated with the project. A number of additional fields can be populated for each drawing. These fields include: Descriptions, Revision number, etc.

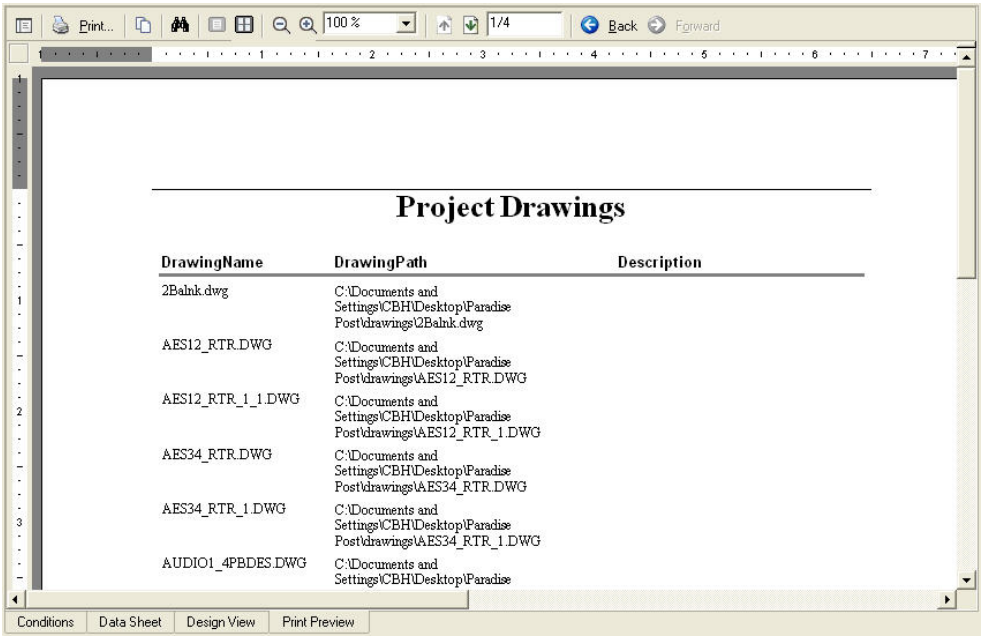


DrawingName	DrawingShort	DrawingLong	CurrentRevision	DrawingType	DrawingNumber	FSCM	SheetPrefix	DrawingPath
2Balnk.dwg			0	0				C:\Document
AES12_RTR.DWG			0	0				C:\Document
AES12_RTR_1_1.DWG			0	0				C:\Document
AES34_RTR.DWG			0	0				C:\Document
AES34_RTR_1.DWG			0	0				C:\Document
AUDIO1_4PBDES.DWG			0	0				C:\Document
AVID_1_AUD.DWG			0	0				C:\Document
AVID_1_AUD_1.DWG			0	0				C:\Document
Avid_1_Vid.DWG			0	0				C:\Document
Avid_1_Vid_1.DWG			0	0				C:\Document
AVID_2_AUD.DWG			0	0				C:\Document
AVID_2_AUD_1.DWG			0	0				C:\Document
Avid_2_Vid.DWG			0	0				C:\Document
Avid_2_Vid_1.DWG			0	0				C:\Document
AVID_3_AUD.DWG			0	0				C:\Document
AVID_3_AUD_1.DWG			0	0				C:\Document
Avid_3_Vid.DWG			0	0				C:\Document
Avid_3_Vid_1.DWG			0	0				C:\Document
AVID_4_AUD.DWG			0	0				C:\Document
AVID_4_AUD_1.DWG			0	0				C:\Document
Avid_4_Vid.DWG			0	0				C:\Document
Avid_4_Vid_1.DWG			0	0				C:\Document
BlockTypeExport.DWG			0	0				C:\Document
Color Wheel.DWG			0	0				C:\Document
Concept.DWG			0	0				C:\Document
Dwg Conventions.DWG			0	0				C:\Document

The Reporting Environment

WireCAD allows database reporting using the Reports form. This form contains several sections that are exposed by clicking the tabs at the bottom of the screen:

- **Preview Tab:** selecting this tab displays the current report in print preview mode.
- **Design Tab:** displays the report designer.
- **Data Sheet Tab:** displays the data associated with the report. This datasheet may or may not be editable depending on the complexity of the retrieved data.
- **Conditions Tab:** allows the application of complex conditions to the data.



Opening and Previewing Reports

With the Reports form active; reports can be opened from the Standard Toolbar, File Menu, and directly from the Project Explorer. The initial view for an opened report is Print Preview. Use the standard view selection buttons to determine display current page, page lay-up, and zoom factor.

Page Setup

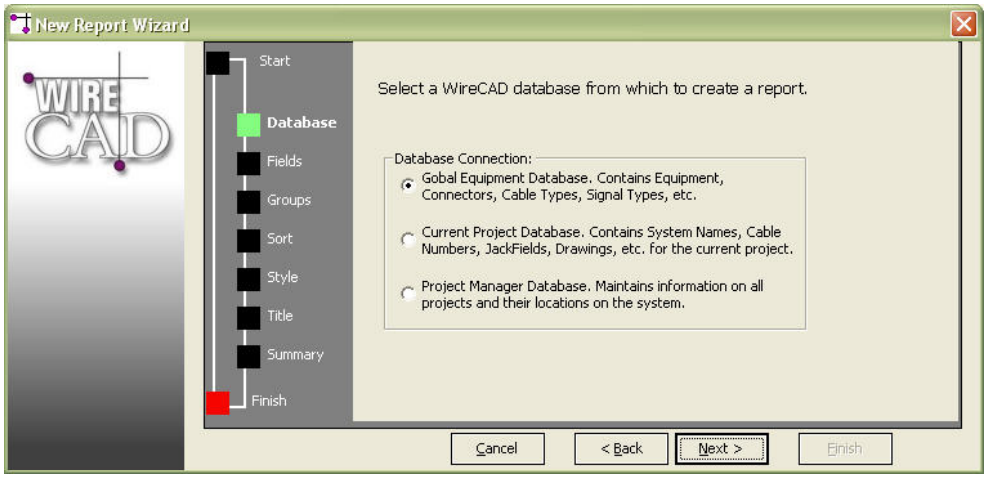
Displays a standard Window Page Setup dialog.

Print Reports  

Print the report to the active Windows printer.

The New Report Wizard

Clicking New Report in the Project Explorer will present the New Report Wizard. This will lead you through the steps of creating a new report. You are given opportunity to select the data from which to create the report, define relationships, sorting, and appearance.



Label Reports

The Report Wizard does not currently support the creation of Labels. The best way to create a new label definition is to modify an existing one. Do this by opening an existing definition and save it as a new name. Next, modify it to accommodate your needs.

The Report Designer

The Report Designer provides a means by which to create and edit report definition files (*.rpx). The WireCAD Report Wizard creates the necessary controls to retrieve the data from the database. This is the ADO control that you see on the report and is named DC1.

Report Sections

The Report Designer permits the grouping of controls by section. Sections can include Header, Detail, Footer. WireCAD also provides additional grouped data sections. These can be added automatically using the New Report Wizard, or by right clicking in the designer and choosing Insert>Group or Insert>Report Header/Footer. The designer must have at least one section named Detail.

Section	Appears	Typically Contains
Report header	Once per report	The report title and summary information for the whole report.
Page header	Once per page	Labels that describe detail fields, and/or page numbers.
Group header	Once per group	Fields that identify the current group, and possibly aggregate values for the group (e.g. total, percentage of the grand total).
Detail	Once per record	Fields containing data from the Source recordset.
Group footer	Once per group	Aggregate values for the group.
Page footer	Once per page	Page number, page count, date printed, report name.
Report footer	Once per report	Summary information for the whole report.

Tools

The Report Designer provides a set of tools for placing Labels, Fields, Shapes, Images, etc. on the layout. Click the tool in the toolbar or menu that you wish to use and Place the item in the appropriate report section. Available tools are:

- Label: Aa used to place unbound text.
- Field: ab| bound/calculated text. Bound Fields can be place from the Field List toolbar. Simply drag the field onto the section and drop. The field will then be bound.
- Checkbox: bound Boolean field.
- Image: bound/unbound image.
- Line: unbound line.
- Shape: unbound shape.
- Frame: unbound frame.
- Sub-report: sub-report use the properties window to bind to appropriate data.
- Page Break: insert a page break.
- OLE container: OLE place an OLE bound or linked object such as a word document etc.
- Barcode: bound barcode field.

Modify

Use tools on the Modify toolbar to change the appearance of the selected item(s).

Align

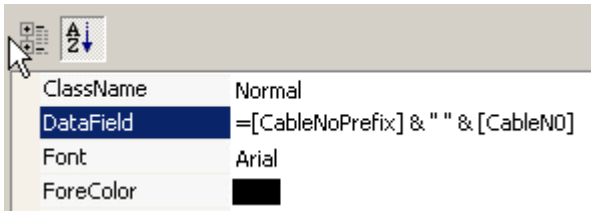
Use tools on the Align toolbar to change the Alignment of the selected item(s).

Calculated Fields

An unbound field can be used to calculate values and concatenate strings. Follow these conventions to calculate fields.

Example:

If we want to create a composite Cable Number using the fields CableNoPrefix and CableNo. We would enter the following data in the DataField property of the selected field. This is found in the Report Properties window.



This will place a space between the two fields that we are concatenating together. The result would look something like: V 1001.

Mathematical operations can be performed as well if the fields in question are numeric. Simply follow the = sign by whatever operators you wish to apply to which fields. The Sum and Count functions are not directly available to calculate fields but are available by assigning the correct values to the SummaryField property. See section below.

Note: field names are not case sensitive.

Summary Fields

Summary fields are fields that provide summary information. A field must first be assigned a DataField to bind to. Next, set the following properties to create the desired behaviour.

- SummaryDistinctField: property sets or returns the name of the field used in a distinct summary function. The summary function will process DataField values based on the distinct value of this field.

  Note: This property is used only when the SummaryFunc value is one of Distinct Summary Functions. When using the summary functions with a field, the CanGrow and CanShrink properties are disabled for the field.

- SummaryFunc: Sets the type of the summary function used to process the DataField values. You can use this function to create sub totals, grand totals and other summary values.

Summary Functions:

Mnemonic	Description
ddSFSum	Calculates the total of all values within the specified summary region (group, page report).
ddSFAvg	Calculates the average of all values within the specified summary region (group, page or report).
ddSFCount	Calculates the count of all values within the specified summary region (group, page or report).
ddSFMin	Calculates the minimum of all values within the specified summary region (group, page or report).
ddSFMax	Calculates the maximum of all values within the specified summary region (group, page or report).
ddSFVar	Calculates the variance of all values within the specified summary region (group, page or report).
ddSFVarP	Calculates the population variance of all values within the specified summary region (group, page or report).
ddSFStdDev	Calculates the standard deviation of all values within the specified summary region (group, page or report).
ddSFStdDevP	Calculates the population standard deviation of all values within the specified summary region (group, page or report).
ddSFDSum	Calculates the total based on the distinct values of another field within the specified summary region (group, page or report).
ddSFDAvg	Calculates the average based on the distinct values of another field within the specified summary region (group, page or report).
ddSFDCount	Calculates the distinct count based on the distinct values of another field within the specified summary region (group, page or report).
ddSFDVar	Calculates the variance based on the distinct values of another field within the specified summary region (group, page or report).
ddSFDVarP	Calculates the population distinct variance based on the distinct values of another field within the specified summary region (group, page or report).
ddSFDStdDev	Calculates the standard deviation based on the distinct values of another field within the specified summary region (group, page or report).
dSFDStdDevP	Calculates the population standard deviation based on the distinct values of another field within the specified summary region (group, page or report).

- SummaryGroup: property sets or returns the name of the group header section that will reset the summarized field value. For example, setting a sum of price for an order group header, will reset the sum to zero for each order group. This property is valid when the SummaryType is set to 3-SubTotal.

Note: When using the summary functions with a field, the CanGrow and CanShrink properties are disabled for the field.

- SummaryRunning: determines whether the summarization will be accumulated or reset for each level (detail, group or page). Setting this property ddSRGroup or ddSRAll will make WireCAD print a running summary of the field at the group or report level.

  Note: When using the summary functions with a field, the CanGrow and CanShrink properties are disabled for the field.
- SummaryType determines the type of summarization on the field if any. WireCAD can summarize the field as:
  - Sub total (group level; reset for each group).
  - Grand total (report level; do not reset until all records are processed).
  - Page total (page level; reset for each page).
  - Or a page count, which is the total number of pages printed.

Summary Types:

Mnemonic	Description
ddSMNone	No summarization.
ddSMGrandTotal	Specifies a report level summary, evaluates the summary function for all records in the report.
ddSMPageTotal	Specifies a page level summary, evaluates the summary function for all records on each page.
ddSMSubTotal	Specifies a group level summary, evaluates the summary function for all records in each group level.
ddSMPageCount	Specifies a Page Count field.

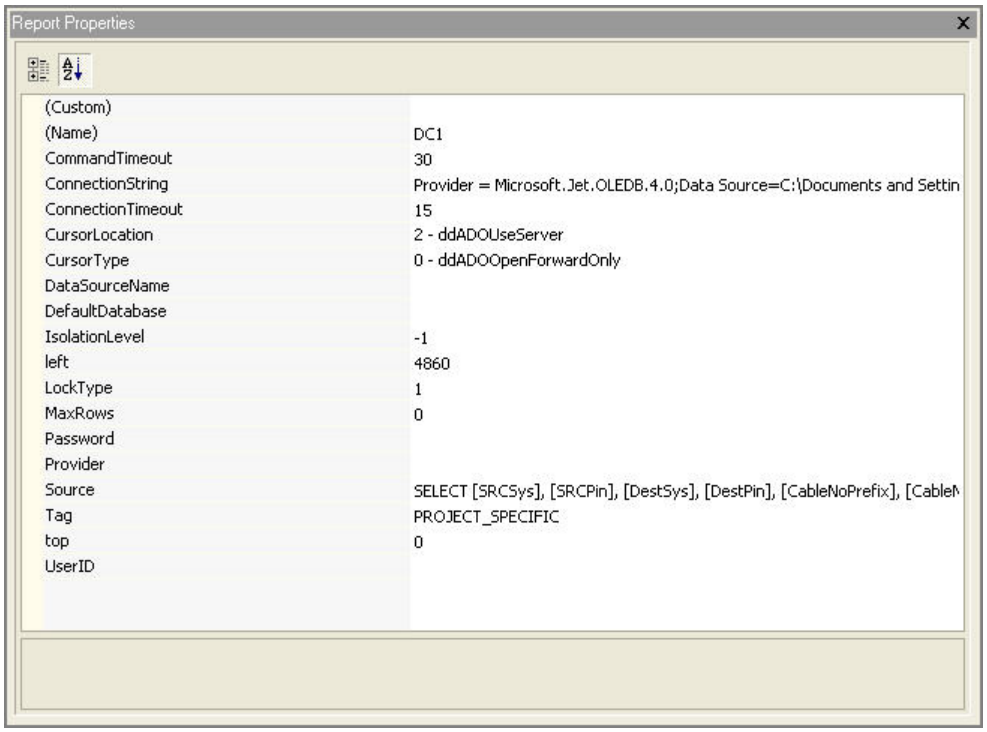
The ADO Control Explained

Every WireCAD report contains one and only one ADO control 

. This control is responsible for retrieving data from the database. WireCAD can only assign this control to one of its databases at a time. If this control is deleted from the report layout it will not provide data, nor will it be able to fill in the Fields List Toolbar.

