

WireCAD v10.1 User Manual

USER MANUAL

© 2025 Holbrook Enterprises, Inc. - WireCAD
Holbrook Enterprises, Inc. - WireCAD

This page is intentionally left blank.
Remove this text from the manual
template if you want it completely blank.

1. Welcome	13
2. Introduction	15
2.1 New in Version 10	17
2.2 Software Activation	31
2.3 License Agreement	36
2.4 Licensing FAQ	38
2.5 Context Sensitive Help	40
2.5.1 Dialog Based Help	40
2.5.2 Form Based Help	42
2.5.3 Tool Panel Based Help	43
2.5.4 Settings Panel Based Help	44
3. WireCAD PRO Manual	47
3.1 Getting Started	48
3.1.1 Grid Basics	49
3.1.1.1 How To: Grid Functions	50
3.1.1.2 How To: Round Trip Export/Imprt on all Grids	55
3.1.2 Setting Up Your Global Data	58
3.1.2.1 Signal Types	59
3.1.2.2 Cable Types	60
3.1.2.2.1 How To: Create a New Cable Type	61
3.1.2.3 How To: Open the Equipment Library	62
3.1.3 Equipment Library	63
3.1.3.1 I/O Tab	64
3.1.4 Display Preferences Tab	64
3.1.5 Creating a New Equipment Definition	65
3.1.5.1 How To: Add Equipment to the Library	65
3.1.5.2 How To: Add Some Inputs and Outputs	68
3.1.6 Creating a New Project	70
3.1.6.1 How To: Create a New Project	70
3.1.7 Creating a New Drawing	77
3.1.7.1 How To: Create a New Drawing	78
3.1.8 Adding Equipment to Drawings	79
3.1.8.1 How To: Add Equipment to Drawings	80
3.1.9 Drawing Cables	82
3.1.9.1 One to One Cables	86
3.1.9.2 One to Many Cables	90
3.1.9.3 Many to One Cables	93
3.1.9.4 Many to Many Cables	94
3.1.9.5 Terminal as Source	95
3.1.9.6 How To: Place a Terminal as a Source	96

3.1.9.7	Terminal as a Destination	96
3.1.9.8	Splice Points	97
3.1.9.9	Adapters	97
3.1.9.10	Working with Pointers (on-sheet/off-sheet reference)	98
3.1.9.10.1	How To: Link Many Pointers at Once	99
3.1.9.10.2	How To: Swap Pointers	100
3.1.10	Defining Locations	101
3.1.10.1	How To: Add a New Location	101
3.1.10.2	How To: Add Multiple Locations	102
3.1.11	Assigning Unique IDs (SysNames)	103
3.1.11.1	How To: Assign a SysName	104
3.1.11.2	How To: Assign Multiple SysNames	104
3.1.12	Assigning Terminals	105
3.1.12.1	How To: Assign Terminals	106
3.1.12.2	How To: Assign Multiple Terminals	106
3.1.12.3	How To: Swap Terminals	107
3.1.13	Assigning Cable Numbers	108
3.1.13.1	How To: Assign a Cable Number	109
3.1.13.2	How To: Assign Multiple Cable Numbers	110
3.1.14	Rack Builder Tool	110
3.1.14.1	How It Works	112
3.1.15	Next Numbers Grid	114
3.2	Advanced Topics	115
3.2.1	Equipment List Visualizer	115
3.2.1.1	How To: Use the Equipment Visualizer	116
3.2.2	Cables Visualizer	117
3.2.2.1	How To: Use the Cables Visualizer	119
3.2.3	AutoScheme Tools	119
3.2.3.1	Auto Block	120
3.2.3.2	RatsNest	122
3.2.4	SysName Formatting	123
3.2.5	Cable Number Formatting	125
3.2.6	Using Your CAD Drawings With WireCAD	128
3.2.6.1	How To: Customize Your CAD Blocks to Work with WireCAD	128
3.2.6.2	How To: Add a Port to an Existing CAD Block	131
3.2.7	Understanding Indexes	133
3.2.8	Sync Equipment with Database	135
3.2.9	Swap Equipment	136
3.2.9.1	How To: Swap Equipment	137
3.2.10	Edit Block in Place	137
3.2.10.1	How To: Edit a Block In Place	138
3.2.11	Rereference Pointers	139
3.2.12	Save and Ripple	140
3.2.13	Spare Cables	142

3.2.13.1	How To: Add a Spare Cable	143
3.2.14	Split Cables	144
3.2.14.1	How To: Split Cables	144
3.2.15	Aux Text	145
3.2.15.1	How To: Apply Aux Text to an Existing Cable	145
3.2.16	Title Block Manager	146
3.2.16.1	How To: Adjust Attributes in the Title Block Manager	146
3.2.17	Understanding Terminals	149
3.2.17.1	How To: Customize a Terminal	150
3.3	Plan View and Layout Tools Dialogs	150
3.3.1	Take Offs	151
3.3.1.1	How To: Pull a Take-Off from a Drawing	152
3.3.2	Location Boundary	154
3.3.2.1	How To: Place a Location Boundary	156
3.3.3	Draw Cable	156
3.3.3.1	How To: Draw a Cable in Planview	157
3.3.4	Populate Cables	157
3.3.4.1	How To: Populate Cables in Planview	159
3.3.5	Draw Prewire	159
3.3.5.1	How To: Draw a Prewire Cable in Planview	160
3.3.6	Populate Equipment	161
3.3.6.1	How To: Populate Equipment in Planview	162
3.4	Reporting	163
3.4.1	How To: Print Reports	163
3.4.2	How To: Print Reports with Parameters	164
3.4.3	How To: Filter Reports	165
3.4.4	How To: Filter a Report Using Parameters	167
3.4.5	Report Design	170
3.4.5.1	How To: Create a Standard Report	170
3.4.5.2	How To: Create a Label Report	174
3.4.5.3	Report Design Basics	176
3.5	Frequently Asked Questions	179
3.5.1	How To: Place Custom Titleblocks (Page Borders)	179
3.5.2	How To: Create Custom Titleblocks	180
3.5.3	Moving Projects (Pack Up/Check-Out)	181
3.5.3.1	How To: Packup/Check-out	184
3.5.3.2	How To: Check In a Project	184
3.5.3.3	How To: Unpack a Project	184
3.5.4	Synchronizing with Another Equipment Database	184
3.5.4.1	How To: Sync Equipment Databases	186
3.5.5	Setting Up On a Network	187
3.5.6	Upgrading from V9	188
3.6	Choosing a Database Format	188

3.6.1	How To: Setup a SQL Server	189
3.7	Included Plugins	190
3.7.1	Pinouts	191
3.7.2	Find and Replace	192
3.7.2.1	How To: Find and Replace a Port Name	192
3.7.3	Translation Manager	194
3.7.3.1	How To: Translate a Form Caption	195
3.7.4	Block Extractor	196
3.7.5	Bulk Block Fixer	198
3.7.5.1	How To: Use a Manufacturer Front Panel	199
3.7.6	Batch Plot	200
3.7.7	Archive Project	202
3.8	Project Rules	203
3.8.1	Expression Syntax	204
4.	WireCAD CMS Tools Manual	207
4.1	CMS Tools Concept	208
4.2	CMS Basics	209
4.2.1	Backbones	210
4.2.1.1	The Backbone Grid	211
4.2.1.1.1	How To: Draw a Backbone	214
4.2.1.2	How To: Create a New Backbone	214
4.2.1.3	Backbone Visualization Settings	219
4.2.1.4	Backbone Preview Export	223
4.2.1.5	Import Backbone Data	223
4.2.2	Blind Patch Tool	226
4.2.2.1	How To: Create a Blind Patch	227
4.2.2.2	How To: Discover Circuits From Blind Patches	227
4.2.2.3	How To: Make a Real Circuit From Blind Patches	228
4.2.3	Circuits	228
4.2.3.1	Circuits Grid	229
4.2.3.2	Circuits Preview Settings	230
4.2.3.3	Outputting Circuits to CAD	233
4.2.3.4	How To: Output a Circuit to CAD	233
4.2.3.5	How To: Output Many Circuits to CAD	235
4.2.3.6	New Circuit Tool	236
4.2.3.6.1	How To: Create a New Circuit	237
4.2.3.6.2	Path Finder Tab	241
4.2.3.7	Combining Circuits	243
4.2.3.8	How To: UnMake a Real Circuit Into Blind Patches	244
4.2.3.9	Import Circuit Data	245
4.2.3.10	Exporting Circuit Data to Excel	249
4.2.4	Next Numbers Grid	249
4.2.5	GPS Mapping Tools	250

4.2.5.1	Using the Map Tool	254
4.2.5.1.1	How To: Associate GPS LAT, LON with your Locations	255
4.2.5.1.2	How To: Show Location Markers in the Map Tool	258
4.2.5.1.3	How To: Show Panel Markers in the Map Tool	259
4.2.5.1.4	How To: Show Backbone Markers in the Map Tool	260
4.2.5.1.5	Using Filters	262
4.2.5.2	Using Your DWG Drawings as Maps	262
4.2.5.2.1	How To: Calibrate (Geo-Reference) Your DWG to the GPS Coordinate System	263
4.2.5.2.2	How To: Copy Calibration Data from another Drawing	265
4.2.5.2.3	How To: Associate GPS LAT, LON with your Locations	266
4.2.5.2.4	How To: Show Location Markers in your Drawing	269
4.2.5.2.5	How To: Show Panel Markers in your Drawing	270
4.2.5.2.6	How To: Show Backbone Markers in your Drawing	272
4.2.5.2.7	Using Filters	274
4.2.5.2.8	HOW TO: Create a Multistory Isometric Drawing to Display GPS Data	275
4.3	CMS Settings and Options	277
4.3.1	User Settings	278
4.3.2	Project Settings	279
4.3.3	Output Settings	280
4.3.4	Status Behaviour	281
4.3.5	Post Process Scripting	283
4.3.6	GPS Settings	285
5.	Reference	287
5.1	Ribbon Tabs and Dialogs	288
5.1.1	The Workspace	288
5.1.2	Ribbon Tabs	296
5.1.2.1	Application Menu	296
5.1.2.2	Project	297
5.1.2.3	File	298
5.1.2.4	Drawing	299
5.1.2.5	CAD Tools	302
5.1.2.6	Advanced Tools	307
5.1.2.7	View	310
5.1.2.8	Database	312
5.1.2.9	Plugins	314
5.1.2.10	Reports	315
5.1.2.11	Plan View and Layout Tools	315
5.1.2.12	Print Preview	316
5.1.2.13	HTML View	316
5.1.2.14	Report Designer	317
5.1.2.15	Toolbox	317
5.1.2.16	Cable Data Selector	317
5.1.2.17	GPS Mapping Tools For Drawings	318
5.1.2.17.1	GPS	318

5.1.2.17.2	Tools	318
5.1.2.17.3	Display	319
5.1.2.18	GPS Mapping Tools for Map Tool	323
5.1.2.18.1	File	323
5.1.2.18.2	Tools	324
5.1.2.18.3	Display	324
5.2	Dialogs	326
5.2.1	Application Menu Dialogs	326
5.2.1.1	New Project Wizard	326
5.2.1.2	Project Save As	330
5.2.1.3	Manage Security	335
5.2.1.4	Packup / Checkout	337
5.2.1.5	Unpack Project	338
5.2.1.6	Check in Project	339
5.2.1.7	Account Info	340
5.2.1.8	Application Setup Wizard	341
5.2.1.9	The Settings Dialog	346
5.2.1.9.1	Application Settings	348
5.2.1.9.1.1	Application Basic	348
5.2.1.9.1.2	WireCAD Security	349
5.2.1.9.1.3	Global Database Location	350
5.2.1.9.1.4	Support Paths	351
5.2.1.9.1.5	Organization	352
5.2.1.9.1.6	Enable/Disable Plugins	353
5.2.1.9.1.7	Todo List	354
5.2.1.9.2	User Settings	354
5.2.1.9.2.1	User Basic	354
5.2.1.9.2.2	Projects List	356
5.2.1.9.2.3	Drawing	357
5.2.1.9.2.4	Drawing (advanced)	360
5.2.1.9.2.5	Command Line Shortcuts	360
5.2.1.9.2.6	CMS	361
5.2.1.9.2.7	Experience Level	362
5.2.1.9.3	Project Settings	364
5.2.1.9.3.1	Project Basic	364
5.2.1.9.3.2	Advanced	365
5.2.1.9.3.3	Locations	366
5.2.1.9.3.4	Delete Handling	366
5.2.1.9.3.5	Enterprise CMS	367
5.2.1.9.3.6	Export Settings CMS	368
5.2.1.9.3.7	Project Rules	369
5.2.1.9.3.8	Status Behaviour CMS	369
5.2.1.9.3.9	GPS Settings CMS	370
5.2.1.9.3.10	Archive Project	372
5.2.1.9.3.11	Cable Text Search Window	373
5.2.1.9.3.12	Drawing Status Items	375

5.2.1.9.3.13	Project Specific Global Database	377
5.2.1.9.3.14	Folder Sync	378
5.2.2	Project Dialogs	379
5.2.2.1	Publish Project to Portal	380
5.2.2.2	Archive Project	381
5.2.2.3	Titleblock Manager	382
5.2.2.4	ToC	385
5.2.3	Drawing Dialogs	389
5.2.3.1	CAD Dialogs	389
5.2.3.1.1	New Drawing Wizard	389
5.2.3.1.2	Print Preview	390
5.2.3.1.3	PDF Export	392
5.2.3.1.4	Export to Visio	393
5.2.3.1.5	Layers	394
5.2.3.1.6	Layouts	395
5.2.3.1.7	Point Styles	397
5.2.3.1.8	Text Styles	397
5.2.3.1.9	Dimension Styles	399
5.2.3.1.10	Model Space Boundaries	400
5.2.3.1.11	Groups	401
5.2.3.1.12	Images	402
5.2.3.1.13	Image Editor	403
5.2.3.1.14	Rectangular Array	404
5.2.3.1.15	Polar Array	405
5.2.3.1.16	Purge	408
5.2.3.1.17	Inserts Dialog	408
5.2.3.1.18	XREF Manager	409
5.2.3.1.19	Swap Equipment	411
5.2.3.1.20	Swap Pointers	412
5.2.3.1.21	Swap Terminals	412
5.2.3.1.22	Edit Attributes	413
5.2.3.1.23	Add Attribute Definition	414
5.2.3.1.24	X Properties Grid	415
5.2.3.1.25	Edit X Properties	416
5.2.3.1.26	Singleline Text Editor	417
5.2.3.1.27	Multi-line Text Editor	418
5.2.3.1.28	Sync Insert Decision	419
5.2.3.1.29	Validate Deletion	420
5.2.3.1.30	Zoom Scale	420
5.2.3.1.31	Alphanumeric Sequence	422
5.2.3.1.32	Text Increment	424
5.2.3.2	Advanced Tools Dialogs	428
5.2.3.2.1	Equipment Library	428
5.2.3.2.1.1	Find	433
5.2.3.2.1.2	Inputs/Outputs	434
5.2.3.2.1.3	Display Preferences	435

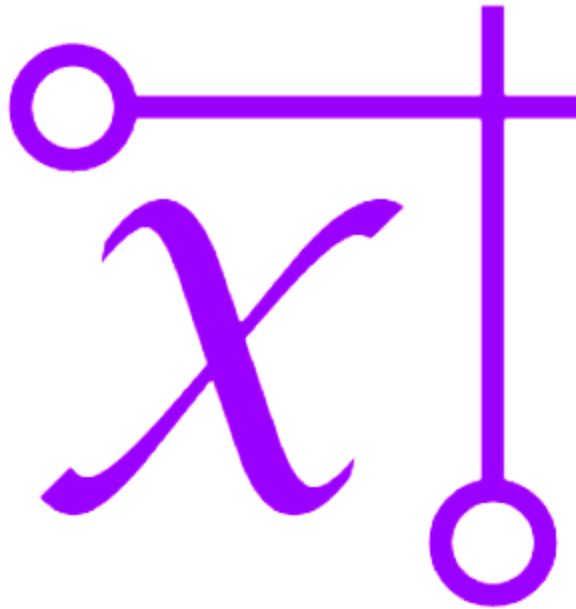
5.2.3.2.1.4	Add Ports	442
5.2.3.2.1.5	Copy Port Data	444
5.2.3.2.1.6	Incoming Signal Types Map	445
5.2.3.2.1.7	Save Equipment As	445
5.2.3.2.1.8	Descriptor Map	446
5.2.3.2.2	Rack Builder Tool	448
5.2.3.2.3	Assign Sysname	449
5.2.3.2.4	Assign Cable Number	451
5.2.3.2.5	Assign Terminals	452
5.2.3.2.6	Assign Terminal No Database Reference	453
5.2.3.2.7	Assign Splice	454
5.2.3.2.8	SysName Error Check	456
5.2.3.2.9	Add Multi-core Cable	456
5.2.3.2.10	Cable Error Check	457
5.2.3.2.11	Auto Block	458
5.2.3.2.12	Ratsnest	461
5.2.3.2.13	WireCADify Block	463
5.2.3.2.14	Add Connection Point	464
5.2.3.2.15	Cable Connection Exists	466
5.2.3.2.16	Add Spare Cable	467
5.2.3.2.17	Error Check Pointers	469
5.2.3.3	Plan View and Layout Tools Dialogs	470
5.2.3.3.1	Take Offs	471
5.2.3.3.1.1	Spare Cable	472
5.2.3.3.2	Location Boundary	474
5.2.3.3.3	Draw Backbone (ENT Only)	475
5.2.3.3.4	Draw Cable	476
5.2.3.3.5	Populate Cables	477
5.2.3.3.6	Draw Prewire	479
5.2.3.3.7	Populate Equipment	480
5.2.4	Data Dialogs	481
5.2.4.1	New Location	481
5.2.4.2	New Cable	483
5.2.4.3	Save Cable Type As	484
5.2.4.4	New Manufacturer	485
5.2.4.5	New Equipment Wizard	486
5.2.4.6	New Cable Type	487
5.2.4.7	Synchronize Global Databases	488
5.2.4.8	Combine Circuits (CMS Only)	490
5.2.4.9	Expression Builder	491
5.2.4.10	Find and Replace	492
5.2.4.11	New Next Number	493
5.2.4.12	Attach Document	494
5.2.4.13	Database Host Settings	495
5.2.4.14	Filter Editor	497
5.2.4.15	New Todo	497

5.2.4.16	Ripple Choice	499
5.2.4.17	Add/Edit Via (CMS Only)	499
5.2.4.18	New Blind Patch	502
5.2.4.19	Titleblock Manager	503
5.2.5	Report Dialogs	505
5.2.5.1	New Report Wizard	506
5.2.5.2	Scan for Discrepancies	506
5.2.5.3	Report Manifest Editor	507
5.2.6	Plugins Dialogs	508
5.2.6.1	Plugin Manager	508
5.2.6.2	Script Editor/Runner	511
5.2.7	PDF Viewer Dialogs	513
5.2.7.1	PDF Print Preview	513
5.2.8	Map Tool Dialogs	514
5.2.8.1	ID GPS Locations	514
5.2.8.2	Print Preview	515
5.3	Forms	517
5.3.1	Application Menu Forms	517
5.3.1.1	Project Information	517
5.3.1.2	Permissions Grid	518
5.3.2	Global Data Forms	519
5.3.2.1	Manufacturers Grid	519
5.3.2.2	Equipment Grid	520
5.3.2.3	Default Signal Types Grid	521
5.3.2.4	Connectors Grid	522
5.3.2.5	Cable Types Grid	523
5.3.2.6	Color Codes Grid	524
5.3.2.7	Pinouts	525
5.3.3	Project Data Forms	526
5.3.3.1	Backbone Grid (CMS ONLY)	526
5.3.3.2	Blind Patch Form (CMS ONLY)	527
5.3.3.3	Circuits Grid (CMS ONLY)	529
5.3.3.4	New Circuit (CMS ONLY)	530
5.3.3.5	New Backbone (CMS ONLY)	530
5.3.3.6	Project Signal Types Grid	531
5.3.3.7	Equipment List	532
5.3.3.8	Cables	534
5.3.3.9	Locations	537
5.3.3.10	Named Paths	538
5.3.3.11	ToDo	539
5.3.3.12	Drawings	540
5.3.3.13	Index Grids	541
5.3.3.13.1	Unassigned Cables	542
5.3.3.13.2	Unassigned SysNames	542
5.3.3.13.3	Cables by Drawing	543
5.3.3.13.4	SysNames by Drawing	543

5.3.3.13.5	Cables by SysName	544
5.3.3.13.6	Orphaned Cables	544
5.3.3.13.7	Orphaned SysNames	545
5.3.3.14	Import Project Data	545
5.3.3.15	Discrepancies	547
5.3.3.16	Next Numbers Grid	548
5.3.3.17	Project Rules	549
5.3.4	Report Forms	550
5.3.4.1	Print Preview	550
5.3.4.2	HTML View	551
5.3.4.3	Report Designer	552
5.3.4.4	Toolbox	553
5.3.4.5	Cable Data Selector	555
5.3.4.6	Bill of Materials Generator	556
5.3.5	PDF Viewer	558
5.3.6	Plugins Forms	559
5.3.6.1	Translation Manager	559
5.3.7	Map Tool	560
5.4	Tool Panels	561
5.4.1	Project Explorer	562
5.4.2	Draw Cables	567
5.4.3	Drawing Properties	568
5.4.4	Find	569
5.4.5	Plan View	571
5.4.6	Favorites	572
5.4.7	Command Line	573
6.	Writing Plugins	575
6.1	Hello World	581
6.2	Getting Started	584
6.3	Registering Your Plugin	585
Index		589

Welcome

1 Welcome

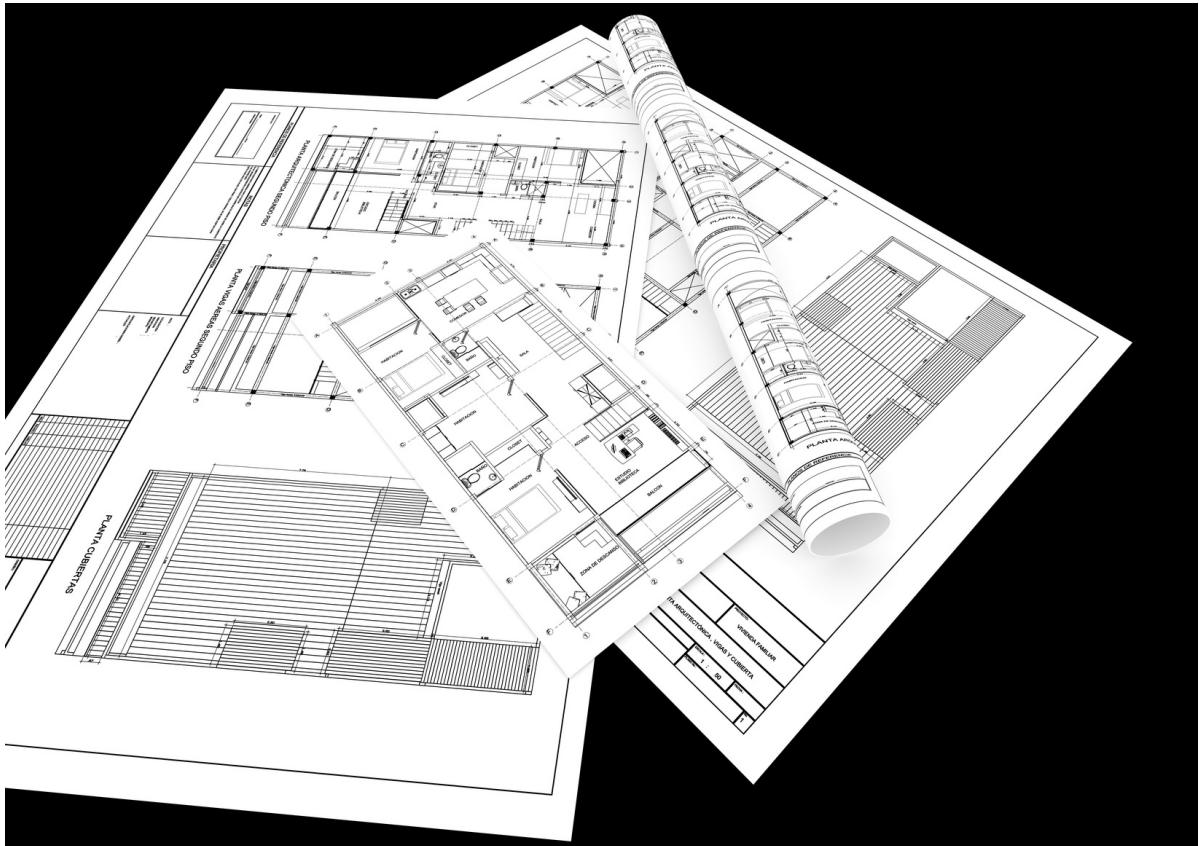


WireCAD v10.1 User Manual

www.wirecad.com

Introduction

2 Introduction



Welcome to WireCAD. WireCAD tools aim to decrease the frustration associated with creating accurate, detailed documentation. WireCAD produces DWG compatible drawings...

Welcome to WireCAD. WireCAD tools aim to decrease the frustration associated with creating accurate, detailed documentation. WireCAD produces DWG compatible drawings accompanied by either VISTADB or SQL Server databases containing all pertinent project data. 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

Things really start to fly when it is time to assign System Names (unique IDs) and Cable Numbers to the equipment in your drawing with our automated tools.

Extensive reporting is available for the project databases including:

- Cable run sheets
- Cable labels
- Project drawings
- Equipment lists
- Bill of Materials
- Power consumption and heat load

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

2.1 New in Version 10

WireCAD v10.1 Changes

WireCAD v10.1 implements a number of security changes; most notably the change from VISTA DB v3 to v6 databases. We did this to remain FIPS compliant. The switch from vdb3 to vdb6 files is seamless with the only drawback being that you cannot use a WireCAD v10.1 project in any earlier version of WireCAD.

We have added the teams.wirecad.com Teams portal with its accompanying Publish Project to Portal tool. All current Assurance members have access to the portal where you can publish projects and invite users to view/redline in any modern browser. Added the Publish Project to Portal tool. You can find it at **Projects>Publish Project to Portal**. This tool prepares and uploads the project to the new WireCAD Team Portal at: <https://teams.wirecad.com>. Create your account with your WireCAD v10 key with active Assurance. From the WireCAD Team Portal you can share your project, its data and drawings with your team. Redline drawings, research and visualize your project data.

WireCAD v10.1 adds project specific global databases. You can, if you choose, relate a global database to a project. This is useful for those that are continually switching global databases to satisfy customer requirements.

- Modified the Attribute Editor to allow modification of additional properties.
- Modified the SysName Assignment dialog to allow selecting of existing SysNames where the make/model match or don't.
- Modified the Inputs/Outputs page of the Equipment Library to support forcing the selection to the top or bottom.
- Added Advanced Tools>Equipment Functions>Next No edit to the drawing UI to allow users to quickly update the next number without having to open the Next Numbers grid.
- Added Advanced Tools>Cable Functions>Next No edit to the drawing UI to allow users to quickly update the next number without having to open the Next Numbers grid.
- Added Advanced Tools>Cable Functions>Show Port Info to allow users to interrogate port information but clicking on the port.
- Modified the Rack Builder dialog to allow exclusion of Systems Involved by adding a checkbox list.
- Modified messaging when a project folder structure is wrong.
- Improved messaging when pointer linking fails.

- Modified the Total Edit Time display on the Start Page and the Project Info page to show the Days, Hours, Minutes, Seconds, Total Hours, Total Minutes.
- Added Table of Contents generator. This can be found on the File menu of any active drawing. This function generates a Table of Contents from the project Drawings table. This is a BETA release of this plugin. This is available in PRO and up.
- Refactored the SysName Format dialog for better stability and usability.
- Refactored the Cable No Format dialog for better stability and usability.
- Added Signalize Group Markers property to the Equipment Library. This property sets whether the Group Markers inherit the color of the pins or the pin names.
- Modified the My Manufacturers list to persist changes to the column structure.
- Reworked the Autoblock tool to align to the the block creation modes.
- Modified the Autoblock tool to use the new Appearance settings.
- Improvements in the Packup and Go function.
- Improvements in the Unpack function.
- Improvements in the Check In function.
- Added Default Single Line of Text Height property to the Drawing Settings. This reduces the number of steps required to enter text in the drawing.
- Added Default MText Height property to the Drawing Settings. This reduces the number of steps required to enter text in the drawing.
- Added the ability to better edit Plan View cables. Add, edit delete and update.
- Removed the Strict naming policy from the Edit Plan View Cables dialog.
- Added a link to the Show Me context menu in drawings that links to the Global Equipment Library for that device.
- Modified the Dimension text height to honor the dwg file format Text Style text height. Removed the Text Height property from the property grid as it is confusing.
- Migrated from VistaDB3 to VistaDB6 databases for speed and FIPS compliance. Please note that the conversion process is automatic. Once you open a project or global database in WireCAD 10.1 you will not be able to revert to WireCAD 10.
- Updated UI components that work better with large displays.
- New drawing engine that opens large drawings.
- Added the ability to set the key on the Serial Keys page of the website to NEVERLOCK. In this mode the key cannot be locked to a machine. Note: in this mode WireCAD requires an internet connection to run.
- Added the ability for a key token to be assigned to a key for better user management. If someone leaves the organization you just change and redistribute a new token.
- Added the ability to disable a Locked license. Note that this only works in WireCAD 10.1 and forward.
- Faster paste operations.
- Implemented post clipboard paste operation to check for an relink pointer that are pasted from another drawing. Works for both On sheet and Off sheet pointers, as well as Copy and Cut operations.
- Added the ability to search grid columns from the Column Chooser.
- Modified the New Project Wizard project Description field to track the project Name field by default.
- Modified the Equipment Library Appearance Settings to allow for custom settings to be saved at the user and project levels. There are no more limits on the numbers of appearance settings you can save. This modification removes existing appearance settings. You will need to remake yours.
- Modified the ColorPicker in the property grid for the drawings and library to match elsewhere.
- Modified the Viewport Frozen Layers select dialog to hide frozen layers from the available layers to select upon initial show of the dialog.

- Modified the Drawing Properties grid to only show selections of 500 objects or less to optimize speed. When a selection set of more than 500 objects is sent to the property grid a message is displayed indicating that the selection set is too large.
- Modified to better support the 3DConnexion SuperMouse button events.
- Updated the New Report Wizard.
- Added filters to the Terminal placement tool panels.
- Added new location based cable run sheet report.
- Added new location based cable pull sheet with cable types and counts.
- Added Find field to Project Explorer.
- Added the ability to control from the Settings dialog whether the Folder Sync mechanism is enabled for SQL projects.
- Added the ability to drag/drop move folders in the Project Explorer

SDK Changes

- Added new event to the Workspace - BeforeUpdateCableText which fires just before the cable number text entities are to be updated handing off the Cables, Polyline, and DrawCableInfo objects for you to set the source and destination texts for the CableNo and the AuxText.
- You will need to recompile any custom plugins.
- Modified the constructors for GridFormBase and GridFormBaseExEx to require that you pass the Workspace to them. This is a breaking change for any of your code that calls these base forms.
- Added a new Select Checked List dialog (frmSelectCheckedList) to the SDK.
- Added a new event: QuerySettingsPanels that is fired upon open of the Settings dialog to load your setting panels that implement the ISettingsPanel interface.
- Modified the EntitySpaces.VistadbProvider dll to support VistaDB 6. Removed the AllowMediumTrust attribute. The assembly will now only allow FullTrust access.

WireCAD v10 Changes

Major Function Changes

- Completely reworked the Equipment Library and the Library SDK.
- Modified the Assign SysName dialog to clarify the different modes of operation.
- Modified the Assign Cables dialog to clarify the different modes of operation.
- Added support for ACAD 2018 drawings.
- Added PDF Import to drawing function.
- Dramatically decreased project load time.
- Dramatically decreased drawing open/save time.
- Dramatically improved full ripple times.
- Faster project switching.
- Added swap Project Location Delimiter function. Allows you to change the LocationDelimiter and ElevationSlotDelimiter variables mid project.
- Modified the Equipment Library to provide port markers. We have provided 38 marker variants and an SDK event to register your own through code.

- Modified the Equipment Library to include Bidirectional ports. If a port is marked as Bidirectional it appears on both the Inputs and Outputs list but can be selected for inclusion in the block from only one side or the other.
- Added AutoWidth property to the display prefs of the Equipment Library.
- Added support for grouped ports
- Implemented round-trip export-import on the project grids.
- Implemented round-trip export-import on the global grids.
- Added Title Block Manager.

Minor Function Changes

- Added Swap Pointer Function. Allows universal change of pointer style.
- Added Swap Terminal Function. Allows universal change of terminal style.
- Added setting to disable progress bar. This helps larger drawings load faster.
- Added Arc Text tool.
- Removed drawing rulers. This was done to stabilize some WIN10 installations that were not displaying the ruler correctly and were throwing exceptions.
- Added project drawing file system watchers for renaming and creating drawings in the active project folder.
- Added splitter bar to Project Systems Grid for better resizing.
- Added the ability to create multiple new drawings from the New Drawing Wizard.
- Modified the Crosshair setting to allow larger sized crosshairs.
- The ModifiedByUser field is now honored in the Equipment Library.
- The ModifiedBy field is now deprecated.
- Removed support for Tags in user defined shapes.
- Moved the tool panel tabs to the left side and added a user setting to allow return to the bottom position.
- Modified the response to missing SysName to better guide the user.
- Added Image field to global Connectors table.

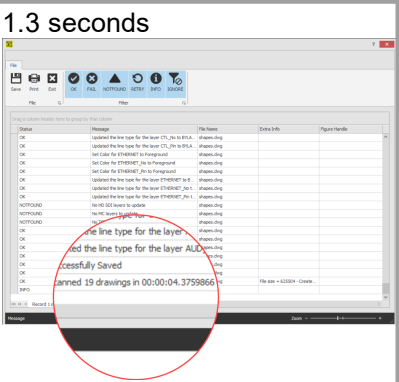
So, let's get started!

Speed

As we mentioned, speed was the watchword for this effort. We have been working much closer to the hardware to take full advantage of multi-core processors and parallel programming techniques to increase the speed of many of the mundane and long playing operations. We now use all processor cores as well as parallel programming techniques to increase program speed.

Below are some of our benchmarks:

Operation	v9 Execution Time	v10 Execution Time
Demo Edit Suite Project Open	30 seconds	2.15 seconds
Edit_1_VID.DWG open	10+ seconds	3.8 seconds
Project Signal Types full ripple	3+ minutes	1.3 seconds



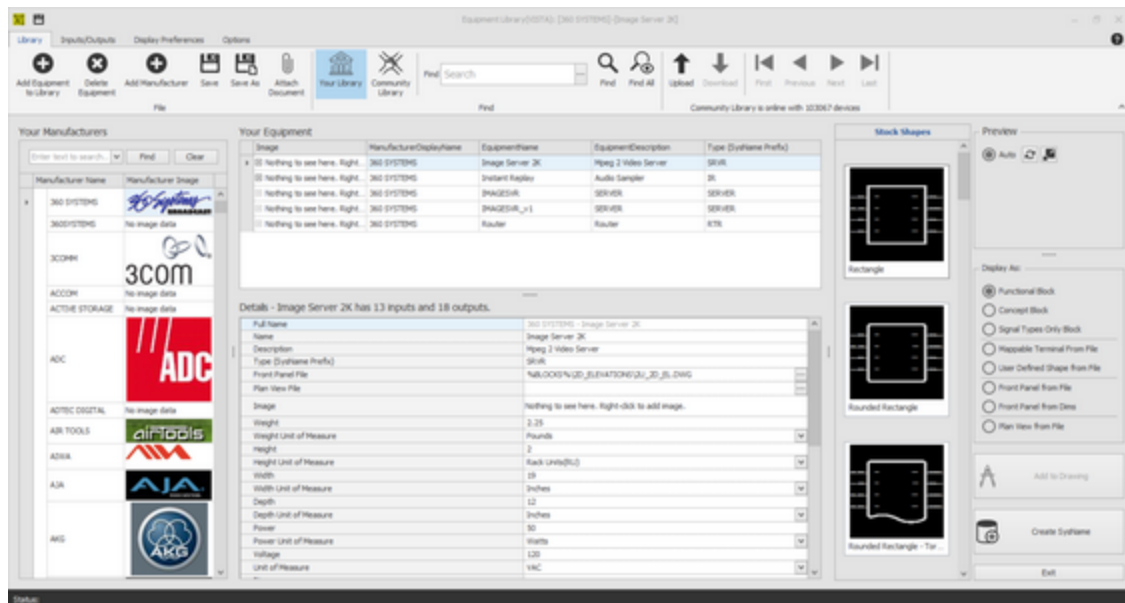
There are similar numbers on all of the ripple operations as well as the drawing save and application close functions.

Equipment Library

We have completely redone the Equipment Library. Completely!

We have made the library faster to load and preview. All of the pertinent information is on the main page. Select a Manufacturer and all of its equipment is shown along with the detail data for the selected device.

We have increased access speed to the Community Library.



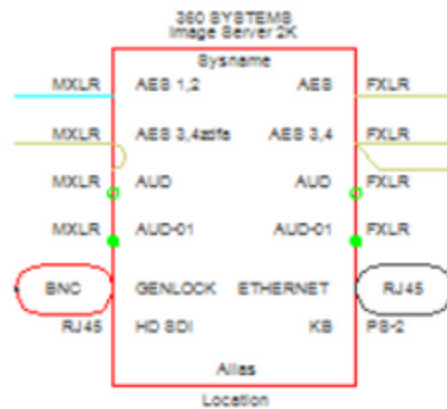
Shapes

The Library now supports many more stock shapes and the ability to code your own in the SDK.



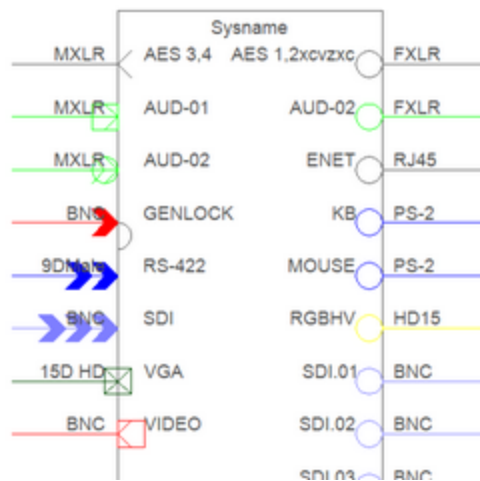
Pin Types

The Library now supports many more pin types and the ability to code your own in the SDK.



Port Markers

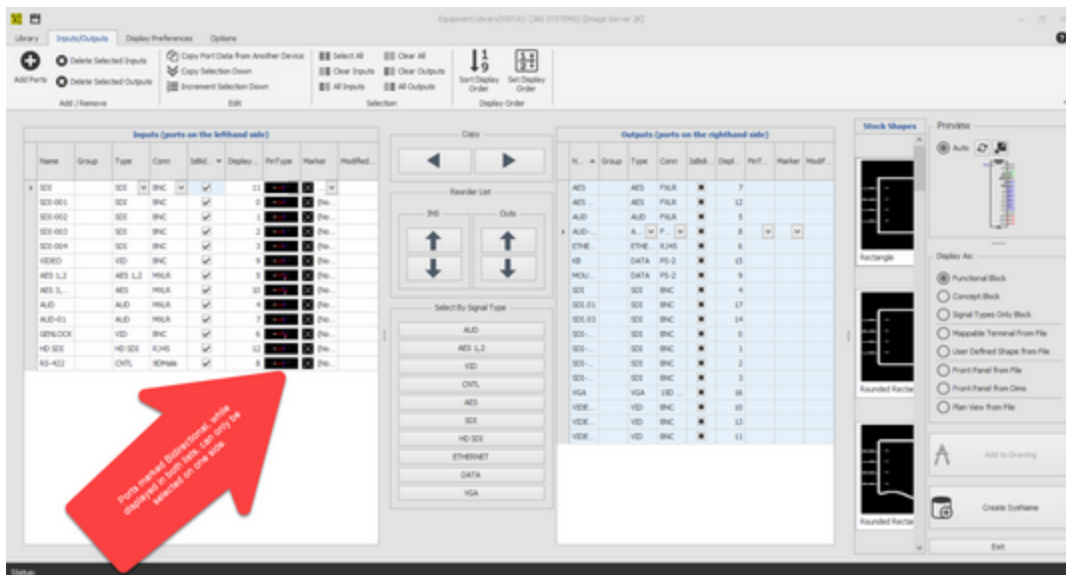
The Library now supports port markers and the ability to code your own in the SDK.



Bidirectional Signal Types

The Library now supports bidirectional signal types. A bidirectional signal type is one that might be displayed on either side of a block depending on document flow.

The new bidirectional port flag is used to ensure that a port so flagged can only be selected once. If it is selected on the input side it cannot be selected on the output side and vice versa.



Port Grouping

The Library now supports port grouping. The groups have several properties that control their appearance:

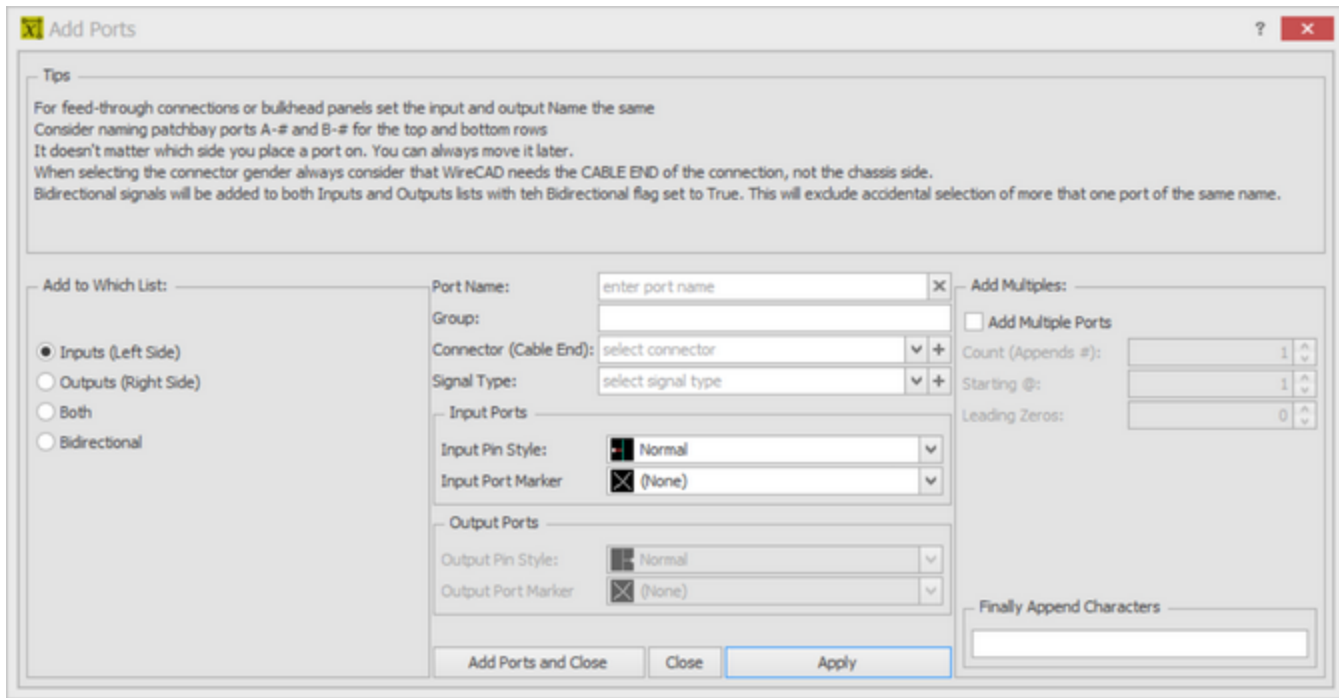
- Input/Output Enable Grouping
- Input/Output Group Spacing - the vertical distance between groups
- Input/Output Group Port Name Concatenation Mode - controls the way the group name is combined with the port name field to represent the combined information in the cables database.
- Input/Output Group Format String - if the **Group Port Name Concatenation Mode = UseFormatString** then the format string will control the port name.
- Input/Output Group Marker



Port Adder Dialog

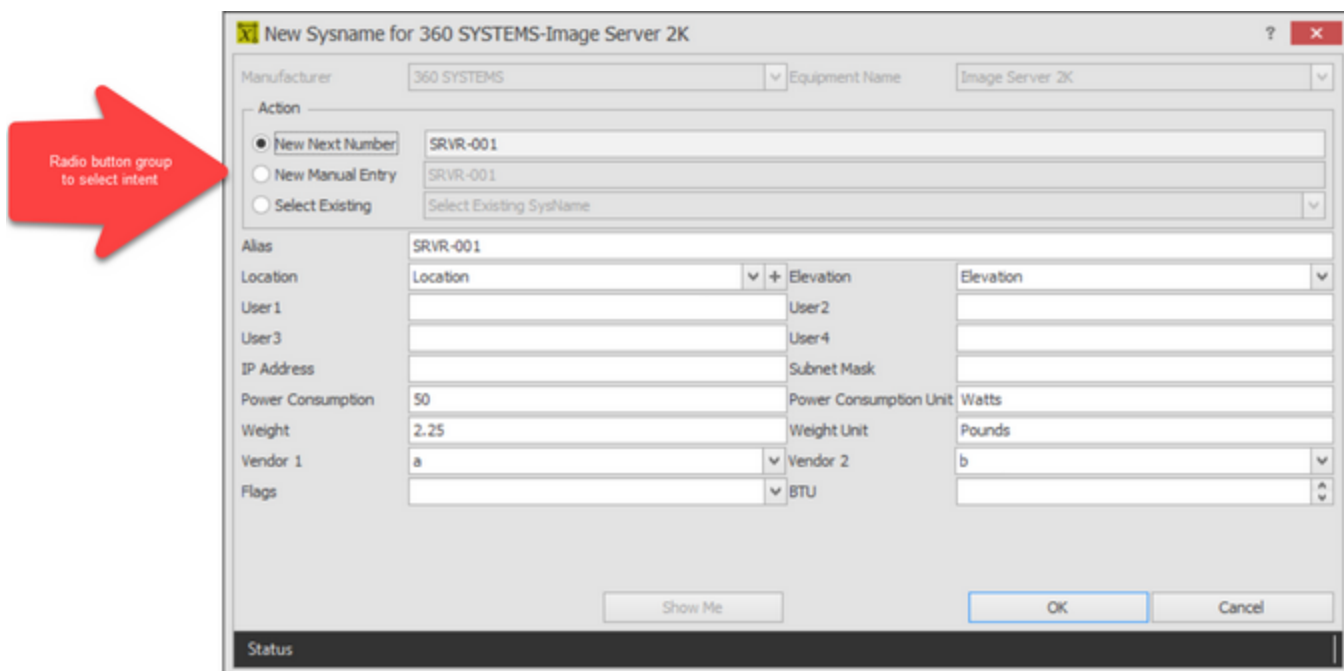
We have reworked the Port Adder dialog to support:

- Bidirectional signal types
- Grouping
- Input Pin Type selection
- Output Pin Type selection
- Input Port Marker Selection
- Output Port Marker Selection



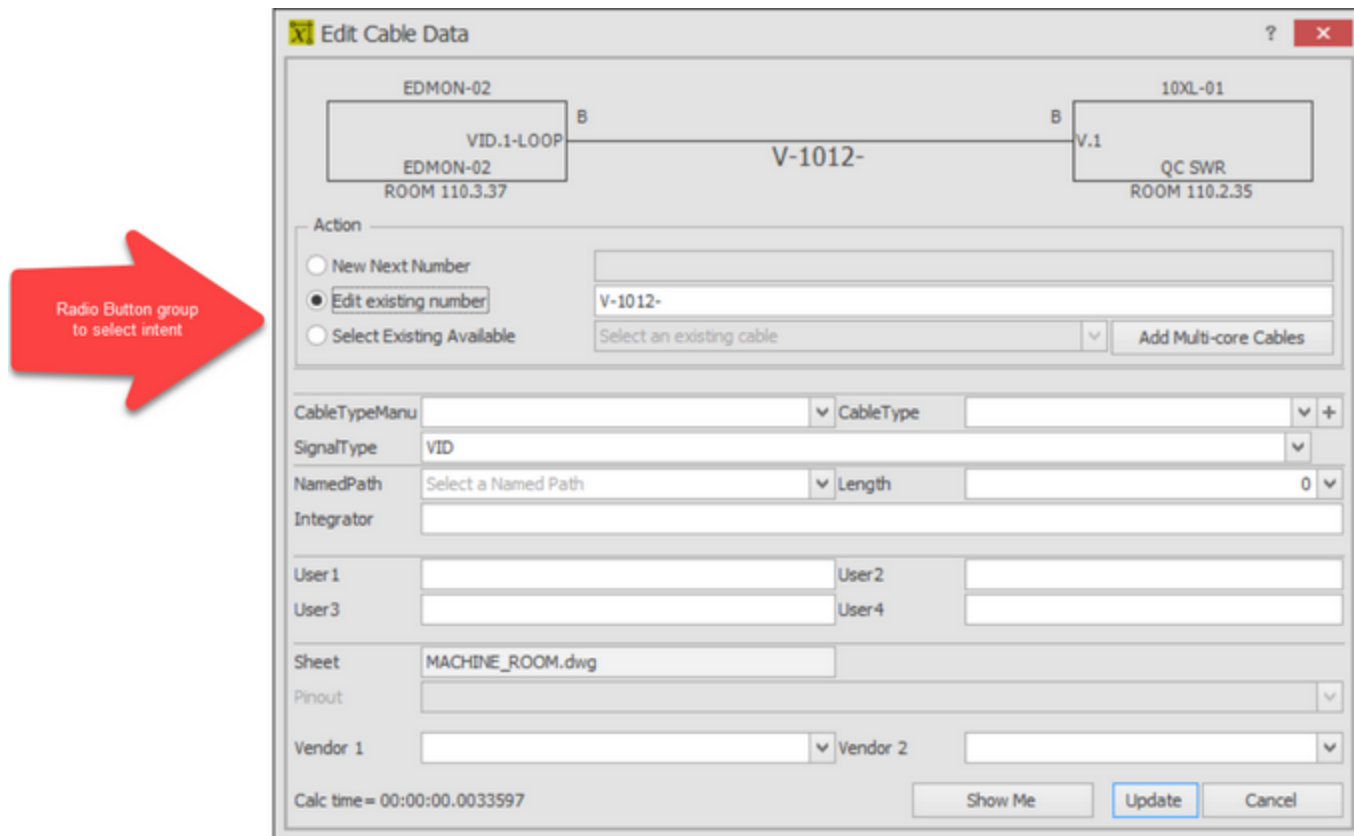
SysName Assignment Dialog

The new SysName Assignment Dialog clarifies the dialog mode via the use of a radio button selection to determine user intent.



Cable Number Assignment Dialog

The new Cable Number Assignment Dialog clarifies the dialog mode via the use of a radio button selection to determine user intent.



Round Trip Export/Import

We have added the ability to export any grid to Excel then modify it in Excel and import the changes back into WireCAD.

In order for this to work you must make sure to export the table's primary key field - usually xxxxGUID where xxxx is the name of the collection that is being exported.

Failing to export the primary key will disable round trip importation and the imported data will be appended to the table.

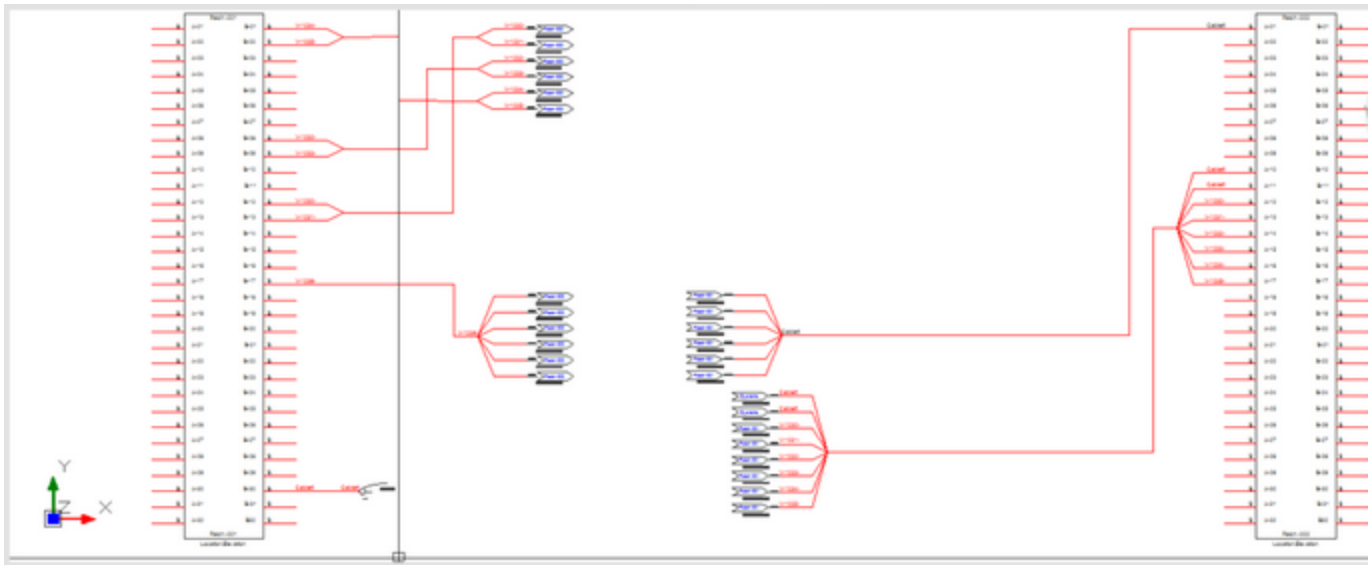
Pointers On Multi-relational Cables

Pointers can now be added to multi-relational cables. The following functions have been enabled:

- Link/Traverse. Double-click to link. [Ctrl]Double-click to traverse. Just like a normal pointer.
- Link Selected Pointers. Link a group of pointers on/off sheet.
- Assign Cable Number. Just like any other cable, double-click the cable to assign.

The following functions have not yet been enabled:

- Reverse linking to a flipped pointer.
- Using a Many-to-One cable to a pointer then pointer to a One-to-Many will not collapse/expand that number of records in the database automatically.

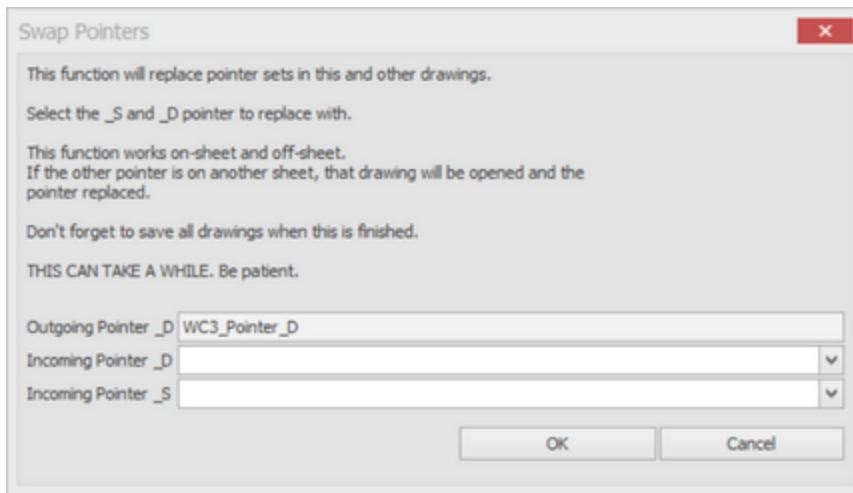


Swap Pointers Function

We have added a long requested feature to allow you to change all pointers of a selected type to that of another.

If you get into the project and decide that something about a pointer does not please your eye you can now change it easily.

Simply select the pointer on the source side of the cable and launch the function. Then select the replacement values. All pointers on and off sheet will be replaced and updated.

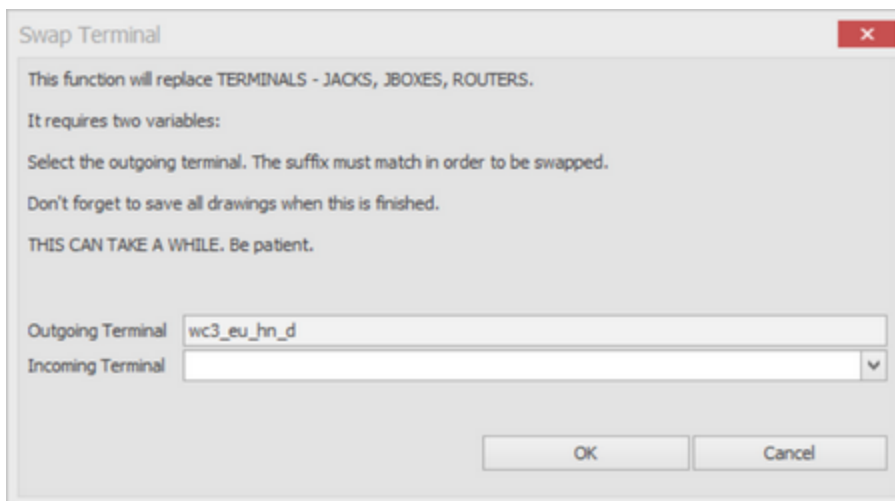


Swap Terminals Function

We have added a long requested feature to allow you to change all terminals of a selected type to that of another.

If you get into the project and decide that something about a terminal does not please your eye you can now change it easily.

Simply select the terminal and launch the function. Then select the replacement value. All terminal on that sheet will be replaced and updated.



Title Block Manager

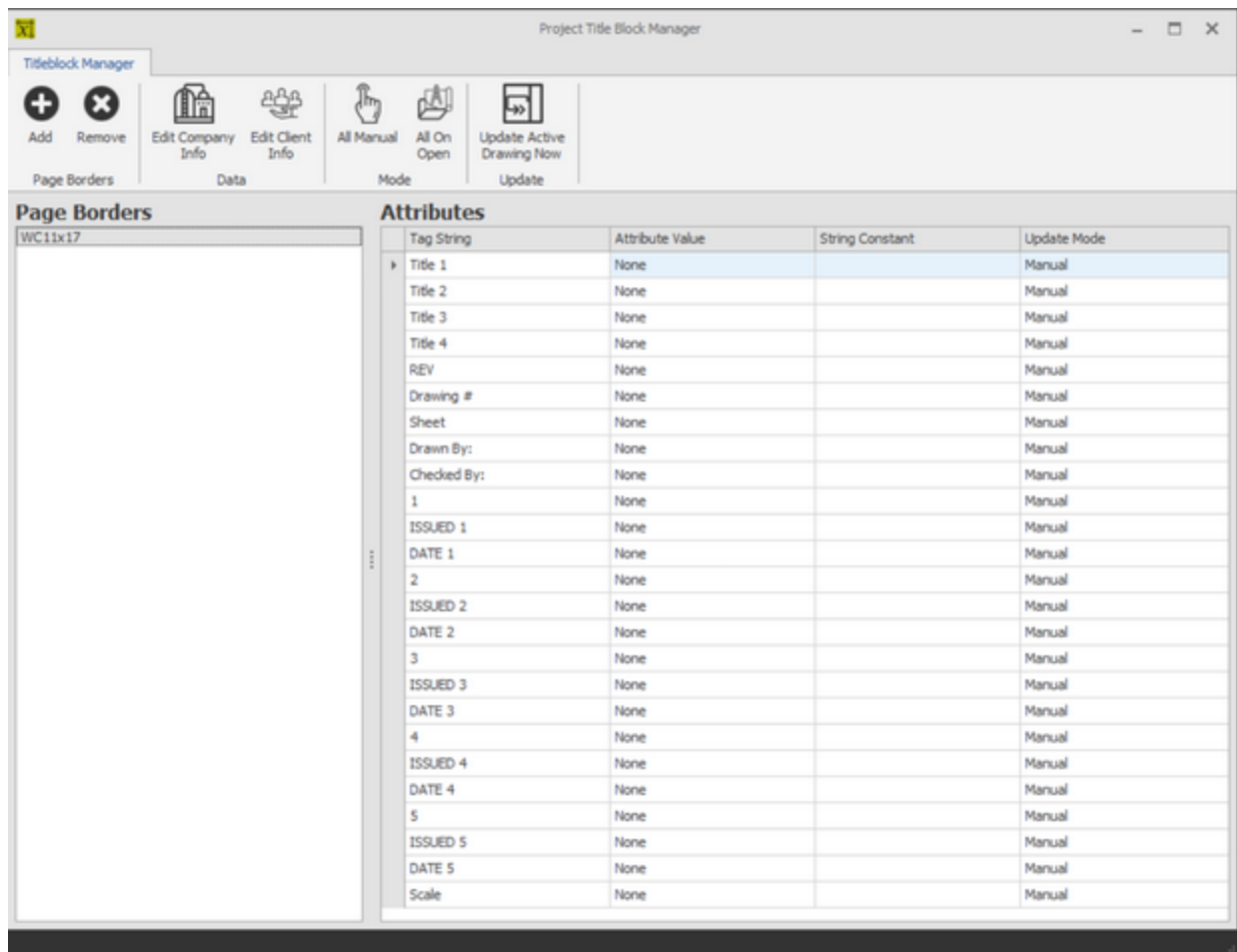
We have added a long requested feature to manage your title blocks throughout the project.

The information is common to all the drawings in the project. You can choose to have the title block data updated manually or on drawing open.

Adding a title block to the tool enumerates the block's attributes, whereupon you assign project variables to the attribute and set the update mode.

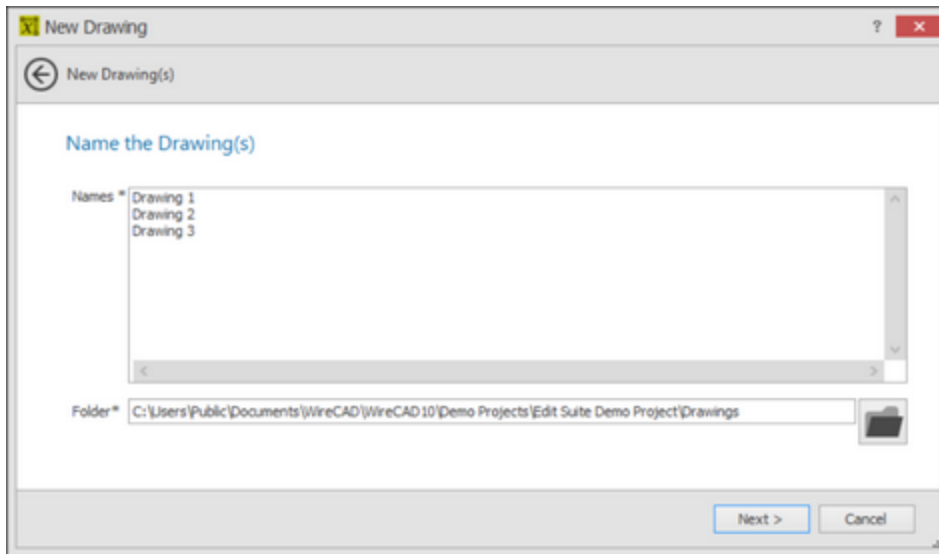
There are many project variables that you can assign to the attributes as well as a string constant value.

In the example below we have assigned the some string constants to the Title1-3 attributes and then assigned project variables to others.



New Drawing Wizard

We have added a long requested feature to allow multiple new drawings to be created from the New Drawing Wizard.



2.2 Software Activation

Navigation: [Introduction](#) ¹⁶ >



WireCAD v10.1 offers 3 program levels with two licensing models:

XLT Perpetual License. This entry level license has several limitations but is a great place to start.

PRO Perpetual License.

PRO Subscription License. This license has a one year term with an expiration date.

CMS Perpetual License. CMS fiber management tools only support the perpetual license model.

WireCAD v10.1 includes server side license management tools that allow you to deactivate a key from www.wirecad.com/serial_keys

From there you can:

- Deactivate a key.
- Set a flag that determines whether a key can lock to a machine or must float.
- Set a token on the key. Consider tokens to be like passwords only they are not encrypted or restricted from view in the interface. These can be use to control who in your organization can use a key. In other words, not only does the user need a key, they need the token as well in order to activate. If someone leaves your organization with the key and you want to restrict their usage, simply add/change the token and distribute the token to everyone else.

An activation key is all that is required to change program levels. If you have questions about the licensing scheme [click here](#) ³⁸.

[Floating Licenses](#) ³¹

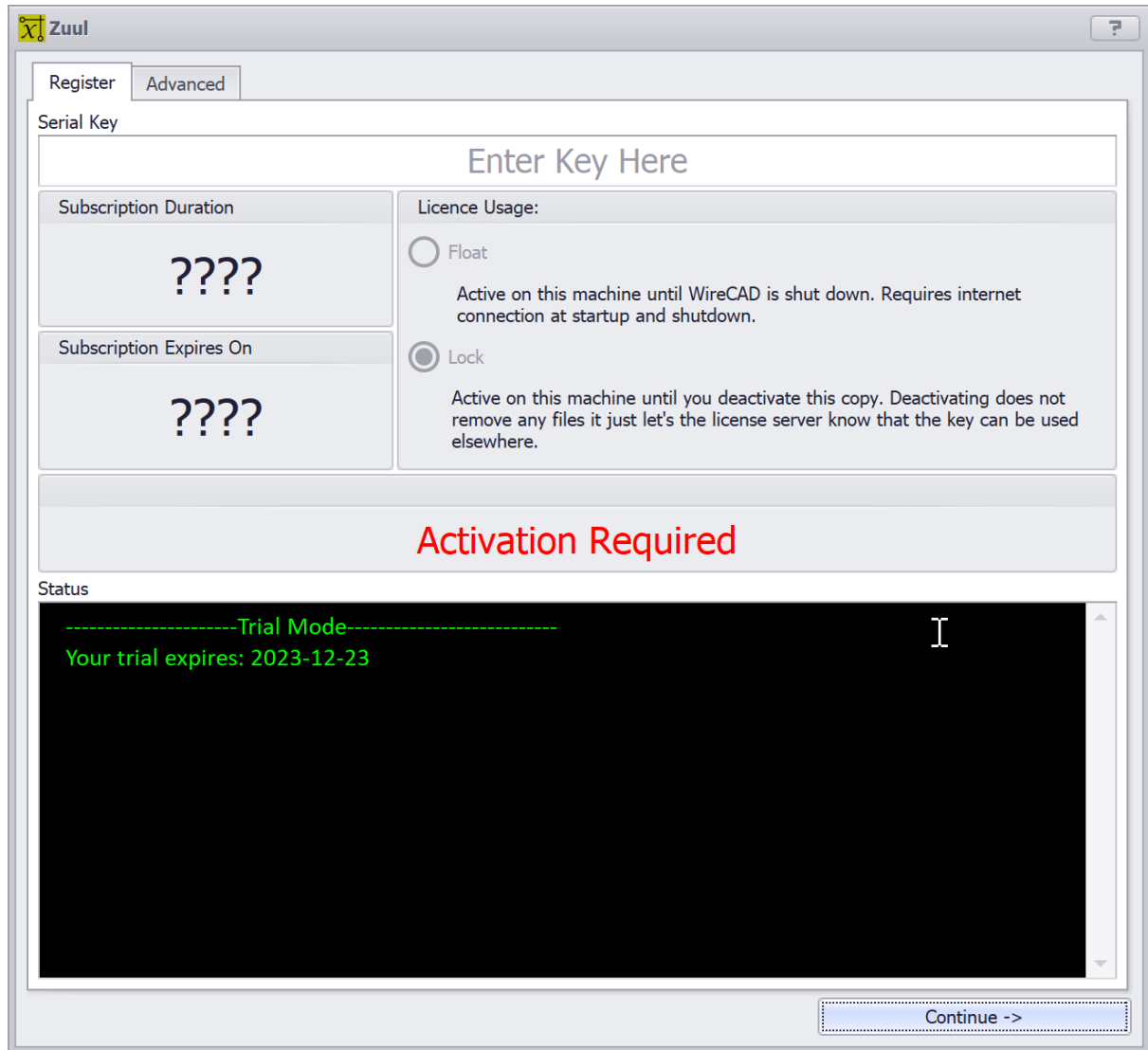
[Floating License Leases](#) ³¹

[How To](#) ³¹

[Troubleshooting Activation](#) 

Activation

Integral to WireCAD is the ability to have a single authorization key activate multiple concurrent machines if your organization has paid for more than one seat. By default, when you purchase a seat of WireCAD your license count for your key will be set to 1 (one). Operational.



The image shows a software activation control dialog box titled "Zuul". It has two tabs: "Register" and "Advanced". The "Register" tab is active. It contains a "Serial Key" field with the placeholder text "Enter Key Here". Below this are two fields: "Subscription Duration" and "Subscription Expires On", both containing "????". To the right of these fields is a "Licence Usage:" section with two radio buttons: "Float" and "Lock". The "Lock" option is selected. Below the "Licence Usage:" section is a red banner that says "Activation Required". At the bottom of the dialog is a "Status" section with a black background and green text that reads "-----Trial Mode-----" and "Your trial expires: 2023-12-23". A "Continue ->" button is located at the bottom right of the dialog.

Software Activation control dialog

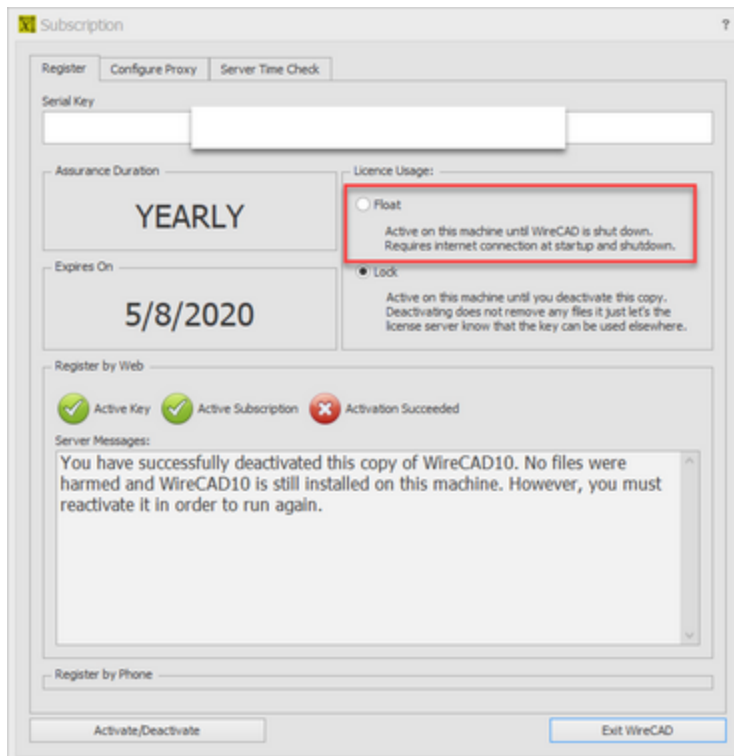
This will allow 1 (one) machine to be active at a time. You may install WireCAD on any number of machines throughout your organization; However, only one (or your license count) machine will be active at any given time.

Floating License Lease

The mechanism by which we float licenses is the license lease. Leases can float or lock to a machine.

In order to change the License Usage you must first Deactivate in order to make the switch between Lock and Float.

NOTE: The point of contact on the account can determine whether the key is allowed to lock to a machine from the www.wirecad.com/serial_keys page.



Once activated the application operates until it is shut down whereupon it released the license(if web connected); thus making the license available to another machine. Upon application startup the license is validated and a lease is acquired automatically if web connected. If you cannot connect to the web you will need to phone WireCAD support at:

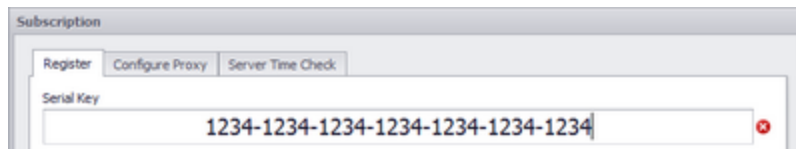
1 661.253.4370 international.

1 866.273.5298 US and Canada toll free.

NOTE: Subscription keys must be activated via the web at least once per subscription period and cannot be activated by phone.

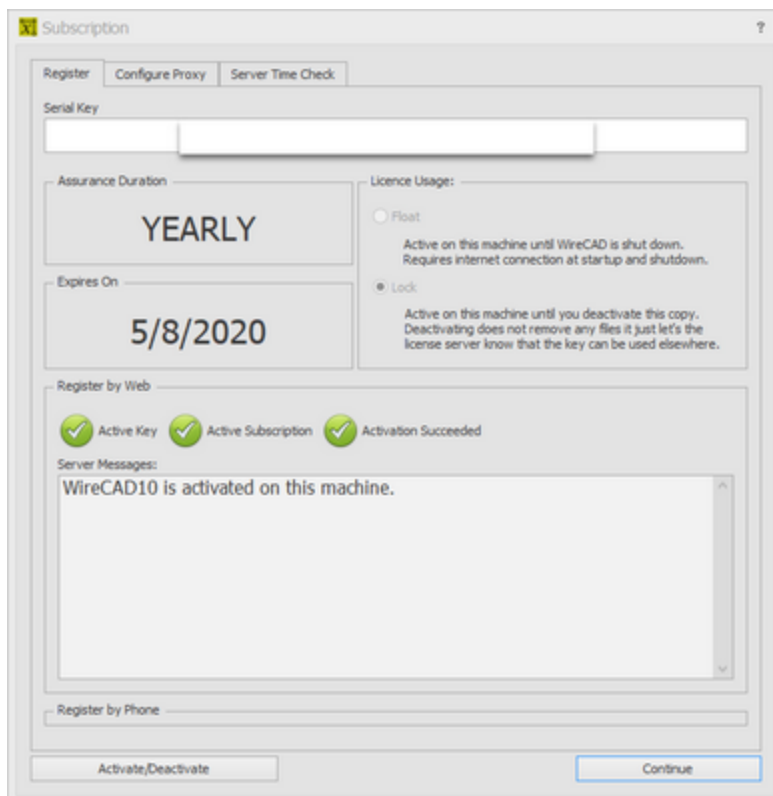
How To: Activate WireCAD

1. Enter your authorization key (it's the really long one that ends in 10. If you fail to enter the key correctly you will not be able to proceed.



2.If you have successfully entered a valid key and you are web connected you will be able to activate by web.

3.Select the Float/Lock mode and click **[Activate/Deactivate]**. If you are successful your screen will look like this:

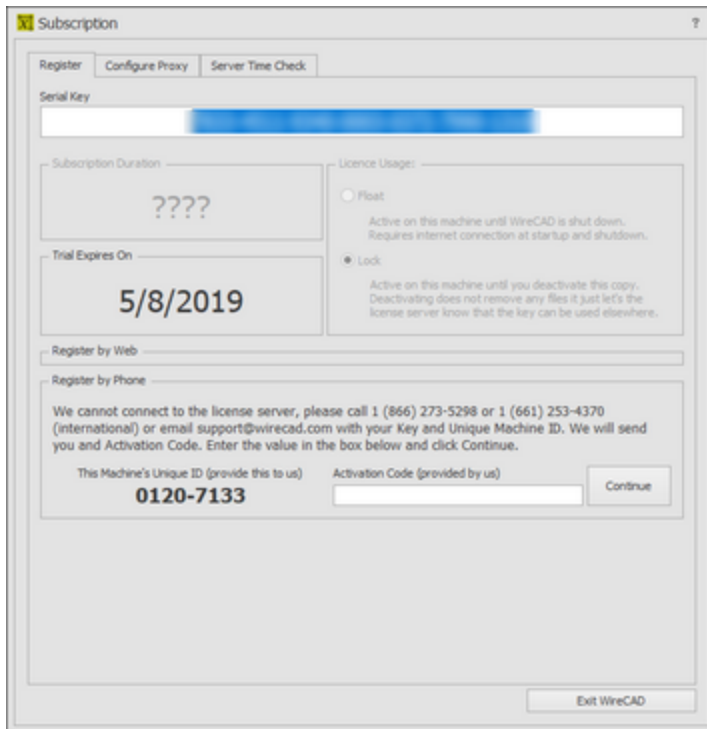


If you are not web connected you will need to call us at:

1 661.253.4370 international.

1 866.273.5298 US and Canada toll free.

If no connection is available you will see the dialog configured like this:



Call the number above. We will need the Machine ID and will read you back an activation code that you will enter in the box.
Then click the **[Continue]** button.

You will receive a message box indicating the **success** of the activation.

Troubleshooting Activation

The following are a few reasons your activation by **web** will fail:

1. Not connected
2. Lease already in use by another machine
3. Authorization key abuse
4. Authorization key not found in the database
5. Your machine Date/Time is more than twenty four hours out of sync with our web server (UTC).
6. License count exceeded.
7. Subscription expired (Applies only to subscription keys).
8. Incorrect Token (if set by POC on the key in the web interface).

The following are a few reasons your activation by **phone** will fail:

1. You have provided an incorrect Machine ID.
2. The WireCAD technician has entered an incorrect value.
3. You have entered an incorrect value in the activation field.

2.3 License Agreement

Navigation: [Introduction](#) 16 >



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.

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..

Automatic Updates

Holbrook Enterprises, Inc. may periodically check the web for updates.
No personal information will be transferred.

LIMITED WARRANTY

Holbrook Enterprises, Inc. warrants for a period of 30 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, but 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.
1112 6th Street South

Nampa, ID 83651
(866) 273-5298 U.S. and Canada
(661) 253-4370 International
support@wirecad.com

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.

2.4 Licensing FAQ

Navigation: [Introduction](#)  >



The following are some frequently asked questions about the WireCAD licensing scheme.

Q: Does WireCAD support floating licenses?

A: Yes, if you choose to use the function it is built in to both PRO and CMS and for subscription keys. You may disable this feature by selecting the Lock option on the License Usage section at activation time. Choosing to do so limits that Authorization Key to that machine only.

Q: How many machines can I install WireCAD on?

A: You may install WireCAD on any number of machines. You will only be able to launch WireCAD on as many machines as your license count supports. The default license count is 1. If you wish to purchase additional licenses you may want to consult with your WireCAD sales professional who will help you decide the best course of action.

Q: I have a laptop and a desktop WireCAD used to let me install on both. How come I have to choose?

A: You don't. WireCAD will still install on both. It will only run on one at a time.

Q: What happens if my machine dies?

A: If your machine dies and you are using the floating license scheme, one of two things will happen:

1. Install WireCAD on the new machine and wait for your lease to expire in whatever lease expiration period you selected (not optimum, but serviceable).

2. Log in to www.wirecad.com/serial_keys and deactivate the lease for the machine that is dead.

Q: I am not connected to the internet very often, can I still use the floating license scheme?

A: We recommend that you use the floating license scheme only if you are regularly connected to the web.

Q: I am going on-site. How do I ensure that my copy of WireCAD will stay active while I am disconnected from the web?

A: If you have already activated WireCAD in Float mode the click the [Activate/Deactivate] button and proceed to deactivate. Then set the License Usage to Lock and Activate.

Q: What are the benefits of the Assurance Subscription?

A: A current Assurance subscription gives you the following premium benefits:

- Free major and minor version upgrades and hot-fixes.
- Priority technical support.
- New samples, tips and how-to topics from time-to-time.
- Discounts on training.
- Access to beta products.
- Assurance Price Lock guarantees that your annual Assurance rate will not increase year-to-year as long as you remain current.

Q: Does the license expire if the Assurance Subscription expires?

A: Depends. If you have been issued a perpetual key, your license does not expire even if your Assurance expires. You can use the products indefinitely even after the Assurance subscription expires.

Q: How long does my Assurance Subscription remain valid?

A: Your subscription duration is for 1 year from date of purchase or renewal.

Q: My Assurance Subscription is about to expire. What should I do?

A: You must renew your subscription to continue to receive the latest versions for free along with all the other benefits of the subscription. To renew your Assurance, contact sales@wirecad.com.

Note that you may or may not receive notifications from Holbrook Enterprises, Inc. dba WireCAD about the pending expiration of your subscription. It is your responsibility to renew your subscription when it is about to expire. You can renew your subscription as early as you want or opt for a monthly credit card payment.

Q: When can I renew my subscription?

A: You must renew your subscription before the expiration of your current subscription. You can renew your subscription anytime before your current subscription has expired; you will not lose any days as the new subscription will come into effect on the day your current subscription ends. In effect, your current subscription will be extended by 1 year.

If you do not renew your subscription before the expiry of your current subscription, your subscription is considered as lapsed and you will not be eligible for free upgrades and other benefits anymore.

Q: My Assurance subscription has lapsed. How can I get the latest version?

A: Maintaining your Assurance subscription and renewing your subscription each year to keep it current is the best and the most cost-effective way to receive all new major and minor versions as they are released. In case your subscription has lapsed and you want to upgrade to the latest version you may simply renew your Assurance subscription at the current rates.

Q: Do you offer academic discounts?

A: We do have academic discounts if WireCAD will be used for education/research purposes. Please contact sales@wirecad.com for more information.

2.5 Context Sensitive Help

Navigation: [Introduction](#)  16 >





WireCAD v10 is tightly coupled to the help file.

When you first start WireCAD we enable **Beginner Mode**. In this mode your moves are reflected in the help system. As you launch functions the help system will display context sensitive help for forms and dialogs.

You can enable/disable Beginner Mode from:
Application Menu>Settings... [User][Expert Mode]

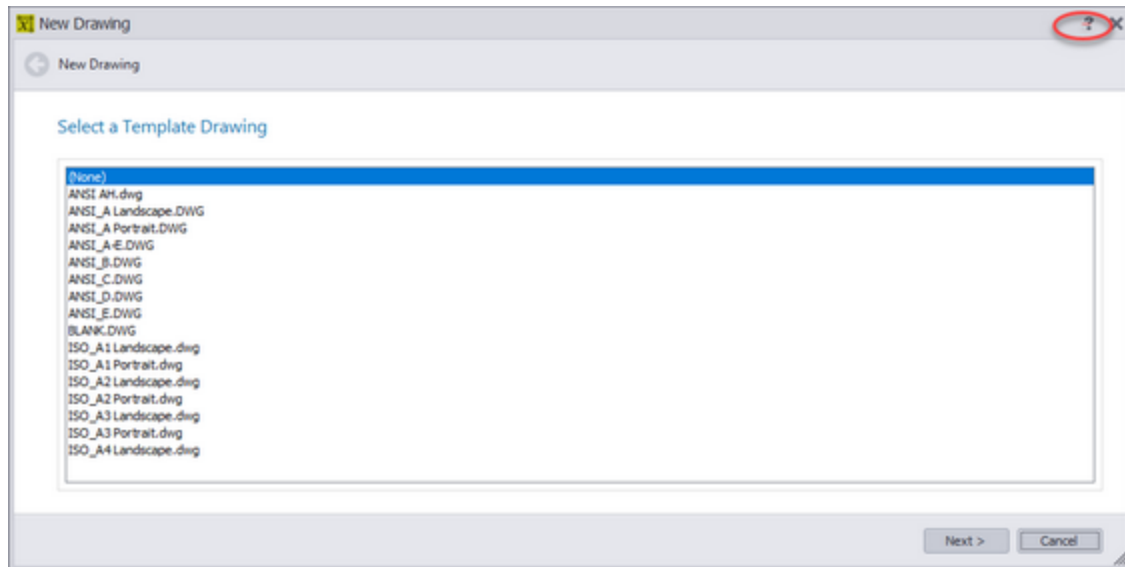
You can get context sensitive help in the following ways from within the application:

2.5.1 Dialog Based Help

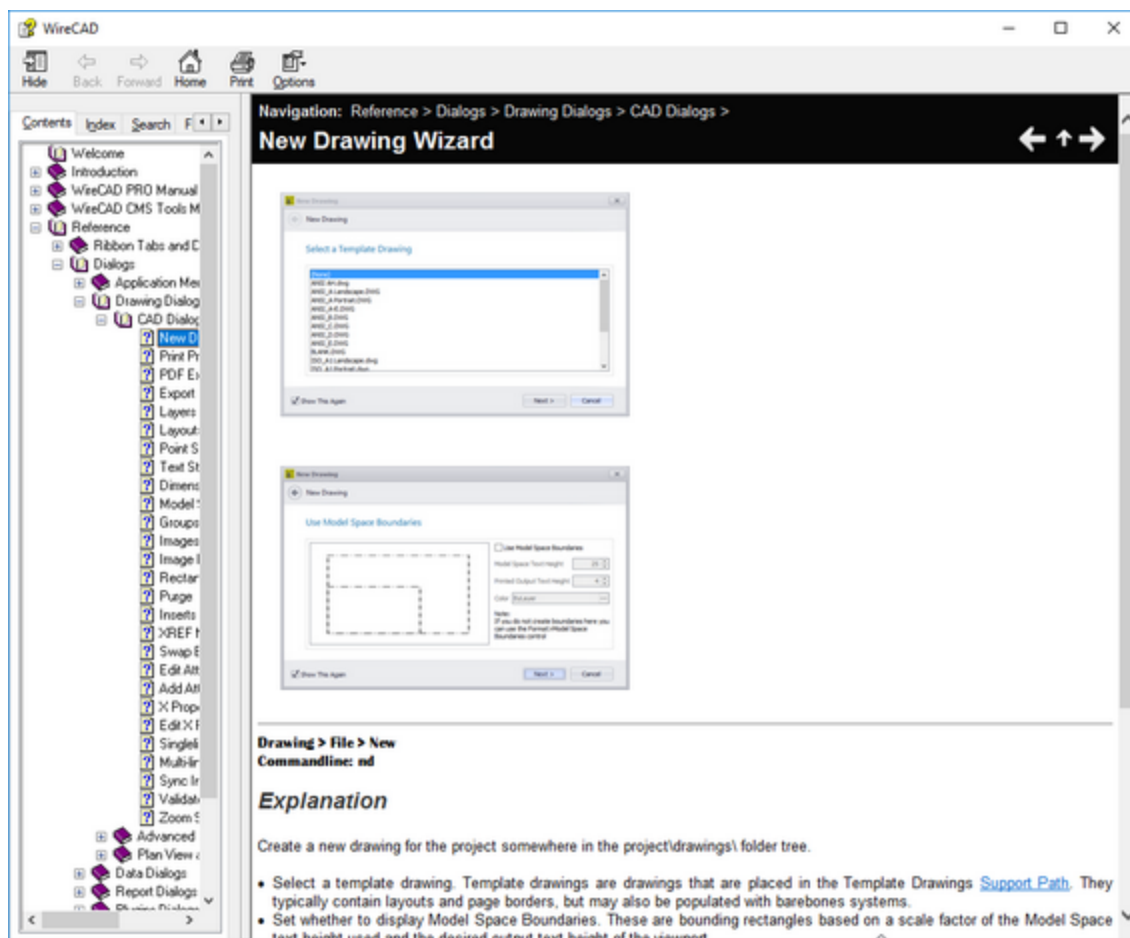
Navigation: [Introduction](#)  16 > [Context Sensitive Help](#)  40 >



Each dialog in WireCAD v10 has a ? in the upper right-hand corner.



Clicking this will open the help file to a section specific to that dialog.

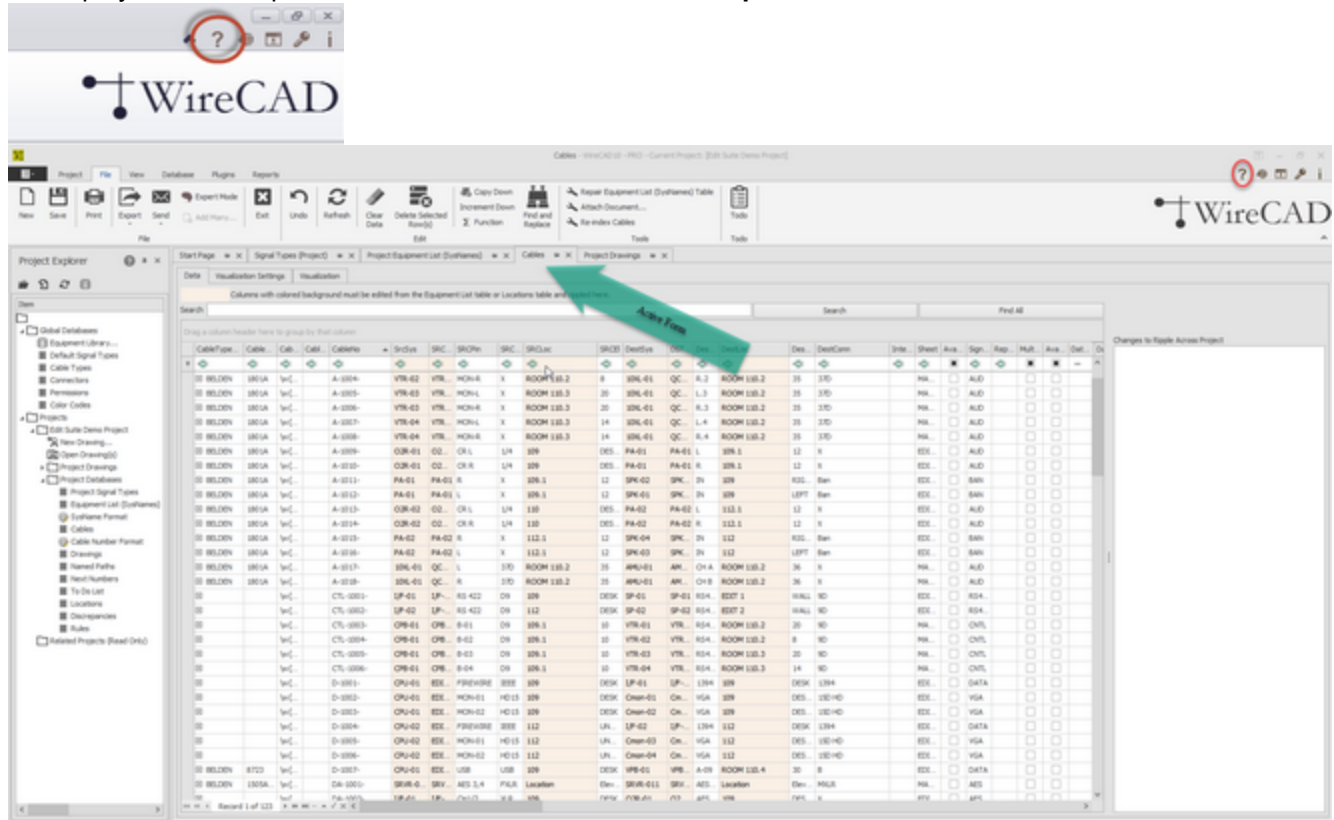


2.5.2 Form Based Help

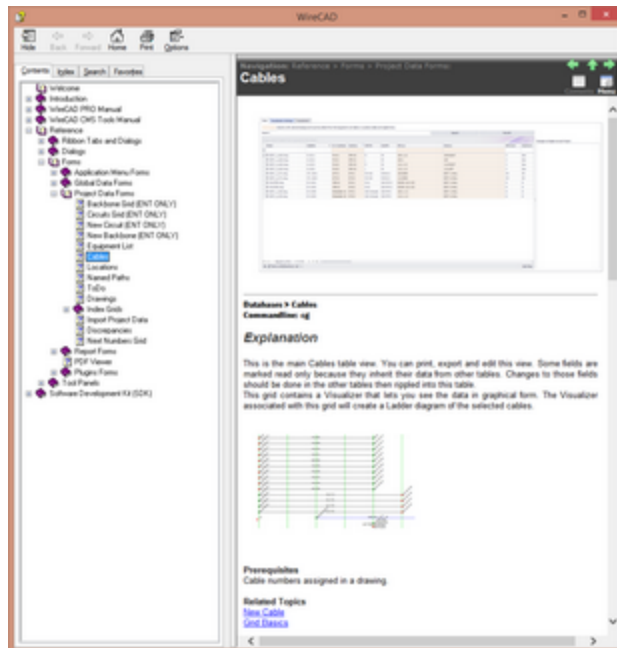
Navigation: [Introduction](#) ¹⁶ > [Context Sensitive Help](#) ⁴⁰ >



To display context help for the active form click **F1** or the **Help** menu item



For example clicking the **Help** menu when the **Cables** database is the active form. Displays the following help topic.

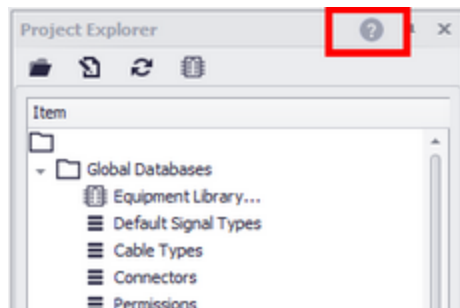


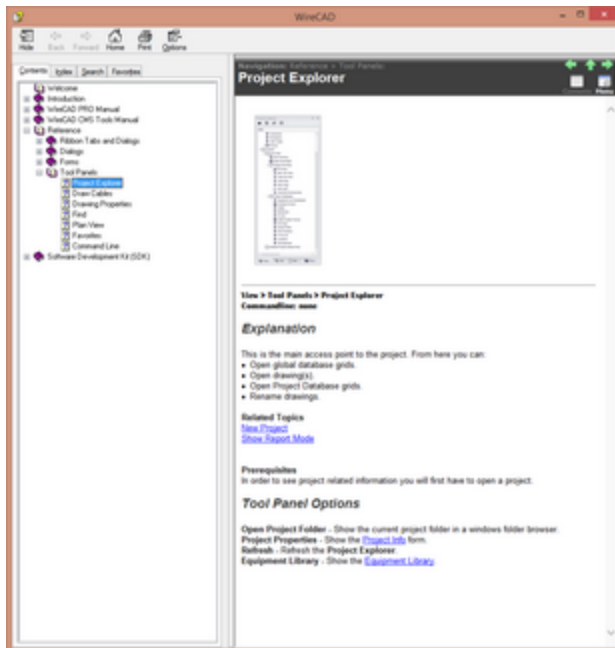
2.5.3 Tool Panel Based Help

Navigation: [Introduction](#) ¹⁶ > [Context Sensitive Help](#) ⁴⁰ >



The same follows for tool panels. Click the ? icon to show a help topic specific to that tool panel.



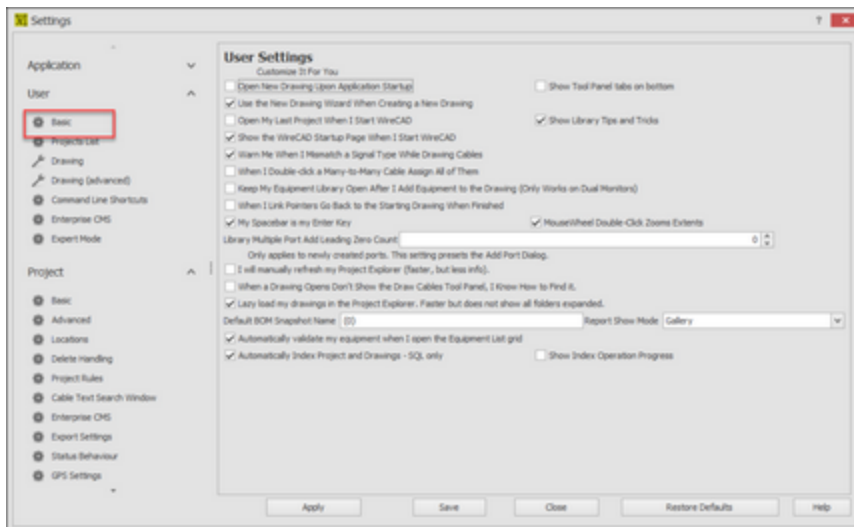


2.5.4 Settings Panel Based Help

Navigation: [Introduction](#) ¹⁶ > [Context Sensitive Help](#) ⁴⁰ >



When the Settings dialog is shown you can get help for each panel by clicking the **[Help]** button.



Clicking the help button will show context specific help for the current settings panel. In the case below we see the help for the **[User][Basic]** panel.

Navigation: Reference > Dialogs > Application Menu Dialogs > The Settings Dialog
> User Settings:
User Basic

Settings

Application: User

User Settings

Customize It For You

- ☐ Open New Drawing Upon Application Startup
- ☒ Use the New Drawing Wizard When Creating a New Drawing
- ☒ Open My Last Project When I Start WireCAD
- ☐ Show Library Tips and Tricks
- ☒ Show the WireCAD Startup Page When I Start WireCAD
- ☒ Warn Me When I Mismatch a Signal Type While Drawing Cables
- ☐ When I Double-Click a Many-to-Many Cable Assign All of them
- ☐ Keep My Equipment Library Open After I Add Equipment to the Drawing (Only Works on Dual Monitors)
- ☐ When I Link Printers Go Back to the Starting Drawing When Finished
- ☒ My Separator is my Enter Key
- ☒ Mousewheel Double-Click Zooms Extends
- Library Multiple Port Add Loading Zero Count
- Only applies to newly created parts. This setting prevents the AddPort Dialog.
- ☐ I will manually refresh my Project Explorer (Faster, but less info).
- ☐ When a Drawing Opens Don't Show the Draw Cables Tool Panel, I know how to find it.
- Default BOM Snapshot Name: (2)
- Report Show Mode: Project/Explorer
- ☐ Automatically validate my equipment when I open the Equipment List grid
- ☒ Automatically Index Project and Drawings
- ☒ Show Index Operation Progress

Save Close Restore Defaults

These are settings for your user profile on this machine.

Basic Panel Options

- Open New Drawing Upon Application Startup** - Opens a blank new drawing when WireCAD starts.
- Use the New Drawing Wizard** - Shows the **New Drawing Wizard** when you click **File > New**. Allows the use of template drawings. If set to false, just creates a new drawing without a template.
- Open My Last Project When I Start WireCAD** - Behaves like earlier versions of WireCAD that opened the last project automatically on startup.
- Show Library Tips** - This shows the Tips and Tricks form in front of the **Equipment Library**.
- Show the WireCAD Startup Page**
- Warn of Signal Type Mismatch** - When you finish drawing a cable WireCAD will check the source and destination port signal types and let you know if they don't match. You can use the new status panel to know the port type if you are confused.

USB IREWIRE HERNET FIBRE JTER

IEEE802.1001 RJ45 SFP

This page is intentionally left blank.
Remove this text from the manual
template if you want it completely blank.

WireCAD PRO Manual

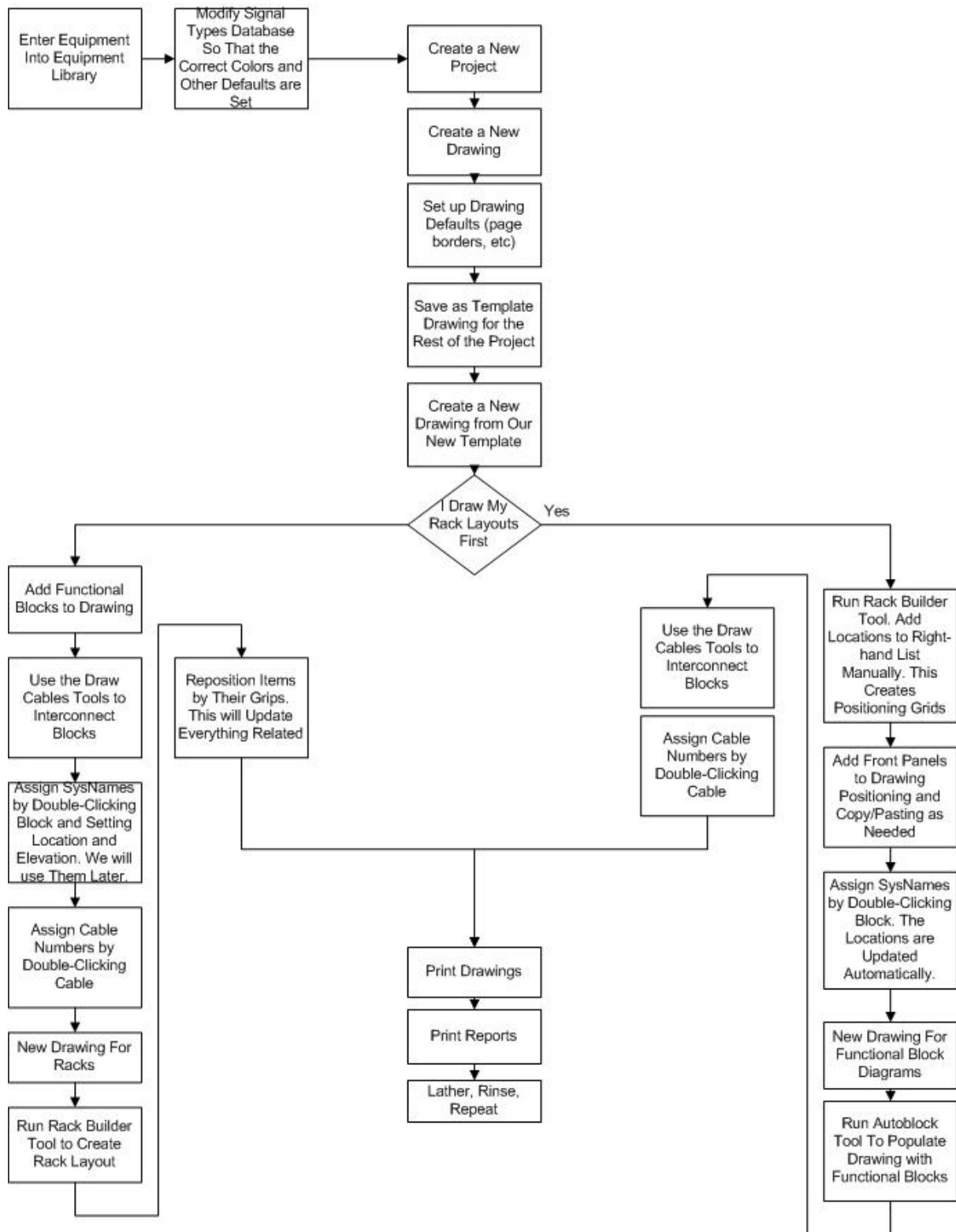
3 WireCAD PRO Manual



3.1 Getting Started

Navigation: [WireCAD PRO Manual](#) 48 >





3.1.1 Grid Basics

Navigation: [WireCAD PRO Manual](#) > [Getting Started](#) >

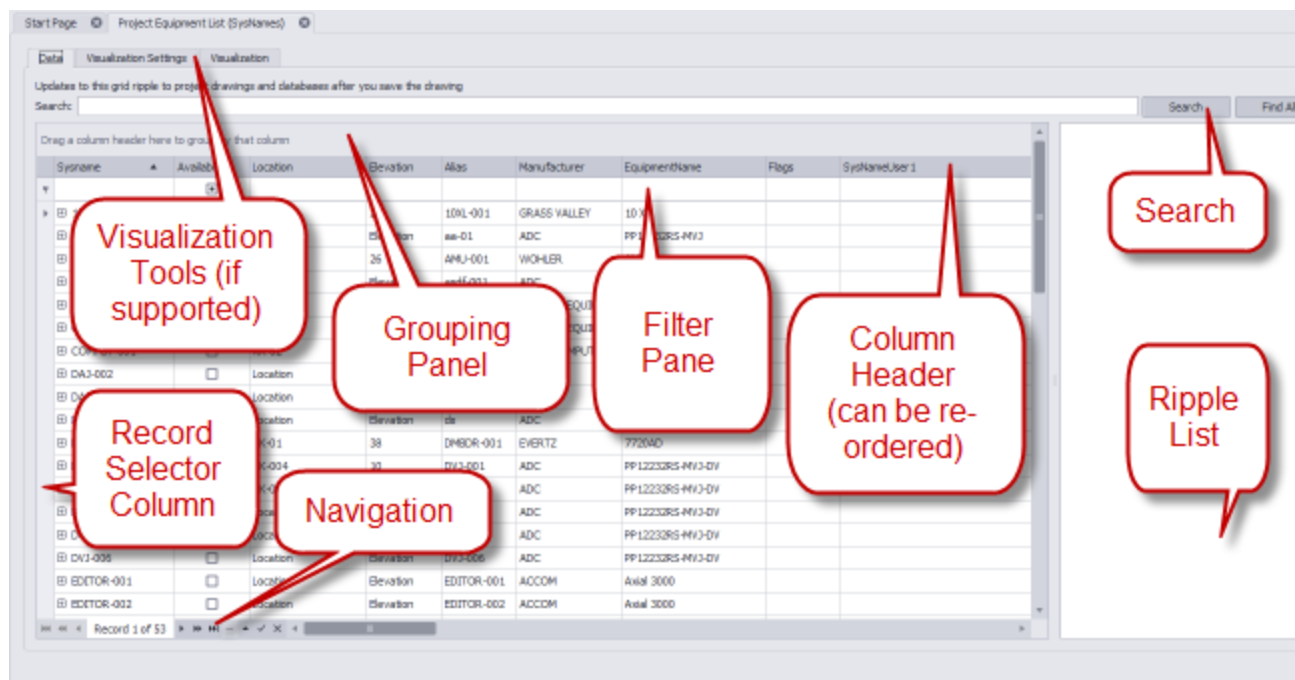


Menu: **various**

Default command line shortcut: **various**

WireCAD displays data from the various databases in grid form. WireCAD grids support the following:

- Searching
- Sorting
- Column re-ordering
- Column hiding/showing
- Column grouping
- Column resizing
- Column filtering
- Copy Selection Down
- Increment Selection Down
- Hierarchical display
- Export to PDF, EXCEL, TXT, HTML, XML and more In addition, several grids support WireCAD Visualization Tools.
- Expression Execution.



Grid Parts

3.1.1.1 How To: Grid Functions

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Getting Started](#) ⁴⁸ > [Grid Basics](#) ⁴⁹ >



•**Grid Searching** - Enter your search text in the **Search** box and click **[Search]**. WireCAD will search all fields in the table for the search term. The % character is the wild card.

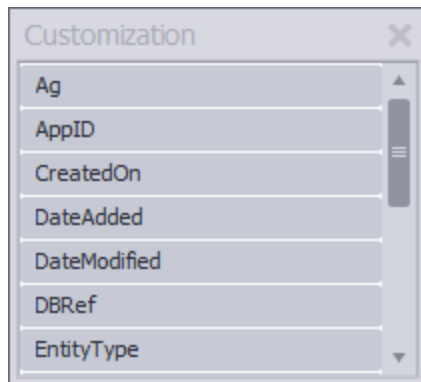
•**Sort** - Clicking on a column header will cause a sort. Clicking it again will reverse the sort order.

•**Column re-ordering/resizing** - Drag a column from its center to initiate a move. Drop it where you want it to display. To resize a column, drag its edge left or right.

•**Column hiding/showing** - To hide a column: right-click the column header and choose **[Remove this Column]** from the context menu.

To show a column:

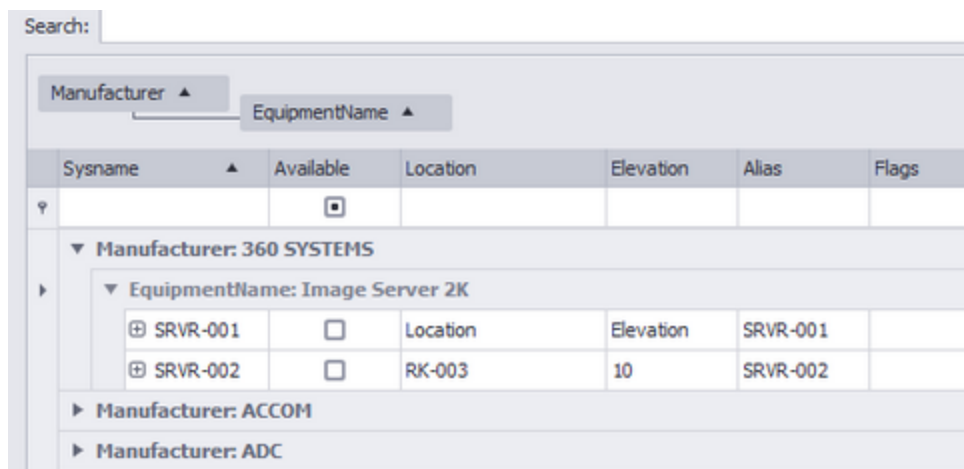
- 1.Right-click any column header
- 2.Click the **[Column Chooser]** context menu item



The Customization form displays hidden column headers

3.Drag the column header you wish to display and drop it on the header bar with the other column headers.

•**Column grouping** - If the grid supports grouping the grid will display a GroupBy panel at the top indicating that column headers may be dropped there to group.

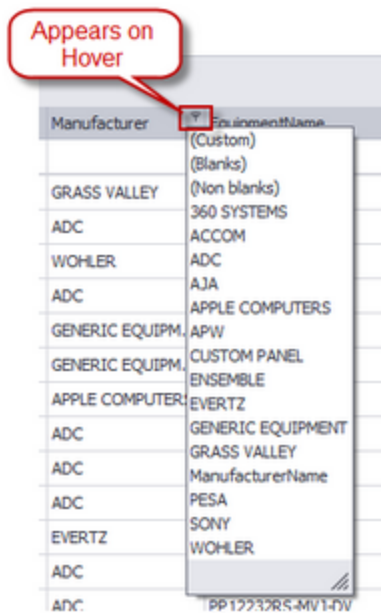


Grid grouped by Manufacturer and EquipmentName fields

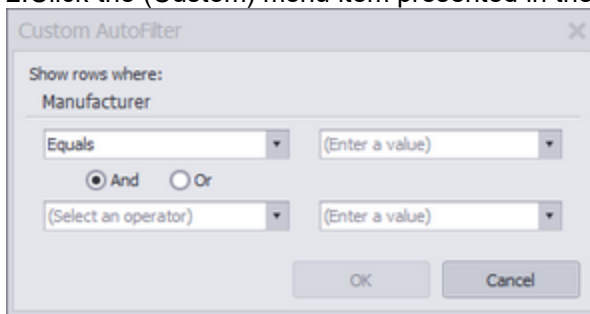
To un-group simply drag the column header back to the header bar.

•**Column filtering** - Columns may be filtered in one of three ways:

- 1.Click the column header filter icon on the column header. You will be presented with a list of unique values found in that column.

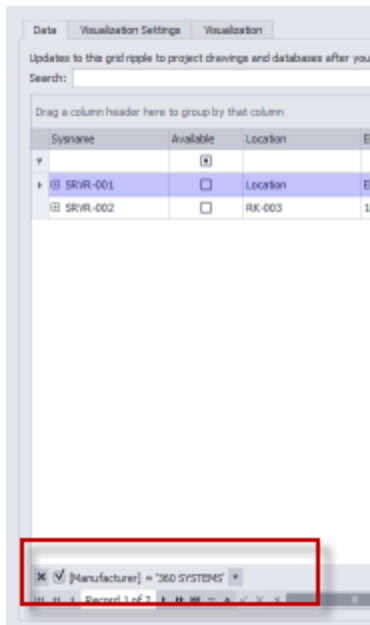


2. Click the (Custom) menu item presented in the column filter list.

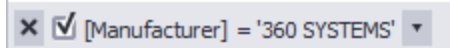


3. If the grid supports it, a filter pane will appear as the top row of the grid. Entering filter criteria in the desired column will cause filtering.

•**Clearing Filter Criteria** - If a filter has been applied to a grid, a filter panel will display on the grid in the lower left hand corner.



You may clear the filter by closing **[X]** the filter panel.



•**Copy Selection Down** - If the grid supports it you will be able to copy the top item in a selection down into all of the selected cells below it. Follow these steps:

1. Create your selection
2. Click **Edit>Copy Selection Down** or press **[Ctrl]+[D]** on your keyboard.
3. Be sure to click **File>Save** to commit your changes to the database.