ADO Properties

This is an expanded view of the ADO control Properties window. Listed below are some of the key functions others are self-explanatory, explained in the description field when the property is highlighted, or non-essential to report creation.



Custom: Presents another version of the property window.

Name: Do not change this from DC1.

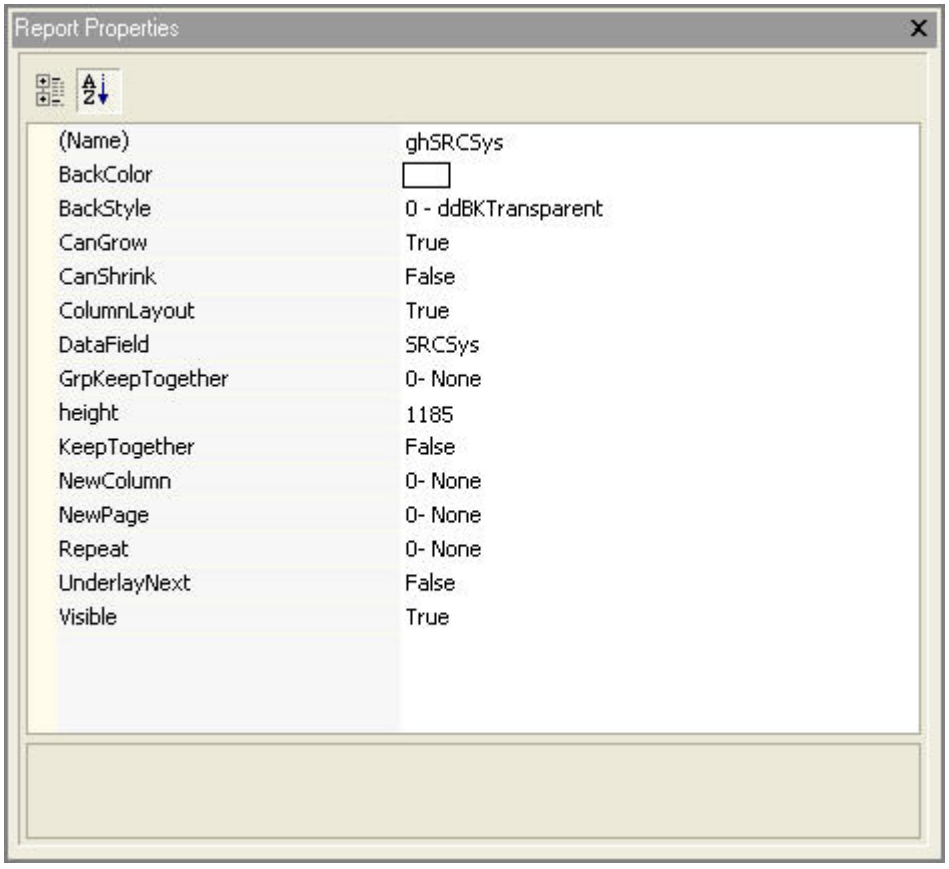
ConnectionString: This is the type, and location of the WireCAD database that we are retrieving data from. ***Do not edit this***.

Source: A SQL select statement used to determine which fields from which tables related to which other tables and in which order to display. Whew! That's a mouthful. While it is beyond the scope of this manual to provide a lesson in SQL suffice it to say that the statement "Select * from [tblCables] ORDER BY [CableNo]" will retrieve all records from the cables table and sort them by cable number. For more information view some of the existing reports.

Tag: Used within WireCAD to identify the data source. ***DO NOT EDIT THIS!***

The Properties Window

Dock-able window used to edit properties associated with the Report Designer. For more information on a property – highlight the property in question and read the description from the description field below.



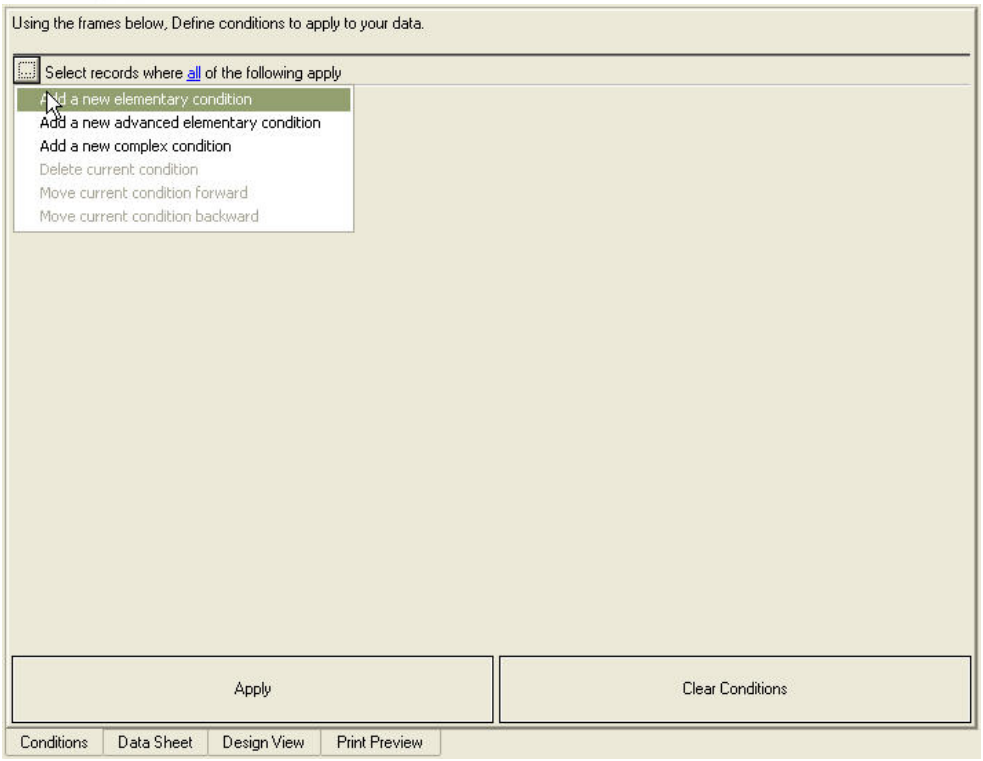
Some Notes About Cable Labels

The fastest way to customize a new cable label is to open an existing one and use the SaveAs function to create a new report file with a different name. See the section on Calculated Fields for information on creating concatenated fields.

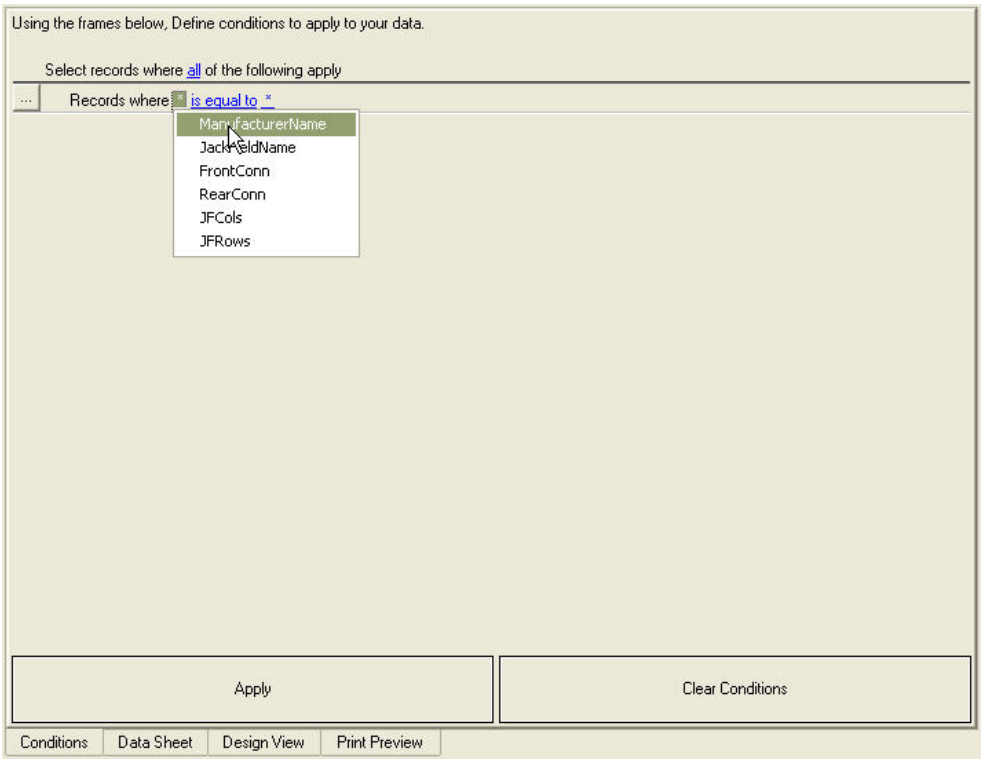
Applying Report Conditions Tutorial

WireCAD provides powerful report Condition generator that allows you to apply complex conditions to you data. Selecting the Condition tab at the bottom of the Reports form displays this form:

Next click on the button with the ellipsis (…) and select Add a new elementary condition.



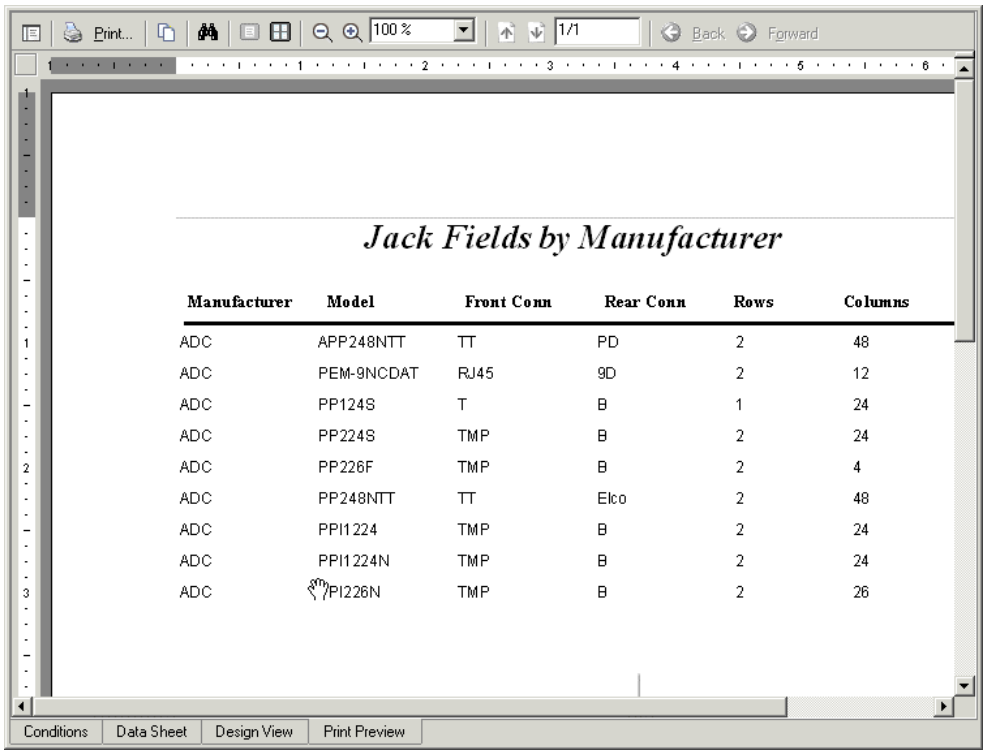
Next, click on the first asterisk to display a list of the fields associated with the report. Select ManufacturerName.



Here we can determine a number of equality operators including <, >, =, Not =, Contains, Starts With, etc. We will leave the default Is Equal To. This will create a condition statement where ManufacturerName is equal to ???

Next, click on the second asterisk to display an edit box and enter ADC in the field:

Next, click the Apply button to apply the conditions and switch to Print Preview Mode.



Thus creating a report limited to the Jack Field Manufacturer ADC.

The Conditions generator is very powerful and can apply multiple conditions. This is useful if you need to print a range of cable numbers or even a range of cable numbers that apply to a given source or destination. You can use it to print equipment pertaining to a certain location.

To clear existing conditions, click the Conditions tab and then click the Clear Conditions button.