•**Increment Selection Down** - If the grid supports it you will be able to increment the top item in a selection down into all of the selected cells below it. Follow these steps:

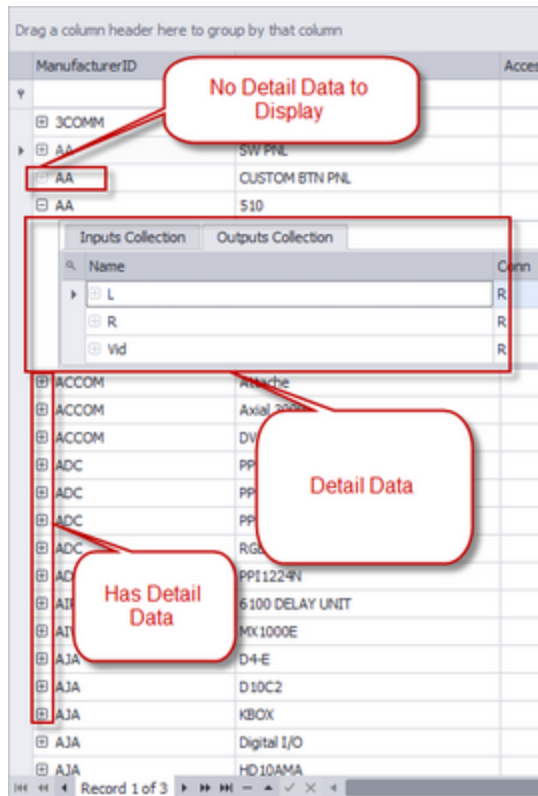
1. Create your selection
2. Click **Edit>Increment Selection Down** or press **[Ctrl]+[I]** on your keyboard.
3. Be sure to click **File>Save** to commit your changes to the database.

•**Deleting Cell Data** - To delete the contents of the selected cells simply click the **[Del]** key on your keyboard.

•**Deleting Selected Rows** - To remove the rows from the grid:

1. Select the rows to delete by clicking on the Record Selector (left-most column without data).
2. Click **Edit>Delete Selected Rows** or click **[Ctrl]+[Del]** on your keyboard.
3. Be sure to click **File>Save** to commit your changes to the database.

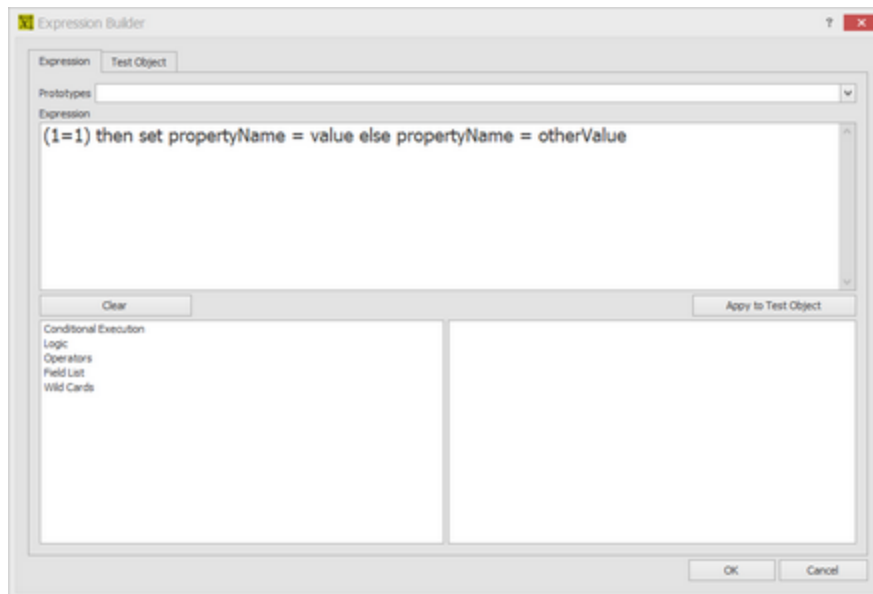
•**Display Hierarchical Detail Data** - If the grid supports it and there is detail data to display you can click the **[+]** button in the left-most row.



•**Export** - Click **File>Export>[File Type]**. WireCAD will then export the data with your current grouping, sorting, and filtering applied to the selected format.

•**Execute Expression** - Manipulate data using the expression engine:




1. Open the grid on which you want to manipulate or transform data.
2. Click **[Function]** button to show the **Expression Builder**.



3. Enter an expression to evaluate. You can test it against a prototype Test Object on which you can set the data before testing.

4. Once happy with the expression click **[OK]** to execute the expression against the grid data.

3.1.1.2 How To: Round Trip Export/Imptr on all Grids

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Grid Basics](#)  >



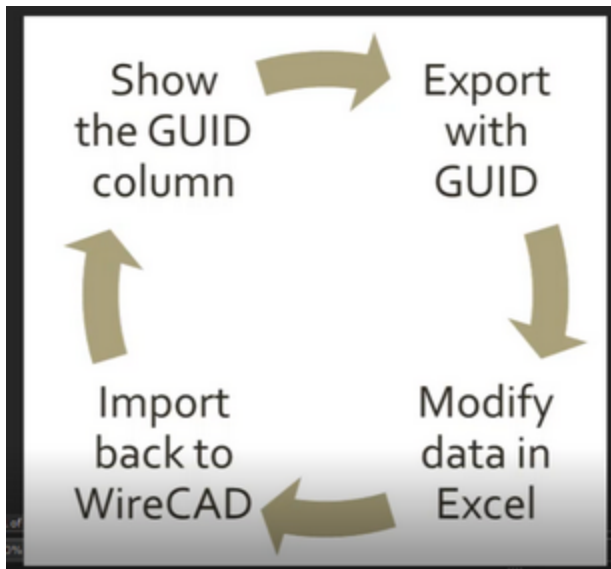
All project and global database grids support the concept that they can be exported to Excel and re-imported into WireCAD.

Users can do a blind import of data ignoring the data that is already in place, or you can do a round trip export/import process.

If the user is using the round trip process the user **MUST** export the table's GUID (Global Unique Identifier) so that there is a database reference.

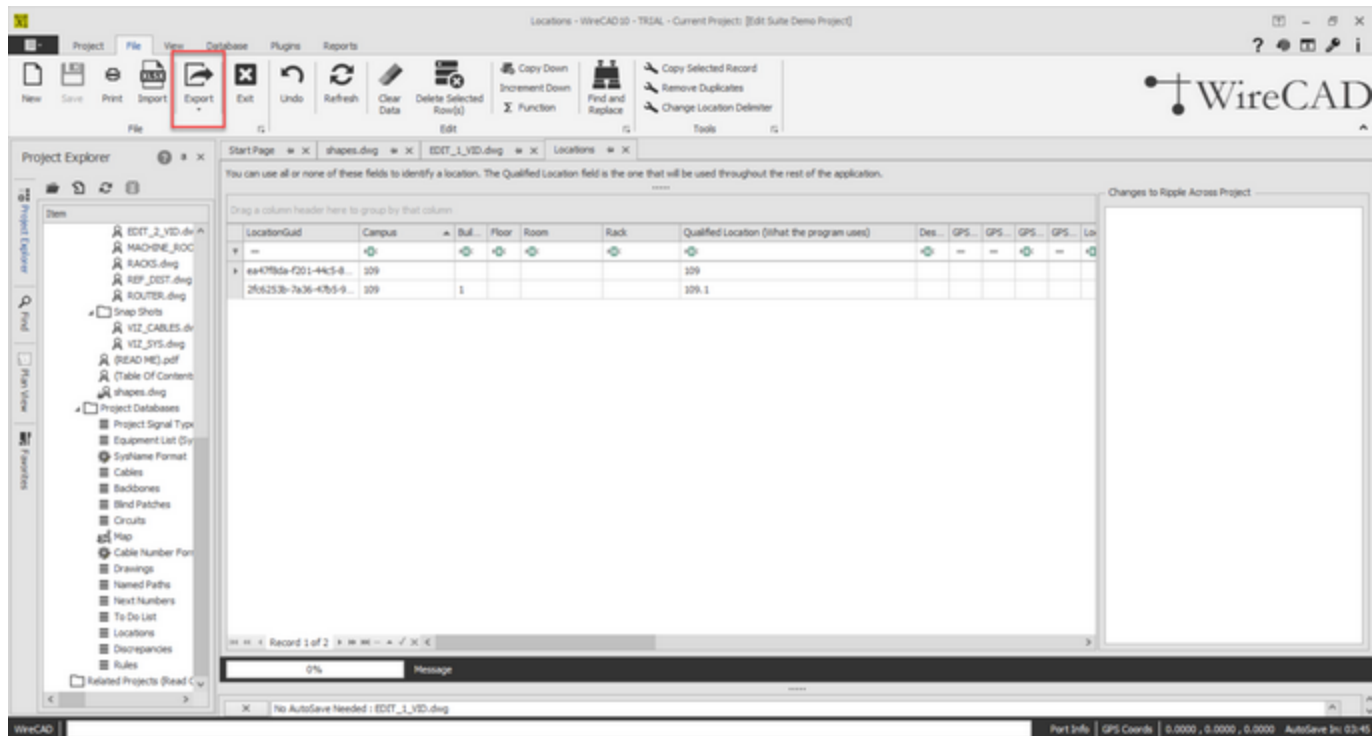
After making changes to the project data, the user can ripple the changes across the project.

Here is the model that you should follow when using the Round Trip Export/Import function:

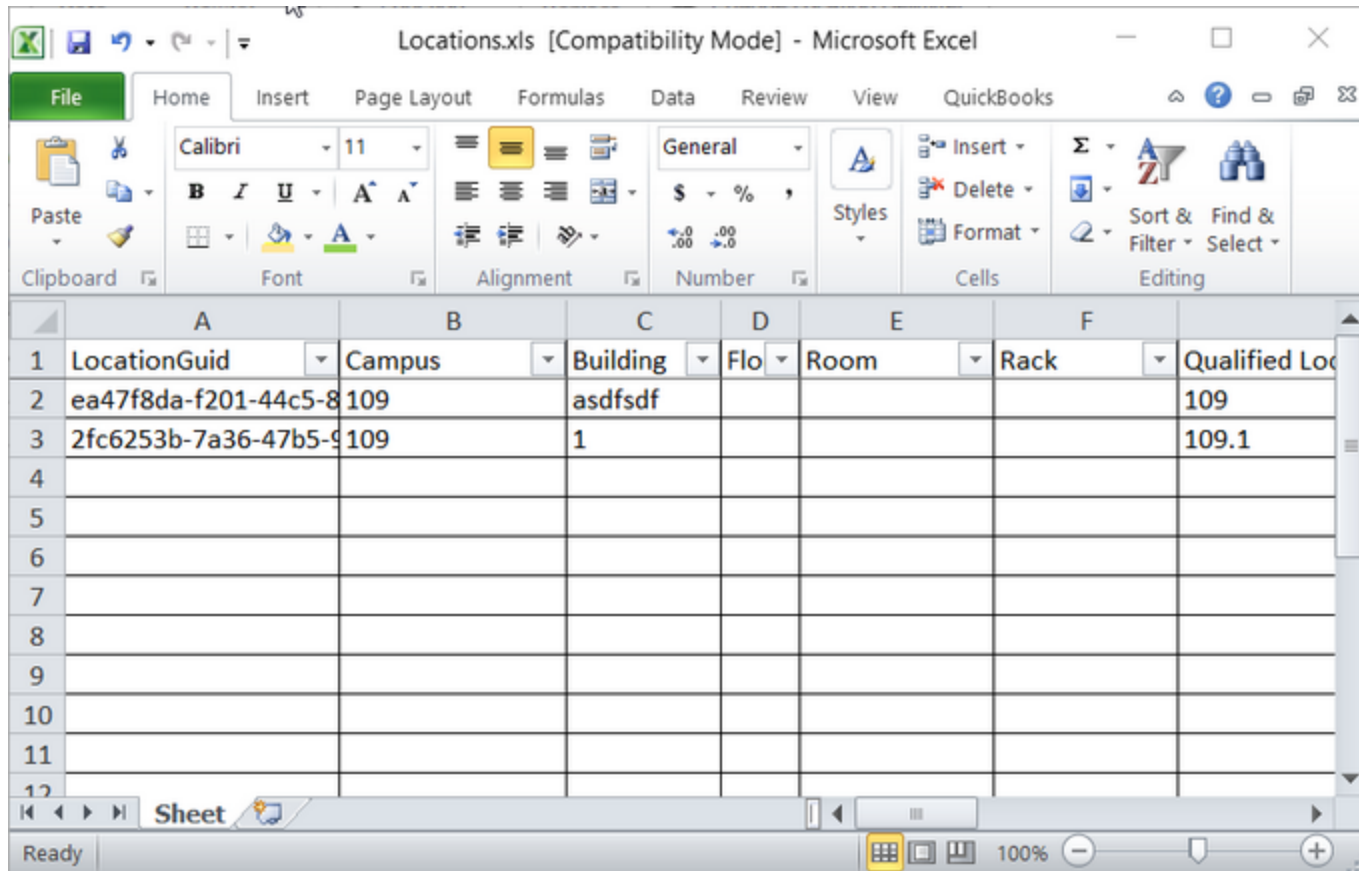


To perform the round trip export/import:

1. Open the grid that you would like to export.
2. In the File menu, Click the **[Export]** drop-down, and select **[To XLS]**

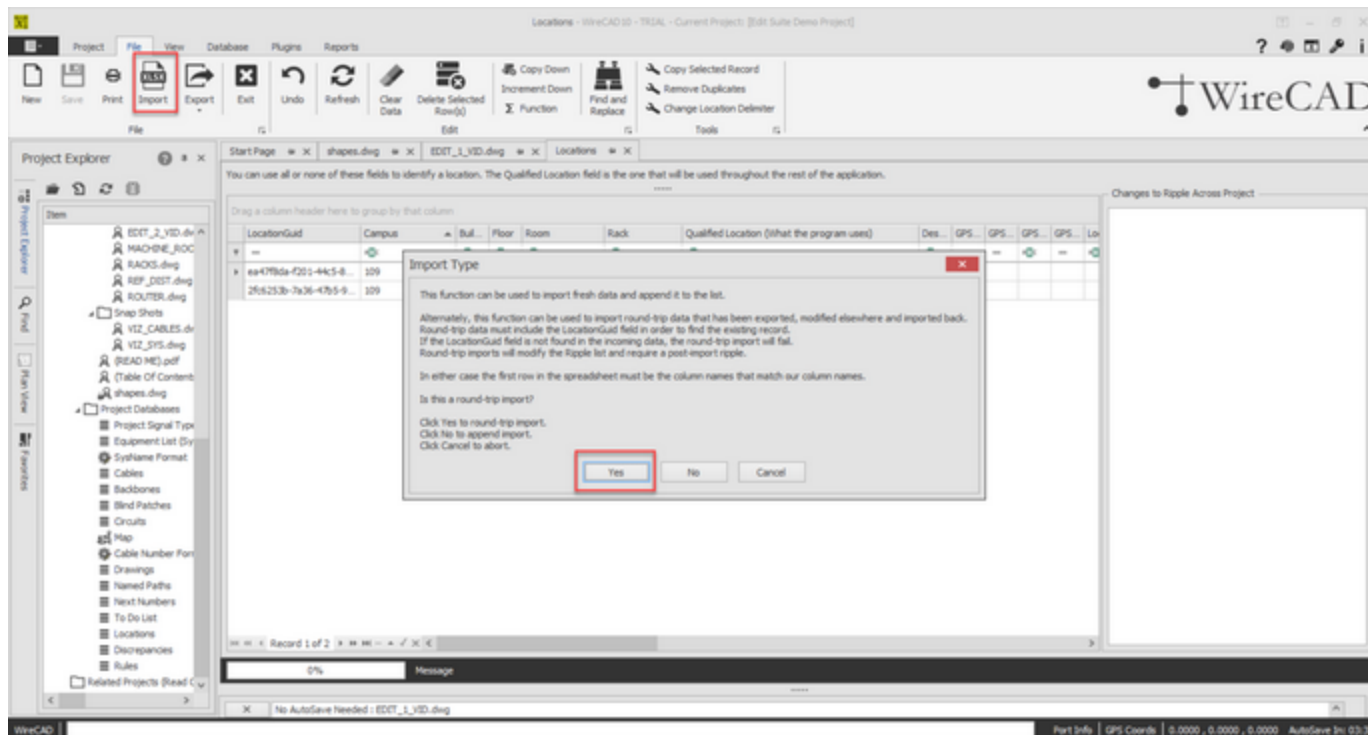


3. Open your Excel file and make a change. Here, we will make a change in the Building field. Save your changes.



	A	B	C	D	E	F	
1	LocationGuid	Campus	Building	Flo	Room	Rack	Qualified Loc
2	ea47f8da-f201-44c5-8109	109	asdfsdf				109
3	2fc6253b-7a36-47b5-9109	109	1				109.1
4							
5							
6							
7							
8							
9							
10							
11							
12							

4. Import your XLS file. **[Import]** > **[Yes]**



5. Once done, you'll get a notification regarding things that may or may not have gone wrong.
6. Ripple your changes across the project.

3.1.2 Setting Up Your Global Data

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  >




Menu: Several

Default command line shortcut: **le**

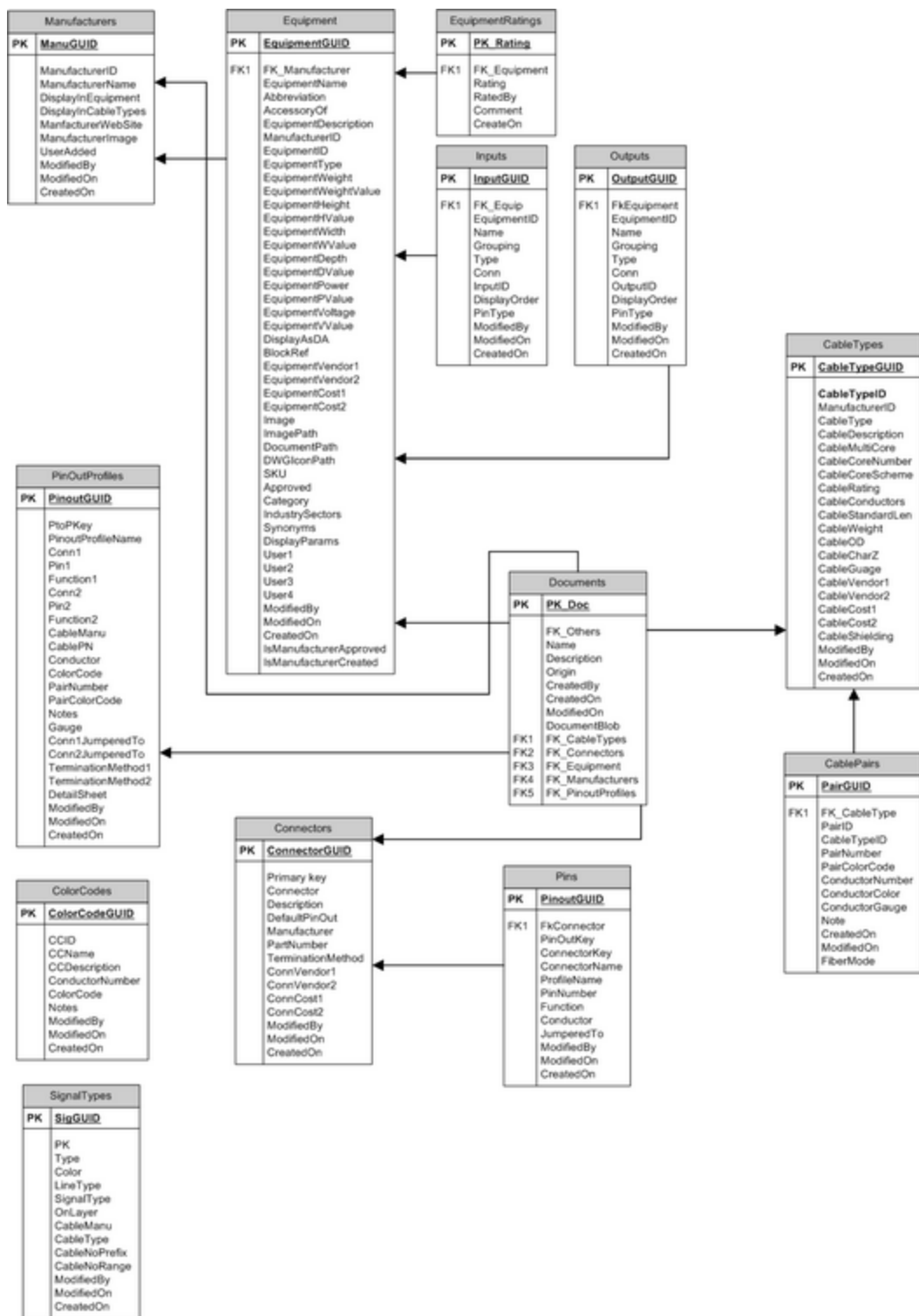
WireCAD maintains a global database with tables representing Manufacturers, Equipment, Signal Types, Connectors, Etc.

Getting started with WireCAD entails setting up the global databases to fit your needs. While the database is populated with data you may find that it suits your needs to purge the data and start fresh. If this is the case we can provide empty databases. At the very least you will want to set up the Signal Types grid with your defaults.

Next you will customize the Equipment Library with the products and IO that you use. In order to do this, you may either download existing products from the WireCAD Community Server or [enter your own](#) .

Structure:

- **Manufacturers** - The topmost table in the hierarchy
- **Equipment** - The Equipment description
- **Inputs** - inputs of a device
- **Outputs** - outputs of a device
- **Signal Types** - signal types and a bunch of defaults.
- **Connectors** - connector types.
- **Pinouts** - pin out definitions (data).
- **Color Codes** - color code look-up. Used by Cable Types.
- **Cable Types** - cable type information.
- **Cable Cores** - cores or conductor data for a cable type.



Global Equipment Library Schema (abrv)

3.1.2.1 Signal Types

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Getting Started](#) ⁴⁸ > [Setting Up Your Global Data](#) ⁵⁸ >



Menu: Database > Default Signal Types

3.1.2.2.1 How To: Create a New Cable Type

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Setting Up Your Global Data](#) [58] > [Cable Types](#) [60] >

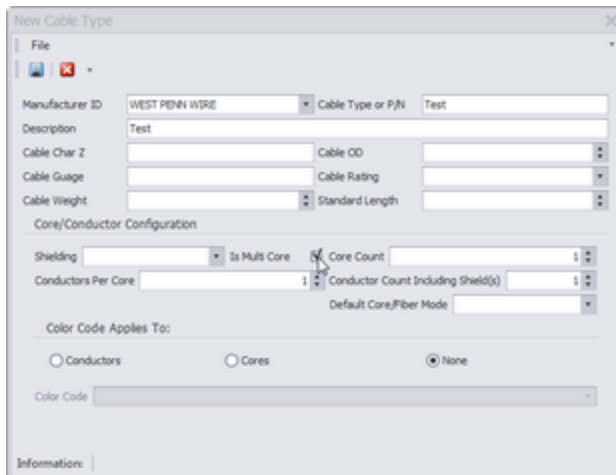


1. Click **Database > Cable Types**.

2. Click **File > New**.

3. Fill in all information about this cable including **Manufacturer**, **Cable Type** or **P/N** & **Description**. If you are working in the CMS tools you will need to define multi-core core configuration.

4. If this cable is a multi core cable, you will need to check the **Is Multi Core** checkbox and set how many cores are in this cable.



5. Select the **Default Core\Fiber Mode**. To label specific cores as SM or MM, see step 8.

6. Click **File > Save**.

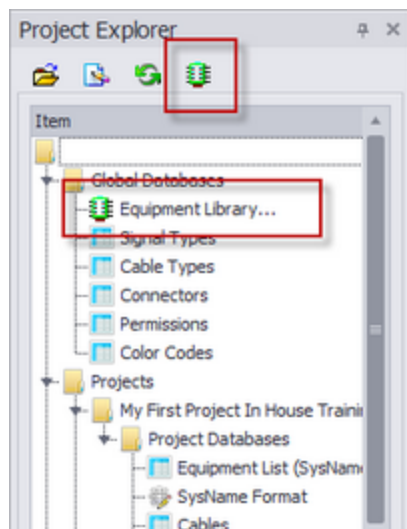
3.1.2.3 How To: Open the Equipment Library

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Setting Up Your Global Data](#) [58] >



The Project Explorer

The Project Explorer allows access to some of the global data grids. You can also access all of these grids from the **Database** menu.



Project Explorer Showing Equipment Library Access

Menu Commands to Access The Equipment Library

•**Database>Equipment Library**

•With an active drawing the Equipment Library can also be opened from: **Advanced Tools>Equipment Library**

•Commandline shortcut: **le**

3.1.3 Equipment Library

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] >



Menu: **Database>Equipment Library**

Menu: **Advanced Tools>Equipment Library**

Default command line shortcut: **LE**

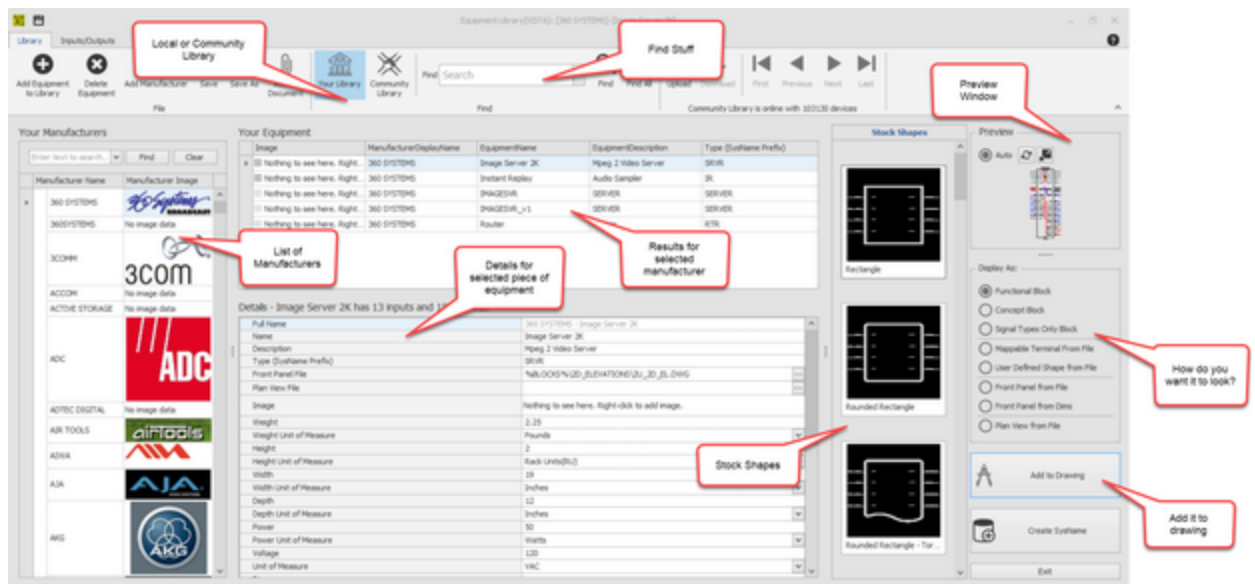
The WireCAD Equipment Library is where you will spend a fair amount of time as you get define equipment that you will use in your designs. This is also where we come to create CAD blocks in our drawings. There are many settings here that let you customize appearance. This chapter is the basics.

Main Library Dialogue

This is where you can find all of the information about a selected device with the exception of input and output information.

Find equipment definitions in the global database or the Community Server. To use a piece of equipment from the Community Library, download it to your local database. Map your signal types from the ones the device creator used.

Preview the block before [adding](#) [79] to drawing.



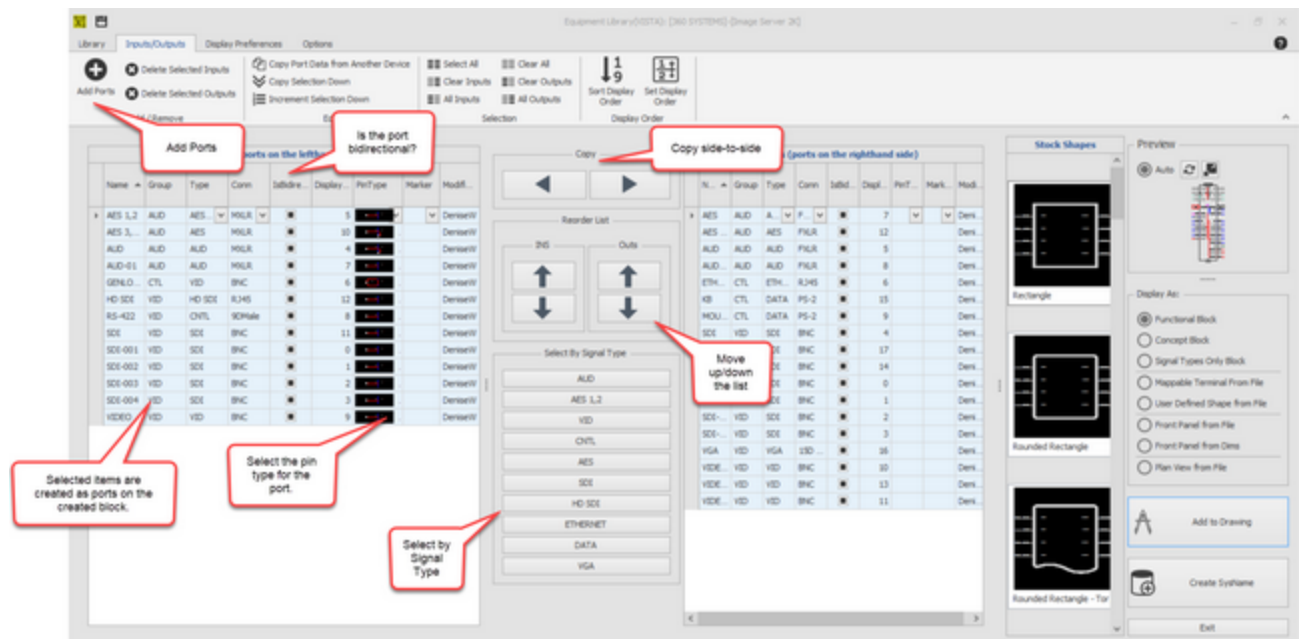
3.1.3.1 I/O Tab

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Equipment Library](#) [63] >



The Inputs/Outputs tab allows for:

- The ability to change pin type
- Users to group ports
- Users to mark ports to allow you to better present or clarify a design
- Users to create bidirectional signal types. If you are not sure which ide to place a prot, you can add a bidirectional port and it will display on inputs and outputs, but you can only use it as one or the other.



Related Topics:

[How To: Add Some Inputs and Outputs](#) [68]

3.1.4 Display Preferences Tab

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] >



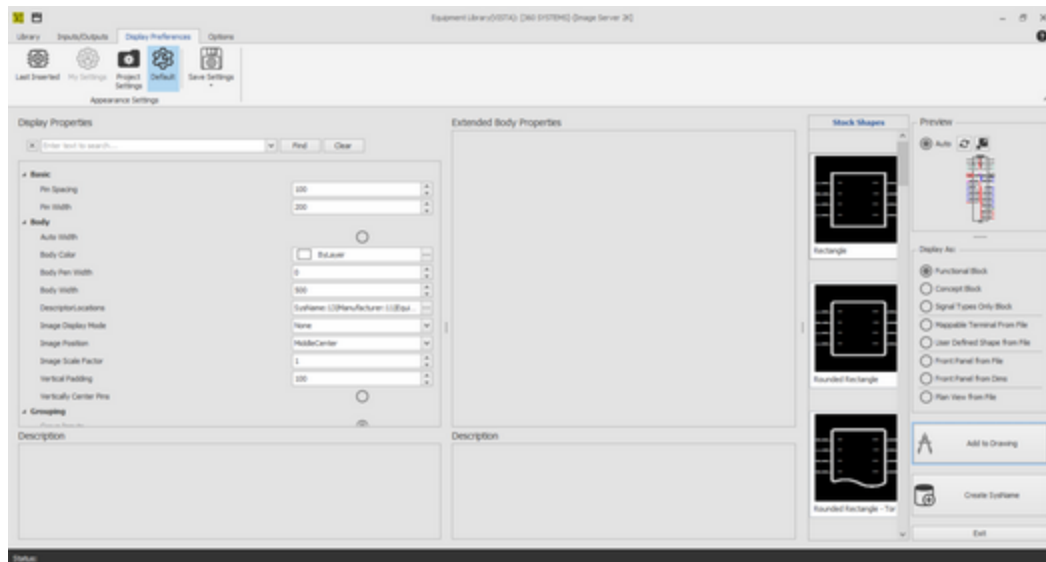
Controls the appearance of the created block.

You can indicate whether or not your port groups are displayed and how the group and port names are concatenated together.

The Display Preferences tab allow you to control the display or groups.

The Equipment Library holds a variety of block shapes to choose from. The width of your block is based on the text size.

Users are able to use the Appearance Settings ribbon menu to customize the appearance of projects, use the default settings, or use settings previously saved in WireCAD.



Related Topics:

[How To: Add Equipment to Drawings](#) 

3.1.5 Creating a New Equipment Definition

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  >



Menu: **Database>Equipment Library[Add Equipment to Library]**

Default command line shortcut: **le**

Equipment definitions are at the heart of WireCAD. They describe equipment down to the I/O. Create a new equipment definition from which to create functional blocks, rack panels, etc.

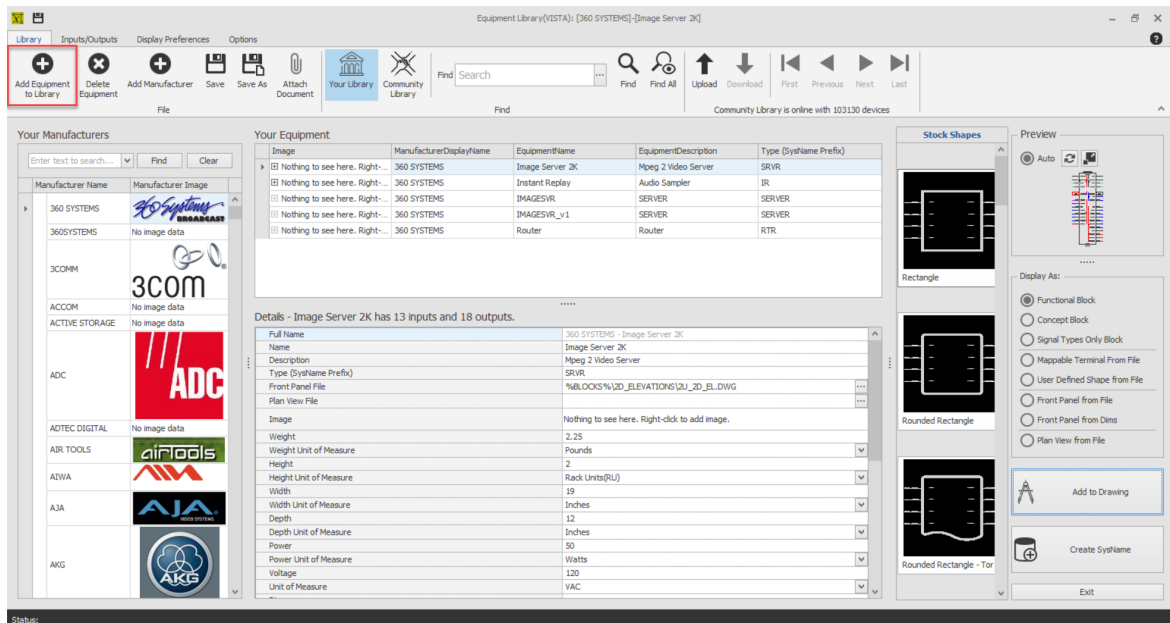
3.1.5.1 How To: Add Equipment to the Library

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Creating a New Equipment Definition](#) 



1. Click **Database>Equipment Library**

2. Click **[Add Equipment to Library]**



3. Select or add a **Manufacturer**.



Who Makes It and What's it Called

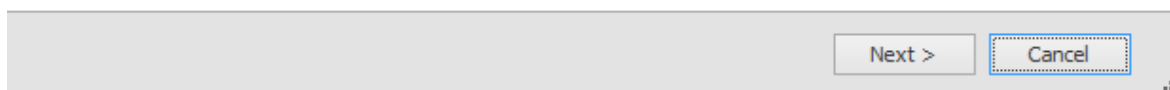
Manufacturer*

EquipmentName/Model/Part Number*

Description*

SysName Prefix*

* = Required



4. Enter **model/pn/name**, **description**, etc.

5. Click [Next >]



There is always some confusion about the **SysName Prefix** (Equipment Type) field. Consider the following:

We have two types of video server -model A and model B - that use similar wiring. We may consider using the **SysName Prefix** - SRVR. Anytime we create an instance of one of these we they will be SRVR-01, SRVR-02, and SRVR-0n. Now consider that model A is not available to use and that we have to install model b instead. No problem. Because they "harness" is named SRVR-n we can interchange Make and Model as needed.

6.This page is optional, but we recommend filling in the **Front Panel File** field.

7.

The screenshot shows a 'New Equipment' dialog box with a title bar containing a question mark and a close button. The main area is titled 'Completing the wizard' and contains several input fields: 'Front Panel File' and 'Plan View File' both with browse buttons (three dots); 'Image' with a large empty box and the text 'No image data'; 'Categories', 'Synonyms', and 'Abbreviation' each with a text input field; and 'Accessory Of' with a text input field. At the bottom right are 'Finish' and 'Cancel' buttons. A back arrow button is in the top left of the main area.



Filling in the Front Panel File field will make the Rack Builder tool work. Leave it empty and you may be frustrated later.

8. Click **[Next >]** to add the new definition.

The wizard completes and you are taken to the Equipment Library to edit the newly added equipment definition.

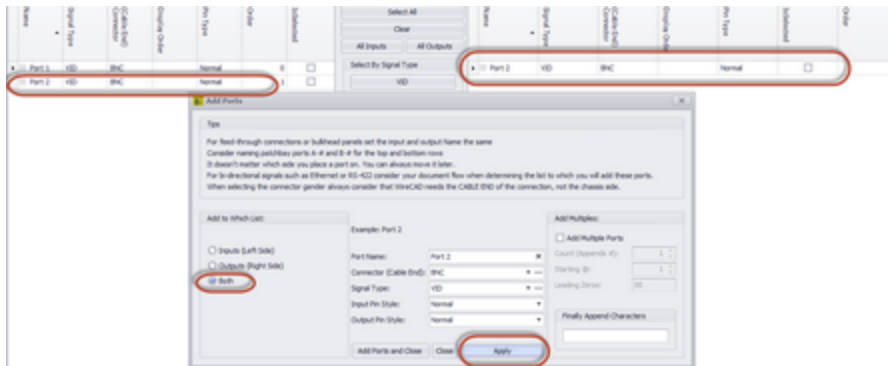
3.1.5.2 How To: Add Some Inputs and Outputs

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Creating a New Equipment Definition](#)  >

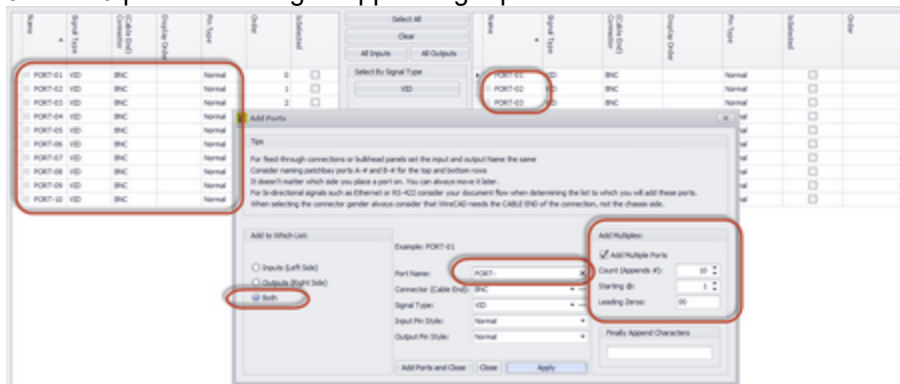


1. Open the **Equipment Library**.
 2. Find an equipment definition to which we will add ports.
 3. Select the **I/O** tab.
 4. Click **[Add Ports]** button.
 5. Enter a port name, connector, signal type.
 6. Select **Inputs**, **Outputs** table or **Both**.
 7. Click **[Add Ports and Close]** or **[Apply]** (if you want to leave the form open).
- Your port(s) will be added to the selected list(s):

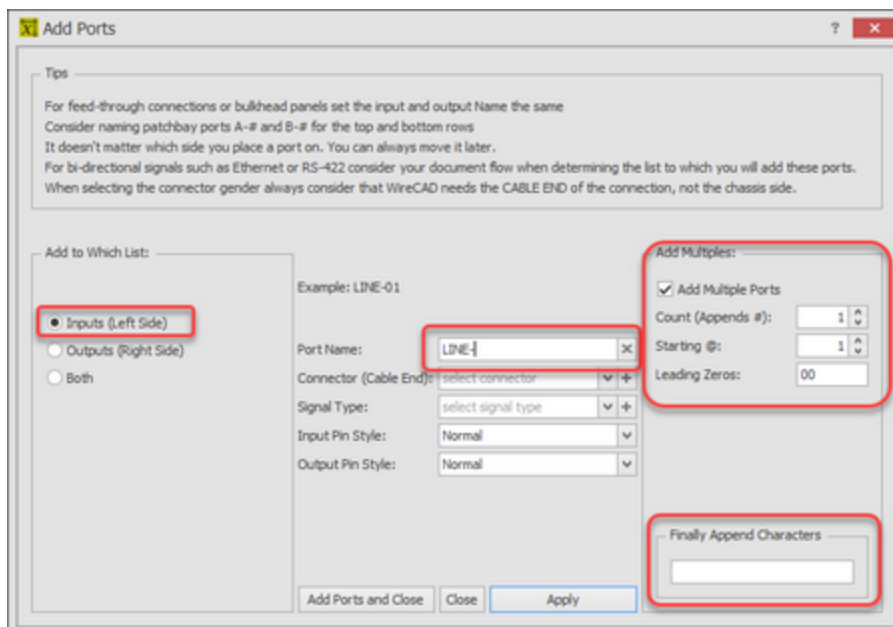
8. Add a single port to the **Inputs** and **Outputs** list at the same time.



9. Add 10 ports to each grid appending a port number to each record.



10. Now let's create the 8 left line inputs of a stereo mixer. We will use the **Finally Append Characters** field to append the characters `-LEFT` after the number is created and appended to the **Port Name**.





Try adding multiple ports. Enter "PORT-" (leave off the quotes). Select any Connector and Signal Type. Select the **BOTH** radio button to add to both lists. Now Check the **Add Multiple Ports** box and enter a **Count** of 8. Click **[Add Ports and Close]**. See below.

3.1.6 Creating a New Project

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  >



Menu: **Application Menu > New Project**

Default command line shortcut: **np**

Create a new WireCAD Project structure. This involves folder structures on your operating system as well as databases and support files.

WireCAD can create projects with a number of different database formats.

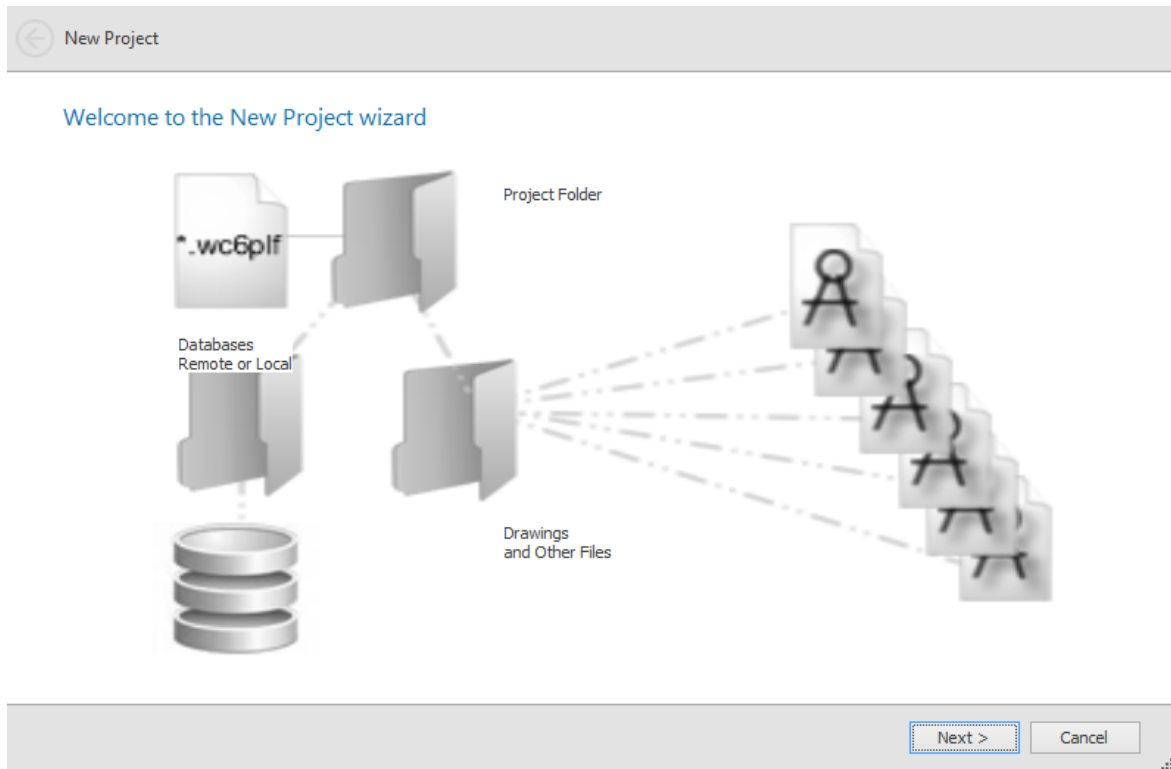
You may choose to create a new project using file based databases for their zero admin capabilities, or SQL Server for an enterprise installation.

3.1.6.1 How To: Create a New Project

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Creating a New Project](#)  >

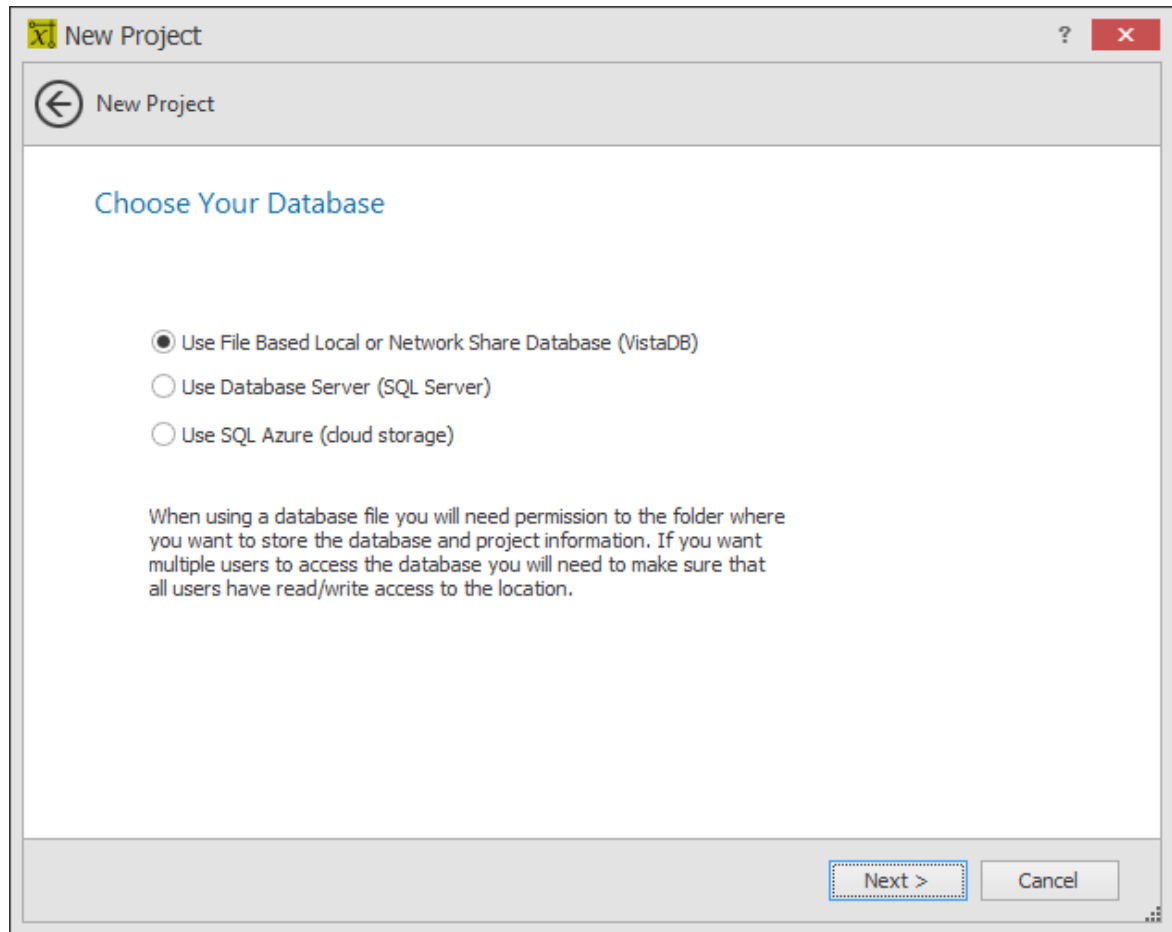


1. Click **[Application Menu] > [New Project]**



2. Click **[Next >]**

3. Select the type of database you wish to use and Click **[Next >]**

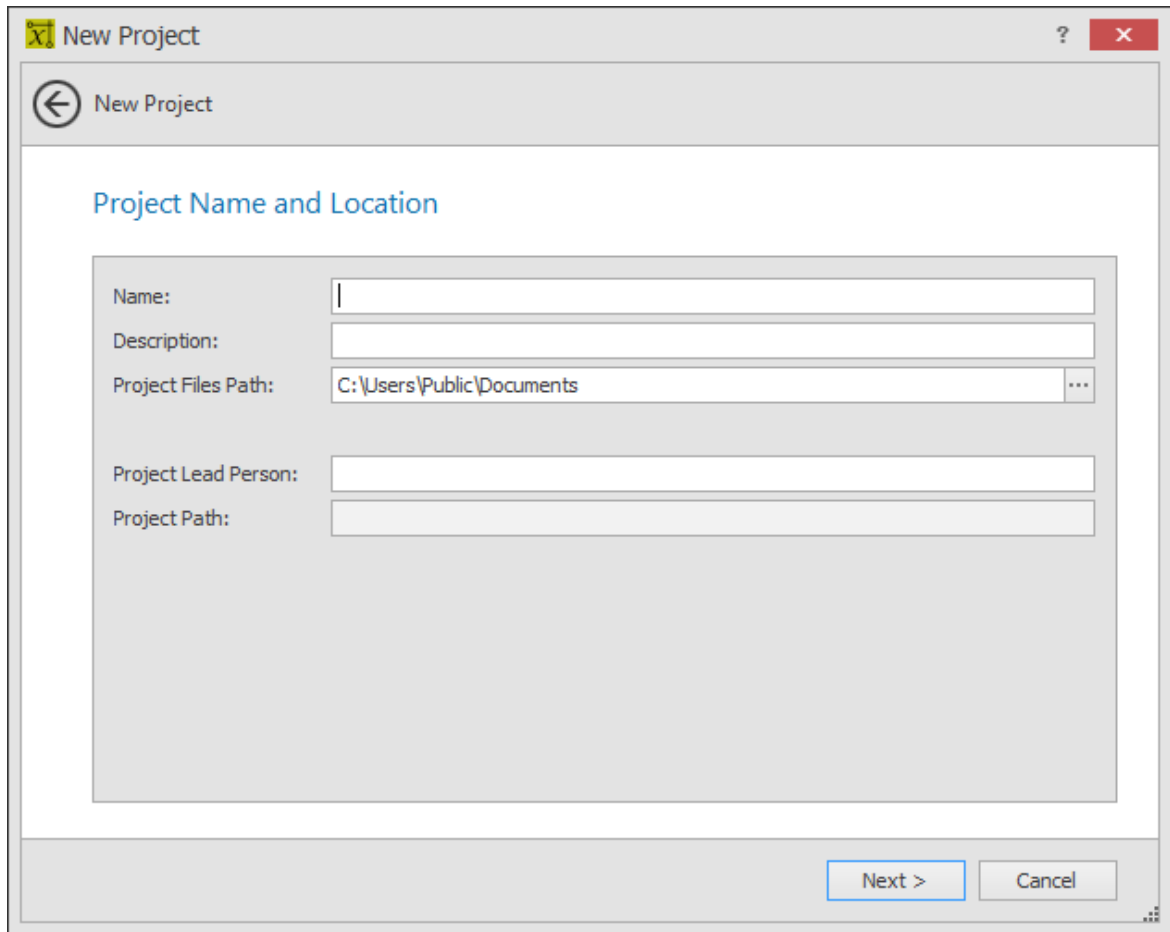


4. Enter the **Name** for the new project. This must follow file naming conventions because we are going to make a folder out of the name and in the Project Path that you choose.



NOTE: We recommend NOT using the possessive ' i.e "Bob's Project." The ' character is the SQL escape character and though we rarely use it in WireCAD it can cause some queries to fail.

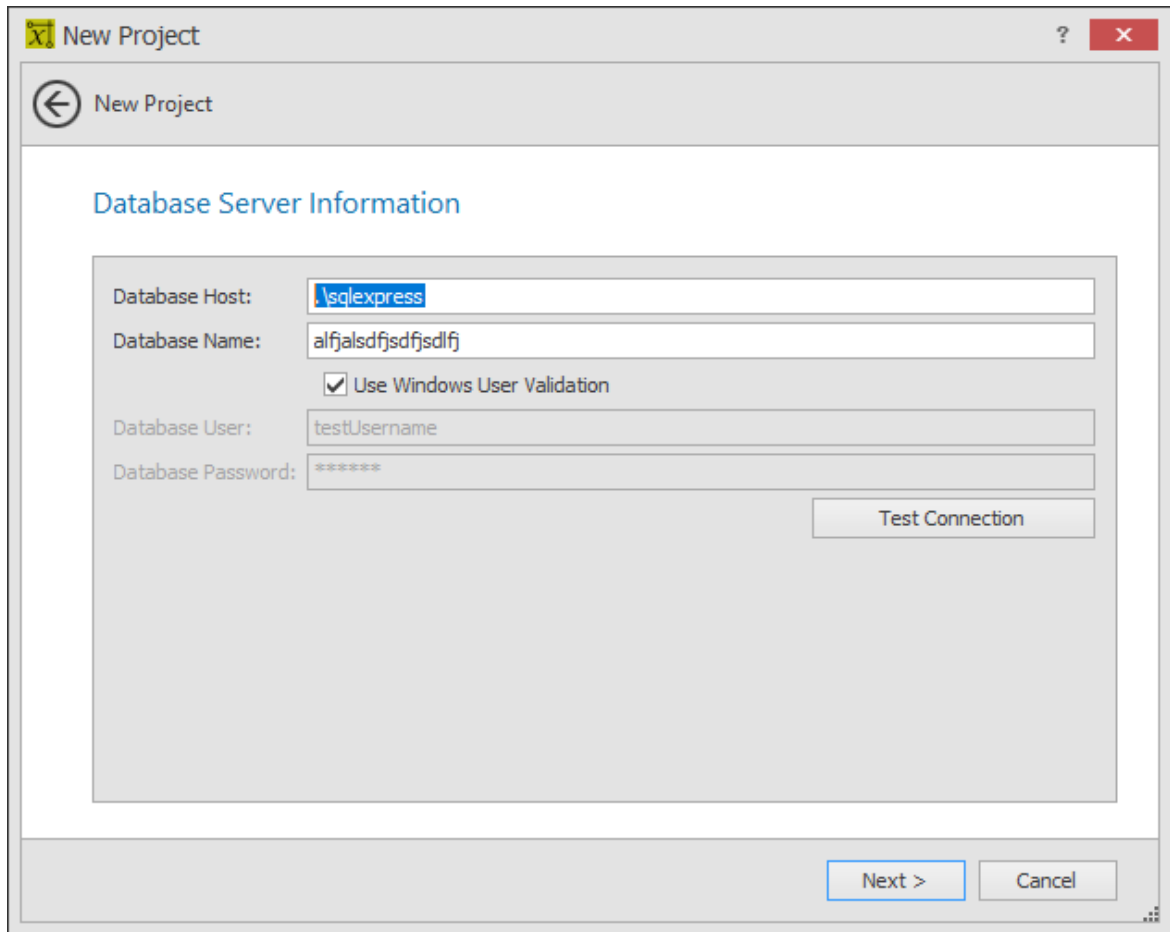
5. Enter a **Description**.



The image shows a 'New Project' dialog box with a title bar containing a yellow 'X' icon, the text 'New Project', a question mark, and a red close button. Below the title bar is a navigation bar with a back arrow icon and the text 'New Project'. The main area is titled 'Project Name and Location' in blue. It contains several input fields: 'Name:' with a text box, 'Description:' with a text box, 'Project Files Path:' with a text box containing 'C:\Users\Public\Documents' and a browse button '...', 'Project Lead Person:' with a text box, and 'Project Path:' with a text box. At the bottom right are 'Next >' and 'Cancel' buttons.

6. Click **[Next >]**

7. If using SQL databases enter the host information. Click **[Next >]**



The image shows the 'New Project' dialog box in WireCAD, specifically the 'Database Server Information' tab. The dialog has a title bar with a yellow 'X' icon and the text 'New Project'. Below the title bar is a navigation bar with a back arrow and the text 'New Project'. The main area is titled 'Database Server Information' in blue. It contains several input fields: 'Database Host' with the value '.sqlexpress', 'Database Name' with the value 'alfjalsdfjsdfjsdlfj', 'Database User' with the value 'testUsername', and 'Database Password' with the value '*****'. There is a checkbox labeled 'Use Windows User Validation' which is checked. A 'Test Connection' button is located to the right of the password field. At the bottom right of the dialog are 'Next >' and 'Cancel' buttons.

New Project

New Project

Database Server Information

Database Host: .sqlexpress

Database Name: alfjalsdfjsdfjsdlfj

☒ Use Windows User Validation

Database User: testUsername

Database Password: *****

Test Connection

Next > Cancel

8. Define some of your project preferences. Here you set your base **Starting Cable** Number for all sequences.

New Project

← New Project

Project Settings (also available under Project>Settings)

Project Settings
Set up defaults for the current project. Each project has its own settings.

Starting Cable Number
Integer used in conjunction with the Cable Number Format tool as a starting

Leading Zero Count ☐ Ignore Cable# Text Color

☐ I Build My Racks From the Top Down Default Rack Height in Rack Units

Default Cable Type Manufacturer and Cable Type To Use When Assigning Cable Numbers

Default Cable Manufacturer Default Cable Type
Used when no cable type is associated with a Signal Type for the cable number being assigned

Default Plan View Block File:

☒ Users (mere mortals) Can Modify These Project Settings

☐ Add Pinout Data to Each Cable. Pinout Data is Defined in the Database>Pinouts Tool

☒ Use the device display settings. Not the Project Settings.

☒ Show duplicate numbers from related projects

9. Click **[Next >]**

10. By default the Next Numbers table will suggest and test the next number. If that number is in use, the **Next Numbers** table is incremented and tested until an available number is found. This behavior can be overridden to force the usage of the number in the Next Numbers table without testing for existence. To do this Check the **Disable Find Next Available**

Project Settings (also available under Project>Settings)

Number Generation

☐ Disable Find Next Available SysName (Use Next Number W/O Checking)

☐ Disable Find Next Available Cable Number (Use Next Number W/O Checking)

The default behaviour is for WireCAD to lookup the next number from the Next Numbers table, then check to see if that number is in use - if so keep incrementing until an available number is found. Disabling this functionality will use the number in the Next Numbers table without checking its availability.

☒ Automatically Consume Cables that are marked as Prewired.

This will use those numbers applied to the PreWire Cable instead of producing a new number.

☒ Strict Multicore Location Assignment. Ensures that all assigned cores go to the same locations.

IP Mask Mode Default Subnet Mask

Relational Data

By default WireCAD attempts to maintain a series of tables that relate the main tables to each other. This allows you to see, for example, which drawings a SysName can be found in etc. The relational data can get out of sync or otherwise corrupt. You may elect here to not maintain it in the project and your project will speed up. Your main Equipment List, Cables, Drawings, and Circuits tables WILL NOT BE EFFECTED.

☒ Maintain Relational Data

☐ Disable Cable Number Edit Dialog Mask

☐ Disable SysName Edit Dialog Mask

Next > Cancel

11. Click **[Next >]**

12. **Enabling Locations Look-up** causes a subtle change in the SysName Assignment process. You will no longer be able to just type a location but rather will select your predefined locations from a drop-down list. If you wish to work as you have in previous versions of WireCAD disable **Locations Look-up**.

New Project

← New Project

Project Settings (also available under Project>Settings)

Location Settings

Location / Elevation String Parsing

Location Delimiter Elevation - Slot Delimiter

Location delimiter is used to separate the various elements of the location field. Example: you might use the dot '.' to separate the location field into its constituent part like: Campus.Building.Floor.Room.Rack.Elevation. The delimiter must be consistent across the project.

Default Location Default Elevation

☐ **Enable Locations Lookup Table**

This allows you to predefine your locations and enforces use of only those predefined locations.

Locations Table Fields

<input checked="" type="checkbox"/> Enable Campus	Campus Field Caption	<input type="text" value="Campus"/>
<input checked="" type="checkbox"/> Enable Building	Building Field Caption	<input type="text" value="Building"/>
<input checked="" type="checkbox"/> Enable Floor	Floor Field Caption	<input type="text" value="Floor"/>
<input checked="" type="checkbox"/> Enable Room	Room Field Caption	<input type="text" value="Room"/>
<input checked="" type="checkbox"/> Enable Rack	Rack Field Caption	<input type="text" value="Rack"/>
<input checked="" type="checkbox"/> Start Numbering at 1. Uncheck to Start at 0		

Next > Cancel

13. Click **[Next >]**

14. Review and click **[Finish]**

WireCAD will then create files and folders on the OS and create a new database on the SQL server if you have elected to use SQL server databases. Otherwise the blank project database will be copied to your project databases folder.

3.1.7 Creating a New Drawing

Navigation: [WireCAD PRO Manual](#) > [Getting Started](#) >



Menu: **File>New Drawing, Project Explorer\Current Project\New Drawing**

Default command line shortcut: **nd**

Related Settings:

Show New Drawing Wizard = true

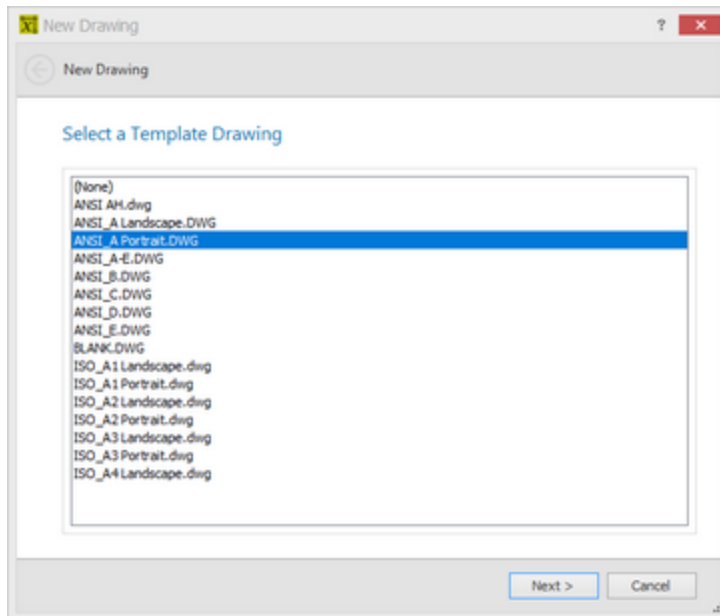
Create a new drawing based on a drawing template, with or without model space boundaries.

3.1.7.1 How To: Create a New Drawing

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Creating a New Drawing](#) [77] >



1. Click **Project>New Drawing**

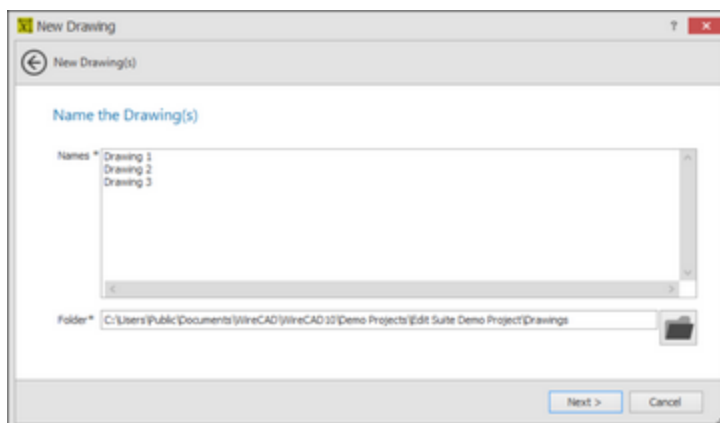


2. Select a template drawing from which to start.

Template drawings are drawings that have been saved in the template drawings folder and already have entities such as page borders, layouts and viewports added to them.

3. Click **[Next >]**

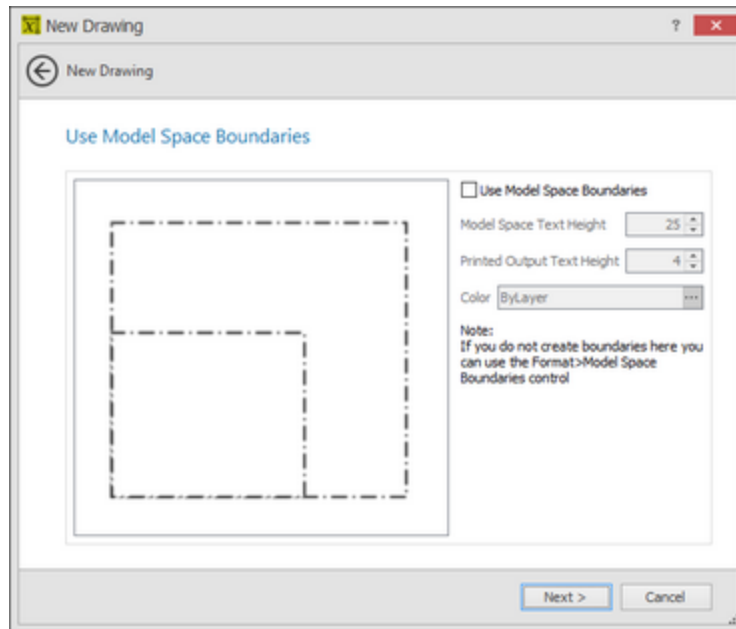
4. Name your drawing(s). We have added feature allowing you to create more than one drawing at a given time. Simply type the name of your drawing into the text box minus the file extension. If you are creating multiple drawings, enter one name per line. Click **[Next>]**



5. Select boundary settings. **Create Model Space Boundaries.** The Model Space Boundaries function takes two arguments, the **Model Space Text Height** and the desired **Printed Output Text Height**. Using these two variables in conjunction with the size of the Viewports in each Layout to create boundary rectangles in the Model space. Each boundary is accompanied by a text

description that describes the Viewport and to which the boundary applies as well as the **text heights** and **scale factor**.

Note: You can add Model Space Boundaries later using the Drawing>Format>Model Space Boundaries ... function



6. Click **[Next >]** to finish and name your drawing.

WireCAD creates the drawing in the project drawings folder tree.

3.1.8 Adding Equipment to Drawings

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] >



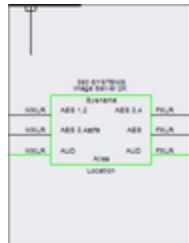
Menu: Database>Equipment Library [Add to Drawing]

Default command line shortcut: le

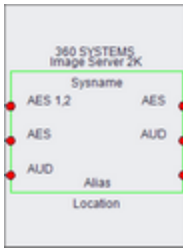
Create a new equipment block from the library and add it to the current drawing.

You can create equipment in one of these modes:

- **Functional I/O**



- **Conceptual I/O (low detail)** - Collapses to the I/O type



•Conceptual I/O (high detail)



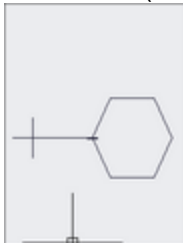
•Front Panel (File Merge)



•Front Panel (From Dims)



•Plan View (File Merge)



•Plan View (From Dims) - Not Yet Implemented

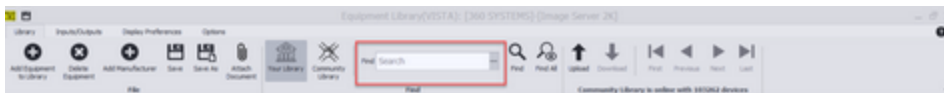
3.1.8.1 How To: Add Equipment to Drawings

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Getting Started](#) ⁴⁸ > [Adding Equipment to Drawings](#) ⁷⁹ >

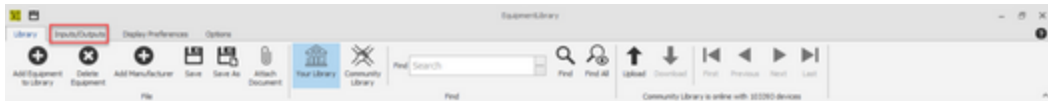


1.Create and open a drawing.

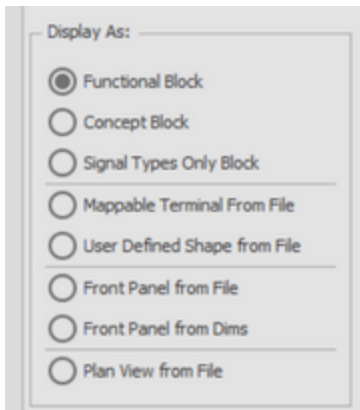
2.Find and select the equipment definition from the Equipment Library.



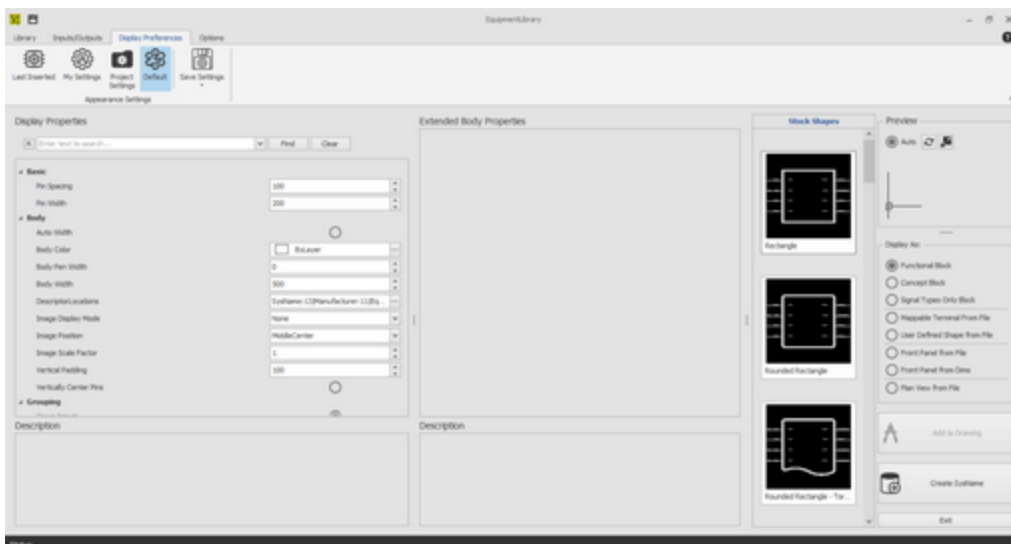
3. Select the IO you want to display in the drawing



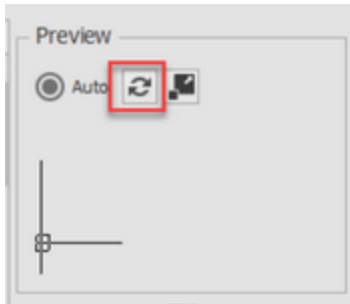
4. Select the Display mode



5. Select the Display Preferences. Everything is parametric. There are settings for Body Width, Pin Spacing, Color, etc.



6. If the **Auto Preview** function is not set, you may wish to click the Refresh button (in the preview window).



7. Click **[Add to Drawing]** (requires an active drawing).
8. Place the newly created block in the drawing. You will be able to place any number of copies. Just keep left-clicking to place. When you are done right-click and click done or click the **[Enter]** key.

3.1.9 Drawing Cables

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Getting Started](#) ⁴⁸ >



WireCAD provides a series of tools to draw cables.

Normally we draw from one WireCAD device to another.

If you need to draw spare cables, see the [Spare Cables](#) ¹⁴² tool.

WireCAD provides a cable auto-routing tool that automatically routes the cable around other devices and, if selected, avoids other cables. The auto-router will always find a path for the cable, even if it means that the cable is drawn through another device or must overlay another cable.

If you do not like the way a cable is routed, you have two choices:

1. Manually draw the cable by selecting Manual Draw and place every point from source to destination.
2. Select the cable as drawn and grab a grip on the cable and move it around. You can use Ctrl+Shift+Click to add vertexes and Ctrl+Click to remove vertexes.

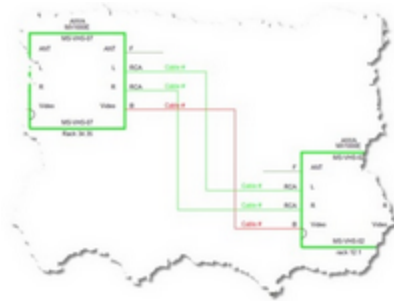


If you manually draw cables or otherwise put them where you want them and then move a device, the auto-router will be invoked and re-route all your changes.

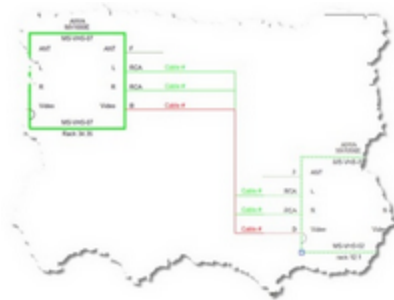
WireCAD Cable Terminology

Devices have inputs and outputs, Cables have sources and destinations.

For purposes of this manual we will refer to Jacks, Junction Boxes, Router Crosspoints, Bulkhead connectors, and On-Sheet/Off-Sheet Pointers, Adapters and Splice Points collectively as terminals.

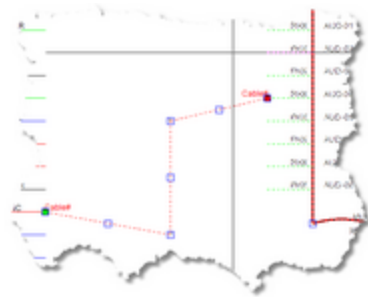


True

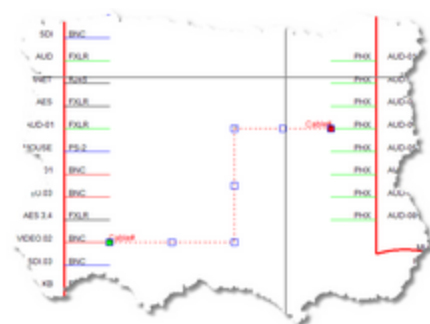


False

- **Tidy Cables** - After moving a grip on a cable the cable is forced ortho.






Before Tidy

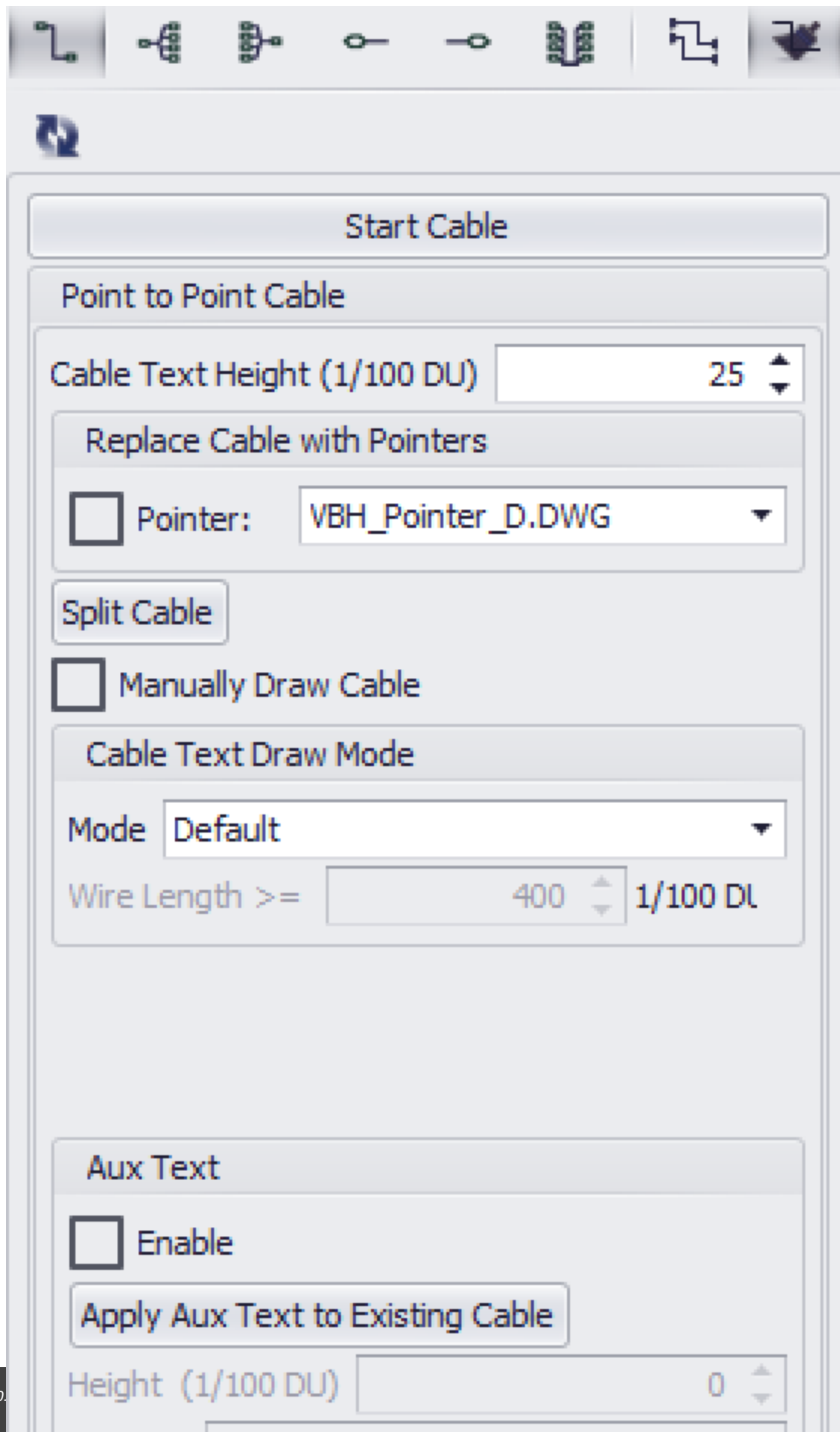


After Tidy

- **Repeat** - Repeat drawing cables until Esc is pressed.

3.1.9.1 One to One Cables

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Drawing Cables](#)  >



The image shows the 'Start Cable' dialog box in WireCAD. At the top is a toolbar with icons for various cable drawing functions. Below the toolbar is a 'Start Cable' button. The main section is titled 'Point to Point Cable'. It contains a 'Cable Text Height (1/100 DU)' spinner set to 25. Below this is a 'Replace Cable with Pointers' section with an unchecked checkbox and a dropdown menu showing 'VBH_Pointer_D.DWG'. There is a 'Split Cable' button and an unchecked 'Manually Draw Cable' checkbox. The 'Cable Text Draw Mode' section has a 'Mode' dropdown set to 'Default' and a 'Wire Length >=' spinner set to 400 with a unit of '1/100 DU'. At the bottom is an 'Aux Text' section with an unchecked 'Enable' checkbox, an 'Apply Aux Text to Existing Cable' button, and a 'Height (1/100 DU)' spinner set to 0.

Start Cable

Point to Point Cable

Cable Text Height (1/100 DU) 25

Replace Cable with Pointers

☐ Pointer: VBH_Pointer_D.DWG

Split Cable

☐ Manually Draw Cable

Cable Text Draw Mode

Mode Default

Wire Length >= 400 1/100 DU

Aux Text

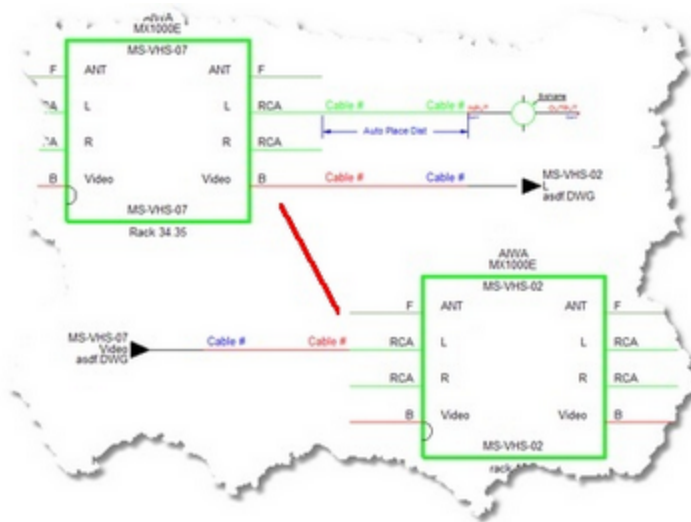
☐ Enable

Apply Aux Text to Existing Cable

Height (1/100 DU) 0



- **Cable Text Height** - The Cable# text entity height in 100/DU.
- **Replace Cable with Pointers** checkbox- When checked this function will place linked pointers between the source and destination ports based on the Default Pointer selection.



[Split Cable] - Splits a cable that is drawn between two ports into a cable with pointers on either side of the ports. This is useful if you need to break up a drawing and copy/paste part of the drawing to another. For more information see [How to: Split Cables.](#)

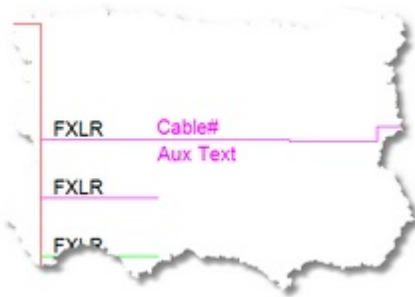
- **Manual Draw checkbox** - Draw every point in the cable.

Cable Text Draw Mode

- **Default** - Draw two Cable# text entities, one at each end of the cable.
- **Left** - Draw only the left side Cable# text entity.
- **Right** - Draw only the right side Cable# text entity.
- **LeftAlwaysRightIf** - Draws the left side always and the right side if the cable polyline is longer than the entered value in 1/100DU.
- **RightAlwaysLeftIf** - Draws the right side always and the left side if the cable polyline is longer than the entered value in 1/100DU.

- **Aux Text Enable** - Enable the placement of Aux Text.

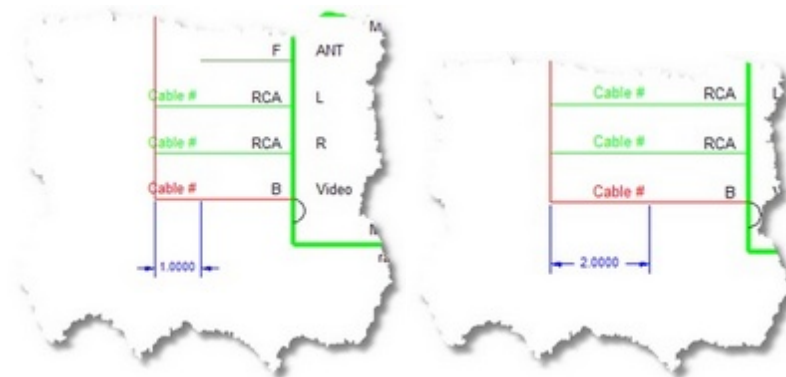
[Apply Aux Text to Existing Cable] - Updates an existing cable to show the Aux Text if not shown.



Aux Text Shown

- **Aux Text Height** - The height in 100/DU of the Aux Text.
- **Location** - The position of the Aux Text relative to the cable polyline.
 - Over** - positioned over the cable polyline.
 - Under** - positioned under the cable polyline.
 - Bubble** - Not yet supported.
- **Variable** - You may choose to populate the Aux Text with the following variables:
 - CircuitNumber** - The Circuit Number as entered in the Cable Edit Dialog.
 - IPAddress** - Not yet implemented.
 - Length** - The Length field as entered in the Cable Edit Dialog.
 - User1** - The User1 as entered in the Cable Edit Dialog.
 - User2** - The User2 as entered in the Cable Edit Dialog.
 - User3** - The User3 as entered in the Cable Edit Dialog.
 - User4** - The User4 as entered in the Cable Edit Dialog.
- **Format** - Used to apply string formatting to the Variable selected. **{0}** represents the data from the selected variable.

Example: the incoming data from the selected variable is 300 and you want to format it to represent meters to the reader. Your format field would be **{0}m**. The output would be formatted as 300m.
- **X Offset** - Horizontal auto-router offset. When drawing cables, WireCAD uses an auto-routing algorithm. The **X Offset** determines how far away horizontally from other equipment and cables a new cable will route.



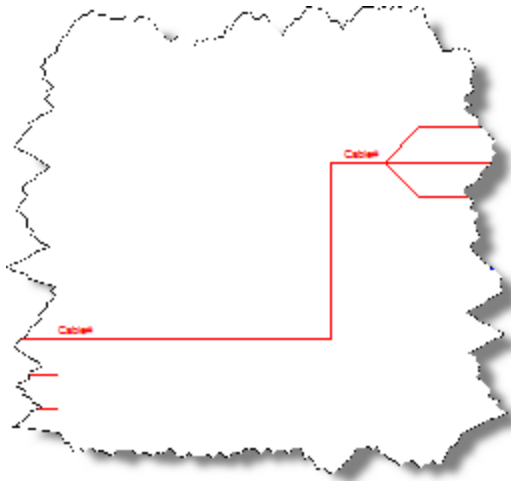
- **Y Offset** - Vertical auto-router offset. When drawing cables, WireCAD uses an auto-routing algorithm. The **Y Offset** determines how far away Vertically from other equipment and cables a new cable will route.



- **Default Pointer** - Select the pointer to use when replacing cable with pointers.

3.1.9.2 One to Many Cables

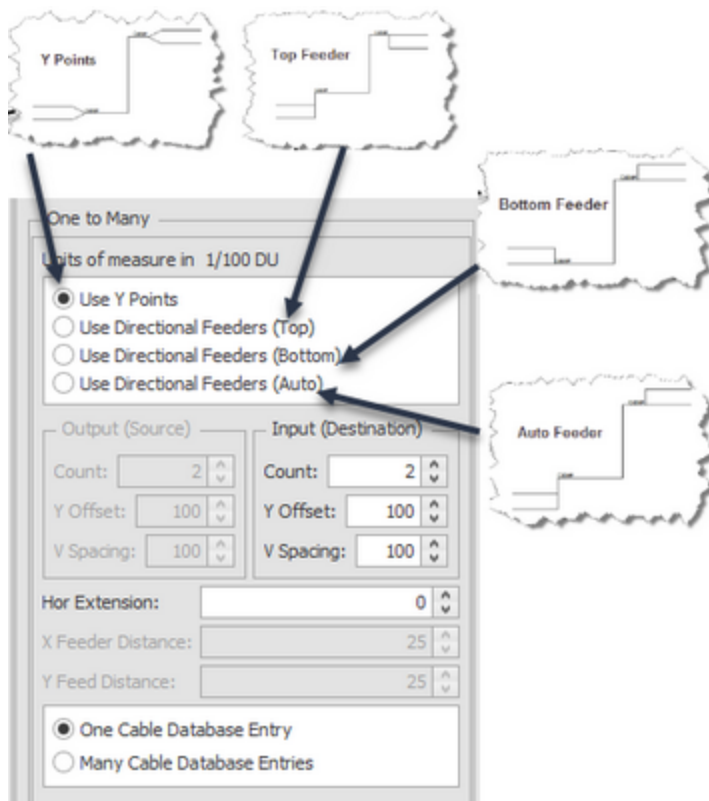
Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Drawing Cables](#) [82] >



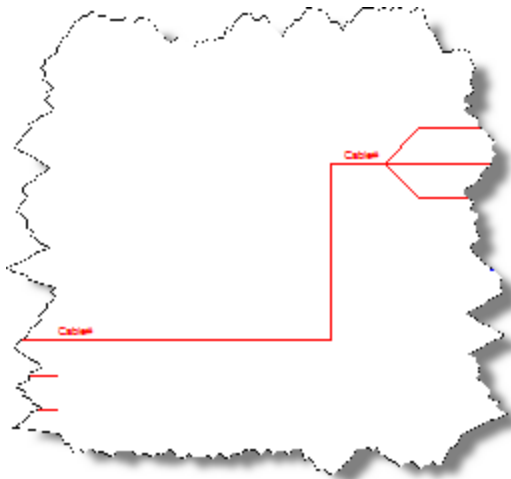
one-to-many cable

Use this set of controls to place a cable from one output to many inputs.

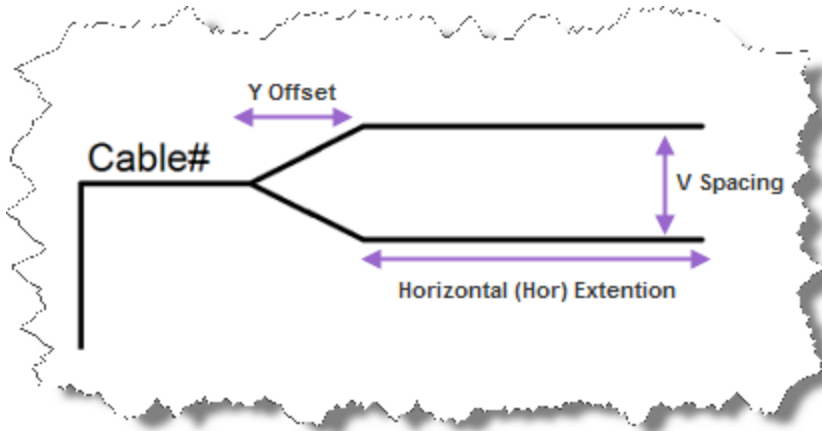
- **Feeder selection** - Determines the display of the Y point(s).



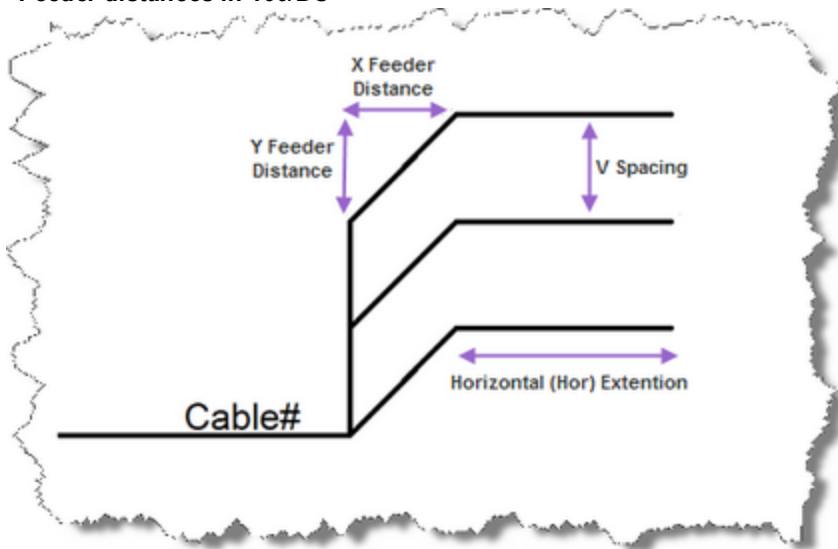
- **Count** - Destination count.



- **Y offset, Y Spacing, Hor Extension in 100/DU**



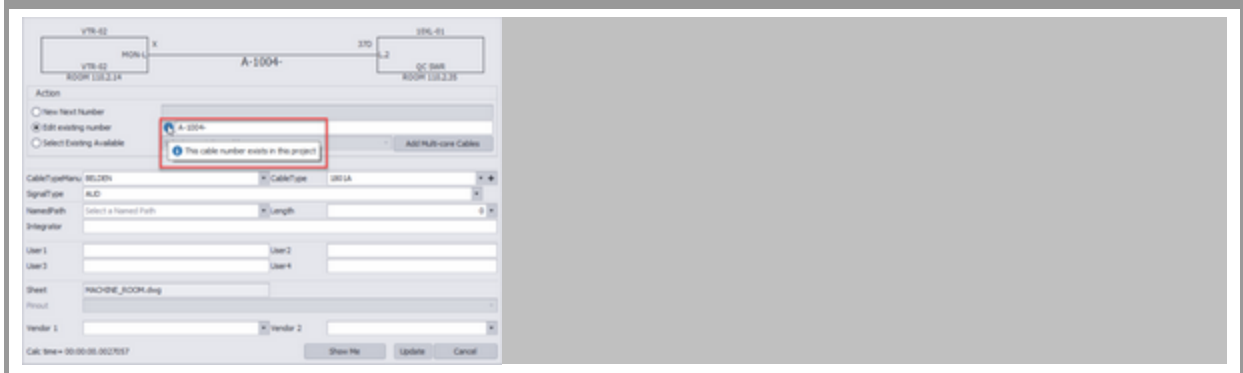
• **Feeder distances in 100/DU**






• **One Cable Database Entry** - Adds a single entry in the cables database and one Cable# text entity at the Y point.

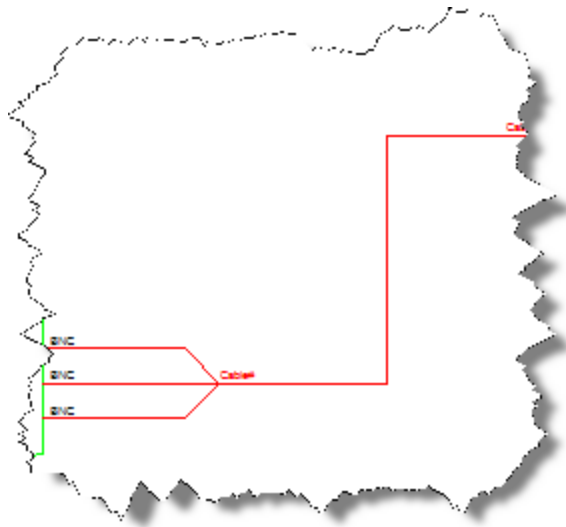
• **Many Cable Database Entries** - Adds a many entries in the cables database and many Cable# text entity at the connection points.

Note: One-to-Many and Many-to-One cables set to Many Cable Database Entries will assign the connection point closest to the cursor when the cable is double-clicked. The first assignment on the cable will enter the database as expected, subsequent assignments will display the Existing Ports dialog prompting you to decide how to number the cables.



3.1.9.3 Many to One Cables

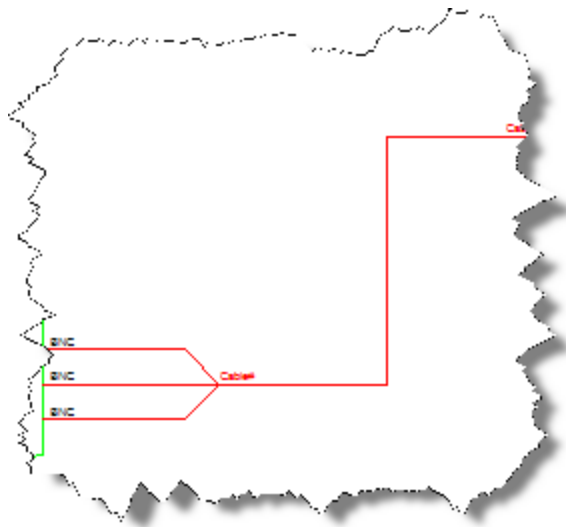
Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Drawing Cables](#)  >



Many-to-One Cable

Use this set of controls to place a cable from many outputs to one input.

- **Count** - Source count.

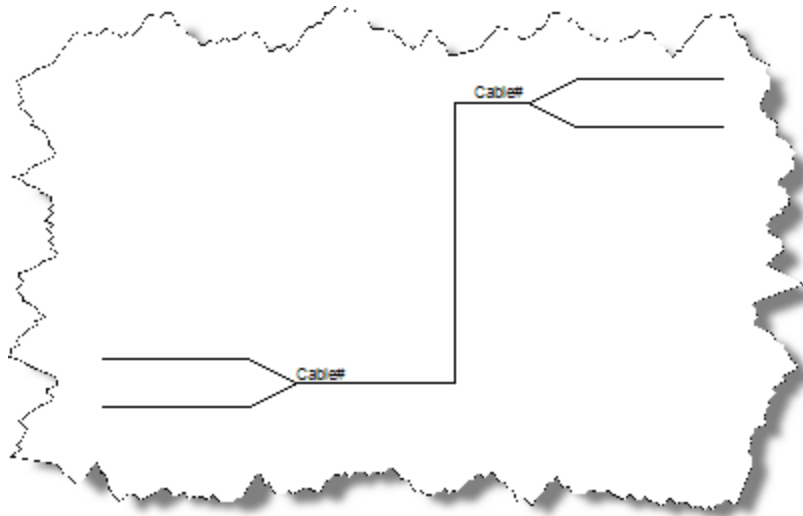


count=3

The remainder of the controls have been explained above.

3.1.9.4 Many to Many Cables

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Drawing Cables](#) [82] >

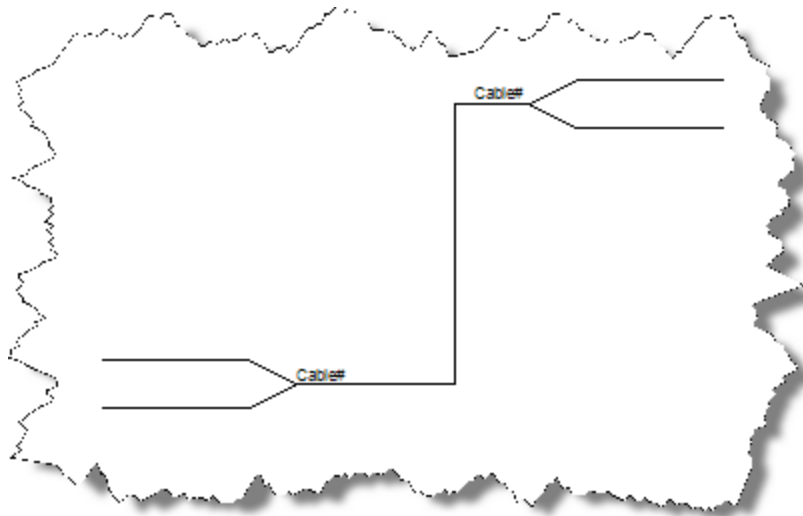


Many-to-Many Cable

Many-to-Many cables behave like a buss. They are a collection of one-to-one cables drawn as a single polyline. When assigning cable numbers the connection point closest to the cursor is used. See the above descriptions for more information about settings.

Related User Settings - **When I double-click a Many-to-Many Cable Assign them All.** This setting overrides the default "point-nearest" behavior.

- **Count** - Source and destination count.

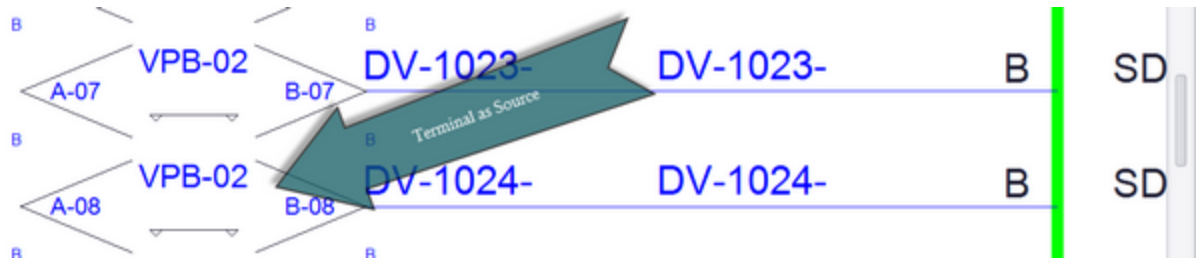


Count = 2
Y feeder

The remainder of the controls have been explained above.

3.1.9.5 Terminal as Source

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Drawing Cables](#) [82] >

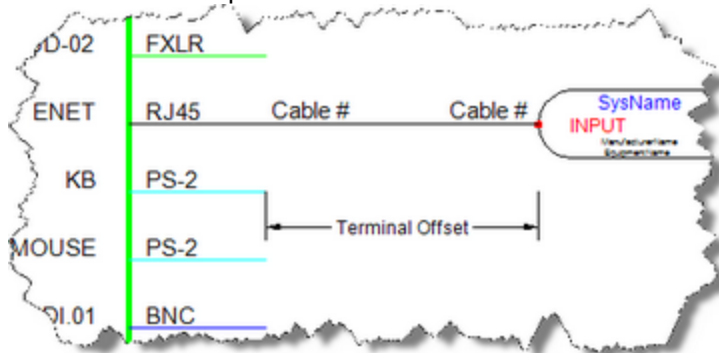


Displays the available Terminals sorted by Terminal style (Jack, Terminal, Pointer).

Note: Terminal file suffixes determine whether the file will be displayed in this window. Files stored in the %BLOCKS%\WireCAD Terminals support path having a _SD.DWG, or _S.DWG suffix will appear in this view.



- **Terminal, Pointer, Jack, Splice, Adapter Gallery** - Displays a gallery of items to click and place in the active drawing.
- **Refresh Button** - Reloads the galleries.
- **Wire Length in 1/100 Drawing Unit(DU).** - The length of the attaching cable or the terminal offset from the attached port.



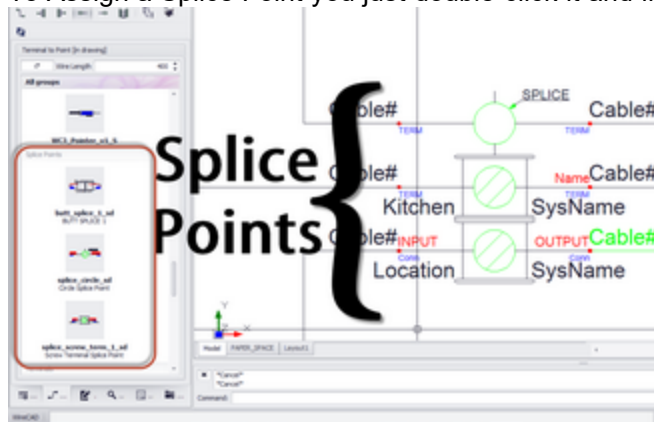
3.1.9.8 Splice Points

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Drawing Cables](#) [82] >



WireCAD supports splice points. If you need a junction in the cable/wire/fiber then use a Splice Point. Splice Points can be found in the Terminals tool panels and are placed like any other Terminal.

To Assign a Splice Point you just double-click it and fill in the data.



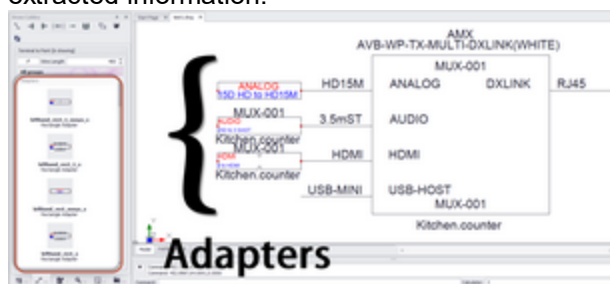
NOTE: if the SysName contains the text `SPLICE` then the duplicate checks and other validations against the Cables and Equipment Lists are ignored. This allows you to have any number of Splice Points all with the same name

3.1.9.9 Adapters

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Drawing Cables](#) [82] >



WireCAD supports the use of Adapters. Adapters are typically used to convert from one connector to another and may also involve the conversion of signal type. Adapters are found in the Terminals tool panels and are placed similar to other terminals. The difference is that instead of a wire being placed between the port and the terminal the Adapter is placed directly on the port. Once placed, the adapter reads the block to which it is being attached and gets the SysName and Port information. It then asks you for the new connector type. Finally it populates the Adapter with the extracted information.



NOTE: if the functional block to which you are attaching the adapter is not yet assigned a **SysName** you will be prompted that the operation cannot complete until you have assigned a **SysName** to the device. You will then need to double-click each attached adapter to get the information into the adapter.

3.1.9.10 Working with Pointers (on-sheet/off-sheet reference)

Navigation: [WireCAD PRO Manual](#) [48] > [Getting Started](#) [48] > [Drawing Cables](#) [82] >



Pointers in WireCAD are a class of Terminal that allows linking two pointers on the same sheet or across sheets.

Unlike other terminals in WireCAD, pointers must be used in pairs.

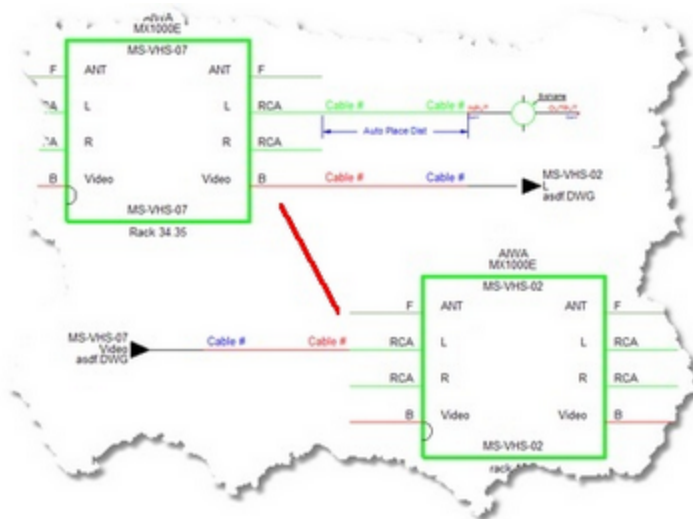
The process of relating one pointer to another is referred to as linking.

There are two pointer families:

1. Pointer_S.dwg and Pointer_D.dwg are the old style pointers. They are still supported but not editable.
2. WC3_<SOMENAME>_S.DWG and WC3_<SOMENAME>_D.dwg are the WC3 family. These are editable. You can change, rename, edit the geometry, etc.

Usage

Pointer pairs that reside on the same sheet can be created and linked automatically by invoking the **[Start Cable]** function while the **Replace Cable with Pointers** checkbox checked.



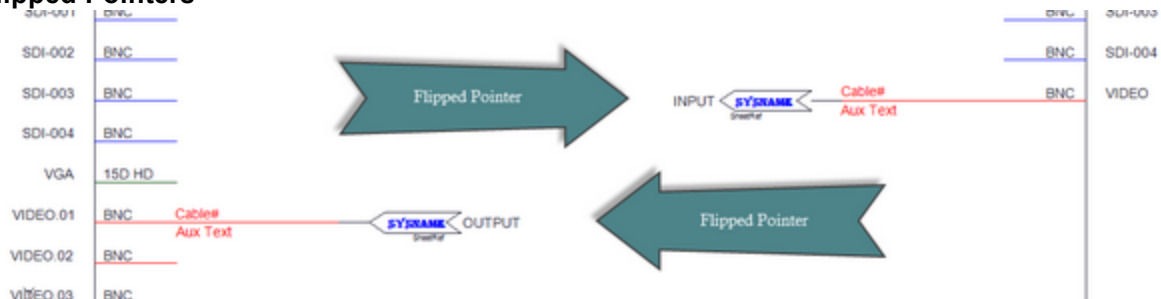
Automatically draw Pointers instead of cables.

In this case the pointer pairs are placed and linked.

Pointer pairs that must be linked across sheets must follow this process:

1. Place the pointer on the source sheet.
2. Place the pointer on the destination sheet.
3. Double-click the pointer on the **source** sheet. You must start on the **source** side.
4. You will be presented with a list of project drawings. Select the one where the destination pointer resides
5. You will be switched to the other drawing. If it is not open it will be first.
6. Find the destination pointer and click it.
7. The pointers will be updated with the complimentary end data.

Flipped Pointers



Occasionally you may find the need to point out of an **Input** or into an **Output**.

In order to do this, WireCAD provides a set of pointers that have a reversed geometry.

The naming convention used is <SomePointerName>_FLIPPED_S.DWG or <SomePointerName>_FLIPPED_D.DWG.

The same process applies as a normal pointer set. Start with the pointer that represents the source side

of the cable and double-click that. Then move to the other side as usual.

3.1.9.10.1 How To: Link Many Pointers at Once

Navigation: [WireCAD PRO Manual](#) > [Getting Started](#) > [Drawing Cables](#) > [Working with Pointers \(on-sheet/off-sheet reference\)](#) >



Explanation

This process links multiple pointers across sheets or on the same sheet at once.

1. Place the pointers on the source sheet.
2. Place the pointers on the destination sheet.
3. Select the pointers on the **source** sheet. You must start on the **source** side.
4. You will be presented with a list of project drawings. Select the one where the destination pointer resides.
5. You will be switched to the other drawing. If it is not open it will be first.
6. Select the same amount of pointers on the destination sheet.

7. The pointers will be updated with the complimentary end data.

The process will fail if:

- The counts do not match.
- The pointer families are found to be different.
- The process is not started on the source side.

3.1.9.10.2 How To: Swap Pointers

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Drawing Cables](#)  > [Working with Pointers \(on-sheet/off-sheet reference\)](#)  >

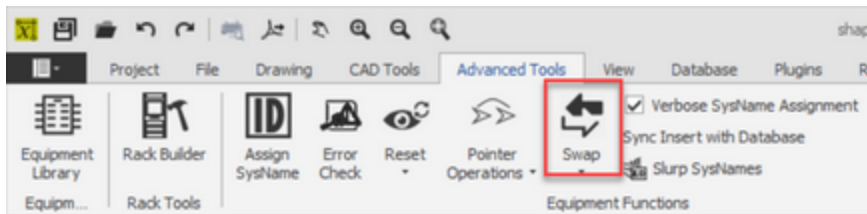


This feature allows you to change all pointers of a selected type to that of another.

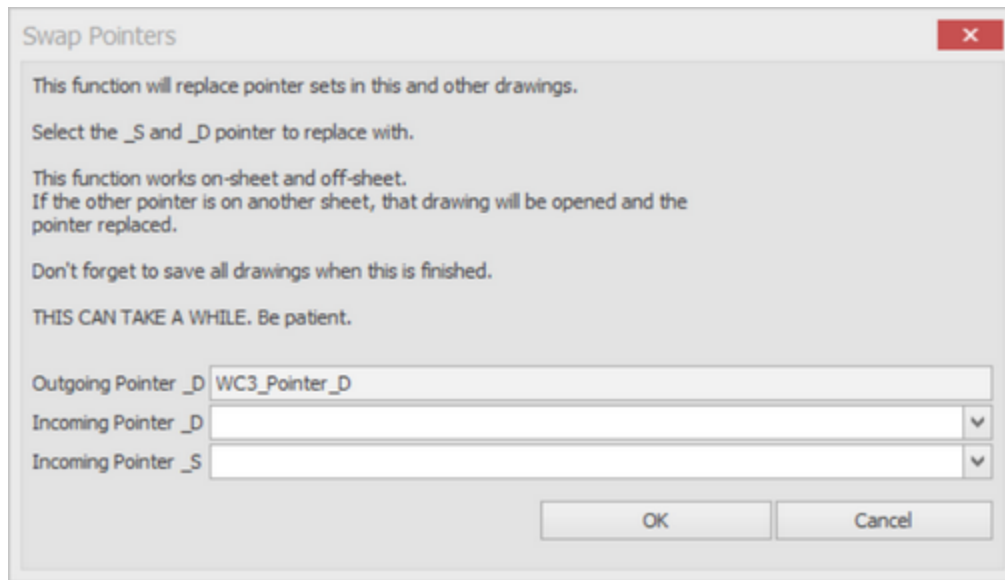
If you get into the project and decide that something about a pointer does not please your eye you can change it easily by following these steps:

1. Select the Pointer you wish to swap.

2. [Advanced Tools], [Swap], [Swap All Pointers]



3. The swap pointers dialogue will open and you can now select the pointer that satisfies your drawing.



4. Click [OK]

3.1.10 Defining Locations

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  >



Menu: **Project Explorer>Project Databases>Locations**

Default command line shortcut: **none**

Related Settings:

Enable Locations Look-up Table = true

You can enable the **Locations Look-up** table. Here you can predefine your locations; thus maintaining referential integrity across all of your locations. In order to use this function you must set **Application Menu > Settings > Project [Locations] Enable Locations Look-up = true**. If this setting is false, WireCAD behaves as it did in previous versions allowing you to type any value in the **Location** field of the **SysName Assignment** dialog.