How To

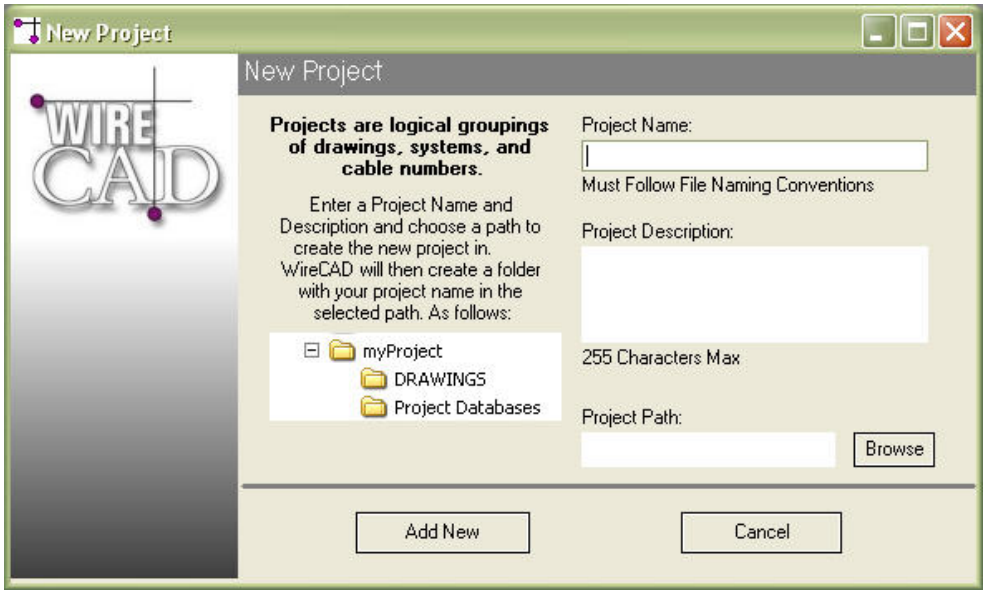
This section provides details on how to perform several basic WireCAD functions. Some of these descriptions appear elsewhere in the text.

How to Create a New Project

Creating a new project in WireCAD creates a new folder with the project name in the directory of your choice and places two additional folders in that folder. As follows:

This process simultaneously creates an entry in the projects database.

Start the function by clicking the Project\New Project menu.  The New Project Dialog will appear:



- Enter a Name for the new project. This name should follow file naming conventions. Do not use more than one period and no ‘ characters.
- Enter a description for the Project. Description are anything your want to use to identify the project.
- Click the 

button and browse to the location on the local drive or network share where you want to store the Project folder.
- Click the Add New button.

Done.

How to Create a New Drawing

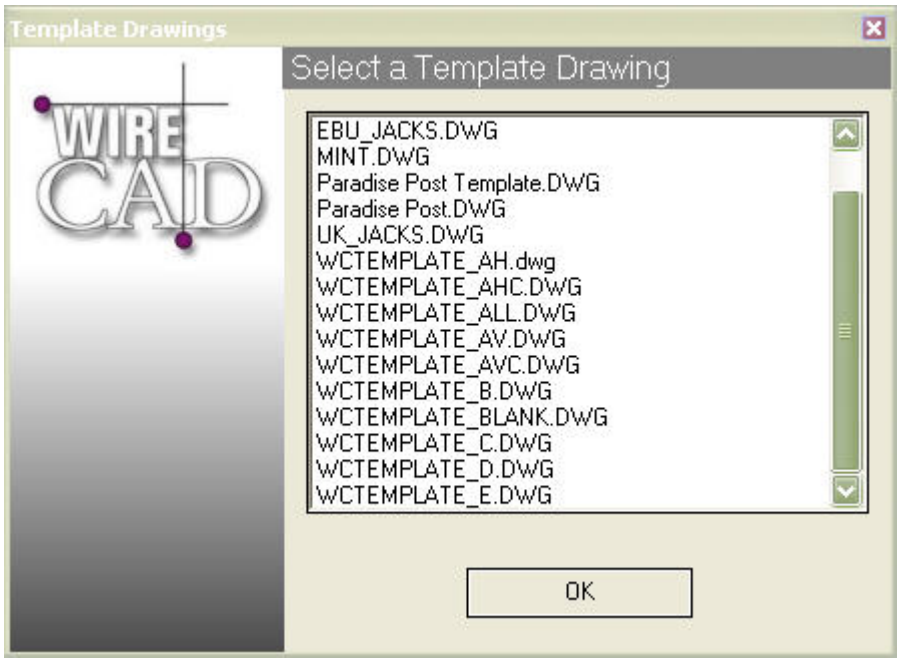
Double-click this icon 

on the Project Explorer or  on the Standard toolbar. You will be prompted to select a WireCAD template drawing.

Template Drawings

When you create a new drawing in WireCAD you are prompted to select from a number of template drawings. These drawings are prepared to receive input from WireCAD. The template drawings also provide a default layout for the ANSI drawing size indicated. To create your own templates, modify an existing template and save the drawing in the WireCAD3\TemplateDrawings folder.

NOTE: You can customize a template drawing with all of your default company and project information. Then save the drawing into the \WireCAD3\TemplateDrawings folder and it will be available from this view. Now you will not have to enter redundant information and can insure symmetry between drawings.

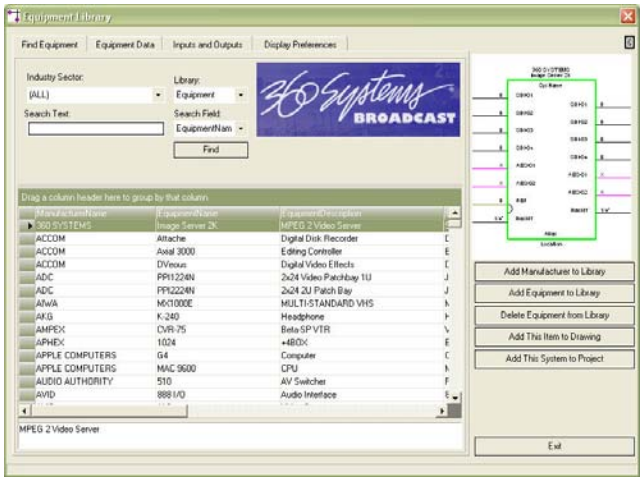


- Click OK
- You will be prompted to name the drawing. Name your drawings something other than drawing##. If WireCAD sees a drawing name that starts with the string “drawing”, you will be prompted every time to save as.

Done.

How to Add Equipment to the Drawing

Start this function by clicking Tools>Equipment>Equipment Library, or the  button. This will display the Equipment Library:



- Select the Search Field upon which to search.
- Type the search text into the Search Text field and Click the Find button.
- Matching records will be displayed in the grid.
- Select the desired Equipment definition. All of the inputs and outputs are automatically selected for inclusion in the block.
- Switch to the Inputs and Outputs tab.
- Modify your selection to suit. Use the Clear Selections button.
- Click on the record selector  to add an Input or Output. If the preview display is not immediately updated, click on the Update Preview button. 

- Once you have selected the desired inputs and outputs, click on the Display Preferences tab.
- Select the desired display style.
- Click on the Add This Item to Drawing Button. 

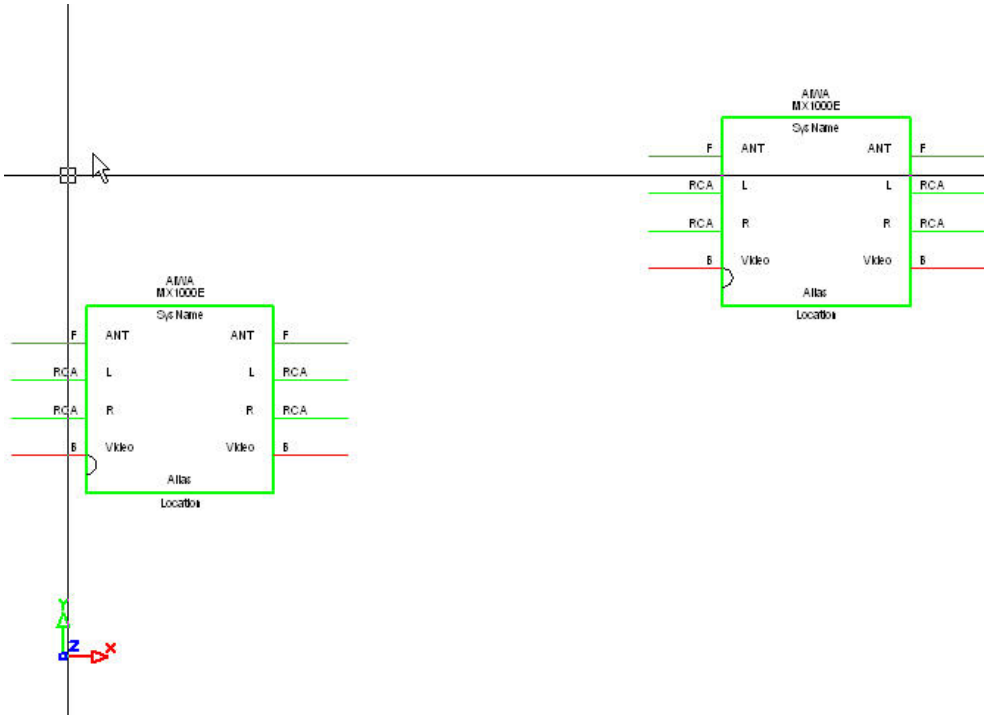
- The Equipment Library will close and you will be prompted to place the new equipment in the drawing.

Done.

How to Draw Cables

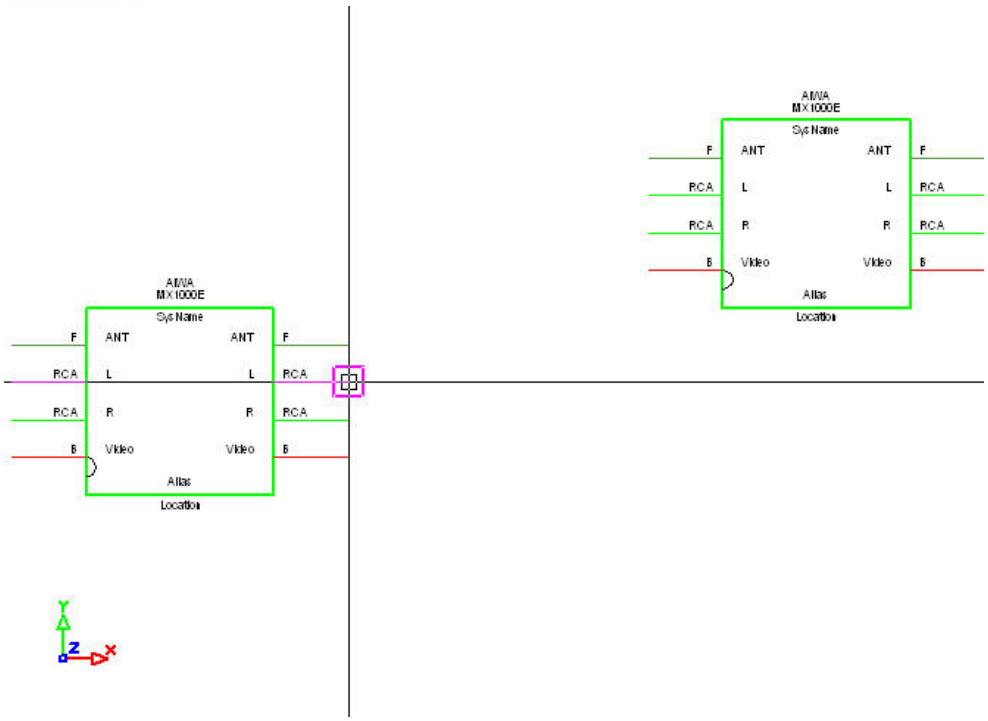
WireCAD drawing conventions dictate that cables are to be drawn from output to input. What follows is a detailed explanation of drawing cables.

- Start with a piece of equipment in the drawing:



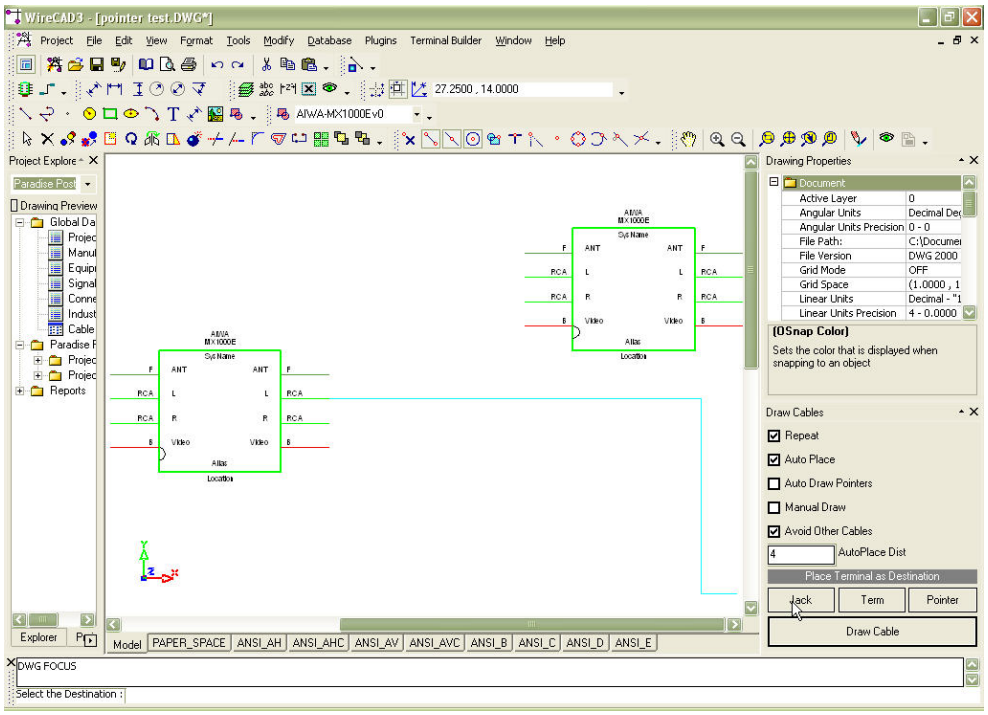
- Select the Draw Cables button: 

- Select an output on the equipment block:

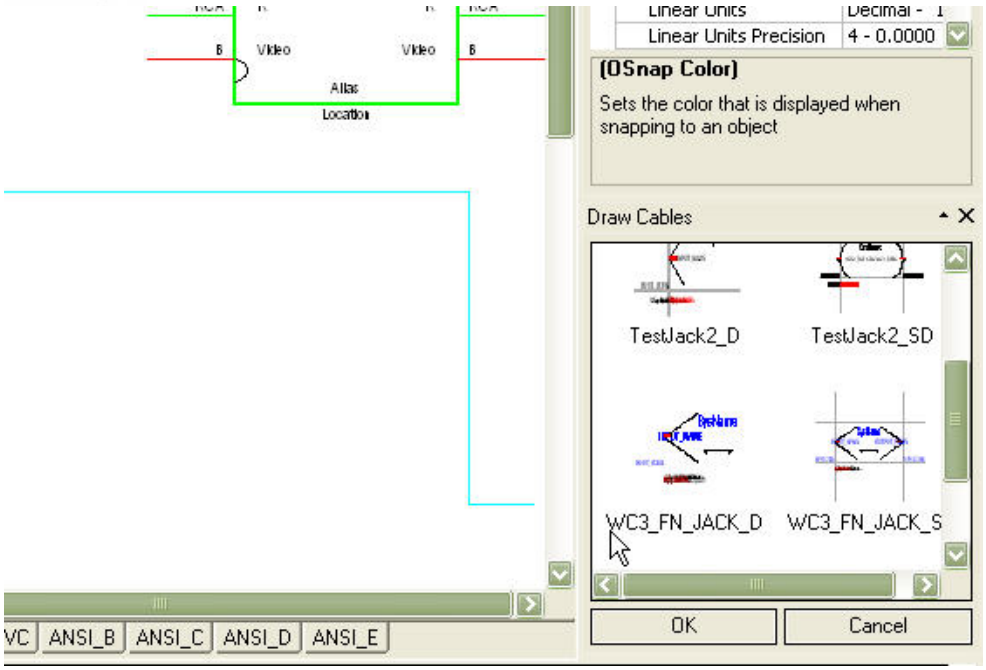


We will now automatically draw a Jack to the right of the selected output.

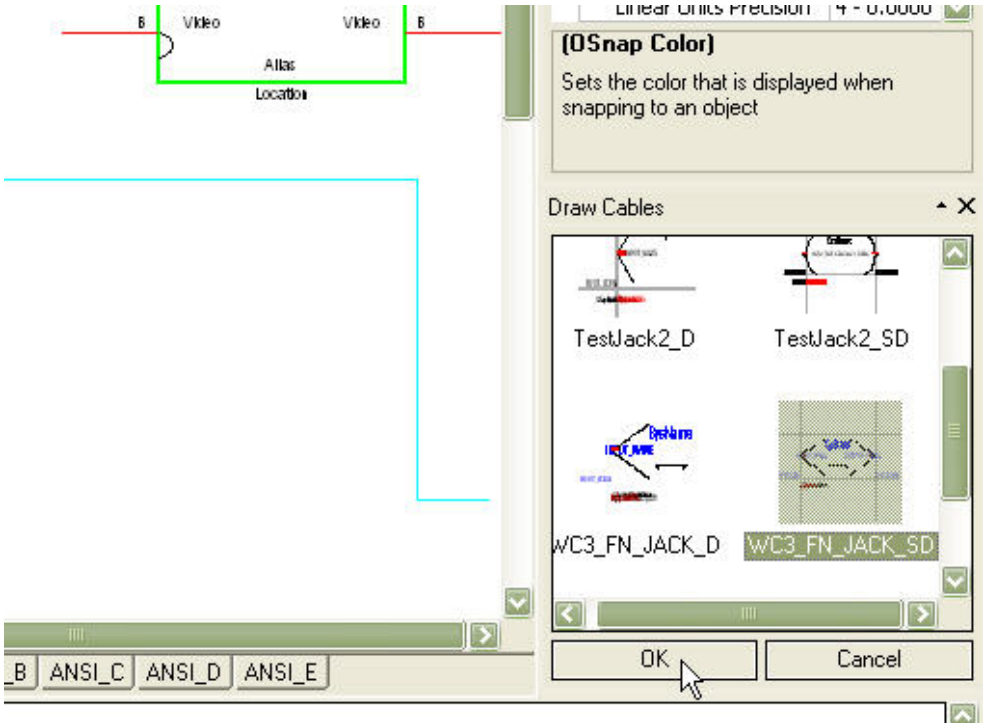
- Select the Jack button:  as follows:



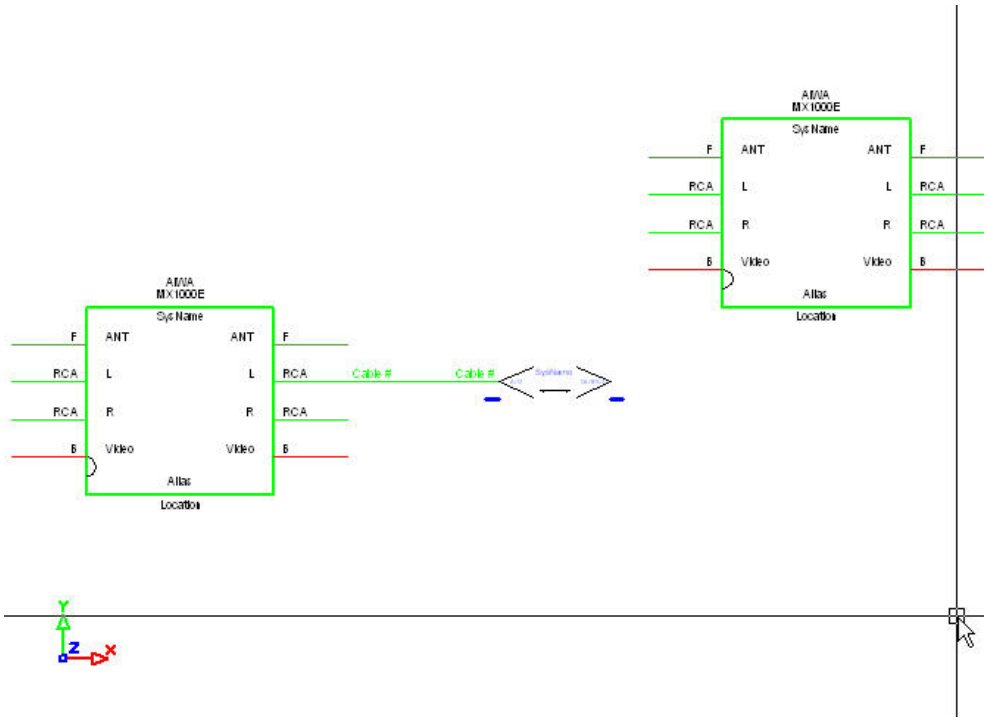
When selecting a Jack, Terminal, or Pointer you are given a choice of styles. Shown here are the Jack types:


Linear Units
Decimal - 1
Linear Units Precision
4 - 0.0000
(OSnap Color)
Sets the color that is displayed when snapping to an object
Draw Cables
TestJack2_D
TestJack2_SD
WC3_FN_JACK_D
WC3_FN_JACK_S
OK
Cancel
Video
Video
Alias
Location
ANSI_B
ANSI_C
ANSI_D
ANSI_E

Select the one called WC3_FN_JACK_SD:

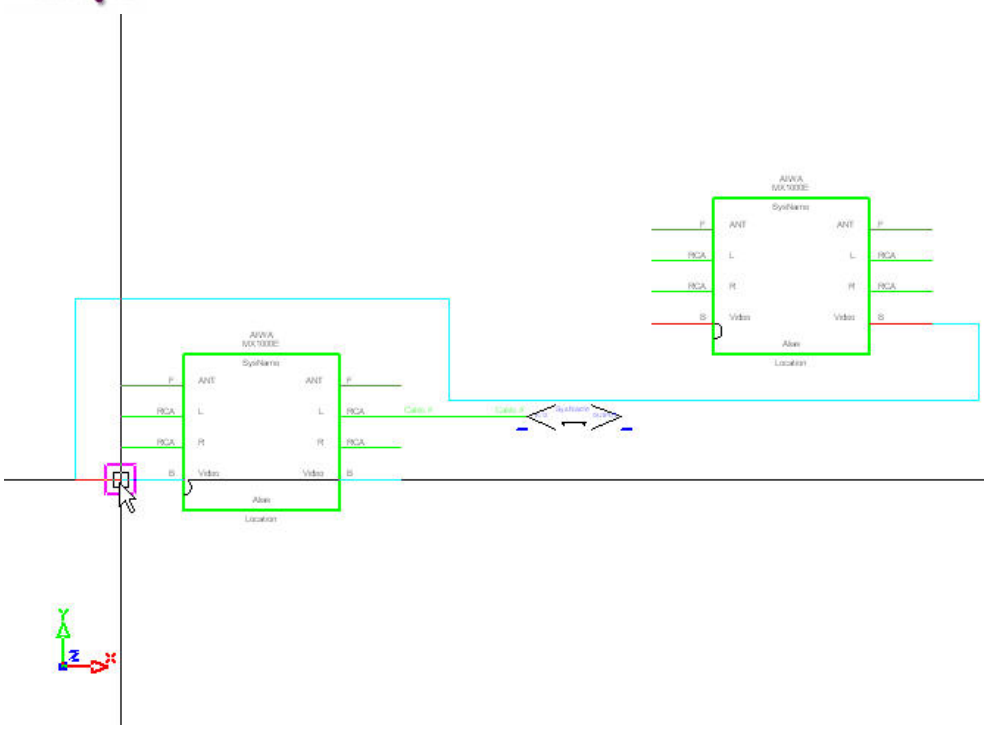


- Then click OK. This will insert a fully normalled jack pair:



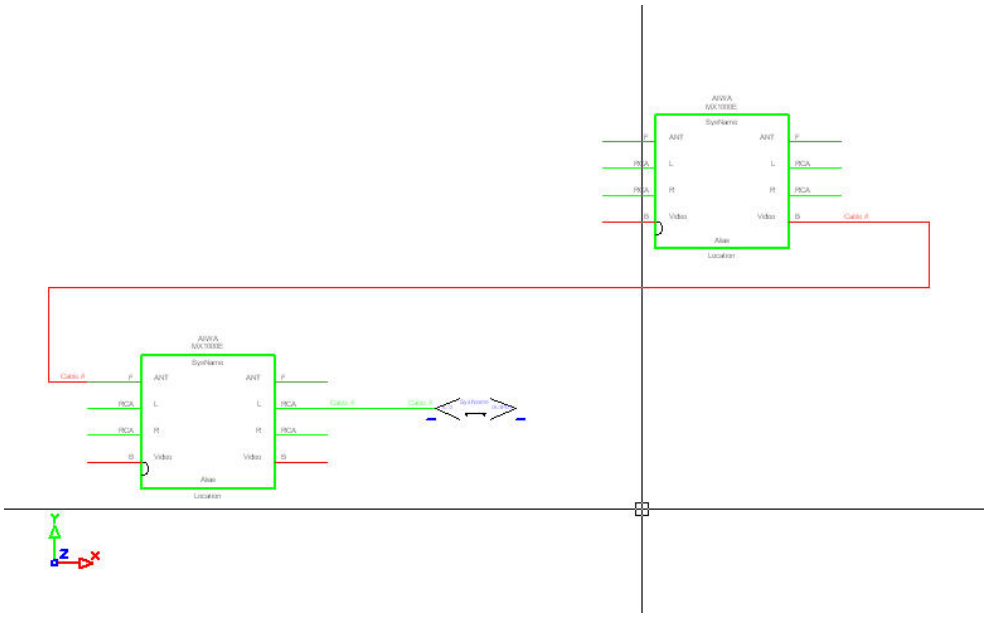
Note that the jack, cable, and cable number text are automatically placed in the drawing 4 drawing (number in the AutoPlace Dist field) units to the right of the output.

If the objective is to draw from output to input on different blocks, then select the output first:



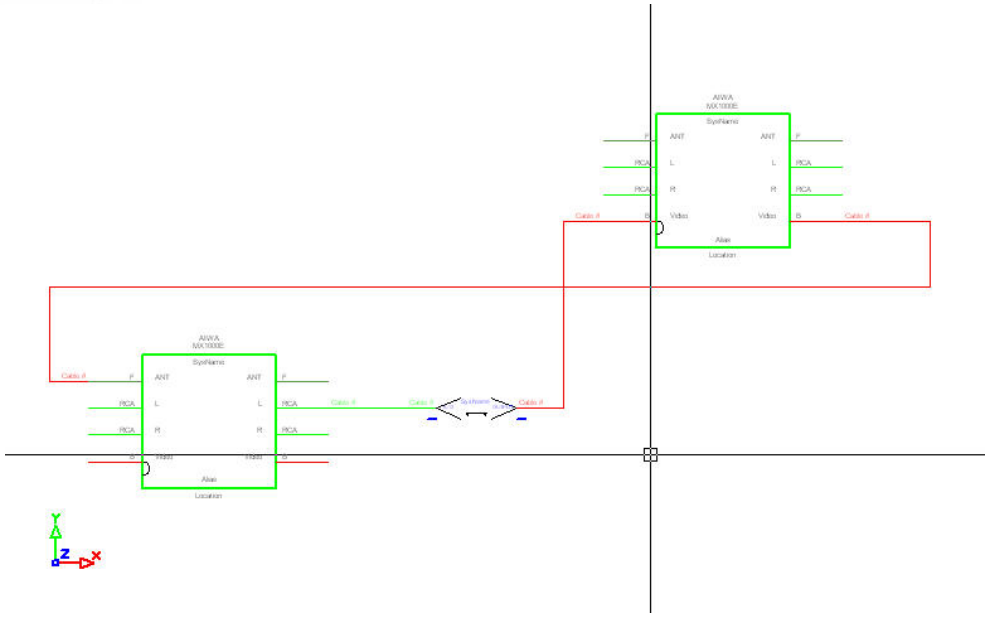
Next, select the input of the device. A cable and cable number text will be placed automatically.

Note: WireCAD will try to avoid equipment and other cables (based on the Avoid Other Cables checkbox). If you need to move a piece of equipment, select it and then grab its grip and drag it to its new location. The wires will remain attached to it and relocate based on the location.