When using the **Locations Look-up** setting you will need to add locations before you can make use of them in the **SysName Assignment** dialog.

There is one other ramification of using the Locations Look-up: in the Cables table are two new fields **SRCEL** and **DESTEL** (Source Elevation, Destination Elevation). These are now populated as you assign cables. This provides an additional level of control as you create reports.

3.1.10.1 How To: Add a New Location

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Defining Locations](#)  >



1. Click **Project Explorer>Project Databases>Locations**. This opens the Locations table.

2. Click **File>New**.

3. Enter your data and click **[Add]**. This will add a single location to the Locations table.

3.1.10.2 How To: Add Multiple Locations

Navigation: [WireCAD PRO Manual](#) ^[48] > [Getting Started](#) ^[48] > [Defining Locations](#) ^[101] >



1. Click **Project Explorer>Project Databases>Locations**. This opens the Locations table.

2. Click **File>New**.

3. Increment the Count field for the location that you want to add multiple of.

For example, let's say we have 10 racks in the location: LA.THE FACTORY.FL-01.RM-101

4. Enter LA in the Campus field.

5. Enter THE FACTORY in the Building field.

6. Enter FL-01 in the Floor field.

7. Enter RM-101 in the Room field.

8. Enter RK- in the Rack field

9. Enter 10 in the Rack Count field.

10. Click **[Add]**. This will add 10 locations to the Locations table:

LA.THE FACTORY.FL-01.RM-101.RK-01

LA.THE FACTORY.FL-01.RM-101.RK-02

...

LA.THE FACTORY.FL-01.RM-101.RK-10

3.1.11 Assigning Unique IDs (SysNames)

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Getting Started](#) ⁴⁸ >



Menu: **Advanced Tools>Equipment Functions>Assign SysName**

Default command line shortcut: **as**

Alternately: double-click the equipment block in the drawing.

Related Project Settings:

[SysName Format](#) ¹²³

Leading Zeros

Next Numbers Database




WireCAD allows you to provide unique names for each piece of equipment in the project. We refer to this name as a SysName. SysNames must be unique by location. WireCAD can enforce a SysName format or allow free form input.

Edit SysNames Dialog Controls

- **Action** - Clarifies the dialog mode via the use of a radio button selection to determine user intent.
- **Manufacturer** - The manufacturer of the device. This field is usually disabled as the dialog has received the Manufacturer information from an upstream process.
- **Equipment Name** - The model/pn/item number of the device being assigned. This field is usually disabled as the dialog has received the EquipmentName information from an upstream process.
- **SysName** - This is our unique-by-location id. If you manually enter a SysName it must follow the format defined in the SysName Format tool. The SysName textbox will be masked to help you follow the format.
- **Alias** - A functional name for the device. Think of it like the friendly name. The SysName is the unique ID the Alias can be duplicated if desired
- **Location / Elevation** - We recommend that you enter a Location and Elevation. Take your best guess. The Rack Builder tool will use your guesses to create a preliminary rack layout that can easily be modified to suit your final design.
- **User 1-4** - For you. You can change these captions from the **Plugins>Translation Manager**. Search for the key **SysNameUser1 - SysNameUser4** and modify the translated field to be what you want.
- **IP Address / Subnet Mask** - For the IP information if any.


- Power Consumption / Weight** - Pre-populated from the Global Equipment database if exists.
- Flags** - Various flags to help you sort the equipment in your Systems table.

3.1.11.1 How To: Assign a SysName

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Assigning Unique IDs \(SysNames\)](#)  >






This function performs the following steps:

1. Gets the next number in the sequence (based on the [SysName Format](#) ) from the Next Numbers table.
2. Prompts the user for input.
3. Updates the drawing
4. Updates the project systems database.

Note: if the project contains related projects, you will be notified of duplicate SysNames in related projects

1. Create and open a project.
 2. Create and open a drawing.
 3. Add equipment to the drawing.
 4. Set the **Verbose SysName Assignment** checkbox = true. **Advanced Tools>Equipment Functions**.
 5. Double-click the equipment in the drawing.
 6. The **SysName** and **Alias** field will be populated with the next **SysName** in the sequence based in the **Number Format** and the **Next Numbers** table.
 7. Enter any additional data and take your best guess at **Location** and **Elevation**.
 8. Click **[OK]**.
- The drawing will be updated and a new record will be added to the **Project Equipment List**.

3.1.11.2 How To: Assign Multiple SysNames

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Assigning Unique IDs \(SysNames\)](#)  >



1. Create and open a project.
2. Create and open a drawing.
3. Add equipment to the drawing.
4. Set the **Verbose SysName Assignment** checkbox = false. **Advanced Tools>Equipment Functions**.
5. Set the **Default Project Location** and **Elevation** in the **Application Menu > Settings [Project] [Locations]**.
6. Create a selection set of devices to assign.
7. Click **Advanced Tools > Equipment Functions > Assign SysName**.
8. The **SysName** and **Alias** field will be populated with the next **SysName** in the sequence based in the **Number Format** and the **Next Numbers** table.
9. The **Location** and **Elevation** fields will be populated with the defaults in the **Project Settings**.

The drawing will be updated and new record(s) will be added to the **Project Equipment List**.

3.1.12 Assigning Terminals

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  >




Menu: **Advanced Tools>Equipment Functions>Assign Terminals**

Default command line shortcut: **ats**

Alternately: **Double-click the terminal in the drawing.**

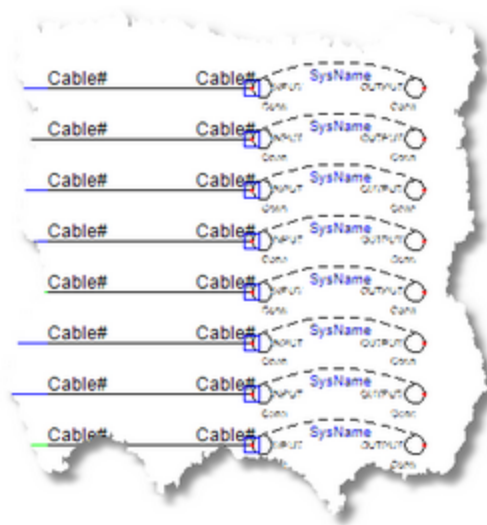
Assumptions:

At least one [SysName](#)  has been assigned.

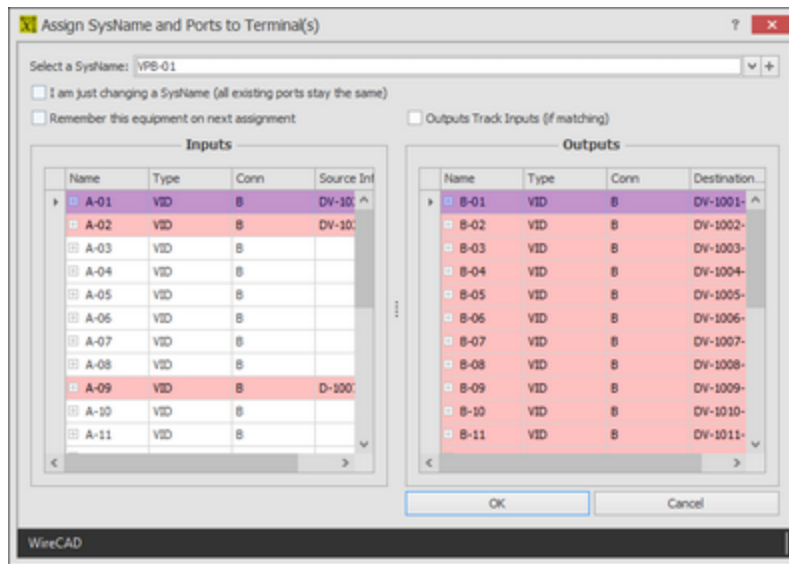
Terminals are one or two port devices. The dwg file defines the geometry.

WireCAD treats terminal assignment slightly differently from a normal SysName assignment.

During a Terminal Assignment we map the existing SysName ID AND I/O port info onto the selected terminal.



Assign Terminal(s) Dialog



- **SysName** - Select the **SysName** or press the **[+]** button to add a new **SysName**
- **I am Just Changing a SysName** - Leaves all port data as currently displayed on a terminal, just changes the SysName. This is useful for modifying existing terminal assignments.
- **Remember this equipment** - Remembers the selected SysName and returns to it on the next terminal assignment.
- **Outputs track inputs** - When you select an input that has a corresponding output of the exact same name, that output will also be selected.
- **Inputs** grid - Active only if the selected terminal(s) have input connection points.
- **Outputs** grid - Active only if the selected terminal(s) have output connection points.

3.1.12.1 How To: Assign Terminals

Navigation: [WireCAD PRO Manual](#) ^[48] > [Getting Started](#) ^[48] > [Assigning Terminals](#) ^[105] >



This function performs the following steps:

1. Opens a port selection dialog based on SysName.
2. Prompts the user to select the SysName and the port or ports to display (or range of ports of multiple terminals are selected).
3. Updates the terminal(s) in the drawing.

1. Create and open a project.
2. Create and open a drawing.
3. Add a terminal to the drawing.
4. Double-click the terminal.
5. Select the **SysName** from the **SysNames** dropdown.
6. Select the **Input** port to apply to the terminal input.
7. Select the **Output** port to apply to the terminal output.
8. Click **[OK]**.

3.1.12.2 How To: Assign Multiple Terminals

Navigation: [WireCAD PRO Manual](#) ^[48] > [Getting Started](#) ^[48] > [Assigning Terminals](#) ^[105] >



This function performs the following steps:

1. Opens a port selection dialog based on SysName.

2. Prompts the user to select the SysName and the port or ports to display (or range of ports of multiple terminals are selected). The order the terminals are added to the selection set is the order that they will be assigned numbers.
3. Updates the terminal(s) in the drawing.

1. Create and open a project.
2. Create and open a drawing.
3. Add terminals to the drawing.
4. Select the terminals to assign.
5. Click **Advanced Tools > Equipment Functions > Assign Terminal(s)** or enter `ats` in the Commandline and click **[Enter]**.
6. Select the **SysName** from the **SysNames** dropdown.
7. Select the top **Input** port to apply to the terminal input.
8. Select the top **Output** port to apply to the terminal output.
9. Click **[OK]**.

WireCAD will assign the terminals from top to bottom in the grid and based on the selection set order.

3.1.12.3 How To: Swap Terminals

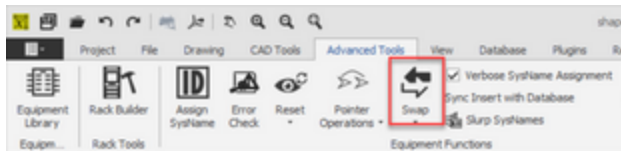
Navigation: [WireCAD PRO Manual](#) ^[48] > [Getting Started](#) ^[48] > [Assigning Terminals](#) ^[105] >



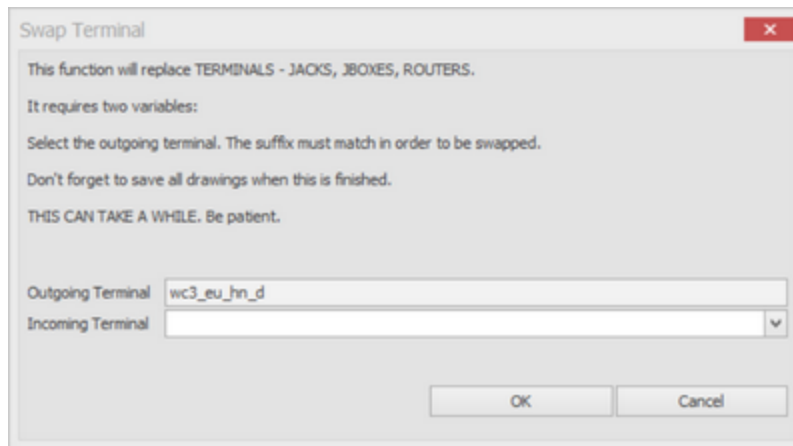
The Swap Terminals feature allows you to change all terminals of a selected type to that of another.

If you get into the project and decide that something about a terminal does not please your eye you can now change it easily.

1. Select the Terminal you wish to swap.
2. **[Advanced Tools], [Swap], [Swap All Terminals]**



3. The swap terminals dialogue will open and you can now select the terminal that satisfies your drawing.



4. [OK]

3.1.13 Assigning Cable Numbers

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  >



Menu: **Advanced Tools>Cable Functions>Assign Cable Number**

Default command line shortcut: **ac**

Alternately: Double-click the cable in the drawing.

Related Settings:

[Cable Number Format](#) 

[Project Settings](#) 

Provides a number for a cable in the drawing and **Cables** table of the project database.


Assumes that the devices on both sides of the cable have first been assigned **SysNames**.

To assign multiple cables at once, create a selection of cables.

The order the cables are added to the select set is the order that they will be assigned numbers.

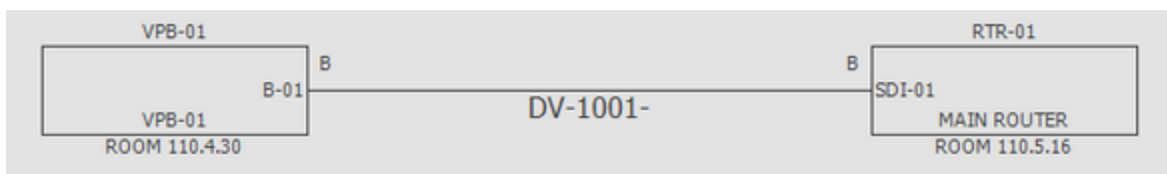
Once you have created a selection of cables, click: **Advanced Tools>Cable Functions>Assign Cable Number**, or type **ac** into the command line followed by the **[Enter]** key.

The Assign Cables function performs the following steps:

1. Gets the next number in the sequence (based on the [Cable Number Format](#) .
2. Prompts the user for input.
3. Updates the drawing.
4. Updates the project systems database.

Edit Cable Numbers Dialog Controls

- **Preview** - This area displays the connection info as found in the drawing.



- **CableNo** - CableNo is a direct entry field as well as a drop down that displays all available cables. When double-clicking an assigned cable, the form will be shown in edit mode. The **[New +]** button will allow the cable number to be fully edited. This field is auto populated with the next number based on the **Cable Number Format** and the **Next Numbers** table.

- **Action** - clarifies the dialog mode via the use of a radio button selection to determine user intent.

- **CableTypeManu, CableType** - Select the **Cable Type Manufacturer** and **Cable Type**. These are populated by the **SignalType** if found. If not, then the project default **Cable Manufacturer** and **Cable Type**.

- **Signal Type** - The incoming signal type. Changing this value may cause a recalculation of the **CableNo** base on the **Cable Number Format**.
- **Integrator** - A user field for assigning the responsibility of the cable.
- **Users1-4** - For you.
- **Sheet** - The drawing file name where the assignment is being made from.
- **Pinout** - If Pinouts are used select the Pinout that matches the two connector types.

3.1.13.1 How To: Assign a Cable Number

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Getting Started](#) ⁴⁸ > [Assigning Cable Numbers](#) ¹⁰⁸ >



1. Create and open a project.
2. Create and open a drawing.
3. Add equipment to the drawing.

4. Add cables to the drawing.
5. Assign **SysNames** to the equipment in the drawing.
6. Set the **Verbose Cable Assignment** checkbox = true. Advanced Tools>Cable Functions. This will display the edit dialog.
7. Double-click a cable.
8. The dialog above will appear with a preformatted cable number in the CableNo field.
9. Add any additional data.
10. Click **[OK]**.

The drawing will be updated and a new record will be added to the **Project Cables** table.

Prerequisites

Two SysNamed devices and one cable.

Related Topics:

[Cable Connection Exists](#) 

3.1.13.2 How To: Assign Multiple Cable Numbers

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Assigning Cable Numbers](#)  >



1. Create and open a project.
2. Create and open a drawing.
3. Add equipment to the drawing.
4. Add cables to the drawing.
5. Assign **SysNames** to the equipment in the drawing.
6. Set the **Verbose Cable Assignment** checkbox = false. Advanced Tools>Cable Functions. This will not display the edit dialog.
7. Create a selection set of cables.
8. Click **Advanced Tools > Cable Functions > Assign Cable Number**.

The drawing will be updated and new record(s) will be added to the **Project Cables** table.

Prerequisites

All selected cables must be attached to devices with SysNames.

Related Topics:

[Cable Connection Exists](#) 

3.1.14 Rack Builder Tool

Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  >



Menu: **Advanced Tools>Rack Functions>Rack Builder**

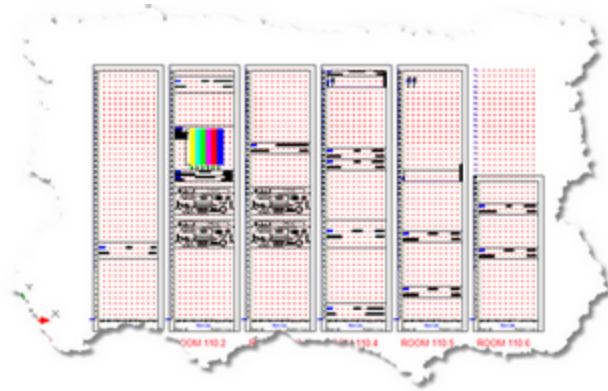
Default command line shortcut: **rb**

Related Settings:

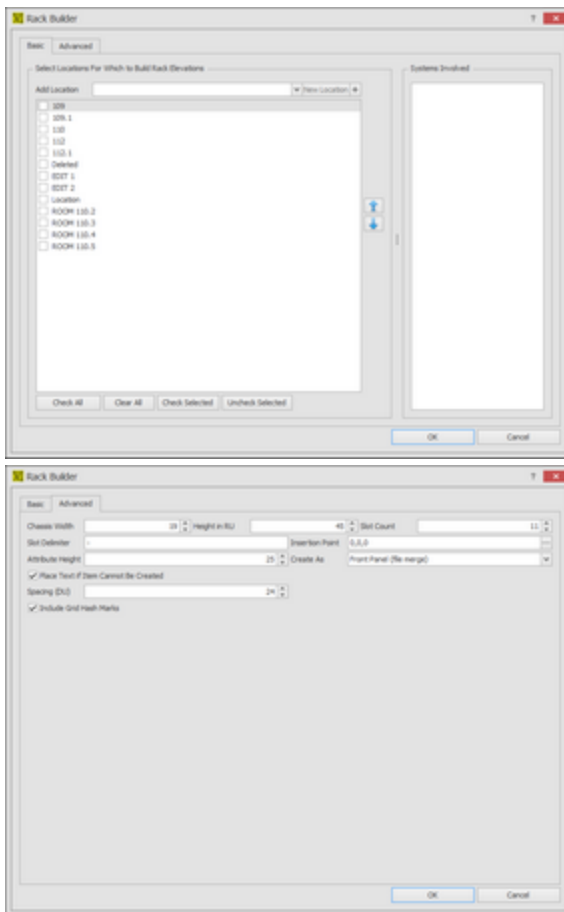
[Default Rack Height](#) 

[Top Down Racks](#) 

The Rack Builder tool utilizes information in the Project Systems table and the global equipment library to place and populate rack elevation views. This process may be run repeatedly as the project progresses.



Rack Builder Dialog



- The **[Basic]** tab allows you to select the locations to include in the Rack Building function. As you select each location, the Systems Involved list will populate.
- Systems Involved** - Displays a list of all the systems that will be placed in the created drawing.
- Chassis Width** - Sets the width of the chassis in DU.
- Height in RU** - Sets the height of the locator grid in Rack Units (RU = 1.75 inches or 4.445cm).
- Slot Count** - Sets the number of slots per locator grid. This is used to position items that may not be located at the insertion point of the rack unit.

- **Slot Delimiter** - WireCAD searches the Elevation field for numeric values first then for the slot delimiter if found it parses the data into two values the elevation and the slot, or in other words how far up in the rack and how far over.

- **Insertion Point** - Where to start the whole process.

- **Attribute Height** - If view rule is not Front Panel (file merge), sets the attribute height of the displayed text.

- **Create As -**

Front Panel (file merge) = use the dwg file found in the equipment definition **BlockRef (Front Panel File)**.

Front Panel (from dims) = use the dimension data from the equipment definition to create a 3D rack block.

From Dims if file not found = Use dimension data if the **BlockRef** is not found.

- **Place Text If Item Cannot Be Created** - If the item cannot be created due to lacking data, place a text marker in the drawing at the location.

- **Include Grid Hash Marks** - This will normally be checked unless you are rebuilding a drawing that already has the locator grids.

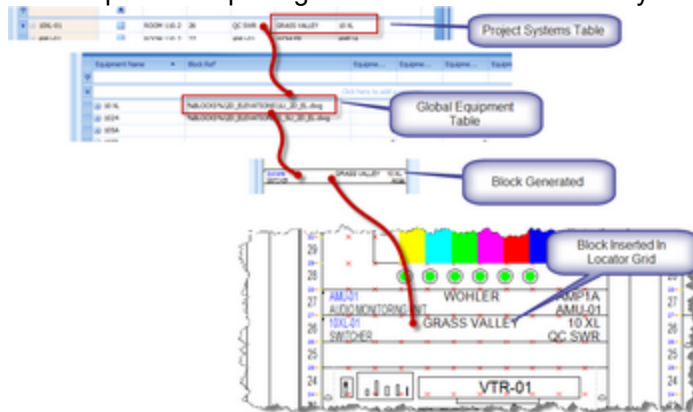
- **Spacing DU** - Sets the location grid spacing in Drawing Units. Measures from left edge to left edge.

3.1.14.1 How It Works

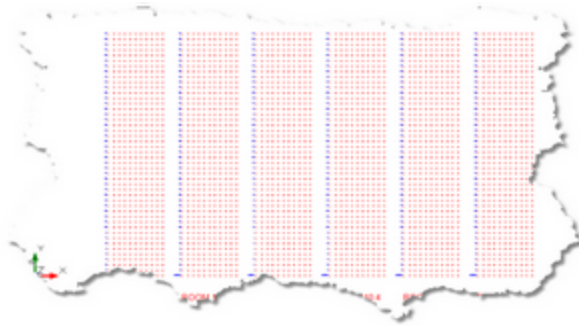
Navigation: [WireCAD PRO Manual](#)  > [Getting Started](#)  > [Rack Builder Tool](#)  >



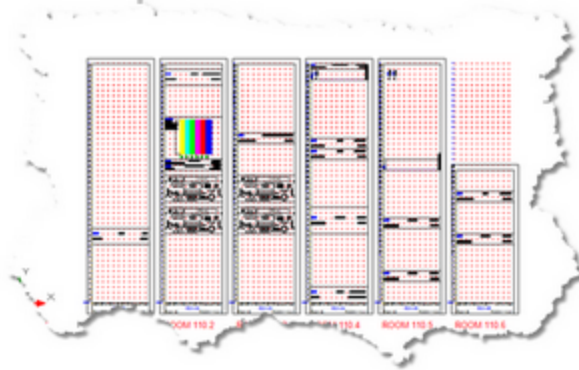
The Rack Builder tool relies on three key pieces of information. First we need the SysName of the device to add to the rack. From the SysName we retrieve the equipment manufacturer and model. Second, using the equipment manufacturer and model, we get the global equipment definition from the global equipment database. If the global equipment definition is complete it will contain either a reference to a front panel dwg file (BlockRef) or dimensional data. If either of these are missing the Rack Builder tool will flag that equipment definition as requiring more information. The Rack Builder tool will perform a pre flight check of all data and let you know what you are missing.



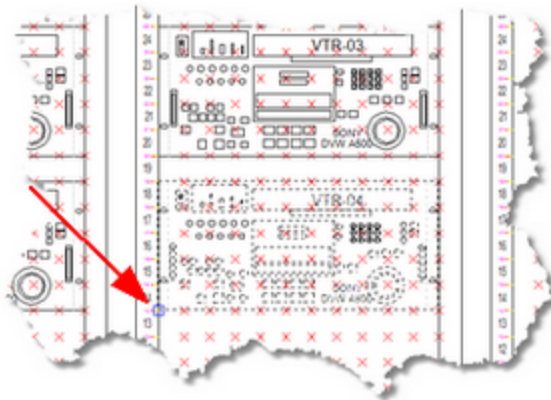
Assuming all of the data fiddly-bits are in the right place, the Rack Builder tool will populate the drawing with one locator grid per location selected. A locator grid is an array of point entities that are spaced horizontally and vertically based upon your selection in the preferences.



Next the Rack Builder tool, based on the Mechanical View Rule, will place either the front panel file or a block created from the dimensional data at the location point defined in the Project System entry.

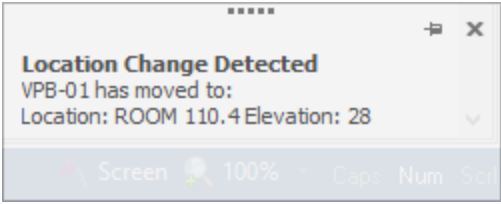


The locator grids facilitate location aware movement of the devices placed on the grid. You may manually place devices created from the equipment library in **Front Panel** mode on the locator grids. To move a device within the rack elevations, select the device, grab it by its grip and move it to the desired location.

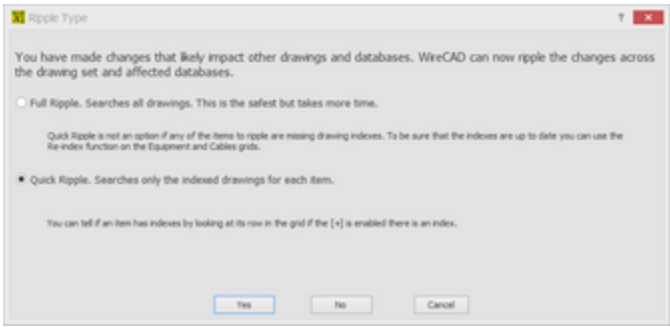


Note: It is desirable to turn on End Point snap and possibly Node Snap while moving devices

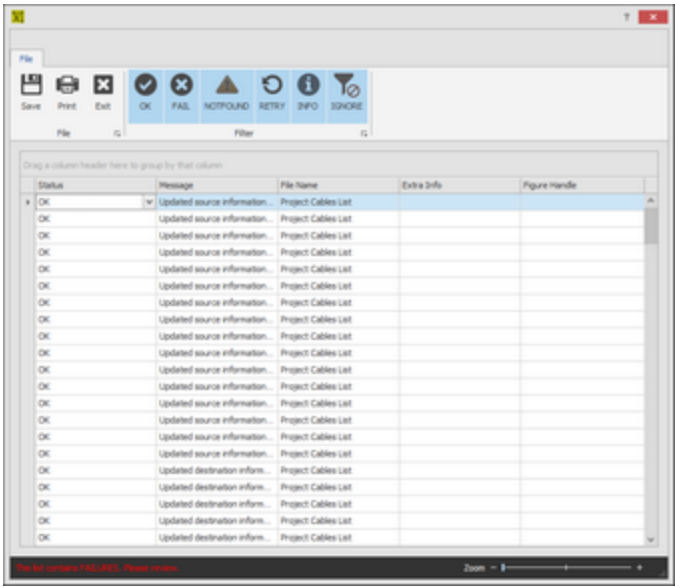
WireCAD will notify you of the location change and update the databases immediately to reflect the change.



It will not update the remainder of the drawing set until you click **File > Save**. You will then be given the opportunity to **Ripple** your changes. Though you are offered the choice you should ALWAYS select **[Yes]** when working in the Rack Builder drawings.



Once changes are made across the drawing set you will be notified of the changed drawings.



3.1.15 Next Numbers Grid

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Getting Started](#) ⁴⁸ >



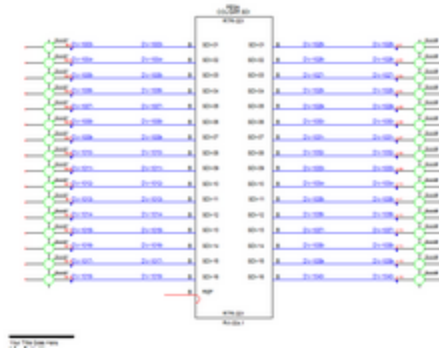
Menu: **Database > Next Numbers**

Default command line shortcut: **NN**

Related Project Settings:

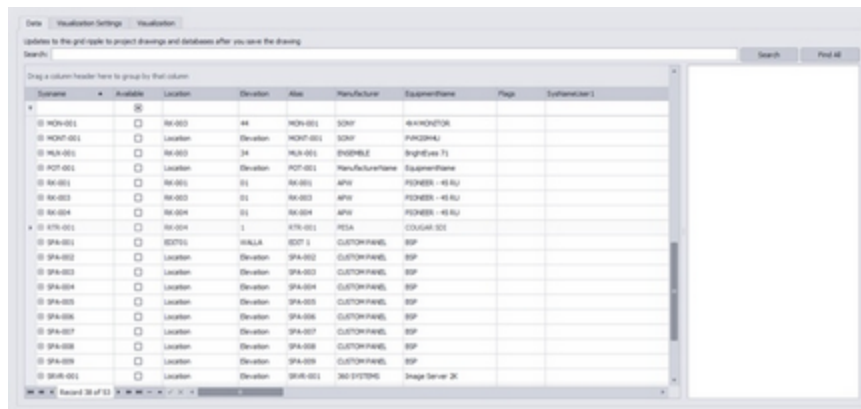
None

Product Level: **ALL**



System Snapshot of all ports and connections

Controls



- **Settings Tab** - Control the output of the Visualizer.
- **Use Last Display Order if Set** - Sorts the ports of the block based on the saved display order if any.
- **Terminal** - The terminal to display.
- **Use Last Saved Appearance** - If there are device level settings saved, use them.
- **Body Width and Pin Spacing** - Appearance settings in 100/DU.
- **Port Data Source** - Pull the port data from the Equipment Library. This will show ALL ports associated with this device definition. Pull the port data from the Cables table. This will show only those ports to which cables have been attached and assigned.
- **Title Block** - Describes the visualization. Salt to taste.
- **Visualization tab** - Does it.

3.2.1.1 How To: Use the Equipment Visualizer

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ > [Equipment List Visualizer](#) ¹¹⁵ >



1. Create a project.
2. Add equipment.
3. Assign **SysNames**.
4. Draw cables.
5. Assign **Cable Numbers**.

6. Open the project Equipment List.
7. Select the **SysName** record to visualize.
8. Click the **Settings** tab.
9. Set the **Terminal**. This is the item that will be displayed as the source and destination of each cable.
10. Click the **Visualize** tab.

3.2.2 Cables Visualizer

Navigation: [WireCAD PRO Manual](#) ^[48] > [Advanced Topics](#) ^[115] >



Menu: **Database>Project Cables>Project Cables Grid**

Default command line shortcut: cg

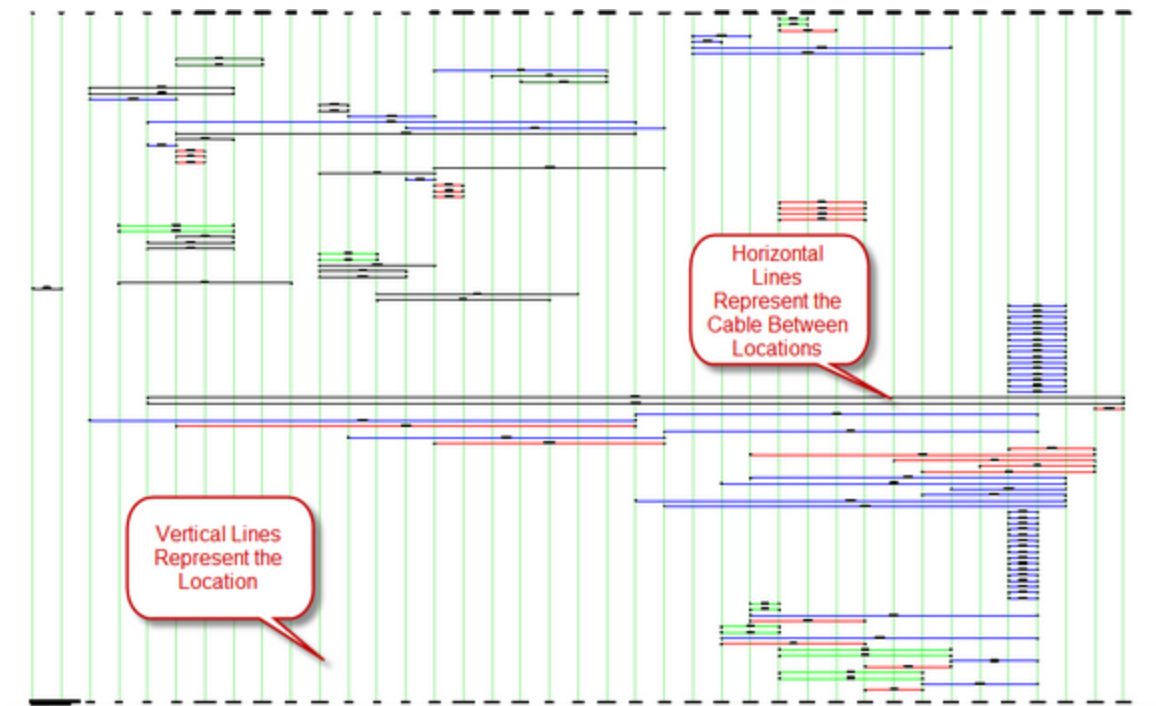
Related Settings:

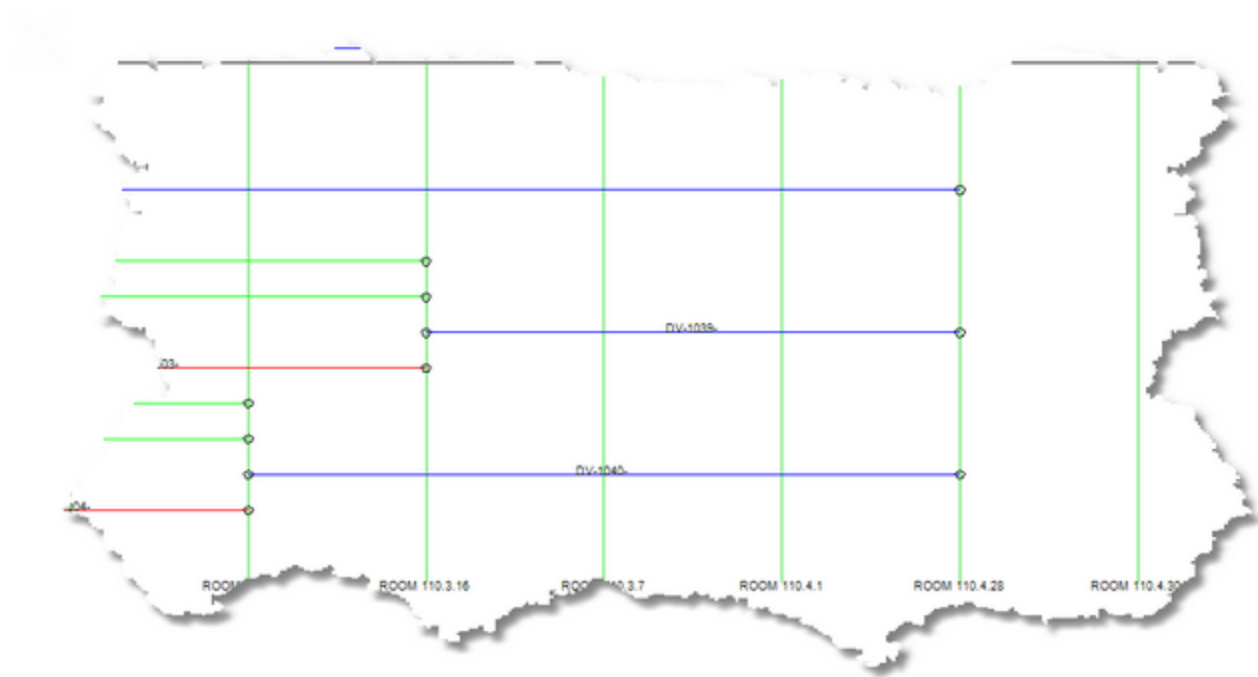
See the Settings tab

The Cables Visualization tool creates a layered diagram (ladder diagram) showing the Locations (vertically) and the cables between them (horizontally).

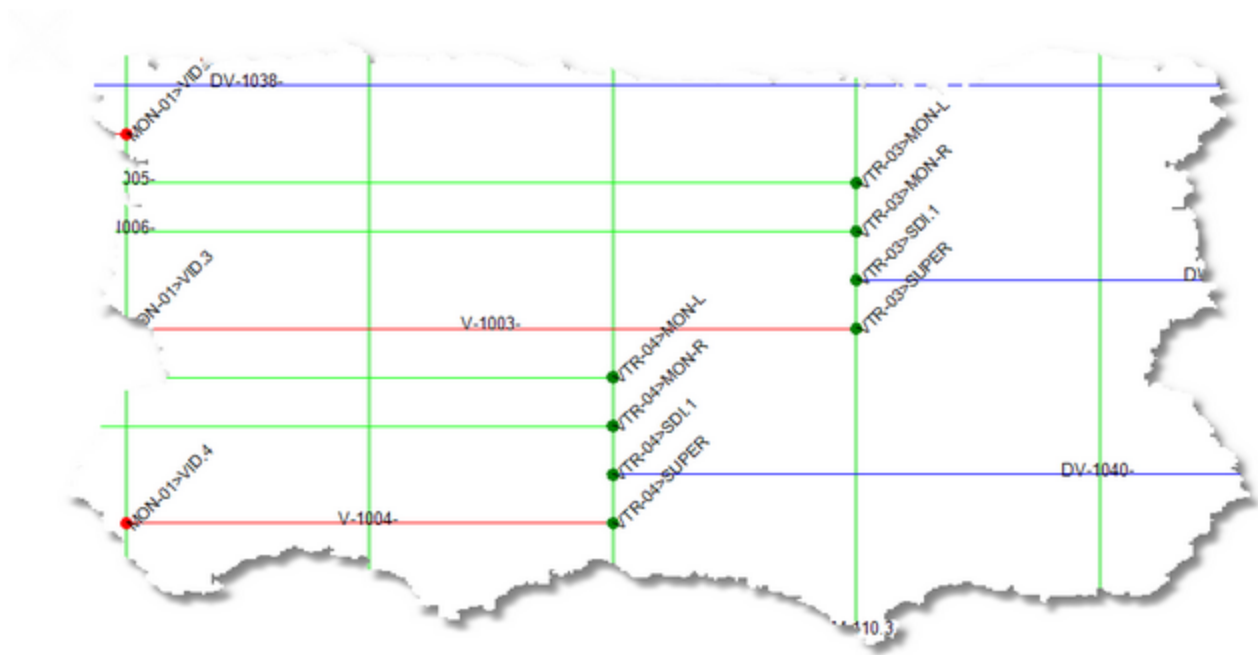
Use this output for:

- Pre-wires.
- Spare inventory.
- General overview.
- Enlighten and amaze your friends.





Visualization without SysNames



Visualization with SysNames

3.2.2.1 How To: Use the Cables Visualizer

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ > [Cables Visualizer](#) ¹¹⁷ >



1. Create a project.
2. Add equipment.
3. Assign **SysNames**.
4. Draw cables.
5. Assign **Cable Numbers**.
6. Open the project Cables grid.
7. Select a group of records to visualize.
8. Click the **Settings** tab.
9. Adjust settings to suit.
10. Click the **Visualize** tab.

3.2.3 AutoScheme Tools

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ >



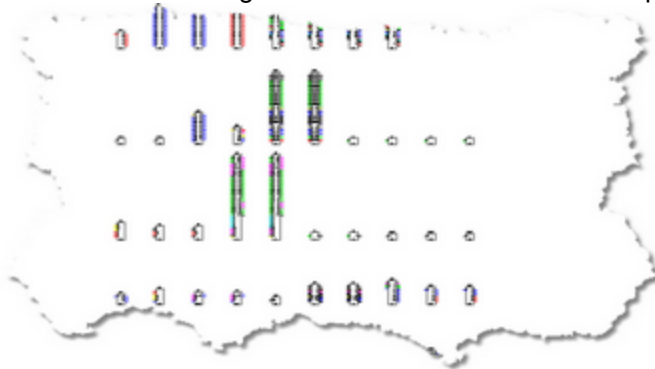
The AutoScheme tools create functional drawings or parts thereof from the data in the project systems and cables databases. This is useful to create detail or overall views.

The AutoScheme tools consist of two utilities. The Auto Block tool automatically places blocks in the drawing on a grid. The Ratsnest tool checks the project cables database against the drawing. If connections defined in the cables database can be reproduced in the drawing because the SysNames and Ports exist, a cable is placed.

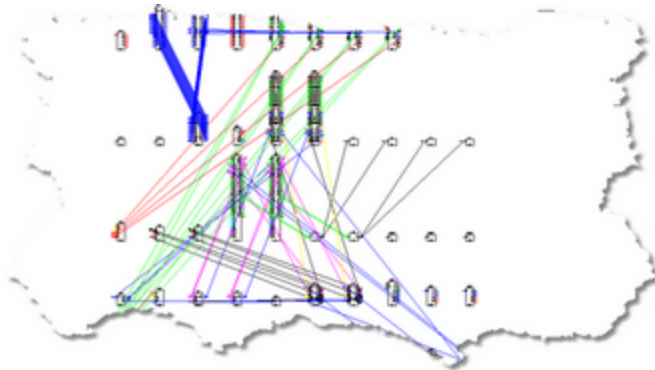
Prerequisites

SysNames and **Cable Numbers** in the project database.

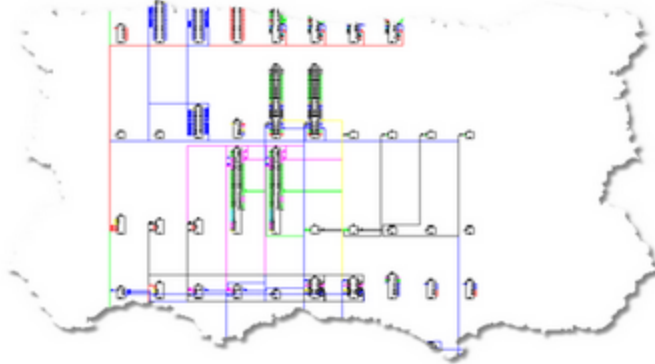
Auto Blocked drawing. Blocks have been created and placed.



After running the Ratsnest tool. This supplies the drawing with cable data and provides a positional reference to help you decide where to move block to better display the drawing.



After running the Cleanup tool from the Ratsnest utility.



3.2.3.1 Auto Block

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ > [AutoScheme Tools](#) ¹¹⁹ >



Menu: **Advanced Tools>AutoScheme Tools>Auto Block**

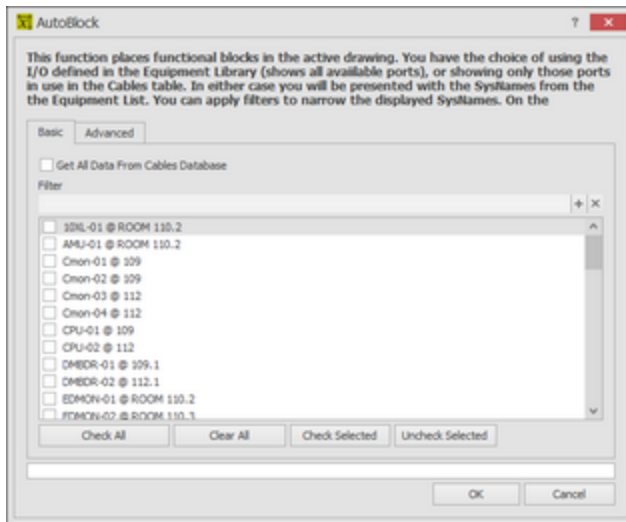
Default command line shortcut: **ab**

Related Settings:

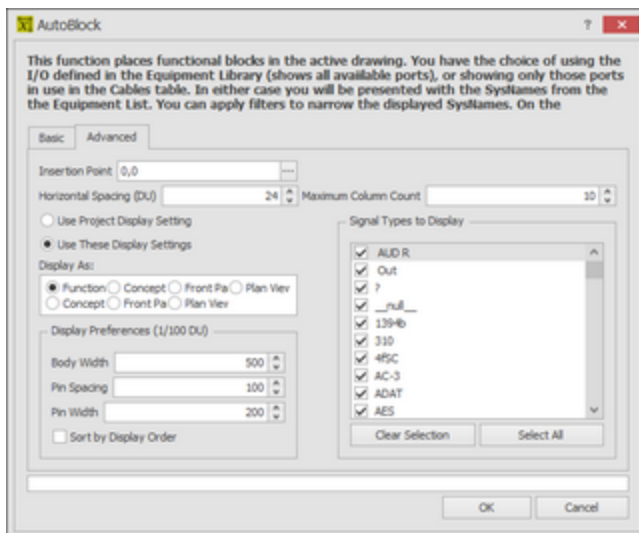
None

The Auto Block tool automatically places functional blocks in the drawing. This tool requires that the Project Systems table be populated.

Controls



- The **[Basic]** tab allows you to determine which systems to add to the drawing.
- Filter** - Filters the list by your criteria.
- The **[Advanced]** tab allows you to refine the behavior of the utility.



- Insertion Point** - The point we start from.
- Horizontal Spacing DU** - How far apart horizontally. The vertical spacing is defined by the height of the highest block in the row.
- Maximum Column Count** - How many columns horizontally.
- Get Port Data From Cables Database** - Select this option to search the cables database for port info instead of the global equipment database. This will effectively show only those ports to which we have attached cables.
- Display As** - How to display the blocks.
- Display Preferences** - If Functional Block or Concept block is selected then set basic display parameters.
- Signal Types to Display** - Filter ports by the selected signal types.

3.2.3.2 RatsNest

Navigation: [WireCAD PRO Manual](#) [48] > [Advanced Topics](#) [115] > [AutoScheme Tools](#) [119] >



Menu: **Advanced Tools>AutoScheme Tools>RatsNest**

Default command line shortcut: **rn**

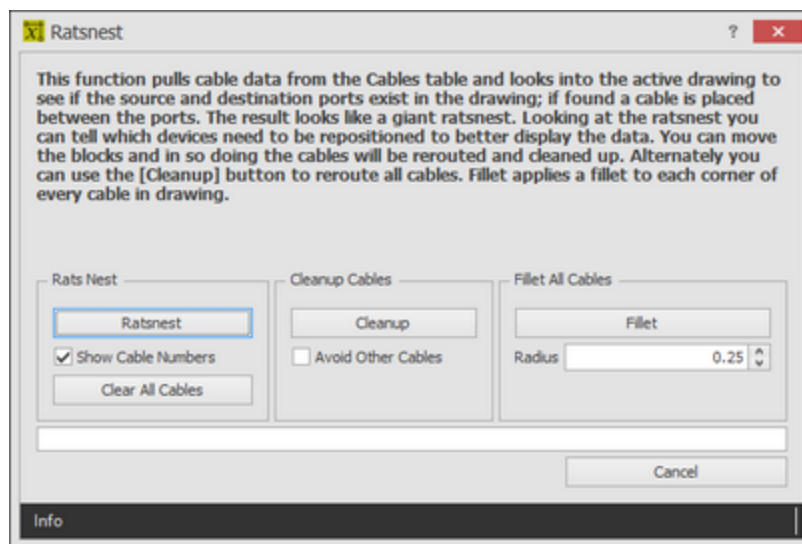
Related Settings:

None

The RatsNest tool checks the project cables database against the drawing. If connections defined in the cables database can be reproduced in the drawing because the SysNames and Ports exist, a cable is placed.

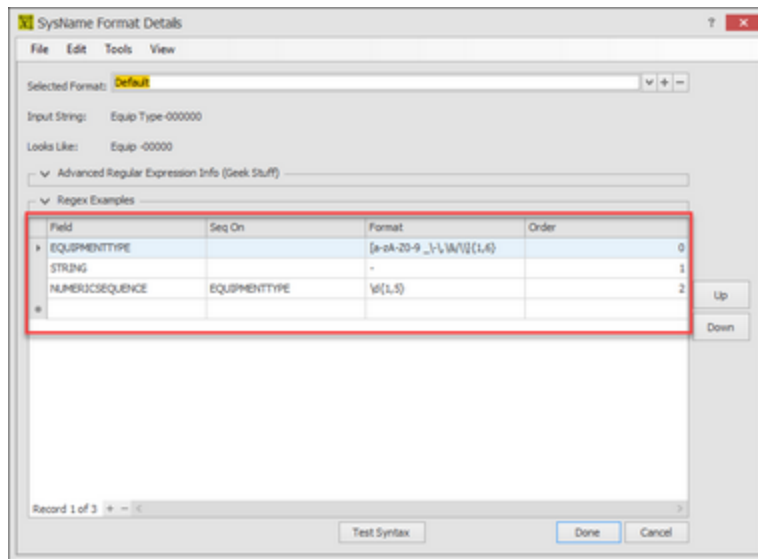
This tool requires that the Project Cables table be populated.

Controls



- This tool has three sections. The RatsNest section does the work of placing the cables in the drawing as defined in the Cables database.
- **[Ratsnest]** - Run the utility to place the cables.
- **Show Cable Numbers** - With or without cable numbers.
- **[Clear All Cables]** - Removes **ALL** cables from the drawing. Not just ratsnested cables. This will remove ALL.
- **[Cleanup]** - Applies the auto-router to all cables in the drawing.
- **Avoid Other Cables** - Auto-router avoids other cables on cleanup.
- **[Fillet]** - Applies fillets to **ALL** cables in the drawing.
- **Distance** - Fillet distance in 100/DU.

•**Variable List** - Displays the sequence of variables in grid view that will be concatenated into the final SysName.



•**SeqOn** - This variable should sequence on another variable's value.

•**Order** - The order in the list of the variable.

•**Available Variables** -

Variable	Description	Can Sequence On	Default (Regex)	Format
MANUFACTURER	Manufacturer Name	X	\w{1,6}	
EQUIPMENTNAME	Equipment Name/PN/Model Number	X	\w{1,6}	
DESCRIPTION	The device description as contained in the global equipment library	X	\w{1,6}	
EQUIPMENTTYPE	The Equipment Type as contained in the global equipment library	X	\w{1,6}	
LOCATION	The location typed in the SysName edit dialog	X	\w{1,6}	
ELEVATION	The location typed in the SysName edit dialog	X	\w{1,6}	
STRING	any string (usually used for delimiters like dashes (-))		-	
NUMERICSEQUENCE	a numeric sequence that starts with the		\d{1,5}	

	project starting number		
ALPHASEQUENCE	an alpha sequence that starts at the letter 'a'		\w{4}
USER1	the global equipment library device definition user1 field		\w{1,6}
USER2	"		\w{1,6}
USER3	"		\w{1,6}
USER4	"		\w{1,6}

More About Regex

The **Format** field uses regular expressions. The following are some simple examples:

\w{1,6}	Any non number, non punctuation string from 1 to 6 characters
\w*	Any non number, non punctuation string any length
[a-zA-Z0-9]{5}	Lower / Upper case and 0 thru 9 exactly 5 characters
\d{1,5}	Digits 1 to 5 digits in length
\d*	Digits any length
\\	a slash "\". The \ is the escape character so you need two
-	a dash except when in [] or {} then a range.
[a-zA-Z0-9_-\.,]{1,4}	Lower / Upper case, 0-9, underscore, dash, and comma.

3.2.5 Cable Number Formatting

Navigation: [WireCAD PRO Manual](#)  > [Advanced Topics](#)  >



Menu: **Project Explorer>Project Databases>Project Cable Number Format**

Default command line shortcut: **fcab**

Related Settings:

None

Allows control of the Cable numbering scheme. Using the Cable Number Format dialog you can select any field that is associated with the Cable Number and concatenate it into your own numbering scheme. Each of the fields selected can be formatted and the numbers can be set to sequence off of another field.

How it Works

The Cable Number format tool is used to generate regular expressions that are then used to parse Cable Number info as well as provide formatting information to masked text boxes.

A note on regular expressions: WireCAD makes extensive use of regular expressions (Regex). Regular expressions are well documented string parsing tools. Much has been written on them. It is not within the scope of this quick start guide to fully explain regular expressions; however, a Google search will tell you more than you ever wanted to know.

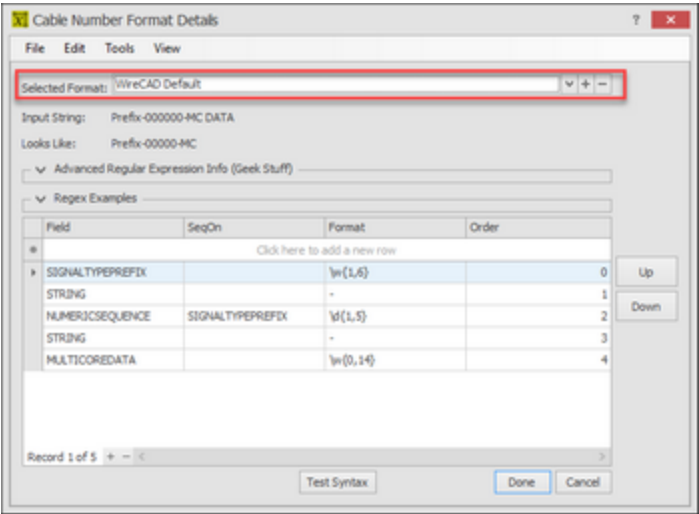
Two forms of regular expression are generated: a simplified form that is used as a mask in both the Cables database and in text boxes, and a complex form that aids in the generation of queries to determine the next number in a sequence. The following are both the simplified and complex Regex for the default cable number format:

```
\w{1,6}-\d{1,5}-\w*

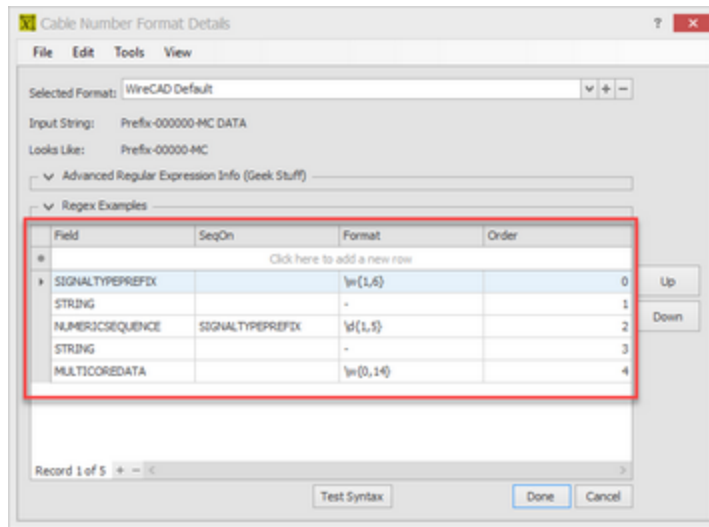
(?<SIGNALTYPEPREFIX>\w{1,6})-(?<NUMERICSEQUENCE>\d{1,5})-(?<MULTICOREDATA>\w*)
```

Controls

- Selected Format** - Selects the current Cable Number format. Allows addition and deletion of formats as well.



- Variable List** - Displays the sequence of variables in grid view that will be concatenated into the final Cable Number.



- SeqOn** - This variable should sequence on another variable's value.
- Order** - The order in the list of the variable.
- Test Syntax** - Performs a basic syntax test on the regular expressions.

•**Available Variables -**

Variable	Description	Can Sequence On	Default (Regex)	Format
SIGNALTYPE	The signal type associated with the cable	X	\w{1,4}	
SIGNALTYPEPREFIX	the signal type prefix associated with the signal type of the cable	X	\w{1,4}	
SOURCESYSTEM	The source SysName	X	\w{1,6}	
SOURCEPIN	The source port or pin name	X	\w{1,6}	
SOURCEALIAS	The source Alias	X	\w{1,6}	
SOURCELOCATION	The source location	X	\w{1,6}	
DESTSYSTEM	The destination SysName		\w{1,6}	
DESTPIN	The destination port or pin name		\w{1,6}	
DESTALIAS	The destination alias		\w{1,6}	
DESTLOCATION	The destination alias		\w{1,6}	
SOURCE_OR_DEST_PIN_IF_JACKFIELD	Not Yet Implemented		\w{1,6}	
MULTICOREDATA	Core data as defined in the associated cable type if any		\w*	

STRING	any string (usually used for delimiters like dashes (-))		-
NUMERICSEQUENCE	a numeric sequence that starts with the project starting number		\d{1,5}
ALPHASEQUENCE	an alpha sequence that starts at the letter 'a'		\w{4}
USER1	the global equipment library device definition user1 field		\w{1,6}
USER2	"		\w{1,6}
USER3	"		\w{1,6}
USER4	"		\w{1,6}

More About Regex

The **Format** field uses regular expressions. The following are some simple examples:

\w{1,6}	Any non number, non punctuation string from 1 to 6 characters
\w*	Any non number, non punctuation string any length
[a-zA-Z0-9]{5}	Lower / Upper case and 0 thru 9 exactly 5 characters
\d{1,5}	Digits 1 to 5 digits in length
\d*	Digits any length
\\	a slash "\". The \ is the escape character so you need two
-	a dash except when in [] or {} then a range.
[a-zA-Z0-9_-\.,]{1,4}	Lower / Upper case, 0-9, underscore, dash, and comma.

3.2.6 Using Your CAD Drawings With WireCAD

Navigation: [WireCAD PRO Manual](#)  > [Advanced Topics](#)  >



With WireCAD you can "WireCADify" your existing non-WireCAD CAD blocks to include WireCAD intelligence; thus making them work with WireCAD. Pretty cool huh?

The process requires at least two steps:

1. WireCADify the block.
2. Add ports to the block.

3.2.6.1 How To: Customize Your CAD Blocks to Work with WireCAD

Navigation: [WireCAD PRO Manual](#)  > [Advanced Topics](#)  > [Using Your CAD Drawings With WireCAD](#)  >



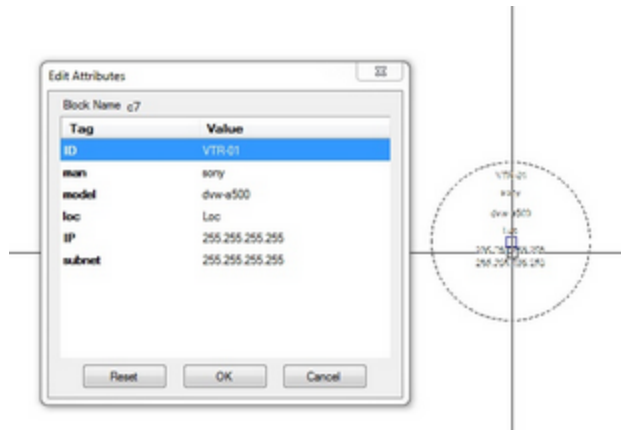
Menu: **Advanced Tools>WireCADify>WireCADify Block**

Default command line shortcut: **wc**

Requires an active drawing with a CAD block.

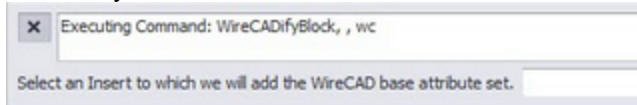
Adds the base WireCAD attribute set to the selected block, mapping your attribute data into our attributes if you choose.

1. Open a drawing with a non-WireCAD CAD block that you want to make work with WireCAD.
2. Click **Advanced Tools>WireCADify>WireCADify Block**.
3. You will be prompted to select a CAD block.

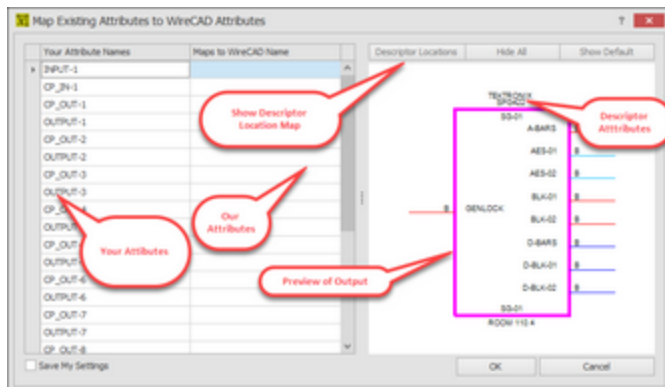


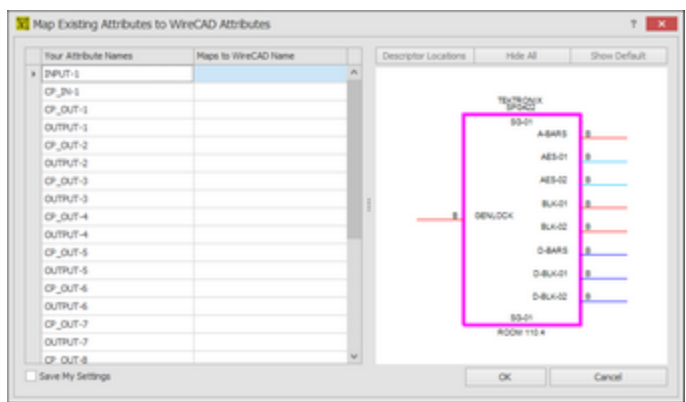
Here we have a CAD block with the displayed attributes (Yours).

4. Select your CAD block. It must be a block and not raw entities.

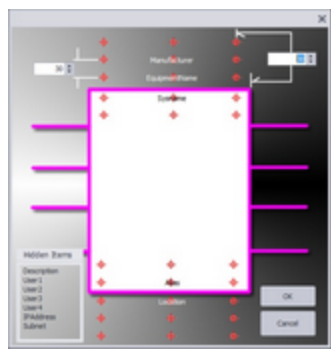


5. Map your attribute to our attributes.

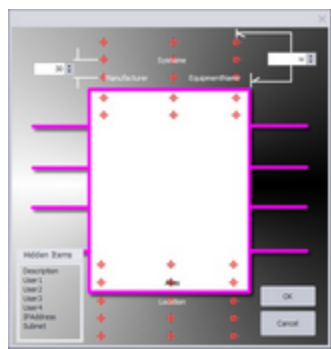




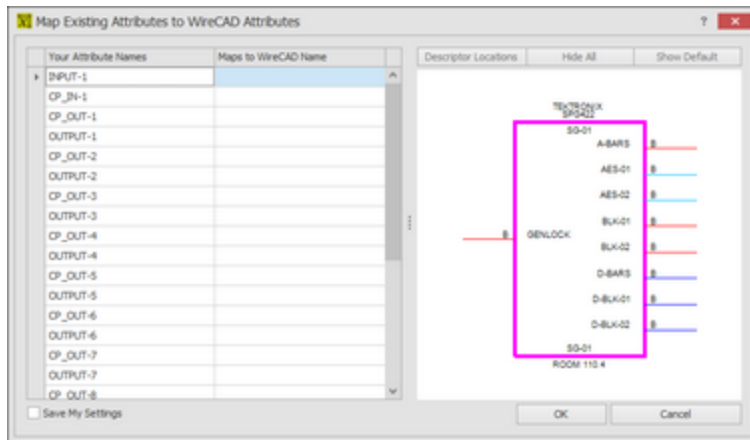
6. Select the [Descriptor Locations] button.



-



Drag to reposition



7. Click **[OK]** to finish.

3.2.6.2 How To: Add a Port to an Existing CAD Block

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ > [Using Your CAD Drawings With WireCAD](#) ¹²⁸ >



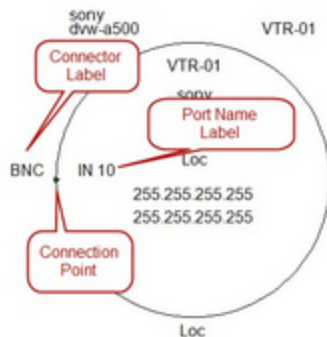
Menu: **Advanced Tools>WireCADify>Add Connection Point**

Default command line shortcut: **none**

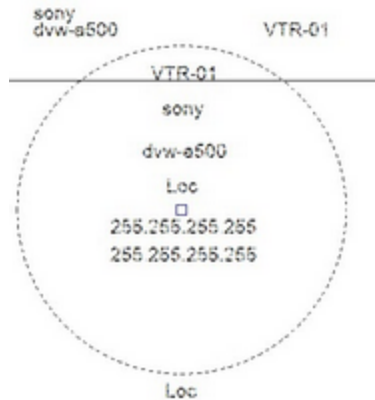
Related Settings:

None

Adds a connection point to the selected block.

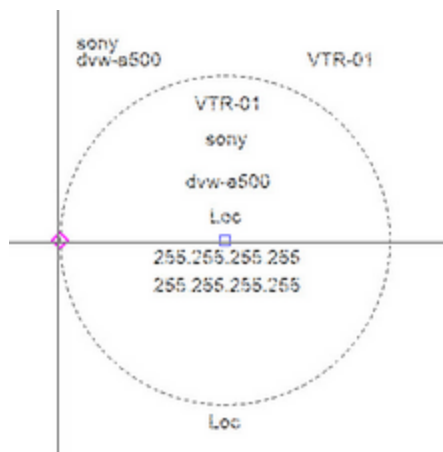


1. Open a drawing with a WireCAD block.
2. Click **Advanced Tools>WireCADify>Add Connection Point**
3. You will be prompted to select a WireCAD block.



Here we have a WireCADified block

4. You will be prompted for the point at which to add the Connection Point.



Here will will choose the arc quadrant

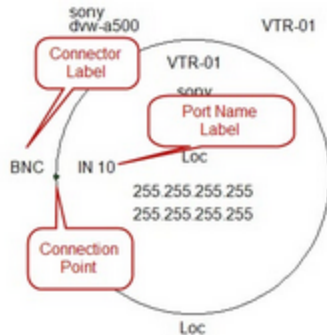
5. Enter your port info into the dialog that is presented.

**Here we will say that the port is an input
called IN10.**

We will show the labels and include a small

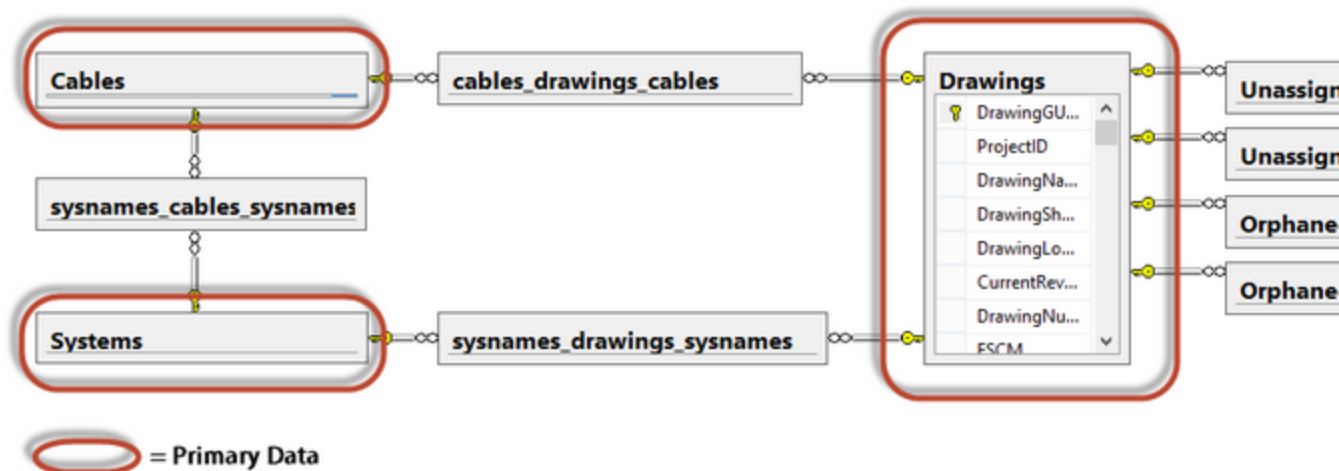
circle to snap to.

6. Click **[OK]** to finish.



3.2.7 Understanding Indexes

Navigation: [WireCAD PRO Manual](#) ^[48] > [Advanced Topics](#) ^[115] >



Related Topics:

- [Unassigned Cables](#) ^[542]
- [Unassigned SysNames](#) ^[542]
- [Cables by Drawing](#) ^[543]
- [SysNames by Drawing](#) ^[543]
- [Cables by SysName](#) ^[544]
- [Orphaned Cables](#) ^[544]
- [Orphaned SysNames](#) ^[545]

WireCAD keeps track of two classes of data.

1. **Primary** - examples of primary data include: SysNames, Cable Numbers, Locations, and Drawings. This data is of cardinal importance. We always want this data to accurately portray what goes on in the drawing/database.

2. **Secondary** - Indexes are secondary or transient data.
The indexes are link data relating the primary data bits to each other.

Index data is transient in nature.

WireCAD consists of a drawing interface with undo/redo and a database which does not. With that as our framework let's discuss a scenario which will put the indexes out of sync quickly.

1. You add a cable to a drawing. A record is added to the Unassigned Cables index.
2. You undo. The cable is removed from the drawing but the database does not have an undo function (nor should it but that's a discussion for another day).
3. We now have a record in the Unassigned Cables index that is in error. It only took two steps to create an inconsistency in the index table.

So rather than trying unsuccessfully to keep every move in sync, we routinely scan the drawing and rebuild the index tables as a background process.

The indexes can be destroyed, rebuilt, and at any given time - out of sync. Deleting all indexes has no effect on primary data.

To insure the most current indexes, run the Rebuild Indexes command.

You can view the indexed data in the primary data grids as sub grids with the [+] button.

You can also view the indexed data in individual grids.

There are seven indexes:

1. **Unassigned Cables by Drawing.** We search the drawing and add any cable that does not yet have a number to this index.
2. **Unassigned SysNames** (equipment) by drawing. We search the drawing for any equipment that does not yet have a SysName and add it to this index.
3. **Equipment by Drawing.** Any equipment that has a SysName and corresponding entry in the Equipment List is added to this index by drawing.
4. **Cables by Drawing.** Any cable that has a number and corresponding entry in the Cables database is added to this index by drawing.
5. **Cables by Equipment.** Any cable that has a number and corresponding entry in the cables database and where both ends of the cable have valid entries in the Equipment list will be added to this index.
6. **Orphaned Cables.** Any cable with a number but no corresponding entry in the Cables database will be added to this index by drawing.
7. **Orphaned SysNames.** Any equipment with a SysName but no corresponding entry in the Equipment list will be added to this index by drawing.

Settings that affect Indexing

The Auto Index setting will scan the project in the background upon project open and any drawing save.

The Show Index Progress setting turns on the main progress bar in the status bar to indicate progress.

3.2.8 Sync Equipment with Database

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ >



Menu: **Context Menu with Equipment Insert Selected>Equipment Functions>Sync Equipment with Database**

Default command line shortcut: **SyncInsertWithDatabase**

Occasionally you will make a decision that puts the data on an insert out of sync with the corresponding record in the **Equipment List**.

This will be manifest by an **Orphaned SysName** record. To remedy the error use the following steps:

1. Select the insert in the drawing that is orphaned. If you are unsure you can always start this function and if the insert matches the database you will be informed as such.
2. Right-click to show the context menu.
3. Click **Equipment>Sync Insert with Database**.

These are the possible outcomes:

1. The insert matches the database. At which point you will be informed as such.
2. The insert partially matches the database. The SysName and Location match but other fields do not. You will be given a choice as to which record to use as the update source.
3. The insert cannot be matched with a record in the database. This results in two other options:
 - a. You will be given the choice to add a new record, or:
 - b. The database is queried for like **Manufacturer** and **EquipmentName** records from which you can select to apply to the insert. Note: only **SysNames** of the same **Manufacturer** and **EquipmentName** will be shown for selection.

Depending on your selection the following will happen:

Outcome	Update Insert in Drawing	Update Database Data	Update Cable Data	Add New Record	Update Attached Pointers
Item Matches Database	-	-	-	-	-
Item Matches SysName and Location Choice to use Insert Data		X			
Item Matches SysName and Location Choice to use Database Data	X		X		X
Item not found in Database Choice is Add				X	

Item not found in database choice is select other existing	X		X		X
--	---	--	---	--	---

3.2.9 Swap Equipment

Navigation: [WireCAD PRO Manual](#) ^[48] > [Advanced Topics](#) ^[115] >



Menu: **Context Menu with Equipment Insert Selected>Equipment Functions>Swap Equipment**

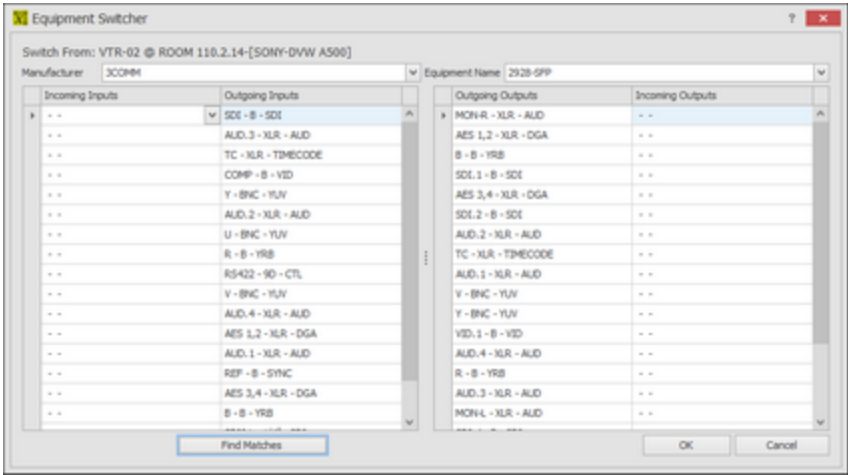
Default command line shortcut: **none**

The Swap Equipment function allows you to swap Manufacturer, Model, and port data on an existing functional block or concept block.

Applies To:
Functional Blocks
Concept (High Detail) Blocks.

Used For:
Switching from one make/model to another.

Controls



- **Manufacturer** - Select the **Manufacturer** to replace the outgoing device with.

- **Equipment Name** - Select the **Equipment Name** / Model / PN to replace the outgoing device with.
- **Inputs** grid, **Outputs** grid - Select the ports to replace.
- **Find Matches** - Matches incoming ports to outgoing ports.
- **OK** - Do it!

3.2.9.1 How To: Swap Equipment

Navigation: [WireCAD PRO Manual](#)  > [Advanced Topics](#)  > [Swap Equipment](#)  >



1. Open a drawing.
2. Select one of the block types in the Applies To section.
3. Right-click to show the context menu.
4. Click **Equipment Functions > Swap Equipment**.
5. Fill in the **Equipment Switcher** form data.
6. Click **OK**.

3.2.10 Edit Block in Place

Navigation: [WireCAD PRO Manual](#)  > [Advanced Topics](#)  >



Menu: **Context Menu with Equipment Insert Selected>Equipment Functions>Edit Block in Place**

Menu: **Context Menu with Equipment Insert Selected>Equipment Functions>Edit Block in Place Ignore Attached Cables**

Default command line shortcut: **ebip, EditBlockinPlaceIgnoreAttachedCables**

Related Settings:

None

Edit Block in Place allows you to modify an existing insert that was created from the Equipment Library. There are two variants of the function. One that attempts to keep all cables attached to the ports from which they started. The other does not.

Because blocks in the CAD file are immutable the function actually creates a replacement block and inserts it into the drawing in place of the other.

Prerequisites

A WireCAD created functional block

Overview

1. The port information is read from the block.
2. The display information is read from the block.
3. Cable information is read from the block.
4. The **Equipment Library** is queried for the incoming device and the Library is placed in Edit Mode.
5. The drawing information is used to create a matching block in the **Equipment Library**.
6. Edit to suit.
7. Update the insert in the drawing.

- 8.Re-route all cables to find their starting ports or not depending on the variant.
- 9.Cables that cannot find their starting ports are modified to show a dashed **LineType**.
- 10.Show a dialog indicating cables that cannot find home.

NOTE: While the port information is being read into the Equipment Library if a port exists on the drawing that does not exist in the Equipment Library it will be added.

3.2.10.1 How To: Edit a Block In Place

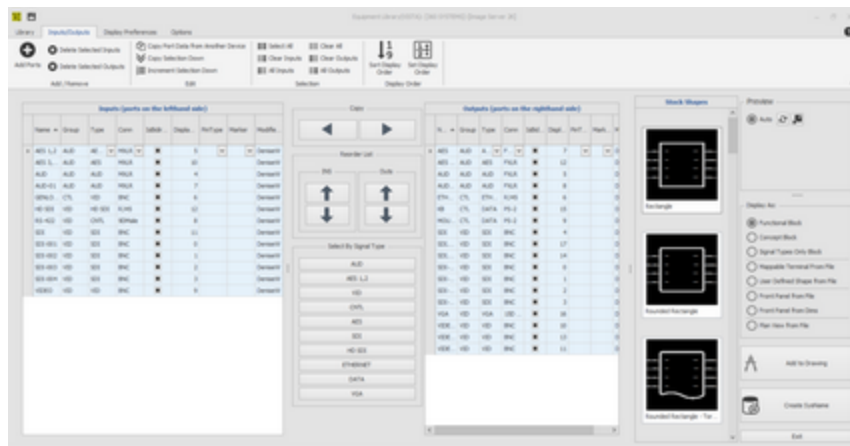
Navigation: [WireCAD PRO Manual](#) ^[48] > [Advanced Topics](#) ^[115] > [Edit Block in Place](#) ^[137] >



- 1.Open a drawing with a WireCAD created functional block.
- 2.Select the block. It must be the only thing in the selection.
- 3.Right-click and select **Equipment>Edit Block in Place**.

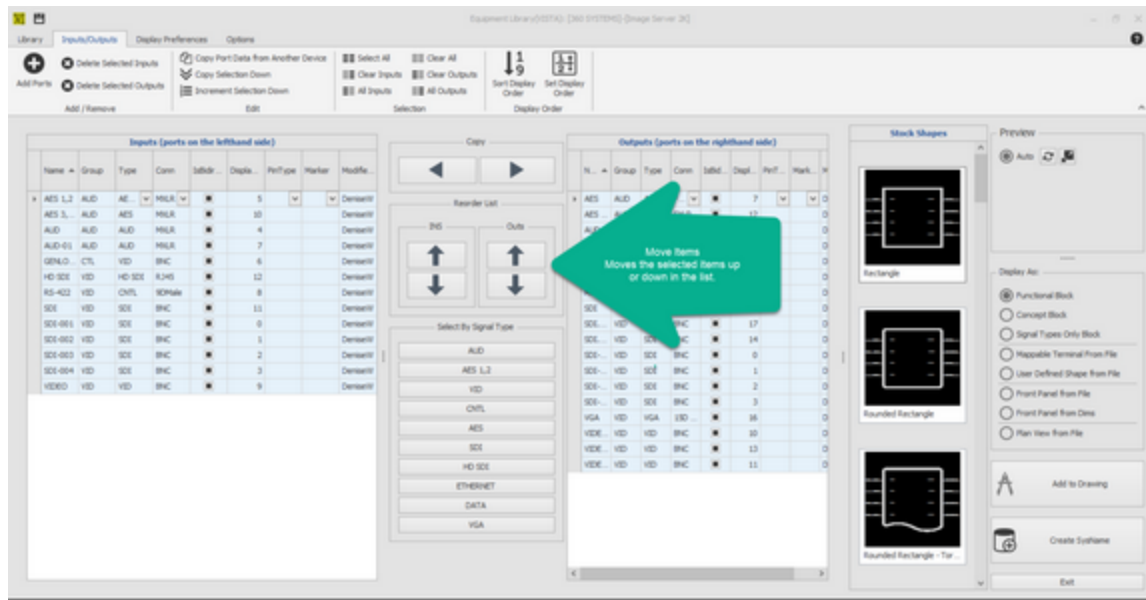


- 4.The block is loaded into the **Equipment Library** for edit.

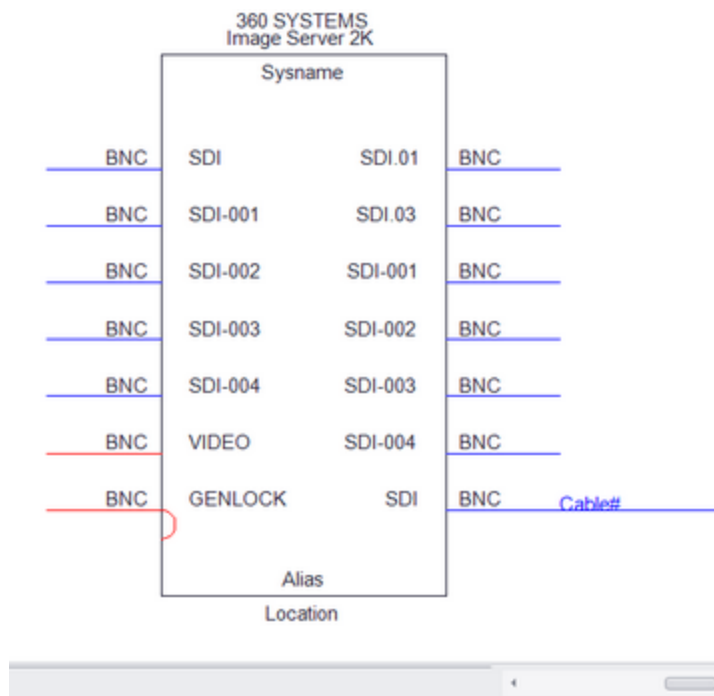


Edit Initiated

- 5.Make changes. Here will move the top SDI port to the bottom.

**List Reordered**

6. Click [Update Insert in Drawing].

**Port moved and cable reattached**

7. Done.

3.2.11 Rereference Pointers

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ >



Explanation

Occasionally you will save pointers to another sheet. This can be because you have done a SaveAs or Copy/Paste.

At that point you will want to change the sheet reference on the pointers. Those pointers that point to the new sheet from other drawings as well as the pointers on this sheet that point to the old drawing will need to be renamed to the new sheet.


Usage

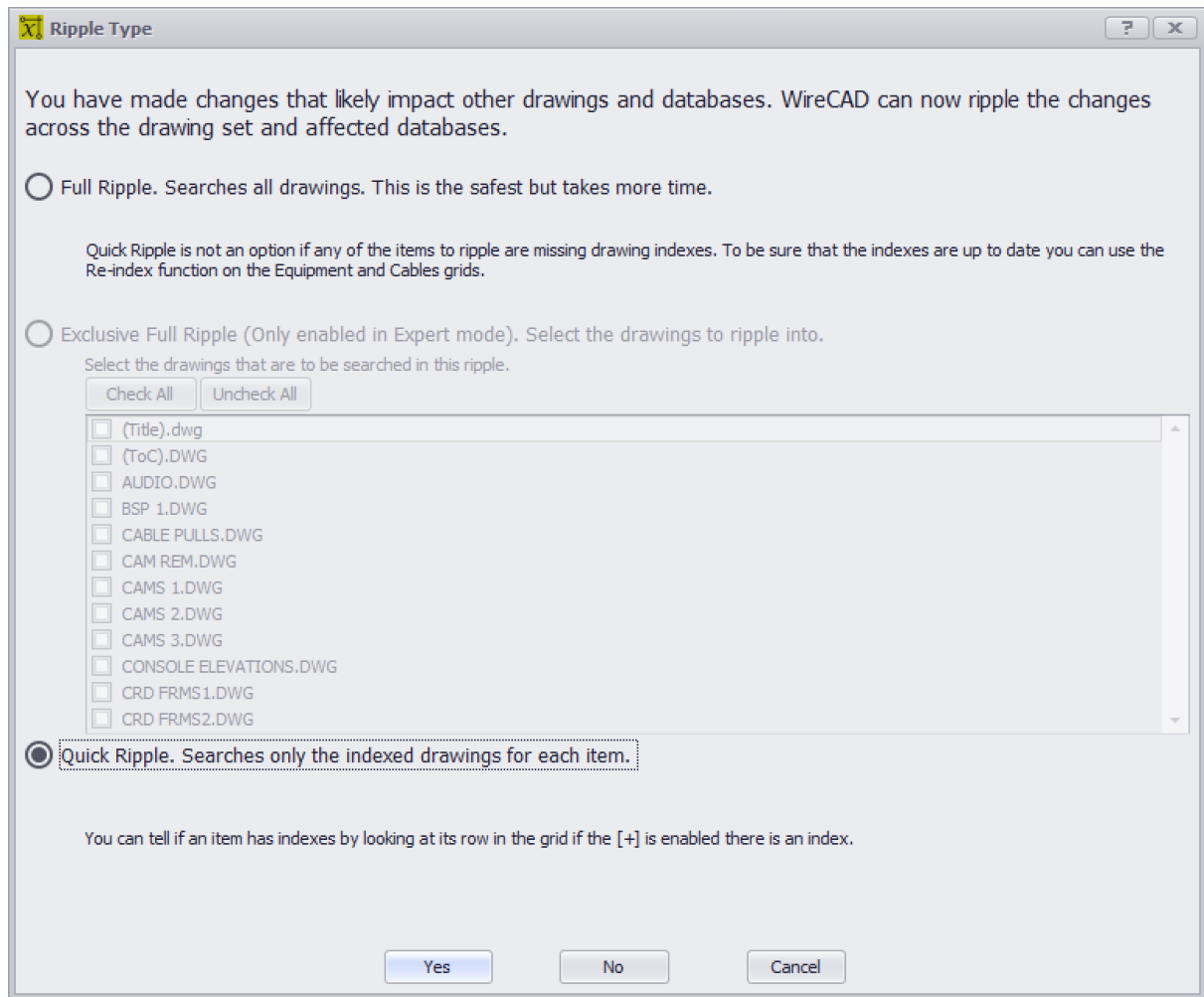
1. Click **Advanced Tools > Rereference Drawing Pointers (Change SheetRef Names)**.
2. **Fill in the dialog** - Set the rename From/To names and where you want to search.

3.2.12 Save and Ripple

Navigation: [WireCAD PRO Manual](#)  > [Advanced Topics](#)  >



Occasionally throughout normal program execution you will be presented with the [Ripple Choice](#)  dialog.



Something has changed that potentially effects data in other drawings and databases. This is your chance to update those other entities in other places throughout the project.

Options

Full Ripple - This is the safest of all ripples and is essentially a find a replace function that searches all drawings. In order for a match to be found the OldSysName and OldLocation fields from the database must match the Sysname and Location field on the block. Then an update to the new information will be made.

Exclusive Full Ripple - [Expert Mode](#)³⁶² only. Don't use this if you don't know exactly what you are doing and why. This option allows you to pick the drawings that you want to do a Full Ripple on.

Quick Ripple - This ripple uses stored index data to update the block by drawing handle. In this ripple we get the index which is a pointer to a block in a drawing and we go get that block and update it.

Applies To:

[Equipment List](#)⁵³²

[Cables](#) ⁵³⁴ database
[Locations](#) ⁵³⁷ database
[Rack Builder](#) ⁴⁴⁸ drawings
[Experience Level](#) ⁴⁵⁶

Related Topics:
[Understanding Indexes](#) ¹³³

3.2.13 Spare Cables

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ >



Menu: **Advanced Tools > Add Spare Cable(s)**
Default command line shortcut: **AddSpareCables**
Related Settings:
None

Place geometry in the drawing and cable(s) in the **Cables** database that represent spare cables.

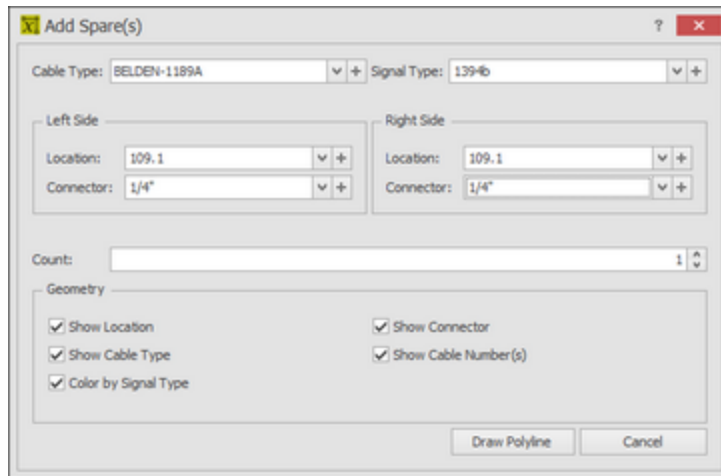
CONF RM	DV-1014- / DV-1018-	LIVEU RK
BNC	1189A	BNC

The following images represent the above settings in both drawing and the **Cables** database.

This function adds **Count** number of cable records to the **Cables** database. The number is automatically formatted in accordance with the [Cable Number Format](#) ¹²⁵ tool and the [Next Numbers](#) ⁵⁴⁸ table.

Used For:
Adding geometry and cable entries in the project Cables database.

Controls



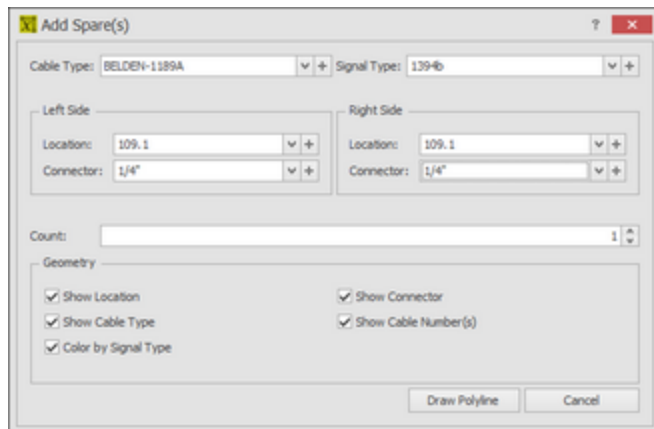
- **Cable Type** - Select the **CableType**.
- **Left Side, Right Side** - set the location and the connector.
- **Count** - how many cable records to enter.
- **Geometry** - What to include in the drawing.

3.2.13.1 How To: Add a Spare Cable

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Advanced Topics](#) ¹¹⁵ > [Spare Cables](#) ¹⁴² >



1. Open a project.
2. Open a drawing.
3. Click **Advanced Tools > Add Spare Cable(s)**.
4. Fill in the dialog.



5. Click **[Draw Polyline]**.
6. The dialog will be dismissed. You are prompted to draw a polyline.
7. Click the points to enter in the polyline.
8. Right-click to finish.

3.2.14 Split Cables

Navigation: [WireCAD PRO Manual](#) [48] > [Advanced Topics](#) [115] >

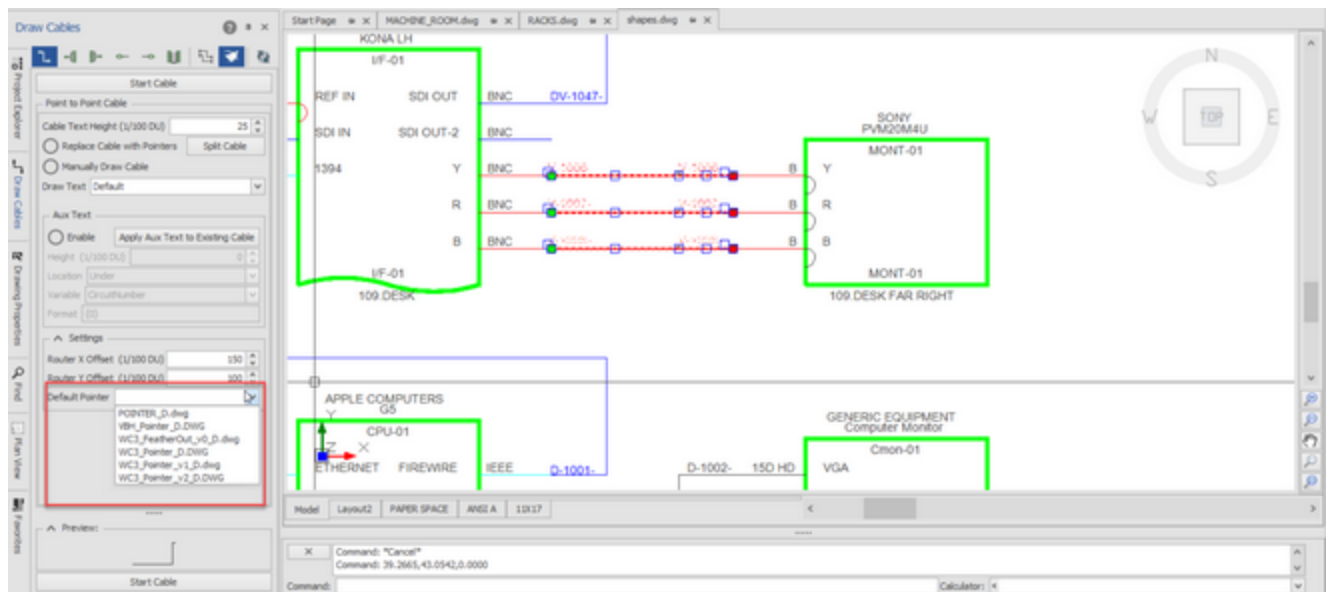


The Split Cables function allows you to replace existing cables, whether assigned or not, with linked pointer sets.

3.2.14.1 How To: Split Cables

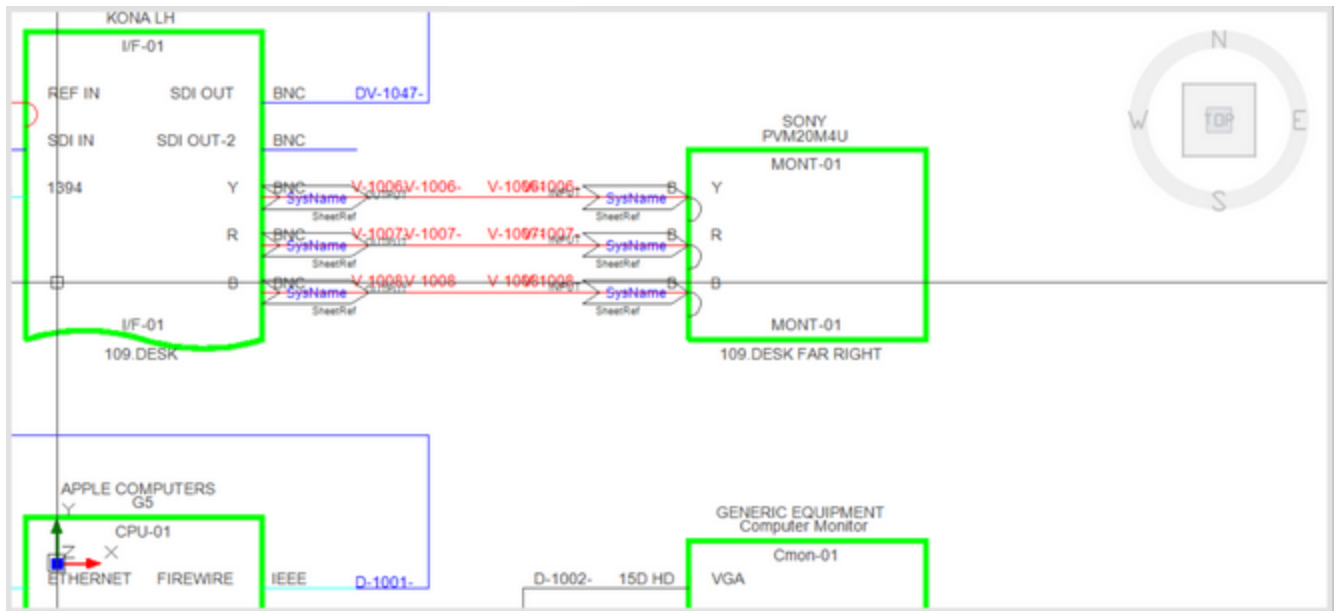
Navigation: [WireCAD PRO Manual](#) [48] > [Advanced Topics](#) [115] > [Split Cables](#) [144] >

1. Select the cable or group of cables in your drawing that you would like to split.
2. Select the Default Pointer that you would like to use. If you select a pointer set that has a full complement of pointers, '_S', '_D' and the flipped '_S' and '_D' versions of your Default Pointer, you will get optimum results out of this function.



3. Select **[Split Cable]**

4. Your end result will look like this:



Hint: Move blocks apart from each other so that you have room for the newly minted pointer. See above example for how not to do it.

3.2.15 Aux Text

Navigation: [WireCAD PRO Manual](#) [48] > [Advanced Topics](#) [115] >



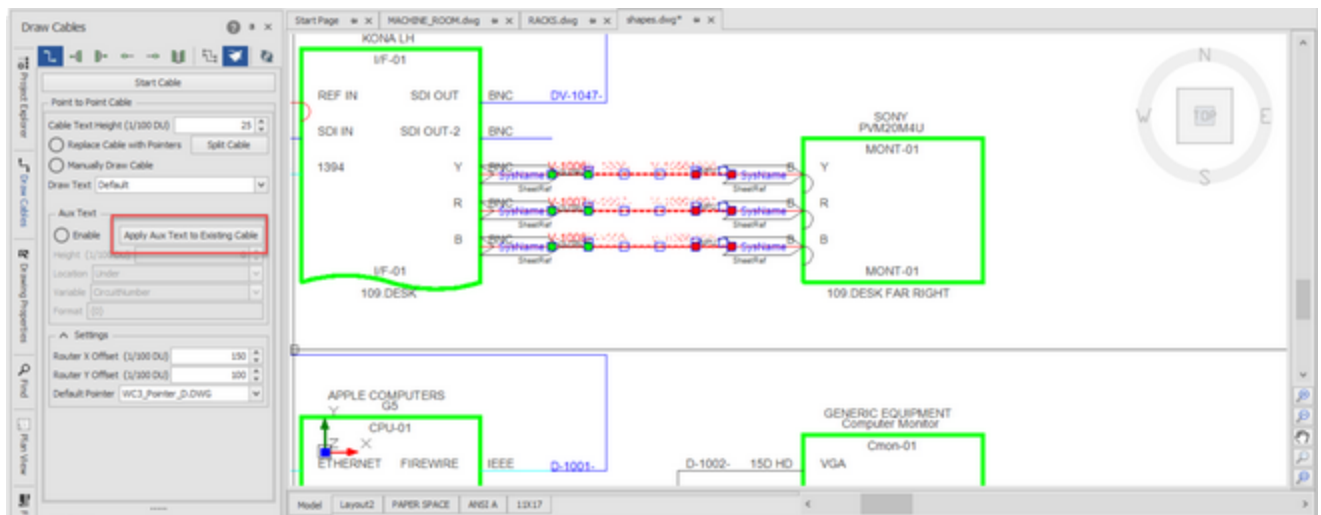
The Aux Text function is for those users that use Aux Text, but forgot to check the box. This function allows you to add the Auxiliary Text without having to redraw a cable.

3.2.15.1 How To: Apply Aux Text to an Existing Cable

Navigation: [WireCAD PRO Manual](#) [48] > [Advanced Topics](#) [115] > [Aux Text](#) [145] >



1. Select the Cables that you would like to Apply Aux Text to.
2. Select **[Apply Aux Text to Existing Cable]** in the Draw Cables menu.



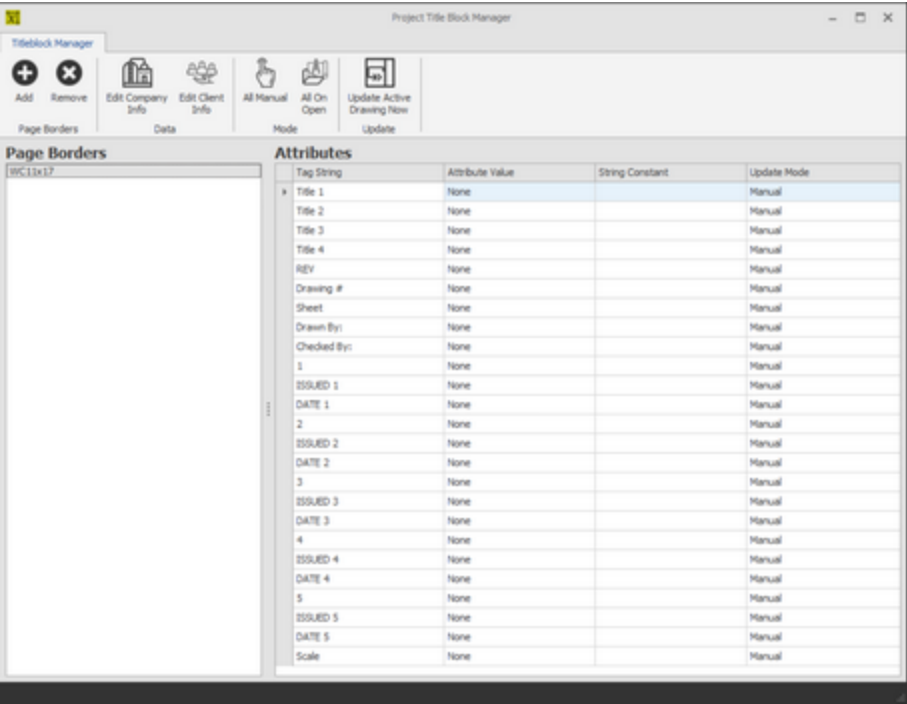
3.2.16 Title Block Manager

Navigation: [WireCAD PRO Manual](#) ^[48] > [Advanced Topics](#) ^[115] >



Menu: **[Project] > [Title Block Manager]**

Default command line shortcut: tbm



The Title Block Manager allows you to edit Attributes in your drawings to reflect across your project.

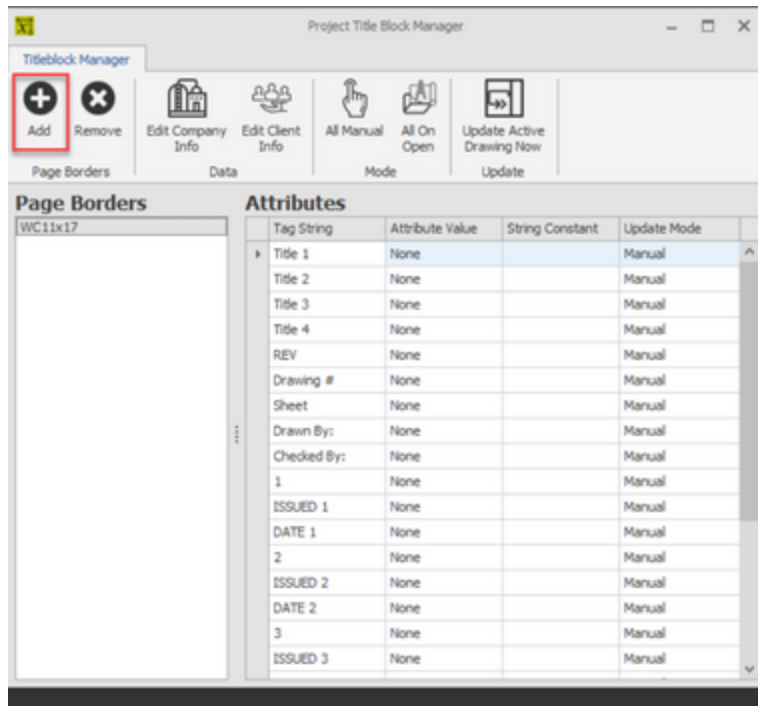
3.2.16.1 How To: Adjust Attributes in the Title Block Manager

Navigation: [WireCAD PRO Manual](#) ^[48] > [Advanced Topics](#) ^[115] > [Title Block Manager](#) ^[146] >

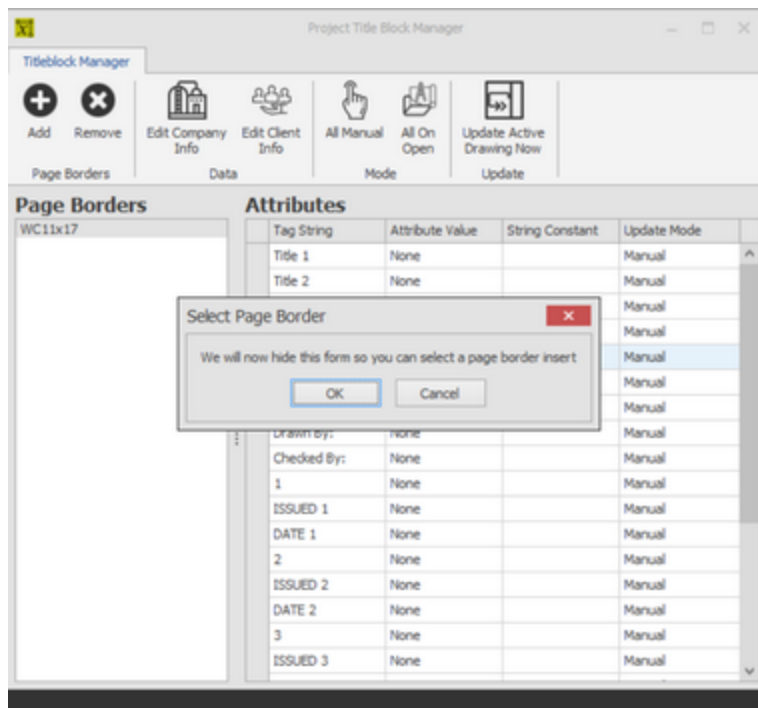


The Title Block Manager will allow you to add a Title Block by adding variables to your available attributes.

1. Open the Title Block Manager. **[Project] > [Title Block Manager]**
If you have not already selected the attributes that you would like to show, simply select **[Add]**

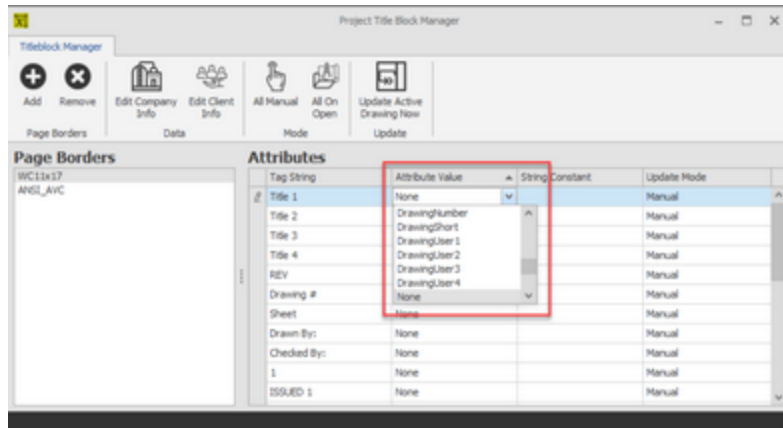


WireCAD will prompt you that it is going to hide the Title Block Manager Dialogue so that you can select a page border insert to work with. Click **[OK]**

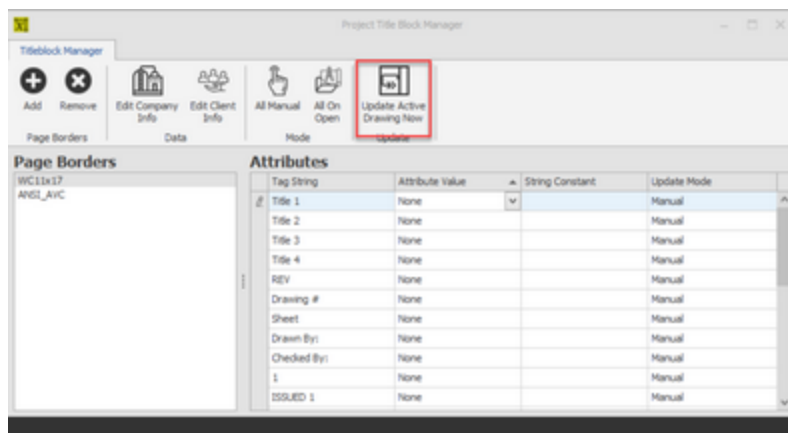


2. Select the Title Block that you would like to manage. The Title Block Manager will re-appear and you will now be able to edit your Attribute Values.

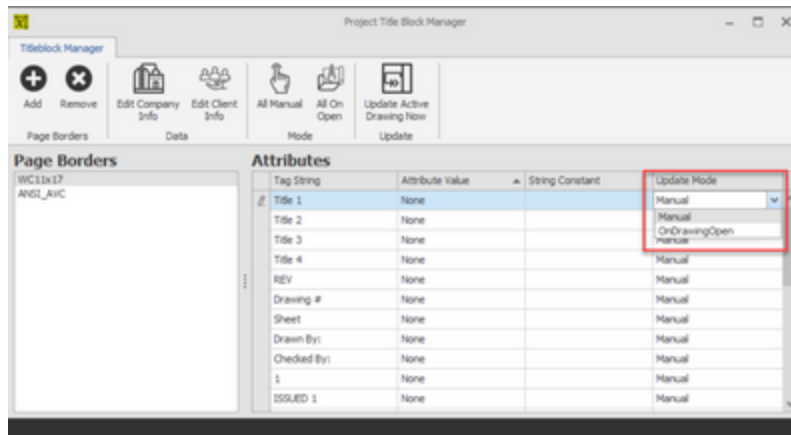
3. Edit your Attribute Values to fit your project needs.



4. Once you have edited your Attribute Values, you can save your changes in one of two ways. The first way you can save your changes is by selecting **[Update Active Drawing Now]**.



You can also set the update settings for each attribute.



5. You are finished.

3.2.17 Understanding Terminals

Navigation: [WireCAD PRO Manual](#) [48] > [Advanced Topics](#) [115] >



Terminals in WireCAD represent a one or two port device and are used to draw compact inline signal flows.

The terminal is a standard CAD block with specific WireCAD attributes. The geometry of the terminal is defined in the CAD drawing file.

In order to be a WireCAD Terminal the following criteria must be met:

- WireCAD Attribute set which includes:

SysName
EquipmentName
ManufacturerName
Location
Alias
SheetRef
Created With:
EquipmentID
WC_TYPE
EquipmentDescription

In addition a Terminal must have at least one invisible Connection Point attribute with the Tag: **CP_IN** or **CP_OUT**.

The **CP_IN** should be placed on geometry that represents the input connection point. Or the point to which we will connect our cable destination.

The **CP_OUT** should be placed on geometry that represents the output connection point. Or the point to which we will connect our cable source.

If a **CP_IN** is defined, the following display attributes may also be used:

INPUT_CONN - optional input connector display name.

INPUT_NAME - optional input display name.

INPUT_TYPE - optional input signal type display name.

If a **CP_OUT** is defined. The follow display attributes may also be used:

OUTPUT_CONN - optional output connector display name.

OUTPUT_NAME - optional output display name.

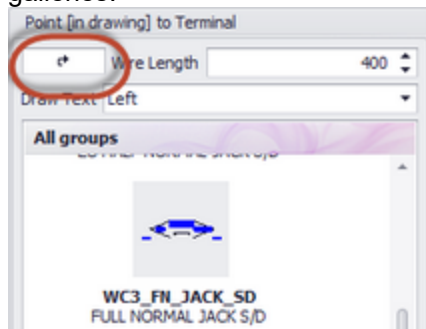
OUTPUT_TYPE - optional output signal type display name.

3.2.17.1 How To: Customize a Terminal

Navigation: [WireCAD PRO Manual](#) ^[48] > [Advanced Topics](#) ^[115] > [Understanding Terminals](#) ^[149] >



1. Open the base CAD file. The terminals are in the %BLOCKS%\WireCAD Terminals path.
2. Modify the geometry and or attributes to suit.
3. Click File > Save As. Remember to keep the _S.DWG and the _D.DWG naming convention intact.
4. Open a drawing that you want to use your shiny new terminal in.
5. Refresh the Terminals tool panel. This will display your new terminal in the place terminals galleries.



6. Click to use as any other terminal.

If the terminal needs changes you will have to do the following:

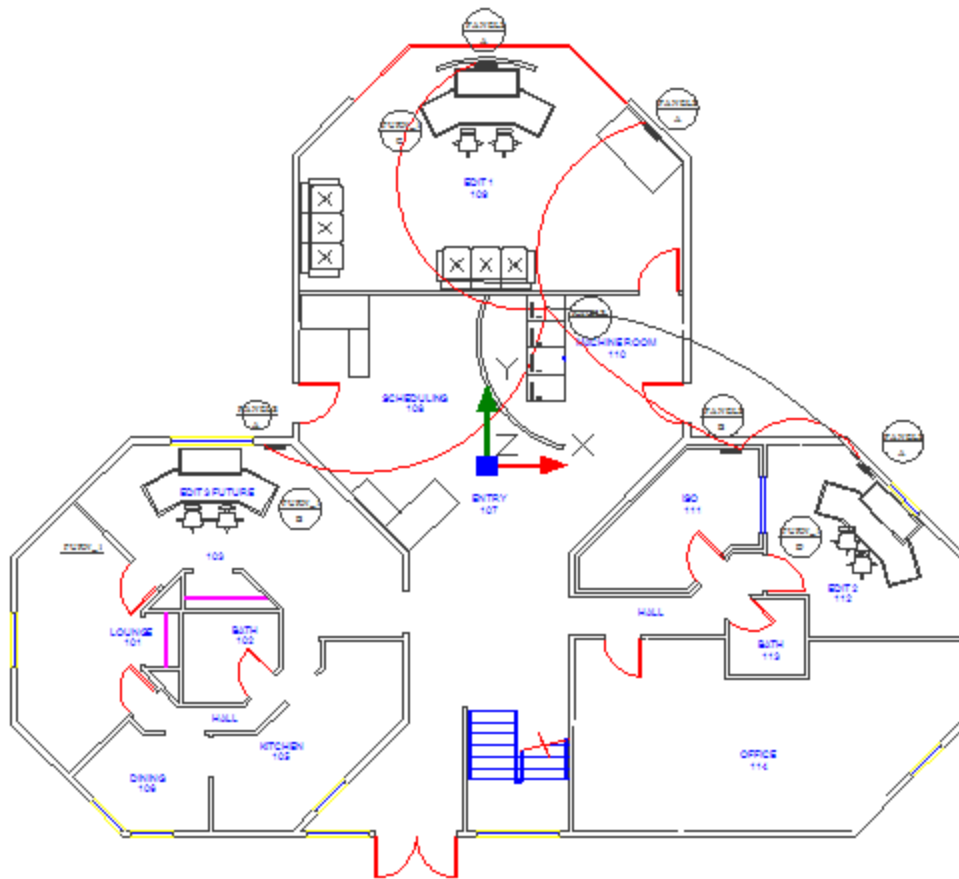
1. Delete all instances of the offending terminal insert from the drawing.
2. Click **File > Purge**.
3. Verify that your file name is in the purge blocks list.
4. Purge the drawing of unused entities.
5. Open the base CAD file again and edit and save.
6. Repeat steps 4 - 6 above.

3.3 Plan View and Layout Tools Dialogs

Navigation: [WireCAD PRO Manual](#) ^[48] >



The following set of tools is targeted at the plan view space.



3.3.1 Take Offs

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Plan View and Layout Tools Dialogs](#) ¹⁵⁰ >



Menu: **Drawing > Plan View and Layout Tools > Take Offs**

Default Commandline shortcut: **showtakeoffs**

Related Settings:

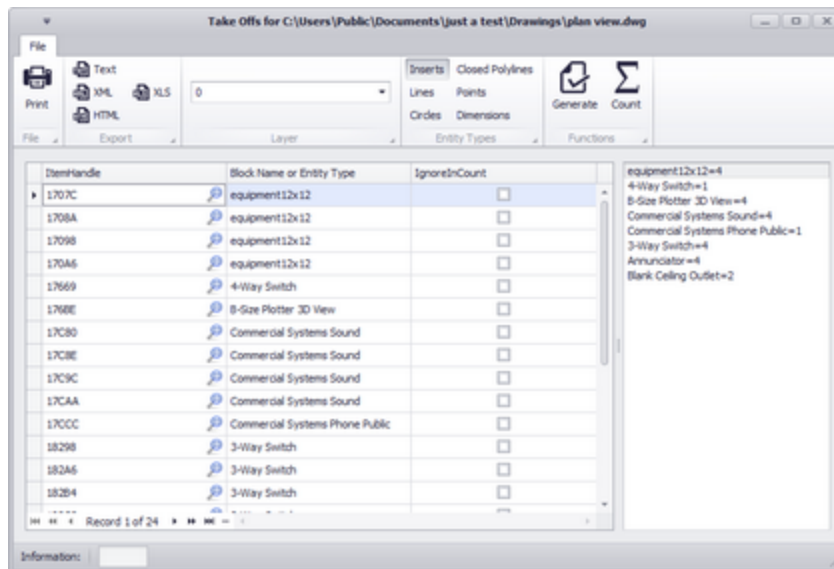
None

Often we need to count items in the plan view space for quoting or other purposes. The Take Offs tool facilitates this process by filtering the drawing for specific entity types on specific layers then counting those instances.

Possible Uses

- Quote preparation.
- Trouble-shooting.

Dialog Options



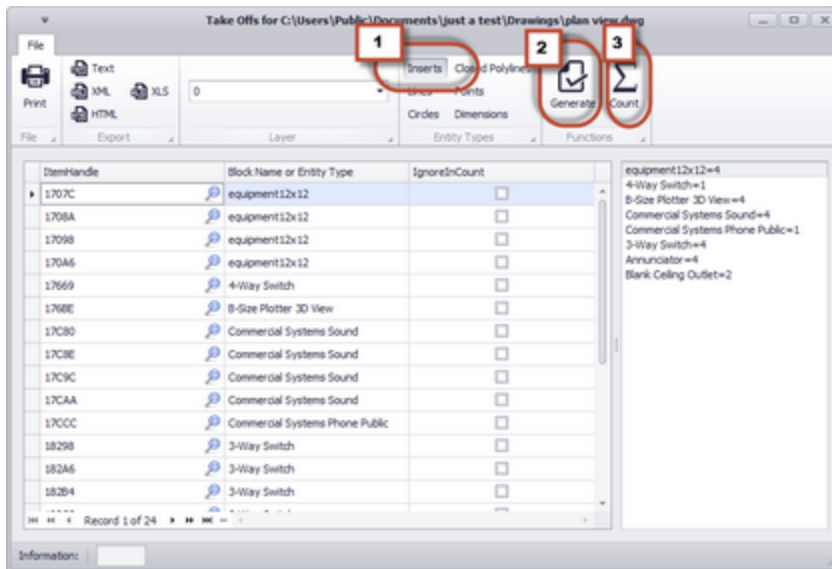
- Print** - Print the grid.
- Export** - Export the grid.
- Layer** - What layer are we searching for the selected entity type(s).
- Entity Types** - What entity type(s) are we searching for.
- Generate** - Do It! The results will be displayed in the grid.
- Count** - Count the items in the list. The results will be displayed in the right-hand list.
- Grid Columns**
 - ItemHandle ShowMe** - Zooms to show the item in the drawing.
 - Block Name or Entity Type** - obvious.
 - IgnoreInCount** - Ignore me when counting.

3.3.1.1 How To: Pull a Take-Off from a Drawing

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Plan View and Layout Tools Dialogs](#) ¹⁵⁰ > [Take Offs](#) ¹⁵¹ >

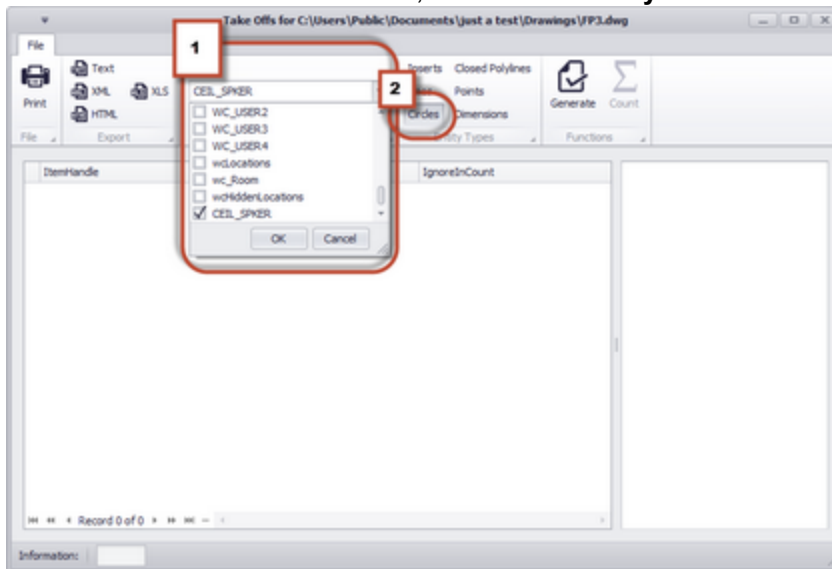


1. Open a plan view drawing.
2. Click **Plan View Tools > Take-Offs**.
3. Select the **Insert** check-button.
4. Then click **[Generate]**. This generates a list of all Inserts in the drawing.
5. Click **[Count]**. This counts the different inserts in the drawing. The results appear in the list on the right.

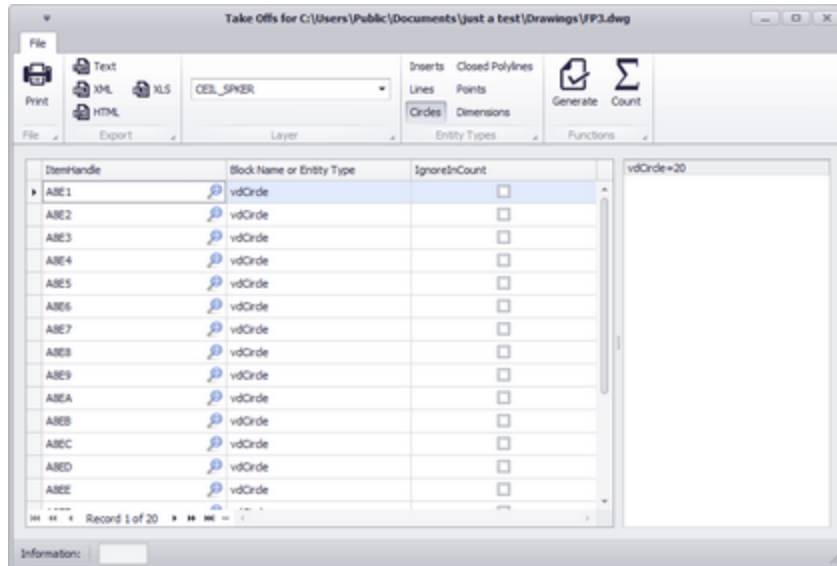


Now let's say we want to count all circles on the CEIL_SPKR layer.

1. Open a plan view drawing.
2. Click **Plan View Tools > Take-Offs**.
3. Select the **Circles** check-button, and select the **Layer** on which to search.



4. Click **[Generate]**. This produces a list of all the circle entities found on the selected layer.
5. Click **[Count]**. This counts them and outputs the result in the right-hand list.



As you can see we have 20 circles that appear on the CEIL_SPKR layer.

3.3.2 Location Boundary

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Plan View and Layout Tools Dialogs](#) ¹⁵⁰ >



Menu: **Plan View and Layout Tools > Add Boundaries from Text**

Default Commandline shortcut: **DrawLocationBoundaryRectangle**

Related Settings:

None

Used to define geometric boundaries for locations important to the design. For example we would want to know where on the drawing the Machine Room is but not necessarily the kitchen (unless part of our cable ends up in the kitchen).

Boundaries can be nested in other boundaries. It is common practice to create the Campus boundary and then nest the Building boundary then the Floor boundary then the Room and Racks boundaries.

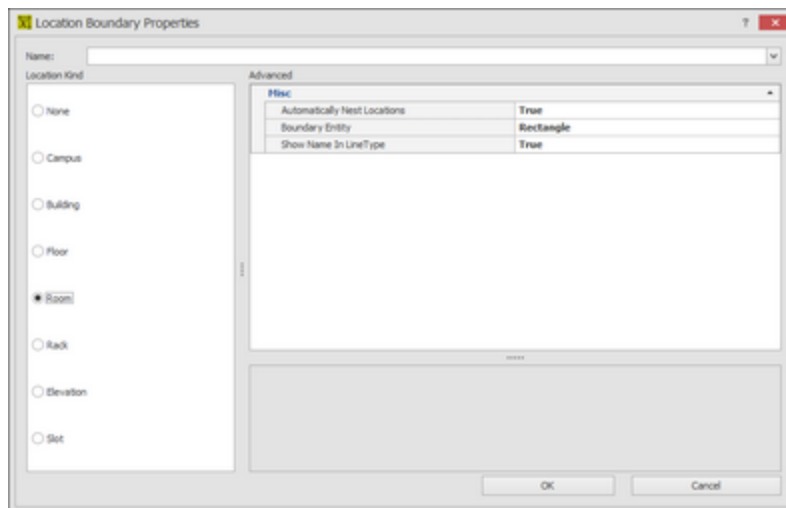


Location Boundary for the room ISO 111

Prerequisites

A plan view drawing.

Dialog Options



- **Name** - The location name.
- **Location Kind** - You will need to tell us the kind.
- **Automatically Nest Locations** - Default is true.
- **Boundary Entity** - Rectangle, Closed Polyline, Circle.
- **Show Name in Linetype** - Renders the entity with the location name as the linetype.

3.3.2.1 How To: Place a Location Boundary

Navigation: [WireCAD PRO Manual](#)  > [Plan View and Layout Tools Dialogs](#)  > [Location Boundary](#)  >



Method 1

Use the selected text entity to populate the dialog.



1. Open a plan view drawing.
2. Select a text entity that represents the location name of the location for which we will create the boundary. This text will populate the dialog **Name** field.
3. Click **Plan View and Layout Tools > Location Boundaries from Text**.
4. Set the **Location Kind**.
5. Set the **Boundary Entity**.
6. Click **[OK]**. The dialog is dismissed and you are prompted to draw the boundary entity.
7. Draw the boundary.

Method 2

Manually populate the Name field.

1. Open a plan view drawing.
2. Click **Plan View and Layout Tools > Location Boundaries from Text**.
3. Enter the **Name**.
4. Set the **Location Kind**.
5. Set the **Boundary Entity**.
6. Click **[OK]**. The dialog is dismissed and you are prompted to draw the boundary entity.
7. Draw the boundary.

3.3.3 Draw Cable

Navigation: [WireCAD PRO Manual](#)  > [Plan View and Layout Tools Dialogs](#)  >



Menu: **Drawing > PlanView and Layout Tools > Draw Cable**

Default Commandline shortcut: **pwdc**

Related Settings:

None

Product Level: **ALL**

This tool allows you to draw a cable in Plan View space that is assigned to the Cables table.

Related Topics:

[Draw Cables](#) 

Dialog Options

- **Cable Type** - Select the Cable Type.
- **Source Information** - The source **SysName** is pulled from the drawing. You will need to select the **Src Pin** and such.
- **Destination Information** - The destination **SysName** is pulled from the drawing. You will need to select the **Dest Pin** and such.
- **Other Stuff** - Enter user information.
- **Cable No** - Click the [...] button to generate a cable number.

3.3.3.1 How To: Draw a Cable in Planview

Navigation: [WireCAD PRO Manual](#) [48] > [Plan View and Layout Tools Dialogs](#) [150] > [Draw Cable](#) [156]



This tool allows you to draw geometry in the Plan View space that represents the path of the cable.

Prerequisites

1. SysNamed source and destination equipment placed in the Plan View space or:
2. Source and destination Location Boundaries placed in the Plan View space.

1. Open a plan view drawing.
2. Click **Plan View and Layout Tools>Draw Cable**
3. Select the source SysNamed equipment.
4. Select the destination SysNamed equipment.
5. Place the points in the polyline to finalize the appearance.

3.3.4 Populate Cables

Navigation: [WireCAD PRO Manual](#) [48] > [Plan View and Layout Tools Dialogs](#) [150] >



Menu: **Drawing > PlanView and Layout Tools > Populate Cables**

Default Commandline shortcut: **popc**

Related Settings:

None

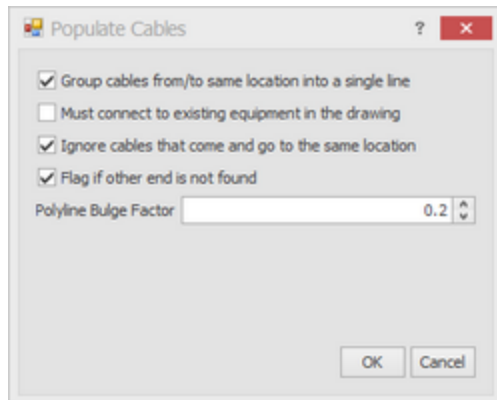
Product Level: **ALL**

This tool pulls the cables from the Cables table into the drawing. The assumption with this tool is that you will use it after you have done your functional drawings and assigned **Cable Numbers** and **SysNames**.

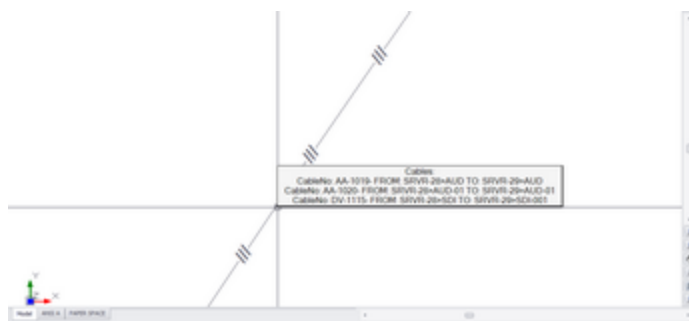
Related Topics:

[Draw Cables](#) 

Dialog Options



•**Group Cables** - Place a single polyline with hash marks to indicate count. All cables that come and go from a single Location or SysName will be grouped. Otherwise a single polyline for each cable will be added.



Grouped Cable

•**Must connect to equipment in the drawing** - The population will fail if the connected equipment SysName is not found. Otherwise the cable will be populated to the drawing based on **Location Boundary**.

•**Ignore cables that come and go to the same location** - Keeps the drawing cleaner.

•**Flag if other is not found** - If both **Locations** or **SysNames** are not found in the drawing a flag will be placed pointing to the other end.

•**Polyline Bulge Factor** - The bulge factor to create an arc instead of a straight line.

3.3.4.1 How To: Populate Cables in Planview

Navigation: [WireCAD PRO Manual](#) [48] > [Plan View and Layout Tools Dialogs](#) [150] > [Populate Cables](#) [157] >

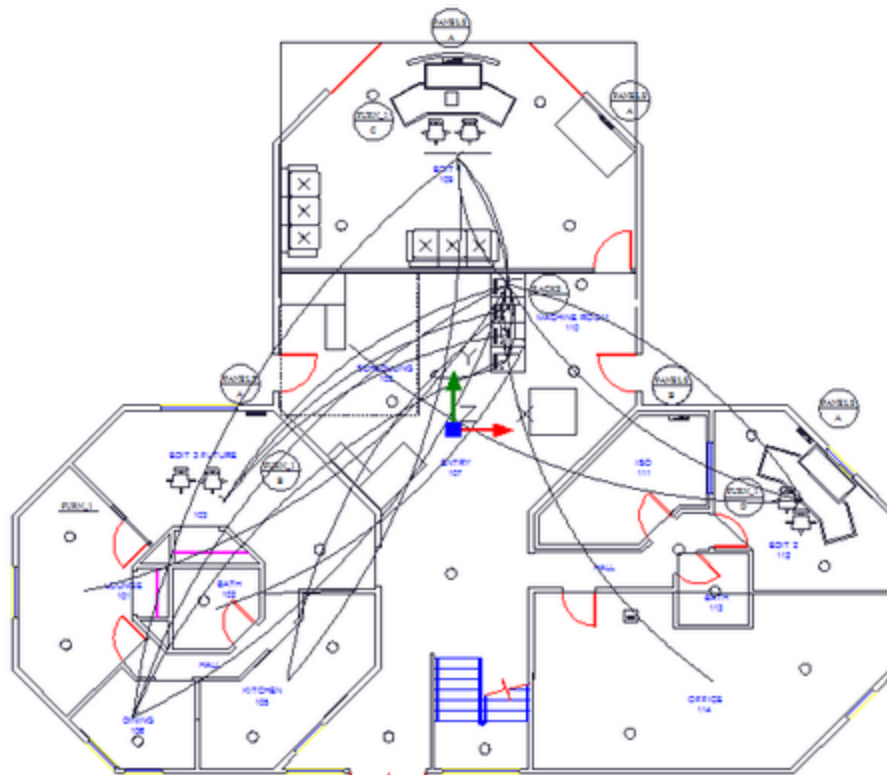


Prerequisites

1. **SysNamed** source and destination equipment placed in the Plan View space **or**:
2. Source and destination **Location Boundaries** placed in the Plan View space.
3. **Cable** data in the Cables table that matches the placed equipment or locations.

Operation

1. Open a plan view drawing.
2. Place **Location Boundaries**.
3. Place plan view equipment and **SysName** - Optional.
4. Click **Plan View and Layout Tools > Populate Cables**.
5. Click **[OK]**.



Planview Populate Cables

3.3.5 Draw Prewire

Navigation: [WireCAD PRO Manual](#) [48] > [Plan View and Layout Tools Dialogs](#) [150] >



Menu: **PlanView and Layout Tools > Draw Prewire Cable**

Default Commandline shortcut: **pw**

Related Project Settings:

Automatically Consume Prewire Cables

Product Level: **ALL**

This tool allows you to draw geometry in the drawing that represents cables that have not yet had their functions assigned. The only thing we know about these cables is the locations from which they start and end and the Cable Type. A record will be added to the Cables table based on the count defined. The record is marked Available and PreWIRE. Prewire cables can be automatically consumed later if the [Automatically Consume Prewire](#) ^[365] setting is checked.

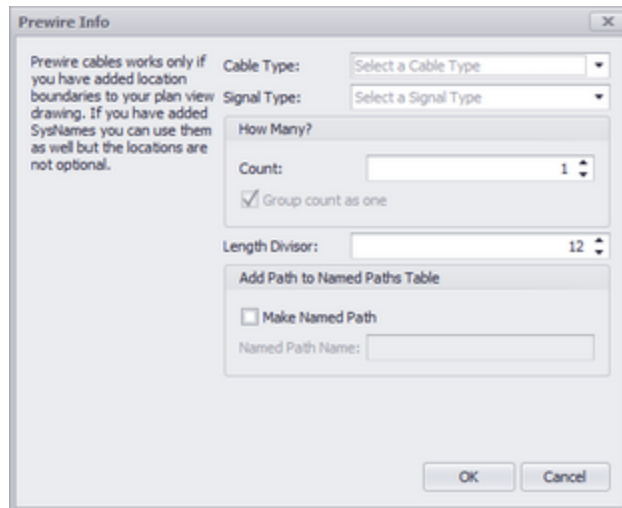
Prerequisites

1. Source and destination Location Boundaries placed in the Plan View space.

Related Topics:

[Draw Cables](#) ^[82]

Dialog Options



•Cable Type, Signal Type

•**How Many** - How many wires of this type take this path. A record will be created for each.

•**Length Divisor** - We will divide the length of the polyline by this value and place that info in the **Length** field of the record. For example, say your Drawing Unit = 1 inch and you want the length in the **Length** field to be displayed in feet. Your **Length Divisor** would be 12.

•**Create Named Path** - Create an entry in the **Named Paths** table with the name provided and length of the polyline.

3.3.5.1 How To: Draw a Prewire Cable in Planview

Navigation: [WireCAD PRO Manual](#) ^[48] > [Plan View and Layout Tools Dialogs](#) ^[150] > [Draw Prewire](#) ^[159] >



Operation

1. Open a plan view drawing.
2. Click **PlanView and Layout Tools > Draw Prewire Cable**.

3. Fill in the form.
4. Click **[OK]**. This will dismiss the dialog and prompt you to draw a polyline.
5. Draw the cable path.
6. **Cables** will be added to the project **Cables** table with a formatted number based on the project **Cable Number Format** and the **Next Numbers** table.
7. The polyline length will be divided by the **Length Divisor** field and added to the cable record **Length** field.

3.3.6 Populate Equipment

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Plan View and Layout Tools Dialogs](#) ¹⁵⁰ >



Menu: **PlanView and Layout Tools > Populate Equipment**

Default Commandline shortcut: **pope**

Related Project Settings:

None

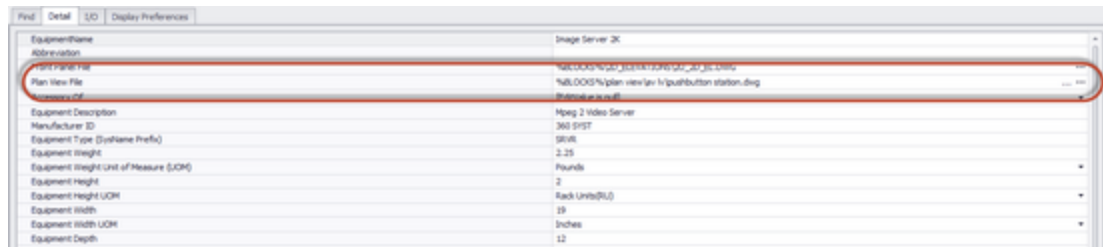
Product Level: **ALL**

There is no UI for this tool.

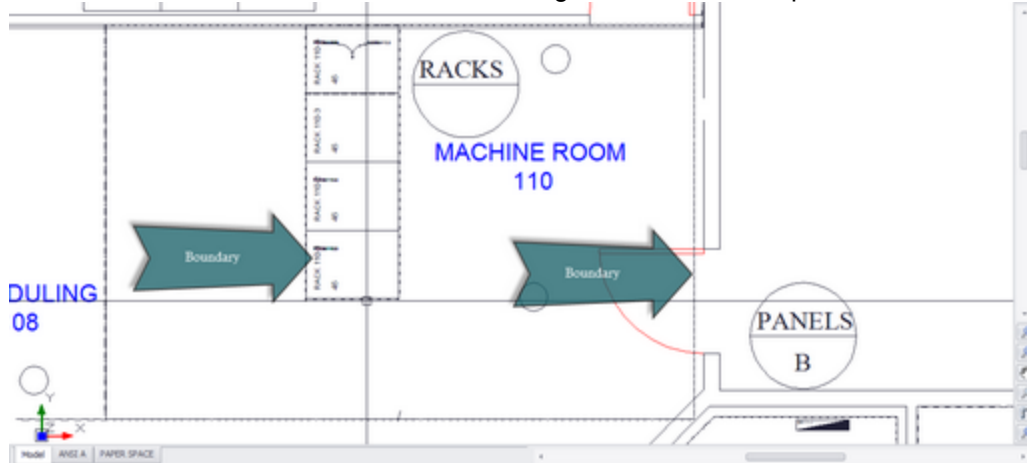
This tool pulls the Plan View version of the equipment from the Equipment List.

The assumption with this tool is that you will use it after you have done your functional drawings and assigned cable numbers and SysNames.

This tool relies on the Plan View File in the device definition to know what to place.

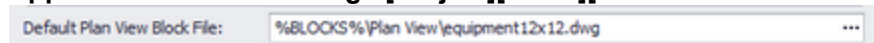


It relies on Location Boundaries in the drawing to know where to place it.



If the Plan View File is blank the Project Default Plan View File will be placed if the Location Boundary is found:

Application Menu > Settings [Project][Basic][Default Plan View Block File]



3.3.6.1 How To: Populate Equipment in Planview

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Plan View and Layout Tools Dialogs](#) ¹⁵⁰ > [Populate Equipment](#) ¹⁶¹ >



Prerequisites

1. SysNamed equipment preferably with an associated **Plan View File** in the global **Equipment Library**.
2. **Location Boundaries** placed in the drawing.

Operation

1. Open a plan view drawing.
2. Place Location Boundaries.
3. Click **PlanView and Layout Tools > Populate Equipment**.
4. A message box will explain the operation of the function. Click **[OK]**. The tool will run.
5. At completion the tool will report the number of placed items.

3.4 Reporting

Navigation: [WireCAD PRO Manual](#)  >



Menu: double-click the report in the **Project Explorer** or Click the report in the **Reports > Reports gallery**

Default command line shortcut: **none**

Related User Settings:

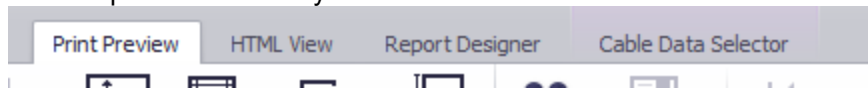
Report Show Mode - Determines where the reports will be shown:

Project Explorer
Ribbon Menu Gallery
Both

Opens a report for preview, printing or export. Reports are designed to connect to the WireCAD DataAccess object that provides access to the entire data model both Global and Project data.

There are four tabs presented when a report is loaded:

1. **Preview** - Shows the print preview that will be output to the selected system printer.
2. **HTML** - Shows the report rendered to HTML for use in web pages.
3. **Design** - Shows the report designer.
4. **Cable Data Selector** - Shows a selector grid for use with the ProjectCables collection. This grid is not implemented for any other data collection.



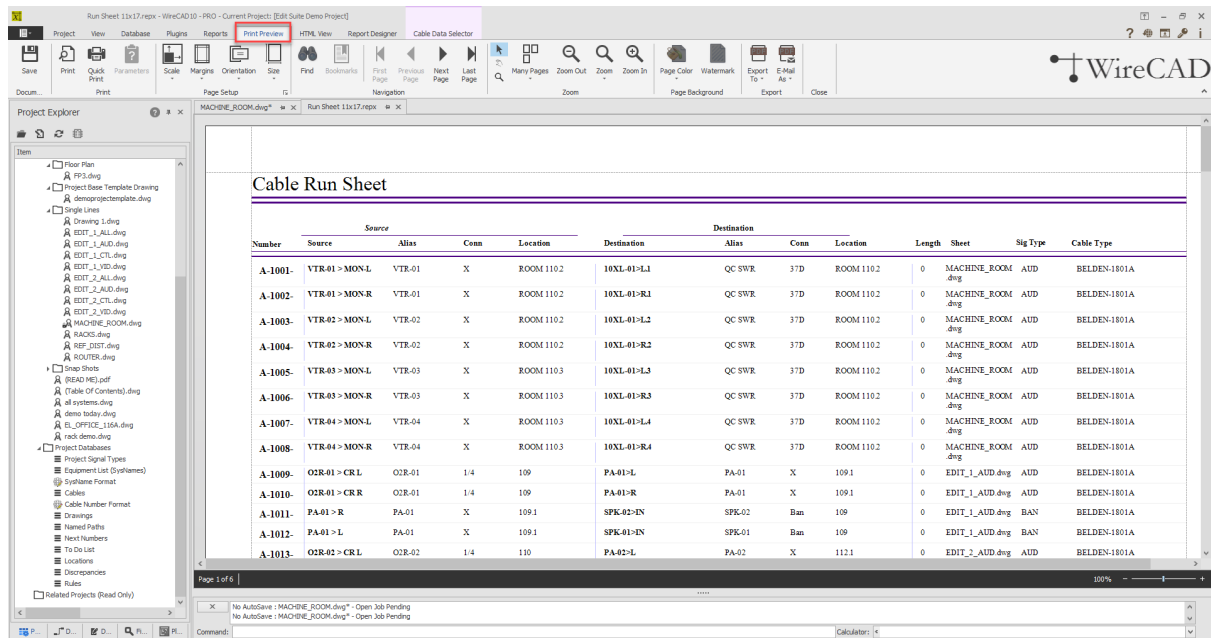
3.4.1 How To: Print Reports

Navigation: [WireCAD PRO Manual](#)  > [Reporting](#)  >



1. Double-click the report in the Project Explorer or Click the report in the **Reports > Reports gallery**.

2. The report will load in **Preview Mode**.



3. Click **File > Print** and select a printer.

3.4.2 How To: Print Reports with Parameters

Navigation: [WireCAD PRO Manual](#) 48 > [Reporting](#) 163 >



Parameters allow the report designer a way to collect input from the user before the report is rendered.

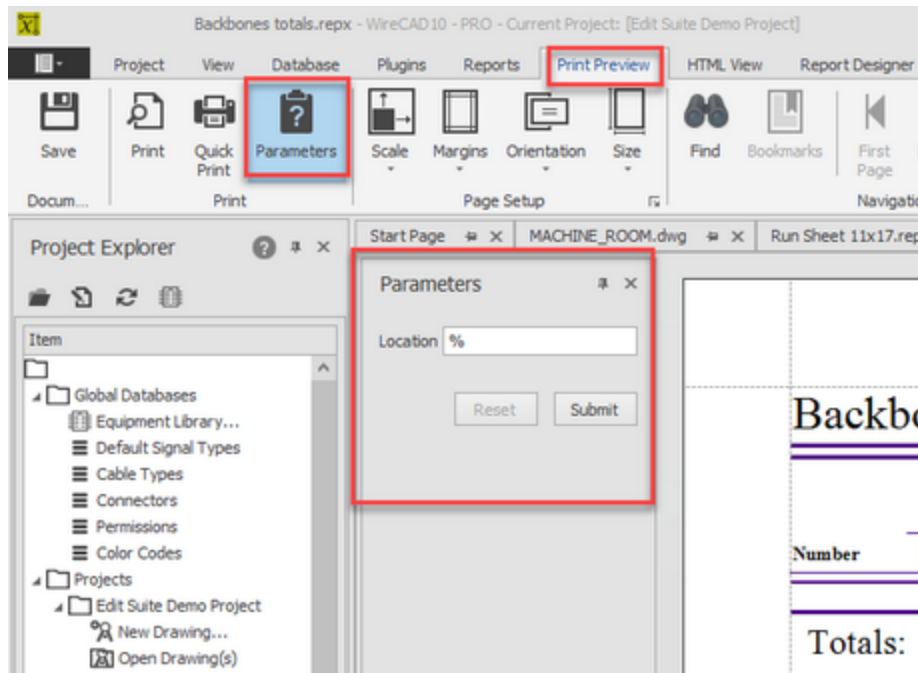
Parameters may be used in the report for several purposes:

- Replace title and descriptive text.
- Provide data to filters.
- Provide data to scripts.

The report designer may force the user to submit parameter data or allow the report to render with default data.

If the report has no parameters defined the **Parameters** menu item will be disabled.

1. Open a report - Double-click the report in the **Project Explorer** or Click the report in the **Reports > Reports** gallery.
2. The report will load in **Preview Mode**.
3. If the report has parameters the Parameters menu item will be enabled. If the report designer determined the report must gather parameters first, the report will not render until you click the **[Submit]** button on the Parameters tool panel.



4. Enter parameter data.
5. Click **[Submit]**. The report will render with the submitted data.
6. Click **File > Print** and select a printer.

3.4.3 How To: Filter Reports

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Reporting](#) ¹⁶³ >

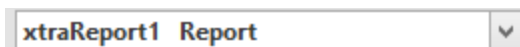


By default a report will query the database for all records in the selected collection. For example if the report displays the project CablesCollection, all records will be displayed. This may or may not be desirable. If you wish to filter the record set to show a specific set of records you have two choices:

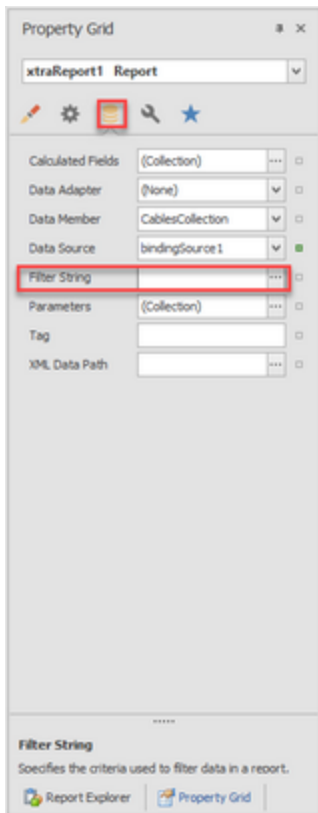
1. If the collection being displayed is the CablesCollection you may use the **Cable Data Selector** tab and make a manual selection there.
2. Otherwise you must create a filter.

How To:

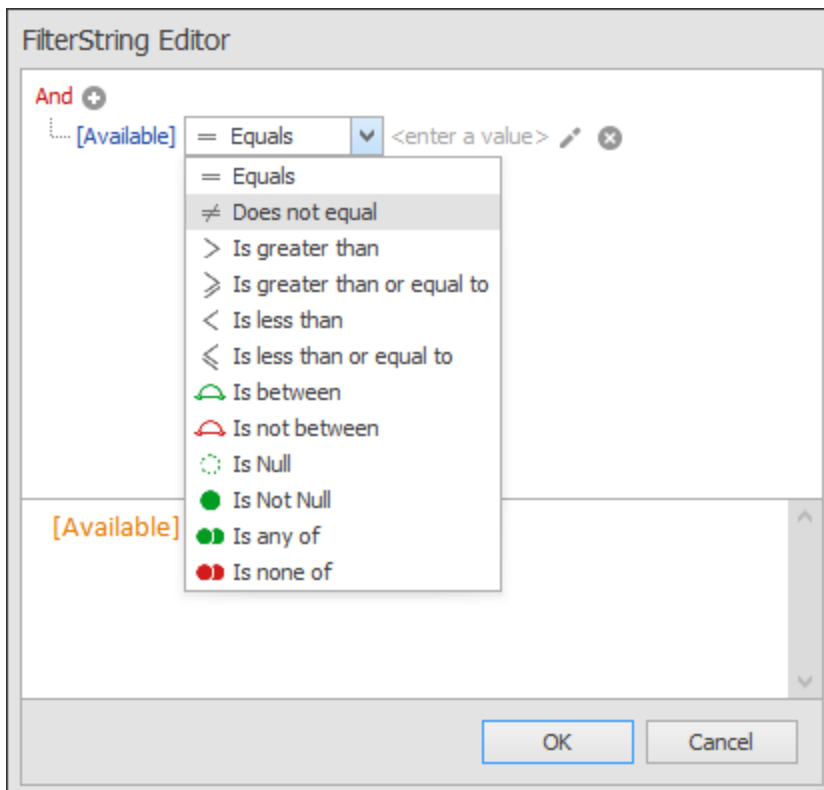
1. Open a report.
2. Switch to the Report Designer tab.
3. Locate the Property window. Make sure that the Property window is displayed. If not click: **Report Designer>Windows>Property Grid**
4. From the Property grid object selector, select **XRReport1**



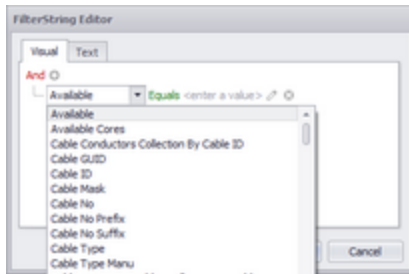
5. From the Property grid select the data tab and then the **Filter String [...]** ellipsis button.



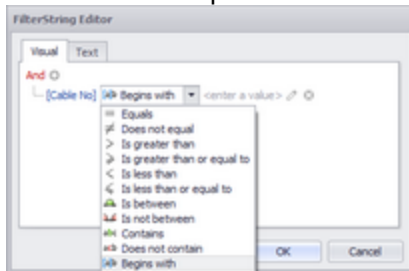
6. This opens the filter dialog from which you may select a number of different filters.



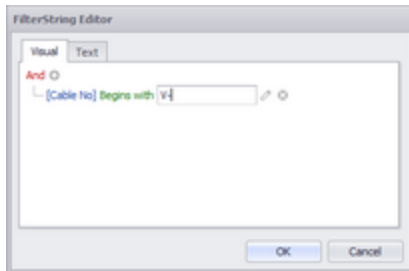
7. Select a field to filter on.



8. Select a filter operator.



9. Enter a value.



10. Click **[OK]** to commit the filter. This filter will show only cables where the CableNo field begins with V-.

11. Select the **Print Preview** tab to render the report with the filter.

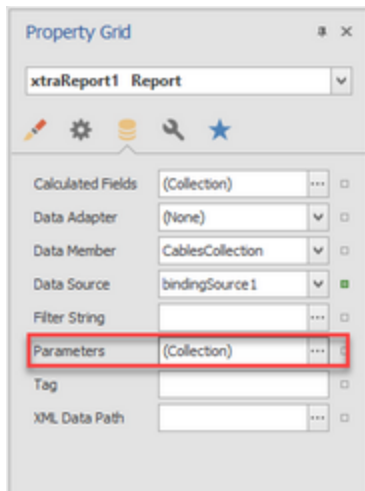
3.4.4 How To: Filter a Report Using Parameters

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Reporting](#) ¹⁶³ >

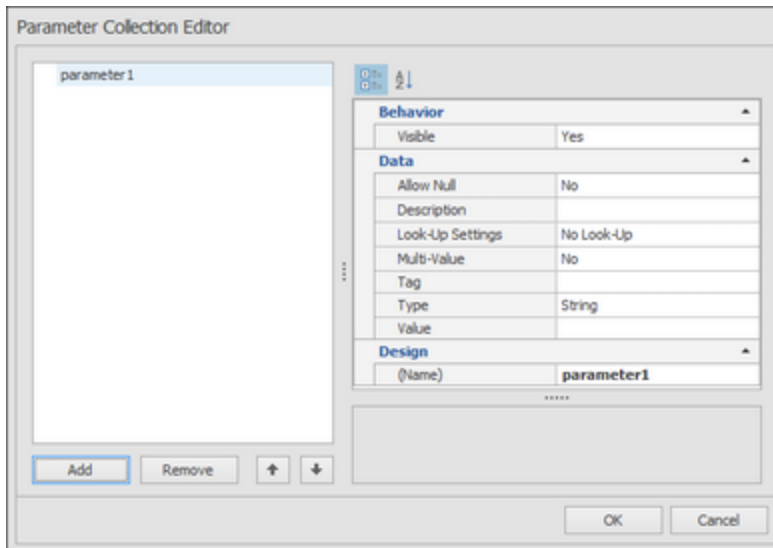


Filters may also contain parameters that the user must enter before the report is generated. In order to make use of this function you must follow these steps:

1. Add a parameter to the report while in Design view. Click the **Parameters (...)** ellipsis button to view the Parameters collection.



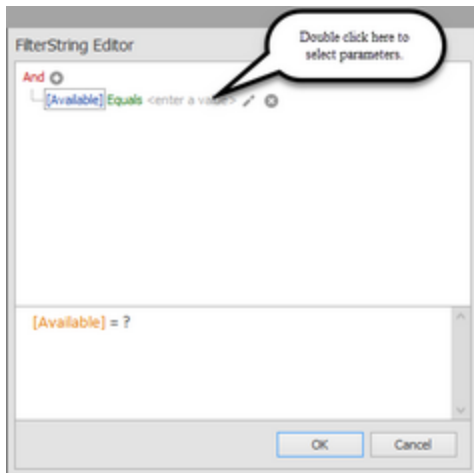
2. Click the **[Add]** button to add a parameter to the collection.



3. Enter description and default value information.

4. Open the filter editor (see above)

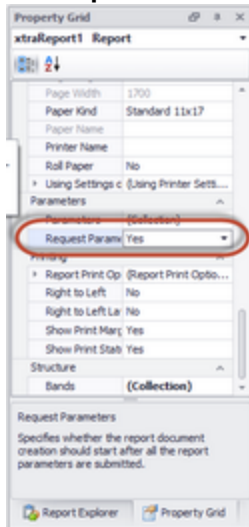
5. Add a Field and Operator, then click on the icon to the right of the field to select parameters.



6. Select your parameter from the parameters list. Above we are selecting parameter1.

7. Click **[OK]** to submit the filter.

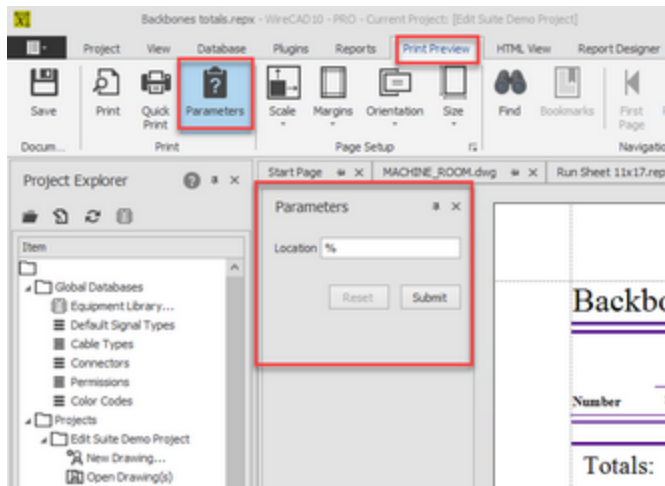
8. Now let's make sure that the parameters are submitted before the report renders. Set the **Request Parameters** property = Yes.



9. Click the **[Print Preview]** tab to preview the report. You will see a **Parameters** pane on the left hand side with a submit button.

10. Enter a value and click **[Submit]**.

11. Select the print preview tab to render the report with the filter.



12. Your report will be rendered with **parameter1** applied to the filter.

3.4.5 Report Design

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Reporting](#) ¹⁶³ >



Menu: **Report>New Report with Wizard**

Default command line shortcut: **rw**

Related User Settings:

Report Show Mode - Determines where the reports will be shown:

Project Explorer
Ribbon Menu Gallery
Both

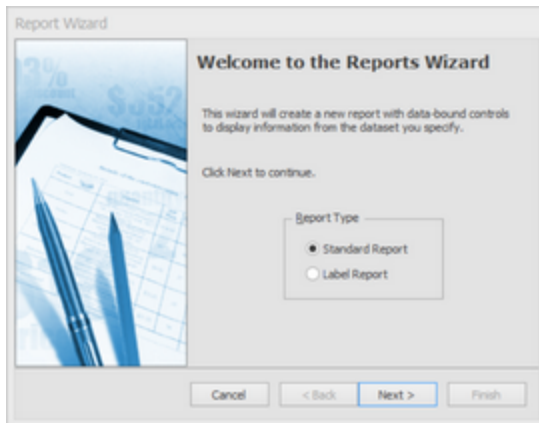
3.4.5.1 How To: Create a Standard Report

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Reporting](#) ¹⁶³ > [Report Design](#) ¹⁷⁰ >



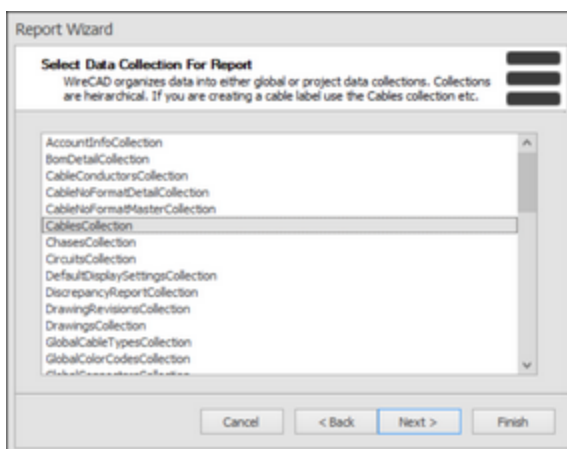
1. Click **Reports>New Report with Wizard**.

2. Select the **Standard Report** option.



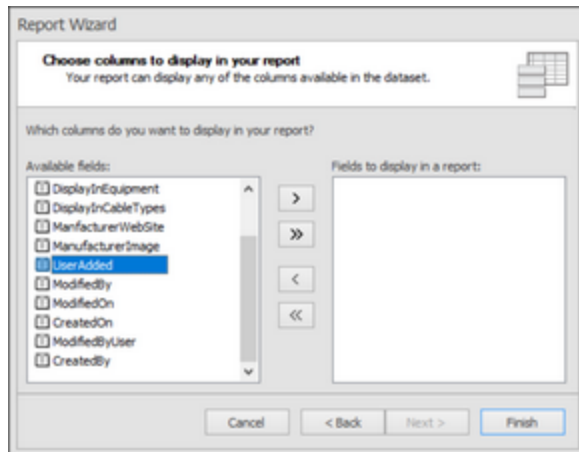
3. Click **[Next >]**.

4. Select a data collection. Data collections access both the global and project databases. For example: say you wanted to show a list of all manufacturers. You would select the ***GlobalManufacturersCollection***.

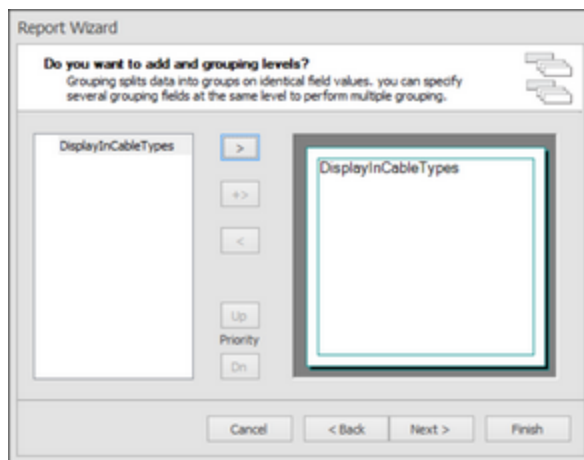


Note: collections are hierarchical to aid in the creation of sub-reports. All collections are populated with data when the report is previewed.

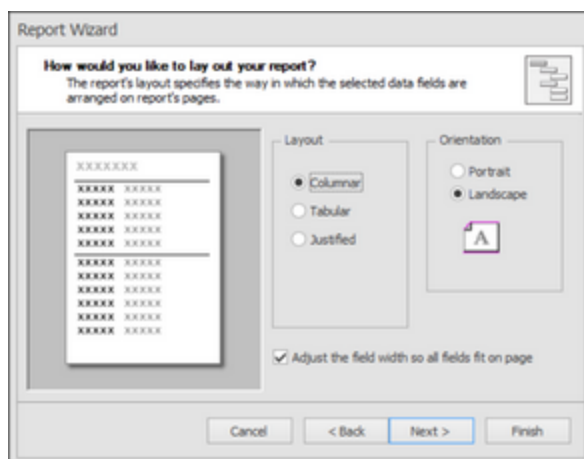
5. Select the fields (**columns**) you wish to display in the report.



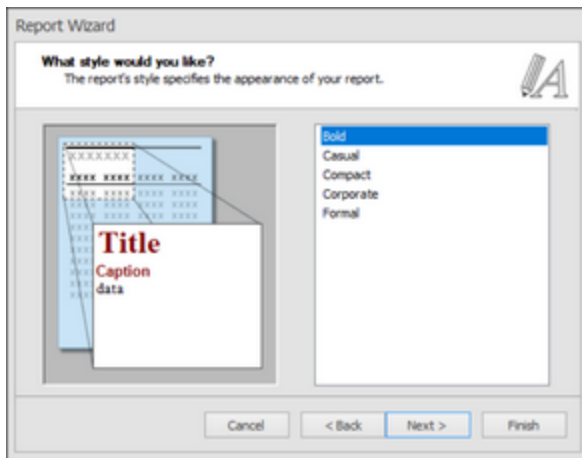
6. Apply any grouping. Each set of records with the same value will become a "group" with its own header.



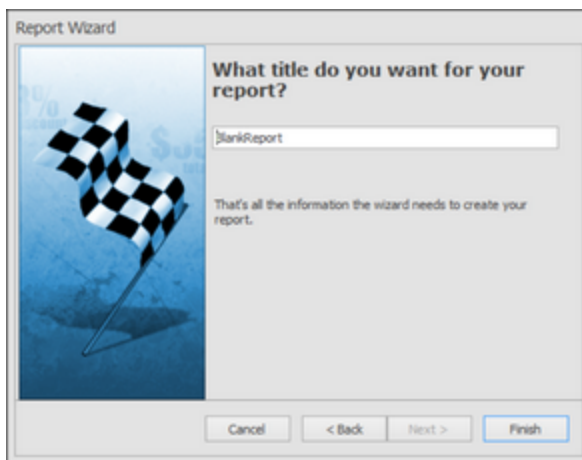
7. Select a **Layout** and **Orientation** for your report. The **Adjust field width to fit** function will force all selected fields on to a single page possibly rendering some of the data unreadable. If you have lots of fields to display, consider using a **Justified** report.



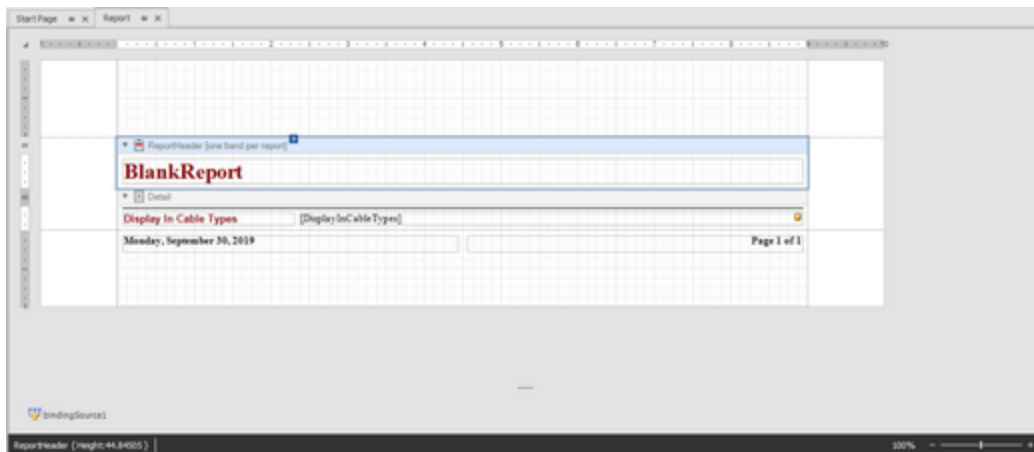
8. Select a **Report Style**. There are several predefined report styles. You can always modify the look and feel later.



9. Title your report.



10. Click **[Finish]** to create your shiny new report.



11. Click **File>Save** and save your work to your reports support path. It can be saved anywhere in the **%REPORTS%** support path folder tree.

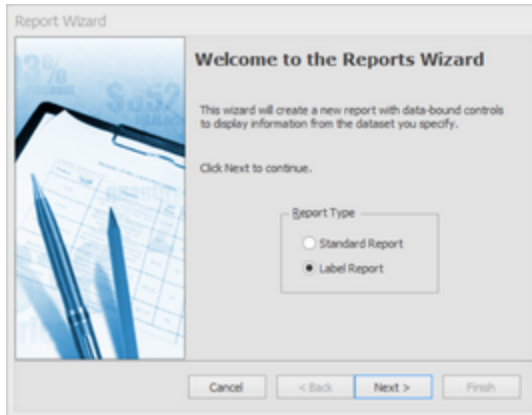
3.4.5.2 How To: Create a Label Report

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Reporting](#) ¹⁶³ > [Report Design](#) ¹⁷⁰ >



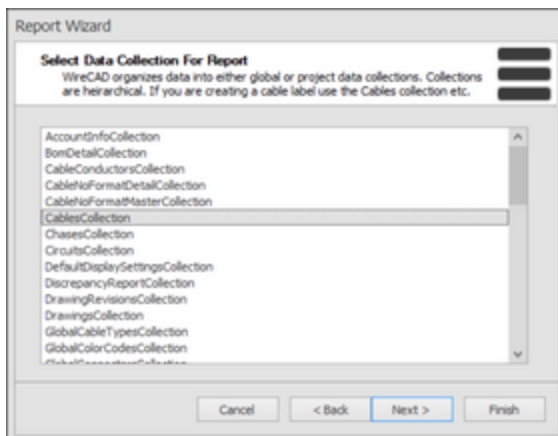
1. Click **Reports>New Report with Wizard**.

2. Select the **Label Report** option.

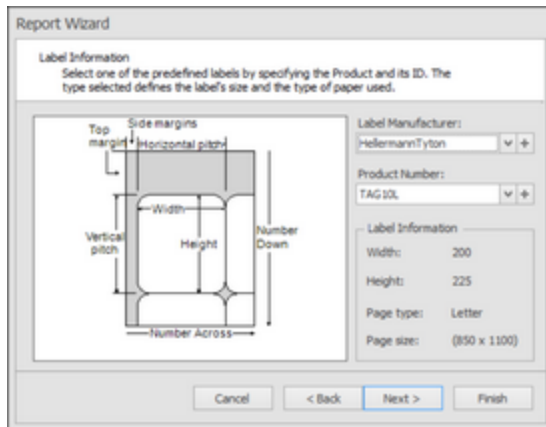


3. Click **[Next >]**.

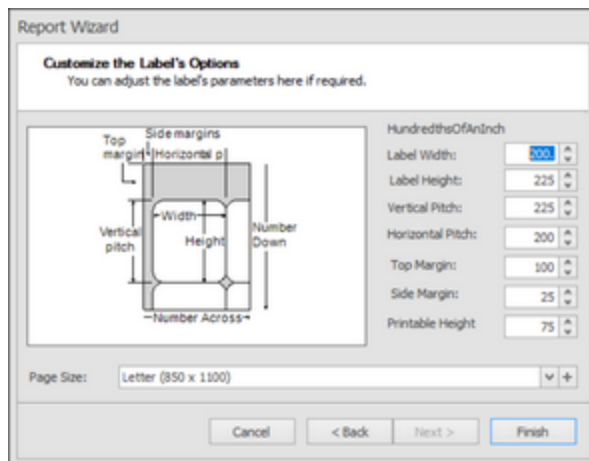
4. Select a data collection. Data collections access both the global and project databases. Note: In the case of labels we probably want to use the **CablesCollection**.



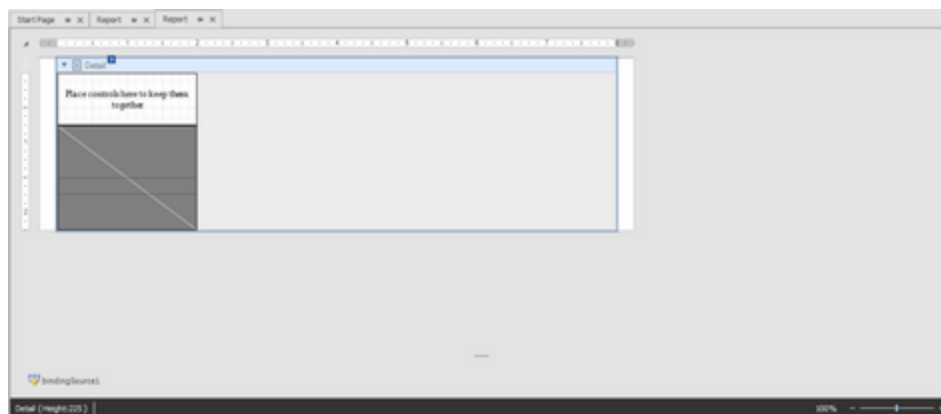
5. Select a label format. WireCAD comes stock with over 1000 label formats from 15 manufacturers including Panduit, Hellerman Tyton and Brady.



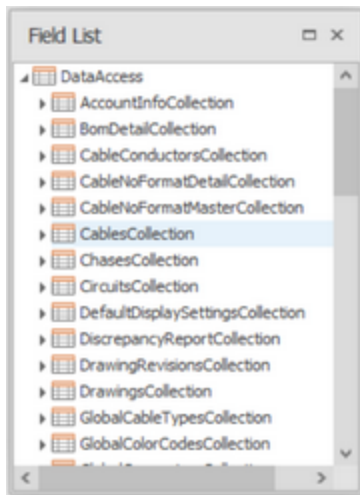
6. Modify height and width of each label and its printable area and margins.



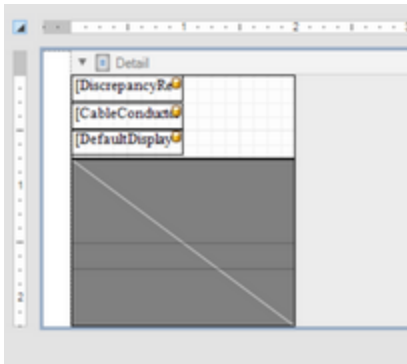
7. Click **[Finish]** to create your shiny new report.



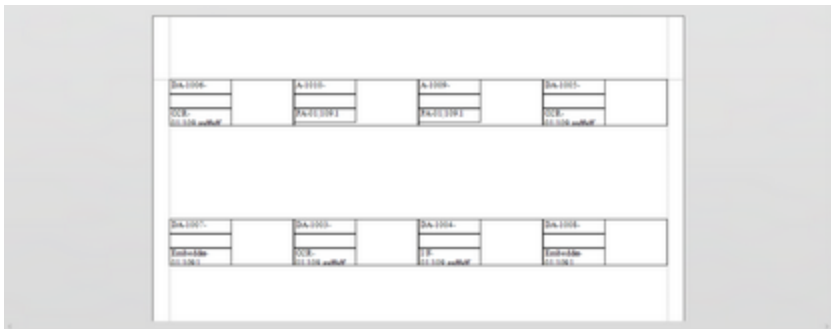
8. Select the **Field List** and drag any fields that you wish to display on to the report design surface. Avoid placing field on the light and dark gray areas. The light gray area will not print and the dark gray is indicative of the clear laminating portion of a cable label.



9. Here we have three fields that we have dragged and positioned on the design surface.



10. Final output looks like this:



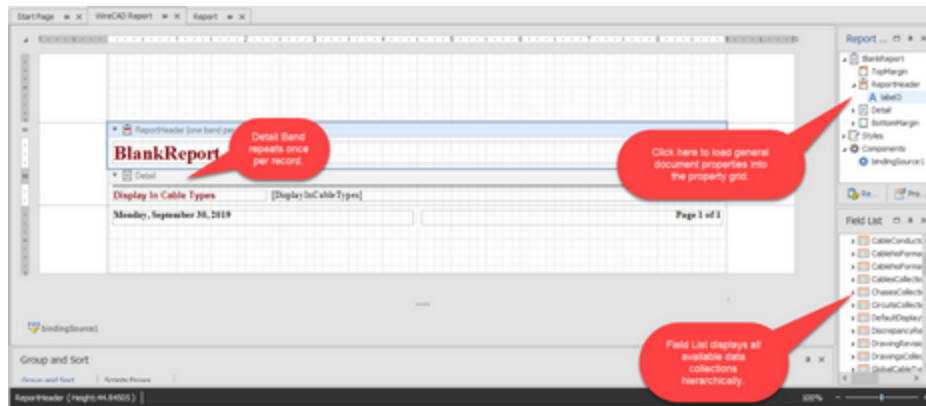
11. Click **File>Save** and save your work to your reports support path. It can be saved anywhere in the **%REPORTS%** support path folder tree.

3.4.5.3 Report Design Basics

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Reporting](#) ¹⁶³ > [Report Design](#) ¹⁷⁰ >

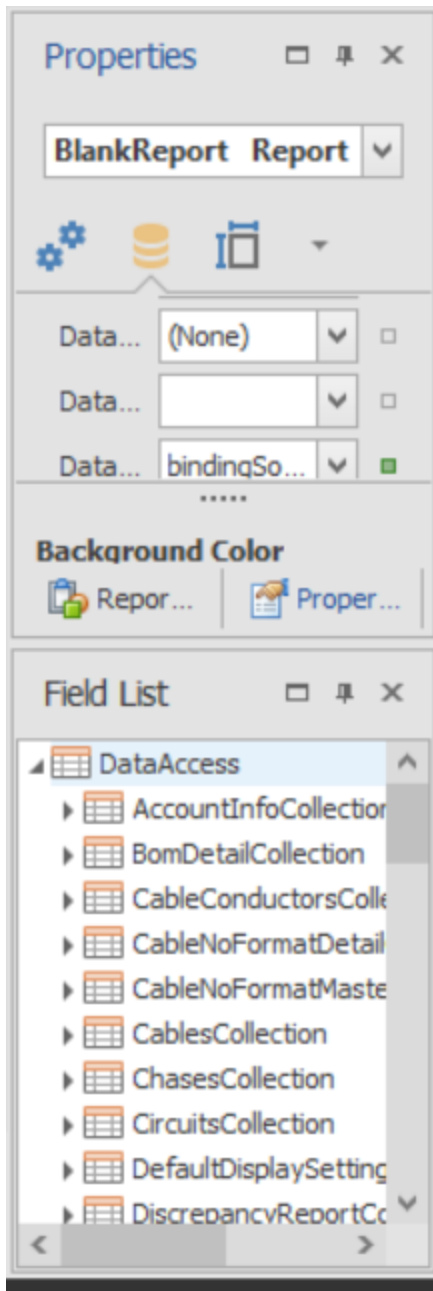


The following topic assumes that you have either opened a new blank report or that you have run the **New Report Wizard**.



If you open a new report without running the wizard you will need to set the report **Data Member** variable.

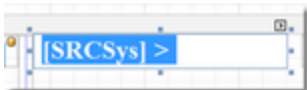
Click in the dark gray area to load the general properties. Select the **Data Member** from the drop down.



You can drag fields from the **Field List** directly to the report designer.

If you want to concatenate multiple fields in a single label you can drag multiple fields onto the same label or edit it directly.

To edit a label double-click it to enter edit mode.



Type directly into the label. Field names must be enclosed in [] brackets.

3.5 Frequently Asked Questions

Navigation: [WireCAD PRO Manual](#) 48 >



3.5.1 How To: Place Custom Titleblocks (Page Borders)

Navigation: [WireCAD PRO Manual](#) 48 > [Frequently Asked Questions](#) 179 >



Menu: **Basic CAD Tools>Blocks>Insert Block Into Drawing**

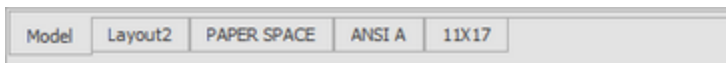
Default command line shortcut: **insert**

Related Settings:

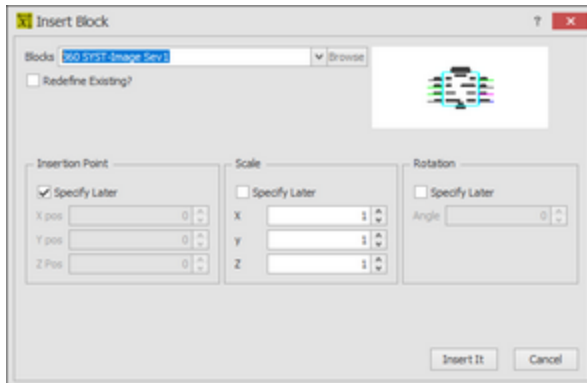
None

This tutorial assumes you have a custom titleblock in dwg format and that it occupies the Model Space of the dwg file. Further that you know the path to the file containing the titleblock file.

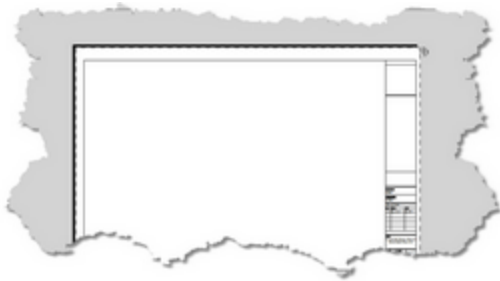
1. Open the drawing where you want to place a custom titleblock.
2. Switch to the Layout in which you intend to place your custom page border.



3. Click **CAD Tools>Blocks and Inserts>Insert Block into Drawing**.



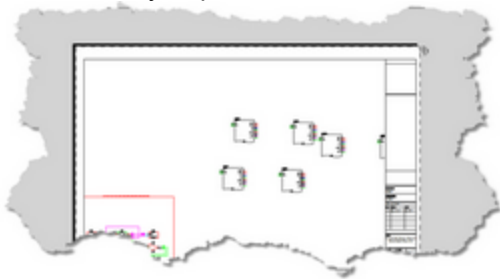
4. Select **[Browse]**.
5. Browse to the dwg that contains your custom title block in the Model Space.
6. Check **Insertion Point - Specify Later**. This will let us manually place the inserted block in the layout space.
7. Uncheck **Scale - Specify Later** and set the X,Y,Z scale to 1.
8. Uncheck **Rotation - Specify Later** and set the **Angle** to 0.
9. Click **[Insert It]**.



10. The dialog will be dismissed so you can drop the insert in the drawing space.

11. Now you will need to place a **Viewport** so you can see the model space.

12. Click **View>New Viewport** and select the logical boundaries of the viewport within the titleblock you placed earlier.



13. Double-click the viewport to activate it for zooming and panning and use the mouse wheel or zoom commands to position the viewport over the model space.

3.5.2 How To: Create Custom Titleblocks

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Frequently Asked Questions](#) ¹⁷⁹ >



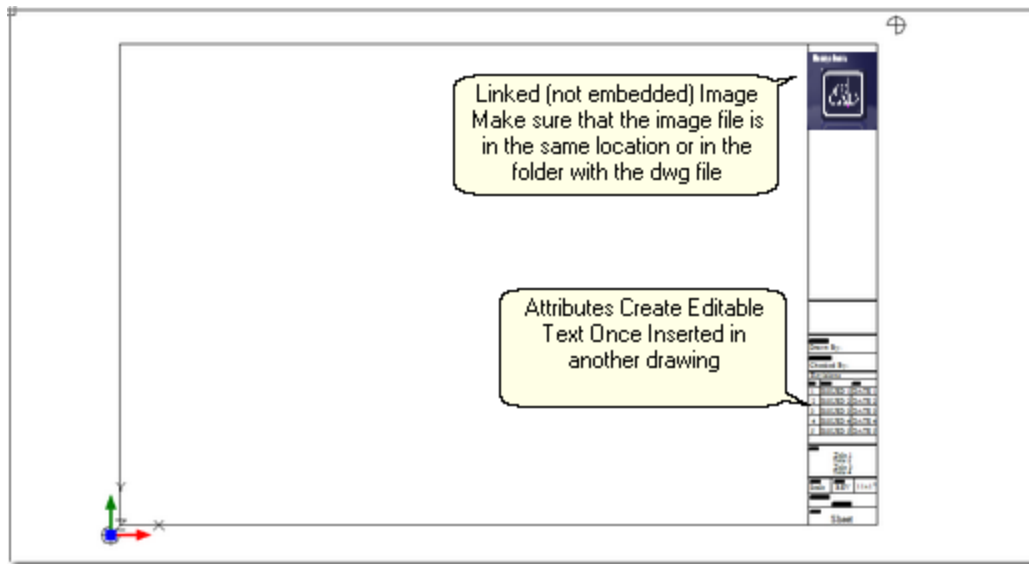
Menu: **CAD Tools>Blocks>Insert Block Into Drawing**

Default command line shortcut: **insert**

Related Settings:

None

1. Create a new drawing with no template.
2. Draw your titleblock in Model space scaling it 1:1 with your printed page size, ie an 11x17 page border would be 11x17 minus your margins.
3. Place any images and attribute definitions.
4. Save the drawing.
5. Follow the steps [here](#) ¹⁷⁹ for placing your title block drawing in any other drawing.



3.5.3 Moving Projects (Pack Up/Check-Out)

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Frequently Asked Questions](#) ¹⁷⁹ >



Menu: **Application Menu > Project Utilities>Packup / Check-out**

Menu: **Application Menu > Project Utilities>Unpack**

Menu: **Application Menu > Project Utilities>Check In**

Default command line shortcut: **packup**

Default command line shortcut: **checkinproject**

When moving projects from machine to machine there are external items upon which the project depends. These are referred to as project and drawing dependencies. The main project dependency is the global equipment database. Drawings may have image, Xref, and font dependencies. When we **Pack Up** a project we are grabbing all of those dependencies (fonts are an exception and are not included) and placing them in the Project folder. We may choose to **Check Out** the project at the same time. This flags the project as read only until such time as the project is **Checked In**.

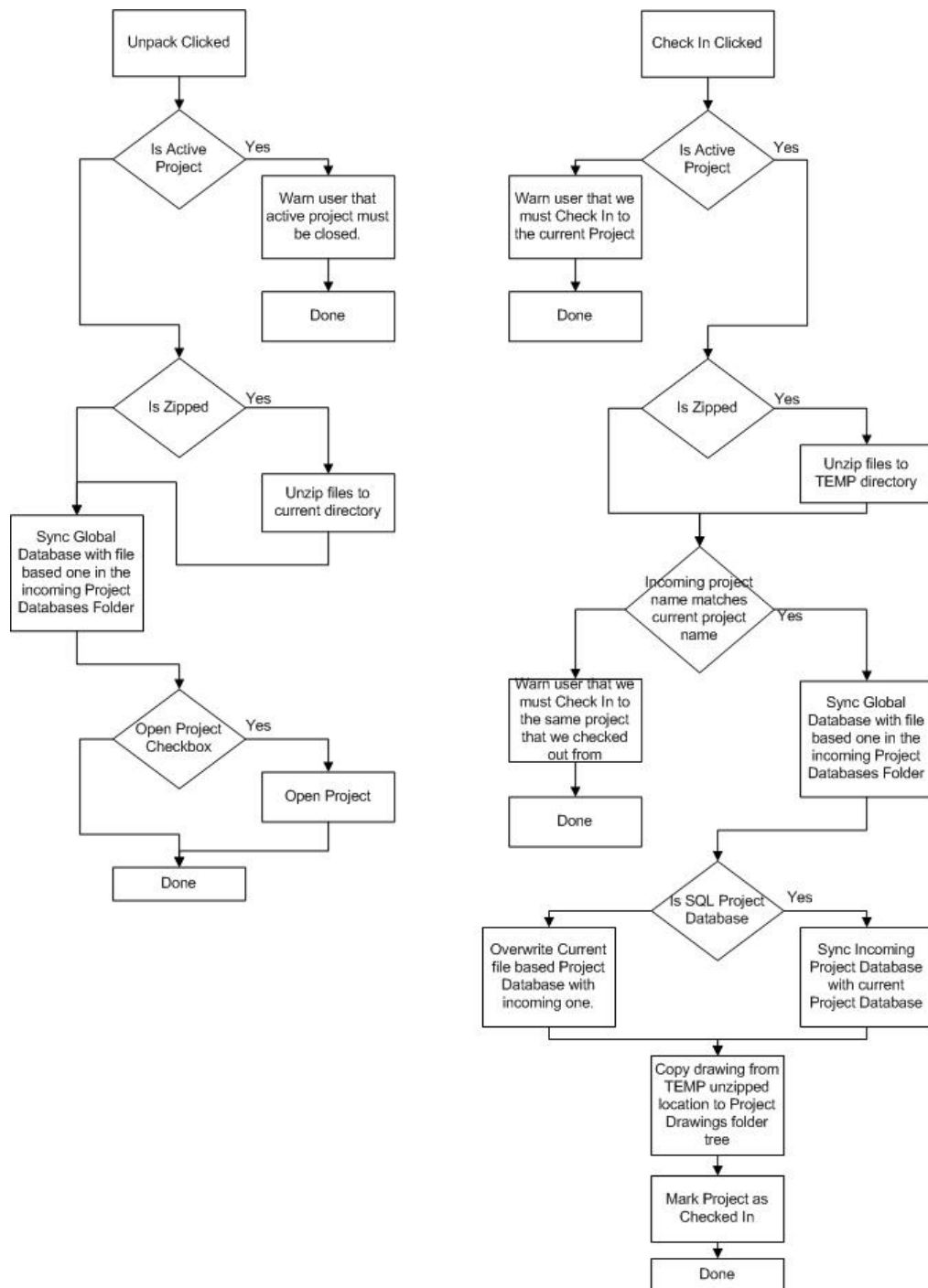
Once **Packed Up** and/or zipped up you are ready to move the project to another machine. Simply copy the Project folder or the zipped file and move it to the new machine. Then use the **Unpack** function.



To Unpack or Check In - That is the Question

The two functions are practically identical with the exception that the **Check In** function won't launch without an active project and once unpacked will mark the active project as **Checked In**. Whether you **Unpack** or **Check In** a project depends on whether you are moving a project to a machine on which that project already resides and whether you have **Checked Out** the Project.

If you have not **Checked Out** the project then there is no need to **Check In** the project.
If you are moving a project to another machine, use the **Unpack** function.



3.5.3.1 How To: Packup/Check-out

Navigation: [WireCAD PRO Manual](#)  > [Frequently Asked Questions](#)  > [Moving Projects \(Pack Up/Check-Out\)](#)  >



Decide whether you are going to **Check Out** the project before starting.

1. Open the project to Pack Up/Check Out.
2. Click **Application Menu > Check In/Out > Pack Up/Check Out**.
3. Click **[Next]** after reading the explanatory text.
4. If you opt to **Check Out** the project you will be prompted to enter an **Expected Return Date**. This will let anyone opening the project subsequent to its **Check Out** who checked it out and approximately when it will be back. In the mean time it will be opened READ ONLY.
5. If you opt to **Compress Output** the final operation will be to create a zipped file of the project folder and place it in the **FileName and Path** location.
6. Click **[Next]**. You will be presented with the progress page.
7. Click **[Finish]**.

3.5.3.2 How To: Check In a Project

Navigation: [WireCAD PRO Manual](#)  > [Frequently Asked Questions](#)  > [Moving Projects \(Pack Up/Check-Out\)](#)  >



To **Check In** a project you must first have the project that we are checking into open. The incoming project should be in a file location that is not the current project path.

1. Open the project into which we will check in the incoming project.
2. Click **Application Menu > Check In/Out > Check In Project**.
3. Click **[Next]** after reading the explanatory text.
4. Browse to the incoming project's .wpi file by clicking the [...] ellipsis button.
5. Click **[Next]**. This presents the progress page.
6. Click **[Finish]**. Wait for the process to finish.

3.5.3.3 How To: Unpack a Project



Navigation: [WireCAD PRO Manual](#)  > [Frequently Asked Questions](#)  > [Moving Projects \(Pack Up/Check-Out\)](#)  >



To **Unpack** a project you must not have an open project. The incoming project should be in a file location WILL BECOME the current project path.

1. Close the current project if any.
2. Click **Application Menu > Check In/Out > Unpack Project ...**
3. Click **[Next]** after reading the explanatory text.
4. Browse to the incoming project's .wpi file by clicking the [...] ellipsis button.
5. Click **[Next]**. This presents the progress page.
6. Click **[Finish]**. Wait for the process to finish.

3.5.4 Synchronizing with Another Equipment Database

Navigation: [WireCAD PRO Manual](#)  > [Frequently Asked Questions](#)  >



Menu: **Database>Synchronize Global Database ...**

Default command line shortcut: **sync**

Related Settings: None

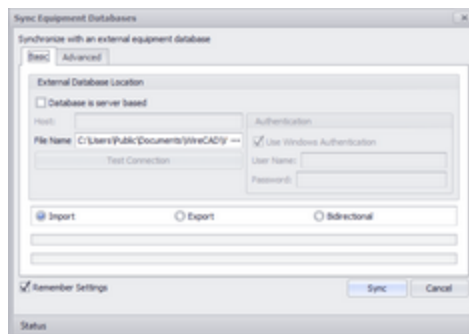
Occasionally you may experience the need to sync with another user's global database. This will copy all of their equipment to your Global Equipment Database. The sync includes manufacturers, equipment, inputs, outputs, signal types, connectors, cable types, cable core data, relational tables, etc.

You may choose within the tool to perform an **import**, **export** or **bidirectional** sync.

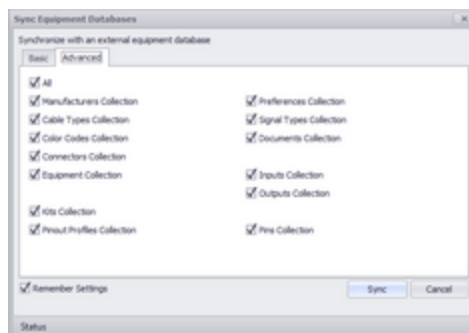
Demystifying Synchronization

Synchronizing data in two tables of the same structure is really very simple. In its most basic form, records that do not exist in one table are added. Records that exist in both tables receive the most current data based on a time stamp. In order that records deleted from one table do not get added back in, a special table is employed to track deleted keys. If the delete is the most current action then the record will likewise be deleted from the other table. In the unlikely event that the records have the exact same time stamp, yet the data is different, those records are flagged as conflict records from which you must pick the most correct.

Controls



Basic tab



Advanced tab

•**Database Location** - Select either a VistaDB file based database or a SQL Server host. If server based you will need to provide host and credentials. If you do not know them contact your SQL Server database administrator.

- **Import, Export, Bidirectional** - Sync direction. Our current global database is the reference. An Import will import records into our current global database.
- **Remember Settings** - Remember database location information.
- **Sync** - Initiates the synchronization with the top progress bar showing the overall progress and the bottom showing detail progress.
- **Status Bar** - Displays status of the sync.
- **Advanced Tab** - Select the collections to be synchronized.

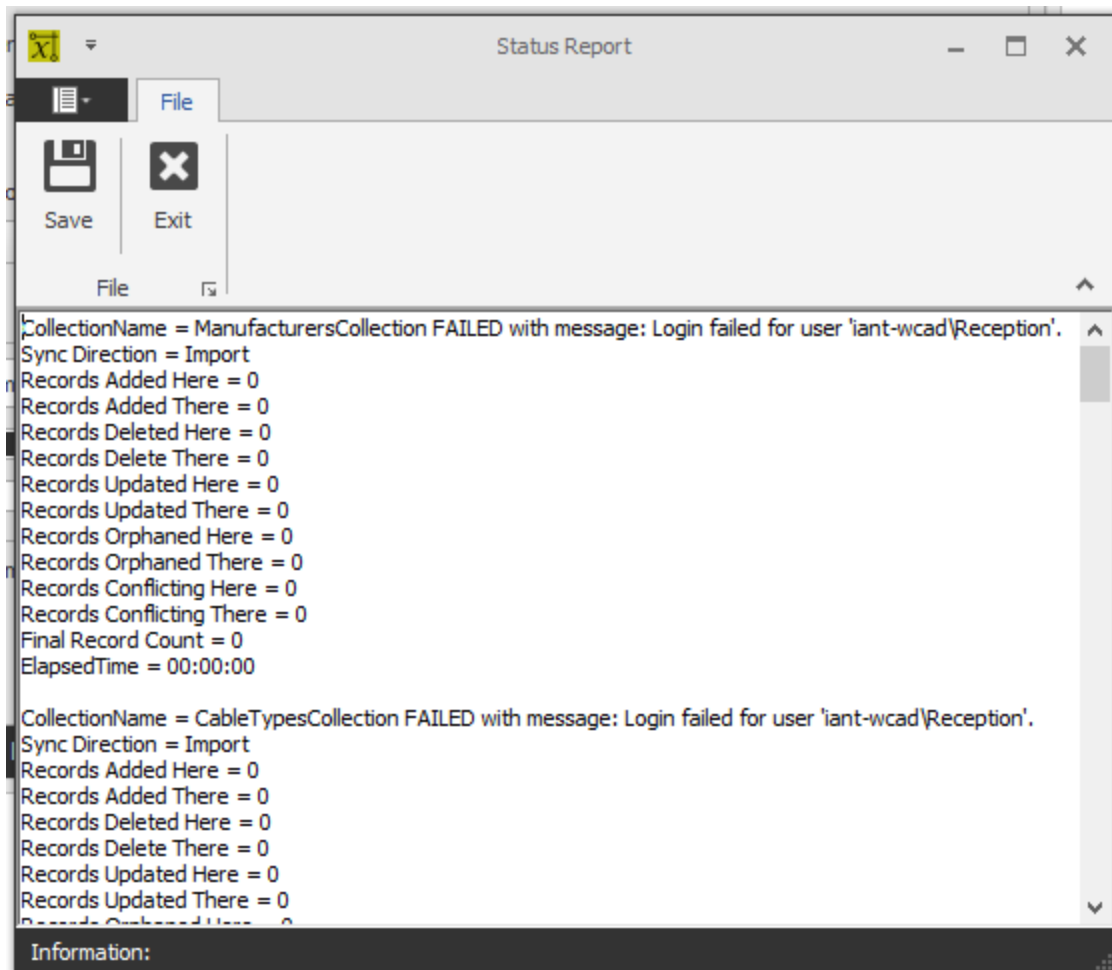
3.5.4.1 How To: Sync Equipment Databases

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Frequently Asked Questions](#) ¹⁷⁹ > [Synchronizing with Another Equipment Database](#) ¹⁸⁴ >



Remember that the Sync Direction is with respect to the current Global Database.

1. Click **Database > Synchronize Global Database ...**
2. Browse to the remote WireCADGlobalEquipment.vdb3 file or enter the SQL host and login for the other SQL server.
3. Set the **Sync Direction**.
4. Click **[Sync]**.
5. Wait for the process to finish.
6. When done you will be presented with a status report. Examine it to make sure that all collections synchronized properly.



3.5.5 Setting Up On a Network

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Frequently Asked Questions](#) ¹⁷⁹ >





What follows is a discussion of several different network topologies and work flows that WireCAD can employ. Regardless of topology or work flow, the following steps should be taken for each WireCAD machine. For purposes of this discussion, we will distinguish between a Windows user (operating under group policy or not) and the WireCAD user. The Windows users will be referred to as OS users. WireCAD users will be referred to as WC users.

1. Install WireCAD
2. Create a network share that is visible to all WireCAD users. Group policy for the OS user of WireCAD should allow the user to read and write the registry (restriction of the registry editor is acceptable), as well as, read and write files on the selected shares, the WireCAD9 folder on the client machine, the OS user's temp directory, the OS user's All Users documents and settings folder trees.
3. Pick one WireCAD client machine from which to copy the global databases and copy from ...
 \WireCAD9\WireCADGlobalEquipment.vdb3 to \
 \YourNetworkShare\FolderForWireCADGlobalDatabases\WireCADGlobalEquipment.vdb3
4. Launch WireCAD on the client machine.
5. Click **Application Menu > Settings{Support Paths}**

6. Modify the support paths for the **Global Equipment** database, and any blocks or reports that you wish to share among all users.
7. Click **[Done]** and relaunch WireCAD.
8. Click **Application Menu > Security > View Permissions**. If you are an administrator or rather if your WireCAD identity is that of an Administrator, you will have edit abilities on this grid.
9. WireCAD uses your Windows groups. You assign permissions to the group. The current user identity is set to the group thus determining their access level.

3.5.6 Upgrading from V9

Navigation: [WireCAD PRO Manual](#)  > [Frequently Asked Questions](#)  >



If you are upgrading from v9 you will need to take the following steps:

1. Install WireCAD v10
2. Setup your global databases (file based or SQL Server)
3. Click **Database > Sync Equipment Libraries**
4. Click the ellipsis (...) and enter the path to the v10 WireCADGlobalEquipment.vdb3 file or set the SQL Server Host info.
5. Click the Import radio button.
6. Click **[Sync]** to sync the two databases.
7. Next convert any projects that you want to work with in v10
8. Click **Application Menu > Open Project** and browse to your v10 <YOURPROJECTNAME>.wc6plf file. WireCAD will open and convert the project. When WireCAD converts projects no drawings are touched. The database schema is modified.

Note: this may take some time



WireCAD can no longer upgrade a v5 or earlier project. Key data access components needed to open and import data from the Microsoft Access databases are no longer available for modern 64bit operating systems.

If you need to bring a v5 or earlier project forward we recommend using a 32bit XP virtual machine with WireCAD v6 installed. Convert the project to v6. Then open it in v10 once converted.

We offer this service if you get in trouble. Call us.

3.6 Choosing a Database Format

Navigation: [WireCAD PRO Manual](#)  >



WireCAD v10 PRO and CMS allows the use of file based and server based databases for the project and global databases. The choice of which to use requires some forethought. Listed here are some basic considerations:

	SQL Azure	SQL Server	VistaDB (File Based)
--	-----------	------------	----------------------

Zero Administration			X
Portable			X
ACID Compliant (atomicity, consistency, isolation, durability)	X	X	X
Database Size	2 Gig	Theoretically unlimited	Theoretical limit is 16 Exabytes (uint64). Practical limit is based upon machine resources. Files are not limited by the database engine, but loading very large databases will require large system resources.
In Process Processing			X
Cloud Based	X		

At first glance that the table above it would seem that the proper choice would be the file based solution. However, take note of the item - In Process Processing means that the WireCAD processes must read and write all data to and from the file based database. Using SQL Server allows us to hand those processes off to the database server creating, in many instances, a significant (read 10X) increase in speed.

Before selecting a database format consider the following questions:



- Will I be moving the project from machine to machine? If yes, consider staying file based on the project.
- Do I have the chops to manage a SQL Server? If no, stay file based. SQL Server requires care and feeding.
- Am I away from my network when I work on WireCAD projects? If you lose connection, WireCAD will become hampered.
- Do I really need the speed enhancements? If you are working on projects with hundreds of thousands of cables, SQL Server is a must.

When you move a project using the **Application Menu > Utilities>Pack Up/Check Out** function and you are using SQL Server databases, the database is converted to a file based version and will remain file based from that point forward.

To set up SQL Server see [here](#) 

Need help with SQL Azure? Contact us we can help.

3.6.1 How To: Setup a SQL Server

Navigation: [WireCAD PRO Manual](#)  > [Choosing a Database Format](#) 



It is not within the scope of this manual to provide an in depth discussion of SQL Server. We will touch on the basics required for use with WireCAD.

The WireCAD distribution includes SQL Server database files for the Global Equipment database. You will need to attach these to the running server. You will then configure WireCAD to look at the SQL Server for the Global Equipment database. WireCAD projects will create a new database (catalog) for every new project.

You will need to set up permissions for each user to allow them dbcreator privileges. This is the default for localhosts but not remote servers.

SQL Server can be set up on a server or on a local machine. WireCAD requires the 2005 version or newer and can use the Express versions.

SQL Server does not provide a graphical user interface so you will want to download the **SQL Server Management Studio Express** as well. Both are free.

The following assumes that you are installing WireCAD on a local machine. If you are installing on a server you will want to copy the global database files in Step 6 below to the server before you attach them to the server.

1. Install **SQL Server**. You can download from the following links:

<https://www.microsoft.com/en-us/download/details.aspx?id=21844>

Be sure to download SQL Server Management Studio Express as well. Both are free.

<http://www.microsoft.com/downloads/details.aspx?FamilyID=c243a5ae-4bd1-4e3d-94b8-5a0f62bf7796&DisplayLang=en>

2. You will be prompted for an instance name. Instance names allow you to have multiple SQL Servers running on the same machine. In addition you will be prompted for a security mode (Windows or SQL), Windows uses your Windows users and groups, SQL ignores these and allows you to manage different users and groups from within SQL Server.

3. Install **SQL Server Management Studio (SQLSMS)**.

4. Launch **SQLSMS** and log in to the server. If the server is on your local machine you can use the shorthand [.\INSTANCENAME] (eliminate the square braces) for the Localhost.

5. Setup a proper login for WireCAD. You may elect to use Windows logins or not. It is beyond the scope of this manual to describe the pros and cons of each authentication method.

6. Attach the WireCADGlobalDatabase.mdf file to the server.

7. You will now need to configure each WireCAD client to look at the SQL Server

8. Launch WireCAD.

9. Click **Application Menu > Application Setup** and follow the steps for SQL Server.

See Also

www.wirecad.com/wiki search SQL Server

msdn.microsoft.com search SQL Server

3.7 Included Plugins

Navigation: [WireCAD PRO Manual](#) 48 >



WireCAD includes a number of Plugins that provide additional functionality.

Each plugin will load and register its own command line commands if the plugin provides command functionality.

For information on customizing WireCAD with your own Plugins see the [SDK](#) ⁵⁷⁶ documentation and the SDK samples.

3.7.1 Pinouts

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Included Plugins](#) ¹⁹⁰ >



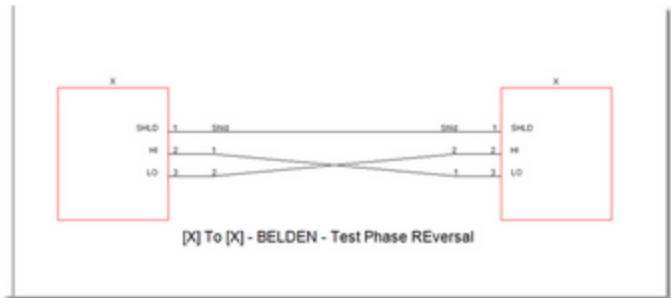
Menu: **Database>Pinouts**

Default command line shortcut: **po**

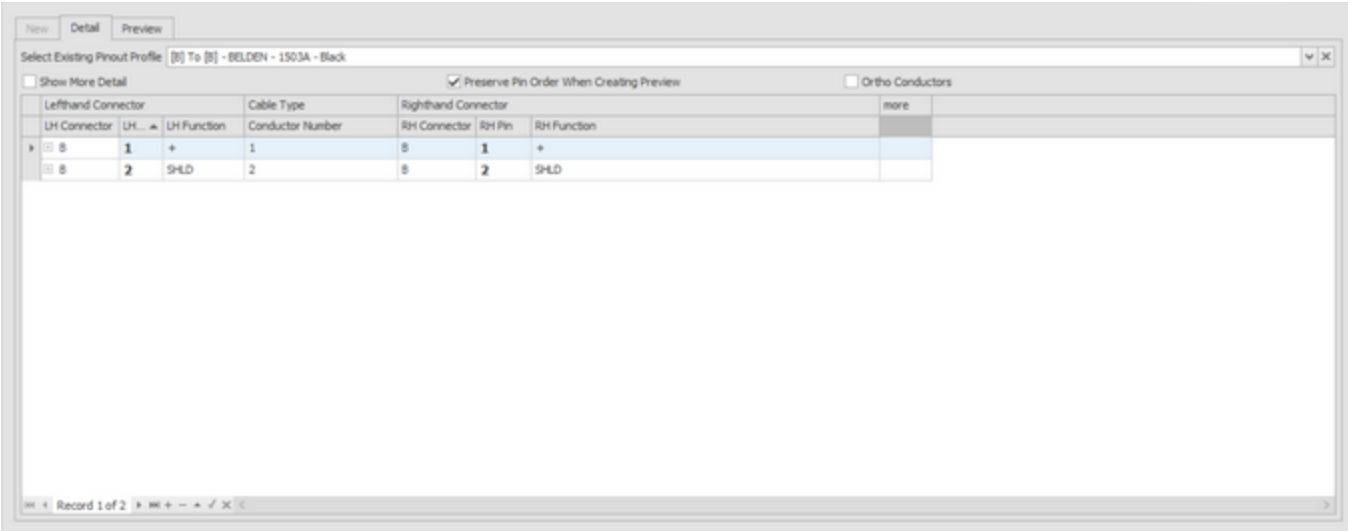
Related Settings:

None

The Pinout utility serves two functions. The first is to generate pinout detail drawings.



The second is to generate data that can be used in the conductors table under any cable that we wish to attach the data to.



3.7.2 Find and Replace

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Included Plugins](#) ¹⁹⁰ >



Menu: **View>Tool Panels - Find**

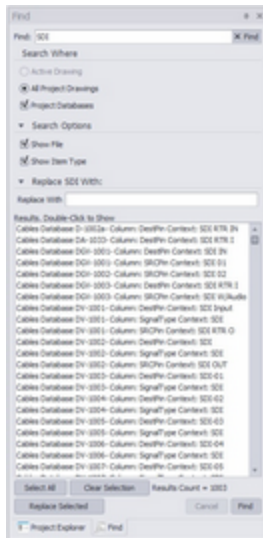
Default command line shortcut: **none**

Related Settings:

None

The Find and Replace tool window allows you to search all of the data collections and drawing. You may then replace found text with that of your choosing.

The Find and Replace tool shows you the context in which the text was found. You can select from the results list which items to replace.



- **Find** - The text to search for.
- **[X]** - Clear the Find text.
- **Search Active Drawing** - Only look in the active drawing.
- **Search All Project Drawings** - Search every project drawing.
- **Show File** - Show file name if any of the context in which the item is found.
- **Show Item Type** - Is it a text entity in a drawing or a field in a database.
- **[Select All]** - Selects all items in the results window.
- **[Clear Selection]** - De-selects all items in the results window.
- **Replace With** - The text to replace the found text with.
- **[Replace Selected]** - Initiate the replacement.
- **[Cancel]** - Cancels the current search.
- **[Find]** - Initiate the search.

3.7.2.1 How To: Find and Replace a Port Name

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Included Plugins](#) ¹⁹⁰ > [Find and Replace](#) ¹⁹² >



In this section we will discuss the finer points of finding and replacing text in WireCAD blocks.

Before You Start

When you see a WireCAD block in a drawing it is referred to as an Insert. WireCAD blocks employ Attributes to display the text values you see. These Attributes are editable. For every port you see in a WireCAD block there is at least one invisible attribute at the connection point. This attribute contains a pipe | delimited string formatted as:

PORTNAME|SIGNALTYPE|CONNECTOR

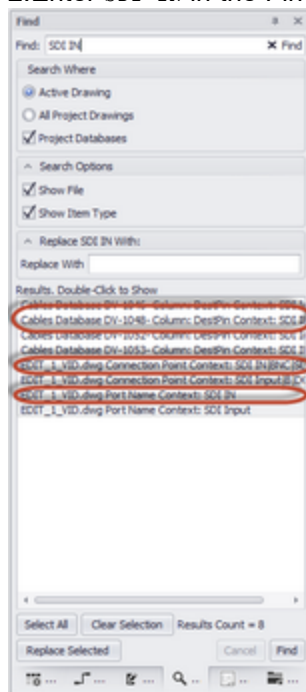
In addition to the connection point attribute there may be a Port Name display attribute.



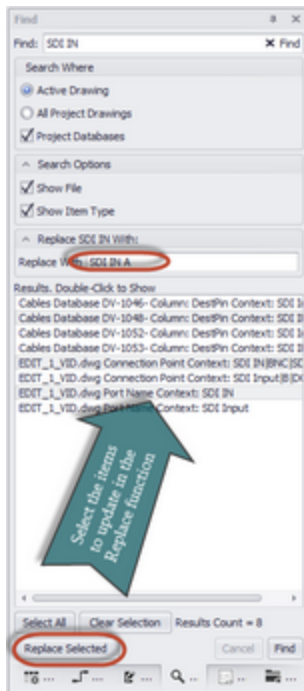
If you change the display attribute **WITHOUT** changing the connection point attribute you **WILL** have grief later. So **ALWAYS** changed both the display attribute and the connection point attribute concurrently.

1. Let's say we want to replace SDI IN on our device with SDI IN A. Further that we have assigned cable numbers to the port in question. We will only look in the active drawing so click the appropriate radio button.

2. Enter SDI IN in the Find field and click **[Find]**. Wait for it....



3. Notice that we have two attributes and a cable all associated with the port SDI IN. If we don't change all of these we will have data integrity issues later.



4. Click **[Replace Selected]**.
5. Done.

3.7.3 Translation Manager

Navigation: [WireCAD PRO Manual](#) ^[48] > [Included Plugins](#) ^[190] >



Menu: **Plugins>Translation Manager**

Default command line shortcut:

Related Settings:

None

All strings in the application are stored in a dictionary with a key and current culture translation. The translation utility provides a means by which all strings, captions, and messages may be translated to a different language.

You can right-click a column header and select the KeyString column to see the base English string that the program searches from.

Controls

Drag a column header here to group by that column

KeyString	en	fr	CalledBy
WireCAD	WireCAD		
UserSettings.xml	UserSettings.xml		
SupportPaths	SupportPaths		
Group	Group		WireCAD.Translation.wcLayoutLocalizer
Checked	Checked		WireCAD.Translation.wcEditorLocalizer
Unchecked	Unchecked		WireCAD.Translation.wcEditorLocalizer
Loaded Plugins:	Loaded Plugins:		WireCAD.frmSplash.labelControl1.Text
PRO	PRO		WireCAD.frmSplash.lblMode.Text
Copyright 2000-2009 Holbrook Enterprises, Inc.	Copyright 2000-2015 Holbrook Enterprises, Inc.		WireCAD.frmSplash.lblCopyright.Text
Serial Number: - Trial	Serial Number: - Trial		WireCAD.frmSplash.lblSN.Text
Info	Info		WireCAD.frmSplash.lblInfo.Text
Lubing up the engine room	Lubing up the engine room		WireCAD.frmSplash
Lubing up the engine room	Lubing up the engine room		WireCAD.Workspace
AutoSave In: {0}	AutoSave In: {0}		WireCAD.frmMain
(Unassigned Items)	(Unassigned Items)		WireCAD.Translation.wcBarLocalizer
(All Items)	(All Items)		WireCAD.Translation.wcBarLocalizer
Custom {0}	Custom {0}		WireCAD.Translation.wcBarLocalizer
layoutControl1	layoutControl1		WireCAD.ucDrawCables.ioJunctions.Text
Junctions	Junctions		WireCAD.ucDrawCables.layoutControlGroup16.Text
Start Cable	Start Cable		WireCAD.ucDrawCables.simpleButton5.Text
Cable Drawing Properties	Cable Drawing Properties		WireCAD.ucDrawCables.gcPrefs.Text
Preview:	Preview:		WireCAD.ucDrawCables.layoutControlGroup17.Text
True	True		WireCAD.ucDrawCables.rgManyToManyDb.Text
layoutControl3	layoutControl3		WireCAD.ucDrawCables.rgManyToManyDb.Text
X Feeder Distance:	X Feeder Distance:		WireCAD.ucDrawCables.layoutControlItem31.Text
Y Feeder Distance:	Y Feeder Distance:		WireCAD.ucDrawCables.layoutControlItem22.Text
Hor Extension:	Hor Extension:		WireCAD.ucDrawCables.layoutControlItem24.Text
Output (Source)	Output (Source)		WireCAD.ucDrawCables.gcSrc.Text

Record 7 of 8062

Here we see the Translation Manager with the English version on the left and the French version on the right. The pane on the right-hand side shows statistics.

- Current Culture is:** - The culture of your machine. If no translation exists, WireCAD defaults to the en (English) language.
- Select Language To Translate** - Selects the language to edit in the right-hand column.
- Show Context** - Select to display a column showing the primary context in which the string or message appears.

3.7.3.1 How To: Translate a Form Caption

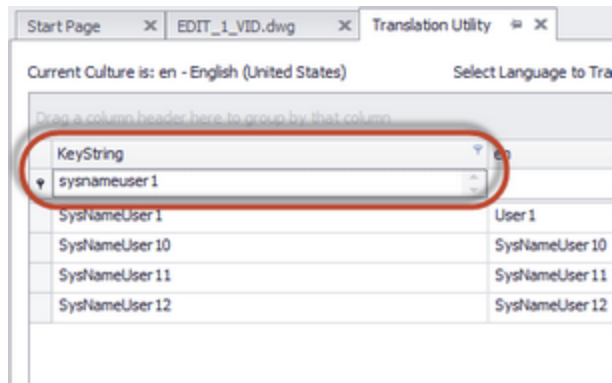
Navigation: [WireCAD PRO Manual](#)^[48] > [Included Plugins](#)^[190] > [Translation Manager](#)^[194] >



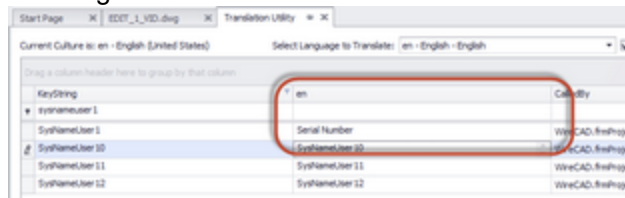
Here we will use the Translation Manager to change the User1 field on the SysName Assignment dialog to display Serial Number.

We are starting with:

1. Click **Plugins > Translation Manager**.
2. Enter `sysnameuser1` in the **KeyString** filter field.



3. Change User1 to Serial Number in the en field.

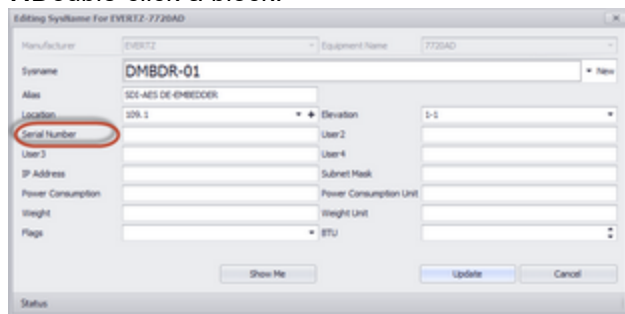


4. Click **File > Save**.

5. Now let's check the result by showing the form.

6. Open a drawing with a functional block.

7. Double-click a block.



3.7.4 Block Extractor

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Included Plugins](#) ¹⁹⁰ >



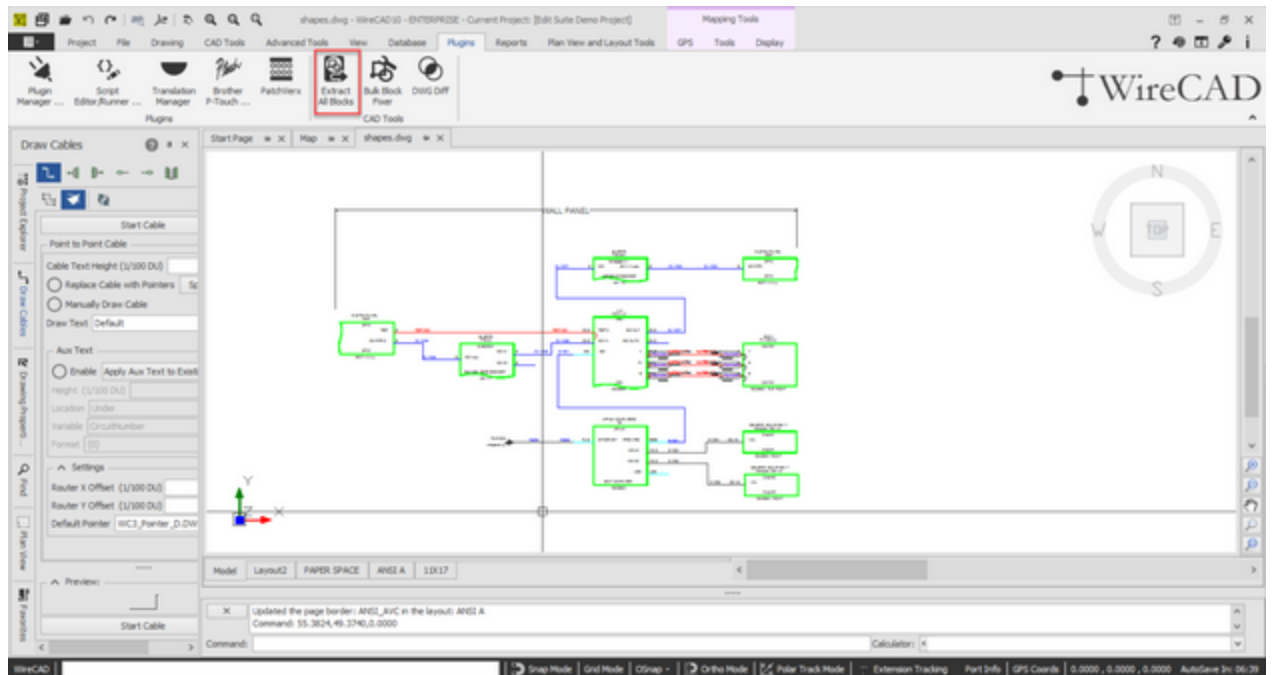
Menu: **Plugins > Extract All Blocks**

Default command line shortcut: **exall**

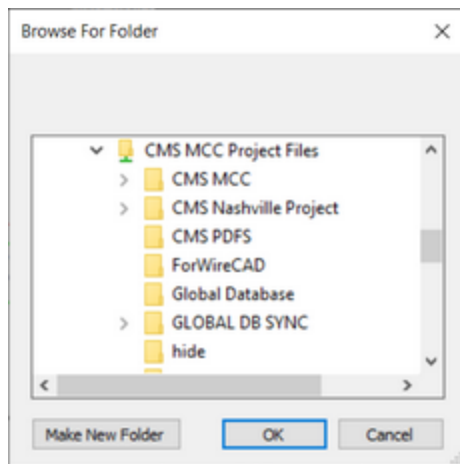
Related Settings:

None

Requires an active drawing.



- Extracts all blocks in the current drawing to the folder of your choosing.



- Files are written out of the drawing as individual dwg files with the name of the block.

Name	Date modified	Type	Size
Boundary Mono.dwg	12/13/2009 10:02	WireCAD6 DWG D...	61 KB
Boundary Stereo.dwg	12/13/2009 10:02	WireCAD6 DWG D...	62 KB
Clip Mic.dwg	12/13/2009 10:02	WireCAD6 DWG D...	63 KB
Contact.dwg	12/13/2009 10:02	WireCAD6 DWG D...	60 KB
Dual Mic.dwg	12/13/2009 10:02	WireCAD6 DWG D...	63 KB
Field Mixer.dwg	12/13/2009 10:02	WireCAD6 DWG D...	81 KB
FDPOLL.dwg	12/13/2009 10:02	WireCAD6 DWG D...	70 KB
Hand Mic.dwg	12/13/2009 10:02	WireCAD6 DWG D...	65 KB
James.dwg	12/13/2009 10:02	WireCAD6 DWG D...	83 KB
Large Mic.dwg	12/13/2009 10:02	WireCAD6 DWG D...	67 KB
Long Mono Shotgun.dwg	12/13/2009 10:02	WireCAD6 DWG D...	65 KB
Long Stereo Shotgun.dwg	12/13/2009 10:02	WireCAD6 DWG D...	66 KB
MICROPHONE NUMBER BIG.dwg	12/13/2009 10:02	WireCAD6 DWG D...	60 KB
MICROPHONE NUMBER.dwg	12/13/2009 10:02	WireCAD6 DWG D...	60 KB
Mini Lapel Directional.dwg	12/13/2009 10:02	WireCAD6 DWG D...	63 KB
Mini Lapel Omni.dwg	12/13/2009 10:02	WireCAD6 DWG D...	62 KB
Mini Shotgun.dwg	12/13/2009 10:02	WireCAD6 DWG D...	64 KB
Radio Mic.dwg	12/13/2009 10:02	WireCAD6 DWG D...	61 KB
Radio Super Shotgun.dwg	12/13/2009 10:02	WireCAD6 DWG D...	83 KB
RF Mic.dwg	12/13/2009 10:02	WireCAD6 DWG D...	61 KB
RF Receiver.dwg	12/13/2009 10:02	WireCAD6 DWG D...	90 KB
Short Mono Shotgun.dwg	12/13/2009 10:02	WireCAD6 DWG D...	62 KB
Short Stereo Shotgun.dwg	12/13/2009 10:02	WireCAD6 DWG D...	62 KB
Super Shotgun.dwg	12/13/2009 10:02	WireCAD6 DWG D...	74 KB
YDORA DEFAULT.T.dwg	12/13/2009 10:02	WireCAD6 DWG D...	60 KB

- The basepoint of the drawing (origin) is the insertion point of the block.