If the objective is to draw from our existing jack to the block, then start the Draw Cable function

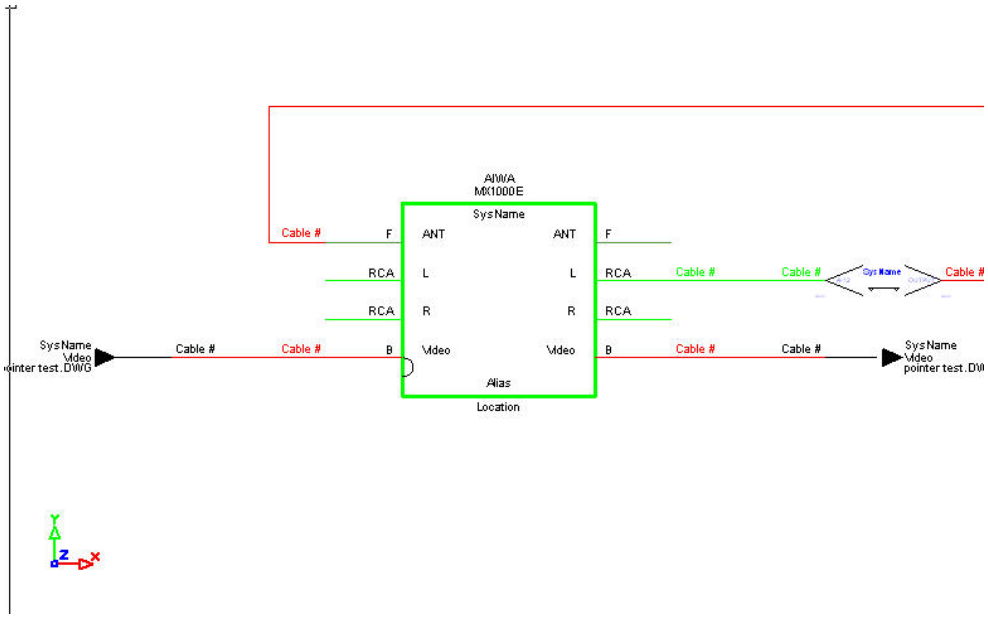
and click on the jack.



Drawing Pointers Automatically

Check the Auto Draw Pointers checkbox. Start the Draw Cables function.

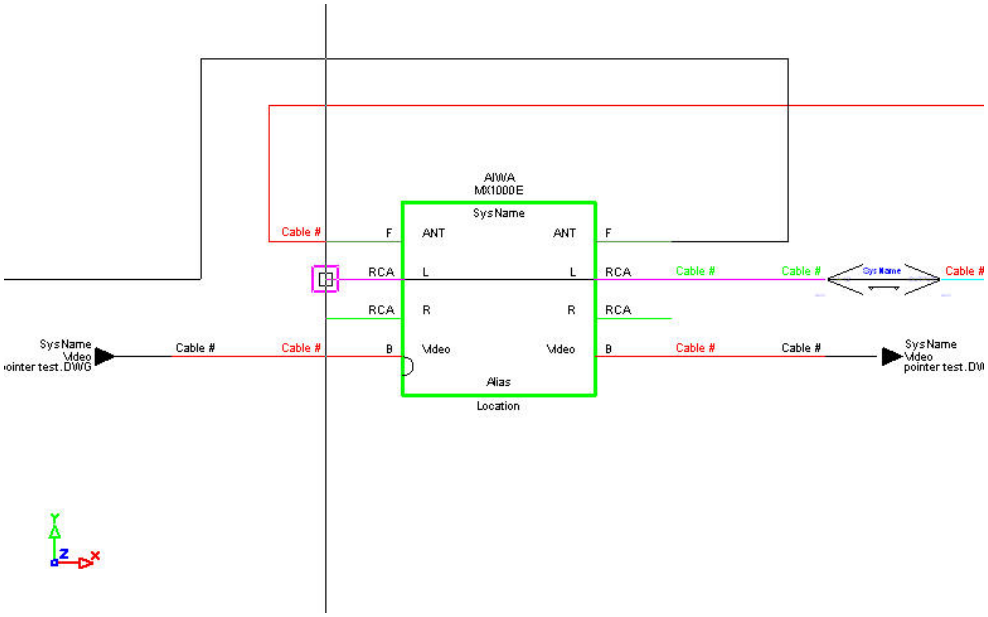
Next, Select the Output followed by the Input.



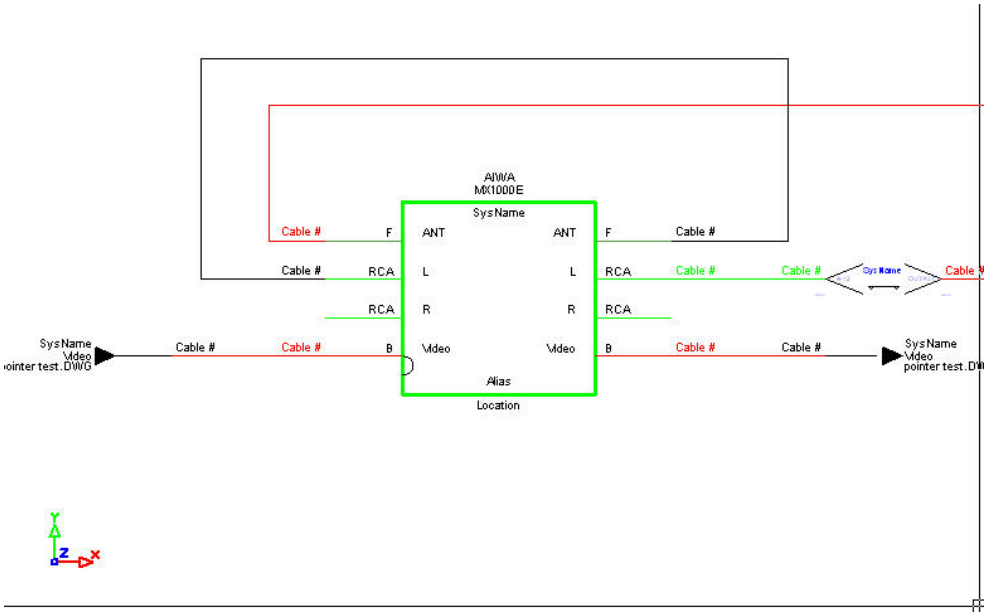
Manually Drawing Cables

Check the Manual Draw checkbox. Start the Draw Cables function.

You will be prompted to start drawing a polyline. Select the Output. Draw the polyline in any direction needed.



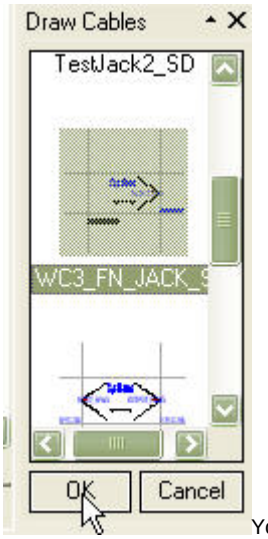
Finish by selecting the desired Input. End the command by right-clicking. This will place the cable# text entities.



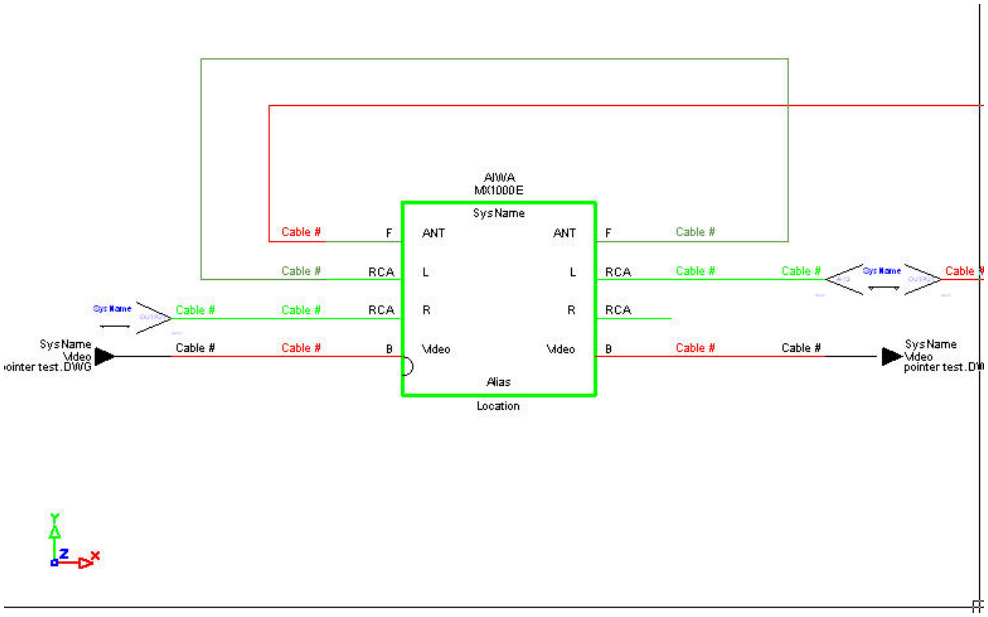
Drawing Jacks, J-Boxes, etc as Sources to the Cable

If you need to have a Jack, Terminal, or Pointer feed an input. Select the terminal button first. 

Note that the Manual Draw function must be unchecked in order to enable the terminal switches. Selecting any of the terminal buttons will start the Draw Cable function.



You are then prompted to select a destination. Upon doing so, if the Auto Place function is checked, the selected Terminal will be placed to the left of the Input based on the number defined in the Auto Place Dist field:

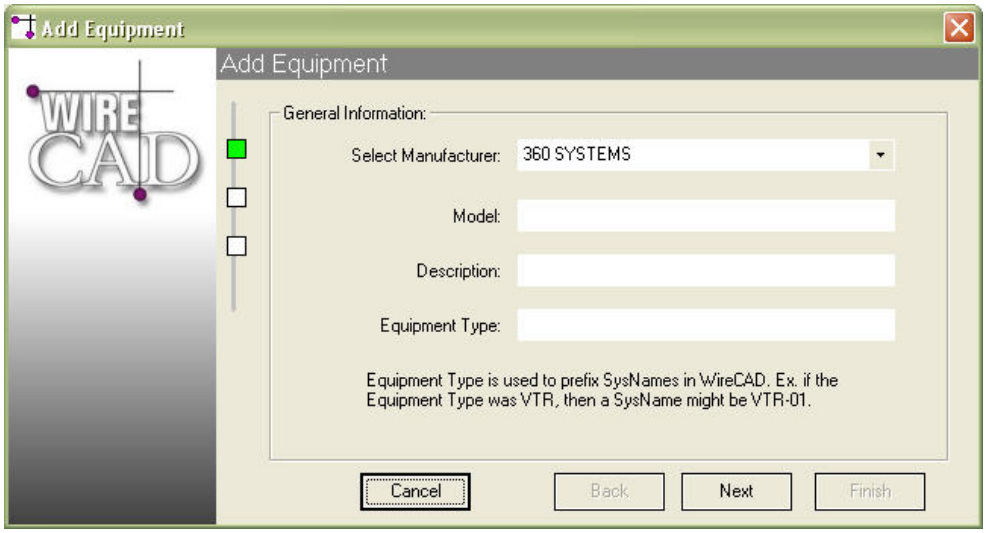


Turning off Auto Place will prompt you for locations to place the terminal devices and then draw cables to the selected location.

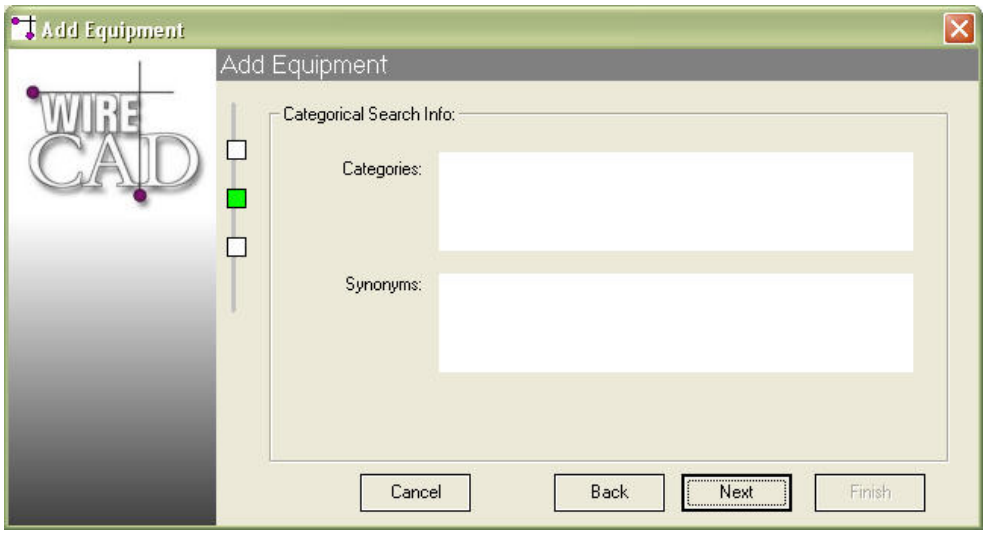
How to Add Equipment to the Library

Start this function by clicking Tools>Equipment>Equipment Library, or the  button. This will display the Equipment Library:

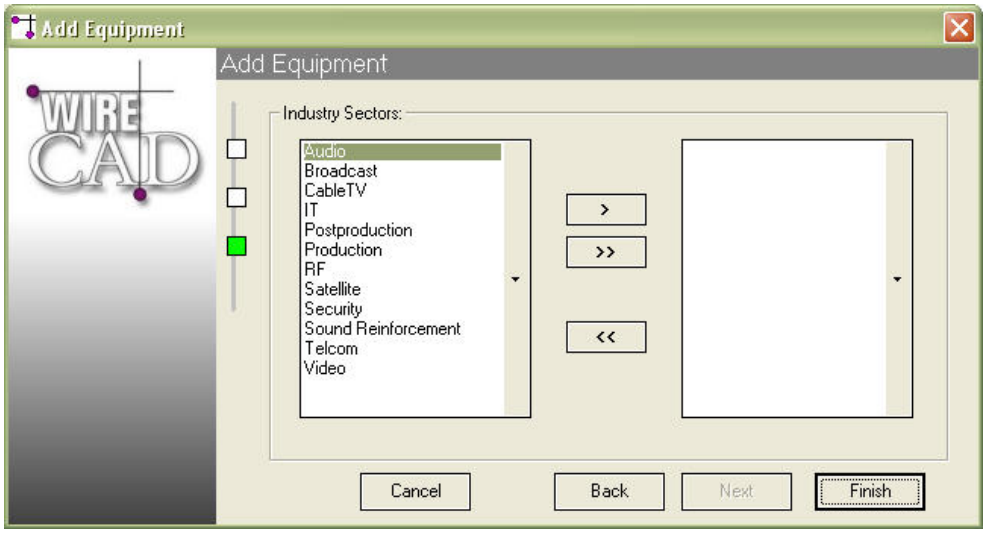
To add a new piece of equipment to the library, click on the Add Equipment to Library button. 