3.7.5 Bulk Block Fixer

Navigation: [WireCAD PRO Manual](#) [48] > [Included Plugins](#) [190] >

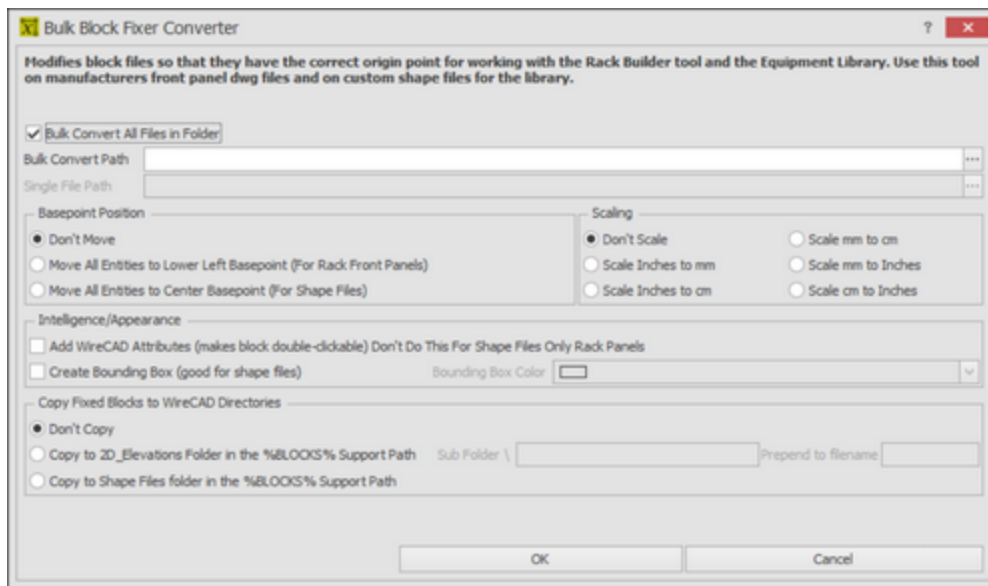


Menu: **Plugins>Bulk Block Fixer**

Default command line shortcut: **blkf**

The **Bulk Block Fixer** utility provides a means by which dwg files may be easily converted for use with WireCAD. The fixer will convert a single drawing or all drawings in a folder.

Bulk Block Fixer Dialog



- Bulk Convert All Files In Folder** - Checked performs selected options on all files in selected folder.

- Basepoint Position** -

Don't Move - Do nothing.

Move All Entities To Lower Left Basepoint.



Before



After

Move All Entities To Center Basepoint.



Before



After

•**Scaling** - Scales all drawing objects based on the selection.

•**Intelligence/Appearance**

Add WireCAD Attributes. - Use this function to add the standard WireCAD attribute set to the dwg file. This makes inserted items double-click-able. This is not required, and should not be used, if you are using the dwg file as a DWGIcon or Custom Shape file.

Create Bounding Box - Creates a rectangle around the entities in the dwg file of the color selected. This is useful for DWGIcons or Custom Shape files

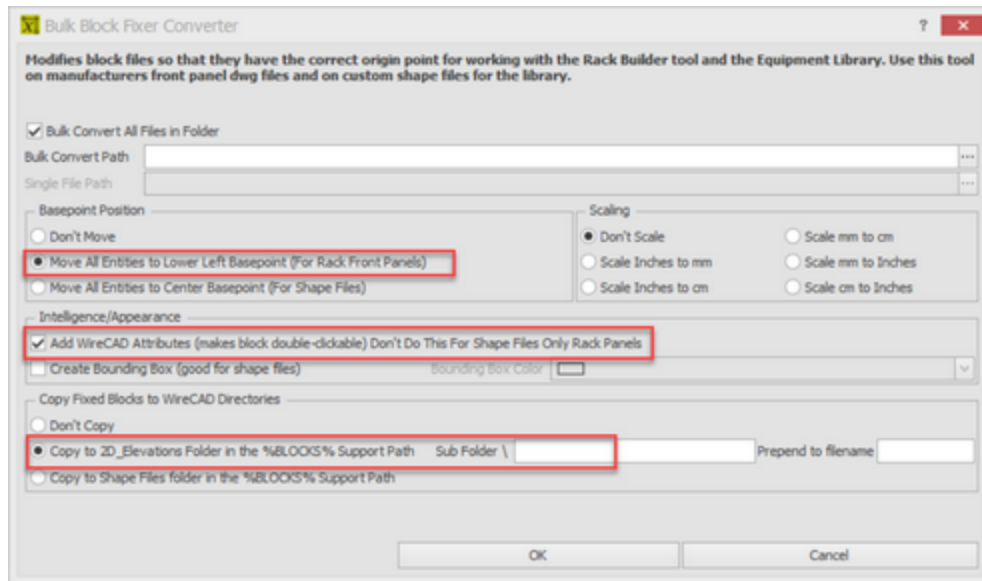
•**Copy** - Copies fixed block files to the selected directories. \Subfolder and Prepend allow you to modify the location and filename upon copy.

3.7.5.1 How To: Use a Manufacturer Front Panel

Navigation: [WireCAD PRO Manual](#) [48] > [Included Plugins](#) [190] > [Bulk Block Fixer](#) [198] >



1. Obtain the manufacturer's front panel dwg file.
2. Open the file in WireCAD and delete anything that is not related to the front panel view.
3. Close the drawing.
4. Click **Plugins > Bulk Block Fixer ...**
5. Uncheck **Bulk Convert All Files** in Folder.
6. Click the **Single File Path [...]** and browse to the manufacturer's front panel file.



7. Select **Move All Entities to Lower Left Basepoint**.
8. Select **Add WireCAD Attributes**.
9. Click **[OK]**.
10. Open the **Equipment Library**.
11. Find the equipment that we want to use our shiny new front panel
12. Set the **Front Panel File** to point to our file.
13. Click **File > Save**.
14. Now you can use it.

3.7.6 Batch Plot

Navigation: [WireCAD PRO Manual](#) ⁴⁸ > [Included Plugins](#) ¹⁹⁰ >



Menu: **Plugins>Batch Plotter**

Default command line shortcut: **bp**

Related Settings:

None

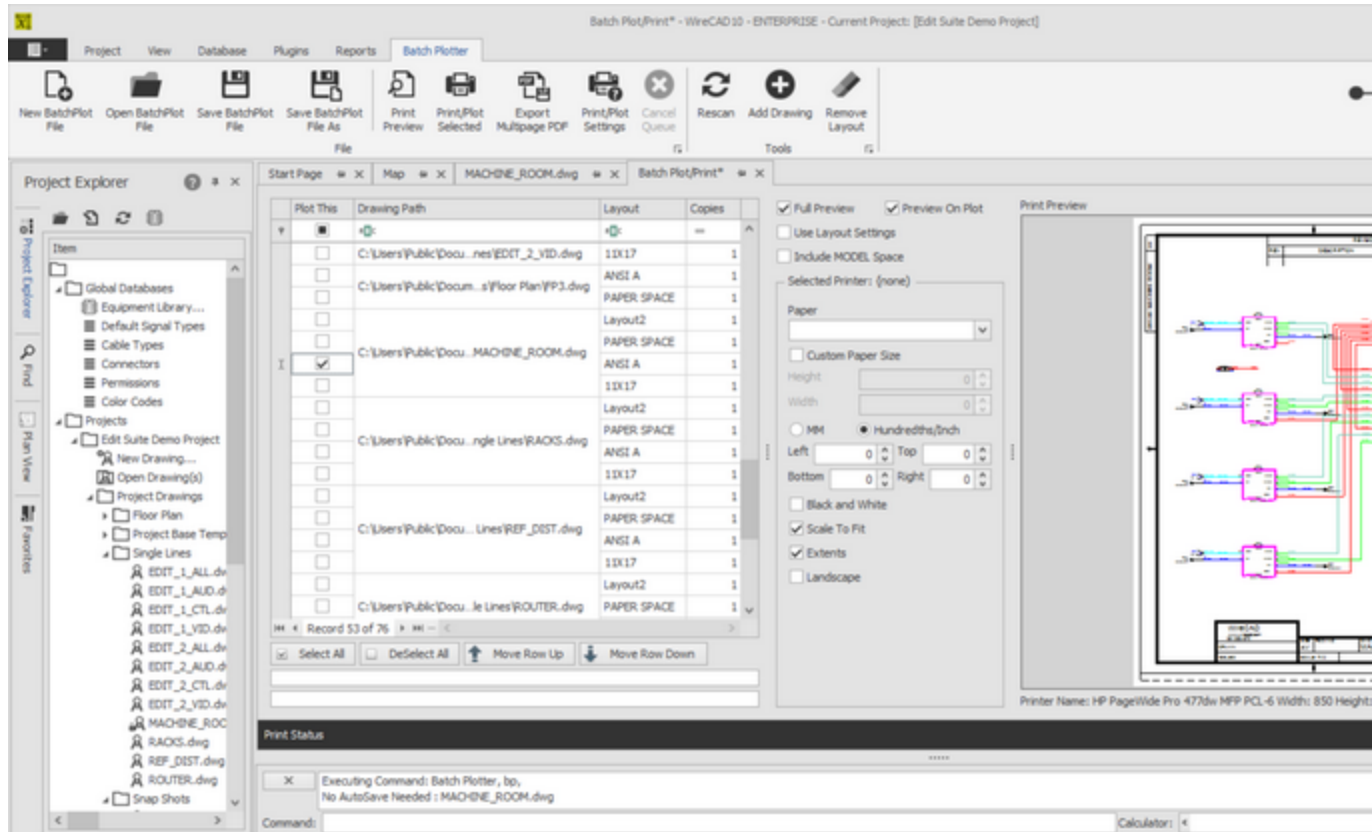
The **Batch Plot** plugin allows you to select any number of layouts to print from project drawings. The first step is to click the **[Rescan]** button. This populates the list with all drawings and layouts. The **Plot This** checkbox determines which layouts to send to the printer. The **[Plot/Print]** button is the same as the **File>Plot/Print** menu command and sends all selected layouts to their respective printers. The **[UP]** and **[Down]** buttons move the selected layout up or down in the list thereby reordering their output order.

If you want to delete a layout. Select the layout in the list and click the minus **[-]** button in the navigation window in the lower left hand corner of the grid

Clicking the **[Preview]** button will preview the selected layout in the **Preview Pane** regardless of whether it is selected to in the **Plot This** column

What it Does

- **Batch Plot** lets you select the drawings and layouts that you want to send to the printer and print them as a batch.
- You may print any layout in any drawing in the current WireCAD project.
- You may also specify how many copies and in what order to print.
- **Batch Plot** allows you to save Batch Plot files for reuse at a future time.



What it Doesn't Do

- **Batch Plot** does not print project reports.
- **Batch Plot** does not collate layouts with multiple copies set.

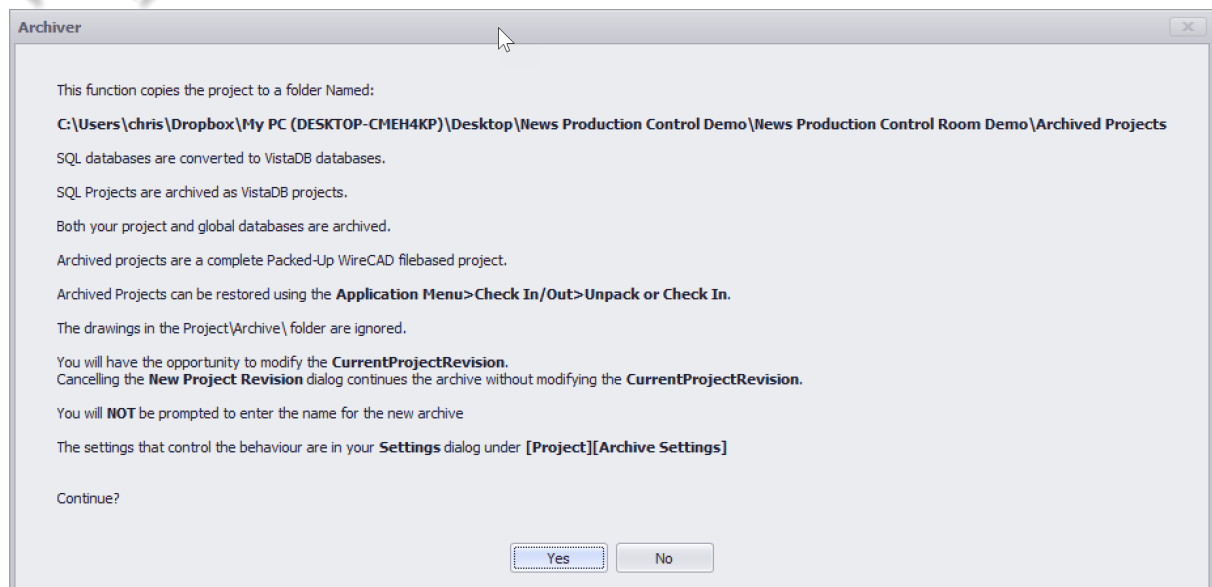
Controls

- **Full Preview** - Render actual output. Otherwise show margin box.
- **Preview On Plot** - Render the full preview (full or not) on output to the printer/plotter.
- **Use Layout Settings** - Ignore the Batch Plotter settings and use those stored with the drawing layout. Selecting this disables printer, paper, and margin settings in favor of those in the layout.
- **Paper** - Printer paper size. If this dropdown is empty, select a printer first.

- Margins** - Set your margin unit of measure first, then the left, right, top and bottom.
- Black and White** - Output in black and white.
- Scale to Fit** - Force all entities in the selected layout onto the page.
- Extents** - Print the extents of the layout.
- Landscape** - Landscape output.
- Select Printer** - Choose the printer that all selected layouts will be sent to.
- New Batch Plot File** - Creates a new file scanning the project drawing set and adding all drawing and all layouts.
- Open Batch Plot File** - Opens a saved Batch Plot file.
- Save** - Saves your settings to a Batch Plot File. This is helpful if you change the default order of the list or the number of copies, etc.
- Save As** - Save your setting as a new file name.
- Print/Plot Selected** - Prints the selected (Plot This) layouts to the printer settings determined in the layout. So be sure to Print Preview (from the drawing layout) your layouts first and set up all of your output settings.
- Export Multipage PDF** - Output the selected (Plot This) layouts with the selected printer settings to a multipage PDF document.
- Cancel Queue** - Stops the function. Does not purge any print jobs already in the windows print queue.
- Exit** - Done.
- Rescan** - Clears the list and re-scans the project drawings for all layouts. This function is like the File>New function but differs in that it does not create a new file name.
- Add Drawing** - It may be desirable to keep all of your settings and add an additional drawing to the list.
- Remove Layout** - Removes the selected layout.

3.7.7 Archive Project

Navigation: [WireCAD PRO Manual](#)  48 > [Included Plugins](#)  190 >



Menu: **Project>Archive Project**


Default command line shortcut: **ap**

Related Settings:

None

The Archive tool copies all of the project data except the drawings in the Project\Archive folder to a new folder. The tool can also create a new CurrentProjectRevision at the same time.

The archived version of the project:

- Copies the drawings and databases.
- SQL databases are archived as VISTADB databases.
- Global and Project databases are included.
- The **Project\Archive** drawings folder is ignored.
- Gives you the choice to increment the [CurrentProjectRevision](#) .

The associated settings panel controls:

- The Archive path.
- The Archive name.
- Automatic naming or user involvement.
- CurrentProjectRevision** inclusion.

The output of the archive tool creates a Packed-Up Project that can be unpacked or checked-in.

Restoring an archived project can be done using the UnPack function or the Check-In function.

Related Topics:

[Project Archive Tool Settings](#) 

[Unpack Project](#) 

[Check-In Project](#) 

3.8 Project Rules

Navigation: [WireCAD PRO Manual](#)  >



WireCAD 10 allows you to define project specific rules.

There are four contexts in which rules will be evaluated. Rules are contained in the **Project Rules** grid.

Rules can be enabled/disabled by use of the **Project>Settings[Project Rules] Enable Project Rules** setting.

A **Rule** consists of:

- 1.The **Expression** to evaluate.
- 2.The **Context** in which to evaluate the expression.
- 3.The **ExecutionOrder**. If multiple rules are defined to evaluate within the same **Context** they will be evaluated in the **ExecutionOrder** order.
- 4.A **Description** of the rule.

Rule Contexts:

- 1.**AssignCableNoAssignBeforeEdit** - executed during a new Cable Number Assignment or existing Cable Number Edit before the dialog is shown.

2. **AssignCableNoAssignAfterEdit** - as above but after the user has clicked Save on the dialog or if **Verbose Cable No Assignment** = false then during the **Cable** save process.
3. **AssignSysNameAssignBeforeEdit** - executed during a new SysName Assignment or existing SysName Edit before the dialog is shown.
4. **AssignSysNameAssignAfterEdit** - as above but after the user has clicked Save on the dialog or if **Verbose SysName Assignment** = false then during the **SysName** save process.

Example

Let's assume that you have the following requirements:

1. SDI SignalType where the Length > 100 should use a different CableType than the default defined in the Project Signal Types Default Cable Type grid.
2. We want to modify the connector type from BNC to KINGS 2065-10-9 if the CableType is 1694

For requirement number 1 we will create a new rule as follows:

- **Context - AssignCableNoAssignAfterEdit.** We use this so that we can get the Length from the user.
- **Expression** - `(SignalType = 'SDI' && Length >=100) then set CableType = '1694'`
- **Description** - My First Rule. Change cable type for SDI if over 100. This is descriptive text for you and your team. It can be whatever.
- **ExecutionOrder** - 0. Rules execute in the order specified. Any Rule with the **ExecutionOrder** < 0 will not execute.

For requirement number 2 we will create two rules as follows:

- **Context - AssignCableNoAssignBeforeEdit.** We use this so we can see our changes in the edit dialog.
- **Expression** - `(SRCConn = 'BNC' && CableType = '1694') then set SRCConn = 'KINGS 2065-10-9'`
- **Description** - BNC to Kings
- **ExecutionOrder** - 0. Rules execute in the order specified. Any Rule with the **ExecutionOrder** < 0 will not execute.
- **Context - AssignCableNoAssignBeforeEdit.** We use this so we can see our changes in the edit dialog.
- **Expression** - `(DestConn = 'BNC' && CableType = '1694') then set DestConn = 'KINGS 2065-10-9'`
- **Description** - BNC to Kings
- **ExecutionOrder** - 1. Rules execute in the order specified. Any Rule with the **ExecutionOrder** < 0 will not execute.

3.8.1 Expression Syntax

Navigation: [WireCAD PRO Manual](#)  > [Project Rules](#)  >



```
(true part [ boolean other true part ]) then set SomeProperty = 'SomeStringValue' [else set
SomeProperty = 'SomeOtherStringValue']
syntax in [] is optional
```

Description	Syntax	Example
Single part boolean that sets	<code>(1=1) then set PropertyName =</code>	Assume the property we want to test is the SignalType field and if true then change the CableType to equal 1505 <code>(SignalType = 'SDI') then set CableType = '1505'</code>

a property if true	'string value'	
Multipart boolean AND that sets a property if true	(1=1 && 2=2) then set PropertyName = 'string value'	Assume the property we want to test is the SignalType field AND the Length field and if true then change the CableType to equal 1505 (SignalType = 'SDI' && Length > 100) then set CableType = '1505A'
Strings	'some string'	
Numbers	1001	
Boolean AND	&&	this and that then (1=1 && 2=2) then ...
Boolean OR		this or that then (1=1 2=2) then ...
Equality	=	
Test Null	is null	(SomeProperty is null) then ...
Set a Property (Assignment)	set PropertyName =	(1=1) then set NumericPropertyName = 1000 or (1=1) then set StringPropertyName = 'string value'
Wildcard (any from here)	%	Used in conjunction with the LIKE operator (CableType LIKE '1505%') then ... will evaluate to true for any value that starts with 1505
Wildcard (any one)	?	Used in conjunction with the LIKE operator (CableType LIKE '1505?') then ... will evaluate to true for any value that starts with 1505 and has one and only one additional character.

Limitations

This syntax has no block structure. If you need to execute multiple set property operations you will need to create multiple rules.

Illustration

The following is **NOT** supported:

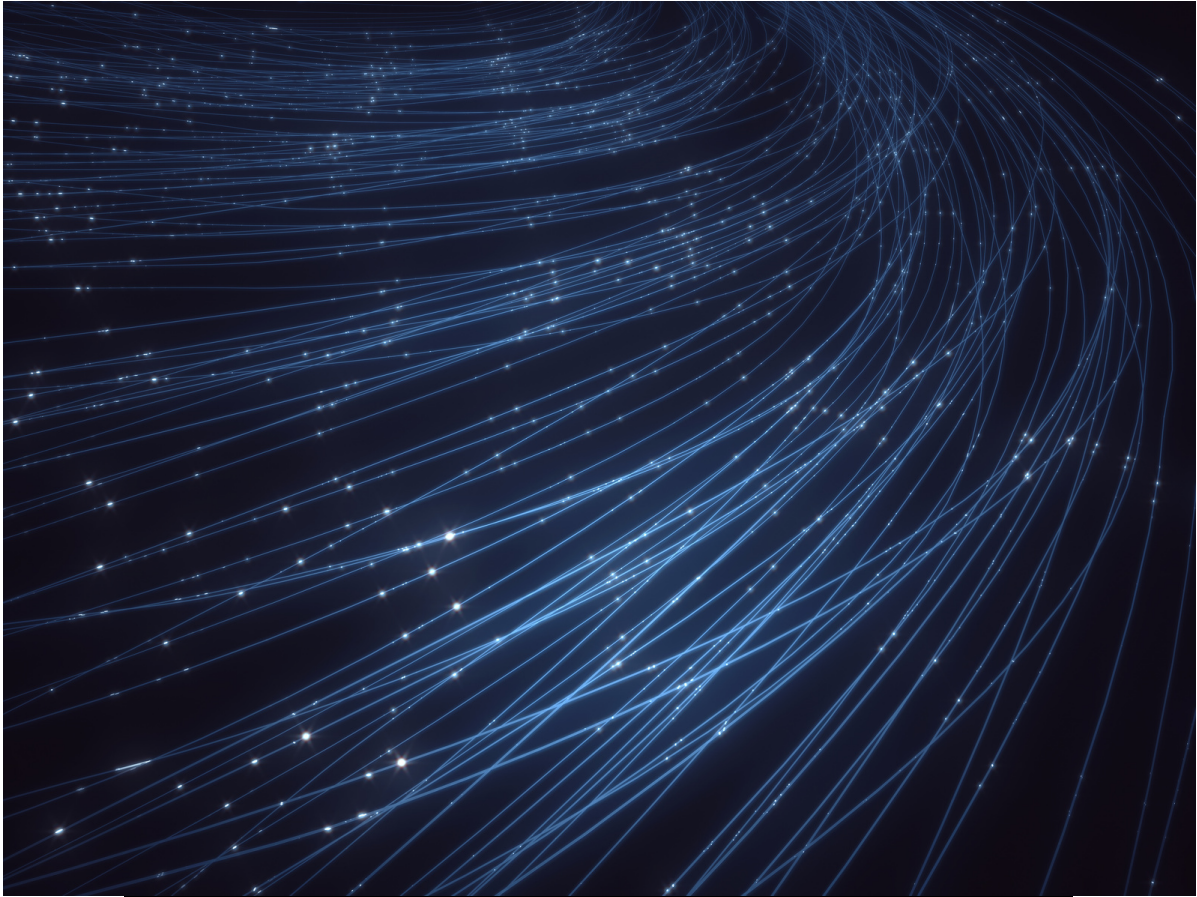
```
(true) then
something;
something else;
some other thing;
else
another thing
or another thing
```

Consider creating a plugin in the SDK if you need to create very complex rules. There is a Project Rules Events project template to use as a prototype to get you started.

This page is intentionally left blank.
Remove this text from the manual
template if you want it completely blank.

WireCAD CMS Tools Manual

4 WireCAD CMS Tools Manual



The WireCAD CMS tools are built upon and extend the WireCAD PRO tool set for the documentation of large scale management of the fiber systems.

This section pertains to the CMS product level.

4.1 CMS Tools Concept

Navigation: [WireCAD CMS Tools Manual](#)   >



WireCAD v10 with CMS tools has introduced the ability to work in a different manner from previous versions of WireCAD.

The standard work flow for WireCAD users has been to create your Functional Block Diagrams first, then assign SysNames and Cable Numbers and lastly, generate reports based on this information.

WireCAD v10 CMS allows you to start by entering your SysNames and Cable Numbers into the Database first, create Backbones and Circuits and then generate a Functional Block Diagram based on the information set in the Database.

The CMS tools work on a declare then visualize approach. You tell the database that the thing exists and WireCAD provides different ways to visualize the data in the database.

If the database item of interest is a fiber patch panel you can visualize all connections to and from that panel.

If the database item of interest is a fiber backbone you can visualize as one of the following:

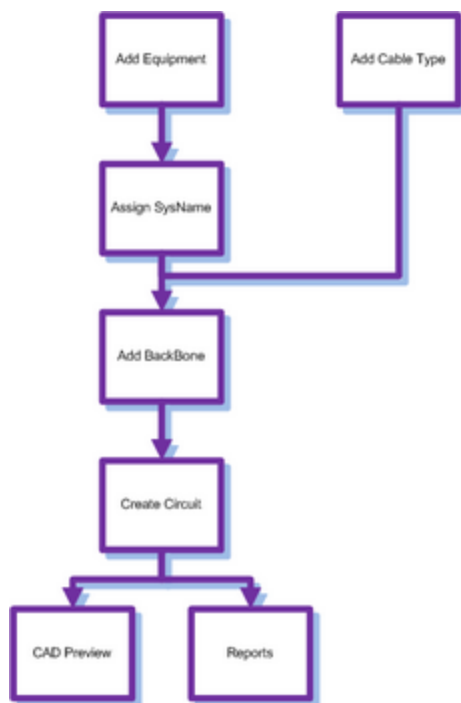
- Riser Diagram
- Ladder Diagram
- Map
- GPS Calibrated (Geo-Referenced) DWG drawing

If the database item of interest is a circuit you can visualize as:

- Circuit Functional Block Diagram
- Map

WireCAD CMS also provides the capability to view your fiber infrastructure in Map view. Using GPS coordinates you can view your plant fiber layout, where fiber patches and panels reside as well as path data for run way-points such as manholes and pull-boxes. You can use the built in Map tool or use your dwg plant drawings (you must [calibrate](#)^[263] the drawing to the GPS coordinate system first).

You can use WireCAD CMS to manage your fiber and copper infrastructures. WireCAD tracks every strand of fiber and every CATx cable and jumper/cross-connect.










4.2 CMS Basics

Navigation: [WireCAD CMS Tools Manual](#)^[208] >



The following is a list of How To's for multiple functions in WireCAD v10 including creating [Cable Types](#)^[523], New Equipment Definitions, Creating Projects and more.

[Creating a New Project](#)  [Creating a New Cable Type](#)  [How To Create a Backbones](#)  [Using The New Circuit Tool](#)  [How to Output a Circuit to CAD](#)  [How to Output Many Circuits to CAD](#)  [Using The Backbone Grid](#) 

The following subjects assume that you understand basic WireCAD procedures:

- Adding Equipment to the Equipment Library
- Adding Signal Types.
- Adding Cable Types.
- Assigning SysNames.
- Adding Locations.

4.2.1 Backbones

Navigation: [WireCAD CMS Tools Manual](#)  > [CMS Basics](#)  >



This is a step by step guide on how to create Backbones. Backbones are represented in the cables database by a collection of cables.

Each cable in the backbone has the same Cable Number Prefix. When you create a new Backbone you are creating a collection of cables based on the selected Cable Type. If you are documenting fibers think one fiber equals one core/conductor/cable. backbone cables are flagged with the boolean value `isBackbone = true`. Doing so distinguishes a cable as being part of a backbone structure.

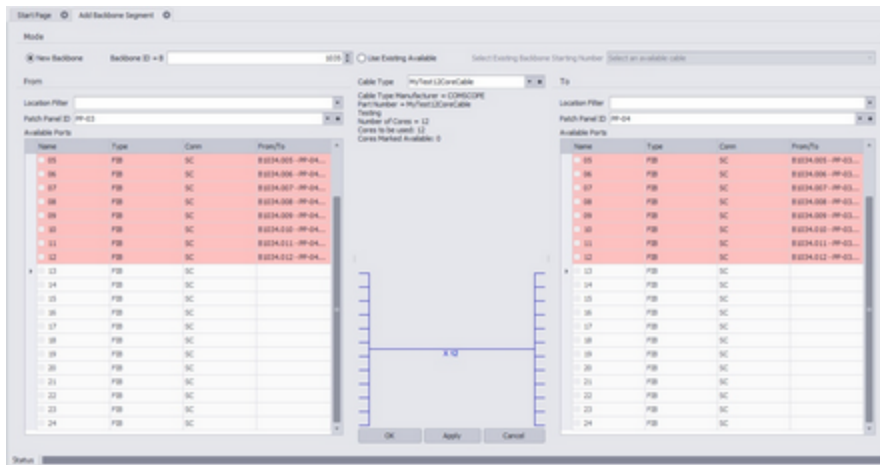
Open the Backbones grid by double-clicking the icon in the Project Explorer **[Project Databases]** **[Backbones]**

Command Line Shortcut: **bbg**

Before You Start

Prerequisites

1. **SysNames** assigned to the panels/equipment to which you will attach the ends of the **Backbone**.
2. **Multi-core Cable Type** representative of the **Backbone**.
3. Before creating a Backbone, you will need to make sure you have created your Equipment and Cable Types associated with this projec



Operation

This tool allows you to draw geometry in the Plan View space that represents the path of the **Backbone**. The process requires the following steps:

3. Draw the polyline that represents the backbone.
4. If locations boundaries are found in the drawing the source and destination locations will be used. Otherwise you will be prompted to define a location for each end.
5. Next you will be presented with the **New Backbone** tool where you will select the source and destination panels/ports and the cable type.
6. Clicking **[OK]** will build a record in the **Cables** table for each core in the multi-core cable type.
7. The polyline length populate the cable record **Length** field.

Dialog Options

- **New Backbone** - Select whether we are creating a new number or assigning an existing, unused core.
- **Backbone ID** - The number.
- **Use Existing Available** - Select an already assigned **Backbone** with available strands to populate the remaining available strands of fiber.
- **Location Filter** - Filter the list by location.
- **From Panel Info** - Select the panel. The ports will be shown in the list.
- **To Panel Info** - Select the panel. The ports will be shown in the list.
- **Cable Type** - Select the Cable Type.

4.2.1.1 The Backbone Grid

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Backbones](#) [210] >



Menu: **Database > Backbones**

Default command line shortcut: **bbg**

Related Project Settings:

Backbone Status Items

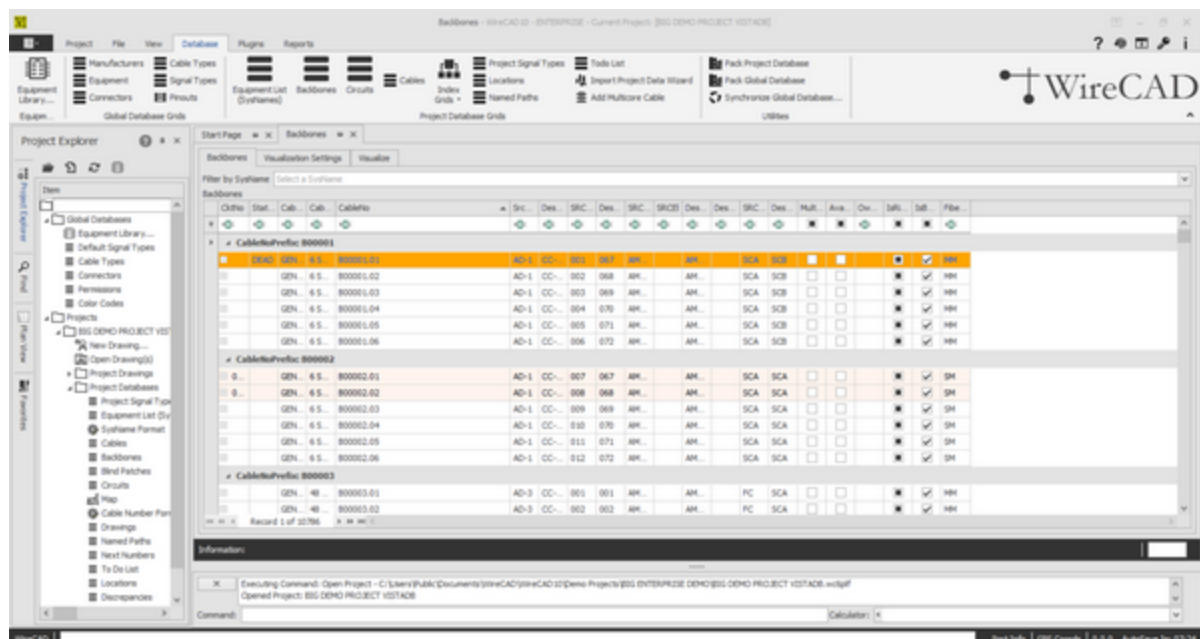
Product Level: **CMS**

The Backbone Grid will show you a list of all the backbone segments in your project and allow you to modify their status or delete the entire backbone cable group at once. The status will refer to whether the backbone is in use, proposed, dead ETC. New backbone status can be created in the project settings menu. There are 3 default status available; in use, proposed & dead.

The Backbone Grid is easier to use than the Cables Database if you are looking for basic information about a backbone such as number or origin and destination information or if you are simply wanting to change the status of the backbone.

To access The Backbone Grid, type **BBG** in the command line.

With The Backbones Grid open, a list of all created backbones and cores will show along numerous columns of information about each backbone core.

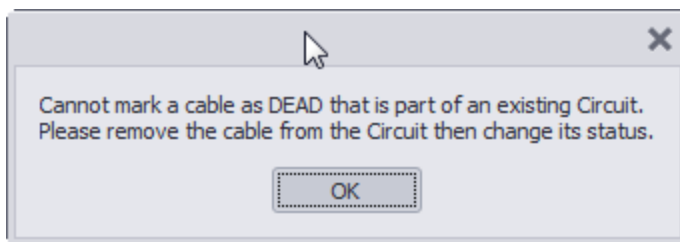
**Controls**

- **Copy Selection Down** - Create a selection vertically in the grid and click **Edit>Copy Selection Down** or **Ctrl+D** and the topmost cell's data will be copied to all selected cells below.
- **Delete Selected Backbone** - This function will delete ALL cables associated with the selected Backbone. If you are just trying to delete a single cable you should do that from the Cables databases.
- **Refresh** - Query the database and reload the data into the grid.
- **Attach Document** - Useful for storing field survey reports and other documents associated with this backbone. You can attach as many documents as you need. The documents are stored in the database. You can click on the link provided in the grid to launch the document into the system document reader for that file extension (MIME Type).
- **Rename Source Connector(s)** - Renames the Source Connector (SrcConn) field and any jumpers or horizontal cables attached to this backbone.

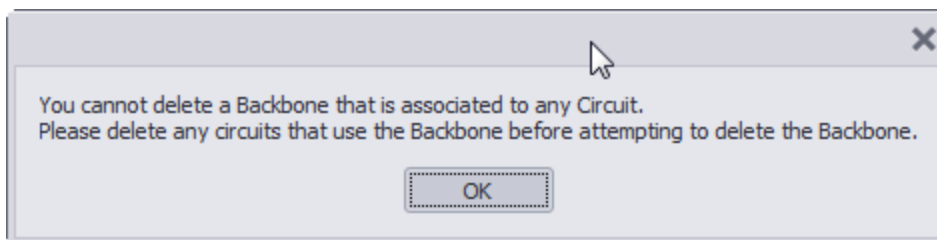
- Rename Destination Connector(s)** - Renames the Destination Connector (DestConn) field and any jumpers or horizontal cables attached to this backbone.
- Rename FiberMode** - Renames the Fiber Mode (FiberMode) field and any jumpers or horizontal cables attached to this backbone.

While a number of fields are shown in *The Backbone grid*, only the **Status** field & **Owner** field are modifiable. All other fields are simply shown for reference.

Note: When marking a backbone as a status that is blocking you must make sure that this cable is no longer used in an active circuit. If WireCAD detects that this backbone is still in use, you will be presented with a warning message stating "Cannot mark a cable as <Blocking Status Name> that is part of an existing Circuit. Please remove the Cable from the Circuit, then change its status. "



Note: You will also receive a warning message if you try to delete a cable that is part of an existing circuit. You must remove the cable/backbone from the circuit before deleting. If you wish to delete individual cores, you can do so from inside the cables database however this information will not persist back into the CAD drawings.



Note: When deleting a backbone in *The Backbone Grid*, the entire group of cables will be deleted even if you have only selected a single core. Make sure that you want to delete the entire backbone segment before selecting delete.

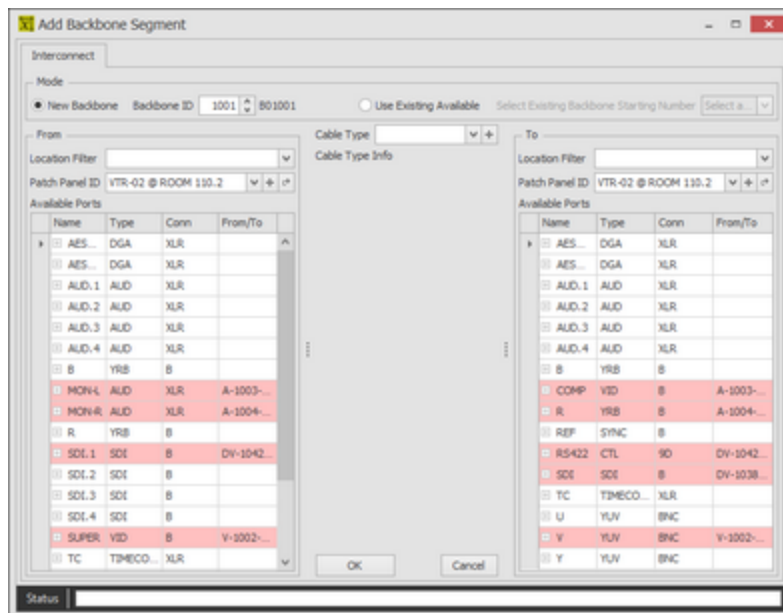
Note: Deleting backbones in *The Backbone grid*, does not adjust number sequence. If you wish to reuse deleted backbone numbers, you will need to manually access the *Next Numbers Grid* and change the next available number to the one you would like to use. Example, you have created backbones 1001,1002,1003 & 1004. The next number in the sequence would be 1005. If you then delete 1002,1003 & 1004, the Add Backbone Segment Tool will label the next backbone as 1005. To reuse 1002, 1003 & 1004, you need to manually select these as next in the sequence by doing so in the [Next Numbers](#) 249 Grid.

4.2.1.1.1 How To: Draw a Backbone

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Backbones](#) [210] > [The Backbone Grid](#) [211] >



1. Open a plan view drawing where you want to draw a plan view polyline.
2. Place source and destination Plan View Equipment or source and destination Plan View Location Boundaries.
3. Click **Plan View and Layout Tools > Draw Backbone**.
4. As prompted select the source equipment or location boundary.
5. Draw the points in the polyline.
6. The **New Backbone** tool is shown.



7. Select the source, destination, cable type and ports.
8. Click **[OK]** to finish.

4.2.1.2 How To: Create a New Backbone

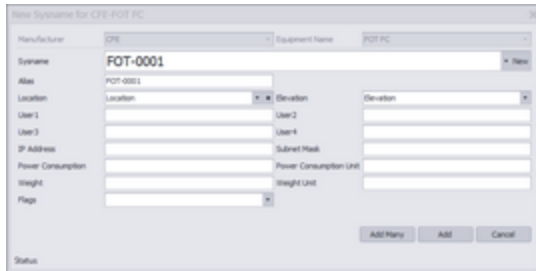
Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Backbones](#) [210] >



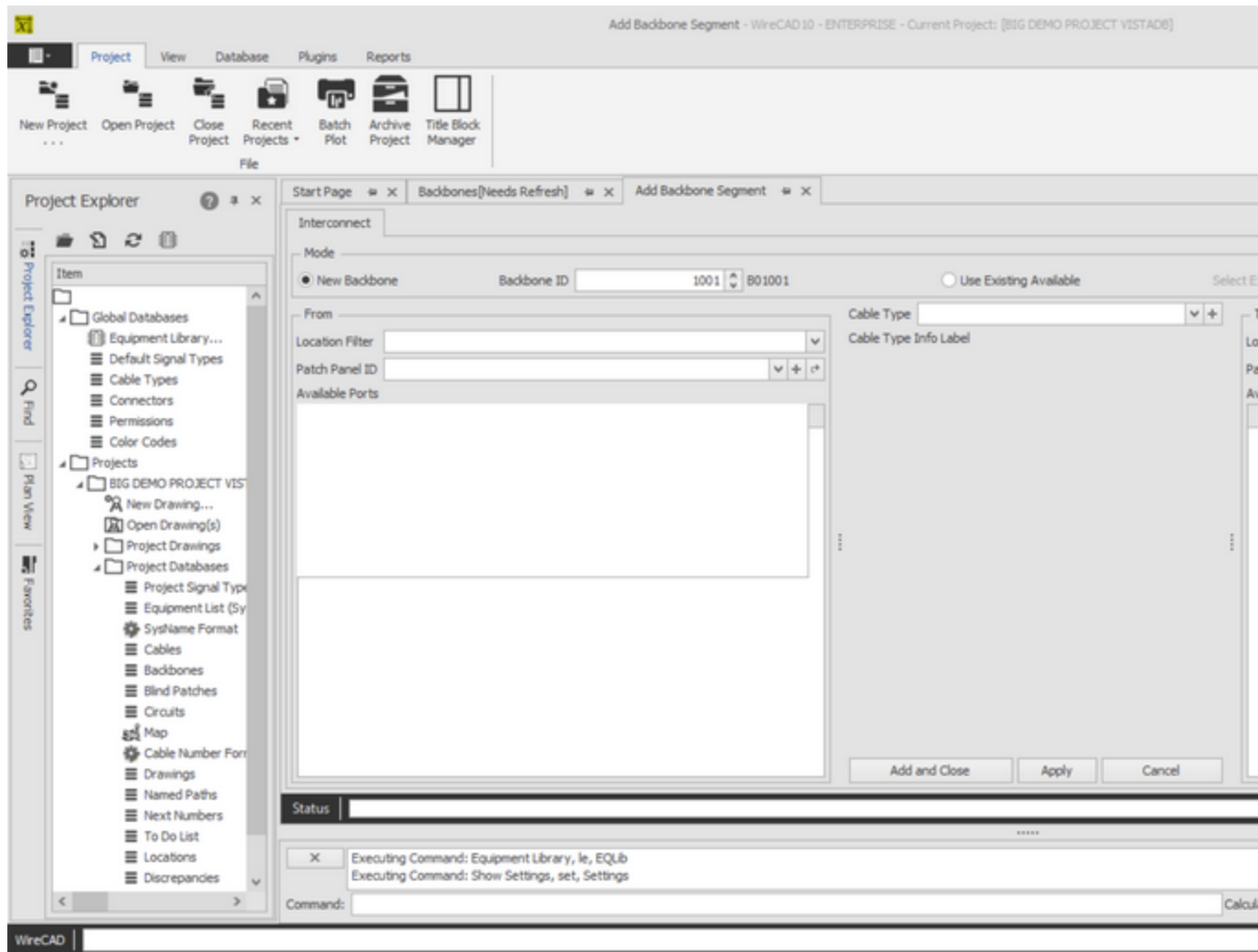
The following procedure details the creation of backbones in the CMS module. Backbones are multi-core cables in the cables database that interconnect two (or more) patch panels.

Note: While this tool is open in your window other WireCAD users will be locked out of the Cables table of the Project database.

1. Make sure you have used the steps above to create your Equipment, Customize your I/O and create your Cable Types.
2. Open the [Equipment Library](#) ⁴²⁸.
3. Find the equipment you would like to use.
4. Click **[Add to Project Database Only]**. You can add as many pieces of equipment you as you would like at this time however you will need to have at least the 2 definitions that the **Backbone** are linked to.



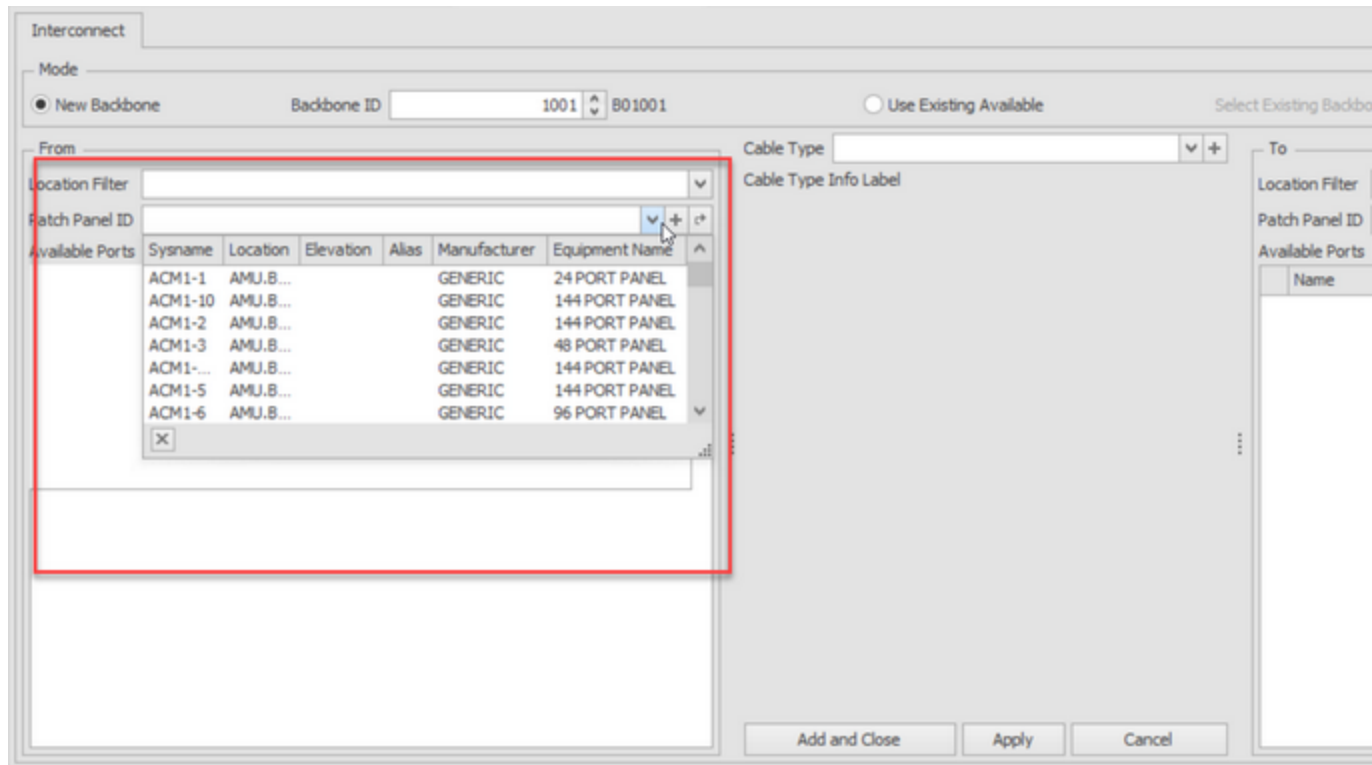
5. If there are multiple pieces of the same equipment in this location, you can select the **[Add Many]** to create multiple pieces of equipment at the same time. Note: WireCAD will automatically SysName each piece of equipment using the next available number in the sequence.
6. Enter in your location for this equipment.
7. Click **[Add]** or **[Add Many]**.
8. Once you have added your equipment into the Project Database, close your [Equipment Library](#) ⁴²⁸.
9. Enter ABS into the command line and press enter or click on the **[Add Backbone]** button on the homepage.
10. The **Add Backbone Segment** window should now be open.



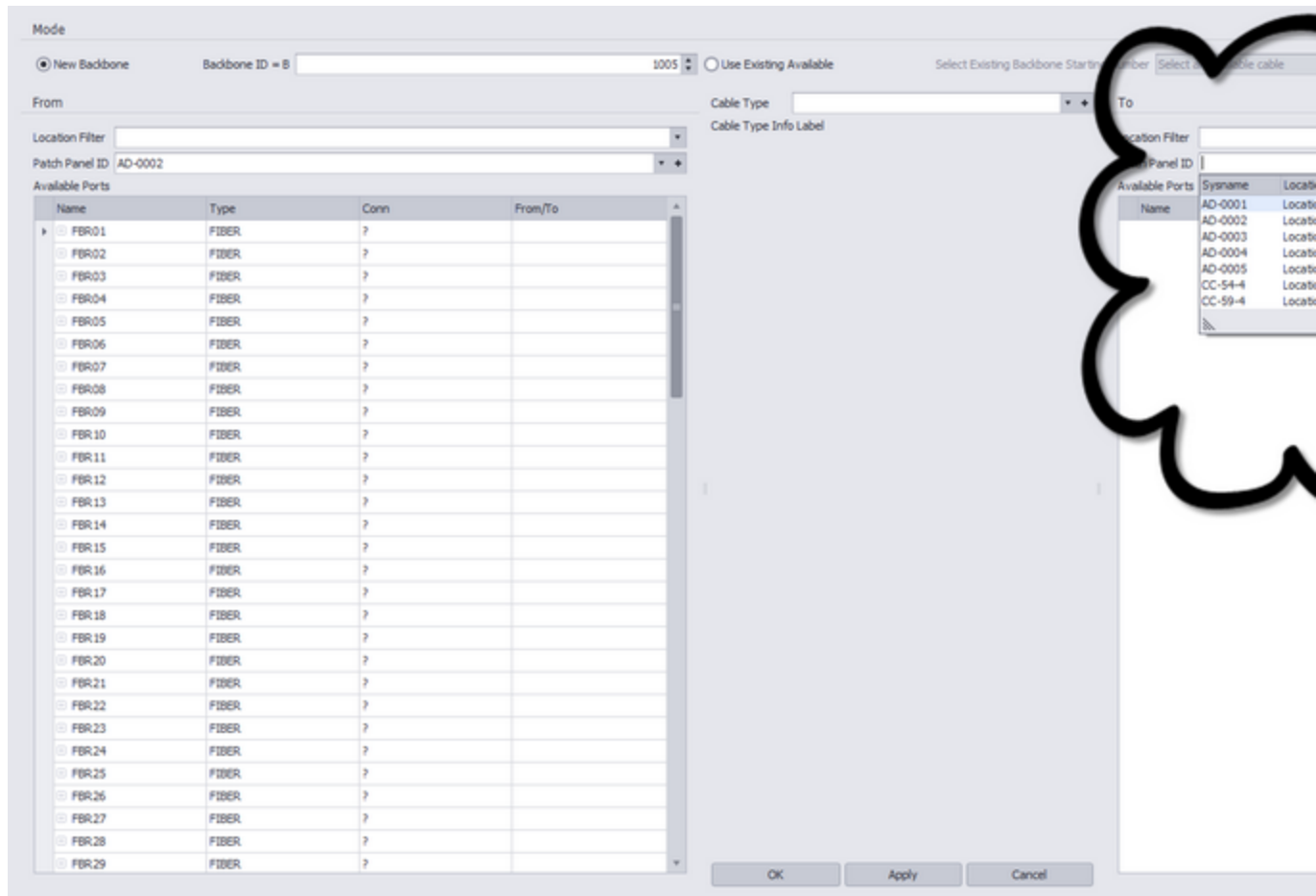
11. First select if you would like to create a **New Backbone** or **Use Existing Available**.



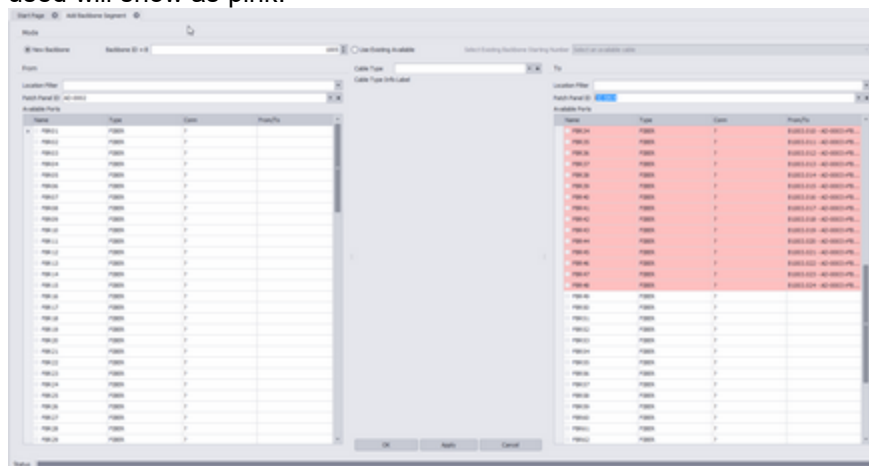
12. On the **From** side, select your equipment from the **Patch Panel ID** field.



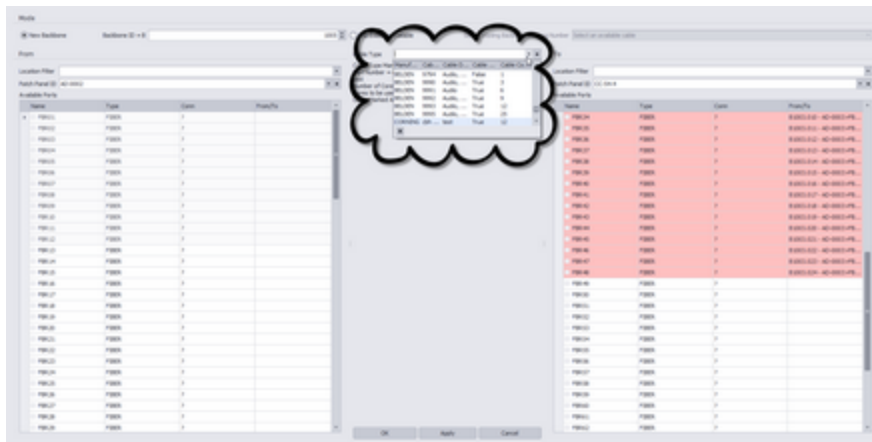
13. On the **To** side, select your equipment from the **Patch Panel ID** field. Again, you can filter your results using the **Location Filter** Field.



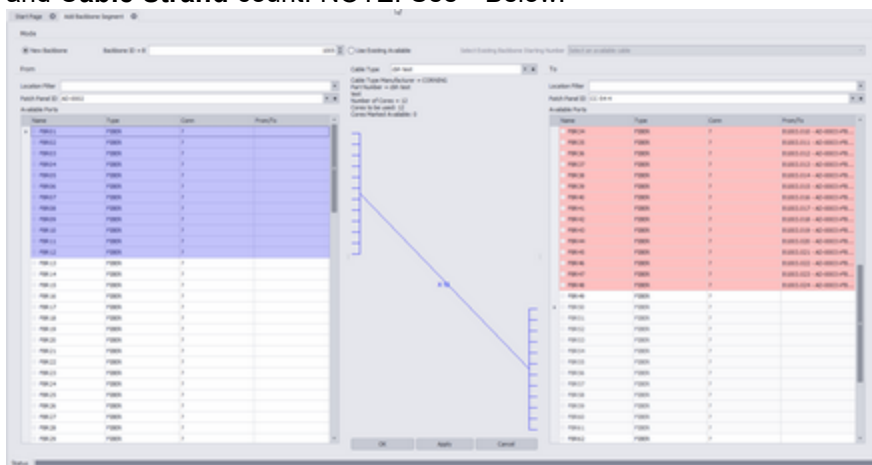
14. On both sides, ports that are available will be shown in white while ports that are currently being used will show as pink.



15. From the **Cable Type** field, select the **Cable Type** you would like to use as your backbone. If you have not already created a **Cable Type** for your backbone, you can do so by clicking the **[+]** sign and following the instructions [here](#).



16. Select the ports you would like to use on the **From** side, then select a starting port on the **To** side. You will see that WireCAD will automatically draw a backbone based on your port selection and **Cable Strand** count. NOTE: See * Below!



17. Click **[Apply]**. WireCAD will create this backbone and then revert to an empty page ready for another backbone assignment.

*Note: Always make sure that you are not overlaying on used ports and that the ports coming in are not the same as the ports going out. Example: If you have a Patch Panel with 24 Ports in and 24 Ports out, you could use ports 1-12 as your incoming ports and 13-24 as your output ports. This allows the other 12 ports on each side to be used for jumpers to other equipment while maintaining a backbone infrastructure.

4.2.1.3 Backbone Visualization Settings

Navigation: [WireCAD CMS Tools Manual](#) ^[208] > [CMS Basics](#) ^[209] > [Backbones](#) ^[210] >



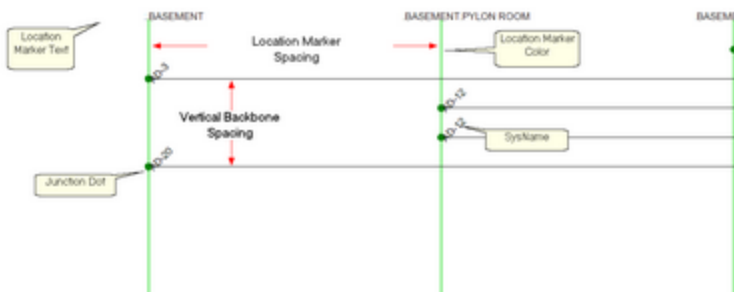
These settings are available on the Backbones Grid [Visualization Settings] tab.

In order to preview Backbones you will first need to [Add Backbones](#) ^[214]

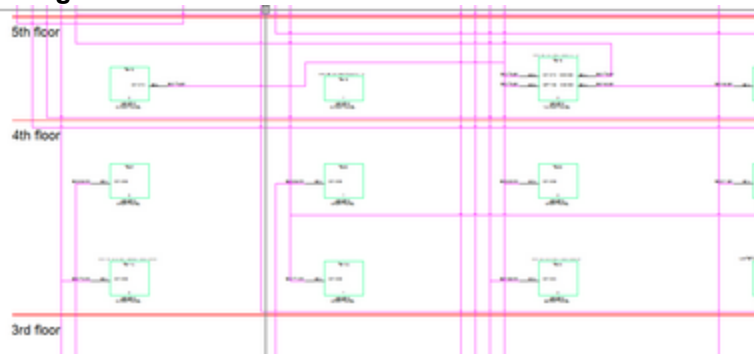
These settings determine the display of the Backbones Visualize tool.

There are three variants:

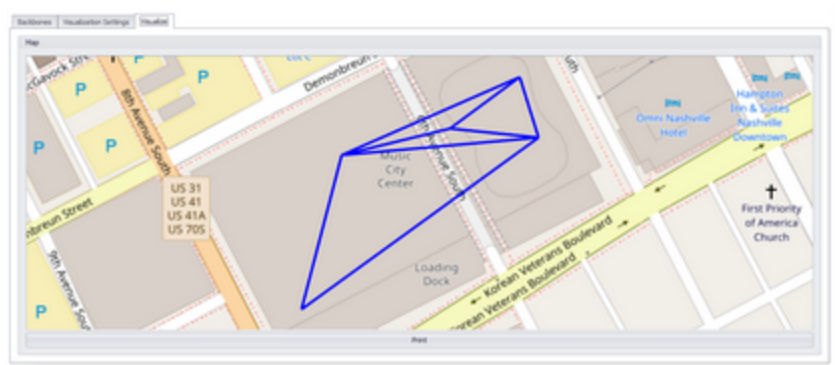
1. Layered Digraph (ladder diagram).



2. Riser Diagram



3. Map

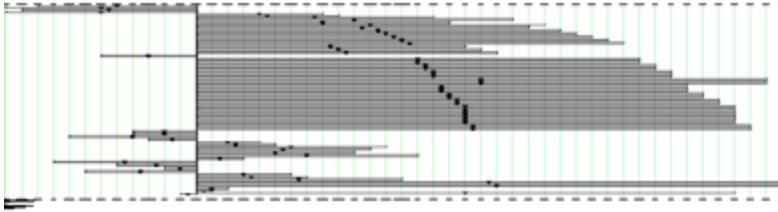


Map Tool - within the Backbones Visualizer

This mode is only viable if you have associated GPS coordinate data in your Locations table. For more information [see this topic](#) ^[266]. In Map view the backbones returned by the Filter Settings are shown on the Map tool. The backbones honor the Backbones Color, and Line Width [settings](#) ^[370]. The Map may be printed or exported through the print preview dialog.

Layered Digraph Mode Data Source

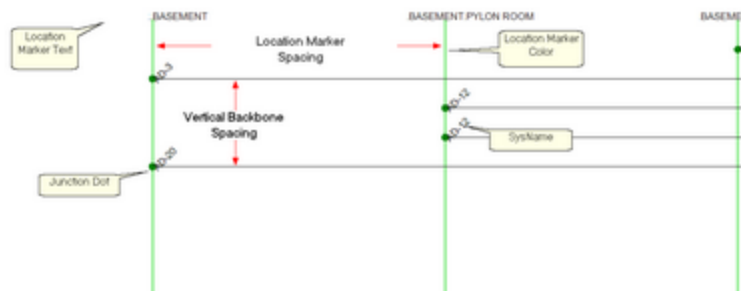
- **Show All** - Shows all backbones in the database in the Visualize tool.
- **Current Record** - Allows selection of only those backbones that touch the selected record.



- **Search Depth** - How many branches deep are we going to search.
- **Search Width** - How many backbones per branch.

Appearance

- **Location Marker Spacing, Location Marker Text Height, Location Marker Color, Junction Dot Radius, Vertical Backbone Spacing(DU).**



- **Center Label Format** - String used to create the center label. Can make use of the following variables:

{0} = Backbone Number
 {1} = Total Count
 {2} = Total Available Count
 {3} = Total Dead Count
 {4} = Single Mode Fiber Count
 {5} = Available Single Mode Fiber Count
 {6} = Dead Single Mode Fiber Count
 {7} = Multimode Fiber Count
 {8} = Available Multimode Fiber Count
 {9} = Dead Multimode Fiber Count

Example:

Assume that our backbone number is 1001 with 12 single mode fibers of which 1 is dead and four are in use.

String:

B{0}-SM COUNT:{4} Avail:{5} Dead:{6}

Output:

B1001-SM COUNT:12 Avail:8 Dead:1

String:

B{0}

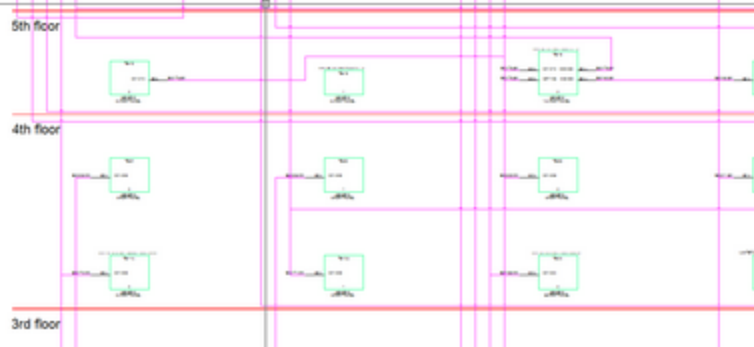
Output:

B1001

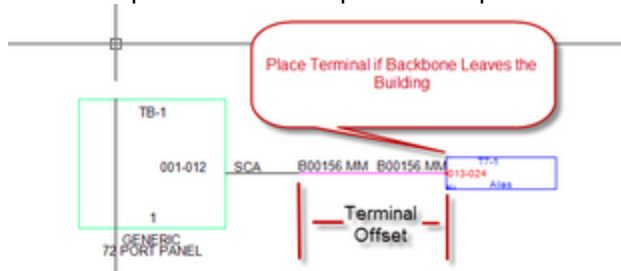
- **Center Label Offset** - Offset from center in DU.
- **Show SysNames** - Shows the SysName labels.
- **Ignore Same Locations** - Hides backbones that originate and terminate in the same location.
- **SysName Rotation Angle** - Sets the rotation angle of the SysName label if shown.
- **SysName Text Height (100th DU)** - Sets the height of the SysName text if shown.
- **SysName Text Offset** - Offset from the endpoint of the backbone.

Riser Diagram Mode

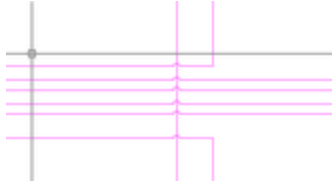
- **Campus and Building** - The Campus and Building for which to build the riser diagram.
- **Layout Max Columns, Column Spacing, Minimum Row Height** - Determine the layout of the panels in the diagram.



- **Body Color, Body Width, Descriptor Locations** - Determines the appearance of the body of the panels in the diagram.
- **Show Terminals if Backbone Leaves the Building, Terminal Offset** - If the other end of the Backbone is not in the selected building a terminal will be placed and a backbone drawn to it. The terminal's position from the port on the panel is determined by the **Terminal Offset** property.



- **Apply Jumps -**



- **Avoid Other Cables** - Instructs the cable autorouter to attempt to avoid other cables.
- **Show Unresolved Backbones** - If the Backbone cannot be placed in the drawing a list is generated. Enabling this setting shows the list as the function completes.

Map Mode

- **Location Filter** - Filter the displayed backbones by location.
- **Panels Filter** - This filter relies on the Locations Filter and provides further filtering by SysName.
- **Backbones Filter** - This filter relies on the Locations Filter and the Panels Filter and provides further filtering by Backbone number.

The map relies on the Locations table GPS fields. If you have not assigned GPS Lat and Lon to the Location the backbones map will not display any data.

To Assign GPS Lat and Lon you can use the Map tool or your GPS calibrated drawing.

Title and Comment Block

- **Show Title** - Sets the visibility of the title/comment block.



MISC.

- **Backbone Color by Signal Type** - Pulls the backbone color from the global Signal Types database.
- **Show Directional Coloring** - Shows green dots for the source end of the backbone and red dots for the destination end if shown.
- **Backbone Color** - Sets all backbones to the color defined.
- **[Reset Default]** - Button to reset the settings to the defaults.

4.2.1.4 Backbone Preview Export

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Backbones](#) [210] >



Exporting the CAD preview of the Backbones Visualization tool is as easy as clicking the **[Export]** button.

The current preview will be exported based on the Export Settings found in the [Application Menu > Settings \[Project\]\[Export Settings\]](#) [280] tool.

4.2.1.5 Import Backbone Data

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Backbones](#) [210] >



Menu: **Database > Backbones**

Default command line shortcut: **bbg**

Related Project Settings:

Backbone Status Items

We have provided an import template that can be found at:

c:\users\public\documents\WireCAD\WireCAD9\Import Templates\import_backbone_template.xlsx.

Command:

Open the Backbones Grid. Then click **File>Import Backbones**.

Prerequisites

You should have a Backbones import template excel file filled in with your backbone data.

Related Topics:

[Import Circuits](#)  245

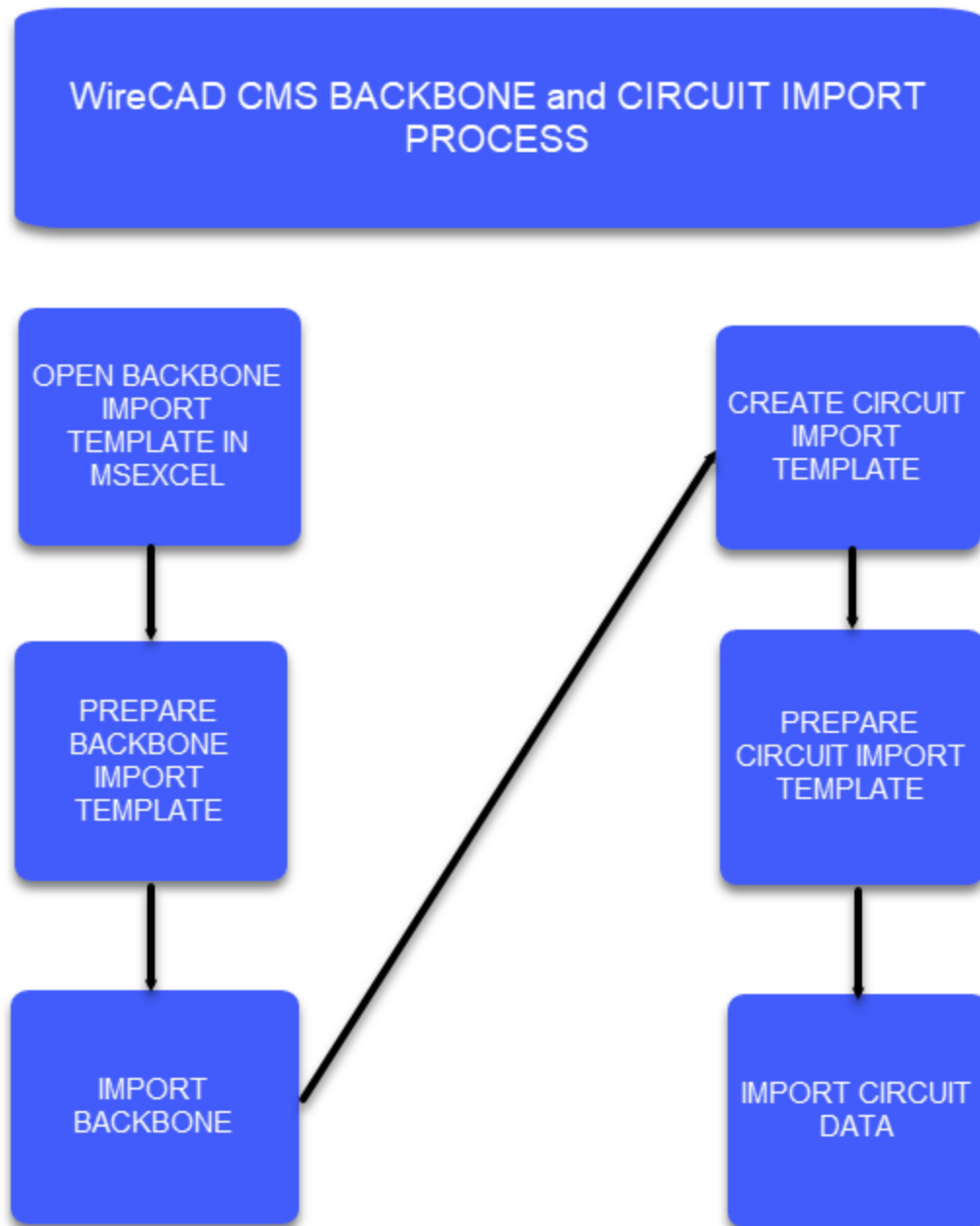
[Backbones Grid](#)  211

This document, when properly filled out will not only import a single line of data that expands to many cables database entries as a backbone but will do the following as well:

- Create global Cable Type and Manufacturer.
- Create global panel data and Manufacturer.
- Create Project SysName data.
- Create Project Location data.

The import tool will perform several checks on the incoming data to ensure that it does not conflict with existing data. Some of these checks include:

- Basic template data check to make sure all required fields are filled in.
- Preflight queries to check against the existing project data. At this point in the import no data has actually been imported. If any errors are found the template row number is returned with error text describing the problem and the import is stopped.
- Post-flight queries to ensure no duplicates were created during the import. If found the error is reported with the opportunity to roll back the import.



4.2.2 Blind Patch Tool

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] >



Menu: **Project Explorer > Project Databases > Blind Patches**

Default command line shortcut: **ShowBlindPatchTool**

Related Project Settings:

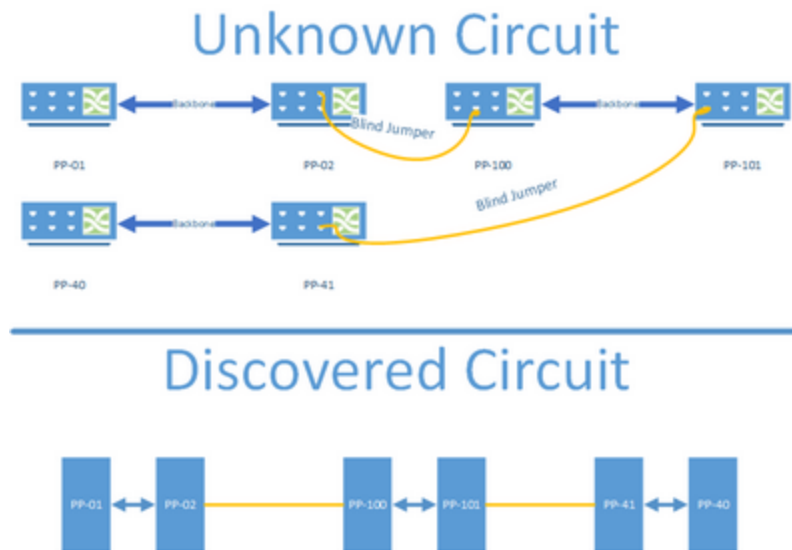
[Status Behavior CMS](#) [369]

Explanation

Unlike the normal WireCAD [New Circuit](#) [238] tool which requires that you know the complete circuit end-to-end, the **Blind Patch Tool** allows you to document your existing patches and then discover circuits from those patches.

You will use the **Blind Patch Tool** to document patches between panels. It is not necessary that you know the whole circuit. Simply create patches from panel to panel as you go. You can use the **[Discover]** tool to find circuits from the blind patches. Once the **Blind Patch Circuits** are discovered you can use the **[Make Circuit]** function to create real **Circuits**.

As you add **Blind Patches** the **Status** of the connected **Backbone(s)** will be mark **BLIND_PATCH**. This status item is a blocking status item. Meaning that other operations will not be able to use or delete this segment of the **Patch Panel** or **Backbone**.



Prerequisites:

[Backbones](#) [211]

Related Topics:

[Blind Patch Tool](#) [527] form.

[New Blind Patch](#) [502] dialog.

4.2.2.1 How To: Create a Blind Patch

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Blind Patch Tool](#) [226] >



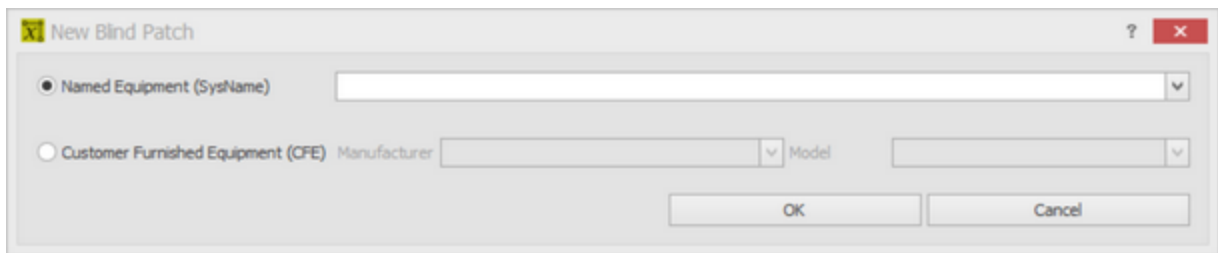
The follow procedure describes creating a Blind Patch. Blind Patches are connections between two panels. We only know the two ends of the patch. We will discover circuits after adding more Blind Patches.

Prerequisites:

At least two backbones.

Procedure:

1. Open the **Blind Patches** tool: Click **Project Explorer>Project Databases>Blind Patches**.
2. Select the Patch Panel (SysName) from the drop-down. This will populate the ToSys in the grid.
3. Select the row in the grid representing the port that you want to patch to on the ToSys.
4. Select either **Simplex** or **Duplex Jumper**. Duplex will require two rows and that the row below the selected row be empty to receive data as well.
5. Click **Blind Patch>New**.



6. Select either **Named Equipment (SysName)** or new **CFE**. Click OK.
7. The new **Blind Patch** is added. **Backbones** that touch the **Blind Patch** will have their **Status** field set to **Blind_Patch**.
8. Done.

4.2.2.2 How To: Discover Circuits From Blind Patches

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Blind Patch Tool](#) [226] >



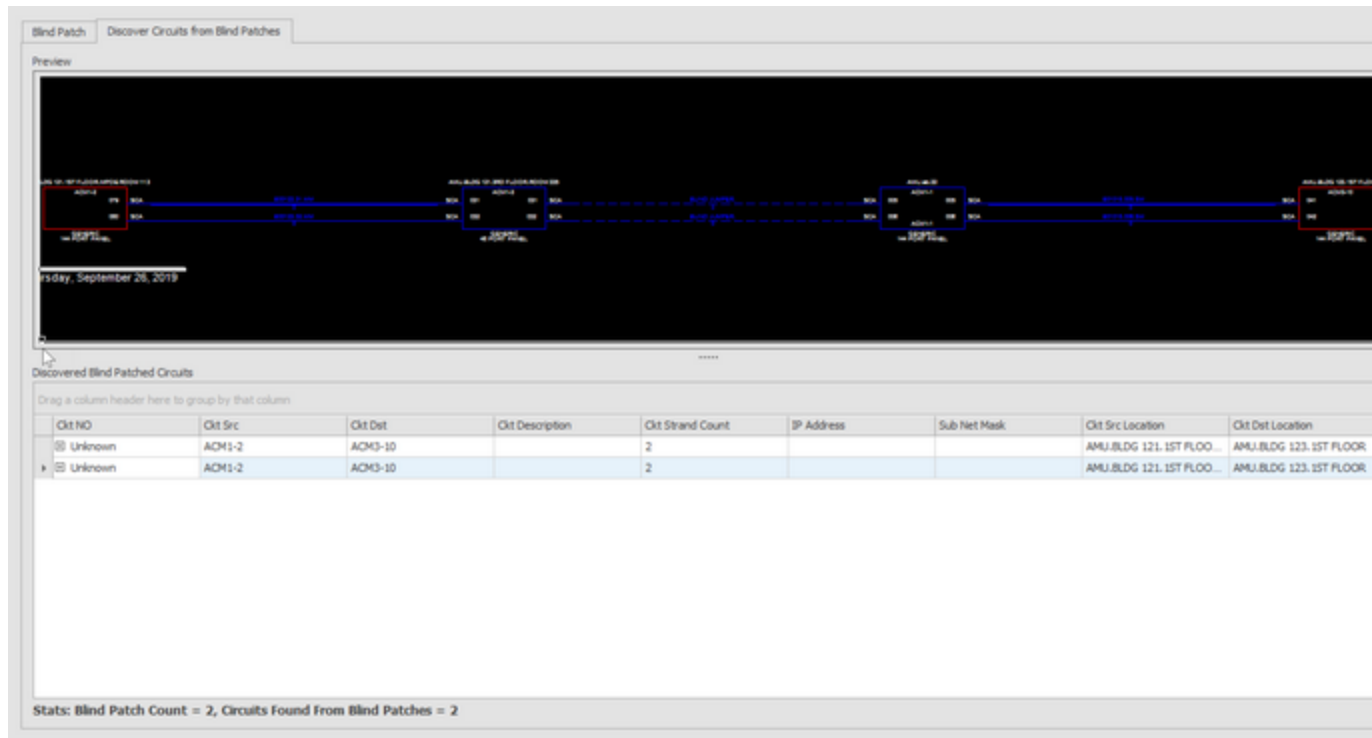
The following procedure describes discovering circuits from Blind Patches.

Prerequisites:

More than one **Blind Patch**.

Procedure:

1. Open the **Blind Patches** tool: Click **Project Explorer>Project Databases>Blind Patches**.
2. Switch to the **Discover Circuits from Blind Patches** tab.
3. Click **[Discover Circuits]**.
4. The grid will be populated.
5. Select a row and the found circuit will be presented in the **Preview** pane (if the **[Preview Selected]** check item is checked).



4.2.2.3 How To: Make a Real Circuit From Blind Patches

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Blind Patch Tool](#) [226] >



The following procedure describes making real circuits from discovered circuits.

Prerequisites:

Blind Patches

Procedure:

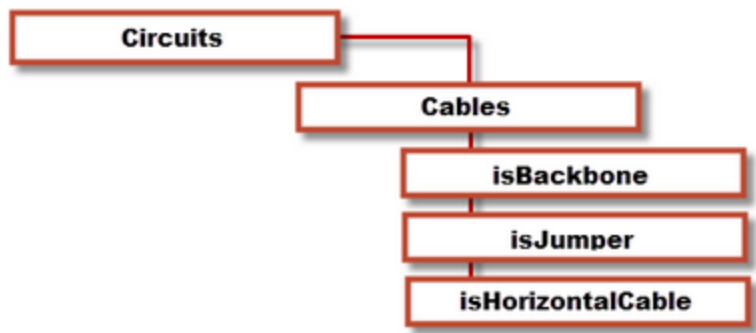
1. Follow the steps above to **Discover Circuits**.
2. Select the circuit in the grid that you wish to make real.
3. Edit the **CktNo** and **CktDescription** fields.
4. Click **[Make Circuit]**.
5. The circuit will be added to the **Circuits** table and removed from the **Blind Patches** list.
6. Done.

4.2.3 Circuits

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] >



A circuit in WireCAD is defined as a collection of cables. Each circuit may have a Name, Description, and other meta data. A circuit also has a strand count to indicate the number of connections made from the originating device on through to the terminating device. Circuits have a one-to-many relationship with the cables in the Cables database. Each circuit may have many cables but a cable can belong to only one circuit.



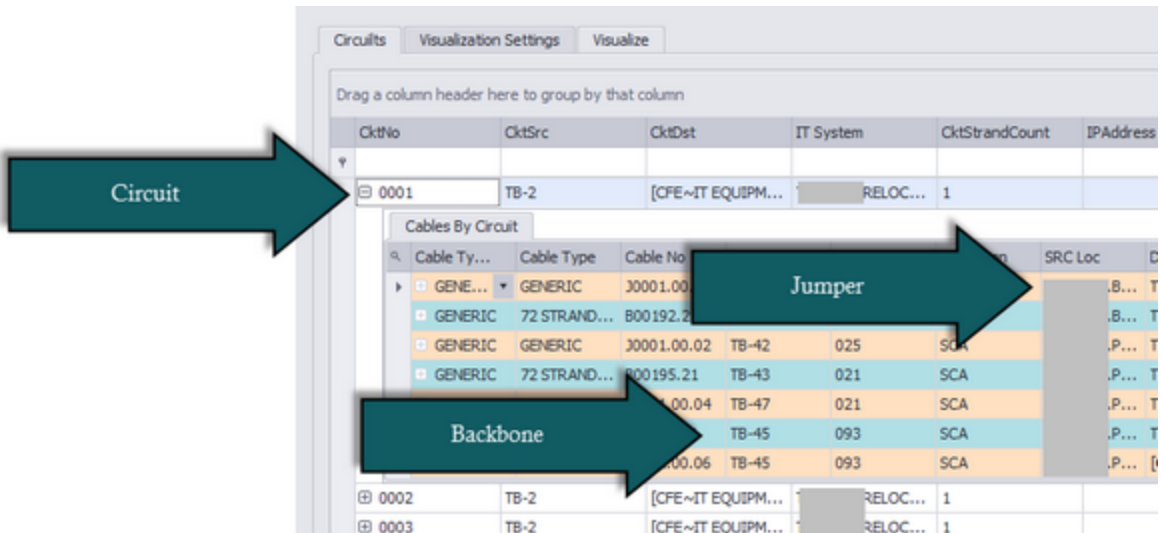
4.2.3.1 Circuits Grid

Navigation: [WireCAD CMS Tools Manual](#) | 208 | > [CMS Basics](#) | 209 | > [Circuits](#) | 228 | >



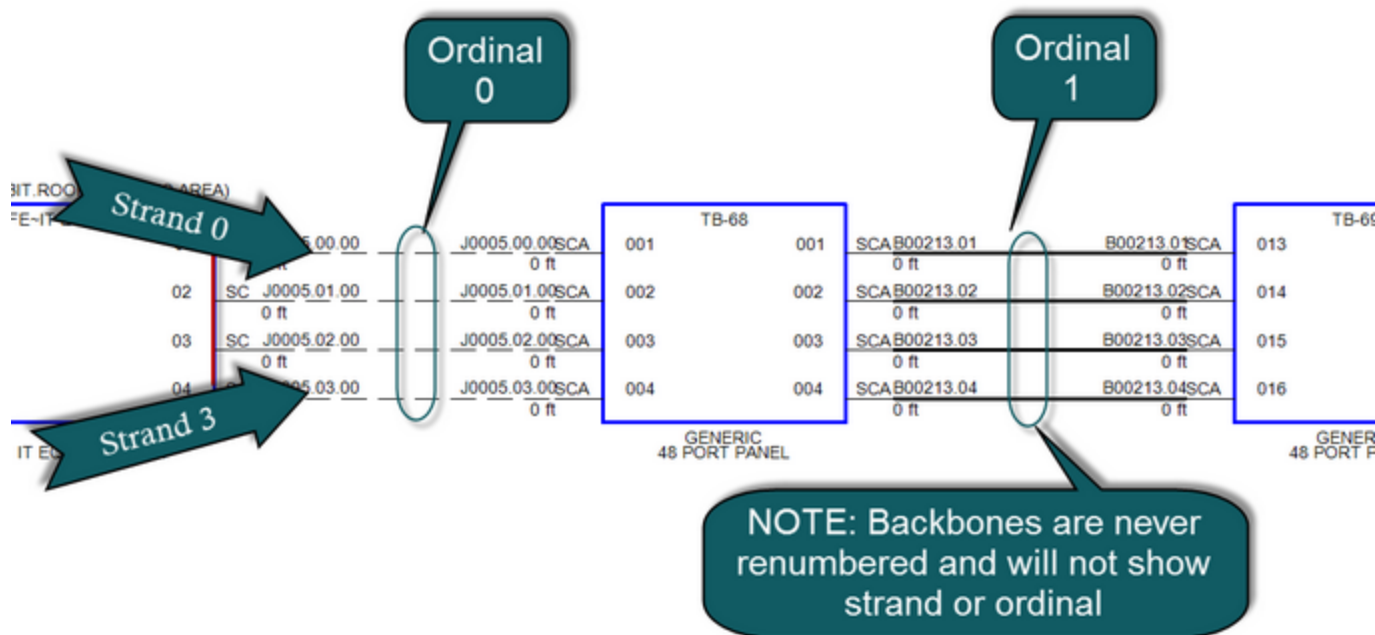
Menu: **Database > Circuits**
Default command line shortcut: **cmscg**
Related Project Settings:
Circuit Status Items
Product Level: **CMS**

The **Circuits Grid** presents the overall circuit data in list form with the child cable data attached for view.



- **Attach Document** - Useful for storing field survey reports and other documents associated with this circuit. You can attach as many documents as you need. The documents are stored in the database. You can click on the link provided in the grid to launch the document into the system document reader for that file extension (MIME Type).
- **Rename Circuit** - Renames a circuit and all of its cables.
- **Combine Circuit** - See the topic [Combining Circuits](#) | 243 |.

Strand Count and Ordinal Explanation



WireCAD uses the strand and ordinal to know the exact position of a cable in a circuit. Strand 0 will always be the topmost cable in the circuit with Strand n being the bottommost cable in a circuit. Ordinal 0 will be the leftmost set of cables in the circuit with Ordinal n being the rightmost set of cables in the circuit.

4.2.3.2 Circuits Preview Settings

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Circuits](#) [228] >



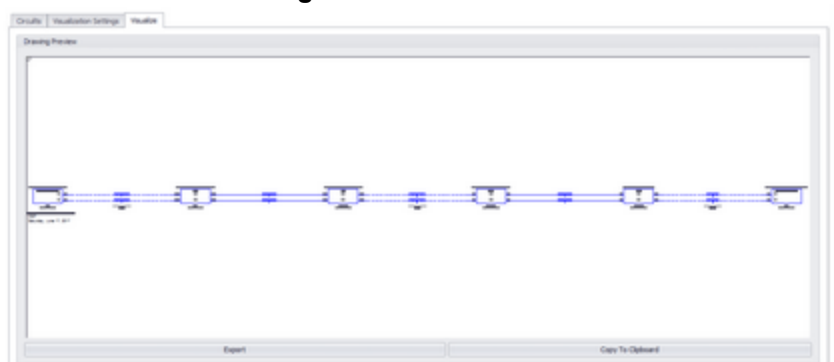
These settings are available on the Circuits Grid [Visualization Settings] tab.

In order to preview Circuits you will first need to [Add Circuits](#) [237]

These settings determine the display of the Circuits Visualization tool.

There are two variants:

1. Functional Block Diagram.



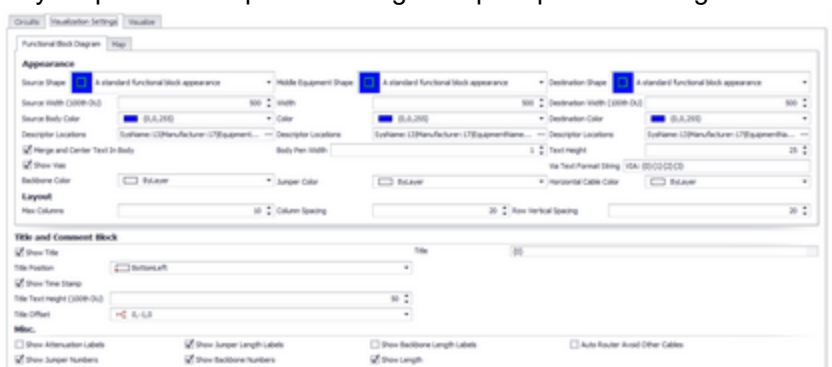
Displays a CAD representation of the circuit as a functional block diagram. The master source and destination are shown with a series of equipment panels, jumpers, and backbones.

2. Map View.



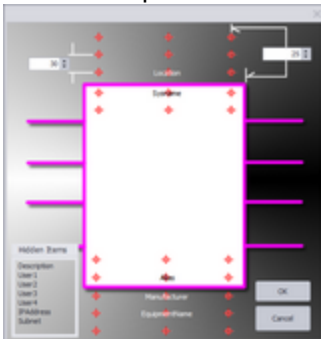
Map Tool - within the Circuit Visualizer

This mode is only viable if you have associated GPS coordinate data in your Locations table. For more information [see this topic](#) ²⁶⁶. In Map view the backbones of the selected circuit(s) are shown on the Map tool. The backbones honor the Backbones Color, and Line Width [settings](#) ³⁷⁰. The Map may be printed or exported through the print preview dialog.



Appearance

- **Source, Middle and Destination Shape** - Choose one of the 16 stock shapes.
- **Source, Middle and Destination shape Width in (100th DU)** - How wide is it.
- **Source, Middle and Destination shape Color** - Set the color of the shape.
- **Source, Middle and Destination shape Descriptor Locations** - Drag the descriptor to the location map or to the **Hidden Items** list to hide.



- **Backbone Color** - Sets the color.
- **Jumper Color** - Sets the color.

- **Horizontal Cable Color** - Sets the color.
- **Merge and Center Text in Body**



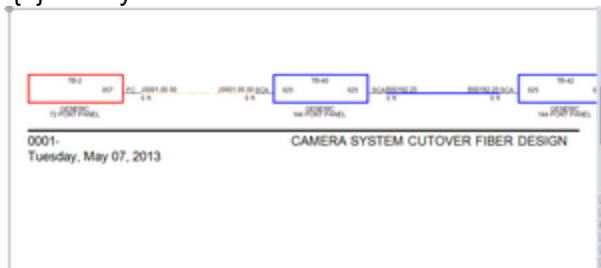
- **Text Height** - Sets the height of all text in the visualization.

Layout

- **Max Columns** - The maximum number of blocks placed before a new row is started below.
- **Column Spacing** - The distance between blocks.
- **Row Spacing** - The Distance between rows.

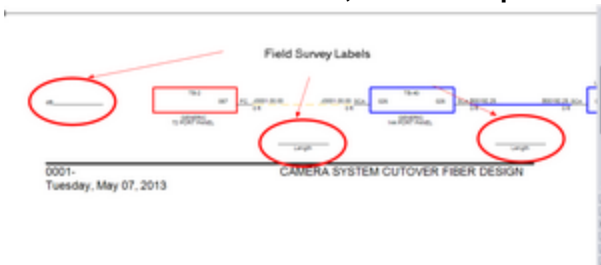
Title and Comment Block

- **Show Title** - Sets the visibility of the title/comment block.
- **Title** - You can type whatever you want in this field. In addition the following variables are available:
 {0} = Circuit Number
 {1} = It System

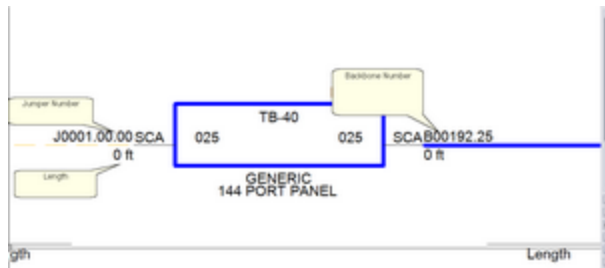


MISC.

- **Show Attenuation Labels, Show Jumper Length Labels, Show Backbone Length Labels**



- **Show Jumper Numbers, Show Backbone Numbers, Show Length**



4.2.3.3 Outputting Circuits to CAD

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Circuits](#) [228] >



Once a circuit has been defined you can create CAD views of single and multiple circuits. The tools to do this are contained in the **Circuits Grid** which you can access from the command line shortcut **CMSCG** or by selecting the **[CAD Preview/Circuits Grid]** on the home page.

Before accessing the **Circuits Grid**, please make sure to setup your export preferences in the settings menu.

Please refer [here](#) [277] for more information on the settings menu.

You can also batch output the selected records by clicking **Tools>Output Selected**.

All export is governed by settings in the **Application Menu > Settings[Project][Export Settings]** tool.



•**[Export]** - Output to file based on the Export Settings found in **Application Menu > Settings [Project][Export Settings]**

•**[Copy to Clipboard]** - Copies the preview to the clipboard so it can be pasted into another drawing.

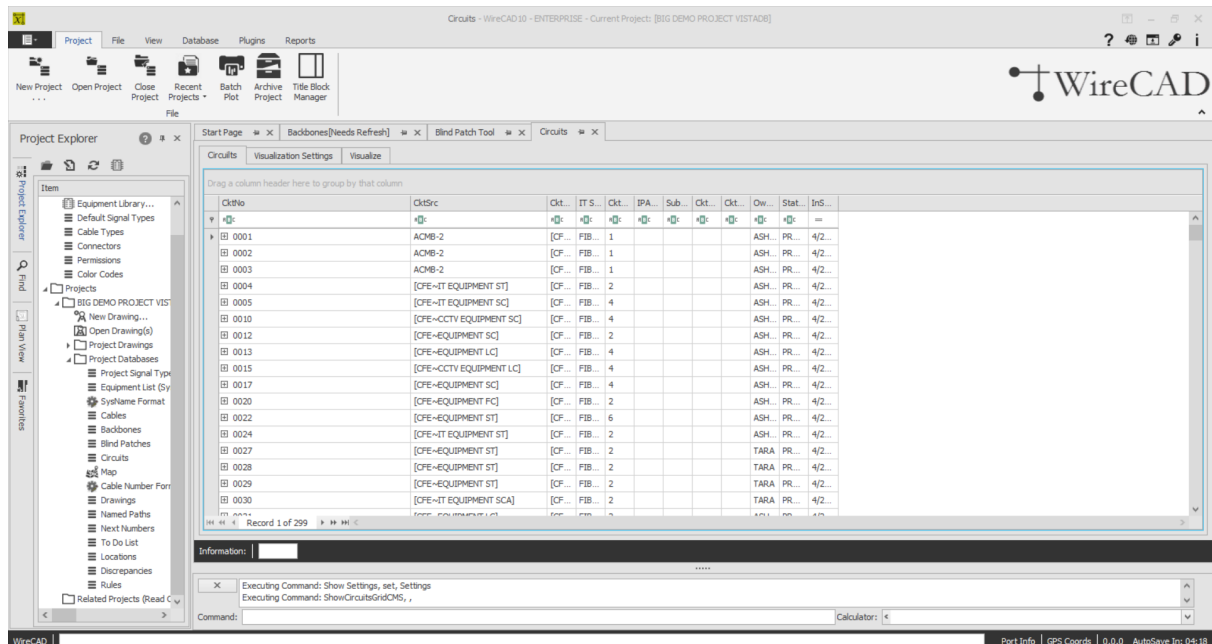
4.2.3.4 How To: Output a Circuit to CAD

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Circuits](#) [228] >



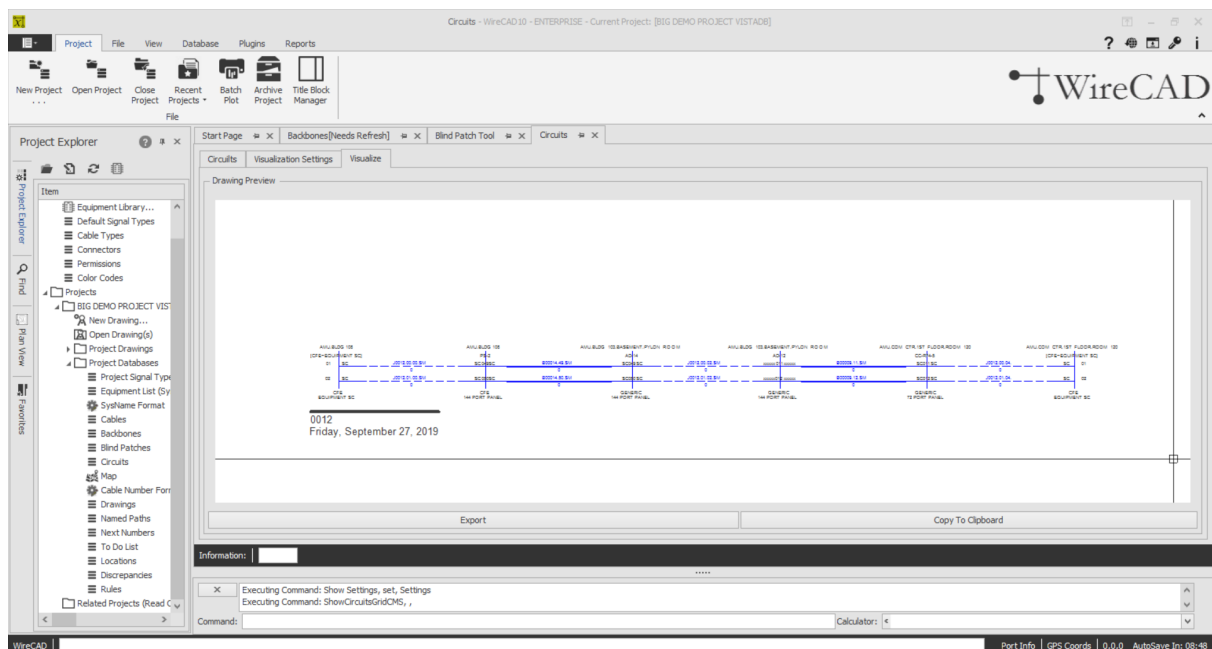
This step by step will guide you on exporting a circuit.

1. Enter **ShowCircuitsGridCMS** in the command line prompt. The **Circuit Grid** Window will open.



2. Select the Circuit Number you would like to Export.

3. Select the **[Visualize]** Tab.



4. Verify the circuit is the one you would like to use and click **[Export]**. A Windows explorer window will come up. Enter the path to export to, what you would like this file to be called and what file type you would like to export as. Note: All of these settings can be automated in the **Settings Menu**.

5. Click **[Save]**. Your exported file will now be created and placed in the location you have specified of the file type you specified in the **Save As Dialog filter**.

4.2.3.5 How To: Output Many Circuits to CAD

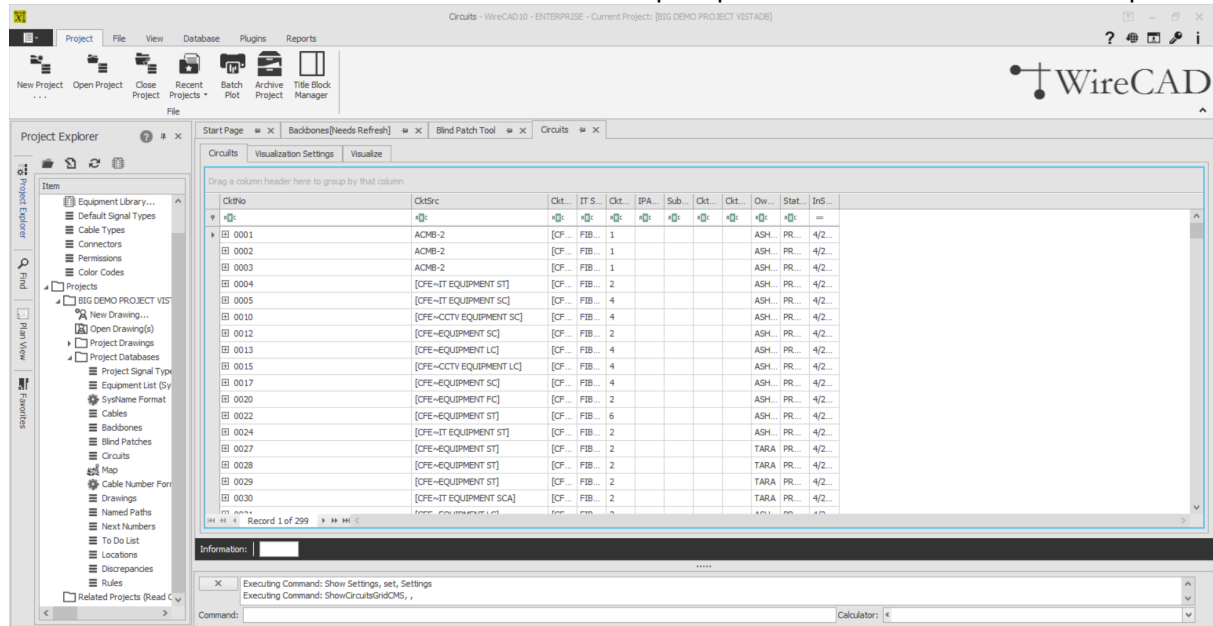
Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Circuits](#) [228] >



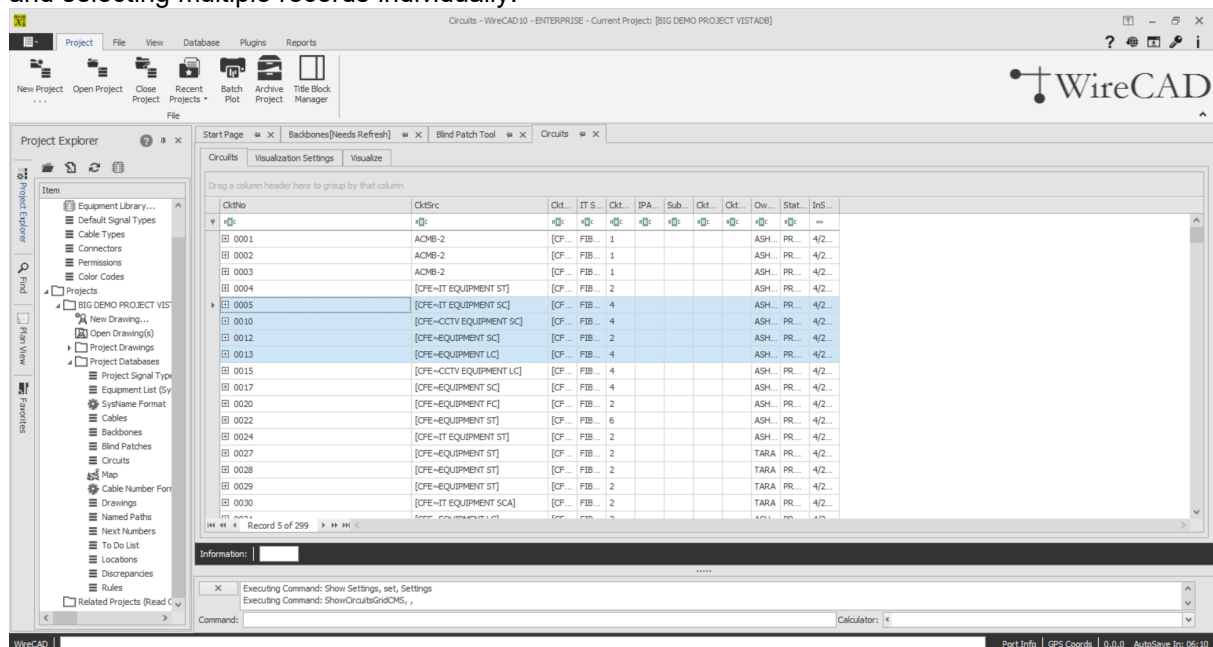
This Step by step will guide you on exporting multiple circuits.

Note: this is different from the command **Tools>Output All Selected Circuits**

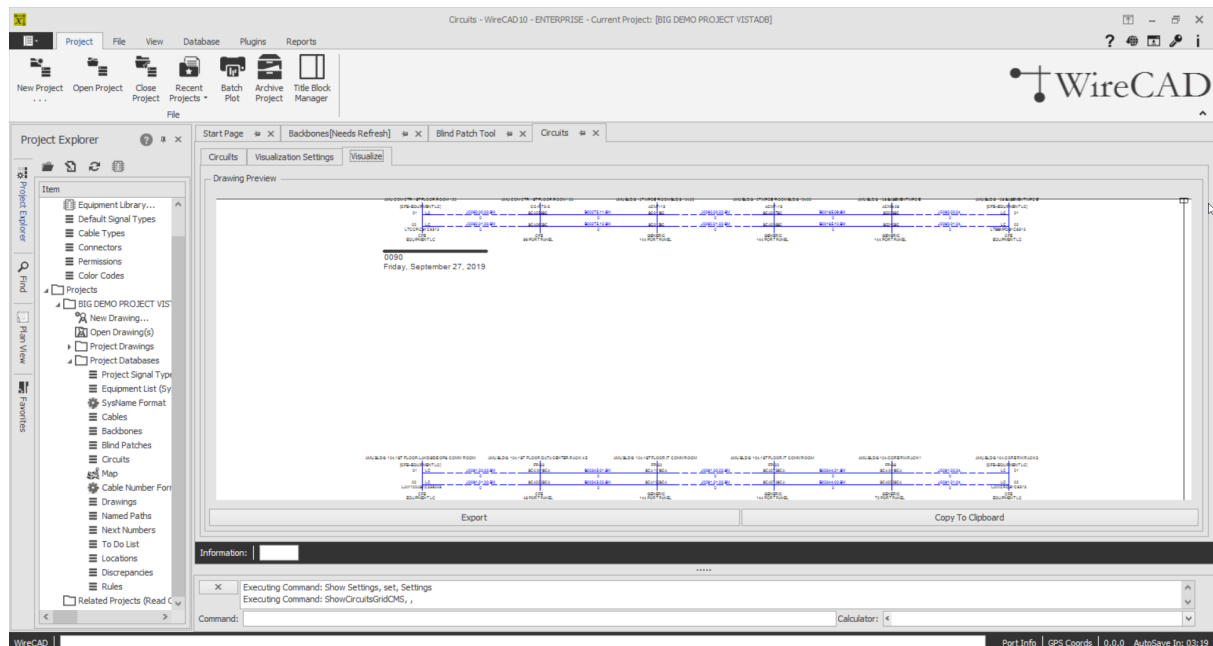
1. Enter **ShowCircuitsGridCMS** in the command line prompt. The Circuit Grid Window will open.



2. Select the Circuit Numbers you would like to Export. You can select multiple circuits by dragging down with your mouse clicked, by holding **[Shift]** and selecting a set or by holding down **[CTRL]** and selecting multiple records individually.



3. Select the **[Visualize]** Tab.



4. Verify the circuits shown are the ones you would like to use and click **[Export]**. A Windows explorer window will come up. Enter the path to export to, what you would like this file to be called and what file type you would like to export as. Note: All of these settings can be automated in the **Settings Menu**.

5. Click **[Save]**. Your exported file will now be created and placed in the location you have specified of the file type you specified in the **Save As Dialog filter**.

4.2.3.6 New Circuit Tool

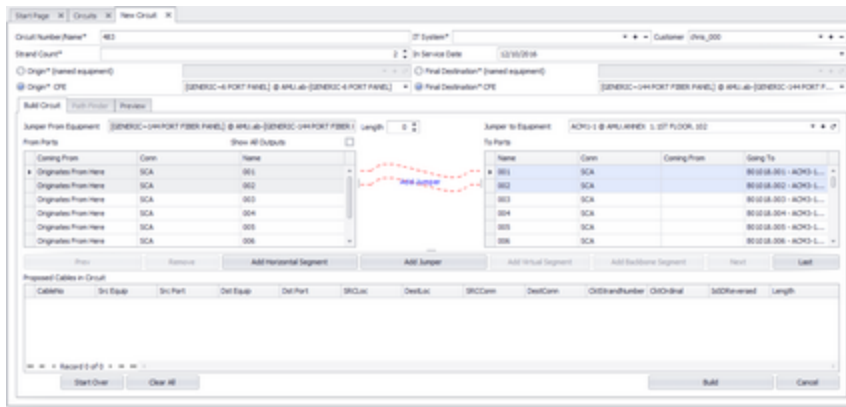
Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Circuits](#) [228] >



The New Circuit tool is used to connect *Field End & Head End* Equipment together by adding jumpers between existing Backbones. Each circuit is comprised of descriptive data, such as **Name**, **Customer**, **IT System** or **Description**, **source** and **destination equipment** (SysNames); as well as a collection of cables in the Cable database. You simply select the ports that you wish to jumper from/to and add the jumpers/back bone segments to the **Proposed Cables list**. A **Path Finder** tool helps you search out possible routes. A **[Preview]** window presents the functional block view of the **Proposed Cables**. Once you are happy with the circuit, simply click **[Build]** and all Proposed cables will be added and associated to the named circuit.

To open the New Circuit tool open the Circuits grid and click **File>New**
Command line shortcut: **nc**

Note: While this tool is open in your window other WireCAD users will be locked out of the Cables table of the Project database.



4.2.3.6.1 How To: Create a New Circuit

Navigation: [WireCAD CMS Tools Manual](#) ^[208] > [CMS Basics](#) ^[209] > [Circuits](#) ^[228] > [New Circuit Tool](#) ^[238] >



Menu: **Database > Circuits**

Default command line shortcut: **cmscg**

Related Project Settings:

Circuit Status Items

Product Level: **CMS**

Click **File > New**.

Before you Start

The use of this tool assumes the following:

- That you have added Backbones to the project.
- That you have added the necessary Cable Types to the global [Cable Types](#) ^[523].
- That you have an idea of what you want to connect together and that you know the first bit of infrastructure to which you will attach.
- That you have created your Customer Furnished Equipment(CFE) in the [Equipment Library](#) ^[428] for the originating and final destination equipment devices and/or that SysNames have been created for each individual piece of equipment to which you will attach.

1. Verify that you have created all your Equipment and Cable Types needed for this circuit.
2. From the Circuits Grid click **[File>New]** to launch the New Circuit tool.
3. Verify that your Circuit **Number/Name** is correct and that it is not duplicating an existing entry.

Start Page | Circuits | New Circuit*

Circuit Number/Name* 483 IT System* Customer chris_000

Strand Count* 2 In Service Date 12/30/2016

Origin* (named equipment) First Enter the Originating Equipment Final Destination* (named equipment)

Origin* CPE [GENER3C-144 PORT FIBER PANEL] @ AMU.ab-[GENER3C-6 PORT PANEL] Final Destination* CPE [GENER3C-144 PORT FIBER PANEL] @ AMU.ab-[GENER3C-144 PORT FIBER I...

Build Circuit Path Finder Preview

Jumper From Equipment [GENER3C-144 PORT FIBER PANEL] @ AMU.ab-[GENER3C-144 PORT FIBER I Length 0

Jumper to Equipment ACM1-1 @ AMU.ANNEX 1. 1ST FLOOR. 102

From Ports Show All Outputs

Coming From	Conn	Name
Originates From Here	SCA	001
Originates From Here	SCA	002
Originates From Here	SCA	003
Originates From Here	SCA	004
Originates From Here	SCA	005
Originates From Here	SCA	006

Prev Remove Add Horizontal Segment Add Jumper Add Virtual Segment Add Backbone Segment Next Last

Proposed Cables in Circuit

Cable#	Src Equip	Src Port	Dest Equip	Dest Port	SrcLoc	DestLoc	SrcConn	DestConn	CktStrandNumber	CktOrdinal	IsDRReversed	Length
3483.00.00	[GENER3C-6 P...	001	ACM1-1	001	AMU.ab	AMU.ANNEX 1...	SCA	SCA	0	0	<input type="checkbox"/>	0
3483.01.00	[GENER3C-6 P...	002	ACM1-1	002	AMU.ab	AMU.ANNEX 1...	SCA	SCA	1	0	<input type="checkbox"/>	0

Record 1 of 2

Start Over Clear All Build Cancel

4. Enter your **IT System** and **Customer** info. IT System is a descriptive name.

5. Select your **Origin** (named equipment) from the drop down menu on the left. Alternately, select the **Origin CFE** (Customer Furnished Equipment) and fill in the detail data in the dropdown.

Origin* (named equipment) Origin* CFE

Build Circuit Path Finder Preview

Jumper From Equipment

From Ports

Coming From Conn

Customer Furnished Equipment is equipment that you connect to that is not tracked in the equipment list. CFE equipment must exist in the Equipment Library with its ports defined so we can make use of them. Simply select the equipment, name and locate it.

Manufacturer Model/PN/Name

Alias

Location

OK

6. Select your **Final Destination System** (named equipment) from the drop down menu on the right. Alternately, select the **Final Destination CFE** (Customer Furnished Equipment) and fill in the detail data in the dropdown.

Final Destination* (named equipment) Final Destination* CFE

Build Circuit Path Finder Preview

Jumper From Equipment

From Ports

Coming From Conn

Customer Furnished Equipment is equipment that you connect to that is not tracked in the equipment list. CFE equipment must exist in the Equipment Library with its ports defined so we can make use of them. Simply select the equipment, name and locate it.

Manufacturer Model/PN/Name

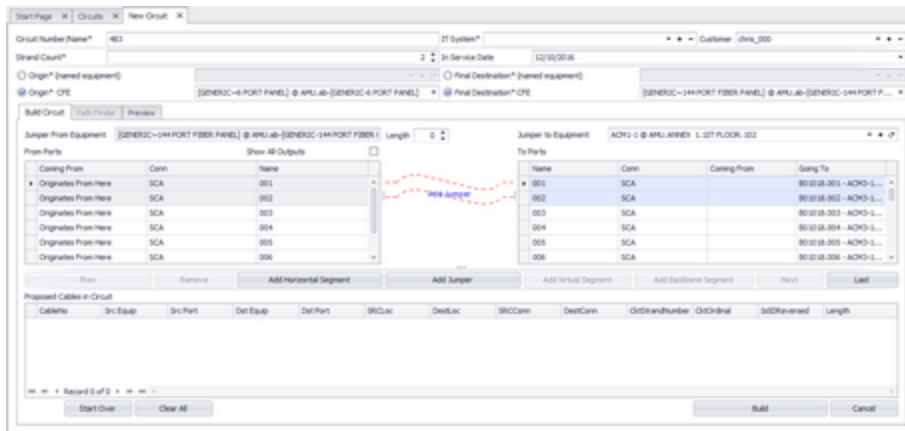
Alias

Location

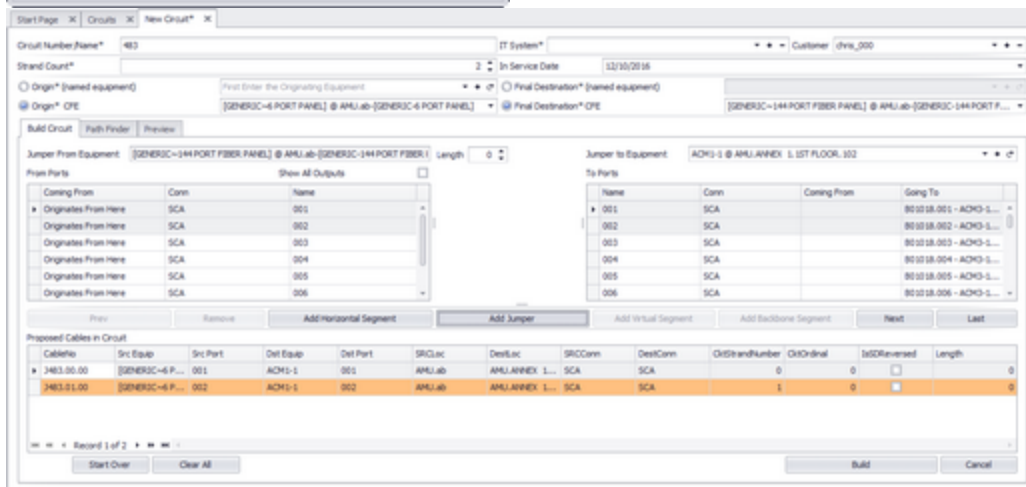
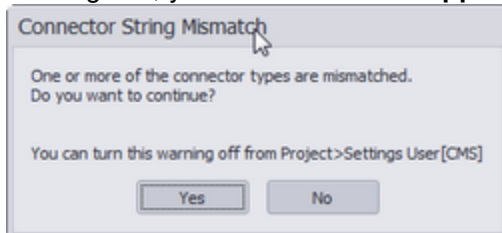
OK

7. Select your **Jumper To Equipment**. This will be the first Patch Panel to which your **Origin** system will connect.

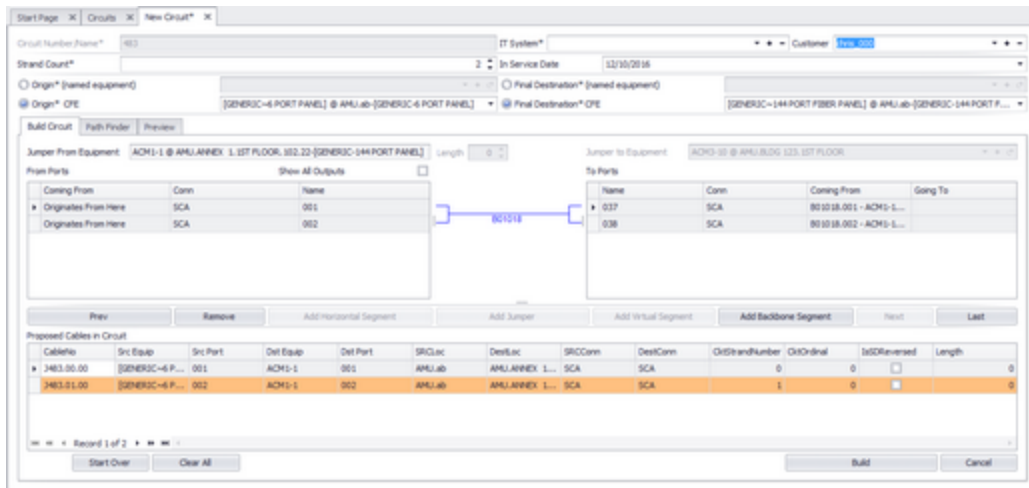
8. Select the ports from your **Origin** equipment and the **Ports** on your **Jumper To** equipment. You will see that WireCAD creates a preview jumper between the 2 for you to verify correct connections.



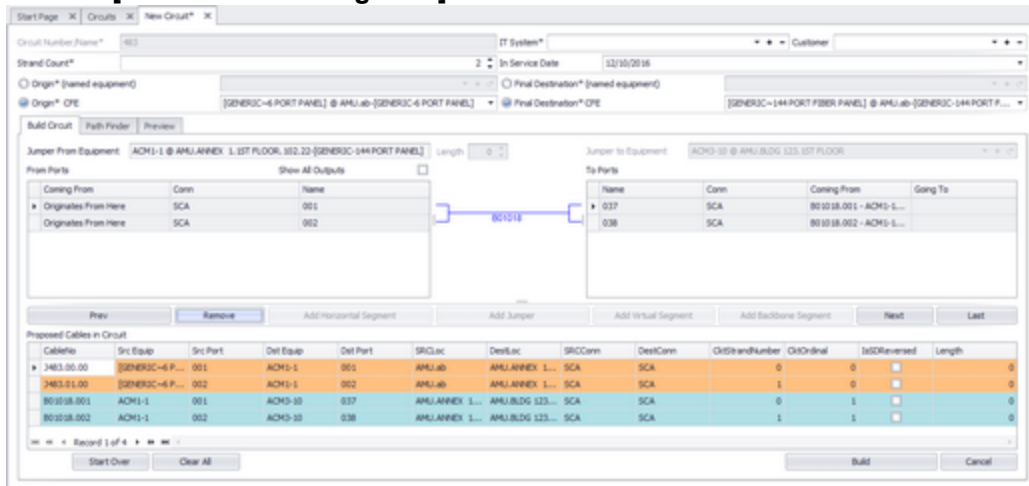
9. Click **[Add Jumper]**. Note: If you have connector types that WireCAD believes do not match, you will be presented with a window stating "Connector String Mismatch". If you want to turn this message off, you can do so in the **Application Menu > Settings>UserCMS**.



10. Click the **[Next>]** button. Note: WireCAD will automatically populate an available route between Patch Panels using existing Backbone structures.



11. Click **[Add Backbone Segment]**.



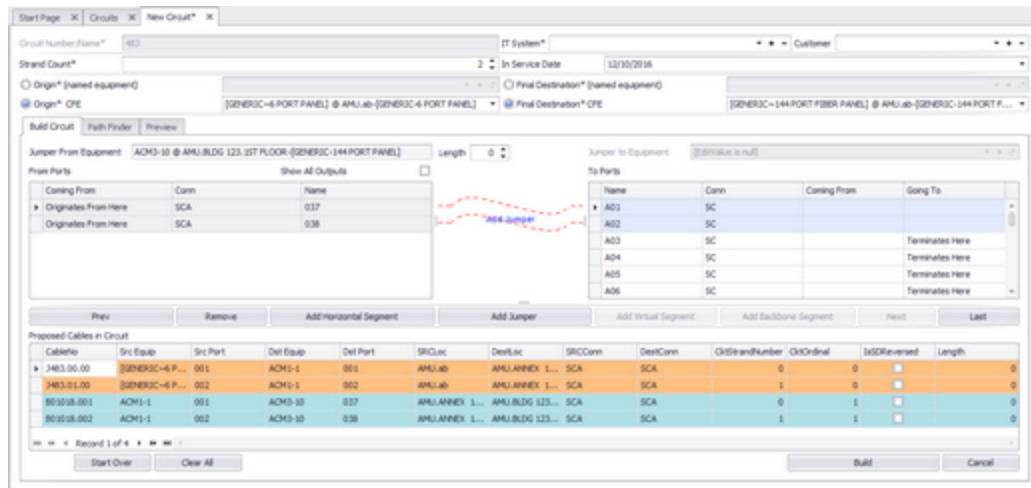
12. Verify the correct cables have been added to the **Proposed Cables In Circuit** list.

13. Then click **[Next]**.

14. Select the **Jumper To** Equipment to populate the **To Ports** list. Alternately you can click the **[Last]** button to populate the **To Ports** list with the ports of the **Final Destination Equipment**. That's what we will do here.

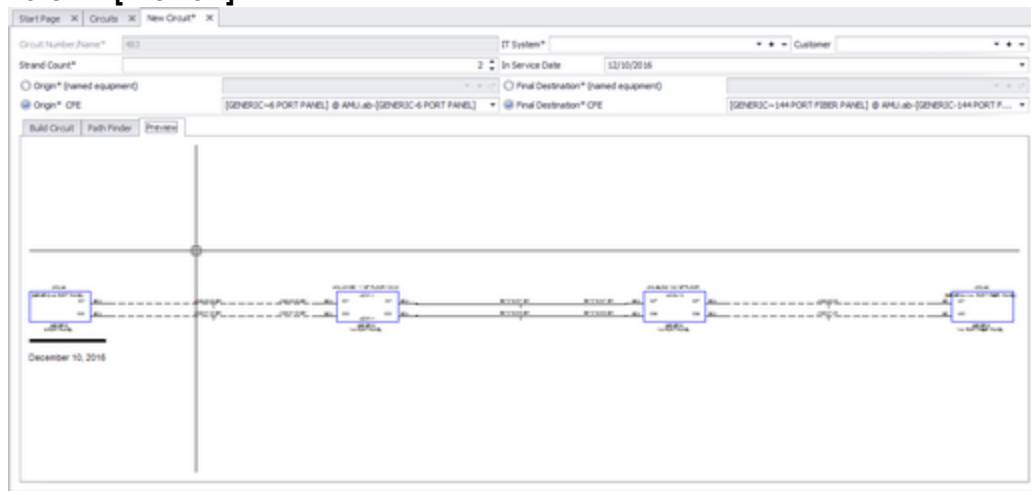
15. Click **[Last]**.

16. Select the top two records.



17. Click **[Add Jumper]**. This places the final jumper in the list.

18. Click **[Preview]**.



19. If everything looks correct, Click **Build Circuit** tab.

20. Click **[Build Circuit]**.

21. Done.

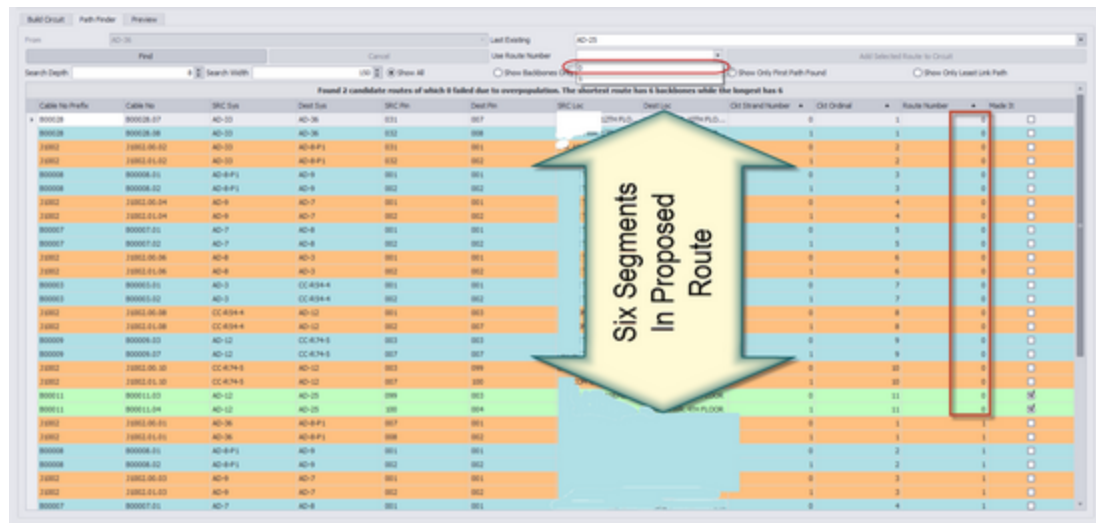
4.2.3.6.2 Path Finder Tab

Navigation: [WireCAD CMS Tools Manual](#)^[208] > [CMS Basics](#)^[209] > [Circuits](#)^[228] > [New Circuit Tool](#)^[236] >

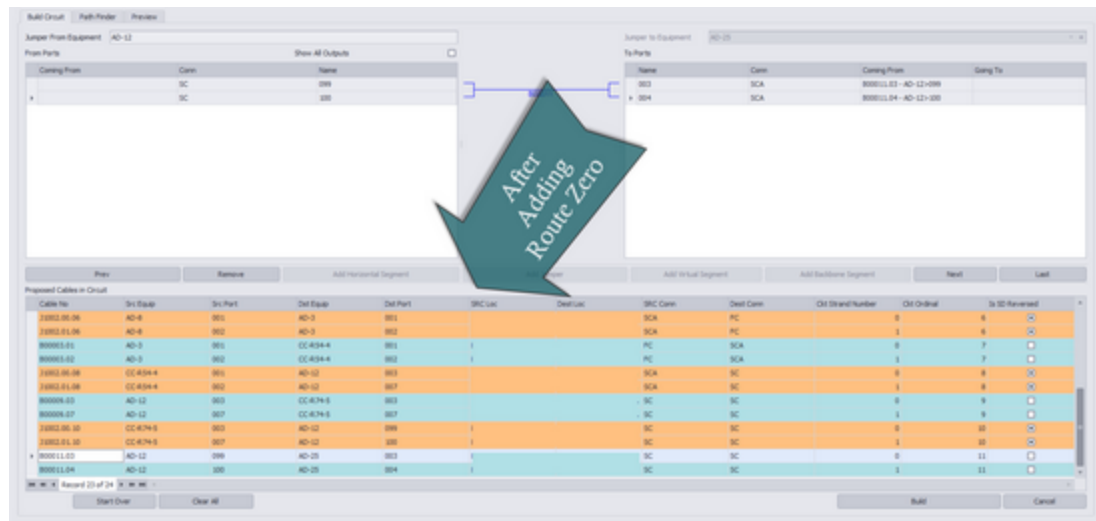


Explanation:

The **Path Finder** tool can be used to find available ports and backbone segments from a source to a destination. The **Path Finder** tool will search the backbone structure for backbones that can be used to get from the source panel to the destination panel, then once a route is established the route candidate is checked for available port count. If all criteria are met the route is returned. If no routes can be established you will be notified.



2 possible routes returned each with 6 segments. We will use Route 0.



After adding Route 0

Controls

- **From** - Prepopulated by your selection with the last jumper added in the Build tab.
- **Last Existing** - Select the last existing piece of infrastructure that we will search for.
- **[Find]** - Start the search.
- **[Cancel]** - Cancel the search.
- **Use Route Number** - Once routes are returned for selection this dropdown will be populated with the available routes.
- **[Add Selected Route to Circuit]** - Once you have selected a route this button will enable for you to add the route to the Proposed Cables to Add to Circuit list on the [Build] tab.
- **Search Depth** - How deep to search more will take longer but find more routes.
- **Search Width** - Maximum candidate routes to search.
- **Show All** - Show all routes found and populated with jumpers.
- **Show Backbones Only (for research)** - Show only the backbones without jumpers. Useful for research.

- **Show Only First Path Found** - Stop searching when the first route is resolved.
- **Show Only Least Link Path** - Search all then prune the list to the route with the shortest number of segments.
- **Cable No Prefix, Cable No, SRC Sys, Dest Sys, Src Pin, Dest Pin, Src Loc, Dest Loc** - Fields pulled from the Cables database or populated by jumpers that would need to be created to complete this circuit along this route.
- **Ckt Strand Number** - For more information about circuit strand and ordinal [see here](#) ^[229].
- **Ckt Ordinal** - For more information about circuit strand and ordinal [see here](#) ^[229].
- **Route Number** - The router may find multiple paths or none. If paths are found they will be returned with a route number. You may then select the which, if any, route to add to the circuit.
- **Made It** - Indicates that the route made it to the destination. All routes shown will show **Made It** on the last segment.

4.2.3.7 Combining Circuits

Navigation: [WireCAD CMS Tools Manual](#) ^[208] > [CMS Basics](#) ^[209] > [Circuits](#) ^[228] >



Menu: **Database > Circuits**

Default command line shortcut: **showcircuitsgridcms**

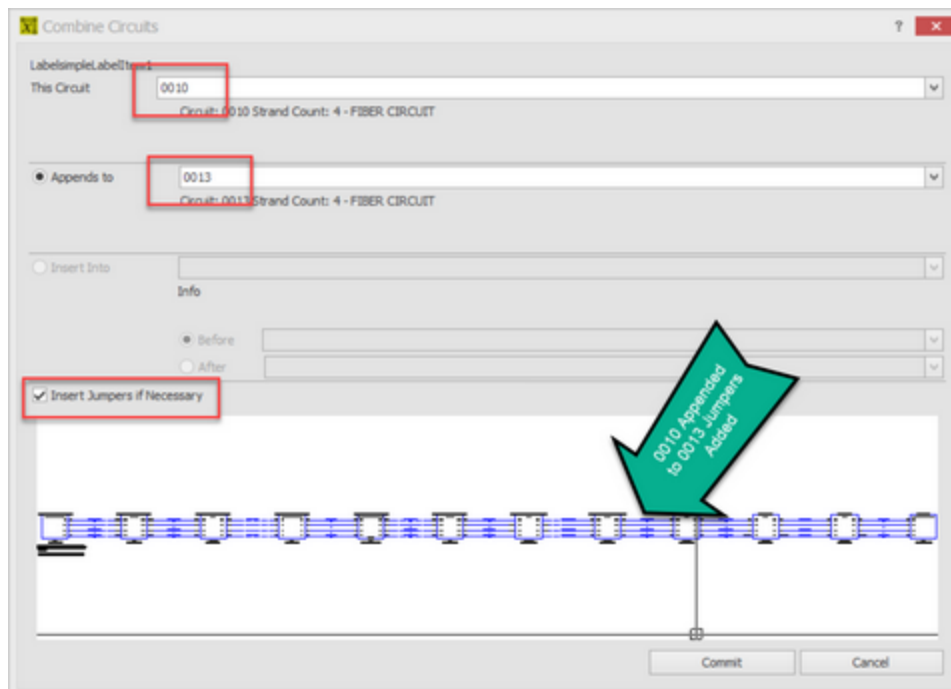
Related Project Settings:

Circuit Status Items

Product Level: **CMS**

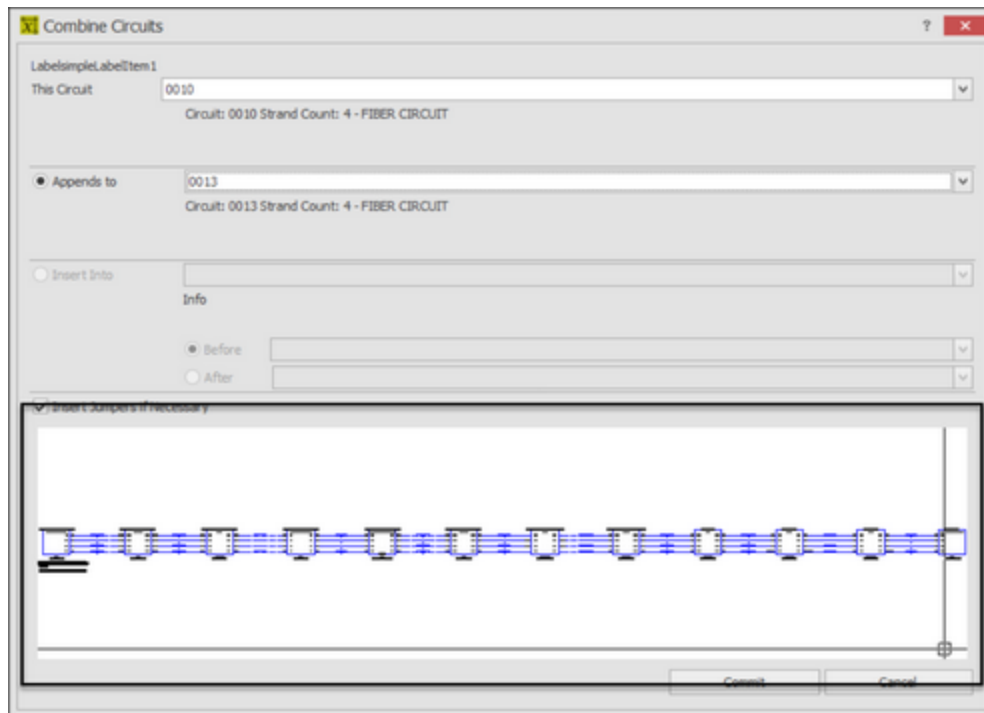
Click **Tools > Combine Circuits**.

WireCAD CMS tools support the concept of combining circuits. Circuits to be combined must have the same Strand Count. The Circuit Name/Number of the Circuit that is being appended or inserted will be discarded.



- **This Circuit** - The Circuit Name/Number of the circuit that will be appended or inserted into another circuit. This circuit Name/Number will be discarded if successfully appended or inserted.

- **Appends To** - The Circuit Name/Number of the circuit that will be appended to.
- **Inserts Into** - The Circuit Name/Number of the circuit that will be inserted into.
- **Before** - Before this SysName.
- **After** - After this SysName.
- **Insert Jumpers if Necessary** - It is possible to create collections of cables in a circuit that do not interconnect. If you do not select this option you will create collection of cables that do not interconnect.
- **Preview**



- **[Commit]** - Do it.

4.2.3.8 How To: UnMake a Real Circuit Into Blind Patches

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Circuits](#) [228] >



The follow procedure describes the process to turn a real circuit into a collection of Blind Patches.

Prerequisites:

At least one Circuit.

Procedure:

1. Open the **Circuits** grid.
2. Select the **Circuit** to remove.
3. Click **Tools>UnMake Circuit**.

The jumpers associated with the circuit will be marked as **IsBlindPatch** = true.

The backbones associated with the circuit will have their **Status** field = Blind_Patch.

4.2.3.9 Import Circuit Data

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [Circuits](#) [228] >



Menu: Database > Circuits

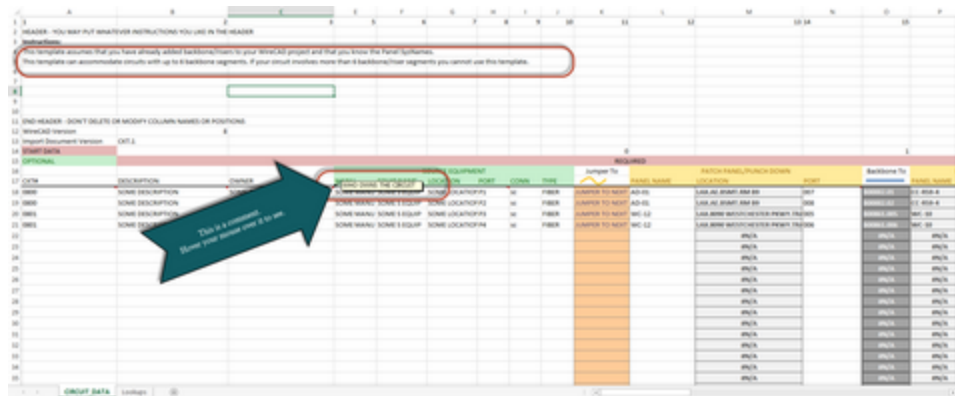
Default command line shortcut: **cmscg**

Related Project Settings:

Circuit Status Items

Product Level: **ENT**

Click **Tools > Import Circuits**.



Command:

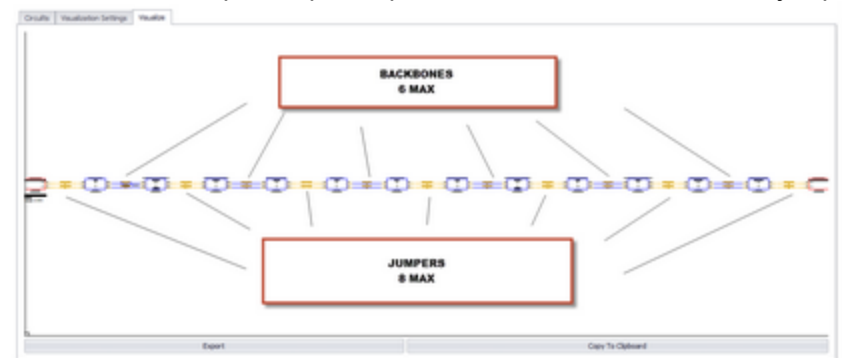
Open the **Circuits Grid**. Then click: **File>Create Import Template**.

Open the **Circuits Grid**. Then click: **File>Import Circuits**.

Limitations

Simplex, duplex or n-strand/core count can be imported. You are limited to 82 rows of circuit data. If you are importing more than 82 rows start another file.

The import template will only accommodate circuits with a maximum of 6 backbones/risers, or in other words a simplex/duplex/n-plex circuit with a maximum of 8 jumpers.



Prerequisites

You must have created backbone data in order to establish circuits.

Related Topics:[Import Backbones](#) ²²³[Circuits Grid](#) ²²⁹**Prerequisites**

You will need to have entered or imported backbone data in order to create circuits.

This is a multi-step process.

Step 1:

Create the import template:

Open the **Circuits Grid**. Then click: **File>Create Import Template**.

This will export the data from your project to the import template. This makes the use of drop downs with project-specific data possible. Without this you will likely get errors if you were to just type the data manually into the import template.

c:\users\public\documents\WireCAD\WireCAD9\Import
Templates\SomeImportTemplateNameWithADateCode.xlsx

NOTE: Be sure to use the most current import template.

Step 2:

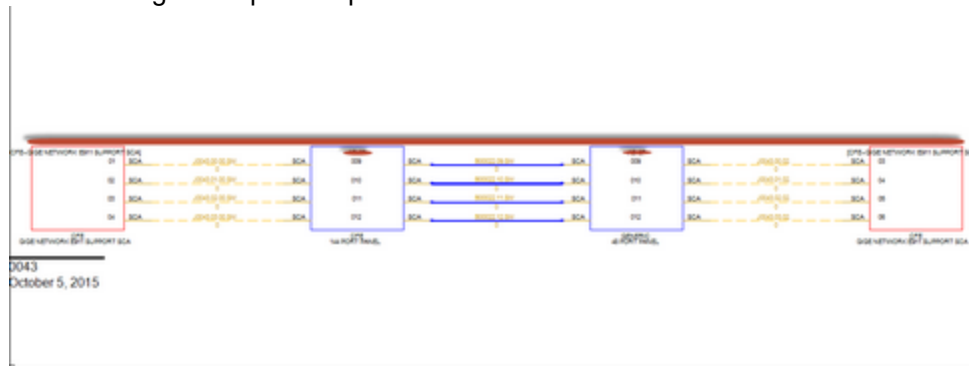
Open the import template in MSExcel and enter your data into the Circuit Import Template created in Step 1 above.

Be sure to fill in all fields.

NOTE: If you leave the **Circuit Number** field blank one will be assigned on import.

A circuit in its simplest form will consist of source and destination equipment with at least one backbone and two jumpers:

The following is a n-plex simple circuit:



This import would require 4 rows in the import template. One row for each horizontal strand/core.

When completing the import template you will need to type the source and destination equipment manufacturers/name and port info.

You will then need to select the first jumper action. The jumper will either jumper to the next backbone or to the final destination.

You will then select the first panel into which we will apply a jumper and the port we will use.

NOTE: The ports displayed in the drop downs are available ports. If you are trying to use a port that is not listed check to see why that port is not listed.

Port and System ID fields are strictly validated. You cannot enter field data in the generated template that is not allowed.

Once you have the first row of the circuit defined you will need to select the Final Action in Column BN. Your choices are:

- CKT CONTINUES ON NEXT LINE
- CKT COMPLETE. NEW CKT NEXT LINE
- CKT COMPLETE. END OF FILE

There can only be one and only one CKT COMPLETE. END OF FILE row in the template.

Once you have entered all of your data. Save the file and proceed to the next step.

Step 3:

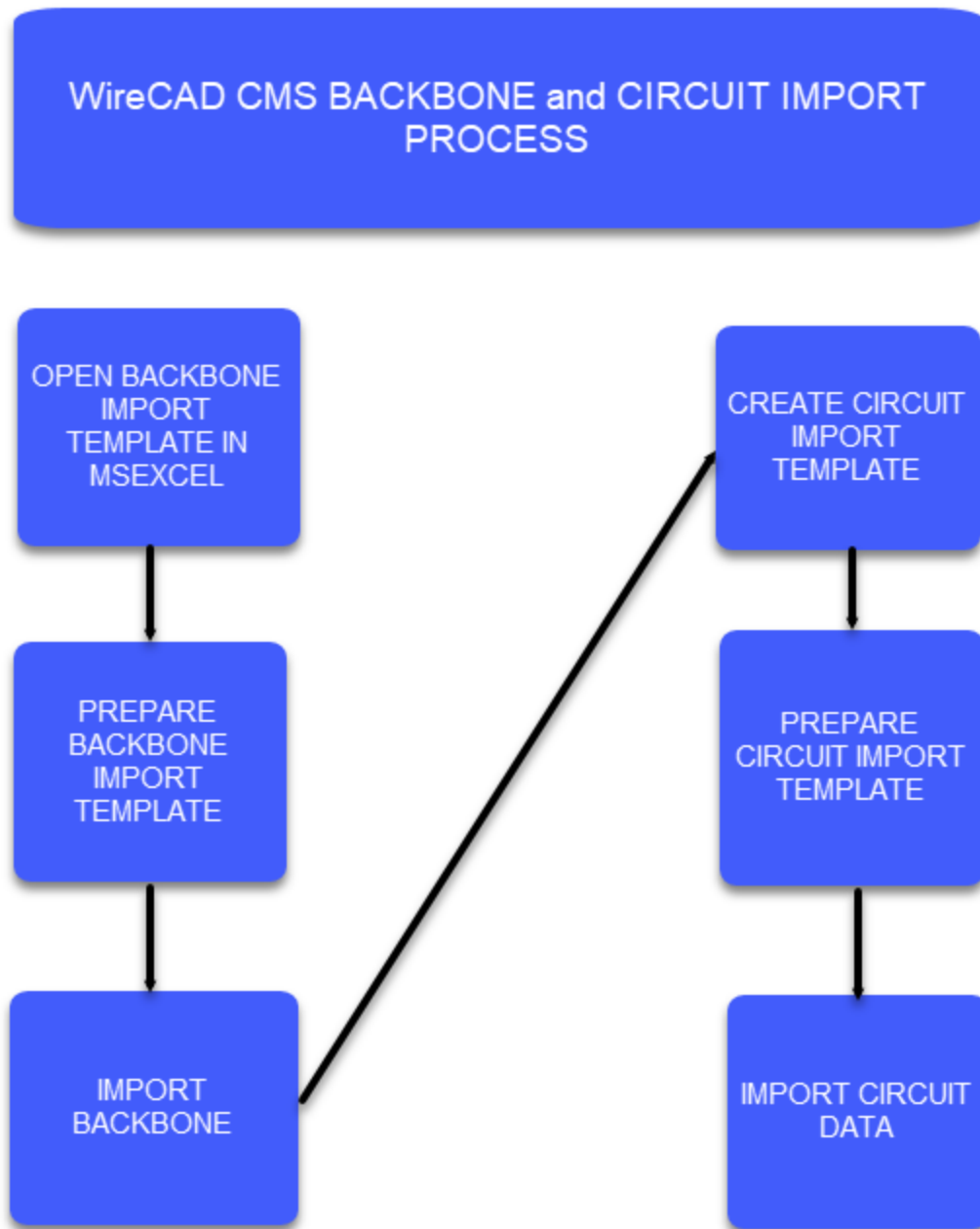
Import your data using the Import Circuits tool.

The Import Circuits tool performs several data integrity checks before the data is actually imported:

- Is the import template the proper format.
- Is all of the necessary data in place.

Then the data is validated against existing data in a preflight check.

- Do the backbones exist.
- Are the ports already assigned to another circuit.
- Is the circuit number available.



4.2.3.10 Exporting Circuit Data to Excel

Navigation: [WireCAD CMS Tools Manual](#)  > [CMS Basics](#)  > [Circuits](#)  >



Menu: **Database > Circuits**

Default command line shortcut: **cmscg**

Related Project Settings:

Circuit Status Items

Product Level: **CMS**

Click **Tools > Export Circuit to Excel**

This function exports the selected circuit(s) to excel. In addition this function also creates a full circuit import template in the same file.

Limitations:

This function will fill up to line 100 in the excel file starting at row 18. If you have selected circuits that expand to more than 82 rows the function will fail.

4.2.4 Next Numbers Grid

Navigation: [WireCAD CMS Tools Manual](#)  > [CMS Basics](#)  >




Menu: **Database > Next Numbers**

Default command line shortcut: **NN**

Related Project Settings:

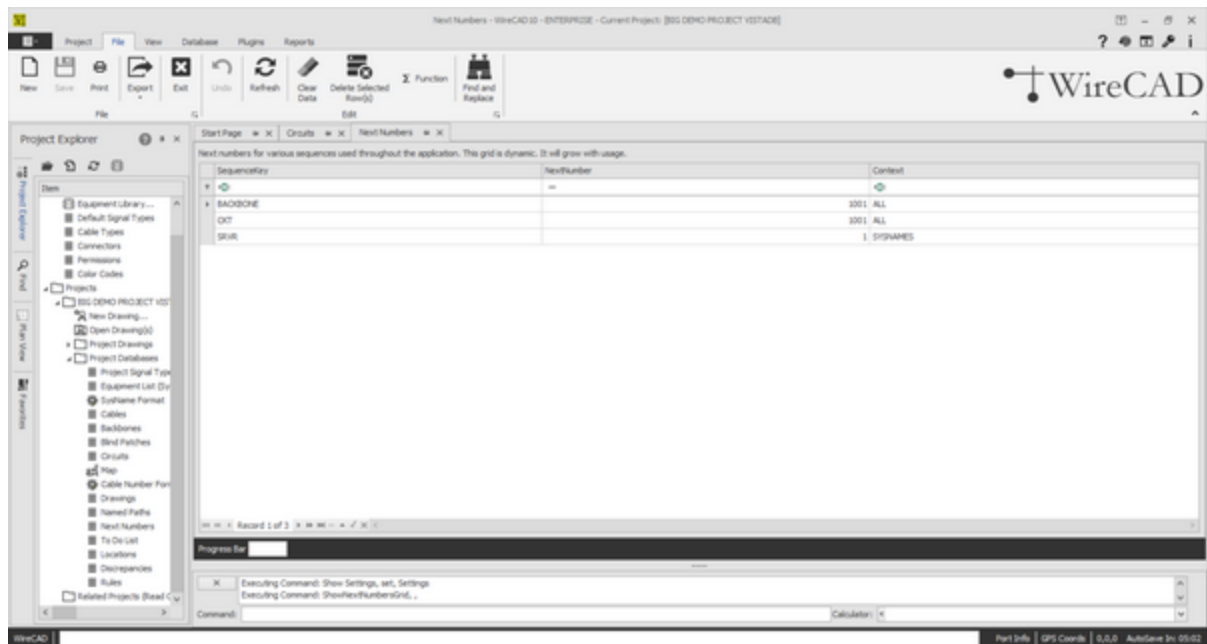
None

Product Level: **ALL**

The **Next Numbers** Grid will show you the next available Cable number based on your [Cable Number Format](#) . To access the **Next Numbers Tool**, type **NN** in the command line prompt.

This grid will show you both **Backbone Cable Numbers** as well as **Standard Cable Numbers**. As you progress through your project, these numbers will change.

To edit these numbers (such as in the event of deleted Backbones) simply click in the Next Number box and fill in the desired number. Be sure that you are not entering a number that is already in use.



Note: Yours will look different as the Next Numbers grid will automatically create entries for each new sequence used by the application.

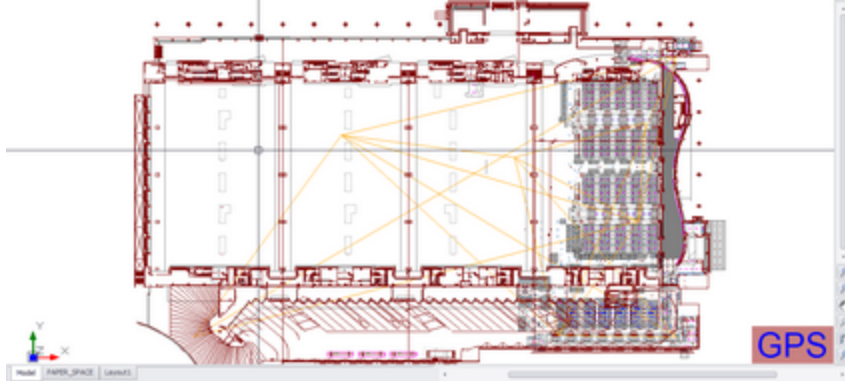
4.2.5 GPS Mapping Tools

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] >

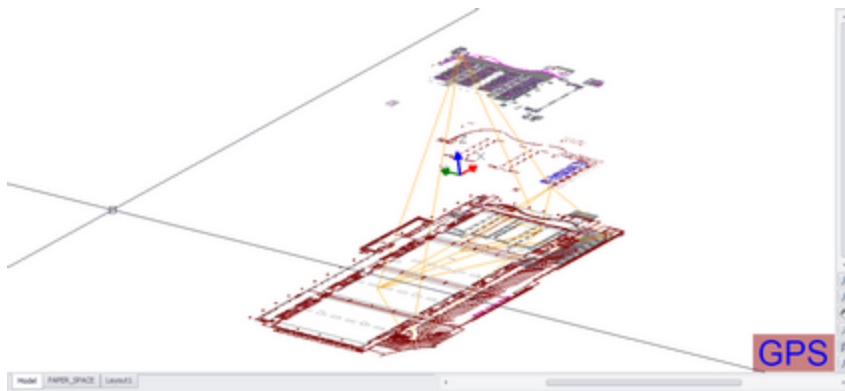


WireCAD v10 CMS provides several ways for you to visualize your locations, panels, backbones and circuits with relation to their GPS coordinates:

1. GPS calibrate (georeference) your dwg drawing and use the tools to pick GPS coordinates from the drawing and display graphical representations of your Locations, Panels and Backbones.



2. You can also set up isometric drawings to show your riser interconnect.



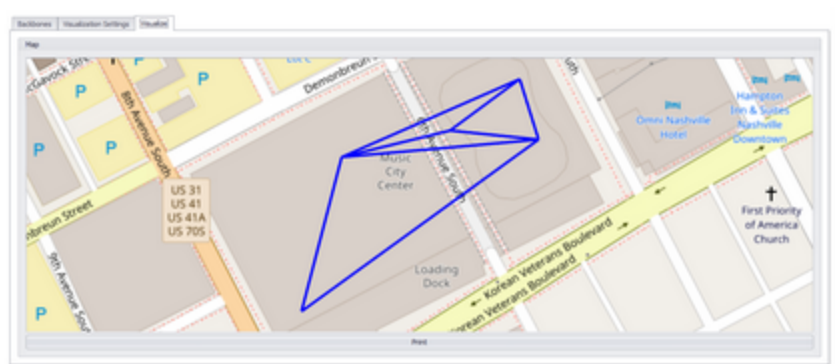
Isometric View Showing Backbone Interconnect

3. Use the Project Map. The functions are the same but the map detail will be less than that of a specific drawing. For example the map may show your building(s) but your CAD drawings will show the detail inside the buildings.



Map Tool - showing Location, Panels and Backbones

4. The Map Tool is available as a [visualization](#) ^[219] in the Backbones grid to show you your Backbones in map view.



Map Tool - within the Backbones Visualizer

5. The Map Tool is available as a [visualization](#) ^[230] in the Circuits grid to show you the GPS path of a selected circuit.



Map Tool - within the Circuit Visualizer

The WireCAD Locations table within the project database contains fields for named locations within your plant. In addition, those named locations can be associated with GPS coordinates.

Drag a column header here to group by that column								
Room	Rack	Qualified Location (What the pr...	GPSLocation	GPSLat	GPSLon	GPSElevation	Description	LocationUs...
101		AMU.ANNEX 1.1ST FLOOR.101	36.1576086,-86.77717961	36.1576086034286	-86.7771796105...	0.0000000...		

Once the project Locations table has associated GPS coordinates you can visualize your data:

- 1.From your GPS calibrated dwg drawing.
- 2.From the Project Map tool.
- 3.From the Backbones grid Visualizer.
- 4.From the Circuits grid Visualizer.

Related Topics:

[GPS Mapping Tools For Drawings](#) ³¹⁸

[GPS Mapping Tools for Map Tool](#) ³²³

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

Accuracy and Precision

There are several factors that effect the accuracy of the GPS coordinate system alignment:

- The accuracy of the drawing.
- The accuracy of the control points.
- The accuracy of the GPS coordinate provided.

From the wikipedia page on decimal precision:

The radius of the semi-major axis of the Earth at the equator is 6,378,137.0 meters resulting in a circumference of 40,075,161.2 meters. The equator is divided into 360 degrees of longitude, so each degree at the equator represents 111,319.9 meters or approximately 111.32 km. As one moves away from the equator towards a pole, however, one degree of longitude is multiplied by the cosine of the latitude, decreasing the distance, approaching zero at the pole. The number of decimal places required for a particular precision at the equator is:

Degree precision versus length

decimal places	decimal degrees	DMS	qualitative scale that can be identified	N/S or E/W at equator	E/W at 23N/S	E/W at 45N/S	E/W at 67N/S
0	1.0	1° 00' 0"	country or large region	111.32 km	102.47 km	78.71 km	43.496 km
1	0.1	0° 06' 0"	large city or district	11.132 km	10.247 km	7.871 km	4.3496 km
2	0.01	0° 00' 36"	town or village	1.1132 km	1.0247 km	787.1 m	434.96 m
3	0.001	0° 00' 3.6"	neighborhood, street	111.32 m	102.47 m	78.71 m	43.496 m
4	0.0001	0° 00' 0.36"	individual street, land parcel	11.132 m	10.247 m	7.871 m	4.3496 m
5	0.00001	0° 00' 0.036"	individual trees	1.1132 m	1.0247 m	787.1 mm	434.96 mm
6	0.000001	0° 00' 0.0036"	individual humans	111.32 mm	102.47 mm	78.71 mm	43.496 mm
7	0.0000001	0° 00' 0.00036"	practical limit of commercial surveying	11.132 mm	10.247 mm	7.871 mm	4.3496 mm
8	0.00000001	0° 00' 0.000036"	specialized surveying (e.g. tectonic plate mapping)	1.1132 mm	1.0247 mm	787.1 µm	434.96 µm

A value in decimal degrees to a precision of 4 decimal places is precise to 11.132 meters at the equator. A value in decimal degrees to 5 decimal places is precise to 1.1132 meter at the equator. Elevation also introduces a small error. At 6,378 m elevation, the radius and surface distance is increased by 0.001 or 0.1%. Because the earth is not flat, the precision of the longitude part of the coordinates increases the further from the equator you get. The precision of the latitude part does not increase so much, more strictly however, a meridian arc length per 1 second depends on latitude at point concerned. The discrepancy of 1 second meridian arc length between equator and pole is about 0.3 metres because the earth is an oblate spheroid.

4.2.5.1 Using the Map Tool

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [GPS Mapping Tools](#) [250] >



WireCAD CMS provides a world map tool. The Map Tool can be opened with the link on the [Project Explorer](#) [562] under the Project Database section.



Concept

The Map tool within WireCAD can be used to associate your Locations table entries to specific GPS coordinates.

Once the Locations table entries have been associated with corresponding GPS coordinates we can automatically place:

- Location Markers
- Panel Markers
- Backbone Markers

Note that the Map Tool requires an internet connection. We are grateful to the folks at OpenStreetMaps for providing the map image data. The map images are © OpenStreetMap contributors.

Pseudo Procedure

The following is a rough procedure for automatically creating a graphic that shows all locations in the Locations table, all SysNames in the Project Equipment List, and all Backbones in the Backbones grid.

1. Use the **Map>Tools>ID GPS Location Coord** [tool](#) [514] to specifically associate your named locations to GPS coordinates.
2. Use the **Map>Display>Location Markers** tool to display all of the Locations in the [Locations](#) [537] table that have GPS coordinates.



3. Use the **Map>Display>Panel Markers** tool to display all of the **SysNames** in the Project [Equipment List](#) ^[532] that have **Locations** in the [Locations](#) ^[537] table that have GPS coordinates.



4. Use the **Map>Display>Backbone Markers** tool to display a polyline representing the [Backbone](#) ^[217] between location/panels that have **Locations** in the [Locations](#) ^[537] table that have GPS coordinates.



Related Topics:

[Map Display](#) ^[324] tab

[GPS Settings](#) ^[370] panel

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.1.1 How To: Associate GPS LAT, LON with your Locations

Navigation: [WireCAD CMS Tools Manual](#) ^[208] > [CMS Basics](#) ^[209] > [GPS Mapping Tools](#) ^[250] > [Using the Map Tool](#) ^[254] >



This topic focuses on the procedure to identify GPS Lat, Lon point from the Map tool.

For a similar procedure using your GPS calibrated dwg drawing see [here](#) ^[266].

Explanation

The WireCAD Locations table within the project database contains fields for named locations within your plant. In addition, those named locations can be associated with GPS coordinates.

Drag a column header here to group by that column

Room	Rack	Qualified Location (What the pr...	GPSLocation	GPSLat	GPSLon	GPSElevation	Description	LocationUs...
101		AMU.ANNEX 1.1ST FLOOR.101	36.1576086,-86.77717961	36.1576086034286	-86.7771796105...	0.0000000...		

There are four fields to contain the GPS data:

- GPSLat - Latitude decimal precision 18 decimals.
- GPSLon - Longitude decimal precision 18 decimals.
- GPSEI - Elevation decimal precision 18 decimals.
- GPSLocation - text convenience field of the Lat,Lon.

Note: We realize that 18 decimal places is far beyond the capabilities of any non-stationary GPS receiver. We just want to be thorough.

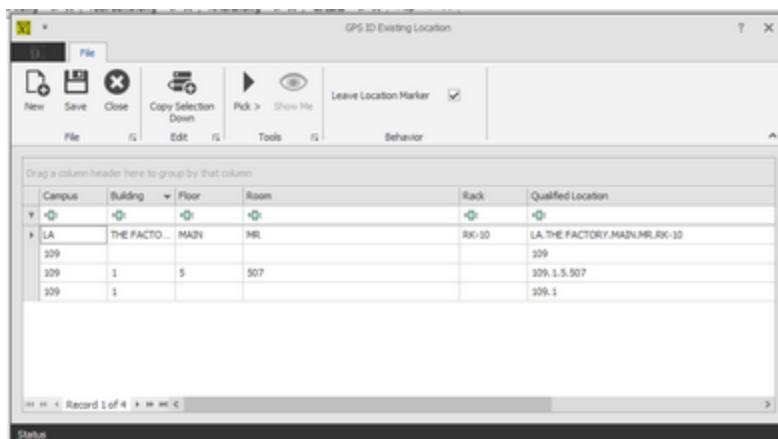
Related Topics:

[ID GPS Locations](#) ^[514] dialog reference.

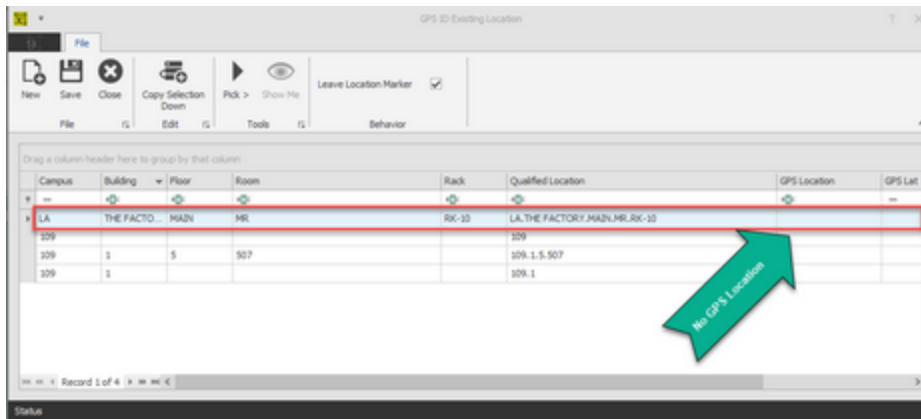
The Map Tool does not give us specific detail of the rooms within our building so some guessing is required if we are going to use this tool to identify Locations within a building. If you have dwg drawing of your building you may be better served to [Calibrate](#) ^[263] those and use the same tool within the drawing to [pick location point](#) ^[266].

WireCAD provides a tool to identify the GPS coordinates from the Map Tool and fill the GPS field data appropriately.

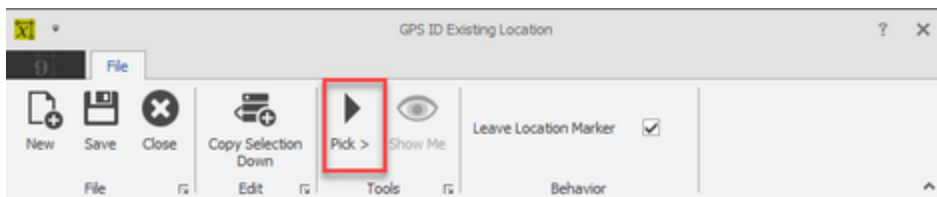
- 1.Open the Map tool.
- 2.Click **Tools > ID GPS Location Coords**. This will open a dialog from which to select the **Location** of interest.



- 3.Select the row(s) of interest within the Locations grid. Here we will define the GPS coordinates for the Location "AMU.COM CTR CORE SWITCH.ROOM 120.5".

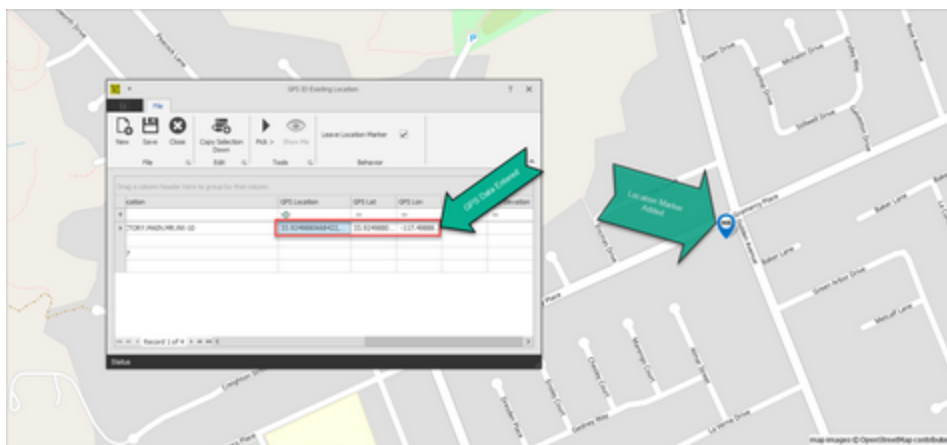


4. Click **Pick >**. The ID dialog is temporarily dismissed to allow you to pick the coordinate from the Map Tool.



5. Pick the coordinate by moving your cursor to the coordinate in the Map Tool and left-click. Here we will select a location in the corner of the building where we estimate the AMU.COM CTR CORE SWITCH.ROOM 120.5 to be. If you have selected **[Leave Location Marker]** as we have here you will leave behind a location marker at the point you have picked.

6. The ID dialog will be shown with the coordinate data in the selected row(s).



7. Click **File > Save**.

8. You are done.

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.1.2 How To: Show Location Markers in the Map Tool

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [GPS Mapping Tools](#) [250] > [Using the Map Tool](#) [254] >



This topic focuses on the procedure to display Locations in the Map tool.
For a similar procedure using your GPS calibrated dwg drawing see this [topic](#) [269].

Explanation

The WireCAD [Map Tool](#) [560] can show your **Locations** for which you have entered GPS coordinate data. Only **Locations** that have GPS data will be considered.

The text displayed in the **Location Marker** is determined by the **Display Format String** of the **Location** section of the [GPS Settings](#) [370] panel.

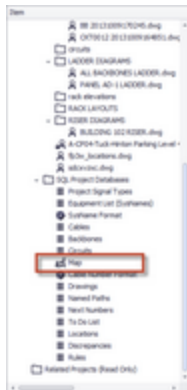
Related Topics:

[Display Tab](#) [324] of the Map Tool interface.

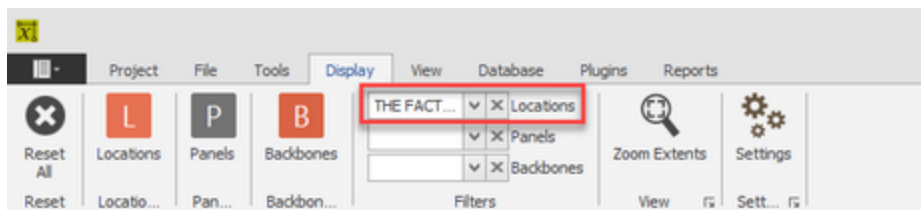
[Map Tool Dialogs](#) [514]

[GPS Settings](#) [370]

1. Open the Map Tool. It can be found in the [Project Explorer](#) [562] under the Project Databases section.



2. Enter any [Filter](#) [262] data that you would like to limit the display of Locations by.



3. Click **Display > Locations**. This will display all locations for which GPS data has been entered. If you have an empty filter all Locations will be shown.



4. You are done.

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.1.3 How To: Show Panel Markers in the Map Tool

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [GPS Mapping Tools](#) [250] > [Using the Map Tool](#) [254] >



This topic focuses on the procedure to display Panels or Equipment in the Map tool. For a similar procedure using your GPS calibrated dwg drawing see this [topic](#) [270].

Explanation

The WireCAD [Map Tool](#) [560] can show your **Panels (SysNames)**. The Location of the SysName will be referenced to the Locations table entry. If the Locations record has GPS data the SysName will be placed. Only **Locations** that have GPS data will be considered.

The text displayed in the **Panel Marker** is determined by the **Display Format String** of the **Panels** section of the [GPS Settings](#) [370] panel.

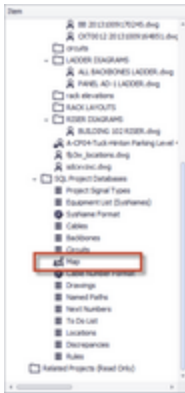
Related Topics:

[Display Tab](#) [324] of the Map Tool interface.

[Map Tool Dialogs](#) [514]

[GPS Settings](#) [370]

1. Open the Map Tool. It can be found in the [Project Explorer](#) [562] under the Project Databases section.



2. Enter any [Filter](#)²⁶² data that you would like to limit the display of Panels (SysNames) by.



3. Click **Display > Panels**. This will display all Panels (SysName) for which GPS data can be found. If you have an empty filter all Panels with Locations with GPS data will be shown.



4. You are done.

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.1.4 How To: Show Backbone Markers in the Map Tool

Navigation: [WireCAD CMS Tools Manual](#)²⁰⁸ > [CMS Basics](#)²⁰⁹ > [GPS Mapping Tools](#)²⁵⁰ > [Using the Map Tool](#)²⁵⁴ >



This topic focuses on the procedure to display Backbones in the Map tool.

For a similar procedure using your GPS calibrated dwg drawing see this [topic](#)²⁷².

Explanation

The WireCAD [Map Tool](#)⁵⁶⁰ can show your **Backbones** for which you have entered GPS coordinate data. Only **Backbones with Locations** that have GPS data will be considered.

The **Backbone Marker** color and Line Width is set in the **Backbones** section of the [GPS Settings](#) ^[370] panel.

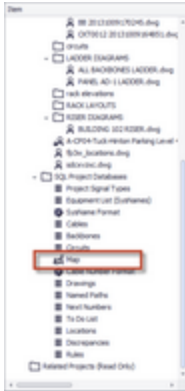
Related Topics:

[Display Tab](#) ^[324] of the Map Tool interface.

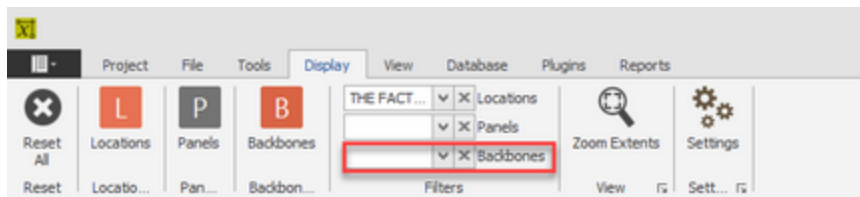
[Map Tool Dialogs](#) ^[514]

[GPS Settings](#) ^[370]

1. Open the Map Tool. It can be found in the [Project Explorer](#) ^[562] under the Project Databases section.



2. Enter any [Filter](#) ^[262] data that you would like to limit the display of Backbones by.



3. Click **Display > Backbones**. This will display all Backbone for which GPS data can be found. If you have an empty filter all Backbone with Location data will be shown.



4. You are done.

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.1.5 Using Filters

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [GPS Mapping Tools](#) [250] > [Using the Map Tool](#) [254] >



The filter section is cascading. Selecting a **Location** will cascade to the **Panels** filter by limiting the **Panels** shown to those that appear at the selected **Location(s)**. In the same way, the **Panels** filter will cascade to the **Backbones** filter. You may select as many of the filterable items you wish.

4.2.5.2 Using Your DWG Drawings as Maps

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [GPS Mapping Tools](#) [250] >

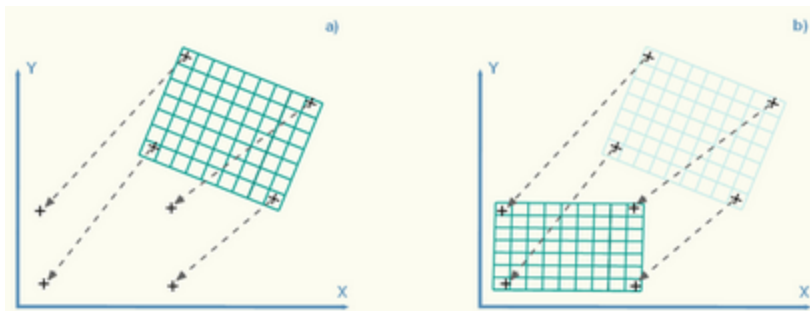


WireCAD CMS supports the use of DWG maps to display equipment and backbones.

Concept

WireCAD can calibrate or georeference any properly scaled drawing to two known control points. These control points typically reference survey markers in the drawing with their associated GPS coordinates. Control points can be any geometry in the drawing for which you know the GPS latitude and longitude. In other words, the control points can be any two points as long as they have known coordinates in both systems.

Note WireCAD uses the term **Calibrate** synonymous with georeference. Calibration or georeferencing is the process of aligning the WCS (world coordinate system) of the drawing with the GPS coordinate system.



Coordinate System Alignment Showing Four Control Points for Illustration. WireCAD Requires Only Two Control Points

Once the drawing has been calibrated, the cursor location is displayed in both Drawing and GPS coordinates.



Further, WireCAD can then be used to associate your **Locations** table entries to specific GPS coordinates.

Once the Locations table entries have been associated with corresponding GPS coordinates we can use that data to automatically draw:

- **Location Markers.** Location markers can be any CAD Block of your choice.
- **Panel Markers.** Panel markers can be any CAD Block of your choice.
- **Backbone Markers.** Backbones will be represented using polylines with the vertexes defined by the GPS coordinates of the end point locations as well as Via locations.

Related Topics:

[Display Tab](#)^[319] of the Drawing Mapping Tools interface.

Pseudo Procedure

The following is a rough procedure for automatically creating a drawing that shows all locations in the Locations table, all SysNames in the Project Equipment List, and all Backbones in the Backbones grid.

Prerequisites

You will need a DWG drawing of your plant or campus that is drawn to scale. The scale used does not matter as long as it is applied consistently. You will need the GPS coordinates of two control points in the drawing.

1. Calibrate the drawing using Mapping Tools>GPS>Calibrate.
2. Use the Mapping Tools>Tools>[ID GPS Location Coord](#)^[514] tool to specifically associate your named locations to GPS coordinates.
3. Use the Mapping Tools>Display>Location Markers tool to display all of the Locations in the Locations table that have GPS coordinates.
4. Use the Mapping Tools>Display>Panel Markers tool to display all of the SysNames in the Project Equipment List that have Locations in the Locations table that have GPS coordinates.
5. Use the Mapping Tools>Display>Backbone Markers tool to display a polyline representing the backbone between location/panels that have Locations in the Locations table that have GPS coordinates.

4.2.5.2.1 How To: Calibrate (Geo-Reference) Your DWG to the GPS Coordinate System

Navigation: [WireCAD CMS Tools Manual](#)^[208] > [CMS Basics](#)^[209] > [GPS Mapping Tools](#)^[250] > [Using Your DWG Drawings as Maps](#)^[262] >



Prerequisites

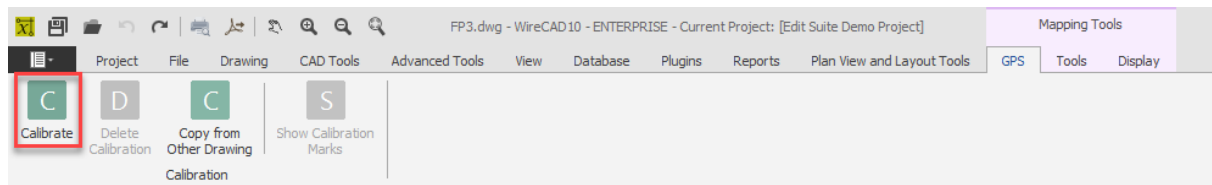
This topic assumes that you have a scaled drawing of your plant.

Further that there exists in the drawing two control points. Control points are typically survey markers for which you have the GPS coordinate data. However, they can be any geometry in your drawing such as the corners of buildings or manhole covers as long as you have reliable GPS data for that point.

Procedure

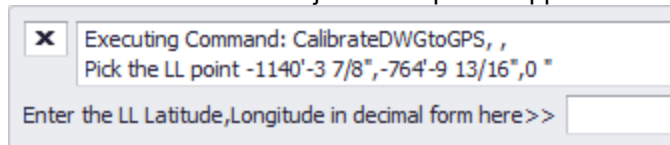
1. Open your plant or campus drawing. If the your plant drawing is very large you may want to consider [Externally Referencing](#)^[409] (XREF) it into a new drawing. This is done to keep the save operation lightweight. Externally referenced drawings do not get saved, only the base drawing.

2. Click Mapping Tools > GPS > Calibrate.



3. You will be prompted in the Command Prompt to pick the first point, or lower left point, from the drawing. Do so.

4. Enter the GPS point data for the lower left point. The control point does not necessarily have to be the lower left corner. We just need points opposite.

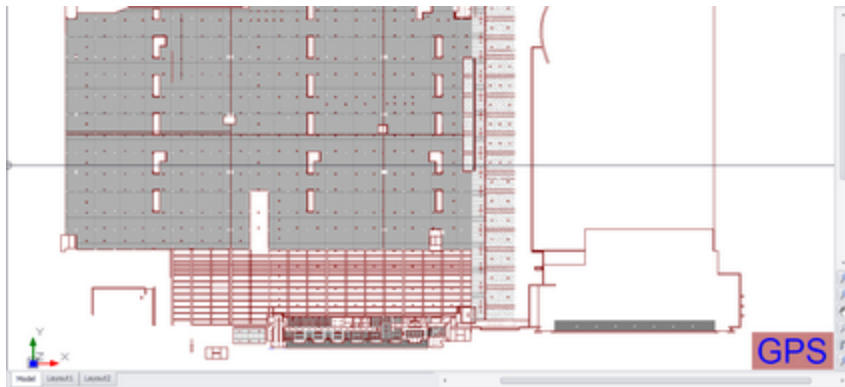


Please note that the coordinate is expected in decimal lat, lon form. **Do not Ctrl+V** to paste your coordinate data as this will initiate the paste function in the drawing not the command line. Use the Right-Click context menu and select Paste there.

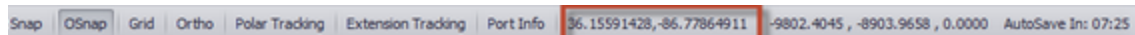
5. Pick the upper right point in the drawing.

6. Enter its GPS coordinate.

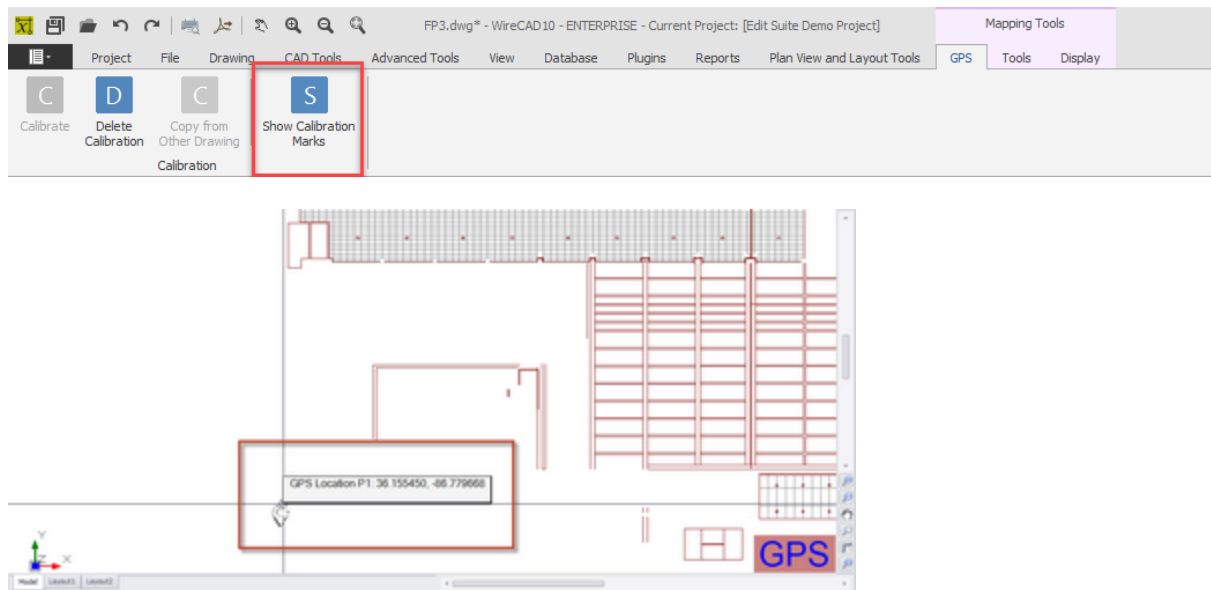
When done the drawing will display GPS in the lower right-hand corner to indicate that it is calibrated.



You will be able to use the GPS tools to pick location points from the drawing. The statusbar text will show you the GPS coordinate of the current cursor position.



Once the drawing is calibrated you can use the **Mapping Tools > GPS > Show Calibration Marks** tool to place a marker in the drawing at your calibration points.

**Related Topics:**

[Display Tab](#) ^[319] of the Drawing Mapping Tools interface.

4.2.5.2.2 How To: Copy Calibration Data from another Drawing

Navigation: [WireCAD CMS Tools Manual](#) ^[208] > [CMS Basics](#) ^[209] > [GPS Mapping Tools](#) ^[250] > [Using Your DWG Drawings as Maps](#) ^[262] >

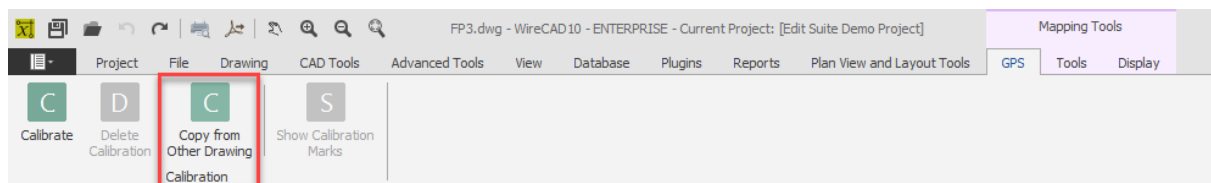


WireCAD provides a tool to copy calibration or georeference data from another drawing. This assumes the following:

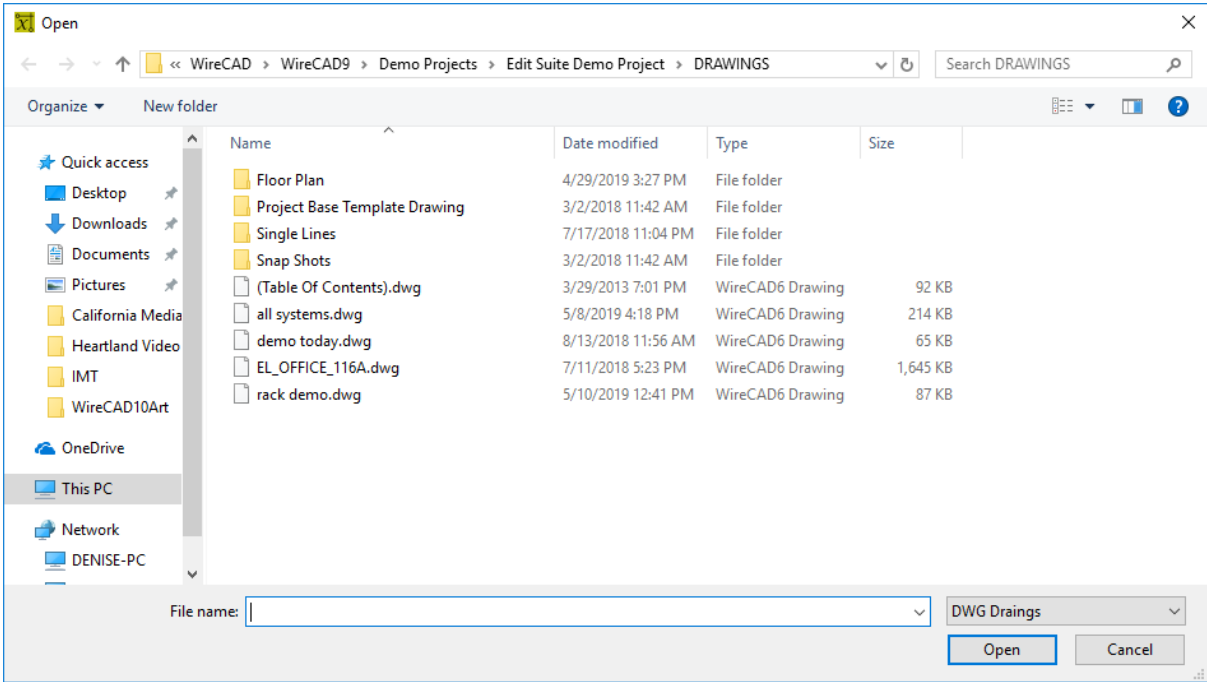
- The drawing you are copying from has been Calibrated. You can tell if the drawing has been calibrated if you open it and it displays the text GPS in the lower right-hand corner.
- The geometry in the two drawings shares the same WCS origin point. This is crucial. If the geometry in the two drawings does not share the same origin point the GPS coordinate system will be off.

Procedure

1. Open the drawing that you want to copy Calibration data into.



2. Click **Mapping Tools > GPS > Copy From Other Drawing**. This will open a file browser dialog.



3. Browse to the Calibrated drawing and select it.
4. Click **[Open]**. This will initiate the process. If the drawing has been calibrated the process will proceed and the commandline history will indicate that the process has been completed.
5. Click **File > Save** to save the calibration data with the drawing.
6. You are done.

4.2.5.2.3 How To: Associate GPS LAT, LON with your Locations

Navigation: [WireCAD CMS Tools Manual](#)^[208] > [CMS Basics](#)^[209] > [GPS Mapping Tools](#)^[250] > [Using Your DWG Drawings as Maps](#)^[262] >



This topic focuses on the procedure to identify GPS Lat, Lon point from a GPS calibrated drawing. For a similar procedure using the Map tool see this [topic](#)^[255].

Prerequisites

This topic assumes that you have a GPS Calibrated drawing of your plant. For more information on calibrating your drawing see this [topic](#)^[263].

Explanation

The WireCAD Locations table within the project database contains fields for named locations within your plant. In addition, those named locations can be associated with GPS coordinates.

Drag a column header here to group by that column								
Room	Rack	Qualified Location (What the pr...	GPSLocation	GPSLat	GPSLon	GPSElevation	Description	LocationUs..
101		AMU ANNEX 1.1ST FLOOR.101	36.1576086,-86.77717961	36.1576086034286	-86.7771796105...	0.0000000...		

There are four fields to contain the GPS data:

- GPSLat - Latitude decimal precision 18 decimals.
- GPSLon - Longitude decimal precision 18 decimals.
- GPSEI - Elevation decimal precision 18 decimals.

- GPSLocation - text convenience field of the Lat,Lon.

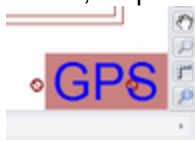
Note: We realize that 18 decimal places is far beyond the capabilities of any non-stationary GPS receiver. We just want to be thorough.

Related Topics:

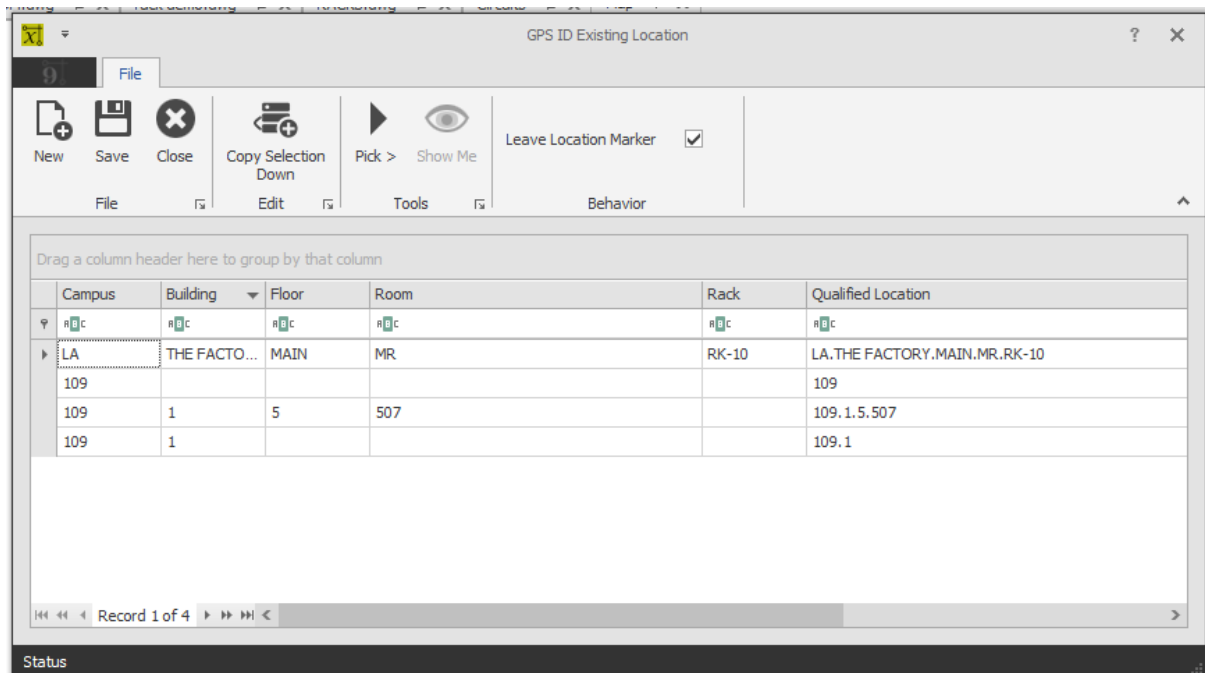
[ID GPS Locations](#) ⁵¹⁴ dialog reference.

WireCAD provides a tool to identify the GPS coordinates from your GPS calibrated drawing and fill the field data appropriately.

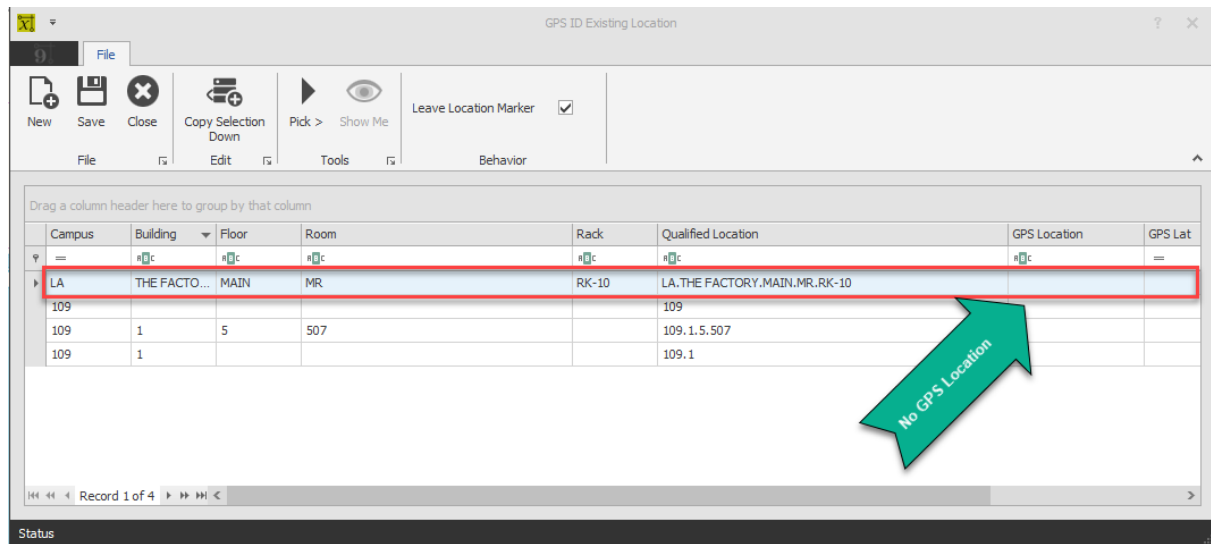
- 1.Open your GPS calibrated drawing. If your drawing does not display GPS in the lower right-hand corner, stop.



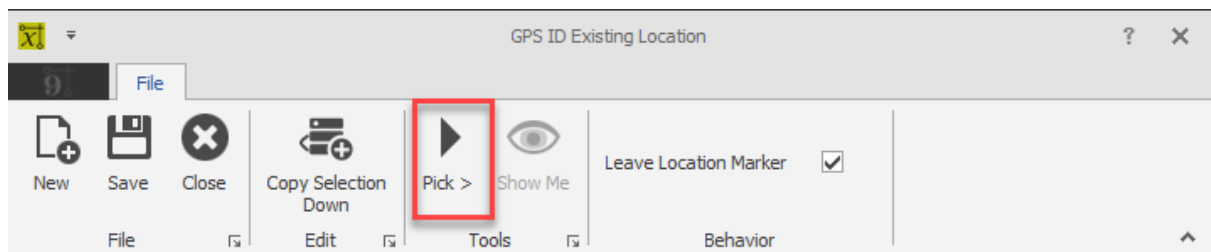
- 2.Click **Mapping Tools > Tools > ID GPS Location Coords**. This will open a dialog from which to select the **Location** of interest.



- 3.Select the row(s) of interest within the Locations grid. Here we will define the GPS coordinates for the Location "AMU.COM CTR CORE SWITCH.ROOM 120.5".

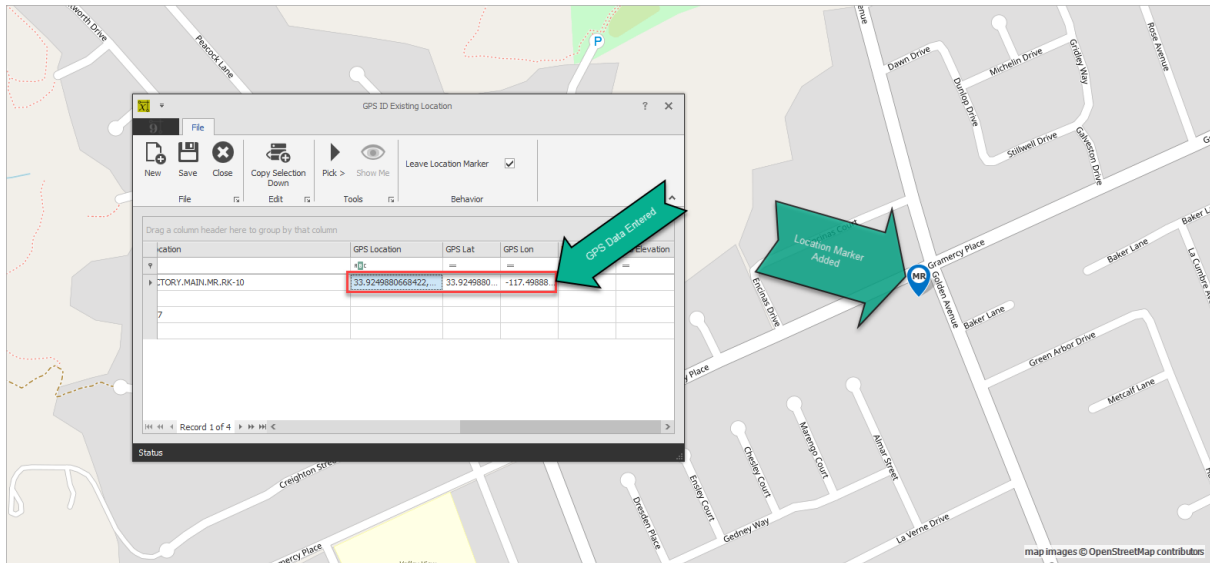


4. Click **Pick >**. The ID dialog is temporarily dismissed to allow you to pick the coordinate from the drawing.



5. Pick the coordinate by moving your cursor to the coordinate in the drawing and left-click. If you have selected **[Leave Location Marker]** as we have here you will leave behind a location marker at the point you have picked.

6. The ID dialog will be shown with the coordinate data in the selected row(s).



7. Click **File > Save**.

8. You are done.

4.2.5.2.4 How To: Show Location Markers in your Drawing

Navigation: [WireCAD CMS Tools Manual](#) ^[208] > [CMS Basics](#) ^[209] > [GPS Mapping Tools](#) ^[250] > [Using Your DWG Drawings as Maps](#) ^[262] >



This topic focuses on the procedure to place Location Markers in a GPS calibrated drawing.

For a similar procedure using the Map tool see this [topic](#) ^[258].

Prerequisites

This topic assumes that you have a GPS calibrated drawing of your plant.

For more information on calibrating your drawing see [here](#) ^[263].

Further, this topic assumes that you have identified GPS coordinates for your Locations in your Locations table. For more information on this topic see [here](#) ^[266].

Explanation

WireCAD has the ability to place a marker for every location in the Locations table for which GPS coordinate data exists. The markers placed are based on the current filter settings. For more information on setting filters see [here](#) ^[274].

The type of marker placed, its scale, and Overlay Mode can be determined in the menu structure.

The marker text, color and Overlay Mode settings can be controlled in the [GPS Settings](#) ^[370] panel.

Related Topics:

[GPS Settings](#) ^[370]

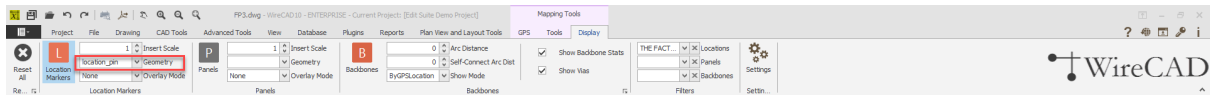
[Display](#) ^[319] Tab

Procedure

1. Open your GPS calibrated drawing. If your drawing does not display GPS in the lower right-hand corner, stop.

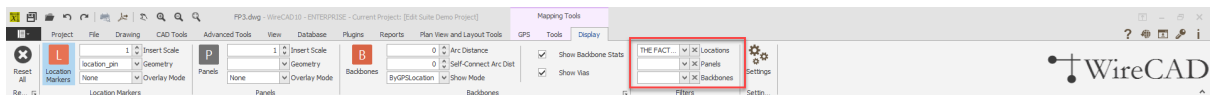


2. Select the **Geometry** and **Scale**. **Geometry** sets the block to insert as a **Location Marker**. For more information on the **Geometry** see this [topic](#)³¹⁹.



3. Select the **Overlay Mode**. For more information on the **Overlay Mode** see this [topic](#)³¹⁹.

4. Enter any [Filter](#)²⁶² data that you would like to limit the display of Locations by. For more information on the **Filters** see this [topic](#)²⁷⁴.



Here we have selected a block that has the appearance of a Location pin



5. Click **Mapping Tools > Display > Locations**. This will display all locations for which GPS data has been entered. If you have an empty filter all Locations will be shown.



6. You are done.

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.2.5 How To: Show Panel Markers in your Drawing

Navigation: [WireCAD CMS Tools Manual](#)²⁰⁸ > [CMS Basics](#)²⁰⁹ > [GPS Mapping Tools](#)²⁵⁰ > [Using Your DWG Drawings as Maps](#)²⁶² >



This topic focuses on the procedure to place Panel Markers in a GPS calibrated drawing. For a similar procedure using the Map tool see this [topic](#) ²⁵⁹.

Prerequisites

This topic assumes that you have a GPS calibrated drawing of your plant.

For more information on calibrating your drawing see [here](#) ²⁶³.

Further, this topic assumes that you have identified GPS coordinates for your Locations in your Locations table. For more information on this topic see [here](#) ²⁶⁶.

Explanation

WireCAD has the ability to place a marker for every Panel in the Locations table for which GPS coordinate data exists. The markers placed are based on the current filter settings. For more information on setting filters see [here](#) ²⁷⁴.

The type of marker placed, its scale, and Overlay Mode can be determined in the menu structure. The marker text, color and Overlay Mode settings can be controlled in the [GPS Settings](#) ³⁷⁰ panel.

Related Topics:

[GPS Settings](#) ³⁷⁰

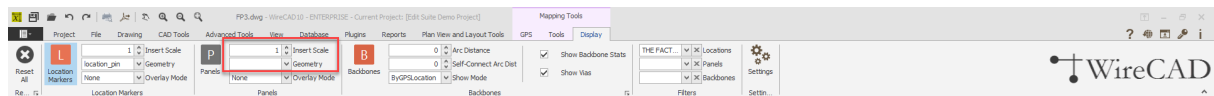
[Display](#) ³¹⁹ Tab

Procedure

1. Open your GPS calibrated drawing. If your drawing does not display GPS in the lower right-hand corner, stop.



2. Select the **Geometry** and **Scale**. **Geometry** sets the block to insert as a **Panel Marker**. For more information on the **Geometry** see this [topic](#) ³¹⁹.

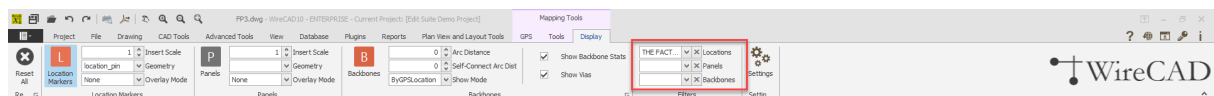


Here we have selected a block that has the appearance of a simple rectangle with an angled line.



3. Select the **Overlay Mode**. For more information on the **Overlay Mode** see this [topic](#) ³¹⁹.

4. Enter any [Filter](#) ²⁶² data that you would like to limit the display of **Panels** by. For more information on the **Filters** see this [topic](#) ²⁷⁴.



5. Click **Mapping Tools > Display > Locations**. This will display all locations for which GPS data has been entered. If you have an empty filter all Panels with found GPS data will be shown.



Here we see the panels displayed with the Radial Overlay mode

6. You are done.

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.2.6 How To: Show Backbone Markers in your Drawing

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [GPS Mapping Tools](#) [250] > [Using Your DWG Drawings as Maps](#) [262] >



This topic focuses on the procedure to place Panel Markers in a GPS calibrated drawing. For a similar procedure using the Map tool see this [topic](#) [260].

Prerequisites

This topic assumes that you have a GPS calibrated drawing of your plant.

For more information on calibrating your drawing see [here](#) [263].

Further, this topic assumes that you have identified GPS coordinates for your Locations in your Locations table. For more information on this topic see [here](#) [266].

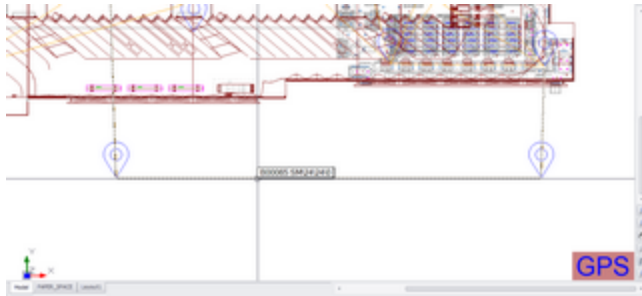
Explanation

WireCAD has the ability to place a polyline for every **Backbone** in the [Backbones](#) [526] table for which GPS location coordinate data exists. The markers placed are based on the current filter settings. For more information on setting filters see [here](#) [274].

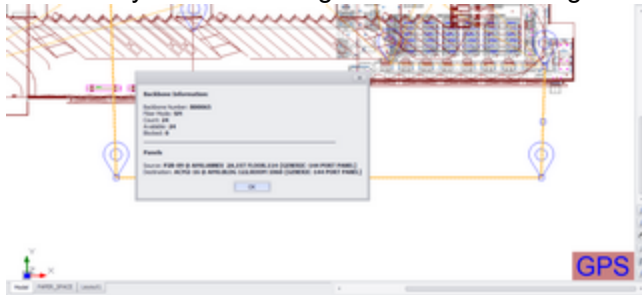
The **Arc Distance**, **Self-Connect Arc Distance** and the **Show Mode** can be determined in the menu. For more information on these settings see this [topic](#) [319].

The marker text, color and other settings can be controlled in the [GPS Settings](#) [370] panel.

The tool can **[Show Backbone Stats]** such as the fiber count, availability, and blocked fiber count. This data, if generated, is displayed in the tooltip of the **Backbone** when the mouse hovers over it. The statistics are presented in the form: Backbone Number FiberMode\Count\Available\Blocked



You can also see **Backbone** stats with or without having the **[Show Backbone Stats]** checkbox checked by double-clicking a **Backbone**. Doing so will present a message box with the stats.

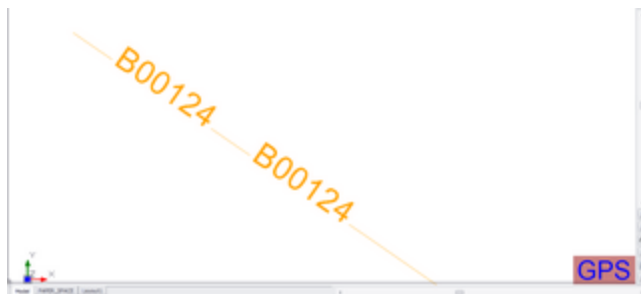


The polyline can be made to show only its end points or if the **Show Vias** is checked it will jump from the starting location through every **Via** location until it reaches the ending location.



Backbone from panel to panel Via Manhole 1 and Manhole 2

The polyline can also be made to show the Backbone number as the polyline LineType



Backbone showing number as Line Type

Related Topics:

[GPS Settings](#) 370

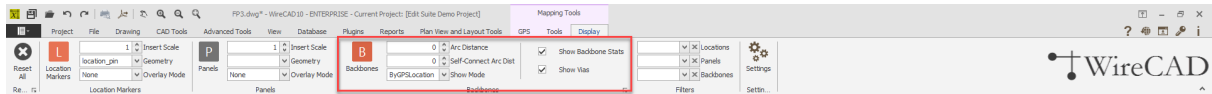
[Display](#) 319 Tab

Procedure

1. Open your GPS calibrated drawing. If your drawing does not display GPS in the lower right-hand corner, stop.

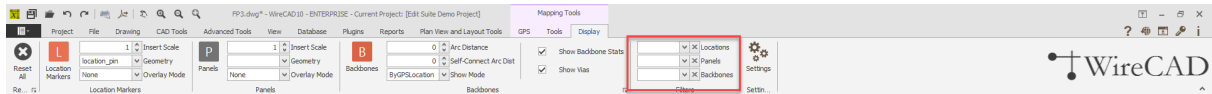


2. Select the **Arc Distance** and **Self-Connect Arc Dist.** **Arc Distance** sets the polyline bulge factor. An Arc Distance of 1 creates an arc of 180°. For more information see this [topic](#)³¹⁹.

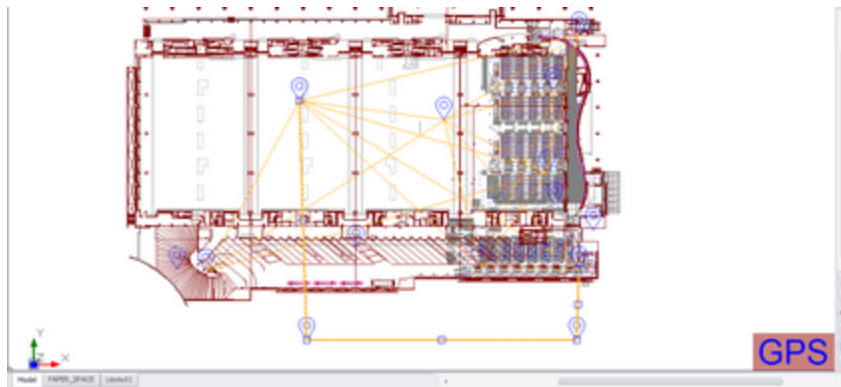


3. Select the **Overlay Mode**. For more information on the **Overlay Mode** see this [topic](#)³¹⁹.

4. Enter any [Filter](#)²⁶² data that you would like to limit the display of **Backbones** by. For more information on the **Filters** see this [topic](#)²⁷⁴.



5. Click **Mapping Tools > Display > Backbones**. This will display all locations for which GPS data has been entered. If you have an empty filter all Panels with found GPS data will be shown.



Here we see the **Backbones** displayed by **GPS Location** and we are showing **Vias**

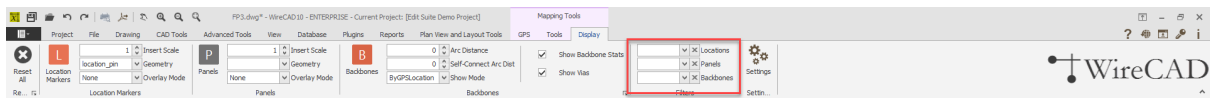
6. You are done.

Note: Although we are showing a real world location in these examples the data is completely fictitious. Any resemblance to any data living or dead is purely coincidental.

4.2.5.2.7 Using Filters

Navigation: [WireCAD CMS Tools Manual](#)²⁰⁸ > [CMS Basics](#)²⁰⁹ > [GPS Mapping Tools](#)²⁵⁰ > [Using Your DWG Drawings as Maps](#)²⁶² >





The filter section is cascading. Selecting a **Location** will cascade to the **Panels** filter by limiting the **Panels** shown to those that appear at the selected **Location(s)**. In the same way, the **Panels** filter will cascade to the **Backbones** filter. You may select as many of the filterable items you wish.

4.2.5.2.8 HOW TO: Create a Multistory Isometric Drawing to Display GPS Data

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Basics](#) [209] > [GPS Mapping Tools](#) [250] > [Using Your DWG Drawings as Maps](#) [262] >

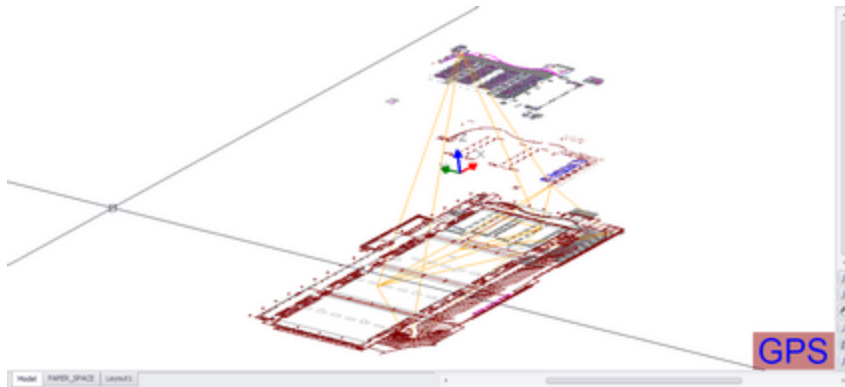


Prerequisites

We will need a new drawing. We will also need the floor plans for each floor one per file all aligned to the same X,Y origin.

Concept

We can use the different floor plans of a structure stacked one on top of another along the Z axis to display our interconnect in an isometric view.



Isometric View Showing Backbone Interconnect

Here we have created a new drawing called `map.dwg` and Externally Referenced each floor on the same X,Y origin but at a different Z axis location.

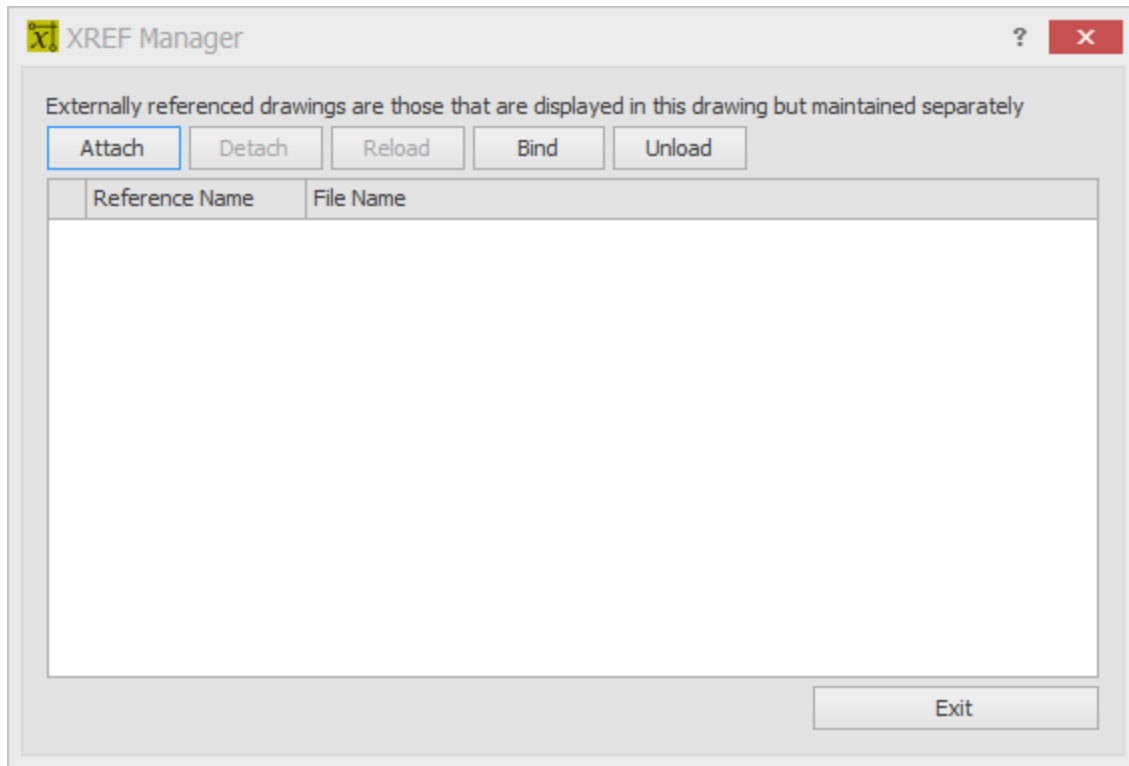
Pseudo Procedure

1. Create a new drawing.
2. XREF the first floor inserting it at 0,0,0.
3. XREF the next floor inserting it at 0,0,n where n is some value along the Z axis far enough away from the origin that when we rotate the drawing into isometric view the two floors do not interfere with each other.
4. Lather, rinse, repeat for each floor inserting the floor at 0,0,n*f where n is the distance along the Z axis and f is the floor number.
5. For ease of use we recommend creating a layer for each floor and assigning the floor to the layer in order to control visibility.
6. [Calibrate](#) [263] the drawing.

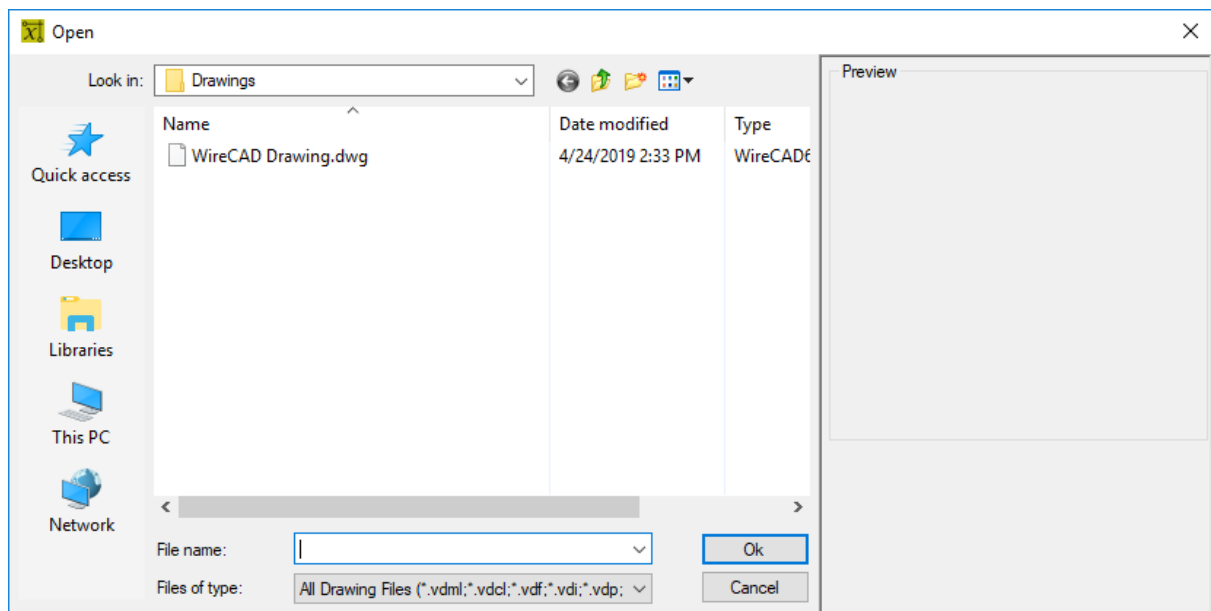
Procedure

Before you start. Collect all of your floor plan files into a known location. We recommend in the same folder as the new drawing you are about to create.

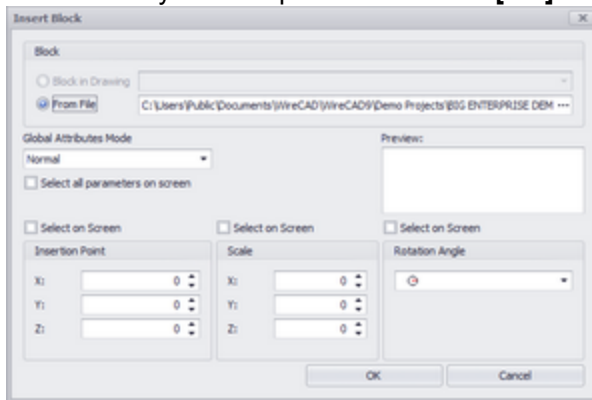
1. [Create a New Drawing](#)⁷⁷. Name it map.dwg or some-such.
2. Make sure you are in Model space.
3. With the map.dwg drawing as the active form. Click **CAD Tools > External Reference Manager (XREF)**. This will open the [XREF Manager dialog](#)⁴⁰⁹.



4. Click the **[Attach]** button. This will open a file browser dialog.

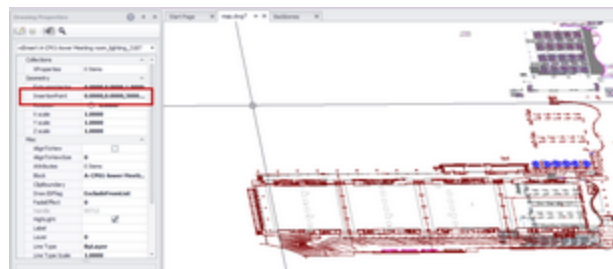


5. Browse to your floor plan file and click **[OK]**. This will display the Insert dialog.



6. For the first floor use the default settings for all other floor enter a Z axis insertion point. Click **[OK]**.

You may need to play with this to find a number that works for the structure you are trying to show. You don't have to reinsert it every time you want to modify the Z axis. Just open the [Drawing Properties](#) ⁵⁶⁸ tool panel and with the XREF selected in the drawing find the Insertion Point property and modify the Z Axis there.



7. This will load the drawing. Note that you may not see the drawing in the current view of Model space until you dismiss all dialogs and click Zoom Extents.

8. Repeat steps 4-7 until you have added all floors.

9. [Calibrate](#) ²⁶³ the map.dwg drawing.

10. Click **File > Save**.

11. You are done.

4.3 CMS Settings and Options

Navigation: [WireCAD CMS Tools Manual](#) ²⁰⁸ >

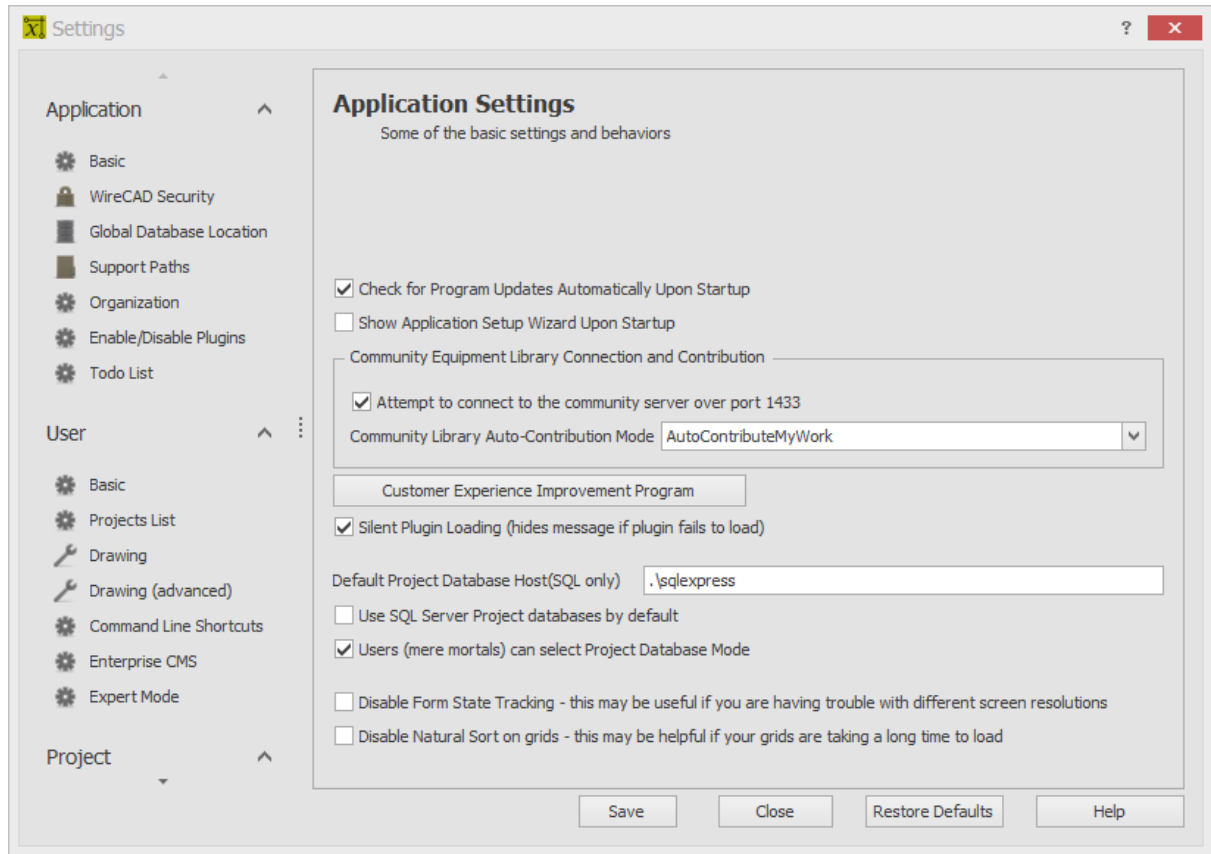


Menu: **Application Menu > Settings**

Default Commandline Shortcut: **set**

To access the settings menu, click **Application Menu > Settings**. Then select one of the three default sets of settings **Application**, **User** or **Project** by clicking on the down arrow.

Note: Third party Plugins may register their own settings categories and panels. Shown are the stock settings.