- Select a manufacturer form the drop-down.
- Enter the Model
- Enter the Description
- Enter the Equipment Type. This field is used to prefix system names. Note: use types that are easily recognizable to the engineering staff. This will aid in keeping your documentation readable. Example: if you are using an AVID non-linear editor (NLE) in your system it may be more readable to use AVID as the prefix instead of NLE; however, this assumes that the system will always by occupied by and AVID and may not be the best way to future proof your system. It's a "salt to taste" issue.
- Click Next



- Enter data in the Categories field. Use keywords that describe the categories of equipment that this equipment belongs to. You can use multiple words separated by a comma.
- Enter data in the Synonyms field. Use keywords that describe the device; such as: CPU, COMPUTER, SERVER.
- Click Next



- Use the > button to move industry sectors from the left-hand list to the right-hand list. Note: this list is populated from the Industry Sector global database.

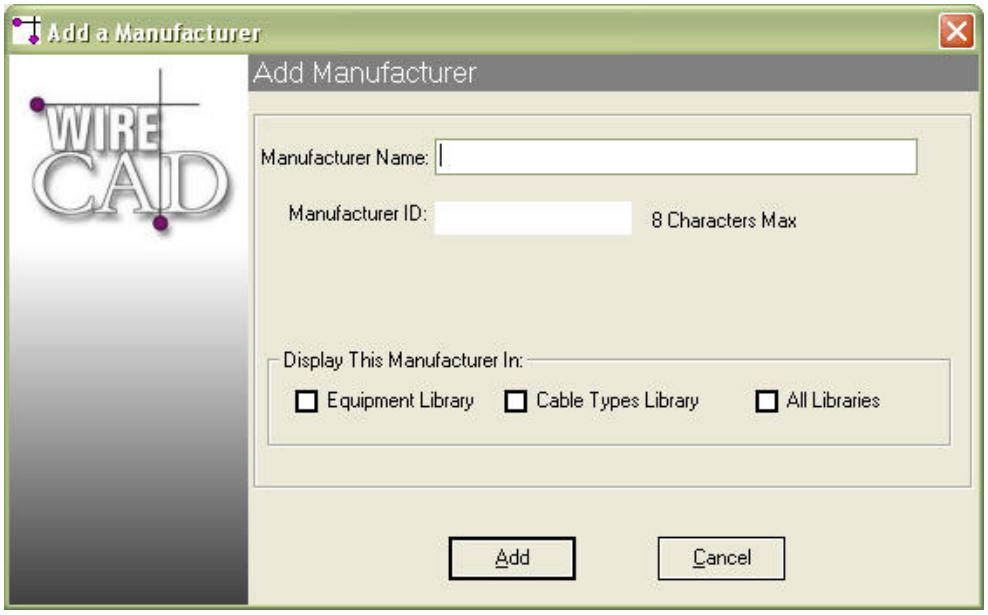
Done.

How to Add Manufacturers to the Library

Start this function by clicking Tools>Equipment>Equipment Library, or the  button. This will display the Equipment Library:

- Click the Add Manufacturer to Library button 

- WireCAD will display the Add Manufacturer Dialog:



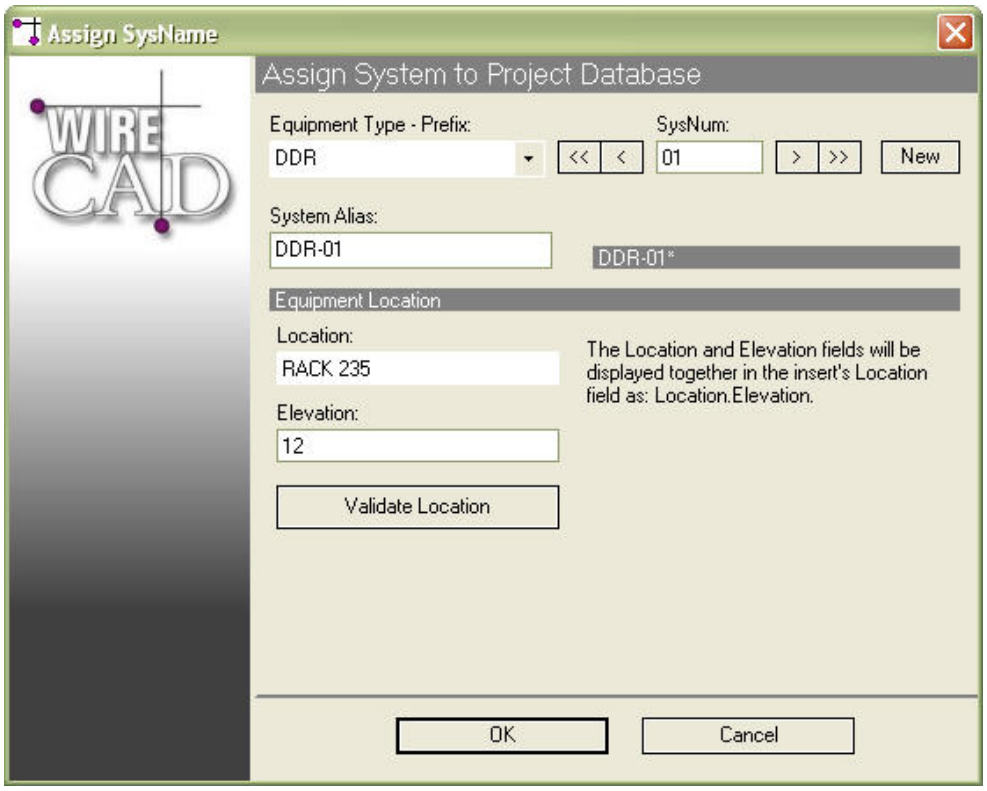
Type the Manufacturer Name in the field. The ManufacturerID will be filled in automatically for you. Select the library where the name is to be displayed. Click Add. If the ManfacturerID already exists, you will be warn and given a chance to modify it.

How to Assign a System Name

Assignment is the term used to indicate that a drawing entity has an associated entry in the database. We refer to several types of assignment:

- SysNames, or system specific identifiers. Thus allowing multiple instances of the same equipment type. This process is where you define the physical location of the equipment in the system.
- Terminals, terminals must first be assigned to the project database. This process is where you define the physical location of the equipment in the system.

Assigning System Names (SysNames), Jacks, Jack Fields, and Cable Numbers is as easy as double-clicking on the entity you wish to assign. Depending on the entity type you will be presented with a series of different dialogs to handle the interface between the drawing environment and the database. If you double-click an equipment block, the following dialog will appear:



Note: A SysName can be assigned to multiple instances of an equipment insert. As an example, assume two drawings – one containing only video, the other containing only audio. Both with a VTR called VTR-01.

Prefix: Sets the prefix for the system Name. Direct entry into this field is possible. This field is referenced to the EquipmentType field in the Equipment Library. The Prefix and the SysNum will be concatenated to form the SysName.

Record Selectors: Used to select an existing SysName. Upon selecting an existing SysName, click OK. You will be informed that the System Name already exists, and prompted whether to proceed with

naming the insert. Selecting the New Record button 

will append a record to the database and increment the SysNum counter.

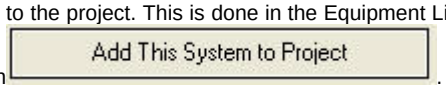
Note that WireCAD v3 automatically assumes a new record as indicated above by the * in the SRVR-01* SysName preview.

Location: User defined location field.

Elevation: User defined elevation field.

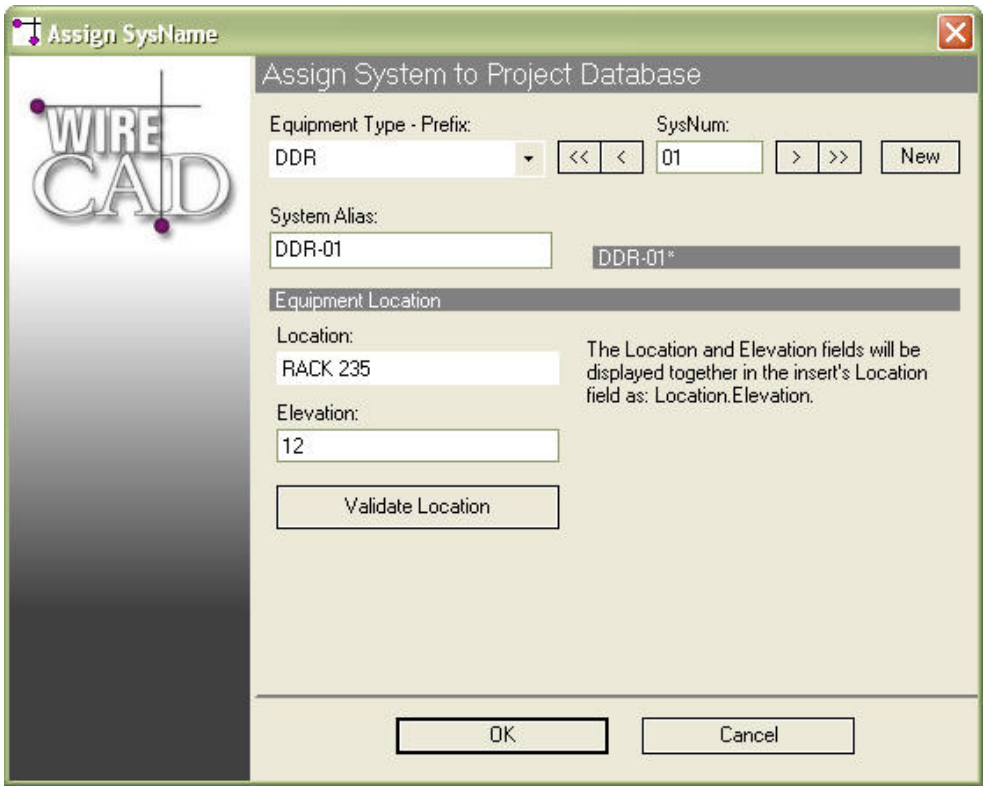
System Alias: Use this field to provide functional or friendly names to systems. Example: suppose you have VTR-450 in your system but the function of this device is Fred's Viewing Deck. Enter "Fred's Viewing Deck" in the alias field to provide an additional description to VTR-450.

Assigning System Names to Individual Terminals

If you have paced WireCAD Terminals in your drawing, these will need to associated with a specific input/output of a specific SysName. For example: suppose we have a jack point in a drawing. That jack point will need to physically occupy a position in a jack-field. The jack-field will have a SysName such as JF-01 and the jack may be the first jack in the top row, say, A-1. The first step in the process is to assign the jack-field to the project. This is done in the Equipment Library  . Find the equipment definition and click the Add This System to Project button 

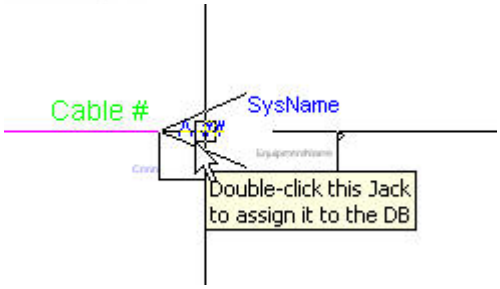
.

This will launch the now familiar Assign Systems dialog mentioned above.

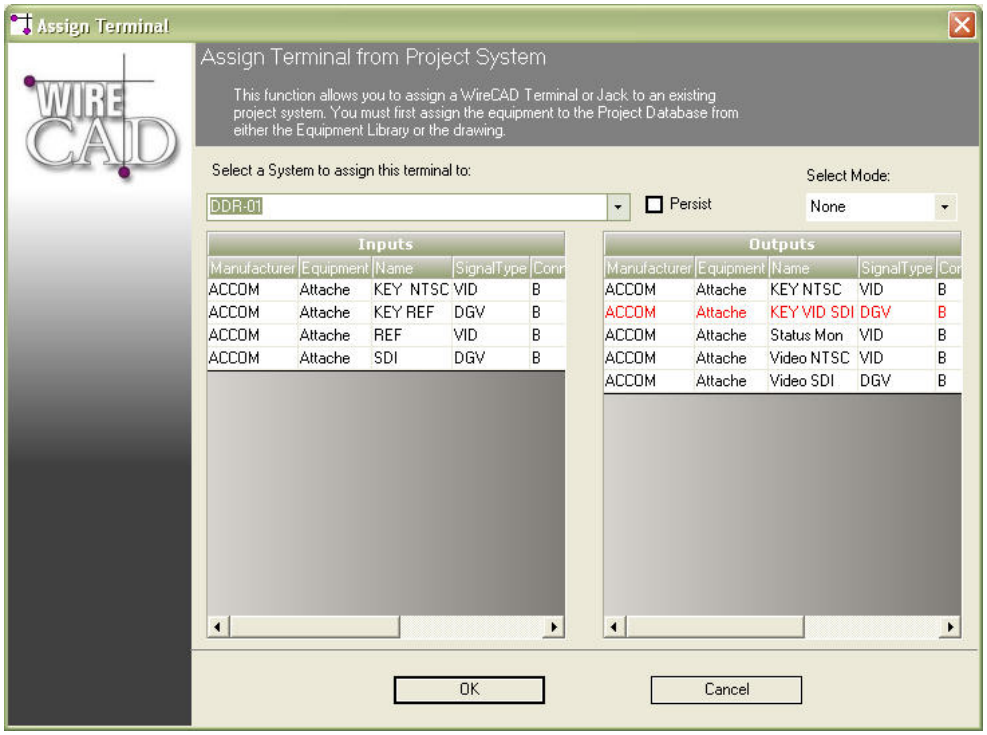


Once you have defined a SysName for your terminal device, you can now assign the individual terminal to the SysName.

To accomplish this, double-click on the terminal in the drawing.:

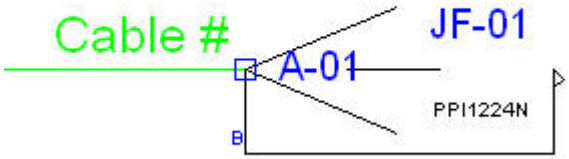


This launches the following dialog where you determine the input/ouput or both that this terminal is to represent:



Select a system from the drop-down. This displays the collection of inputs and outputs for the selected system. Records that are displayed in red are already assigned cable numbers.

Clicking OK will update the information in the drawing:



How to Assign a Cable Number

See the topic: Assigning Cable Numbers

Appendix

Table Structures

dbEquip.mdb Table Definitions – Equipment Database

Table Name	Description	Related To:
SignalDefault	Future	NA
SignalUser	Future	NA
tblCablePairs	Listing of cores and core IDs for Multi-core Cable Types	tblCableTypes
tblCableTypes	Cable Types Data	tblCablePairs
tblConnectors	Connector Data	NA
tblEquipment	Equipment Data excluding input and output info	tblManufactureres, tblInputs, tblOutputs
tblInputs	Equipment Input Data	tblEquipment
tblOutputs	Equipment Output Data	tblEquipment
tblJackFields	Jack Field Data	tblManufacturers
tblManufacturers	Manufacturer Data	tblEquipment, tblJackFields, tblRTR, tblCableTypes
tblPreferences	Preference Data	TblUsers

tblRackUnits	Future	NA
tblRTR	Router Data (future)	tblRTRInputs, tblRTROutputs
tblRTRInputs	Routers Input Data(future)	tblRTR
tblRTROutputs	Router Output Data(future)	tblRTR
tblUsers	User Data	TblPreferences

dbPM.mdb Table Definitions – Project Manager Database

Table Name	Description	Related To:
tblClientInfo	Maintains Data About	tblProjects

	Your Clients	
tblCompany	Your Company Info	
tblProjects	Project Data	tblDrawings, tblDwgTypes, tblDefaultDwgTypes

ProjectDB.mdb Table Definitions – Project Specific Database

Table Name	Description	Related To:
tblJFRowCol	Project Jack Field Individual Jacks Data	tlbJF
tblCables	Main Cable Database	tblChildCables
tblChildCables	Child Cable Database (Not Used – Legacy)	tblCables
tblJF	Project Jack Fields	JFRowCol
tblPID	Pointer ID Generator	
tblRTR	Project Routers (future)	tblXP
tblSystems	Project Sysnames	
tblXP	Project Router Cross-Points	tblRTR
tblBOMMaster	Contains the BOM FileName	tblBOM
tblBOM	BOM table	TblBOMMaster
TblDrawings	Project Drawings	NA

WireCAD Necessary Layers

The following layers are necessary to proper operation of WireCAD:

Layer	Description
Alias	Equipment Block Alias attribute
Cables	Base Cable layer. Most cables will change off of this layer based on Signal Type.
Comments	Future
Connectors	Equipment Block connectors
Equipment	Equipment Block EquipmentName attribute
Location	Equipment Block Location attribute
Manufacturer	Equipment Block Manufacturer attribute
Pinnames	Equipment Block I/O attributes
Sysnames	Equipment Block System Name attribute

WireCAD Necessary Text Styles

The following text styles are necessary to proper operation of WireCAD:

Text Style	Description
Simplex	Jacks, J-Boxes, Router Cross-points, Pointers
WC_EQUIPMENT	Equipment Block EquipmentName attribute
WC_MANUFACTURER	Equipment Block Manufacturer attribute
WC_PIN	Equipment Block I/O attributes
WC_CABLENO	CableNo text entities
WC_SYSNAME	Equipment Block SysName attribute
WC_ALIAS	Equipment Block Alias attribute
WC_LOCATION	Equipment Block System Location attribute
WC_CONNECTOR	Equipment Block System Connector attribute

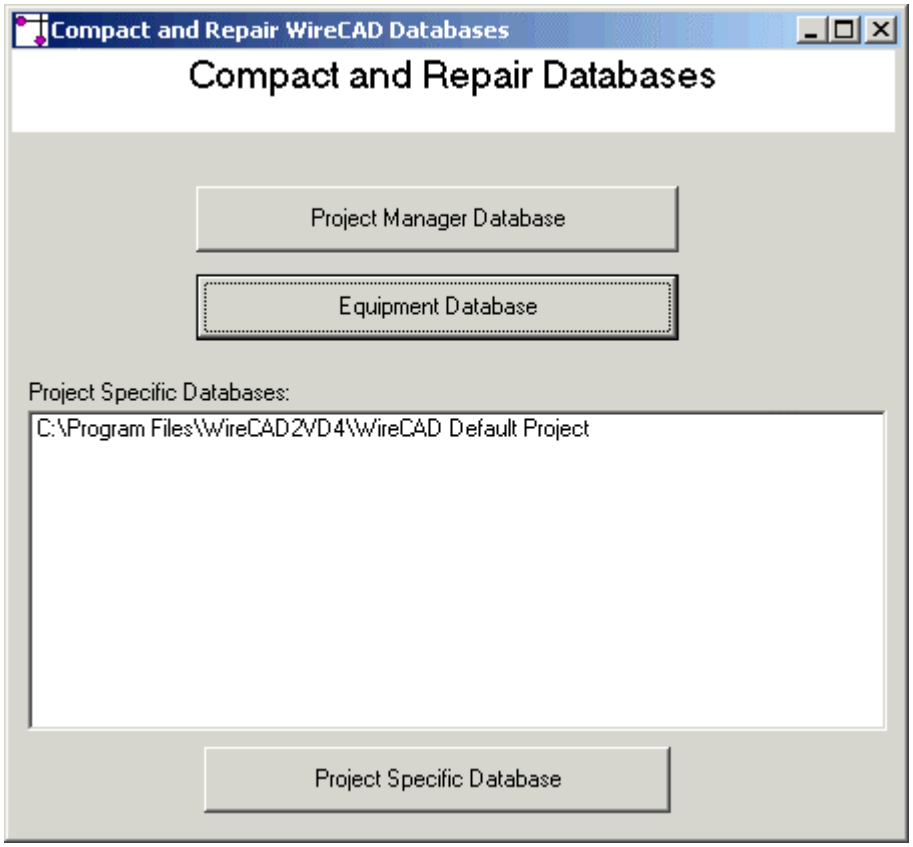
WireCAD Necessary Blocks

The following Blocks are necessary to proper operation of WireCAD:

FN_JACK_SD	Full normal jack set
FN_JACK_S	Full normal source jack
FN_JACK_D	Full normal destination jack
HN_JACK_SD	Half normal jack set
HN_JACK_D	Half normal source jack
HN_JACK_S	Half normal destination jack
JACK1_1_SD	Non normal jack set
JACK1_1_S	Non normal source jack
JACK1_1_D	Non normal destination jack
JBOX_S	J-Box source side
JBOX_D	J-Box destination side
JBOX_1_S	J-Box source side
JBOX_1_D	J-Box destination side
LOOPIN	Looping input
POINTER_S	Pointer source side
POINTER_D	Pointer destination side
RTR_S	Router source side
RTR_D	Router destination side
TERM_50	Terminator 50 ohm
TERM_75	Terminator 75 ohm
TERM	Terminator unspecified impedence

Database Utilities

WireCAD provides a utility for compacting and repairing databases called CompactDBs.EXE located in the WireCAD2 directory. In order for this function to work, all WireCAD v2 users connected to the databases must close the application. If used on a single user machine, close all instances of WireCAD v2 before running this utility. Not doing so will cause the utility to fail.



Project Manager Database: Compact and repair the project manager database (dbPM.mdb) located in the WireCAD2 directory.

Equipment Database: Compact and repair the equipment database (dbEquip.mdb) located in the WireCAD2 directory.

Project Specific Database: This button works in conjunction with the Project Specific Databases list box. Compact and repair the selected project specific database (ProjectDB.mdb) located in the project directory.

Note: these functions will create a backup of the compacted database appending OLD to the filename. Example: dbEquip.mdb will be named dbEquipOLD.mdb.

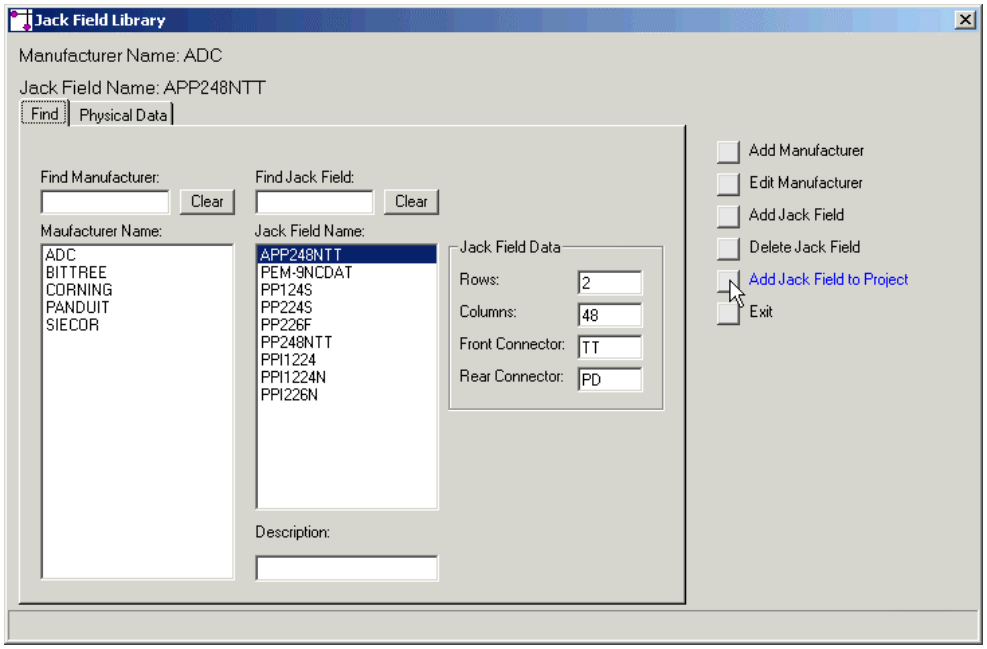
Version 2 Jacks and Jackfields

This section is copied from the version 2 manual and is here for legacy compatibility.

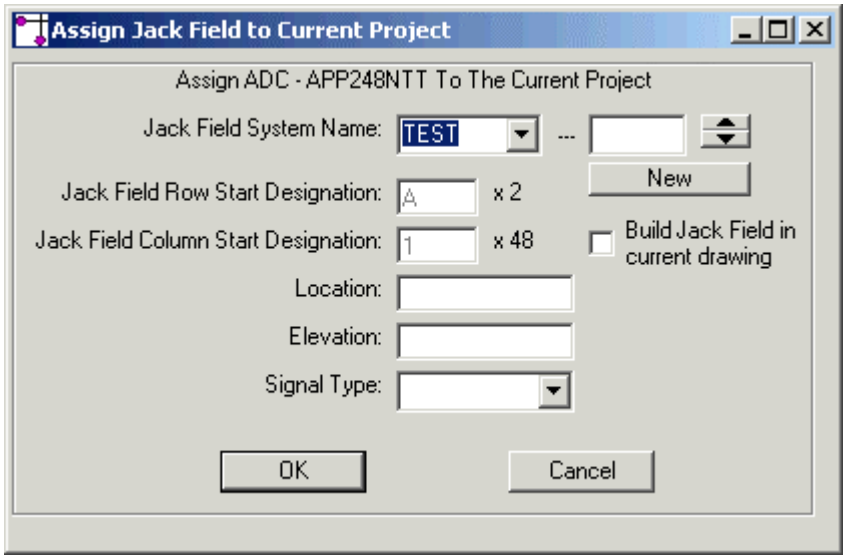
Assigning Jack Fields to the Current Project

In order to make individual jacks available for assignment a Jack Field must first be assigned to the project. During this process you will be given the opportunity to build the entire jack field in the current drawing. This process inserts every jack in the jack Field in the drawing and fills in all of the information regarding the Jack.

This function is available from the Jack Fields Library:



Select a manufacturer and Jack Field to assign. Next, click the Add Jack Field to Project button. You will be presented with the following dialog:



Jack Field System Name: Similar to the Prefix and SysNum fields in the Assign Equipment Function Above. You may type directly in these fields.

Note: the recommended practice is to type or select the prefix and click the New button to get the next number in the series.

New: Gets the next number in the series associated with the selected prefix. If the prefix has no associated entries then the New function will return 01 as the starting number.

Jack Field Row Start Designation: Future.

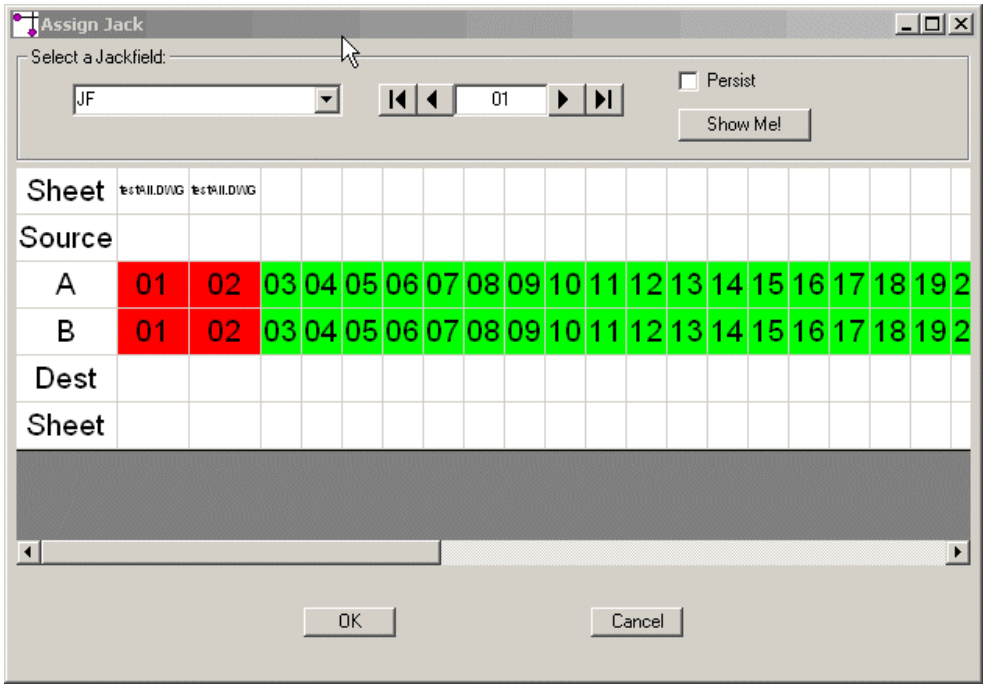
Jack Field Column Start Designation: Future.