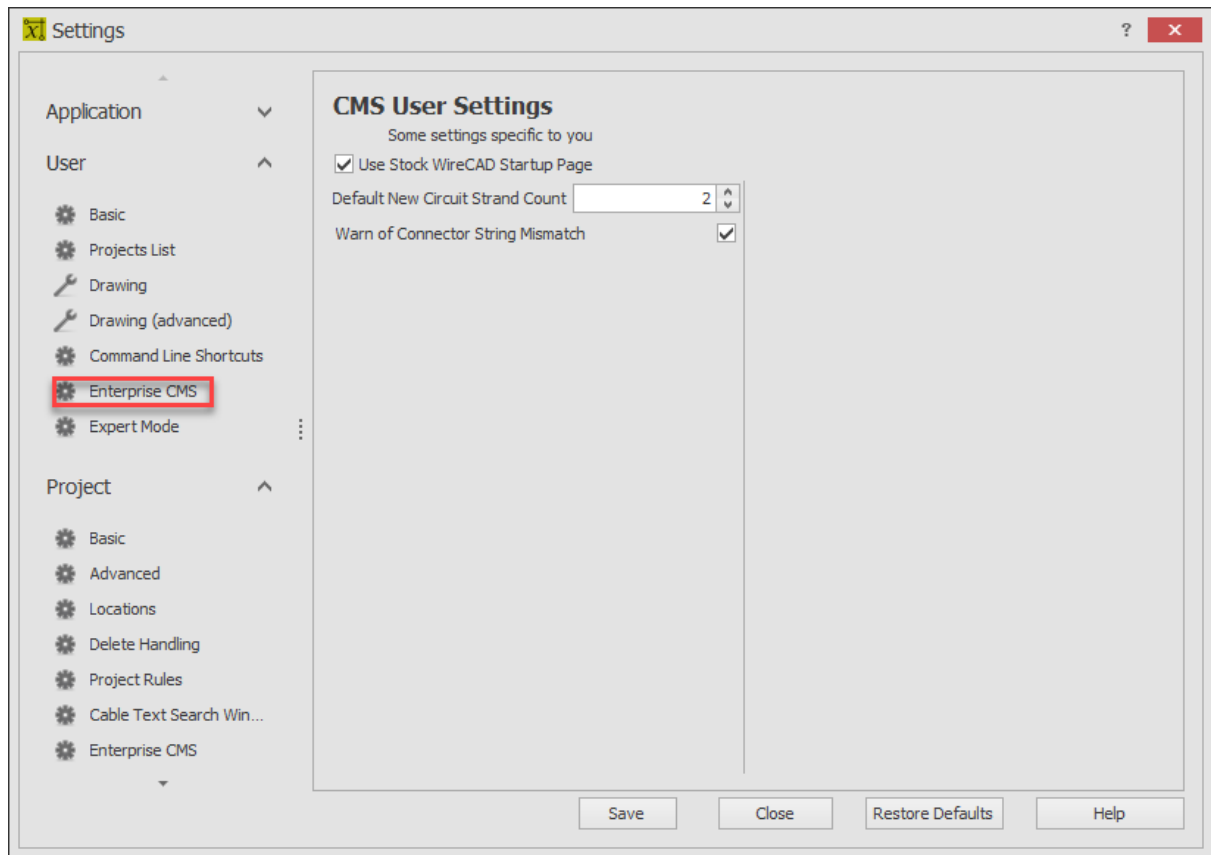
4.3.1 User Settings

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Settings and Options](#) [277] >



Menu: **Application Menu > Settings[User][CMS User]**

These are the additional user settings that pertain specifically to the CMS module.



- **Use Stock WireCAD Startup Page** - Not used anymore.
- **Default New Circuit Strand Count** - When creating a new circuit, a default strand count for jumpers is created. This will change that strand count to a different value.
- **Warn Of Connector String Mismatch** - Having this box checked will allow a warning message to display anytime WireCAD detects that 2 ports being connected have a different connector type.

4.3.2 Project Settings

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Settings and Options](#) [277] >

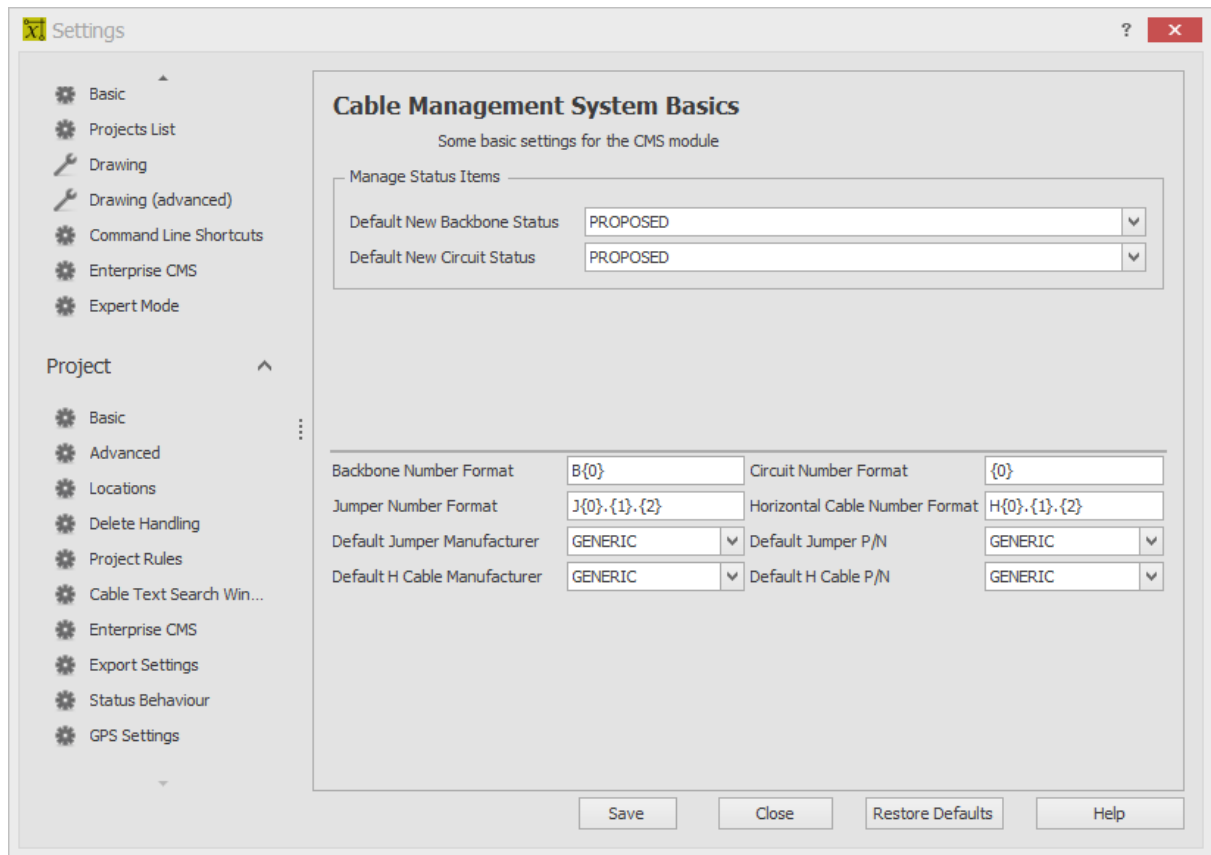


Menu: **Application Menu > Settings[User][CMS User]**

These are the additional project settings that pertain specifically to the CMS module.

Note: You will need to have an open project in order to access these settings.

Note: These settings are project specific and will need to be created for each new project.



•**Default New Backbone Status** - When creating a backbone, you have the ability to mark a "status" on that backbone such as "In Use", "Proposed" ETC. Selecting a status will cause that to become the default for all backbones in this project. Status Items are defined in the **Status Behavior** Project Settings panel.

•**Default New Circuit Status** - Just like above, when creating a Circuit, you have the ability to mark a status on that circuit.

•**Backbone Number Format** - The variable {0} contains the next number in the Next Numbers grid for Backbones.

•**Circuit Number Format** - The variable {0} contains the next number in the Next Numbers grid for Circuits.

•**Jumper Number Format** -

The variable {0} contains the Circuit base number.

The variable {1} contains the Strand Number of the circuit for this jumper

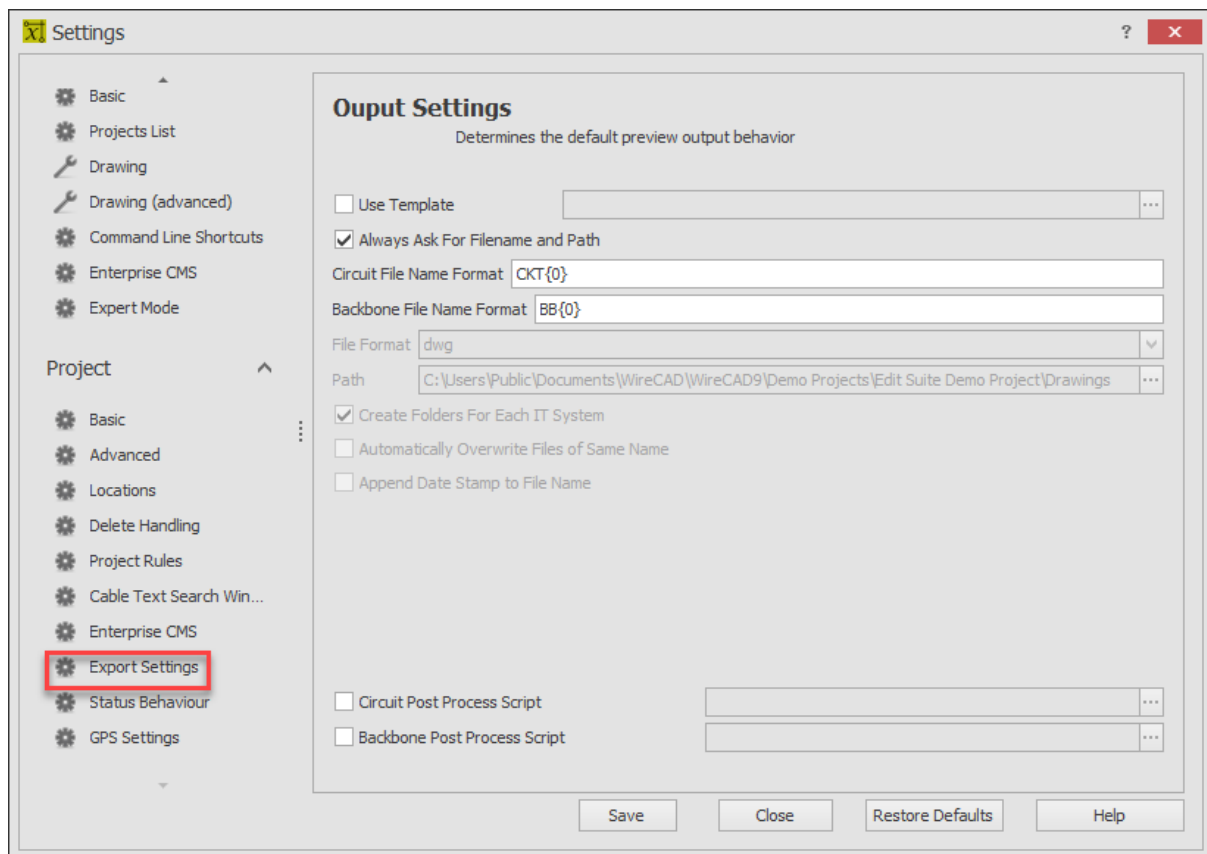
The variable {2} contains the Ordinal Number of the circuit for this jumper.

4.3.3 Output Settings

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Settings and Options](#) [277] >



Menu: **Application Menu > Settings[Project][Export Settings]**



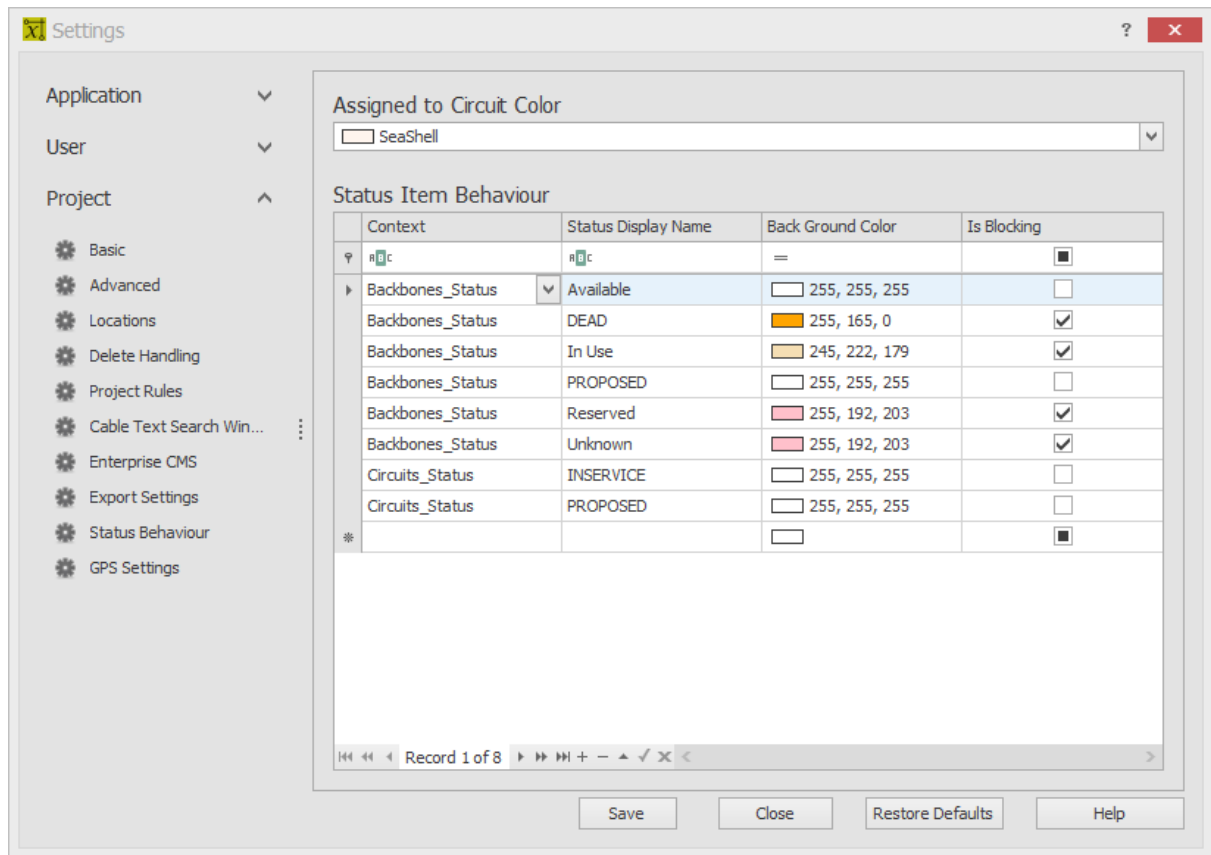
- **Use Template** - Path to a template drawing that the output will be exported into. Your template drawings can contain any page borders or layouts and settings that you wish.
- **Always Ask for Filename and Path** - You are involved in the file name selection and path.
- **Circuit File Name Format** - Sets the file name format for Circuit output. {0} = Circuit Name.
- **Backbone File Name Format** - Sets the file name format for Backbone output. {0} = Backbone Number.
- **File Format** - Presets the output format.
- **Path** - Where do we output.
- **Create Folders for Each IT System** - Create a new folder for each IT System and output the preview to that folder.
- **Automatically Overwrite Files of Same Name** - Self-explanatory.
- **Append Date Stamp to File Name** - Self-explanatory.
- **Circuit Post Process Script** - Path to a c# file which will be run post export but pre write-to-disk. See the Post [Process Scripts](#)^[283] topic for more information.
- **Backbone Post Process Script** - Path to a c# file which will be run post export but pre write-to-disk. See the Post [Process Scripts](#)^[283] topic for more information.

4.3.4 Status Behaviour

Navigation: [WireCAD CMS Tools Manual](#)^[208] > [CMS Settings and Options](#)^[277] >



Menu: **Application Menu > Settings[Project][Status Behavior]**



• **Assigned to Circuit Color** - The color to display in the Backbones grid if the record is assigned to a circuit. Here we see circuit 0232 using the backbone B000003.41.

CktNo	CableType...	CableType	CableNo	SrcSys	DestSys	SRCPin	DestPin	SRCLoc	SRCEI	DestLoc	DestEI	SRConn	DestConn	MultiCore	AvailableC...
	GENERIC	48 STRAND...	B00003.36	AD-3	CC-R54-4	036	036	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.37	AD-3	CC-R54-4	037	037	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.38	AD-3	CC-R54-4	038	038	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.39	AD-3	CC-R54-4	039	039	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
0232	GENERIC	48 STRAND...	B00003.41	AD-3	CC-R54-4	041	041	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.42	AD-3	CC-R54-4	042	042	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.43	AD-3	CC-R54-4	043	043	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.44	AD-3	CC-R54-4	044	044	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
0242	GENERIC	48 STRAND...	B00003.45	AD-3	CC-R54-4	045	045	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
0243	GENERIC	48 STRAND...	B00003.46	AD-3	CC-R54-4	046	046	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.47	AD-3	CC-R54-4	047	047	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	48 STRAND...	B00003.48	AD-3	CC-R54-4	048	048	AMU.BLDG ...		AMU.COM ...		FC	SCA	<input type="checkbox"/>	<input type="checkbox"/>
CableToPrefix: B00004															
	GENERIC	24 STRAND...	B00004.01	AD-4	AD-5	001	001	AMU.BLDG ...		AMU.BLDG ...		ST	ST	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	24 STRAND...	B00004.02	AD-4	AD-5	002	002	AMU.BLDG ...		AMU.BLDG ...		ST	ST	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	24 STRAND...	B00004.03	AD-4	AD-5	003	003	AMU.BLDG ...		AMU.BLDG ...		ST	ST	<input type="checkbox"/>	<input type="checkbox"/>
	GENERIC	24 STRAND...	B00004.04	AD-4	AD-5	004	004	AMU.BLDG ...		AMU.BLDG ...		ST	ST	<input type="checkbox"/>	<input type="checkbox"/>

- **Context** - Backbones_grid or Circuits_grid.
- **Display Name** - The name of our status.
- **Background Color** - Self-explanatory.
- **Is Blocking** - A blocking status is one that blocks operations from proceeding. For example: if we mark a backbone fiber as DEAD we would not want to use that in a Circuit. If the status item is set as Is Blocking = true then the New Circuit tool will not allow a circuit to involve that strand of fiber.

4.3.5 Post Process Scripting

Navigation: [WireCAD CMS Tools Manual](#) [208] > [CMS Settings and Options](#) [277] >



Post Process Scripting of the output of the Visualizers in WireCAD allow you to customize the final appearance of the created document.

What follows are two example scripts, one for the Circuit output and the other for the Backbone output. The process is the same for both. Only the method signature is changed.

Circuits Export Script Example

```

////////////////////////////////////
using System;
using System.Data;
using System.Text;
using System.Windows.Forms;
using System.Diagnostics;
using System.Reflection;
using System.IO;
using WireCAD;
using WireCAD.Interfaces;
using VectorDraw.Professional.vdObjects;
using VectorDraw.Professional.vdFigures;
using VectorDraw.Professional.vdPrimaries;
using VectorDraw.Professional.vdCollections;
using WireCAD.ProjectFile.DAL;
using VectorDraw.Geometry;

/// <summary>
/// Fill your titleblock with your data
/// </summary>
//Your class name
public class TitleBlockFiller
{
    //the method signature is key so this next line must appear as shown
    public static void Run(Workspace ws, Circuits c, vdDocument doc)
    {
        string drawingName = Path.GetFileName(doc.FileName);

        //change this next line to be your titleblock name
        string BlockNameToUpdate = "ansi";

        //Get the layout by name that we want to update
        vdLayout layOut = doc.LayOuts.FindName("ANSI_B");

        //loop through the entities in the layout
        foreach (vdFigure figure in layOut.Entities)
        {
            //if we find a viewport let zoom it to the extents of our model
            if (figure is vdViewport)
            {
                vdViewport vp = figure as vdViewport;
                vp.ZoomExtents();
                vp.Update();
            }

            //now look for our titleblock
            if (figure is vdInsert)
            {
                //cast our base figure into an insert object so we can access its properties
                vdInsert insert = (vdInsert) figure;
                //now we will test to see if the insert name is one that

```

```

        //we know to be a titleblock
        //Modify this to meet your needs
        if (insert.Block.Name.ToLower().Contains(BlockNameToUpdate))
        {
            //need to fix it.
            //this next bit makes it insert get the latest version from the blocks table
            insert.Update();
            insert.Invalidate();

            //we got one so let's set the attributes
            //we will use the safeSetAttributeValue function so that if the attribute doesn't
exist it won't
            //fail
            //Modify this to meet your needs
            //Enter your Attribute names and the values you want to fill them with

            ws.Utilities.SafeSetAttributeValue(insert, "SHEET", layOut.Name);
            ws.Utilities.SafeSetAttributeValue(insert, "Drawing_Name",
Path.GetFileNameWithoutExtension(drawingName));
            ws.Utilities.SafeSetAttributeValue(insert, "DWG_Number", c.CktNO);
            ws.Utilities.SafeSetAttributeValue(insert, "COMPANY", "Slate Gravel Co");
            ws.Utilities.SafeSetAttributeValue(insert, "ADDRESS", "Number 1 Quarry Way");
            ws.Utilities.SafeSetAttributeValue(insert, "ADDRESS2", "Bedrock - The
World");
            ws.Utilities.SafeSetAttributeValue(insert, "DATE",
DateTime.Now.ToShortDateString());
            ws.Utilities.SafeSetAttributeValue(insert, "Scale", "NTS");
            insert.Update();
            insert.Invalidate();
            doc.Redraw(true);
        }
    }

    //Now create a text item that displays our circuit number in the upper righthand corner
    vdText t = new vdText(doc);
    t.TextString = string.Format("CKT - {0}\n{1}", c.CktNO, c.CktDescription);
    t.HorJustify = VectorDraw.Professional.Constants.VdConstHorJust.VdTextHorRight;
    Box bb = layOut.Entities.GetBoundingBox(true, true);
    t.Height = .25d;
    t.InsertionPoint = new gPoint(bb.Right-3, bb.Top-2);
    layOut.Entities.AddItem(t);

    doc.ActiveLayout = layOut;
}
}

```

Backbones Export Script Example

The only difference in the two examples is the method signature. Use the code above replacing:

```
public static void Run(Workspace ws, Circuits c, vdDocument doc)
```

With:

```
public static void Run(Workspace ws, CMSSettings settings, vdDocument doc)
```

4.3.6 GPS Settings

Navigation: [WireCAD CMS Tools Manual](#)  > [CMS Settings and Options](#)  >



Menu: **Application Menu > Settings[Project][GPS Settings]**

See the Reference section regarding the [GPS Settings](#)  panel.

This page is intentionally left blank.
Remove this text from the manual
template if you want it completely blank.

Reference

5 Reference



Here you will find a listing of all menu functions, forms, and dialogs.

5.1 Ribbon Tabs and Dialogs

Navigation: [Reference](#)  >





This section provides a reference to the individual controls in the dialogs accessed in the Ribbon Toolbar.

Only dialogs that need explaining are documented

Please note that although this reference is quite complete, only dialogs that really need explaining are documented. For example, we assume that you can figure out what menu entries like Save and Save As do for yourself.

This also applies to simple dialogs whose functions are explained elsewhere. Some of these dialogs are listed, but only with references to the relevant topics in the Procedures sections.

5.1.1 The Workspace

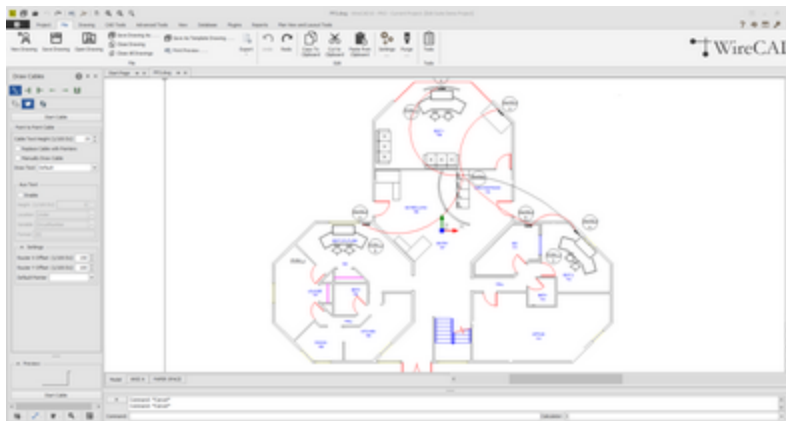
Navigation: [Reference](#)  > [Ribbon Tabs and Dialogs](#)  >



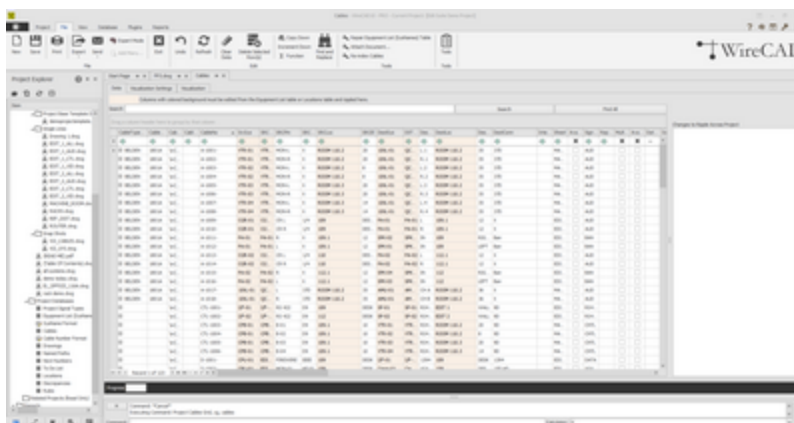
The WireCAD workspace has four main components: The Application Menu at the top of the screen, the Ribbon Toolbar where you access all program functions, the Tool Panels collection for

navigating and managing your projects and the Main Window where you edit the content of your projects. The Main Window can display four distinct environments:

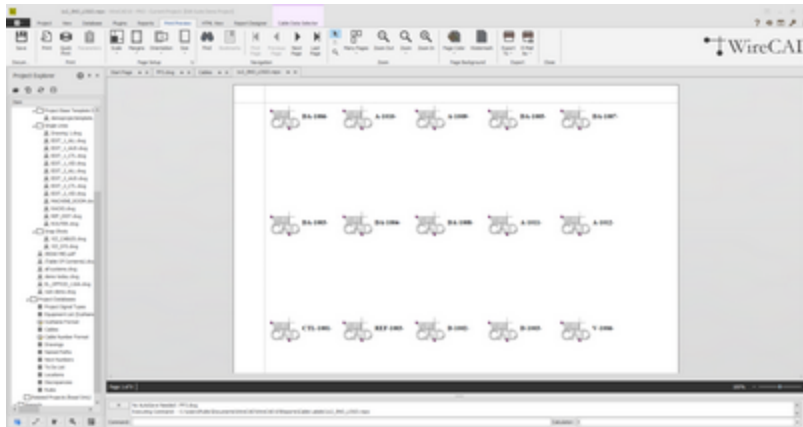
1.The Drawing environment



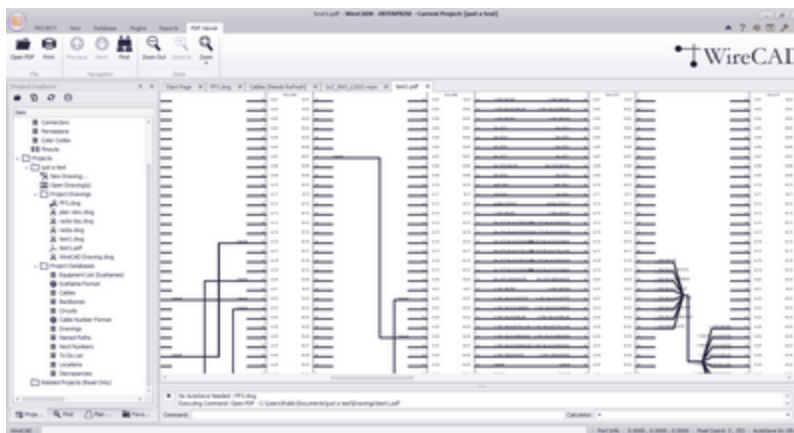
2.The Data environment



3.The Reporting environment



4. The PDF viewer environment



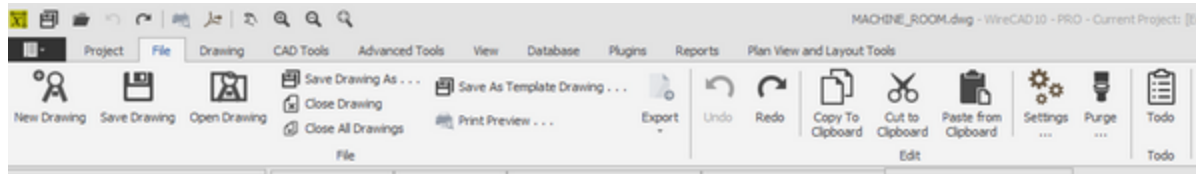
5. The Application Menu and Quick Access Toolbar



The Application Button in the top left corner of the Help & Manual window is one of the most important controls. It provides access to the functions normally accessed in the File menu in menu-based programs. This is where you open existing projects, create new projects, save your projects under other names and so on.

The Quick Access Toolbar next to the Application Button is a place for frequently-used tools.

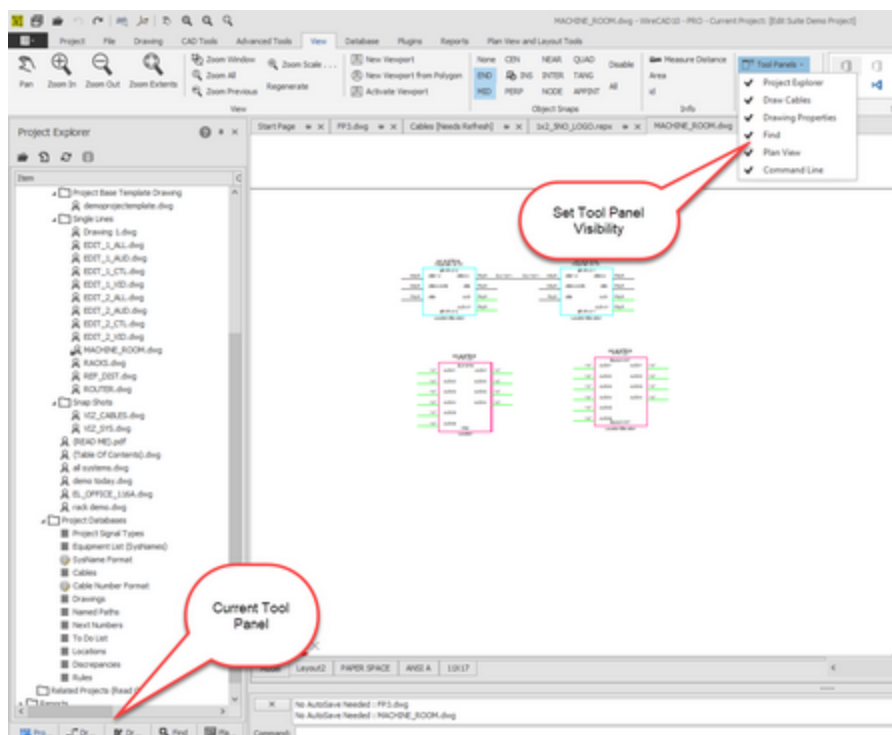
6. The Ribbon Toolbar



The **Ribbon Toolbar** is the control center where you access virtually all of WireCAD's functions. If you use Microsoft Office 2007 you will already be familiar with the Ribbon interface. It is context-sensitive, automatically displaying functions relevant to what you are currently doing.

Tip: The Ribbon can also be operated almost entirely via the keyboard. To display the accelerator keys just press and release the ALT key once the keys will be displayed in icons superimposed on the Ribbon.

7. Tool Panels

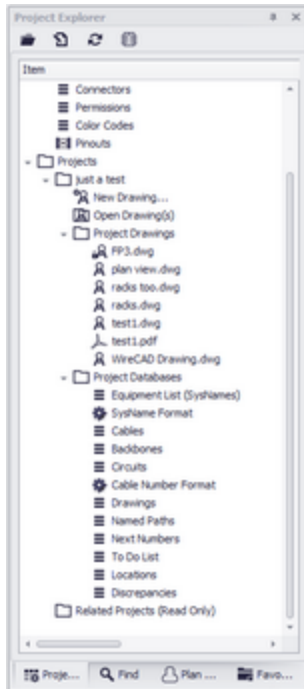


Tool Panels are organizations of controls. There are different Tool Panels depending on the current environment.

8. The Project Explorer Tool Panel



The **Project Explorer** is like Windows Explorer for WireCAD projects. When you load or create a project all its contents are displayed here, including both the Global Equipment tables and the Project Databases and files.

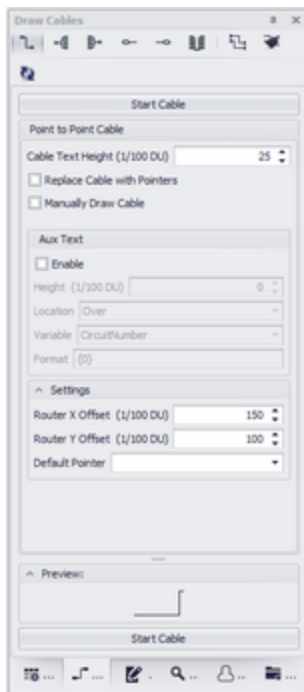


Note: This tool panel is active at all times.

9. The Draw Cables Tool Panel



The **Draw Cables Tool Panel** contains all the tools for drawing cables. You can draw cables with different relationships. You can also set the drawing behavior of cables.

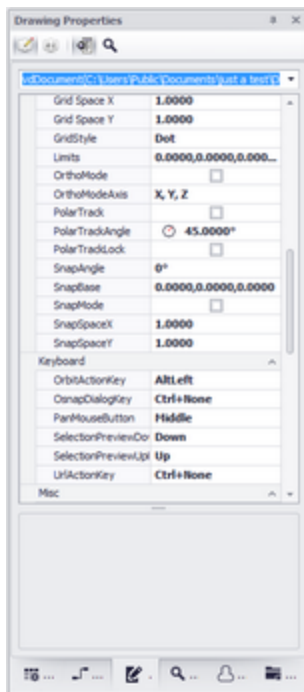


Note: This tool panel is only active when the current environment contains a drawing.

10. The Drawing Properties Tool Panel



The Drawing Properties Tool Panel allows editing of granular drawing settings. If no entity selection exists in the drawing the Drawing Properties window will display the general document properties.

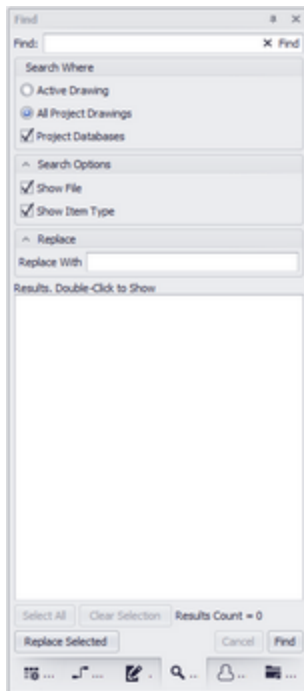


Note: This tool panel is only active when the current environment contains a drawing.

11. The Find and Replace Tool Panel



The **Find and Replace Tool Panel** allows you to search the current drawing, all drawings and the project databases for text and replace that text with your entered values.



Note: This tool panel is active at all times.

12. The Plan View Tool Panel




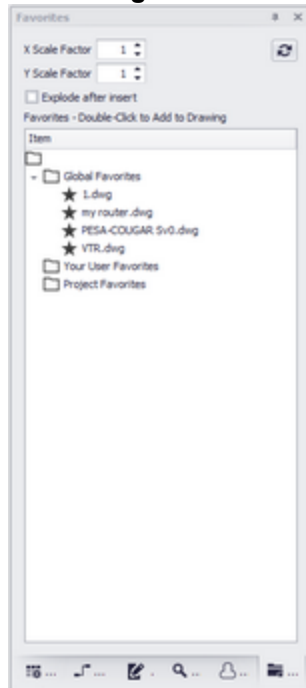
The Plan View Tool Panel allows you to place plan view symbols in your drawing.



Note: This tool panel is only active when the current environment contains a drawing.

13. The Favorites Tool Panel

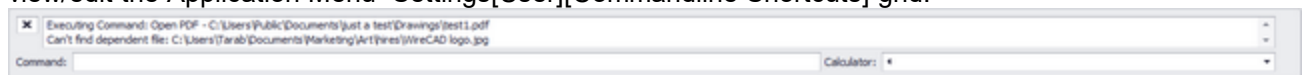
 The Favorites Tool Panel holds blocks that you have saved as favorites (right-click entity in drawing>Add to Favorites).



Note: This tool panel is only active when the current environment contains a drawing.

14. The Command Line Interface

The Command Line Interface allows you to see the command history, receive prompts from the application and execute commands directly. For a full list of command line command you can view/edit the Application Menu>Settings[User][Commandline Shortcuts] grid.



5.1.2 Ribbon Tabs

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ >



5.1.2.1 Application Menu

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >

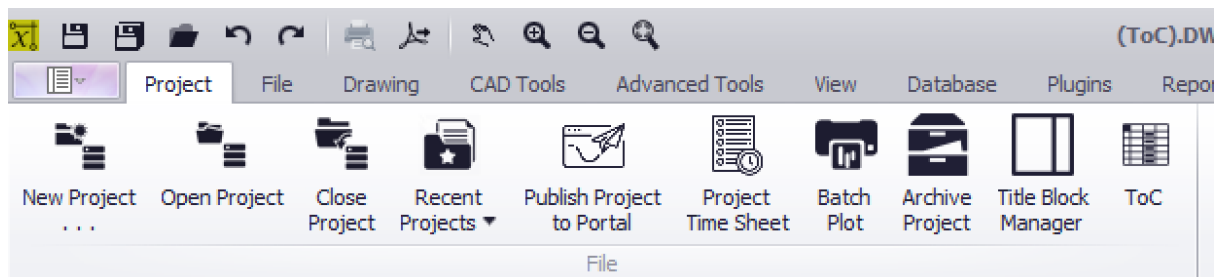


The Application Menu contains the following static menu functions:

- **New Project...** - Displays the [New Project Wizard](#).^[326]
- **Open Project** - Display a file open dialog. Browse to and select a *.wc6plf file to open the WireCAD Project.
- **Save Project As...** - Displays the [Save Project As](#)^[330] dialog. Use this to create a copy of the current project in a different location.
- **Close Project** - Closes the current project.
- **Frequently Used Folders >**
 - Project Folder** - Opens the root folder of the current project in a window explorer window.
 - Global Data Folder** - Opens the global data folder. C:\Users\Public\WireCAD\WireCADx
- User App Data Folder** - Opens C:\Users\<YOUR USER NAME>\AppData\Local\WireCAD\WireCADx
- **Security Tools >**
 - View Permissions** - Shows the permissions grid.
 - Manage Security...** - Shows the [Users Groups and Permissions dialog](#)^[335].
- **Check In/Out >**
 - Pack Up / Check Out** - [Packup/check](#)^[337] out utility. Collects all info for the project and makes is portable.
 - Unpack Project ...** - [Unpack a project](#).^[338]
 - Check In Project** - [Check in a Checked Out project](#)^[339].
- **Project Info** - [Project specific info](#)^[517] like the current revision and related projects list.
- **Account Info** - [Who is the project for](#)^[340].
- **Application Setup** - Displays the [Application Setup Wizard](#)^[341]. This is shown once on startup but you can review and change settings here.
- **Settings** - Displays the main [Settings](#)^[346] dialog.
- **Exit** - Exits the program.

5.1.2.2 Project

Navigation: [Reference](#)^[288] > [Ribbon Tabs and Dialogs](#)^[288] > [Ribbon Tabs](#)^[296] >



The Project Ribbon Tab is a static menu structure. It does not change with environment changes. The Project Ribbon Tab contains the following menu items:

- New Project ...** - Display the [New Project Wizard](#)^[326].
- Open Project** - Display a file open dialog. Browse to and select a *.wc6plf file to open the WireCAD Project.
- Close Project** - Close the current project.
- Recent Projects** - Most recently used projects list. This selection is based on the **Project List** setting in the **Application Menu>Settings[Project List]**.

Batch Plotter - Launch the [Batch Plotter](#)^[200] tool.

Archive Project - Launch the [Archive](#)^[381] tool.

Titleblock Manager - Launch the [Titleblock Manager](#)^[382] tool to assign common variables to all of your title blocks in the project.

ToC - Launch the [Table of Contents](#)^[385] generator that creates a Table of Contents from the data in the project Drawings table.

Publish Project to Portal - Launch the [Publish Project to Portal](#)^[380] tool.

Other

While WireCAD does not currently merge any other tools with the Project menu, authors of WireCAD plugins have the ability to merge their own tools onto this ribbon tab. Consult their documentation for information.

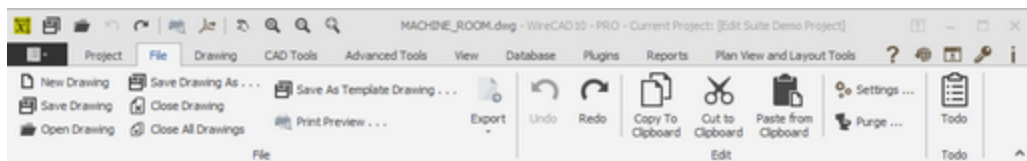
5.1.2.3 File

Navigation: [Reference](#)^[288] > [Ribbon Tabs and Dialogs](#)^[288] > [Ribbon Tabs](#)^[296] >



The File Ribbon Tab is a dynamic menu that changes its structure with the current environment.

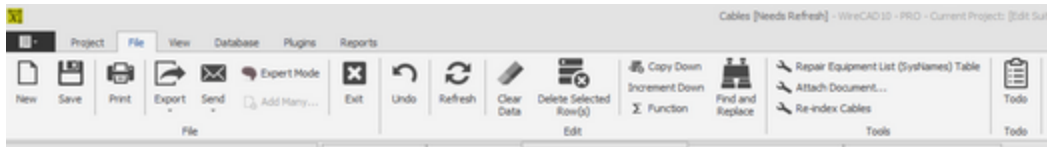
Drawing Environment



Again, we will not explain the obvious ones.

- **New Drawing ...** - Shows the [New Drawing Wizard](#)^[389].
- **Save As Template Drawing** - Any drawing may be used as a template from which other drawings may be created. Simply **Save As Template Drawing** to make your drawing usable in the **New Drawing Wizard** as a starter drawing.
- **Print Preview ...** - Display the [Print Preview](#)^[390] dialog.
- **PDF Export ...** - Display the [PDF Export](#)^[392] dialog.
- **Export to Visio** - [Create smart Visio drawings](#)^[393] where the WireCAD cable becomes live active Visio connections.
- **Settings ...** - Display the main [Settings](#)^[346] dialog. Same as clicking **Application Menu>Settings**.
- **Purge ...** - [Purge](#)^[408] unused entities from the drawing.
- **Todo** - Create a [New Todo](#)^[497] linked to this drawing.

Data Environment



For an general discussion on grid basics [see this topic](#) ⁴⁹¹.

- **New** - Create a new item on the collection. The dialog presented will differ based on the collection.
- **Clear Data [Del]** -
- **Delete Selected Rows [Ctrl + Del]** -
- **Copy Down [Ctrl]+[D]** -
- **Increment Down [Ctrl]+[I]** -
- **Find and Replace** -
- **Function** - Displays the [Expression Builder](#) ⁴⁹¹ dialog so you can manipulate data using the Expression engine.
- **Tools** - See Grid Specific Functions for a listing of the tools available to each data grid.

Other

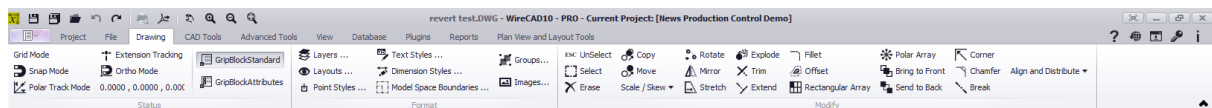
While WireCAD does not currently merge any other tools with the File menu, authors of WireCAD Plugins have the ability to merge their own tools onto this ribbon tab. Consult their documentation for information.

5.1.2.4 Drawing

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >



Active only when the current environment is a drawing.



Grid Mode - Toggle the visible grid **[F7]**. Note: the visible grid is rendered only for the **Drawing Limits** which can be set in the Drawing Properties Tool Panel.

Snap Mode - Toggle the invisible snap grid **[F9]**. The invisible snap grid is based on the **SnapX** and **SnapY** settings in the Drawing Properties Tool Panel.

Polar Track Mode - Toggle PT mode. When enabled and in a function requesting a point from the drawing the selectable polar angle will be restricted to the **Polar Track Angle** in the Drawing Properties Tool Panel.

Extension Tracking -

Ortho Mode - Toggle restrict movement to horizontal/vertical only.

Coordinate - Displays the cursor position in current coordinates. Coordinates are expressed in drawing units (DU). Drawing units do not express particular units (meters, inches etc).

In this part user have to make some assumptions in order to define that the coordinates of the drawing mean particular units (meters, inches etc).

For example:

For a mechanical drawing we can make the assumption for example: where one drawing unit defines one millimeter(1 D.U=1mm).

For a architectural/technical drawing we can make the assumption for example: where one drawing unit defines one meter(1 D.U=1m) or one foot.

This can be very helpful in designing,dimensioning,retrieving information from the drawing(distances,area calculations).

GripBlockStandard - Show only one grip per insert.

GripBlockAttributes - Show one grip per attribute in the selected insert. Grips can be used to reposition the attribute if the Attribute is set to LockPosition = false.

Layers ... - Shows the [Layers](#) ^[394] dialog.

Layouts ... - Shows the [Layouts](#) ^[395] dialog.

Point Styles ... - Shows the [Point Styles](#) ^[397] dialog.

Text Styles ... - Shows the [Text Styles](#) ^[397] dialog.

Dim Styles ... - Shows the [Dimension Styles](#) ^[399] dialog.

Model Space Bounds - Shows the [Model Space Boundaries](#) ^[400] dialog.

Groups ... - Shows the [Groups](#) ^[401] dialog.

Images ... - Shows the [Images](#) ^[402] dialog.

UnSelect - Clears the current selection. Same as pressing {esc}.

Select - Start a selection.

Erase - Erase the current selection. If no selection then you will first be prompted to select entities.

Copy - Copy the current selection. This is a base-point or offset copy and is preferred over [Ctrl][C] then [Ctrl][V] if copying in the same drawing as it is lighter weight. If no selection then you will first be prompted to select entities.

Move - Move the current selection. If no selection then you will first be prompted to select entities.

Scale / Skew - Scale the current selection. If no selection then you will first be prompted to select entities.

Scale / Skew >Scale (uniform) - Scale the X and the Y directions to the same scale factor.

Scale / Skew >Scale To (uniform) - Scale from the current size to a specific number. Allows you to scale without doing the math to calculate the scale factor.

Scale / Skew >Scale X - Scale the entities in the selection set in the horizontal direction only.

Scale / Skew >Scale Y - Scale the entities in the selection set in the vertical direction only.

Scale / Skew >Skew X - Skew the entities in the selection set in the horizontal direction.

Scale / Skew >Skew Y - Skew the entities in the selection set in the vertical direction.

Rotate - Rotate the current selection. If no selection then you will first be prompted to select entities.

Mirror - Mirror the current selection. If no selection then you will first be prompted to select entities.

Stretch - Stretch polylines in the current selection. If no selection then you will first be prompted to select entities.

Explode - Explode the current selection. This function will reduce complex object to their primitive constituent parts. For example; exploding an insert unlinks all of the entities back one level. Exploding a polyline reduces the polyline to a collection of line segments. If no selection then you will first be prompted to select entities.

Entities that can be exploded:

- vdDimension explode to vdLines, vdText(s), and vdInserts (the arrows).
- vdInsert explode to the entities that is consist of. (If there are Inserts inside Blocks then you may need to apply more than 1 explode to get the base entities).

- vdPolyHatch explode to vdPolyline(s).
- vdPolyline explode to vdLine(s) and/or vdArc(s).
- vdRect explode to vdPolyline.
- vdText explode to vdPolylines(only texts with fontfile SHX).TTF(true type font) texts are not exploded.

Trim - Trim objects at a cutting edge defined by other objects the current selection. 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 do not function if the objects do not intersect.

Extend - Extend lines,arcs,polylines until they intersect with some other object which is used as limit of the extension. Firstly you have to select the objects that consist the limits of the extension. Then you have to choose a point at an object that you want to extend. If the object you want to extend does not intersect with above objects then nothing will happen.

Fillet - Connect two lines , two arcs, or one arc with a line(these two objects must have at least one common point either visible, or in their extension) , with an arc with a specific radius. The value of the radius has some restrictions depending the position of the objects. If radius=0 then simply the objects are either extended until they intersect each other in one point (if there was not an intersection point) either trimmed (if an intersection point is visible).

Offset - Create a new object in parallel direction and in a specified distance from the original object which is used as pattern for the new object. When you execute an offset command, you are prompted to select an object. Then you have to specify the offset distance which is the distance that the new object will be drawn from the original object. Then you have to set the side that the object will be drawn because there are two sides.

Rectangular Array - Creates multiple copies of objects in a rectangular pattern. Shows the [Rectangular Array](#) dialog. With the rectangular array you can create an array defined by a number of rows and columns of copies of the selected object. First you have to select the objects. Then you have to define number of rows and number of columns of the rectangle, the distance between rows and the distance between columns. If no selection then you will first be prompted to select entities.

Polar Array - Same as R Array but copies radially around a center point. Select the objects to copy. Then set the center point and next define the numbers of the copy objects that will be created and the fill angle. Lastly choose if the object will be rotated or not. [See the Polar Array dialog here](#).

Bring to Front - Bring to front. This changes the Z order of the objects in the render engine bringing the current selection to the front. If no selection then you will first be prompted to select entities.

Send to Back - Send the current selection to the back of the Z order. If no selection then you will first be prompted to select entities.

Corner - Causes lines that can intersect to intersect. Start the command then select the first line then the second line.

Chamfer - Chamfer the polyline, line, distance or angle object at two intersecting points by the entered distance.

Break - Break a line, polyline, circle, arc, etc. based on the selected distance.

A lign and Distribute

The align functions require a selection set with at least two entities selected. The first item in the selection is used as the reference. The others in the selection are then aligned to the reference point.

If the two selected entities are WireCAD blocks with cables attached, the cables will be rerouted.

Align and Distribute>Align Lefts - Left edges.

Align and Distribute>Align Rights - Right edges.

Align and Distribute>Align Tops - Top edges.

Align and Distribute>Align Bottoms - Bottom edges.

Align and Distribute>Align Middles - Middles (vertically).

Align and Distribute>Align Centers - Centers (horizontally).

The distribute functions require a selection set with at least three entities. The entities can be distributed using uniform spacing between entities or center-to-center spacing of the entities. The eyeball tends to see the space between the entities especially if the entities are not uniform in size.

Align and Distribute>Distribute Y Centers - Distribute the selection vertically based on center-to-center spacing.

Align and Distribute>Distribute Y Spacing - Distribute the selection vertically based on uniform space between entities.

Align and Distribute>Distribute H Centers - Distribute the selection horizontally based on center-to-center spacing.

Align and Distribute>Distribute H Spacing - Distribute the selection horizontally based on uniform space between entities.

Align and Distribute>Increase Y Spacing - Increase the vertical spacing. You will be prompted for the spacing distance.

Align and Distribute>Decrease Y Spacing - Decrease the vertical spacing. You will be prompted for the spacing distance.

Align and Distribute>Increase X Spacing - Increase the horizontal spacing. You will be prompted for the spacing distance.

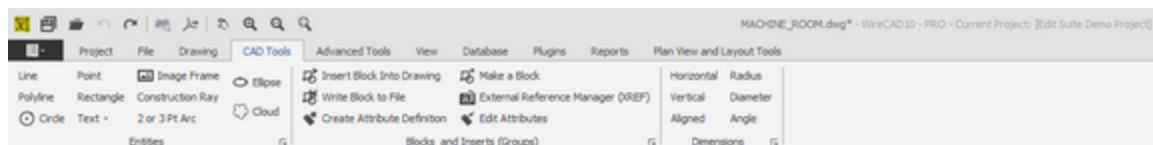
Align and Distribute>Decrease X Spacing - Decrease the horizontal spacing. You will be prompted for the spacing distance.

5.1.2.5 CAD Tools

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >

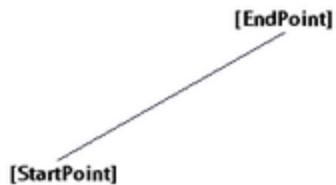


Active only when the current environment is a drawing.



Entities Page Group

•**Line** - A single line segment. Lines can be one segment or a series of connected segments, but each segment is a separate line object.



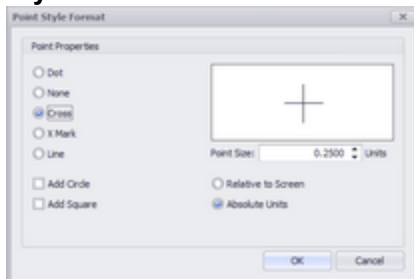
•**Polyline** - 2D/3D line composed of line and arc (bulges) segments. Polyline is specified by an array of Vertexes (points). When the Polyline has thickness then the extrusion vector of the polyline defines the direction of the thickness. Polylines can be Open or Closed, can be SPLine and be filled with a color or a hatch.



•**Circle** - A full circle is defined by its CenterPoint and its Radius. Circle is drawn in the plane that is defined by CenterPoint and ExtrusionVector.



•**Point** - An object which occupies a single point in the coordinate system. The display of the point is inherited from the document Point Style property and can be set by clicking **Drawing>Point Styles...**



•**Rectangle** - A rectangular object defined by an insertion point, height and width.


**•Text >**

Single Line of Text - Text inherits its style from the document **Text Styles** collection which can be edited by clicking **Drawing>Text Styles...** Text entities do not have font properties directly but inherit the font and font properties from the associated Text Style.



Multiline Text (MTEXT) - A multiline text entity.




Alphanumeric Sequence - Create a sequence of text entities. See Alphanumeric Sequence [dialog here](#). 

FRED-1

FRED-2

FRED-3

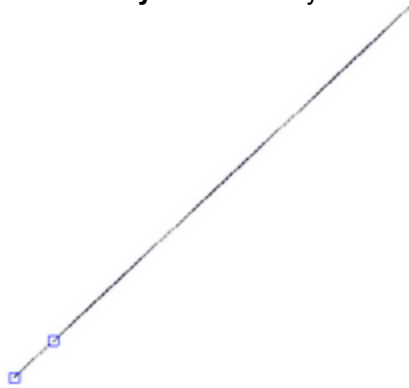
FRED-4

Text Increment - Modify an existing selection of text entities with a sequence of numbers. For more information see [Text Increment](#)  dialog here.

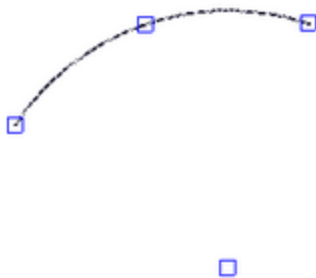
•Image Frame - Used to render images in the document. Images may either be linked or imbedded in the drawing file. The image frame defines the location and size of the displayed image. To imbed an image you must edit the document Images collection **Drawing>Images**.



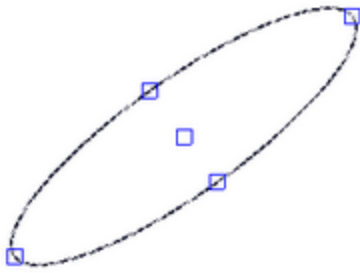
- **Construction Ray** - A line entity defined by its starting point and an angle.



- **2 or 3 Pt Arc** - A circular arc 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. The ExtrusionVector is always vertical to the arc. Arc is drawn in the plane that is defined by CenterPoint and ExtrusionVector.



- **Ellipse** - Draw an Ellipse.



- **Cloud** - Draw a cloud.



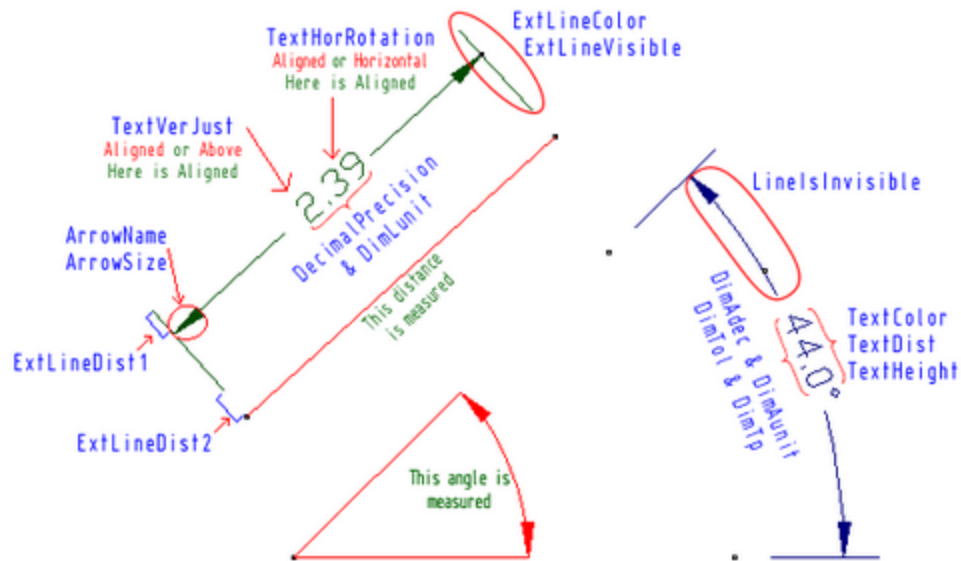
Blocks and Inserts Page Group

- **Insert Block Into Drawing** - Opens the [Inserts](#) ^[408] dialog. You can then insert an existing block into the drawing or browse to another file to insert in the drawing.
- **Write Block to File** - Writes the selected insert out to its own file prompting you for the file save as name and location as well as the new basepoint for the drawing.
- **Create Attribute Definition** - A tag/value data pair that can be included in a block to display editable text. See the [Add Attribute Definition](#) ^[414] dialog.
- **Make a Block** - Select entities and group them into a unit called a block. The block resides in the document Blocks table. To display a block we Insert an instance of the block into the drawing at some point, scale, and rotation.
- **External Reference Manager** - Other drawings may be displayed within the drawing space and maintained as separate files. These are called Externally Referenced (XREFS) drawings. The externally referenced drawing cannot be edited in the current drawing, only viewed, positioned, scaled and rotated. This function opens the [External Reference Manager](#) ^[409] dialog.
- **Edit Attributes** - Allows editing of the attributes of the selected insert of a block. Here you can adjust the display value, style, color. The dialog also supports updating other instances of the same insert, or all inserts with matching Tags. [See here](#) ^[413].

Dimensions Page Group

Dimensioning is the process of adding measurement annotation to a drawing. There are 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. There are three basic types of dimensioning: linear, radial, and angular. Dimensions can be horizontal, vertical, aligned, rotated, and angular. A linear dimension measures 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.

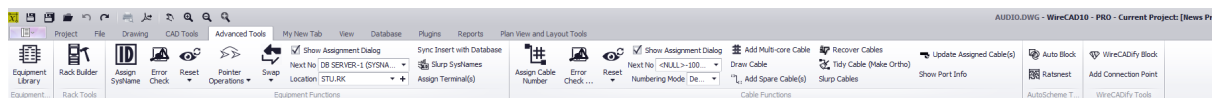


5.1.2.6 Advanced Tools

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >



Active only when the current environment is a drawing.



Equipment Library page group

Equipment Library - Opens the [Equipment Library](#) ⁴²⁶.

Rack Tools page group

Rack Builder - Opens the [Rack Builder](#) ⁴⁴⁸ tool.

Equipment Functions page group

Assign SysName - Assigns a [SysName](#) ^[449] to all selected inserts that are assignable.

Error Check - Shows the [Drawing SysName Error Check](#) ^[456] dialog.

Reset>Reset All Systems - Reset will remove the SysName data from the insert and do nothing with the database. The Reset functions are useful if you need to reuse a drawing and reassign new numbers.

Reset>Reset Selected Systems - Reset will remove the SysName data from the insert and do nothing with the database. The Reset functions are useful if you need to reuse a drawing and reassign new numbers.

Reset>De-Assign SysName - De-Assign will remove the SysName data from the insert and mark the record in the database deleted/available. See also: [Project Delete Handling](#) ^[366] settings.

Pointer Operations>Link Selected Pointers - Allows you to link a selection of pointers with another selection of pointers.

Pointer Operations>Rereference Drawing Pointers (change sheet reference name) - Allows you to change the name of the sheet to which pointer(s) are pointed.

Pointer Operations>Error Check Pointers - Launches the Error Check Pointers [dialog](#) ^[469]. This tool tests the links between pointers. It does not test the connection to ports, nor does it test the validity of the connections, only whether the pointer is properly in both direction.

Swap>Swap Equipment - Allows you to change your equipment out for something else. See the [Swap dialog here](#) ^[411].

Swap>Swap All Pointers - Allows you to change your pointer type if the current pointer displeases your eye. See the Swap Pointers [dialog here](#) ^[412].

Swap>Swap All Terminals - Allows you to change your terminal type if the current terminal displeases your eye. See the Swap Terminals [dialog here](#) ^[412].

Show Assignment Dialog - When checked displays the [SysName Assignment](#) ^[449] dialog. Otherwise the assignment proceeds with all default information.

Next No - Dropdown to set the next number of the selected sequence.

Location - Dropdown to set the default Location. You can also add a new Location with the [+]. This presets the Location field in the New [SysName assignment](#) ^[449] dialog.

Sync Insert with Database - Checks the selected insert to see if the SysName, Alias, Manufacturer, EquipmentName, Location and User fields match the Equipment List. If they do not you will be presented with the [Sync Equipment Decision](#) ^[419] dialog.

Slurp SysNames - Gets all of the SysName data from the drawing and adds it to the database if not found in the database.

Assign Terminal(s) - Starts the terminal assignment dialog for the current selection of assignable terminals.

Cable Functions page group

Assign Cable Number - [Assigns the cable](#) ^[451] a number based on several factors:

[The Project Settings\[Starting Number\]](#) ^[364].

[Project Cable Number Format](#) ^[125] tool.

[Next Number](#) ^[548] sequence.

Show Cable Assignment Dialog - See below.

The process of assignment will update drawing with the number as well as place an entry in the project **Cables** table.

Error Check - Shows the [Error Check](#) ^[457] dialog.

Reset>Reset All

Reset>Reset Selected Cables - Removes the Cable data from the drawing and do nothing to the database. The Reset functions are useful if you need to reuse a drawing and reassign new numbers. This function does not work across pointers.

Reset>De-Assign Cable Number - De-Assign will remove the Cable data from the drawing and mark the record in the database deleted/available. This function works across pointers. In other words if the cable that is being de-assigned is pointed to another cable/drawing the complementary pointer is de-assigned as well. See also: [Project Delete Handling](#) ^[366] settings.

Show Assignment Dialog - When checked displays the [Cable Assignment](#) ^[451] dialog. Otherwise the assignment proceeds with all default information.

Next No - Dropdown to set the next number of the selected sequence.

Numbering Mode - Numbering sequence mode (**Default** (no skipping), **Evens**, **Odds**, **Fives**, **Tens**). Use to skip numbers.

Add Multicore Cable - Displays the [Add Multi-core Cable](#) ^[456] dialog. There you select the cable type number base, etc. and build an entry in the **Cables** table for every core in the associated cable type marking the records available.

Draw Cable - Starts the [Draw Cable](#) ^[567] function.

Add Spare Cable(s) - Add spare cable(s) that do not connect to anything yet but can have. See the [Add Spares](#) ^[467] dialog.

Recover Cables - The Recover Cables function does the following:

- Gets all cable data from the drawing.
- Compares it to the Cables table of the project.
- Updates the Cable Number information in the drawing for matches. A match is found when the source and destination SysNames, Locations, and Port names match.

Tidy Cable - Forces the selected cable to be orthogonal (horizontal or vertical only).

Slurp Cables - Gets all of the Cable data from the drawing and adds it to the Cables table if not found in the database.

Update Assigned Cable(s) - Ensure that the database matches the currently attached ports. Can be used after a cable has been moved to another port.

Show Port Info - Click a port to show all port data Name - Connector - Type.

AutoScheme Tools page group

Auto Block - Shows the [Auto Block](#) ^[458] dialog. With Auto Block you can place functional blocks in the drawing on a grid simply by selecting which SysName to place.

Ratsnest - Shows the [Ratsnest](#) ^[461] dialog. Used in conjunction with Auto Block to wire the placed blocks. Pulls cable data from the Cables table, searches the drawings for matches and when found places a cable.

WireCADify Tool page group

WireCADify Block - Shows the [WireCADify Block](#) ^[463] dialog. Add the WireCAD attribute set to your CAD blocks. You do not need to use this command on any WireCAD generated block.

Add Connection Point - Shows the [Add Connection Point](#) ^[464] dialog. Add WireCAD Connection Points to blocks that you have WireCADified. You do not need to use this command on any WireCAD generated block.

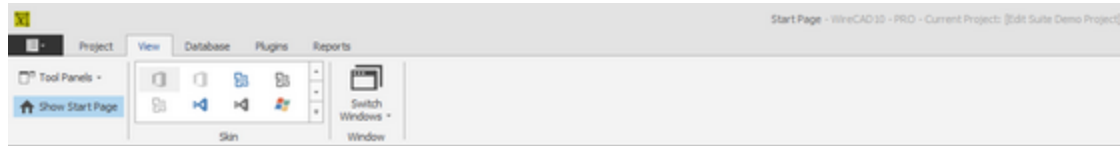
5.1.2.7 View

Navigation: [Reference](#) ^[288] > [Ribbon Tabs and Dialogs](#) ^[288] > [Ribbon Tabs](#) ^[296] >



The View tab is dynamic. It merges tools from the current environment.

Static View



•Tool Panels >

Project Explorer - Toggle the visibility of the [Project Explorer](#) ^[562] Tool Panel.

Draw Cables - Toggle the visibility of the [Draw Cables](#) ^[567] Tool Panel.

Drawing Properties - Toggle the visibility of the [Drawing Properties](#) ^[568] Tool Panel.

Find and Replace - Toggle the visibility of the [Find and Replace](#) ^[569] Tool Panel.

Plan View Tools - Toggle the visibility of the [Plan View Tools](#) ^[571] Tool Panel.

Favorites - Toggle the visibility of the [Favorites Tool](#) ^[572] Panel.

Command Line - Toggle the visibility of the [Command Line](#) ^[573] Tool Panel.

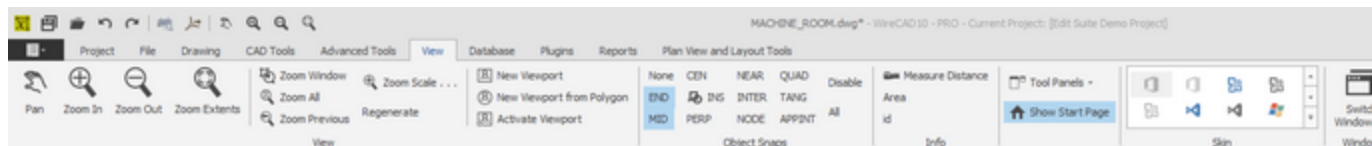
There may be others depending on the plugins that are loaded. These are the default tool panels.

•**Show Start Page** - Toggle the visibility of the Start Page.

•**Skin Gallery** - Select the look and feel of the application.

•**Switch Windows - {Current Window Set}** - Toggle the active window.

Drawing Environment View



View page group

•**Pan** - Reposition of the view. This can be better done with the mouse wheel of your mouse. Depress the mousewheel to click the button underneath it to activate the PAN mode. NOTE: this relies on your mouse driver middle button set to default.

•**Zoom Commands** - Zoom command allows the user to increase or decrease the apparent size of objects , so the user can control the part of the drawing that is included in the screen. Zoom commands are transparent. Transparent commands are commands that can be invoked when another command is active.

Remarks:

There are several ways to execute the zoom command:

"E"(Extends) Zooms to display the drawing extents

"P"(Previous) Zooms to display the previous view

"W"(Window) Zooms to display an area specified by two opposite corners of a rectangular window.

User must specify these two corners.

"A"(All) zooms to the drawing limits or current extents, whichever is greater.

•**Zoom Scale** - Show the Zoom Scale dialog. Zooms the display at a specified scale factor. 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.

•**Regenerate** - Re render the entire drawing.

•**New Viewport commands** - Viewports are areas that display different views of your model. As you work, you can split the drawing area into one or more adjacent rectangular views known as model viewports. In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Also, errors you might miss in one view may be apparent into others. ViewPorts are treated as rectangle drawing objects which display views and can be moved or resized. They can be created only in a layout and not in Model space. You can also attach viewports to closed polygons (polylines, circles, ellipses, rectangles).

Object Snaps page group

Object Snaps - OSnaps provide an easy way to pick geometric points from the drawing.

OSnap	Function
End	End Point snap
Mid	Mid Point snap
Center	Center snap
Ins	Insertion Point snap
Perp	Perpendicular To snap
Near	Nearest snap. This one overrides all others.
Inter	Intersection snap
Node	Point snap
Quad	Quadrant snap
Tang	Tangential To snap
AppInt	Apparent Intersection snap

Disable	Toggle currently selected OSnap state
All	Enable/Disable all.

Info page group

- **Measure Distance** - Measure the distance in Drawing Units(DU) between two points selected from the drawing.
- **Area** - Measure the area of a selected object.
- **ID** - Get the drawing ID of the selected object.

Other

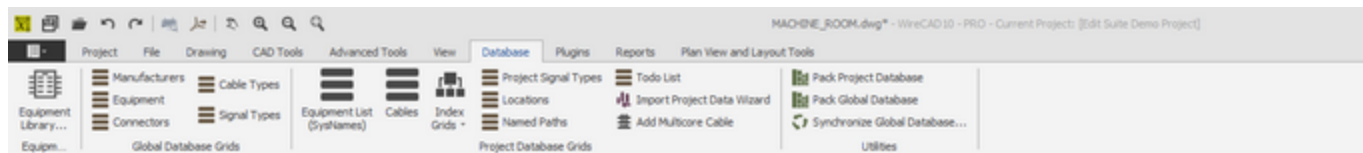
While WireCAD does not currently merge any other tools with the View menu, authors of WireCAD plugins have the ability to merge their own tools onto this ribbon tab. Consult their documentation for information.

5.1.2.8 Database

Navigation: [Reference](#) ^[288] > [Ribbon Tabs and Dialogs](#) ^[288] > [Ribbon Tabs](#) ^[296] >



No environments currently merge with this tab. This is the default view but Plugins may register additional commands.



Equipment Library page group

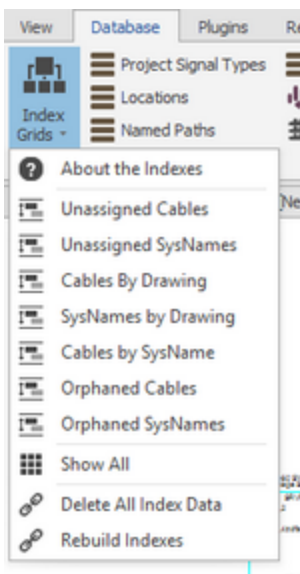
- **Equipment Library** - Show the global [Equipment Library](#) ^[428].

Global Database Grids page group

- **Manufacturers** - Show the global [Manufacturers](#) ^[519] Grid.
- **Equipment** - Show the global [Equipment](#) ^[520] Grid.
- **Connectors** - Show the global [Connectors](#) ^[522] Grid.
- **Cable Types** - Show the global [Cable Types](#) ^[523] Grid.
- **Default Signal Types** - Show the Default [Signal Types](#) ^[521] Grid. **This grid defines many default behaviors like cable color and default cable type.** For more information see this topic on [Signal Types](#) ^[59].
- **Pinouts** - Show the global [Pinouts](#) ^[525] tool.

Project Database Grids page group

- **Equipment List** - Show the project [Equipment List](#)^[532]. This is the collection of all SysNamed equipment across all drawings in the project.
- **Backbones** - Show the project **Backbones** grid. (ENT ONLY). This is the collection of all **Backbones** in the project.
- **Circuits** - Show the project **Circuits** grid. (ENT ONLY). This is the collection of all **Circuits** in the project.
- **Cables** - Show the project [Cables](#)^[534] grid. This is the collection of all **Cables** in the project.
- **Index Grids** > For more information see [this topic](#)^[133]. This sub menu will show any or all of the index grids and allow you to delete and rebuild the indexes.



- **Project Signal Types** - Show the [Project Signal Types](#)^[531] grid. This grid has priority for **Signal Types**. If a signal type is not found in this database it is pulled from the [Default Signal Types](#)^[521] grid.
- **Locations** - Show the project **Locations** grid. This is the collection of all **Locations** in the project.
- **Named Paths** - Show the project **Named Paths** grid. This is the collection of all **Named Paths** in the project.
- **Todo List** - Show the project Todo List.
- **Import Project Data Wiz** - Show the project **Import Data Wizard**. This tool allows you to import cable and equipment data from other sources.
- **Add Multicore Cable** - Displays the [Add Multi-core Cable](#)^[456] dialog. There you select the cable type number base, etc. and build an entry in the Cables table for every core in the associated cable type marking the records available.

Utilities page group

- **Pack Project Database** - This file base database utility will compress and reorganize your project database.
- **Pack Global Database** - This file base database utility will compress and reorganize your global database.

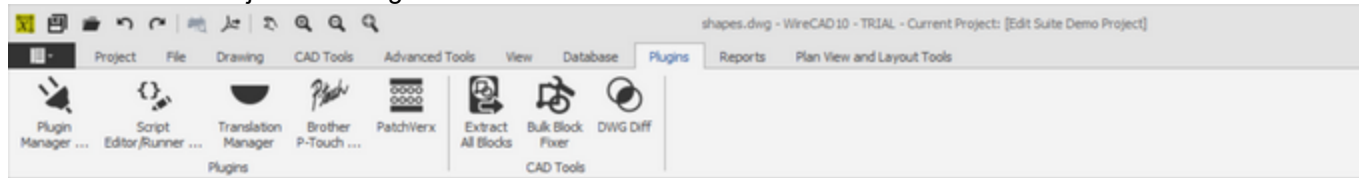
- **Synchronize Global Database** - Show the global database [Synchronizer](#)^[488] tool. This tool allows you to import/export/merge two global databases of either SQL, VISTA, or SQL Azure.

5.1.2.9 Plugins

Navigation: [Reference](#)^[288] > [Ribbon Tabs and Dialogs](#)^[288] > [Ribbon Tabs](#)^[296] >



This tab is dynamic. Depending on product level and installed third party or WireCAD plugins these menu items are subject to change.



Plugins page group

- **Plugin Manager ...** - Shows the [Plugin Manager](#)^[508] dialog where you can control which plugins are loaded.
- **Script Editor Runner** - Shows the [Script Editor/Runner](#)^[511] dialog. This tool allows you to create custom scripts to perform tasks. You can view/edit existing scripts that show how to do a bunch of things by browsing to the script folder in c:\users\public\wirecad\wirecad\scripts. Therein you will find many example scripts. Scripts are written in C# and the editor provides intellisense hints.
- **Translation Manager** - Shows the [Translation Manager](#)^[194]. While not technically a plugin we have always shown it here so why break with tradition. All text (strings) in WireCAD are held in a dictionary. The dictionary is editable through the Translation Manager. All strings default to English if the machines current culture cannot find a translation in the dictionary. You have the ability to change the displayed messages, column headings, and all other strings.
- **Batch Plot** - Show the [Batch Plot](#)^[200] tool. This tool allows you to scan the drawings for layouts and select the layouts to print/plot in a batch. You can save/load your settings for the next time around.
- **Brother P-Touch!** - Show the Brother P-Touch! plugin to print directly to P-Touch! printer with USB ports that support direct print. (NOTE: not all P-Touch! printers with USB ports support direct print).
- **PatchVerx** - If you have installed PatchVerx on your machine this icon will be present so you can run the worlds best patchbay designation strip tool inside of WireCAD and pull data directly from your Cables table and place in on your patchbay label.

CAD Tools page group

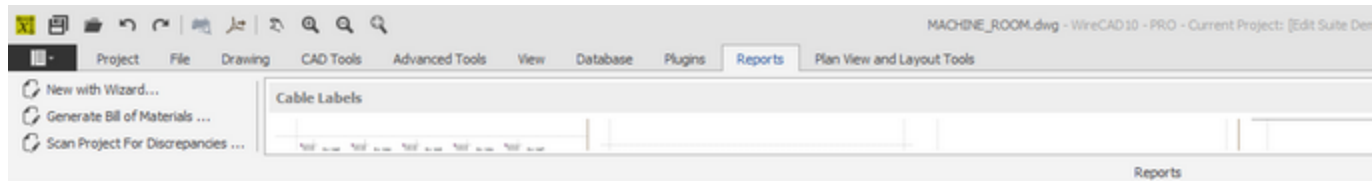
- **Extract All Blocks** - [Extracts all blocks](#)^[196] from the active drawing and writes them out as individual dwg files to the selected folder.
- **Bulk Block Fixer** - If you want to use someone else's CAD blocks you will need to fix them so they work with WireCAD. This is the tool.
- **DWG Diff** - Drawing Differencing tool. Compare two similar drawings and create three views A not B, B not A, and Common to Both. Compares geometry and drawing collections like layers, linetypes, etc.

5.1.2.10 Reports

Navigation: [Reference](#) ^[288] > [Ribbon Tabs and Dialogs](#) ^[288] > [Ribbon Tabs](#) ^[296] >



Items on this tab are static. Whether or not the gallery displays is based on the **Application Menu>Settings[User][Basic]ReportShowMode**.



Reports page group

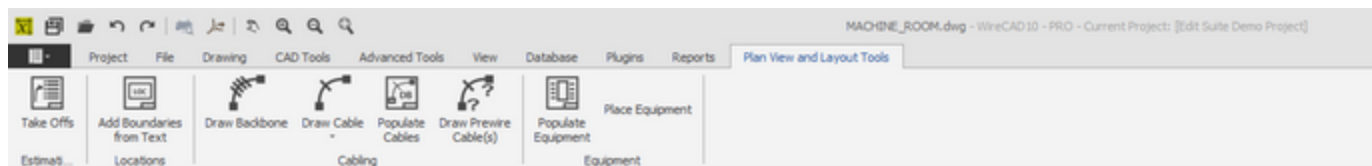
- **New with Wizard...** - Create a new report using the New Report Wizard. We recommend that you find a report in the existing Reports Gallery that is close and save that as your new name then edit your changes there. You will save a bunch of time.
- **Generate Bill of Materials** - Shows the [BOM generator](#) ^[556] which counts equipment, cable, and connectors in the project. NOTE be sure to run this tool before you run any of the BOM reports or they will be empty.
- **Scan Project for Discrepancies** - Scan for common problems. Shows the [Discrepancy Scanner](#) ^[506] dialog. When it finishes you can see the results in the **Project Discrepancy** list from the **Project Explorer**.
- **Reports Gallery** - Lists all reports in the **%REPORTS%** support path and its subdirectories. Clicking a gallery item will load the report into a report form for preview/design/export.

5.1.2.11 Plan View and Layout Tools

Navigation: [Reference](#) ^[288] > [Ribbon Tabs and Dialogs](#) ^[288] > [Ribbon Tabs](#) ^[296] >



Active only when the current environment is a drawing.



For more information see the [Plan View and Layout Tools](#) ^[150] topic.

Estimating page group

- **Take Offs** - Show the [Take Offs](#) ^[471] tool. This tool assists in counting items in the drawing.

Locations page group

•**Add Boundaries from Text** - Add [Location Boundaries](#)⁵³⁷ using a text entity to derive the location name. Once you select a text entity the Boundaries form is displayed allowing you to set the remainder of the properties.

Cabling page group

•**Draw Backbone - ENT only**. Draw a polyline representing the backbone in the plan view space. Then assign a number, cable type, source and destination ports and add that to the database.

•**Draw Cable** - Drawing a polyline representing a cable from a location boundary to another location boundary or from placed equipment to placed equipment. Cable information is then created in the Cables table.

•**Populate Cables** - Automatically places arched cables from location to location as defined in the Cables table.

•**Draw Prewire Cables** - Draw a polyline representing prewire cables and add the data to the Cables table marking the IO as PREWIRE. Later, you can assign cables to those PREWIRE entries.

Equipment page group

•**Populate Equipment** - Automatically place **SysNamed** equipment in the drawing in the **Location Boundaries** if found.

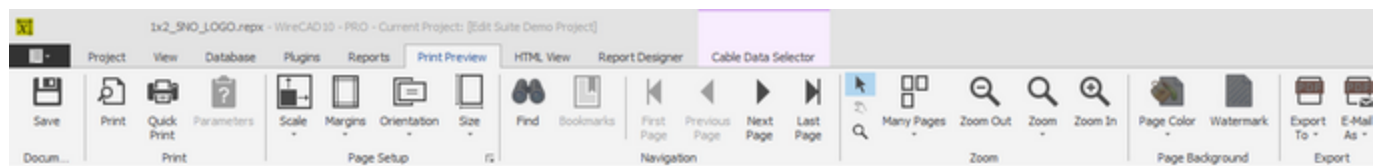
•**Place Equipment** - Place unnamed plan view equipment in the drawing. Preferably after **Location Boundaries** have been defined.

5.1.2.12 Print Preview

Navigation: [Reference](#)²⁸⁸ > [Ribbon Tabs and Dialogs](#)²⁸⁸ > [Ribbon Tabs](#)²⁹⁶ >



Visible only when the active environment is a report.



Related Topics:

[Print Preview Reports Form](#)⁵⁵⁰

5.1.2.13 HTML View

Navigation: [Reference](#)²⁸⁸ > [Ribbon Tabs and Dialogs](#)²⁸⁸ > [Ribbon Tabs](#)²⁹⁶ >



Visible only when the active environment is a report.



Related Topics:

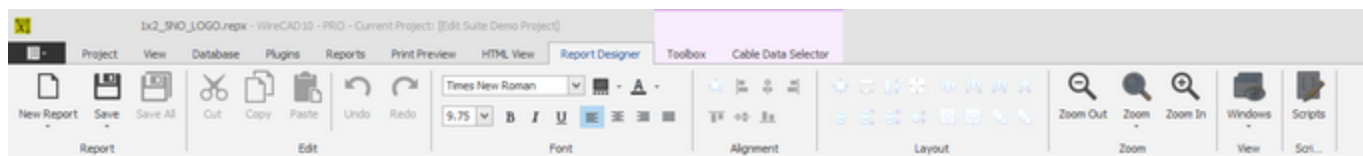
[HTML Preview Reports Form](#) ⁵⁵¹

5.1.2.14 Report Designer

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >



Visible only when the active environment is a report.



Related Topics:

[Report Design Form](#) ⁵⁵²

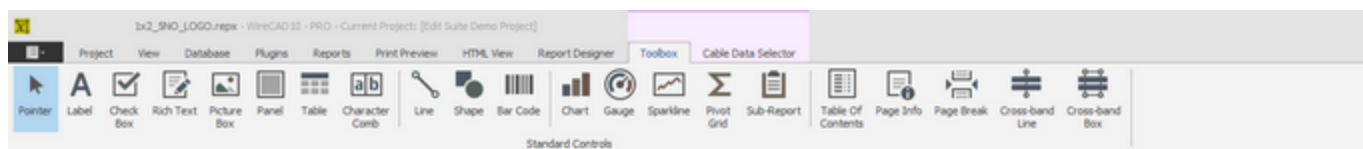
[A discussion on report design can be found here](#) ¹⁷⁰.

5.1.2.15 Toolbox

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >



Visible only when the Report Designer Tab is active.



Related Topics:

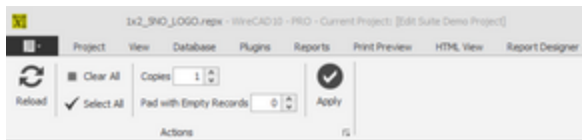
[Report Design Toolbox Form](#) ⁵⁵³

5.1.2.16 Cable Data Selector

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >



Visible only when the active environment is a report.



Related Topics:

[Cable Data Selector Form](#) ⁵⁵⁵

5.1.2.17 GPS Mapping Tools For Drawings

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ >



Active only when the current environment is a drawing.

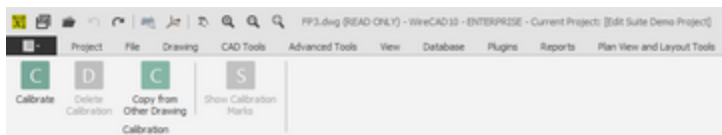
The GPS Mapping Tools section provides access to the functions necessary to use your DWG plant drawings with the GPS coordinate system.

5.1.2.17.1 GPS

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ > [GPS Mapping Tools For Drawings](#) ³¹⁸ >



Active only when the current environment is a drawing.



Calibrate: Align the world coordinate system of the drawing to the GPS coordinate system.

Delete Calibration: Removes the calibration data from the drawing. Only available if the drawing has current GPS calibration data.

Copy from Other Drawing: Allows you to copy calibration data from a drawing that shares the same coordinate data. The drawing you are copying from must have GPS calibration data or this function will fail.

Show Calibration Marks: Place location markers at the two calibration points in the drawing.

Related Topics:

[How To: Calibrate Your DWG to the GPS Coordinate System](#) ²⁶³

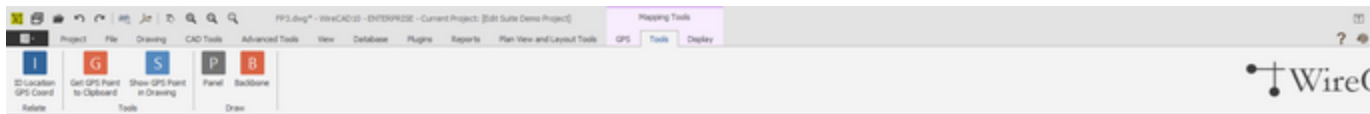
[How To: Copy Calibration Data from another Drawing](#) ²⁶⁵

5.1.2.17.2 Tools

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ > [GPS Mapping Tools For Drawings](#) ³¹⁸ >



Active only when the current environment is a drawing.



Relate Section

ID Location GPS Coord: Associate a Locations table record with a specific GPS coordinate point.

Related Topics:

- [How To: Associate GPS LAT, LON with your Locations](#) ^[255] Map tool version
- [How To: Associate GPS LAT, LON with your Locations](#) ^[266] DWG version.
- [ID GPS Locations](#) ^[514] dialog reference.

Tools Section

Get GPS Point to Clipboard: Allows you to select a point in the drawing. Once a point is selected its GPS lat, lon are copied to the clipboard as text. You can paste this data into any other application as text.

Show GPS Point in Drawing: Prompts for the GPS Lat,Lon to show. Enter the Lat,Lon in the command line. A location marker will be placed and zoomed into view.

Draw Section

Panel: Place a new SysNamed Panel or other equipment at a specific location within the drawing using the GPS Location markers as the location for the equipment. Requires that at least one Location Marker is displayed. For more information on displaying **Location Markers** see this [topic](#) ^[269].

Backbone: Draw a new Backbone between two panels. The panel markers must exist in the drawing.

Related Topics:

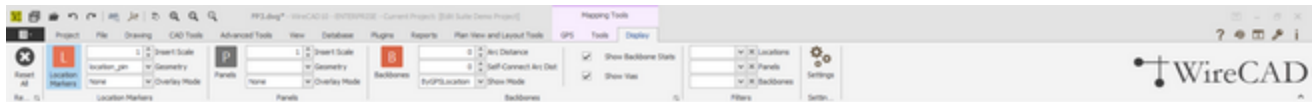
- [How To: Show Panel Markers in your Drawing](#) ^[270].
- How To: Draw a Backbone in your Drawing.

5.1.2.17.3 Display

Navigation: [Reference](#) ^[288] > [Ribbon Tabs and Dialogs](#) ^[288] > [Ribbon Tabs](#) ^[296] > [GPS Mapping Tools For Drawings](#) ^[318] >



Active only when the current environment is a drawing.



Reset Section

Reset All: Removes existing Location, Panel and Backbone markers.

Location Marker Section

Location Marker: Toggle the view of the Location Markers in the GPS calibrated drawing.



Location Markers are inserts that are placed at the GPS coordinates of the drawing. The tooltip of the marker displays the Location name as defined in the **Display Format String** of the [GPS Settings](#) ^[370] tool as well as any equipment found at that location.

Location Markers are placed in the drawing on a layer named "GPS_LOC_PIN_TEMP". We do not recommend placing any other entities on this layer. When the Location Marker toggle button is set to unchecked, all layer entities are deleted without prompting.

Double-clicking a Location Marker in the drawing will show its statistics.

Insert Scale: Sets the scale of the block defined in the Geometry setting.

Geometry: Sets the block to insert as a Location Marker. The Geometry dropdown is populated by dwg files in the **%BLOCKS%** ^[351] **GPS\Location Markers** directory.

In a stock installation this will be:

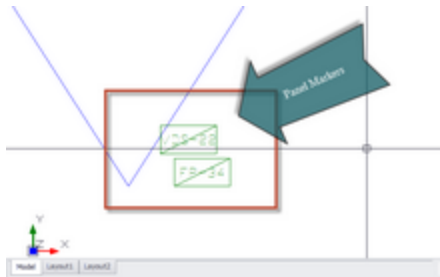
C:\users\public\documents\WireCAD\WireCADx\Blocks\GPS\Location Markers.

Overlay Mode: Determines the display behavior of locations that share the same GPS coordinates. The modes are:

- None:** Do nothing. Overlay marker upon marker. In this mode the tooltip data is aggregated to the topmost marker.
- Offset:** Offset the markers based on the OffsetX and OffsetY of the [GPS Settings](#) ^[370].
- Rotate:** Rotate the markers around the point.
- Radial:** Rotate the markers based on the Radial Angle and Radial Distance of the [GPS Settings](#) ^[370].

Panel Marker Section

Panel: Toggle the view of the Panel Markers in the GPS calibrated drawing.



Panel Markers are inserts that are placed at the GPS coordinates of the drawing. The tooltip of the marker displays the Sysname as defined in the **Display Format String** of the [GPS Settings](#)^[370] tool as well as any Backbones found on that SysName.

Panel Markers are placed in the drawing on a layer named "GPS_LOC_PANEL_TEMP". We do not recommend placing any other entities on this layer. When the Panel toggle button is set to unchecked, all layer entities are deleted without prompting.

Double-clicking a Panel Marker in the drawing will show its statistics.

Insert Scale: Sets the scale of the block defined in the Geometry setting.

Geometry: Sets the block to insert as a Panel Marker. The Geometry dropdown is populated by dwg files in the [%BLOCKS%](#)^[351] \GPS\Panel Markers directory.

In a stock installation this will be:

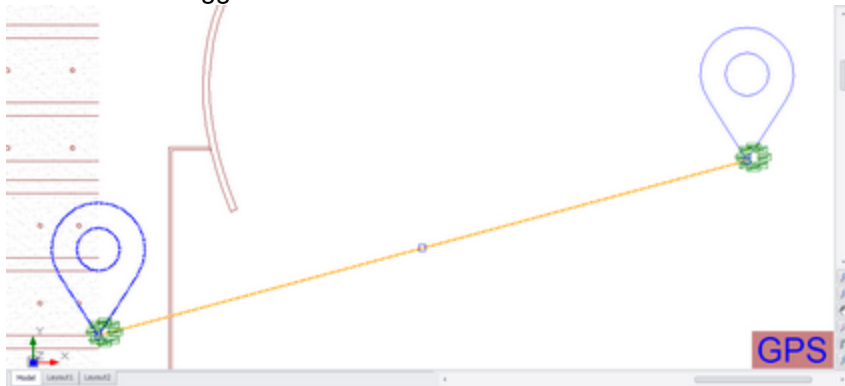
C:\users\public\documents\WireCAD\WireCADx\Blocks\GPS\Panel Markers.

Overlay Mode: Determines the display behavior of panels that share the same GPS coordinates. The modes are:

- None:** Do nothing. Overlay marker upon marker. In this mode the tooltip data is aggregated to the topmost marker.
- Offset:** Offset the markers based on the OffsetX and OffsetY of the [GPS Settings](#)^[370].
- Rotate:** Rotate the markers around the point.
- Radial:** Rotate the markers based on the Radial Angle and Radial Distance of the [GPS Settings](#)^[370].

Backbones Section

Backbones: Toggle the view of the Backbones Markers in the GPS calibrated drawing.

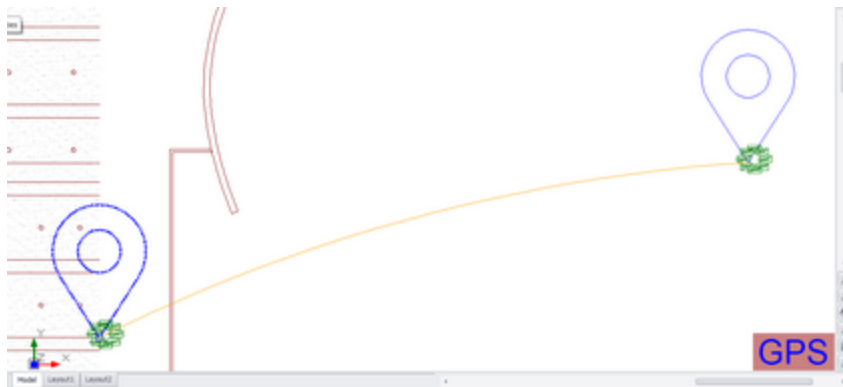


Backbone Markers are polylines that are placed at the GPS coordinates of the drawing. The tooltip of the marker displays the Backbone either the basic data of the first strand of fiber in the Backbone or the full Backbone statistics if the **Show Backbone Stats** checkbox is checked.

Backbone Markers are placed in the drawing on a layer named "GPS_LOC_BB_TEMP". We do not recommend placing any other entities on this layer. When the Backbone toggle button is set to unchecked, all layer entities are deleted without prompting.

Double-clicking a Backbone Marker in the drawing will show its statistics.

Arc Distance: Sets the polyline bulge factor of the Backbone. A bulge factor of 1 will create an arc of 180°.



Arc Distance of .1

Self-Connect Arc Distance: Sets the polyline bulge factor of polyline that share the same source and destination location GPS coordinates.

Show Mode: Determines the display behavior of backbones. The modes are:

- ByGPSLocation:** Place the source and destination points of the polyline based on GPS coordinates.
- ByLocationMarker:** Place the source and destination points of the polyline based on the GPS coordinates of Location Markers found in the drawing. This can be used to limit the display of Backbones.
- ByPanelMarker:** Place the source and destination points of the polyline based on the GPS coordinates of Panel Markers found in the drawing. This can be used to limit the display of Backbones.



ByPanelMarker

Show Backbone Stats: Generate the backbone statistics such as total fiber count, available fiber count and blocked fibers. This data is then displayed in the tooltip when the mouse hovers over the Backbone polyline.

Show Vias: If the Backbone contains related Via data with Locations (with GPS data) the polyline traverse from source to destination and every Via point.

Filter Section

The filter section is cascading. Selecting a location will cascade to the Panels filter by limiting the Panels shown to those that appear at the selected Location(s). In the same way, the Panels filter will cascade to the Backbones filter.

Locations: Filter by Location.

Panels: Filter by Panels.

Backbones: Filter by Backbones.

Settings Section

Settings: Display the [GPS Settings](#) ^[370].

5.1.2.18 GPS Mapping Tools for Map Tool

Navigation: [Reference](#) ^[288] > [Ribbon Tabs and Dialogs](#) ^[288] > [Ribbon Tabs](#) ^[296] >



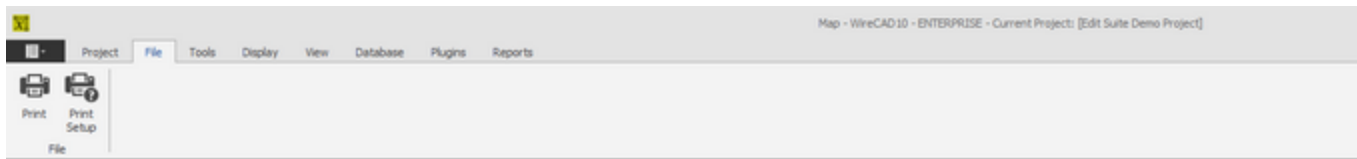
Active only when the current environment is the Map Tool.

5.1.2.18.1 File

Navigation: [Reference](#) ^[288] > [Ribbon Tabs and Dialogs](#) ^[288] > [Ribbon Tabs](#) ^[296] > [GPS Mapping Tools for Map Tool](#) ^[323] >



Active only when the current environment is the Map tool.



Print Preview: Show the Print Preview dialog. From the Print Preview dialog you can set the paper size, scale, orientation, and printer. You can also export to several formats from here.

Print Setup: Show the printer setup dialog where you can set the printer and print preferences.

Related Topics:

[Print Preview](#) ³¹⁶

5.1.2.18.2 Tools

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ > [GPS Mapping Tools for Map Tool](#) ³²³ >



Active only when the current environment is the Map tool.



ID GPS Location Coord: Show the ID GPS Location Coordinate dialog.

Copy Point to Clipboard: Select a point and the GPS coordinates are copied as text to the clipboard. The text format is: Lat,Lon in decimal form.

Related Topics:

[ID GPS Locations](#) ⁵¹⁴

5.1.2.18.3 Display

Navigation: [Reference](#) ²⁸⁸ > [Ribbon Tabs and Dialogs](#) ²⁸⁸ > [Ribbon Tabs](#) ²⁹⁶ > [GPS Mapping Tools for Map Tool](#) ³²³ >



Active only when the current environment is the Map tool.



Settings: Show the Settings dialog with the [GPS Settings](#) ^[370] panel shown.

Related Topics:

[GPS Settings](#) ^[370]

5.2 Dialogs

Navigation: [Reference](#) ^[288] >



Dialogs differ from Forms in that they do not occupy the full client docking area of the main window. Dialogs can be floating or modal. When a dialog is modal no other interaction with elements outside the modal dialog can take place.

Dialogs will remember their position and size unless you enable the [setting](#) ^[348]: Application [Basic] Disable Form State Tracking.

Related Topics:

None

5.2.1 Application Menu Dialogs

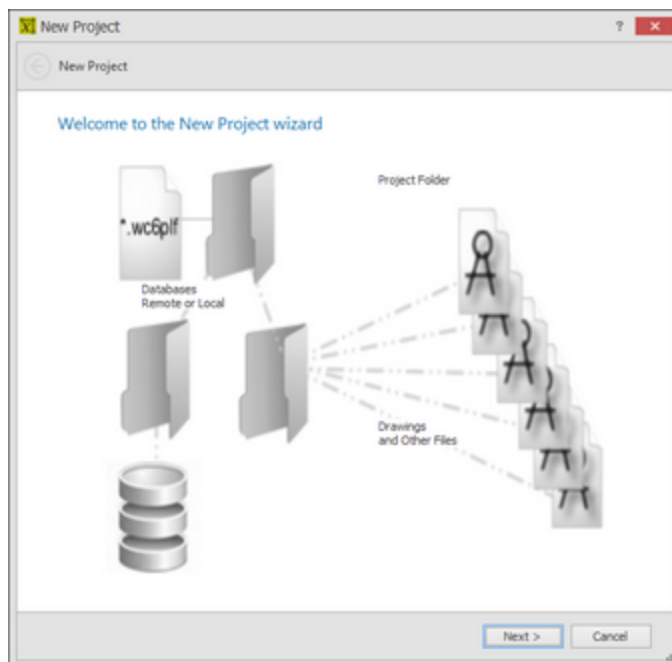
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] >



The following is a set of dialogs associated with Application Menu.

5.2.1.1 New Project Wizard

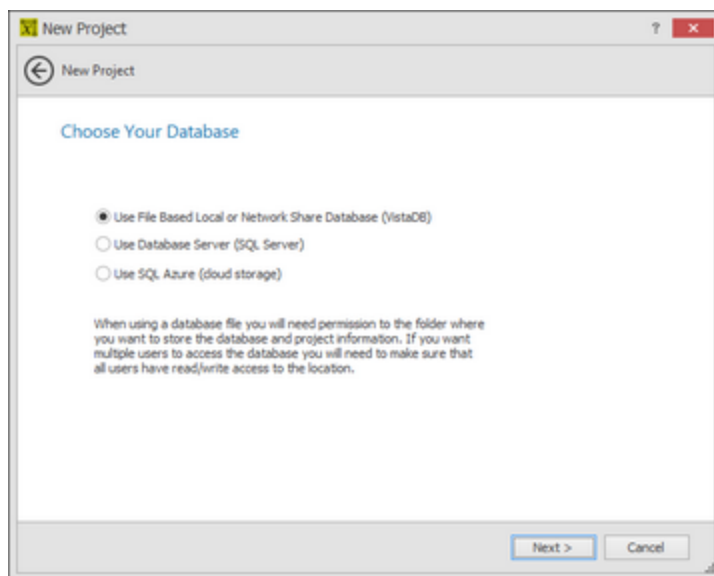
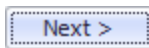
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] >



Application Menu > New Project...**Commandline:** np***Explanation***

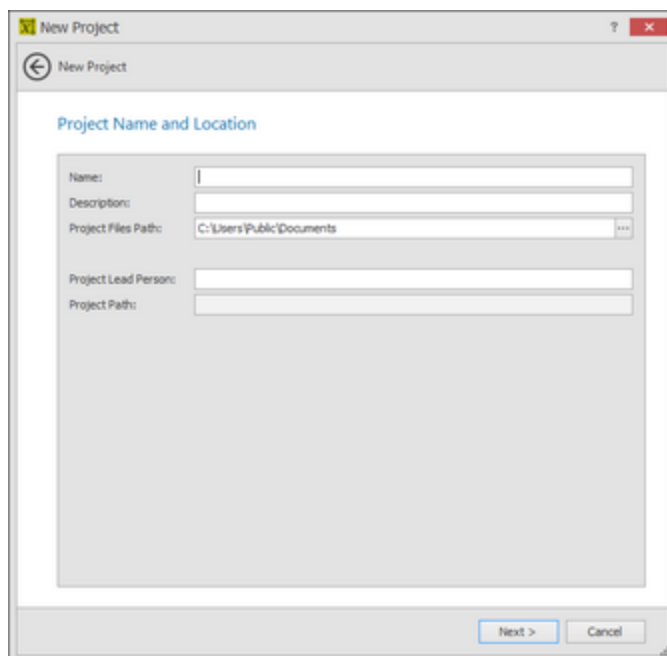
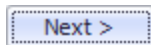
WireCAD Projects are a loosely coupled collection of files, folders and databases. The New Project Wizard will guide you through the steps of creating a new Project with several different options for different database types.

For a more extensive discussion see the [How To: Create a New Project](#) ⁷⁰ topic.

Related Topics:[Create a new Project](#) ⁷⁰***Dialog Options*****•Navigate Forward, Navigate Backward*****Explanation*****Related Topics:**[Choose your Database Type](#) ¹⁸⁸

Choose Database Page Options

- **Use File Based Local or Network Share Database (VISTADB)** - File based databases.
- **Use Database Server (SQL Server)** - You will need the host name and log in info to proceed.
- **Use SQL Azure (cloud storage)** - You will need a Microsoft Azure account for this option. Don't have one? Contact us. We can help.

A screenshot of the "New Project" dialog box. The title bar says "New Project". Inside, there's a section titled "Project Name and Location". It contains several input fields: "Name:" (empty), "Description:" (empty), "Project Files Path:" (containing "C:\Users\Public\Documents" with a browse button "..."), "Project Lead Person:" (empty), and "Project Path:" (empty). At the bottom right, there are "Next >" and "Cancel" buttons.

Explanation

Name, describe and locate the project.

NOTE: SQL, and SQL Azure do not like project names that start with a number.

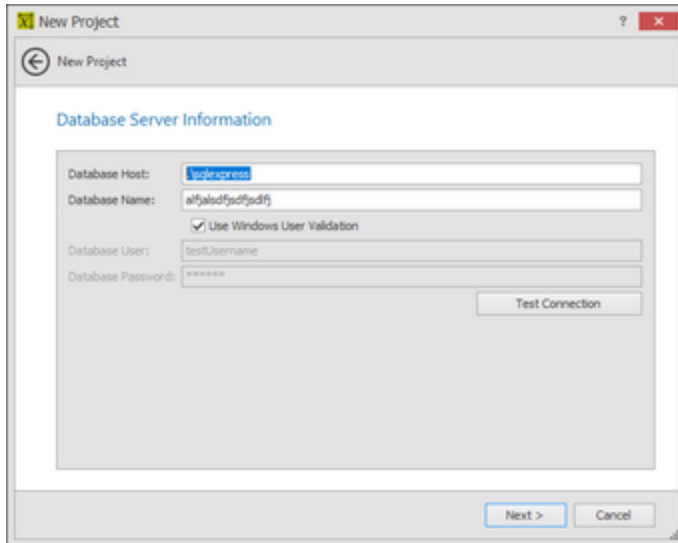
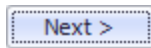
Related Topics:

[Default Project Location](#) 

Name, Description, Location Page Options

- **Name** - 50 Characters max. Must not have any operating system illegal characters. We also recommend not using the ['] apostrophe.

- Description** - Required.
- Project Files Path** - Pulled from the Settings, you can change it here.
- Project Lead Person** - Who is in charge here.



Explanation

How do we log in to the SQL Server. There is a subtle distinction to be made between SQL And SQL Azure:

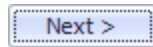
- SQL Server** the database name will be the Project Name.
- SQL Azure** the database name will be the name you gave your SQL Azure database. We will create a schema on that database with the project name as the schema name.

Related Topics:

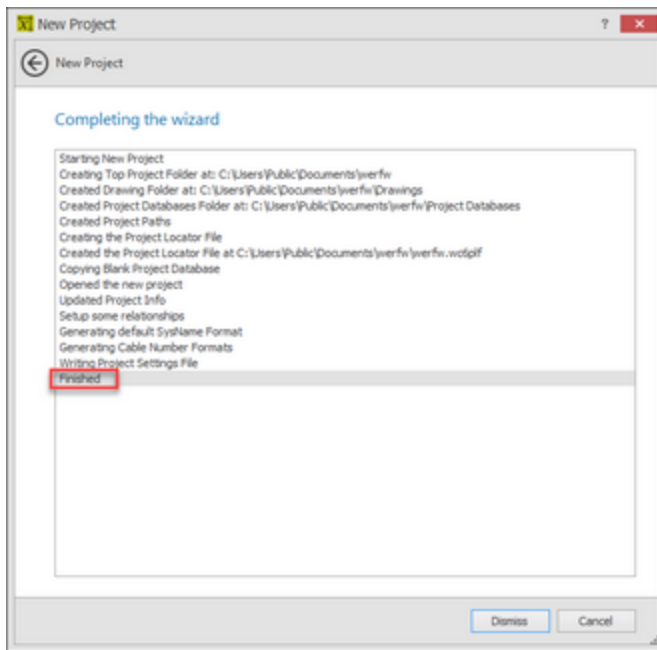
[Choose your Database Type](#) 

Database Host Page Options

- Host Name** - The host name and instance of your SQL Server.
- Database Name** - The database name. See the explanation above.
- Use Windows Security** - Not valid on SQL Azure.
- User Name and Password**
- Test** - Can we connect?



There are several pages of project settings that can also be found in the **Application Menu>Settings[Project]** dialog.



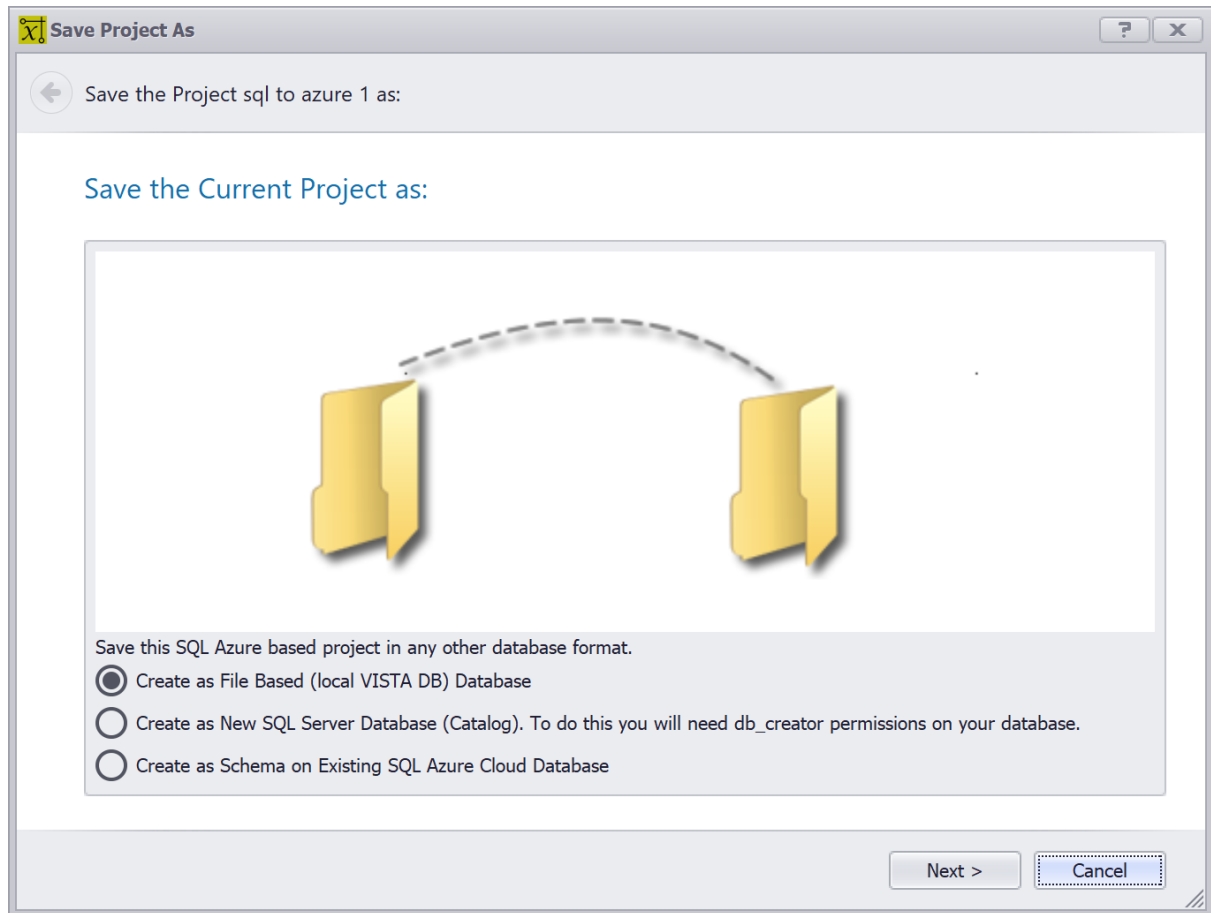
Finalization

Clicking Finish starts the project creation process. Make sure that at the end the log tells you that it finished. If not the project was not properly created and you may experience errors later.

5.2.1.2 Project Save As

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] >





Application Menu > Save Project As ...
Commandline: save project as

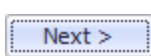
Explanation

This tool allows you to save a project as a new name in a new location. The project database can be changed from any supported format to any other supported format.

For example: if you have a file based database (VISTA DB) and you want to put that data in a SQL Server, you can use the Application Menu>Save Project As tool to do this.