Location: User defined location field.

Elevation: User defined elevation field.

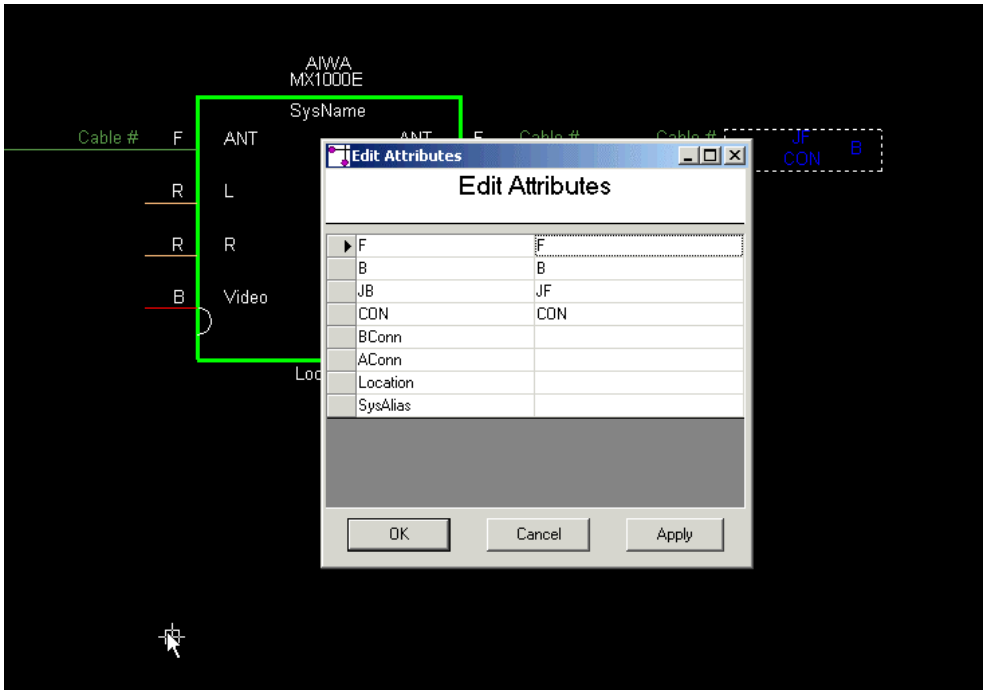
Assigning Individual Jacks

Double-click on an individual jack to assign it to the database. You are presented a dialog representing the jack field. Green squares represent un-assigned jacks, while red squares let you know that the jack is already assigned and in which drawing.



Assigning System Names to J-Boxes and Router Cross-points

Once a J-box or Router Cross-point is placed in the drawing, simply edit its attributes by double-clicking on it. The following dialog will appear allowing editing:



The following is a table of attributes associated with J-Boxes and Router Cross-points:

J-Box		**Router**	
Attribute	Description	Attribute	Description
JF	SysName	RTR	SysName
CON	Connector	Layer	Router Layer
F	Front or Back of Panel	A	Cross-point Number
B	Front or Back of Panel	B	Cross-point Number
AConn	Left Side Connector	AConn	A Cross-point Connector
BConn	Right Side Connector	BConn	B Cross-point Connector
Location	Location	Location	Location
SysAlias	System Alias	SysAlias	System Alias

Keyboard Shortcuts

Keyboard shortcuts are defined in a file called “User.ks” in the APPPath\Users\User.ks. User.ks follows this form:

```
'////////////////////////////////////////////////////////////////////////////
'/WireCAD keyboard shortcuts. (c)2003
'/Holbrook Enterprises, Inc. All rights reserved
'/
'/RULES: Modify this file to include your shortcuts
'/Place your shortcut after the comma, replacing any
'/existing text. Note that two letter shortcuts
'/are executed first, next three letter shortcuts.
'/Any Shortcut longer than three letters must be
'/executed by typing the <ENTER> key.
'/
'/Example: CMD_LINE,LI would be executed immediately,
'/as would CMD_CIRCLE,CIR. However, CMD_OSNAP,OSNAP would
'/require that you press the <ENTER> key to execute it
'/Note further, that commands cannot contain similar
'/strings i.e. CI and CIRCLE could not be defined as shortcuts
'/since anytime CI is typed it will execute immediately
'/
'/WARNING! THERE IS NO TYPE CHECKING OR VALIDATION ON THIS FILE
'/NOT FOLLOWING THE ABOVE RULES WILL LEAD TO ERRATIC BEHAVIOR
'////////////////////////////////////////////////////////////////////////////



'////////////////////////////////////////////////////////////////////////////
'/Drawing Tools
'////////////////////////////////////////////////////////////////////////////
CMD_LINE,DL
CMD_CIRCLE,DCC
CMD_CIRCLE_2PT,DC2
CMD_CIRCLE_3PT,DC3
CMD_PLINE,DP
CMD_ARC,DAA
CMD_ARC_2PT,DA2
CMD_ARC_3PT,DA3
CMD_RECT,DR
CMD_TEXT,DT
CMD_ATTDEF,DAT
CMD_POINT,DP
CMD_IMAGE,DI
'////////////////////////////////////////////////////////////////////////////
'/Dimension Tools
'////////////////////////////////////////////////////////////////////////////
CMD_DIM_ALIGNED,DDA
CMD_DIM_H,DDH
CMD_DIM_V,DDV
CMD_DIM_RADIUS,DDR
CMD_DIM_ANGLE,DDG
'////////////////////////////////////////////////////////////////////////////
'/File Operations
'////////////////////////////////////////////////////////////////////////////
CMD_DWG_OPEN,FO
CMD_DWG_CLOSE,FC
CMD_DWG_SAVE,FS
CMD_DWG_NEW,FN
CMD_DWG_PRINT,FP
CMD_DWG_PREVIEW,PP
'////////////////////////////////////////////////////////////////////////////
```

```
'/Entity Modification
'////////////////////////////////////////////////////////////////////////////////
CMD_DELETE,MD
CMD_COPY,MC
CMD_MOVE,MM
CMD_ROTATE,MR
CMD_SCALE,MS
CMD_MIRROR,MI
CMD_STRECTH,ME
CMD_EXPLODE,MDA
CMD_TRIM,MT
CMD_EXTEND,MX
CMD_SENDTOBACK,MTB
CMD_BRINGTOFRONT,MTF
CMD_BREAK,MB
CMD_FILLET,MF
CMD_ARRAY,MA
'////////////////////////////////////////////////////////////////////////////////
'/Object Snaps NOT SUPPORTED YET
'////////////////////////////////////////////////////////////////////////////////
CMD_OSNAP_END,OSE
CMD_OSNAP_MID,OSM
CMD_OSNAP_CEN,OSC
CMD_OSNAP_NEAR,OSN
CMD_OSNAP_INSERT,OSI
CMD_OSNAP_PERP,OSP
CMD_OSNAP_TANG,OST
CMD_OSNAP_QUAD,OSQ
CMD_OSNAP_INT,OSX
CMD_OSNAP_POINT,OSPT
'////////////////////////////////////////////////////////////////////////////////
'/WireCAD Drawing Tools and Equipment Libraries
'////////////////////////////////////////////////////////////////////////////////
CMD_ASSIGN_CABLE,AC
CMD_ASSIGN_SYSTEM,AS
CMD_ASSIGN_JF,AF
CMD_ASSIGN_JACK,AJ
DLG_EQUIP_LIB,LE
DLG_JF_LIB,LJ
DLG_CT_LIB,LC
DLG_PCABLES_DS,LPC
DLG_PSYSTEMS_DS,LPS
'////////////////////////////////////////////////////////////////////////////////
'/VIEW Functions
'////////////////////////////////////////////////////////////////////////////////
CMD_ZOOM_EXTENTS,ZE
CMD_ZOOM_WINDOW,ZW
CMD_ZOOM_LAST,ZL
CMD_ZOOM_IN,ZI
CMD_ZOOM_OUT,ZO
CMD_REDRAW,VR
CMD_REGEN,VG
CMD_MODEL,VM
CMD_PAPER,VP
CMD_MEASURE,VD
CMD_CALC,VC
'////////////////////////////////////////////////////////////////////////////////
'/Format Dialogs
'////////////////////////////////////////////////////////////////////////////////
DLG_POINTSTYLES,SP
DLG_TEXTSTYLES,ST
DLG_DIMSTYLES,SD
```

```
DLG_LAYERS,SL
DLG_INSERTS,SI
DLG_LAYOUTS,SO
'////////////////////////////////////////////////////////////////////////////
'/Toolbars
'////////////////////////////////////////////////////////////////////////////
TLB_DWG_TOOLS,TBT
TLB_DWG_FORMAT,TBF
TLB_DWG_MODIFY,TBM
TLB_DWG_OSNAP,TBO
TLB_DWG_VIEW,TBV
TLB_DWG_SINGLE_LINE,TBS
TLB_DWG_DIMS,TBD
TLB_DWG_COMMAND,TBC
```

License Agreement

YOU SHOULD CAREFULLY READ THE FOLLOWING TERMS AND CONDITIONS BEFORE OPENING THIS PACKAGE AND/OR BY USING THE SOFTWARE. OPENING THIS PACKAGE OR USING THE SOFTWARE INDICATES YOUR ACCEPTANCE OF THESE TERMS AND CONDITIONS. IF YOU DO NOT AGREE WITH THEM, YOU SHOULD PROMPTLY RETURN THE PACKAGE TO THE LOCATION WHERE YOU PURCHASED THE SOFTWARE, UNOPENED WITH PROOF OF PURCHASE, AND YOUR MONEY WILL BE REFUNDED.

Holbrook Enterprises, Inc. provides this program and licenses its use. Holbrook Enterprises, Inc. retains the ownership of this product. This Agreement shall be governed by the laws of the state of Delaware.

LICENSE

Permitted Uses/You May:

* Use the software on any computer provided the software is used on only one computer and by one user at a time.
* Copy the program into any machine readable or printed form for backup or modification purposes in support of your use of the program on a single machine.

Prohibited Uses/You May not:

* Make copies of the documentation or software, except as noted above.
* Distribute, rent, sub-license, transfer, or lease the software or documentation.
* Alter, modify or adapt the software or documentation, including, but not limited to, translating, decompiling, disassembling, or creating derivative works.

This license and your right to use the software automatically terminate if you fail to comply with the provisions of this License Agreement.

TERM

The license is effective until terminated. You may terminate it at any other time by destroying the program together with all copies, modifications and merged portions in any form. It will also terminate upon conditions set forth elsewhere in this Agreement or if you fail to comply with any term or condition of this Agreement. You agree upon such termination to destroy the program together with all copies, modifications and merged portions in any form.

IF YOU TRANSFER POSSESSION OF ANY COPY, MODIFICATION OR MERGED PORTION OF THE PROGRAM TO ANOTHER PARTY, YOUR LICENSE IS AUTOMATICALLY TERMINATED.

TRADE SECRET

You acknowledge that the software constitutes valuable trade secret information that is the exclusive property of Holbrook Enterprises, Inc..

LIMITED WARRANTY

Holbrook Enterprises, Inc. warrants for a period of 90 days from the date of original delivery to you that the program will substantially conform to the published specifications and to the documentation, provided that it is used on the computer hardware and with the operating system for which it was designed. Holbrook Enterprises, Inc. warrants the diskette(s) on which the program is furnished, to be free from defects in materials and workmanship under normal use for a period of ninety (90) days from the date of delivery to you as evidenced by a copy of your receipt. This warranty gives you specific legal rights. You may have other rights that vary from state to state.

DURING THE WARRANTY PERIOD, IF THE SOFTWARE DOES NOT PERFORM AS WARRANTED, YOUR EXCLUSIVE REMEDY SHALL BE TO SEND THE SOFTWARE TO HOLBROOK ENTERPRISES, INC. WHICH SHALL, AT ITS OPTION, EITHER REFUND TO YOU THE PRICE PAID OR REPAIR OR REPLACE THE SOFTWARE.

To the extent permitted by applicable law, Holbrook Enterprises, Inc. disclaims all other warranties, either or implied, including, buy not limited to, warranties of merchant ability and fitness for a particular purpose. Holbrook Enterprises, Inc. makes no warranty as to title. No Holbrook Enterprises, Inc. dealer, distributor, agent, or employee is authorized to make any modification or addition to this warranty.

LIMITATION OF LIABILITY

Because programs are inherently complex and may not be completely free of errors, you are advised to validate your work. TO THE EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT WILL HOLBROOK ENTERPRISES, INC. OR ANY OF ITS PRINCIPALS OR AGENTS BE LIABLE FOR DIRECT, INDIRECT, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES WHATSOEVER ARISING OUT OF THE USE OF OR INABILITY TO USE THE PROGRAM OR DOCUMENTATION, even if advised of the possibility of such damages. Specifically, Holbrook Enterprises, Inc. is not responsible for any costs or damages including, but not limited to, those incurred as a result of lost profits or revenue, loss of use of the computer program, loss of data, the costs of recovering such programs or data, the cost of any substitute program, claims by third parties, or for other similar costs. IN NO CASE WHATSOEVER SHALL HOLBROOK ENTERPRISES, INC.'S LIABILITY EXCEED THE PRICE PAID FOR THE LICENSE TO USE THE SOFTWARE.

SOME STATES DO NOT ALLOW THE LIMITATION OR EXCLUSION OF LIABILITY FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES SO THE ABOVE LIMITATION OR EXCLUSION MAY NOT APPLY TO YOU.

Should you have any questions concerning this Agreement, you may contact Holbrook Enterprises, Inc. by writing to: Holbrook Enterprises, Inc. 24421 Chestnut Street Suite 101, Newhall, CA 91321

You acknowledge that you have read this agreement, understand it and agree to be bound by its terms and conditions. You further agree that it is the complete and exclusive statement of the agreement between us which supersedes any proposal or prior agreement, oral or written, and any other communications between us relating to the subject matter of this agreement.

I

J

K

L

M

O

P

R

S

T

U

V

W

Z