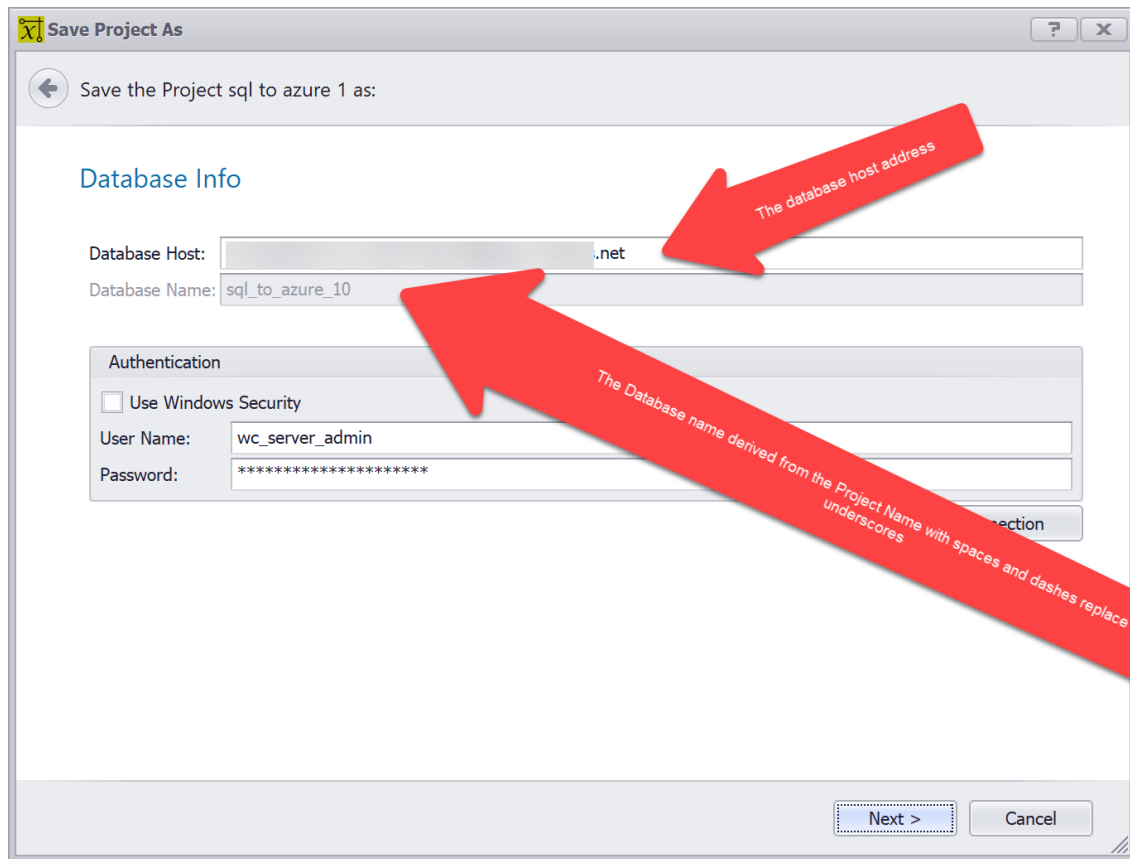
Dialog Options

- **Navigate Forward, Navigate Backward**



- **Create as File Based (local VISTA DB) Database.** This is the easiest and most portable form of WireCAD project.
- **Create a New Server Database (Catalog). To do this you will need db_creator permissions on the database server.** This will create a new database on your database server and copy the data into it.
- **Create as Schema on Existing SQL Azure Cloud Database.** This will create a new Schema on an existing database. This is typically done to keep costs contained by using a single database to house multiple projects.
- **Save As:** the new project name.
- **Save As Project Path:** The new project path.
- **Host:** The database host address. This is typically provided by your system database administrator. If you are doing this yourself you should know the address or IP address of the server that is hosting your sql instance.
- **Database Name:** This is the name of the new database for a standard SQL installation or the name of the existing database for the Azure host address.
- **Schema:** This should be empty for standard SQL Server and the project name for SQL Azure.
- **Use Windows Security:** This determines whether you are submitting your Windows credentials or the SQL login credentials. When in doubt - ask.
- **User Name and Password.** Used if Windows Authentication is not used.

If you are using either of the SQL options you will be presented with a database server Host and Login page.



Standard SQL Server project with the new database name derived from the project name

The Rules

- Project names must follow file naming conventions.
- Projects should not be nested in other project.
- SQL Server projects cannot start with a number (digit).

Subtle Differences Between SQL and SQL Azure

When we are creating a new SQL project, we create a new separate database on the database server.

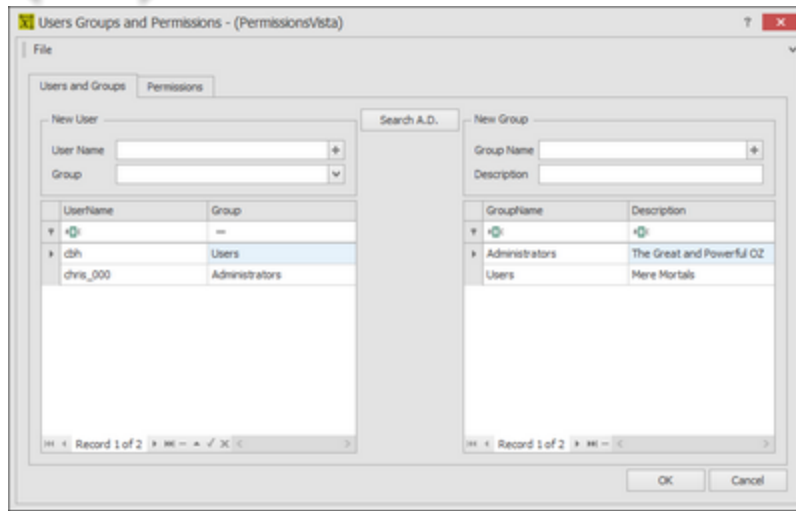
When we are creating a new SQL Azure project, we are using an existing database that has been set up for us. This database is capable of distinguishing table data of the same name by separating them using different "Schema".

Think of database schema as a folder tree, we can have tables named "Cables" in schema1 and schema2. We access the different "Cables" tables through dotted notation like: schema1.Cables, schema2.Cables.



5.2.1.3 Manage Security

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Application Menu Dialogs](#) [326] >



Application Menu > Security Tools > Manage Security ...

Commandline: security

Explanation

Administrators can manage users, groups and permissions. If WireCAD Security is enabled (see related topics). The person that enabled the Security option is the Administrator (the Great and Powerful Oz). All other users will be added automatically to the Users (mere mortals) group. Oz may:

- Change the group to which the user belongs.
- Create new groups.
- Modified group permissions.
- Create other Ozes.

So to be clear. Users belong to Groups. Groups have Permissions.

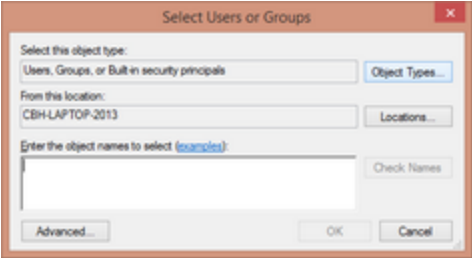
NOTE: No action is required to add a user. They simply need to open WireCAD. If their user profile does not yet exist in the WireCAD Security database, it will be added to the Users group.

NOTE: Oz can proactively add users by searching the Active Directory and adding the user to the group of his choosing.

Related Topics:
[Application Setting - WireCAD Security](#)
[Application Setup Wizard](#)

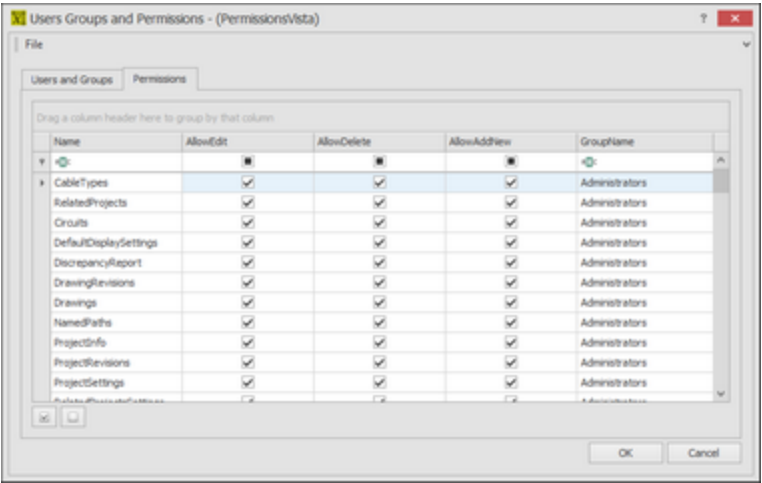
Users and Groups Tab Options

- New User** - Create a new user and assign to a group.
- New Group** - Create a new Group and associate Permissions set. After creating the new Group, switch to the Permissions tab and set the permissions on the collections and drawings.
- Search Active Directory** - Using the standard AD search box.



- User Grid** - Assign the User to a Group.

Permissions Tab Options

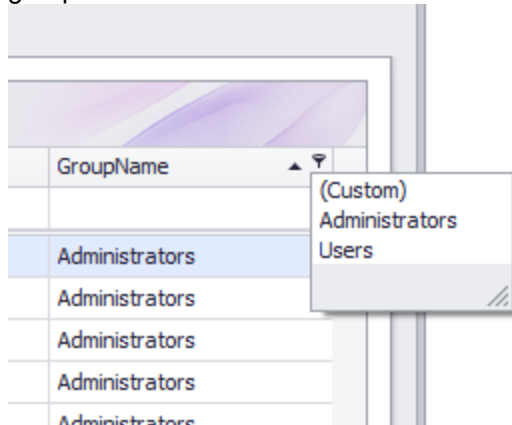


Explanation

Each data collection in WireCAD has Edit, Delete, and Create permissions. This grid enumerates each collection for each group.

Related Topics:[Grid Basics](#) ⁴⁹

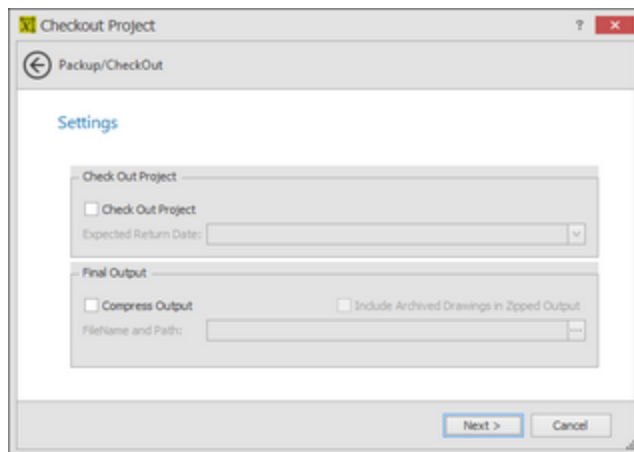
- **Collection Name** - This is usually the table name in the database.
- **Permissions** - Edit, Delete, AddNew.
- **GroupName** - You may find it handy to filter by group name then set the permissions for the group.



- **Select All and Clear Selection.**

**5.2.1.4 Packup / Checkout**

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ >



Application Menu > Check In/Out > Pack up /Check Out ...
Commandline: packup

Explanation

A WireCAD project is a loosely coupled group of files, databases and settings. The global database is maintained separately from the project database, and drawings may have linked images. In order to move the project to another machine we will need to gather all of the resources used to create the project into a central location (the project folder).

The **Packup / Checkout** Tool performs the following functions:

- Copy a file based version of the Global Database into the Project\Project Databases folder and sync it with the SQL Server version if necessary.
- Copy all linked images to the drawings directories.
- If **[Check-out]** then flag the project as read only.
- If **[Compress Output]** then zip all the project folder items into a single zipped file with the name: **[PROJECT_NAME.ZIP]**

NOTE: This process can take some time on large projects.

Related Topics:

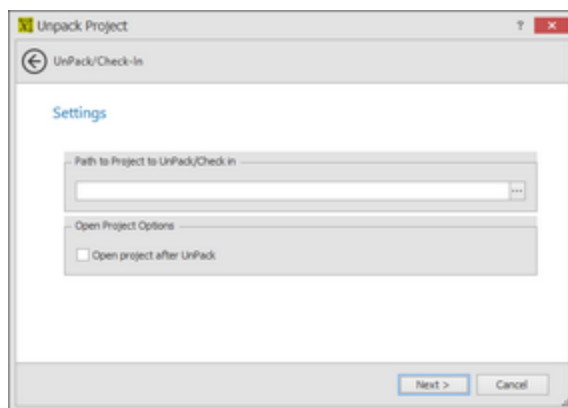
[Moving Projects](#) 

Dialog Options

- Check Out Project** - Flag the Project Read Only. This is useful if the project is going out into the field for commissioning. The version that stays in the office should not be edited.
- Expected Return Date** - Let the people in the office know when you think you will have the project checked in.
- Compress Output** - Zip it.

5.2.1.5 Unpack Project

Navigation: [Reference](#)  > [Dialogs](#)  > [Application Menu Dialogs](#)  >



Application Menu > Check In/Out > Un Pack ...
Commandline: unpack

Explanation

The complementary function to the Pack Up is the Un Pack. This function performs the following:

- Unzip the compressed file (if necessary).
- Sync the local Global Database with the one found in the incoming Project Databases folder.
- Open the project for use.

NOTE: This process can take some time on large projects.

Prerequisites

A project that has been Packed Up. Using the Pack Up / Check Out tool.
No project can be open or the tool will not run.

Related Topics:

[Pack Up / Check Out](#) ³³⁷

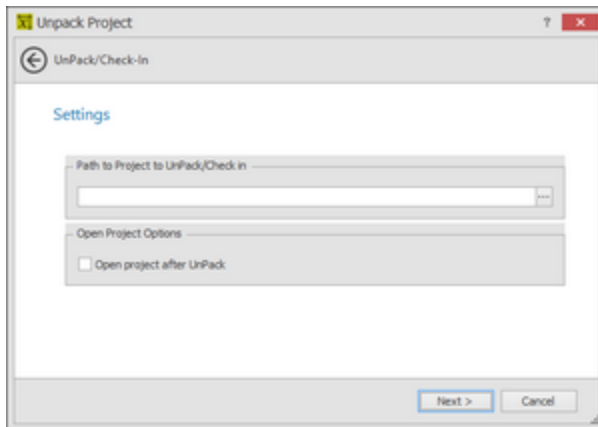
[Moving Projects](#) ¹⁸¹

Dialog Options

- Project Path** - Browse to the zipped file or the .wc6plf file.
- Open project after unpack** - Of course.

5.2.1.6 Check in Project

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ >



Application Menu > Check In/Out > Check In...

Commandline: packup

Explanation

The complementary function to the Check Out is Check In. This function performs the following:

- Unzip the compressed file (if necessary).
- Sync the local Global Database with the one found in the incoming Project Databases folder.
- Syncs the Project Database of the incoming project with the location database.
- Overwrite all drawings with the incoming ones.
- Remove Read Only flag.

NOTE: This process can take some time on large projects.

Prerequisites

A project that has been Checked Out. Using the Pack Up / Check Out tool.

The Checked Out local project must be open before this tool will run.

The project names of the local and incoming projects must match or the tool will fail.

Related Topics:

[Pack Up / Check Out](#) ³³⁷

[Moving Projects](#) ¹⁸¹

Dialog Options

•**Project Path** - Browse to the zipped file or the .wc6plf file.

•**Open project after unpack** - Of course.

5.2.1.7 Account Info

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ >



Application Menu > Account Info

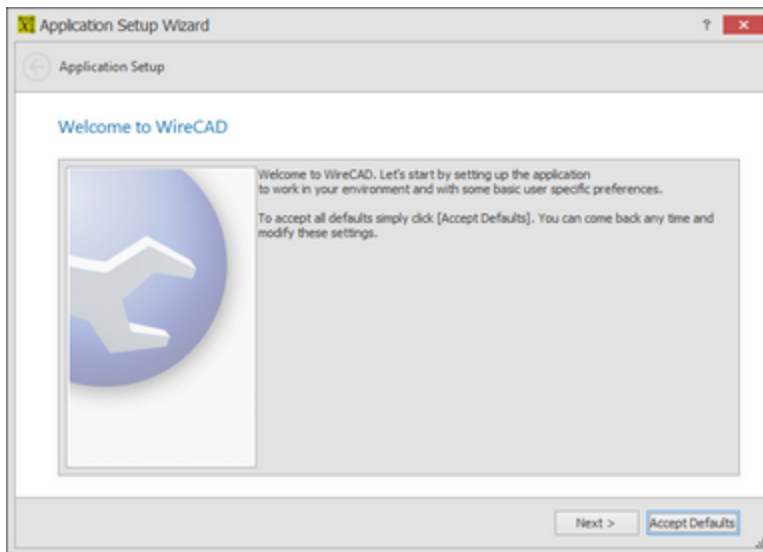
Commandline: Edit Account Info

Explanation

This dialog allows you to enter account specific data for use in reports.

5.2.1.8 Application Setup Wizard

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] >



Application Menu > Application Setup Wizard...

Commandline: `showappsettings`

Explanation

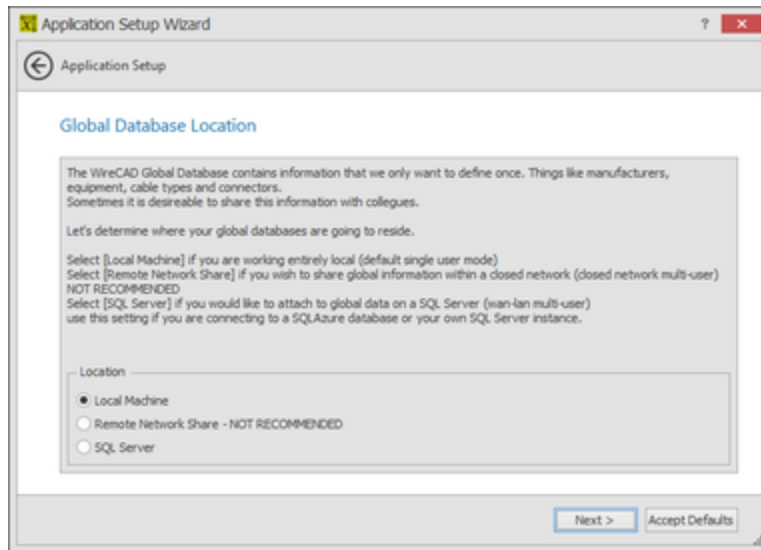
This wizard runs once at the start of WireCAD. You can always run it again to change the settings, but most of the settings in the wizard are contained in the Settings dialog.

Dialog Options

•Navigate Forward, Navigate Backward

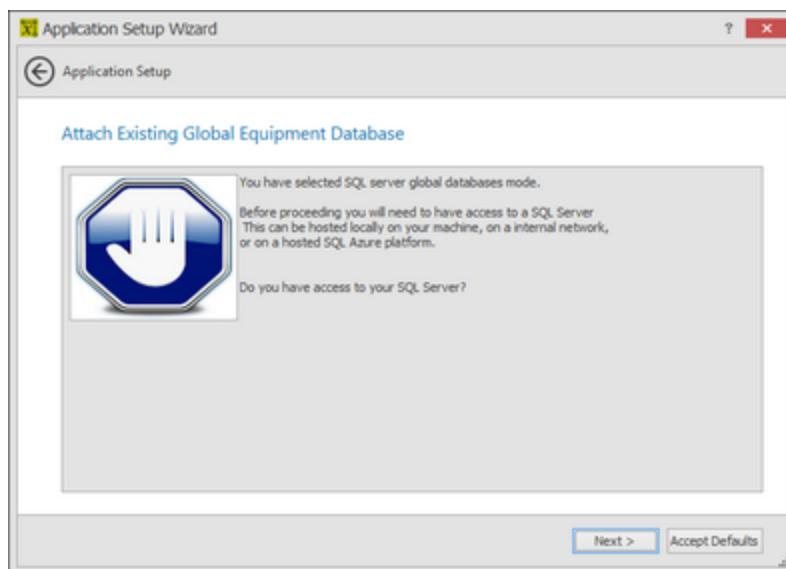


•Show This Again - Show this dialog again.



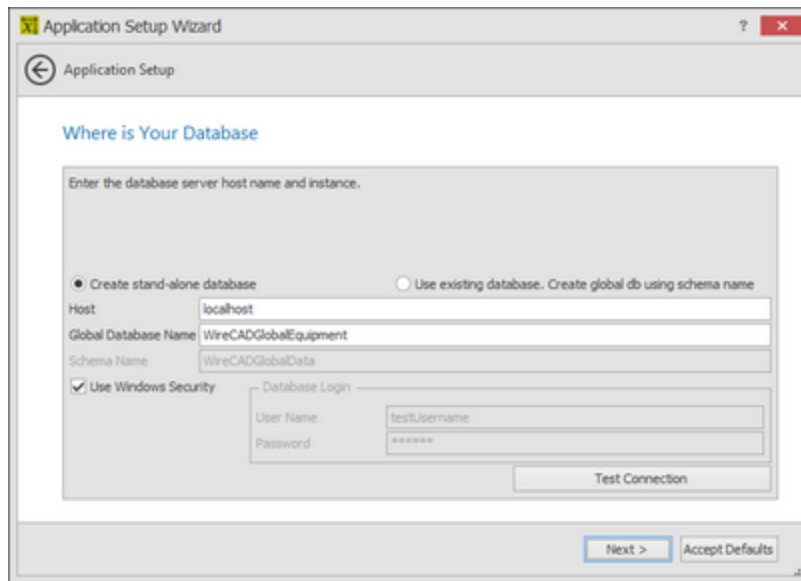
Global Database Location Page Options

- **Local Machine** - The default location is c:\users\public\WireCAD\WireCADx\WireCADGlobalEquipment.vdb3. You may rename the file in order to better organize your global data by client. See this topic on the [Global Database Location](#) ³⁵⁰.
- **Remote Network Storage** - This is not recommended but is possible. Why do we do this? Well we hate to limit you. Here are the ramifications of using file based storage over a network:
 The file is not network fault tolerant like SQL Server.
 Data can become locked and in some cases corrupt.
 You may find yourself having to Pack (compact and repair) the database if you get locked out of it.
- **SQL Server** - This is the best option for shared storage, but you lose portability.



Database Path Page Options

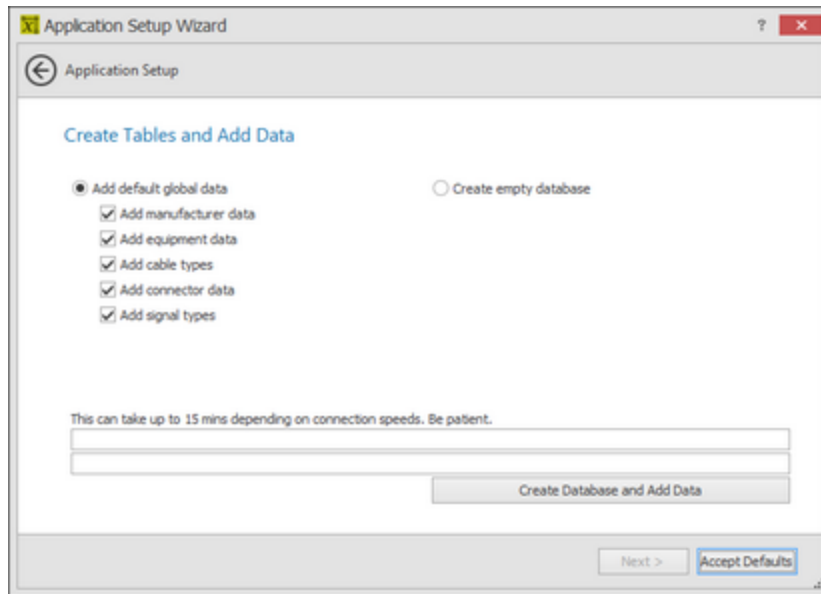
•**Path to Shared Global Database** - Show if the first selection is Remote Network Storage. Enter the Path to the WireCADGlobalEquipment.vdb3



Show only if the initial selection is SQL Server.

Database Host Info Page Options

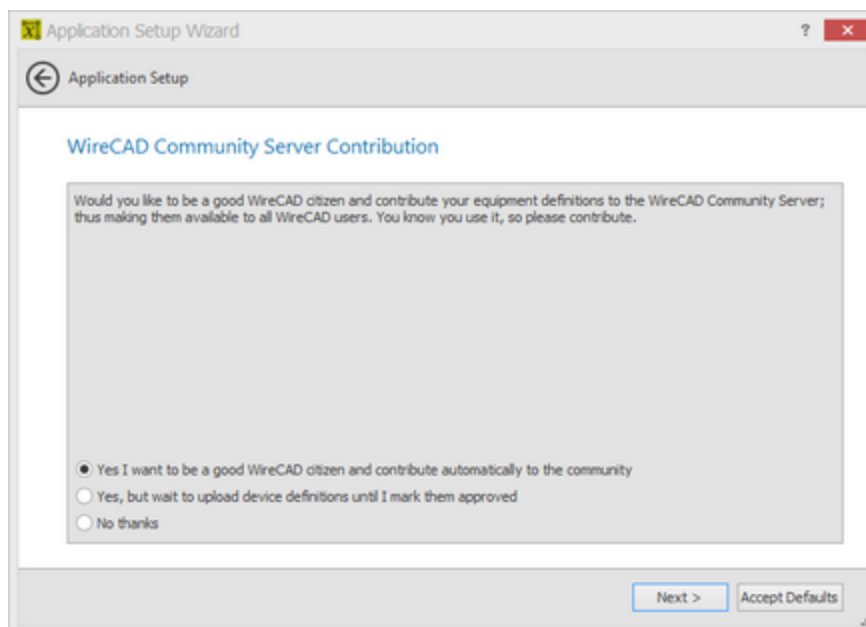
- Create stand-alone database** - Use this option if you are installing on an internal network.
- Use existing database. Create global db using schema name.** - Use this option if you are installing on a SQL Azure database. The schema name is used to separate different data structures on the same database.
- Host** - The hostname\instance name or IP address\instance name.
- Global Database Name** - Either the name of the existing global database or the name of the SQL Azure database on which we will create our global database schema.
- Schema Name** - Name this something like WireCADGlobalData.
- Use Windows Security** - Not available on SQL Azure.
- User Name, Password** -
- Test** - See if we can connect.



Shown only if the SQL Server Option is selected.

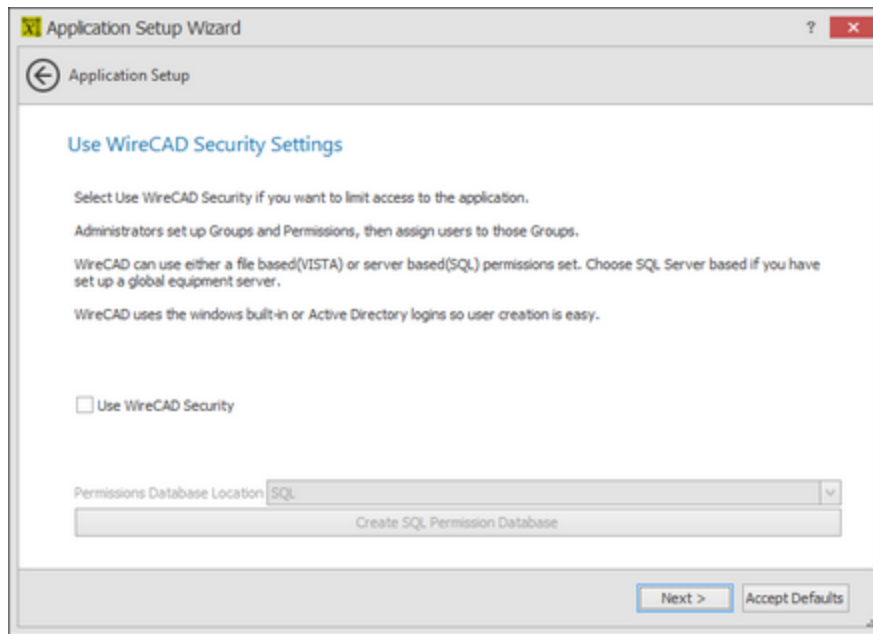
Create Tables and Add Data Page Options

- **Add default global data** - Create the database on the host and add the selected default data.
- **Create empty database** - Create the database on the host with no data.
- **Create Database and Add Data** - Do it!



WireCAD Community Server Contribution Mode Page Options

- **Yes I want to contribute**
- **Yes but wait until I check the Approved field**
- **No Thanks** - You can still manually upload from the Equipment Library in this mode.

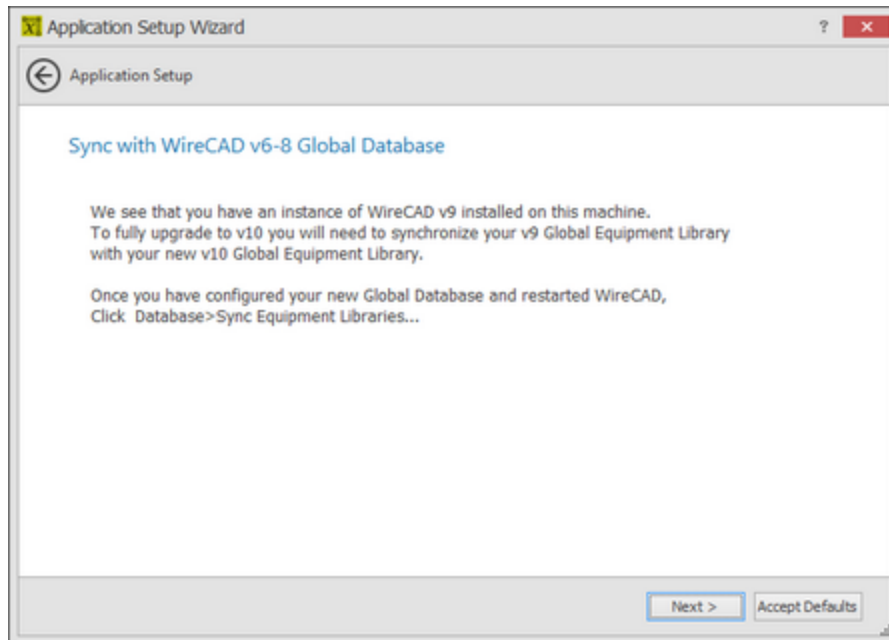


Explanation

WireCAD Security is used to restrict user access to key application sections and functions. If enabled, the user launching WireCAD will be subject to the rules of the group to which they belong. The user that initiates WireCAD security will be the only Administrator and will need to assign all other user permissions. This is managed through the [Manage Security](#) ³³⁵ dialog.

Use WireCAD Security Setting Page Options

- **Use Security** -
- **Database Type** -
 - VISTADB** - while you can use VISTADB as the security database it is only application to this machine.
 - SQL** - we will create the permissions database on your SQL Server using the SQL host and login information.
- **Create SQL Permissions Database** - Do It!



Explanation

We found a WireCAD previous version. In order to use the work that you have already done you will need to synchronize with the previous version Global Database. There are many variables that can be configured to locate the Global Database elsewhere. When you finish the Application Setup Wizard restart WireCAD then launch the Database >Sync Global Database tool and set the remote database to look at your previous instance.

Related Topics:

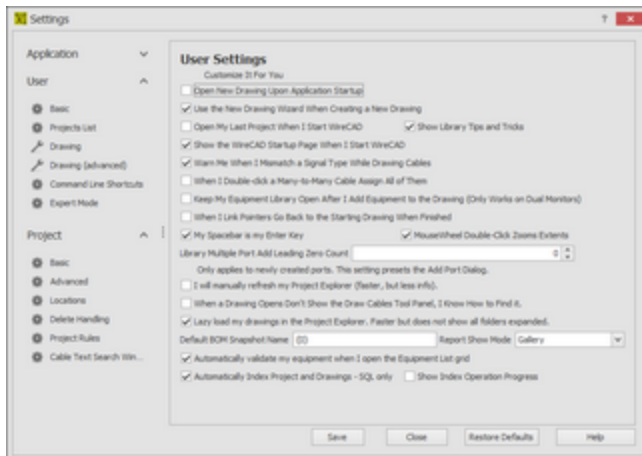
[Synchronizing with Another Global Database](#) 

[Sync Global Database dialog](#) 

5.2.1.9 The Settings Dialog

Navigation: [Reference](#)  > [Dialogs](#)  > [Application Menu Dialogs](#)  >





Application Menu > Settings
Commandline: set

Explanation

The Settings dialog is comprised of the following sections:

- **Application** - settings that determine the behavior of WireCAD for this machine. These settings are stored in the Global Application Data folder as ApplicationSettingsX.xml.

[Basic](#) ^[348]

[WireCAD Security](#) ^[349]

[Global Database Location](#) ^[350]

[Support Paths](#) ^[351]

[Organization](#) ^[352]

[Enable/Disable Plugins](#) ^[353]

[Todo List](#) ^[354]

- **User** - settings specific to your user profile on this machine as UserSettingsX.xml.

[Basic](#) ^[354]

[Projects List](#) ^[356]

[Drawing](#) ^[299]

[Drawing \(advanced\)](#) ^[360]

[Command Line Shortcuts](#) ^[360]

[Enterprise CMS \(ENT Only\)](#) ^[361]

[Expert Mode](#) ^[362]

- **Project** - settings specific to the current project. These settings are stored in the project folder as ProjectSettingsX.xml.

[Basic](#) ^[364]

[Advanced](#) ^[365]

[Locations](#) ^[366]

[Delete Handling](#) ^[366]

[Enterprise CMS](#) ^[367]

[Export Settings](#) ^[368]

[Project Rules](#) ^[369]

[Status Behavior](#) ^[369]

NOTE: These are the stock basic settings panels. Third party developers may register their own settings panels. See their documentation.

Prerequisites

None

Dialog Options

- **Save** - Commit your changes and issue a SettingsChanged command to the application.
- **Close** - Do nothing and discard your changes.
- **Restore Defaults** - Restore the defaults of the current settings panel.

5.2.1.9.1 Application Settings

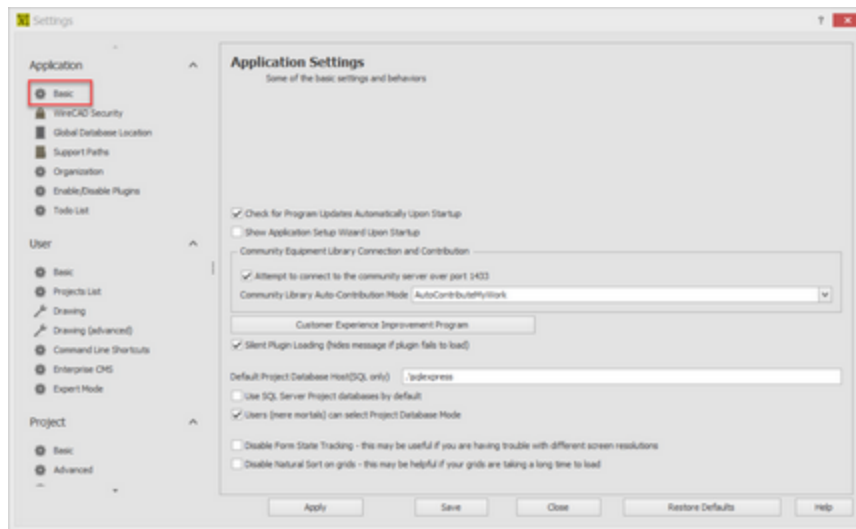
Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ > [The Settings Dialog](#) ³⁴⁶



Application settings that determine the behavior of WireCAD for this machine. These settings are stored in the Global Application Data folder as ApplicationSettingsX.xml.

5.2.1.9.1.1 Application Basic

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ > [The Settings Dialog](#) ³⁴⁶ > [Application Settings](#) ³⁴⁸

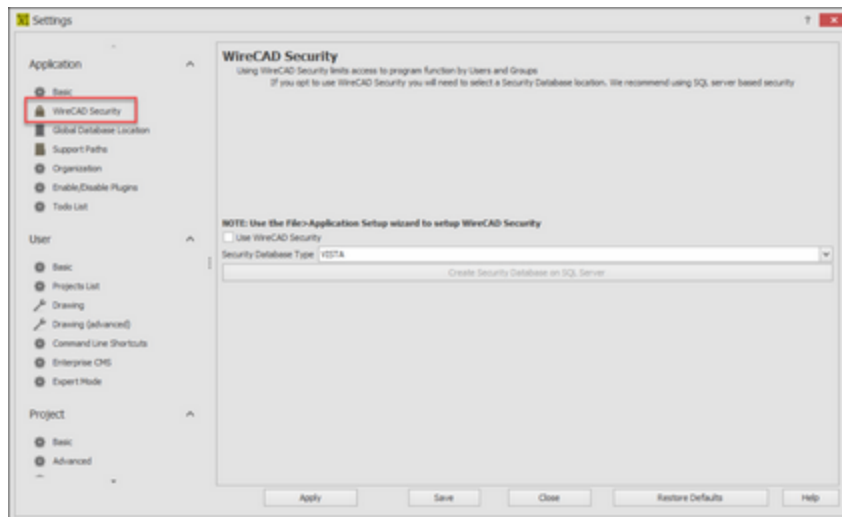


Basic Panel Options

- Check for Program Updates Automatically on Startup** - If updates are available you will see a banner pop up in the lower right-hand corner of the screen.
- Show Application Setup Wizard on Startup** - This is usually unchecked except the first run of the application. You can always start the **Application Setup Wizard** by clicking **Application Menu > Application Setup Wizard**.
- Community Server Contribution** - How will you contribute to the **WireCAD Community**?
- Customer Experience Improvement Program** - You must opt in to the **CEIP**. We will then gather usage and error data. The generic data will be uploaded to our servers and aggregated so we can bring you a better application.
- Silent Plugin Load** - Hide all error messages on startup.
- Default Project Database Host** - Applies only to SQL projects. Presets the **New Project Wizard**.
- Use SQL Server Project Databases by Default** - Presets the **New Project Wizard**.
- Users (mere mortals) can select Project Database Mode** - Some organizations may wish to limit selection of project database type.
- Disable Form State Tracking** - The setting disables the functions that remember form position and sizing. This can be helpful if you are experiencing trouble with different resolution monitors in a multi-monitor setup.

5.2.1.9.1.2 WireCAD Security

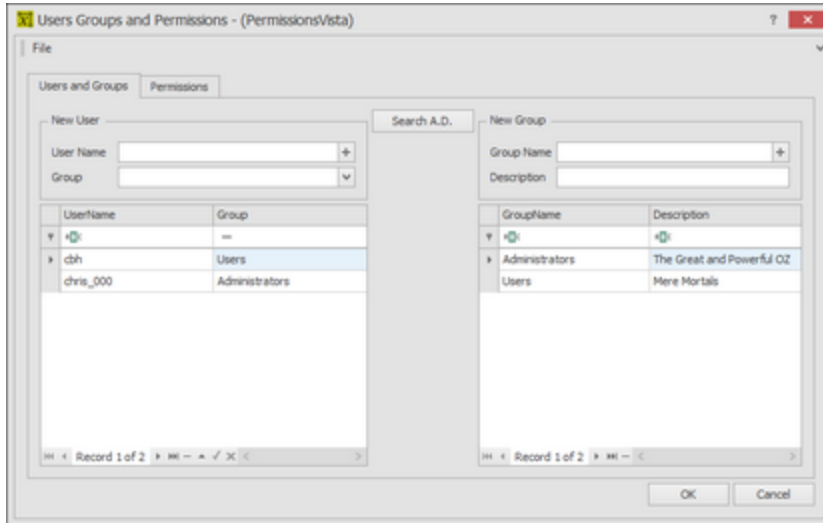
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Application Settings](#) ^[348] >



WireCAD Security Panel Options

- Use WireCAD Security** - Enable Security. When enabled the WireCAD security system uses the current login to determine program access. If your user is a member of the Administrators group

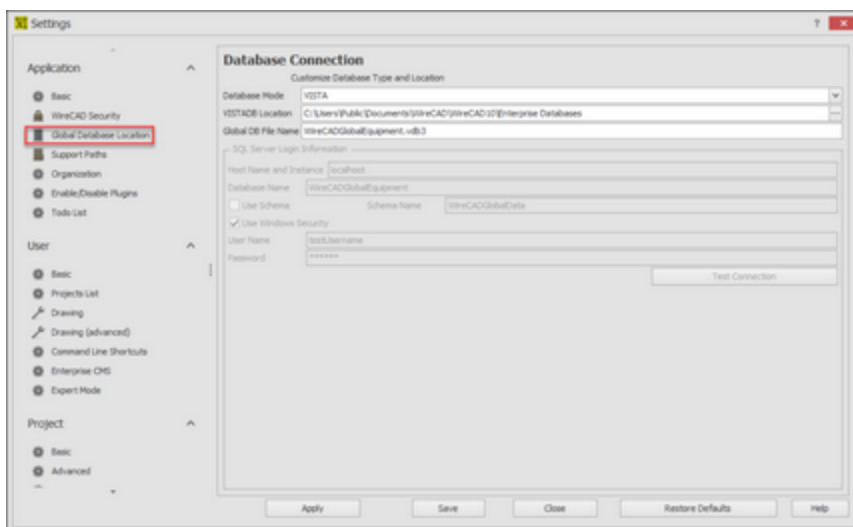
you will be able to control the access of other users via the **Application Menu > Security > Manage Security...** dialog.



•**Database Type** - Typically you will only use security in a multi-user environment with a SQL host, but the capability exists to use file based VistaDB databases.

5.2.1.9.1.3 Global Database Location

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Application Settings](#) ^[348] >

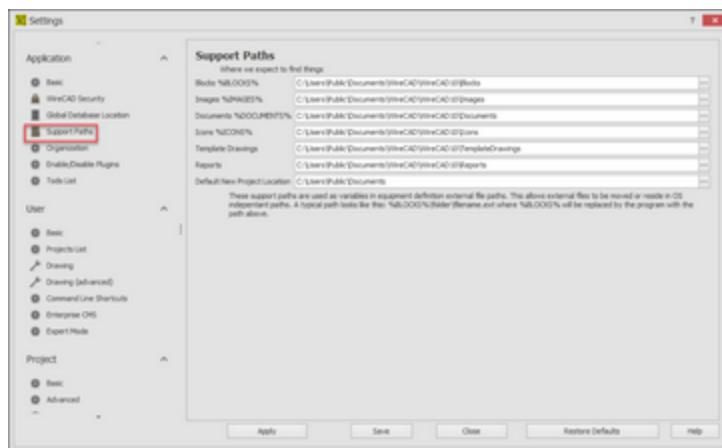


Global Database Location Panel Options

- Database Mode** - VISTA/SQL. NOTE: SQL will work for SQL AZURE as well. We will need the database name and schema name.
- VISTADB Location** - Path to the VISTADB file that is your WireCADGlobalEquipment.vdb3 file.
- Global DB File Name** - Note that this is the default name given to the global database file but you may change it.
- Host Name and Instance** - This can be a fully qualified path, ip address, or host name.
- Database Name** - The name of the global database to connect to. This is WireCADGlobalEquipment by default but you may choose to rename it.
- Use Schema, Schema Name** - WireCAD supports multiple schema in the same database file. This is useful if you are using a cloud based SQL Server service that charges by the database. You can have a schema for your global database and one for each of your project databases. All contained within the same database file.
- Use Window Security** - Use this if you wish Windows security to manage the login. This will not work with SQL Azure.
- User Name, Password**
- Test Connection** - Test to see if you can connect and have the appropriate privileges.

5.2.1.9.1.4 Support Paths

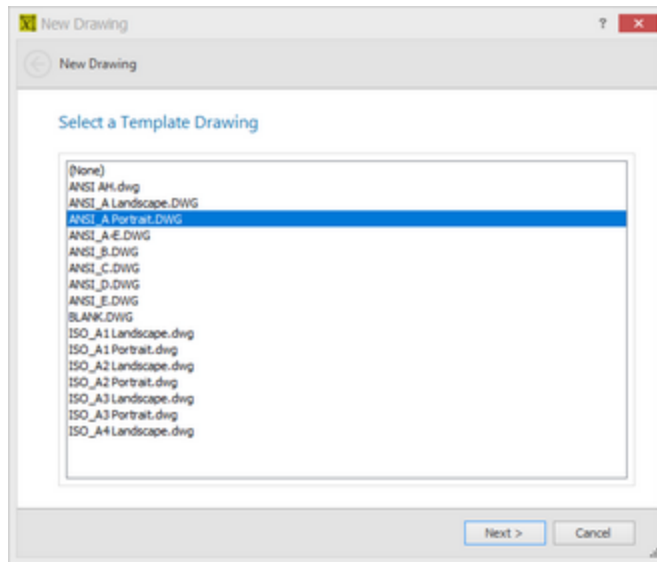
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Application Settings](#) ^[348] >



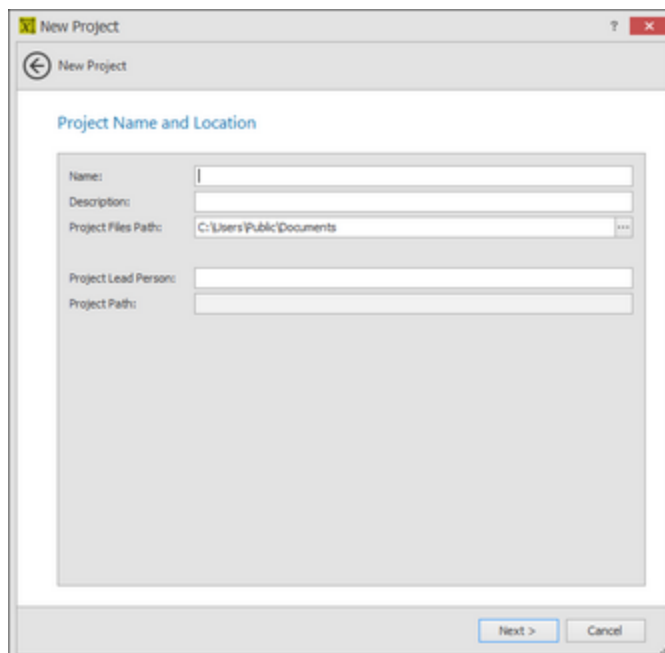
Application - Support Paths Panel Options

- Block %BLOCKS%** - The path that will replace the %BLOCKS% variable in the equipment library.
- Images %IMAGES%** - The path that will replace the %IMAGES% variable in the equipment library.
- Documents %DOCUMENTS%** - The path that will replace the %DOCUMENTS% variable in the equipment library.
- Icons %ICONS%** - The path that will replace the %ICONS% variable in the equipment library.

- **Template Drawings** - The path to the **Template Drawings** folder. This is the folder where **Template Drawings** are saved when using the **File > Save As Template Drawing** function is used and the folder that the **New Drawing Wizard** searches to fill the Templates list:



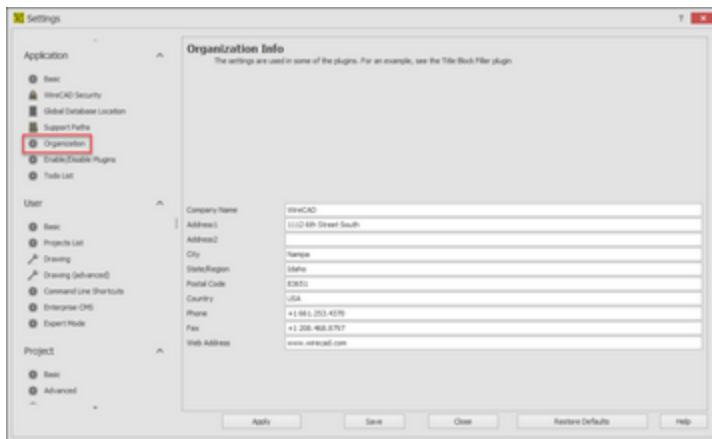
- **Reports** - The path to the top reports folder. This is a recursive search and will enumerate all subfolders and files.
- **Default New Project Location** - Presets the New **Project Wizard**.



5.2.1.9.1.5 Organization

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ > [The Settings Dialog](#) ³⁴⁶ > [Application Settings](#) ³⁴⁸ >



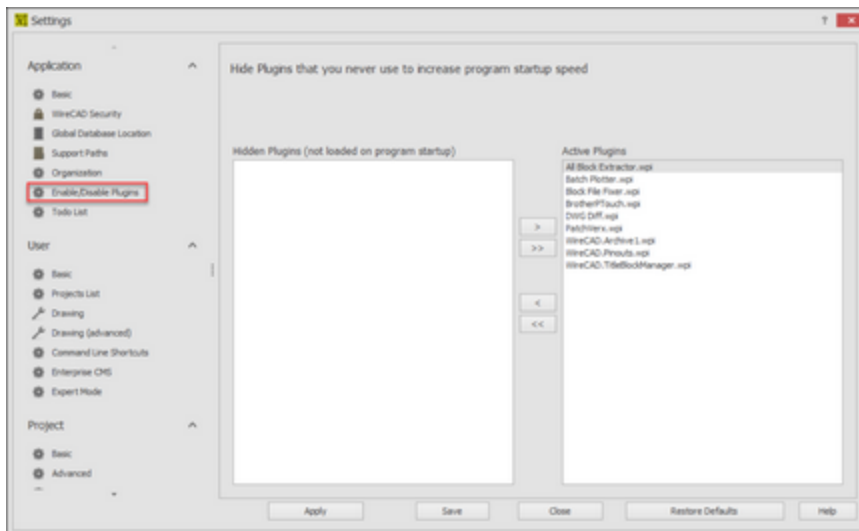


Application - Organization Panel Options

This one's self-explanatory

5.2.1.9.1.6 Enable/Disable Plugins

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Application Settings](#) ^[348] >



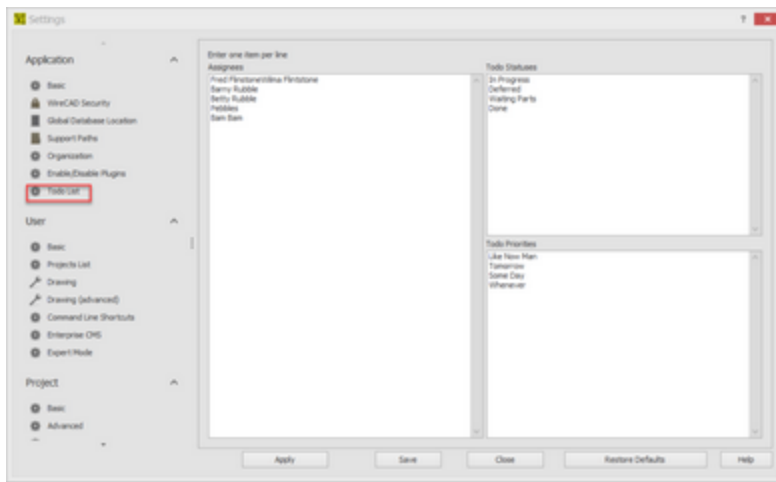
Enable/Disable Plugins Panel Options

Move items to the left-hand list to disable.
Move items to the right-hand list to enable.

This panel only works with Plugins that support the wpi manifest file. Auto-discovered Plugins cannot be disabled from here.

5.2.1.9.1.7 Todo List

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Application Settings](#) ^[348] >



Explanation

Sets the drop downs in the Project Todo List

Application - Todo Panel Options

- **Assignees** - List of people to whom a task may be assigned.
- **Statuses** - List of Todo item statuses. Modify this to suit your needs.
- **Priorities** - List of priorities. Again, make it fit your needs.

5.2.1.9.2 User Settings

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] >



User settings specific to your user profile on this machine. The path to these settings is usually:
C:\Users\<YOUR_USER_NAME>\AppData\Local\WireCAD\WireCAD9\UserSettings.xml.

Your system administrator has the ability to modify this.

You can use:

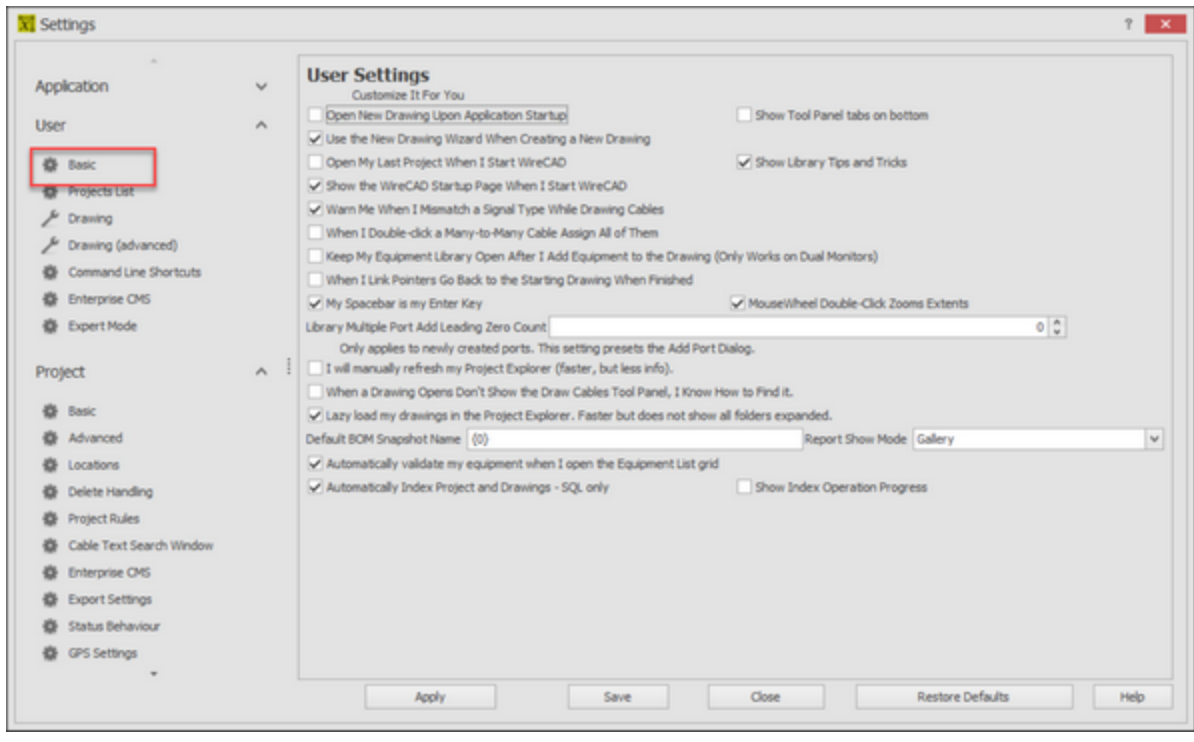
Application Menu > Frequently Used Folders > User Application Data Folder

To know the path we are using.

5.2.1.9.2.1 User Basic

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [User Settings](#) ^[354] >

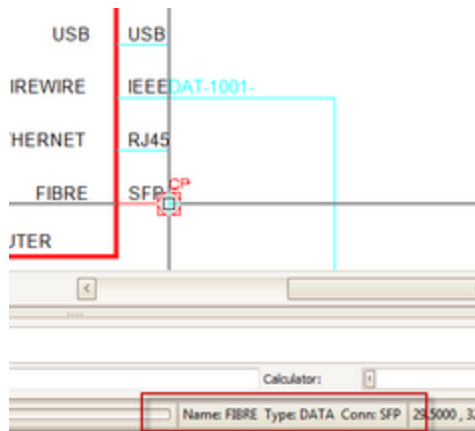




These are settings for your user profile on this machine.

Basic Panel Options

- **Open New Drawing Upon Application Startup** - Opens a blank new drawing when WireCAD starts.
- **Use the New Drawing Wizard** - Shows the **New Drawing Wizard** when you click **File > New**. Allows the use of template drawings. If set to false, just creates a new drawing without a template.
- **Open My Last Project When I Start WireCAD** - Behaves like earlier versions of WireCAD that opened the last project automatically on startup.
- **Show Library Tips** - This shows the Tips and Tricks form in front of the **Equipment Library**.
- **Show the WireCAD Startup Page**
- **Warn of Signal Type Mismatch** - When you finish drawing a cable WireCAD will check the source and destination port signal types and let you know if they don't match. You can use the new status panel to know the port type if you are confused.



• **When I Double-click a Many-to-Many Cable Assign All of Them** - Many-to-Many cables that are set for Multiple Database Entries can either be assigned all at once by a single double-click, or if this setting is turned off a double-click will find the closest port to the cursor and assign that.

• **Keep Equipment Library Open** - This is useful if you are using dual monitors and want to have the **Equipment Library** stay open on the other monitor.

• **When I Link Pointers Go Back to the Starting Drawing When Finished** - The default Link Pointer function will leave you on the second sheet. If you set this to true it will jump you back to the first sheet.

• **My Spacebar is my Enter Key** - This is helpful when executing command line commands.

• **MouseWheel Double-click Zooms Extents**

• **Library Multiple Port Add Leading Zero Count**

• **I Will Manually Refresh the Project Explorer** - True to disable autorefresh on the **Project Explorer**. This may be helpful if your project grows to have hundreds of drawings.

• **When a Drawing Opens Don't Show the Draw Cables Tool Panel**

• **Lazy Load my drawings in the Project Explorer** - True to disable recursive search of all drawing subfolders. This may be helpful if your project grows to have hundreds of drawings.

• **Default BOM Snapshot Name** - Presets the BOM Generator. The {0} represents the current date.

• **Report Show Mode** - Where are we seeing the reports.

Project Explorer - appends the reports tree after the project.

Reports Gallery - populates the Reports Gallery with thumbnails and descriptions from the accompanying report manifest xml files (if any).

• **Automatically validate my equipment when I open the Equipment List grid** - Default = true but can be expensive on large projects.

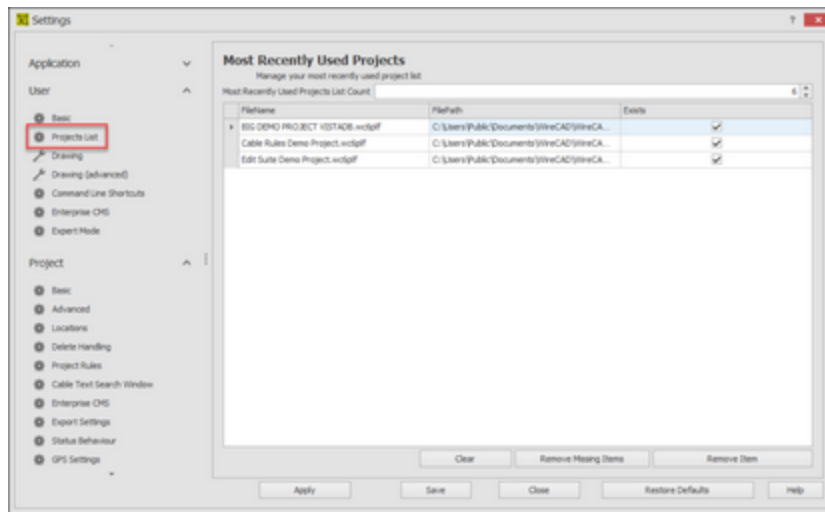
• **Automatically Index Project and Drawings** - For more information see the [Understanding Indexes](#)^[133] topic. True will re-index the project on open and drawings on save.

• **Show Index Operation Progress** - Displays the progress bar during index operations.

5.2.1.9.2.2 Projects List

Navigation: [Reference](#)^[288] > [Dialogs](#)^[326] > [Application Menu Dialogs](#)^[326] > [The Settings Dialog](#)^[346] > [User Settings](#)^[354] >





The Most Recently Used (MRU) projects list. The depth of the list is set. As soon as the list count is exceeded the oldest item is discarded.

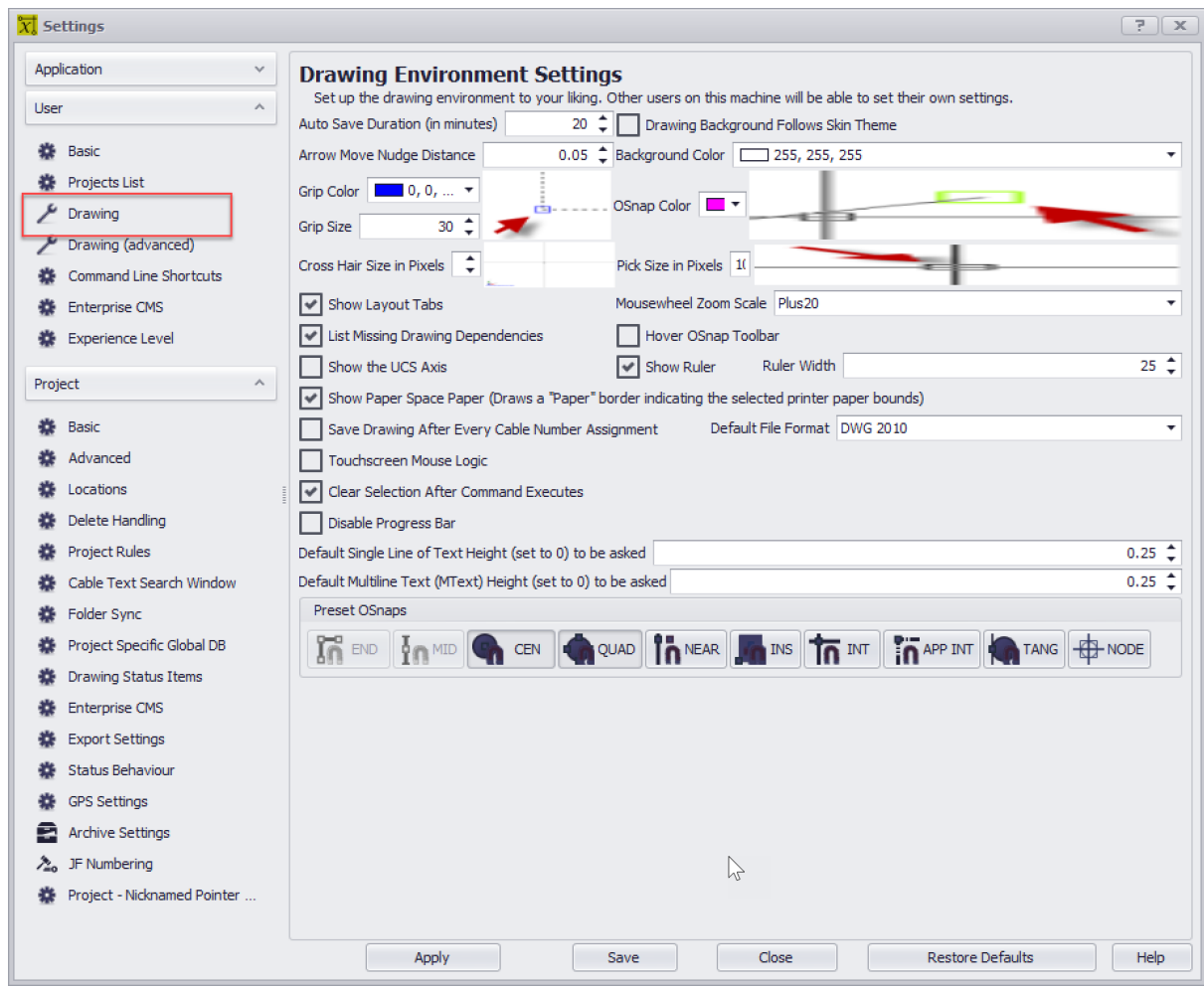
Projects List Panel Options

- **Most Recently Use Projects List Count** - How many items allowed in the list.
- **Projects List** - displays the name path and whether the project can be reached.
- **Clear** - Reset the list.
- **Remove Missing Items** - Remove items that cannot be found because the path has changed or is currently unavailable.
- **Remove Item** - Remove the selected row.

5.2.1.9.2.3 Drawing

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [User Settings](#) ^[354] >





Common user specific drawing environment settings.

User - Drawing Panel Options

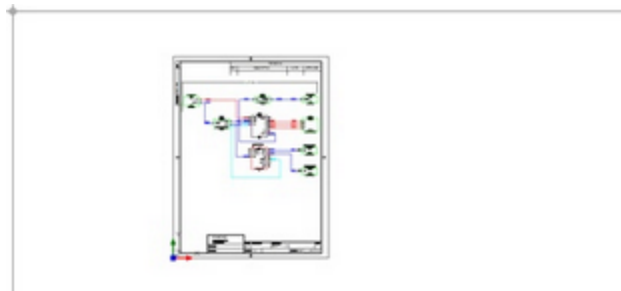
- **Auto Save Duration** - Sets the Auto Save Duration. Auto Save saves only drawings that are modified but not currently involved in a function.
- **Arrow Move Nudge Distance** - When nudging a selection how far do we move in the direction.
- **Grip Color, Grip Size, Cross Hair Size**
- **Show UCS Axis** - Shows the UCS Axis:



- **Show Paper Space Paper** - When a layout is selected, show the current page size as returned by the selected printer as a white "page."



True



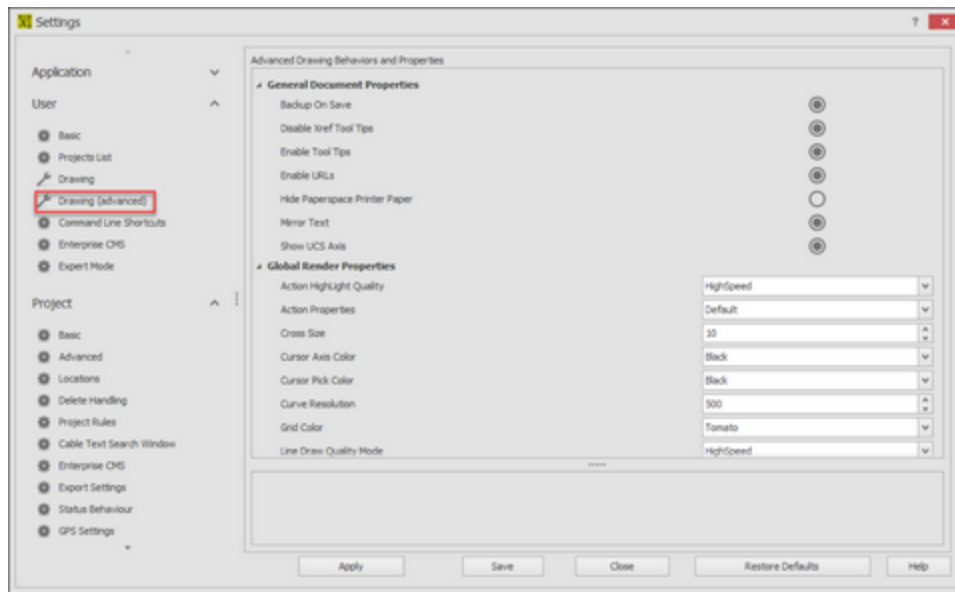
False

- **Save Drawing After Every Cable Number Assignment** - Setting this to true is not recommended but will speed the assignment process. You are responsible then for saving the drawing to ensure drawing/database parity.
- **Touchscreen Mouse Logic**
- **Clear Selection After Command Executes** - Overrides the default behavior of leaving the selection set in tact after an operation.
- **Drawing Background Follows Skin Theme, Background Color, OSnap Color**
- **Pick Size** - Sets the pick size in pixels. The pick determines the search window aperture when clicking the cursor.
- **Disable Progress Bar** - turn off the main form progress bar during drawing loading.
- **Default Single Line of Text Height** - Sets the TextHeight property of a single line of text in DU if non zero.
- **Default Multiline Text (MText) Height** - Sets the TextHeight property of a MText in DU if non zero.

- **Preset OSnaps** - Set the default OSnaps whenever WireCAD turns on the OSnaps it will base the selection on these presets.

5.2.1.9.2.4 Drawing (advanced)

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Application Menu Dialogs](#) [326] > [The Settings Dialog](#) [346] > [User Settings](#) [354] >



These are really granular controls of the render engine. It is beyond the scope of this manual to explain each we will hit some, but play with them if you are interested and see how they behave.

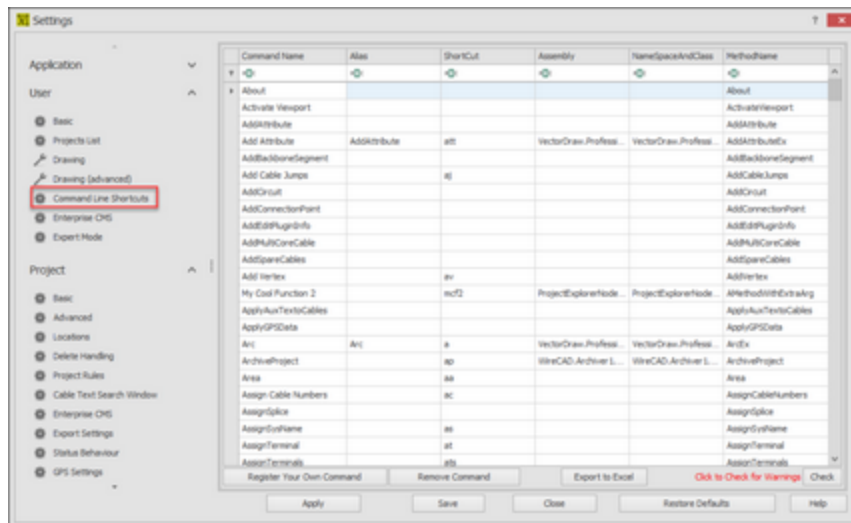
User - Drawing Advanced Panel Options

- **BackupOnSave** - Saves the file with the ~.bak extension before overwriting the existing file.
- **DisableShowPrinterPaper** - Same as **Show Paper Space Paper**.
- **DisableXrefToolTips** - Sometimes XREF tool-tips can be annoying.
- **EnableToolTips** - Show tool-tips or not.
- **EnableURLs** - URLs of object will be opened.
- **MirrorText** - Controls how the Mirror command deals with text.
- **CurveResolution** - How many segments to a circle.
- **PickAdd** - Sets whether a user selection action replaces the current selection or adds to it.
- **SelectActionKey** - Set a key value used in combination with mouse-down to add or remove selected item(s) from a selection.

5.2.1.9.2.5 Command Line Shortcuts

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Application Menu Dialogs](#) [326] > [The Settings Dialog](#) [346] > [User Settings](#) [354] >





User - Command Line Shortcuts Panel Options

- **Register Your Own Command, Remove Command** - Developer function.
- **Check** - Checks the shortcuts to see if you have any duplicates.
- **Commands List** - You may freely edit any of the columns:

CommandName

Alias

ShortCut

Even though the grid allows you to edit the following fields if you are not a developer you should **NOT** edit any of the columns:

Assembly

NameSpaceAndClass

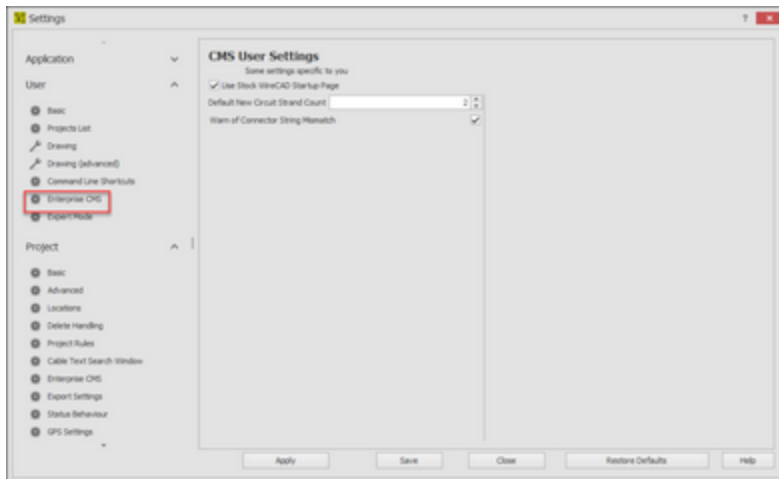
MethodName

You will break WireCAD.

5.2.1.9.2.6 CMS

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [User Settings](#) ^[354] >





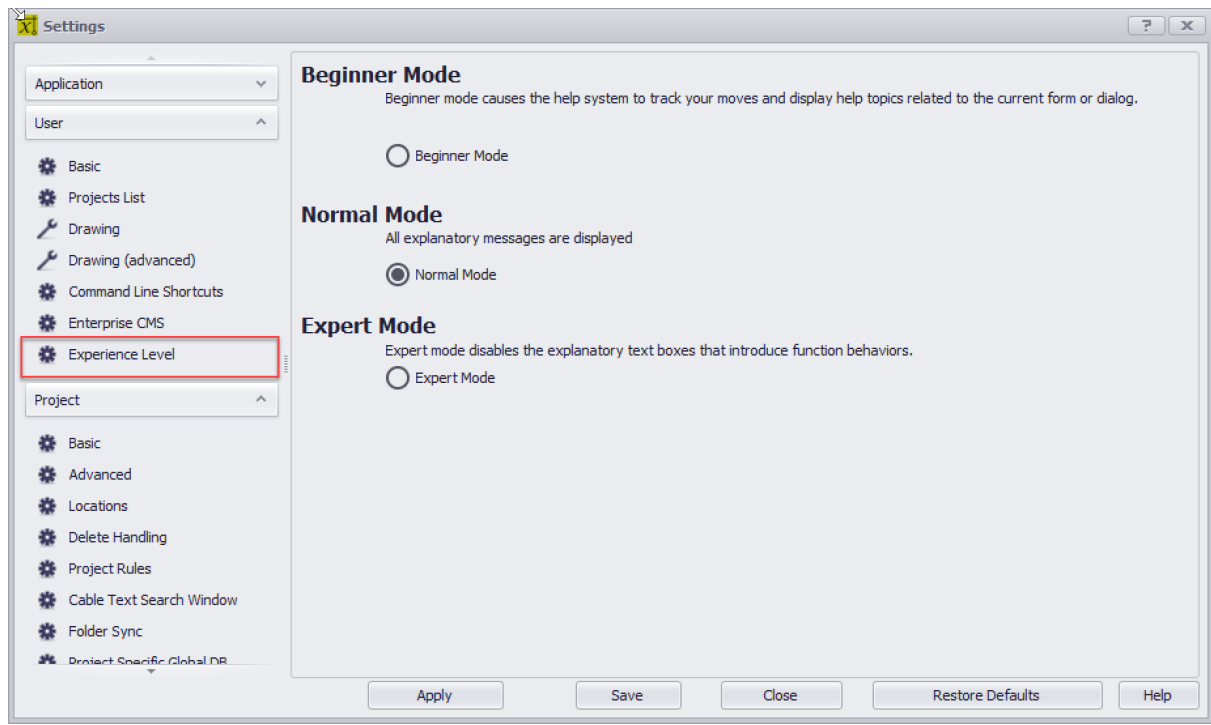
Enterprise CMS Panel Options

- **Use Stock WireCAD Startup Page** - No longer active. Has no effect.
- **Default New Circuit Strand Count** - When creating a new circuit, a default strand count for jumpers is created. This will change that strand count to a different value.
- **Warn of Connector String Mismatch** - Having this box checked will allow a warning message to display anytime WireCAD detects that 2 ports being connected have a different connector type.

5.2.1.9.2.7 Experience Level

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ > [The Settings Dialog](#) ³⁴⁶ > [User Settings](#) ³⁵⁴ >





User - Experience Level Panel Options

Explanation

Fine tune WireCAD for you.

- **Beginner Mode** - tightly couples the help system with your actions in the application. The help system will display topics based on the current form or dialog.
- **Normal Mode** - Does not automatically couple to the help system but does show all explanatory dialogs.
- **Enable Expert Mode** - Once you learn WireCAD functions, the explanatory dialogs that show before a function executes can be disabled.

While not a comprehensive list (there are roughly 100 instances in the code that reference the Expert mode), some of the functions that will execute without explanation are:

- Rack Builder tool.
- De-assign Cable.
- De-assign SysName.
- Reset All SysNames.
- Reset Selected SysNames.
- Reset All Cables.

- Reset Selected Cables.
- Project switching will not inform you that the current project needs to be closed.
- Equipment Library tips will not be shown.
- Cables Grid>File>Expert Mode.
- Batch Plot.
- Copy, move, delete in drawings with selections made outside the visible window will not warn you.
- Pack Project Database.
- Pack Global Database.

5.2.1.9.3 Project Settings

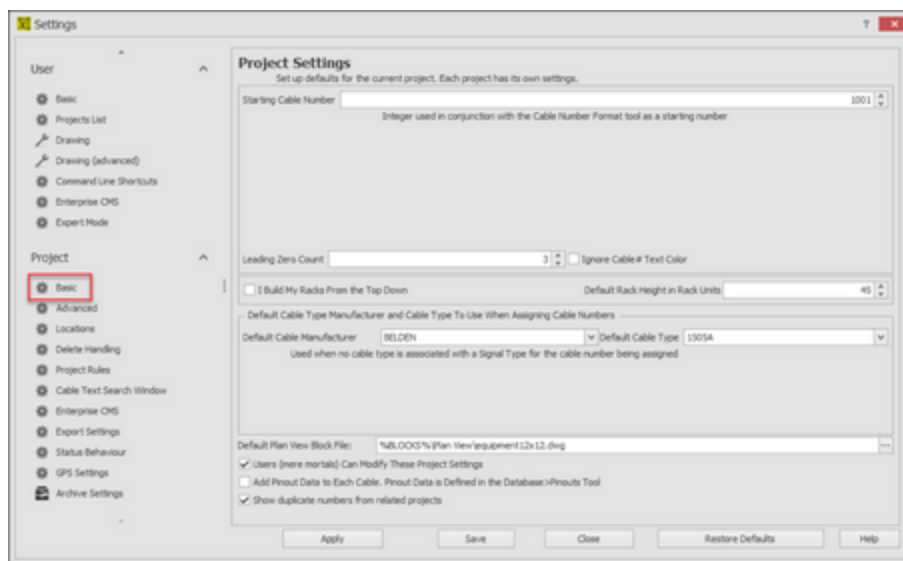
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] >



Settings specific to the current project. These settings are stored in the project folder as ProjectSettings9.xml.

5.2.1.9.3.1 Project Basic

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >



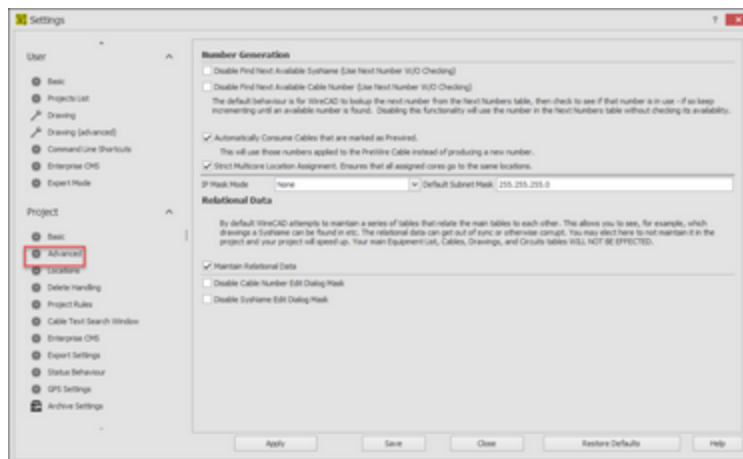
Basic Panel Options

- **Starting Cable Number** - The default starting number of ALL sequences.
- **Leading Zero Count** - How many leading zeros in SysNames and Cable Numbers.

- **Ignore Cable # Text Color** - Does not color the Cable Number text entities.
- **Build My Racks From the Top Down, Default Rack Height**
- **Default Cable Type** - This is the default of defaults. The Project Signal Types table can override this.
- **Default Plan View Block File** - If no Plan View File is defined on the Equipment definition then this will be used by the Populate Equipment function in the Plan View Tools.
- **Users (mere mortals) Can Modify These Project Settings** - If you can see and edit this you are a WireCAD Administrator or have not set up WireCAD Security.
- **Add Pinout Data to Each Cable** - Allow pinout data to be attached to a cable as it is being assigned. The pinout criteria must match in order to be a candidate for inclusion in the list of available pinouts for a cable.
- **Use the device display settings** - This setting prioritizes any Equipment Library device settings saved with the device over the settings saved for the project.
- **Show duplicate numbers from related projects** - Only if Verbose assignments are being done.

5.2.1.9.3.2 Advanced

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >



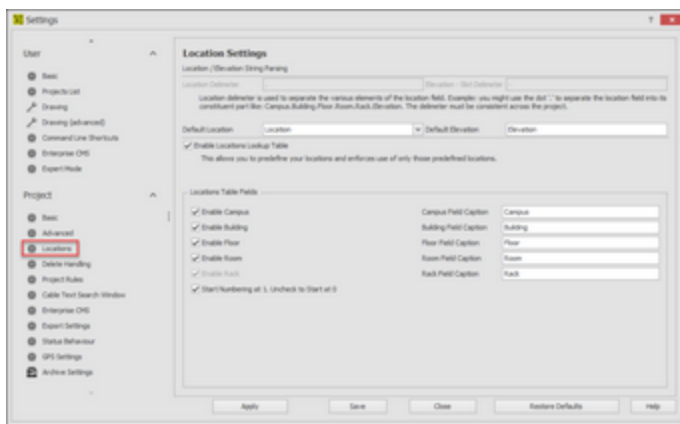
Project - Advanced Panel Options

- **Disable Find Next Available SysName, Disable Find Next Available Cable Number** - The default behavior is for WireCAD to look-up the next number from the Next Numbers table, then check to see if that number is in use - if so keep incrementing until an available number is found. Disabling this functionality will use the number in the Next Numbers table without checking its availability.
- **Automatically Consume Prewire Cables** - This will use those numbers applied to the PreWire Cable instead of producing a new number.
- **Strict Multi-core Assignment** - Enforces a policy that all cores in each end of a multi-core cable must originate from the same location.
- **IP Mask**

- Maintain Relational Data** - See this discussion on Understanding Indexes. Disabling this will not disable the indexing function if you call Re-index but will not attempt to initiate the data in real time.
- Disable Cable Number Edit Dialog Mask** - If you find the masking of the Edit Cable Data Cable Number textbox to be cumbersome you can disable it here. This also disables the mask on the SysName assignment dialog.

5.2.1.9.3.3 Locations

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >



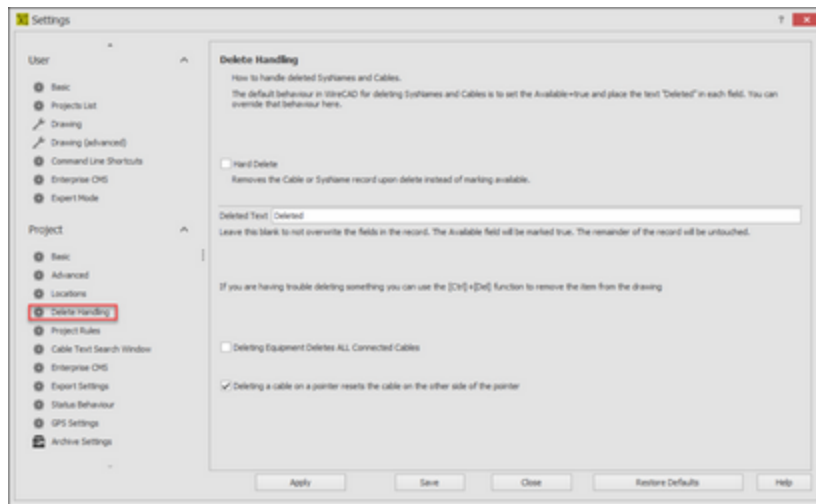
Project - Locations Panel Options

- Location Delimiter** - Settable on project creation only this is the delimiter that separates CAMPUS from BUILDING, BUILDING from FLOOR, etc. The default is a [.]. This will create Qualified Locations in the form of: Campus.Building.Floor.Room.Rack.
- Elevation-Slot Delimiter** - The delimiter that separates the elevation numeric from the slot numeric. For example: a device located in elevation 20 at slot 5 with a delimiter of a dash [-] would be typed and 20-5.
- Enable Locations Look-up** - If disabled the locations field may be typed into directly. This may lead to confusion or referential issues, ie; user a calls something Rack 1 and user b calls the same thing RK 01.
- Locations Table Fields** - Enable those that you will use and call them what you will.
- Start Numbering at 1** - Only applies if you are auto generating locations.

5.2.1.9.3.4 Delete Handling

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >



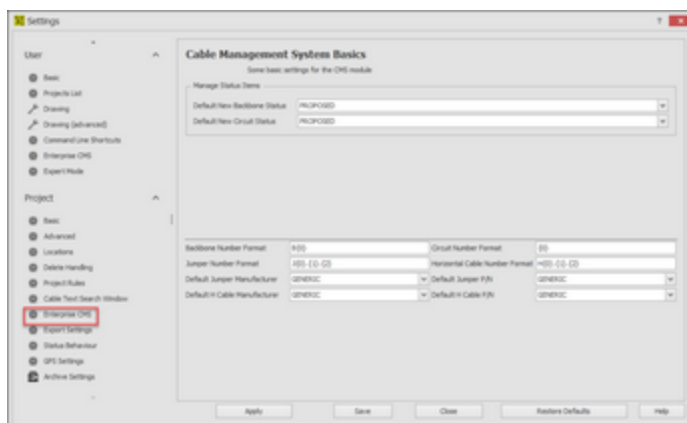


Delete Handling Panel Options

- **Hard Delete** - Overrides the default behavior which is to mark a deleted cable Available=true and place the Deleted Text in key fields. Enabled this removes the row from the table upon deletion.
- **Deleted Text** - If deleted and not Hard Deleting the text to place in fields. Leave empty to leave existing text in place.
- **Deleting Equipment Deletes ALL Connected Cables** - Works with functional blocks, concept blocks, and terminals.
- **Deleting a cable on a pointer resets the cable on the other side of the pointer** - The other sheet will be opened and updated. This can take some time.

5.2.1.9.3.5 Enterprise CMS

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >

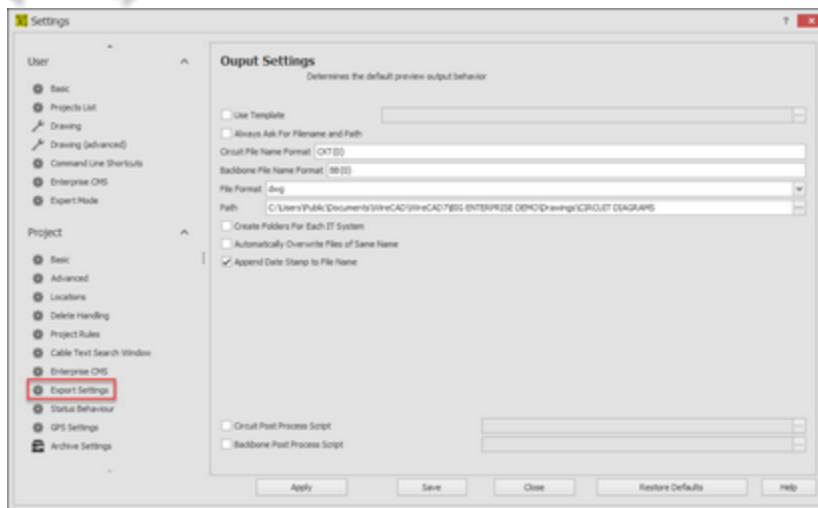


Enterprise CMS Panel Options

- **Manage Status Items** - When creating a backbone, you have the ability to mark a "status" on that backbone such as "In Use", "Proposed" ETC. The Status Behavior settings panel defines the Status Items in the list.
- **Backbone Number Format** - The variable {0} contains the next number in the Next Numbers grid for Backbones. Define the string format of the next Backbone number.
- **Jumper Number Format** -
The variable {0} contains the Circuit base number.
The variable {1} contains the Strand Number of the circuit for this jumper
The variable {2} contains the Ordinal Number of the circuit for this jumper.
Define the string format of the next Jumper number.
- **Default Jumper Manufacturer and P/N, Default Horizontal Cable Manufacturer and P/N**
- **Circuit Number Format** - The variable {0} contains the next number in the Next Numbers grid for Circuits. Define the string format of the next Circuit number.
- **Horizontal Cable Number Format** - The variable {0} contains the next number in the Next Numbers grid for Horizontal Cables. Define the string format of the next Horizontal Cable number.

5.2.1.9.3.6 Export Settings CMS

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Application Menu Dialogs](#) ³²⁶ > [The Settings Dialog](#) ³⁴⁶ > [Project Settings](#) ³⁶⁴ >



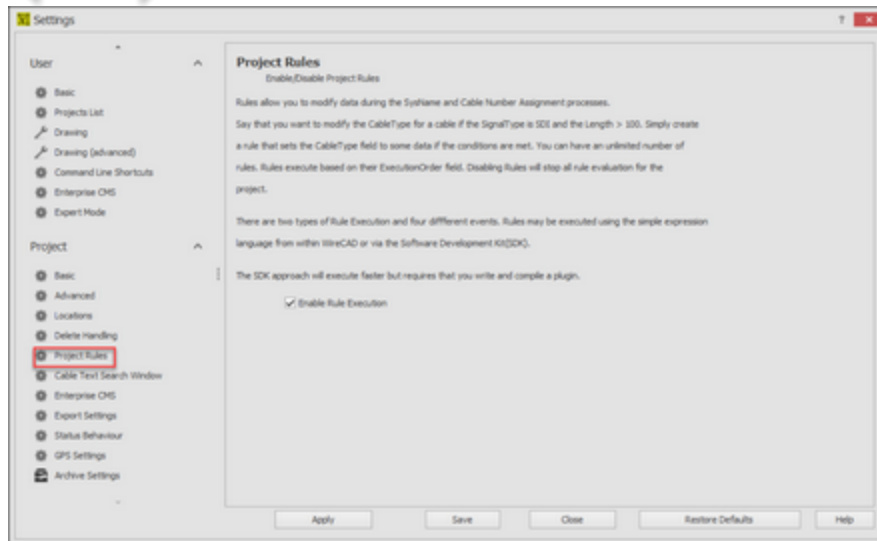
Project - Export Settings Panel Options

- **Use Template** - Path to a template drawing that the output will be exported into. Your template drawings can contain any page borders or layouts and settings that you wish.
- **Always Ask for FileName and Path** - You will be involved in the filename selection and path.
- **Circuit FileName Format** - Sets the file name format for Circuit output. {0} = Circuit Name.
- **Backbone FileName Format** - Sets the file name format for Backbone output. {0} = Backbone Number.
- **File Format** - Presets the output format.

- **Path** - Where do we output.
- **Create Folder for Each IT Systems**
- **Automatically Overwrite Files of Same Name**
- **Append Date Stamp to FileName**
- **Circuit Post Process Script, Backbone Post Process Script** - Path to a c# file which will be run post export but pre write-to-disk. See the [Post Process Scripts](#) ^[283] topic for more information.

5.2.1.9.3.7 Project Rules

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >



Project - Project Rules Panel Options

Explanation

Rules allow you to modify data during the SysName and Cable Number assignment processes. There are four events or contexts in which rules may execute:
 SysName assignment pre and post dialog show/dismiss.
 Cable Number assignment pre and post dialog show/dismiss.

Related Topics:

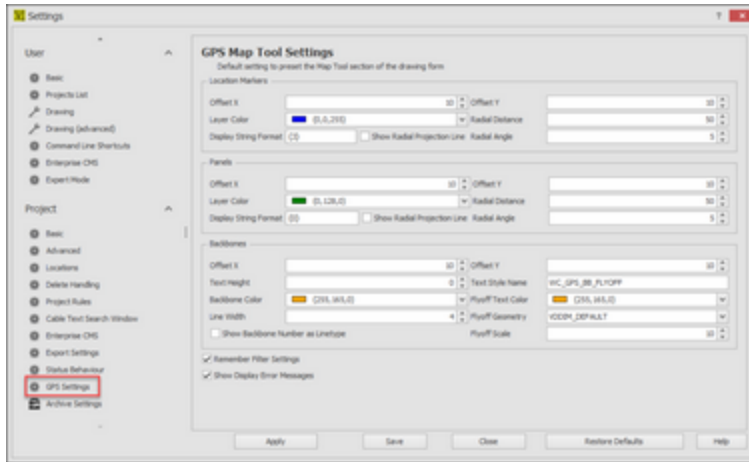
[Project Rules](#) ^[203]
[Expression Syntax](#) ^[204]

- **Enable Rule Execution** - Enable/Disable all rule and rule event execution.

5.2.1.9.3.8 Status Behaviour CMS

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >

Menu: **Application Menu > Settings[Project][GPS Settings]**



Location Marker Settings

Offset X, Y: Used by the **Location Overlay Mode** = Offset.

Layer Color: sets the color the location marker will paint.

Radial Distance: Used in conjunction with the **Location Overlay Mode** = Radial and the **Show Radial Project Lines** to paint a line from the radial arc center to the location marker.

Display String Format: Sets the variable used to display the name of the location on the marker in the drawing or map.

The following variables can be used:

- {0} = Campus.
- {1} = Building.
- {2} = Floor.
- {3} = Room.
- {4} = Rack.
- {5} = Qualified Location.

Show Radial Project Lines: Used in conjunction with the **Location Overlay Mode** = Radial. Sets the display of a projection line from the arc center to the **Radial Distance** and location marker.

Radial Angle: Used in conjunction with the **Location Overlay Mode** = Radial. Sets the rotation angle of the radially displayed location markers.

Panel (Equipment) Marker Settings

Offset X, Y: Used by the **Panel Overlay Mode** = Offset.

Layer Color: sets the color the panel marker will paint.

Radial Distance: Used in conjunction with the **Panel Overlay Mode** = Radial and the **Show Radial Project Lines** to paint a line from the radial arc center to the panel marker.

Display String Format: Sets the variable used to display the name of the panel on the marker in the drawing or map.

The following variables can be used:

- {0} = All.
- {1} = SysName.
- {2} = Alias.

{3} = Location.
 {4} = Manufacturer.
 {5} = Equipment Name/Model.

Show Radial Project Lines: Used in conjunction with the **Panel Overlay Mode** = Radial. Sets the display of a projection line from the arc center to the **Radial Distance** and panel marker.

Radial Angle: Used in conjunction with the **Panel Overlay Mode** = Radial. Sets the rotation angle of the radially displayed panel markers.

Backbone Marker Settings

Offset X, Y: Used by the **Backbone Show Mode** not = ByGPSLocation to offset the flyoffs.

Text Height and Style: Sets the appearance of flyoff text.

Backbone Color: sets the color the backbone marker will paint.

Backbone Text Color: sets the color the backbone flyoff marker will paint.

Line Width: Sets the pen stroke width of the backbone.

Flyoff Geometry: Sets the block to use when a flyoff is created.

Flyoff Scale: Sets the scale factor of the **Flyoff Geometry**.

Show Backbone Number as Linetype: Creates a new linetype with the backbone text as part of the linetype.

General Settings

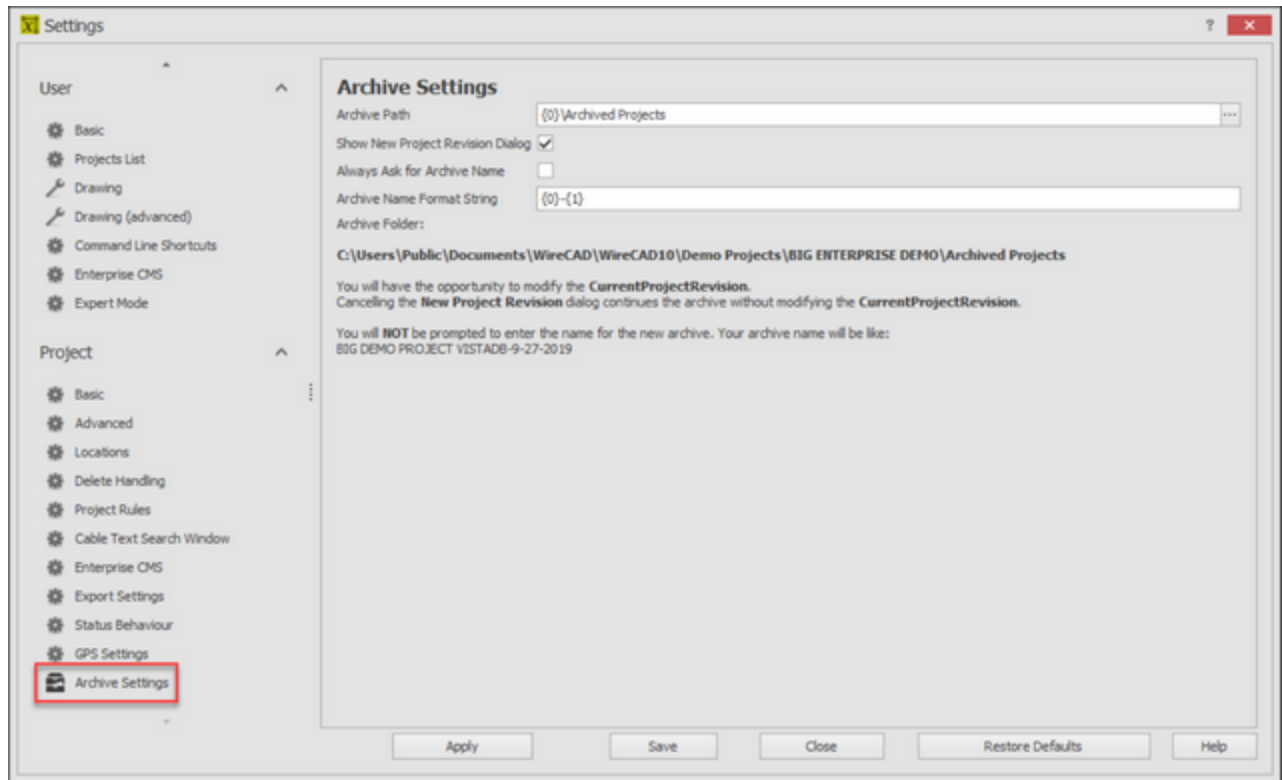
Remember Filter Settings: Persist the filter data across forms and sessions.

Display Error Messages: Show a list of the problems found when showing Panels and Backbones.

5.2.1.9.3.10 Archive Project

Navigation: [Reference](#)  [Dialogs](#)  [Application Menu Dialogs](#)  [The Settings Dialog](#) 
 > [Project Settings](#)  >





Project - Archive Project Panel Options

Settings that affect the Archive tool.

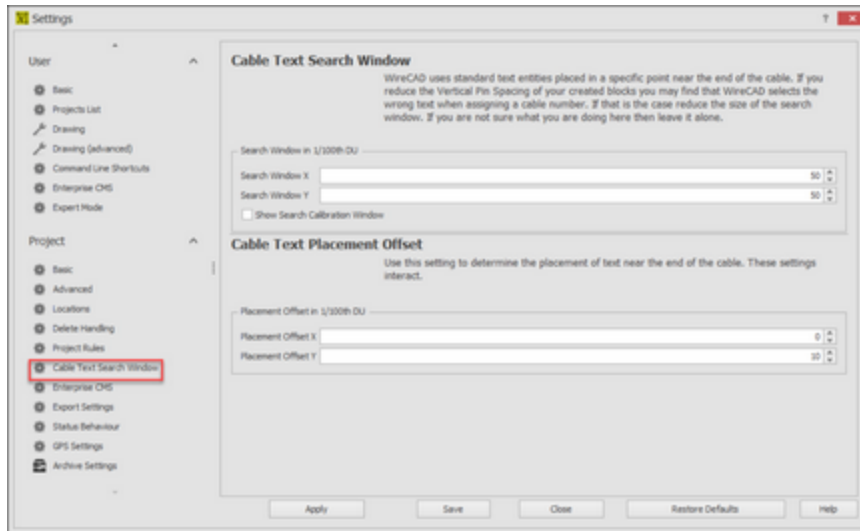
- **Archive Path** - path to the archive root folder. The Archive Name will be used to create a folder within this root folder. Use {0} for the current project path.
- **Show New Project Revision Dialog** - Set whether the **New Project Revision** dialog is shown and populated during the archive process. If the **New Project Revision** dialog is canceled the **CurrentProjectRevision** will not be incremented.
- **Always Ask for Archive Name** - Sets whether you are asked for the archive name or whether the archive is automatically named using the **Archive Name Format String**.
- **Archive Name Format String** - The name of the folder the project is contained within as well as the archived project name.
 {0} = Current Project Name.
 {1} = UTC short date string.
 {2} = UTC short time string.

Related Topics:

[Archive Project](#) ^[202]

5.2.1.9.3.11 Cable Text Search Window

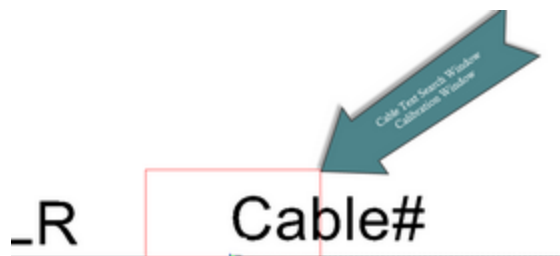
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Application Menu Dialogs](#) ^[326] > [The Settings Dialog](#) ^[346] > [Project Settings](#) ^[364] >



Project - Cable Text Search Window Options

WireCAD uses standard text entities placed in a specific point near the end of the cable. If you reduce the Vertical Pin Spacing of your created blocks you may find that WireCAD selects the wrong text when assigning a cable number. If that is the case reduce the size of the search window.

- **Search Window X** - Sets the x offset from the end of the cable that we search for text.
- **Search Window Y** - Sets the y offset from the end of the cable that we search for text.
- **Show Search Calibration Window** - Renders a red rectangle representing the search window. Be sure to turn this off before doing actual work in the application.

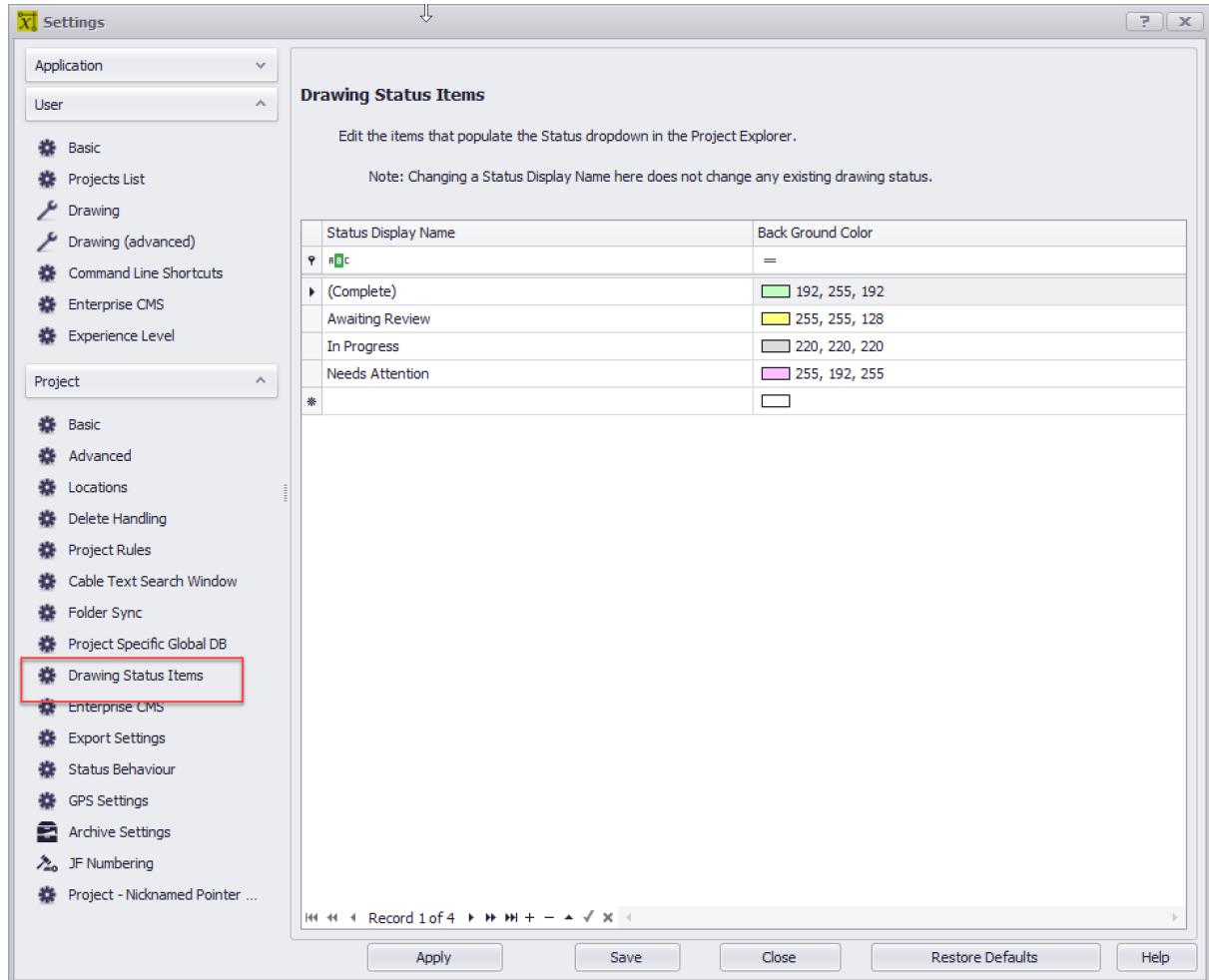


Search Calibration Window

- **Placement Offset X** - Sets the x offset of the Cable Number text placement. This setting interacts with the **Search Window X**. If you set the **Placement Offset X** outside the **Search Window X** you will need to modify the **Search Window X** to accommodate the new placement.

• **Placement Offset Y** - Sets the y offset of the Cable Number text placement. This setting interacts with the **Search Window Y**. If you set the **Placement Offset Y** outside the **Search Window Y** you will need to modify the **Search Window Y** to accommodate the new placement.

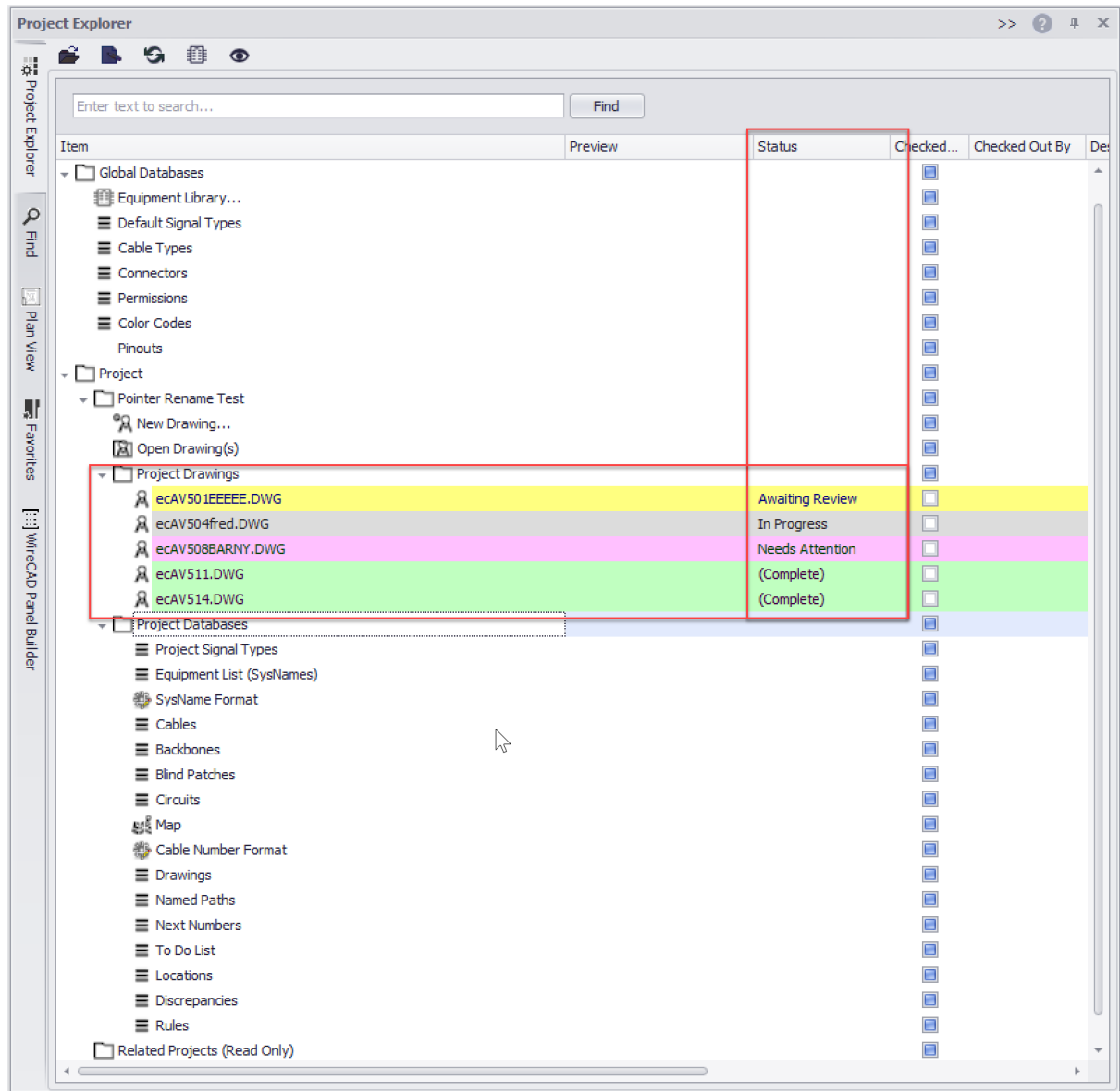
5.2.1.9.3.12 Drawing Status Items



Explanation:

List of **Drawing Status Items** to help organize your drawings.

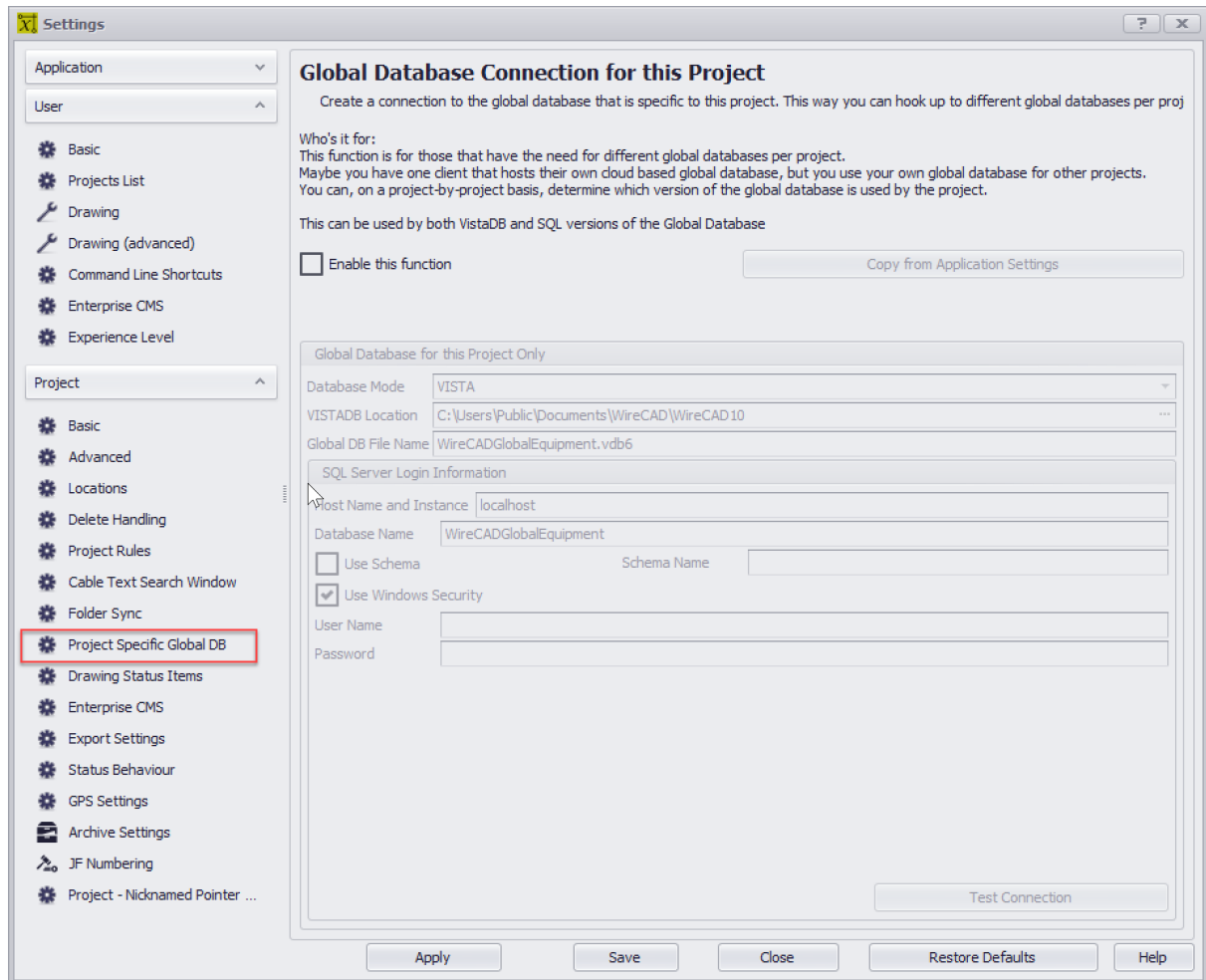
These Status Items are used by the Project Explorer [Status] field.

**Properties**

Status Display Name - Set the caption to be displayed in the drop-down.

Background Color - Set the color to paint the row.

5.2.1.9.3.13 Project Specific Global Database



Explanation:

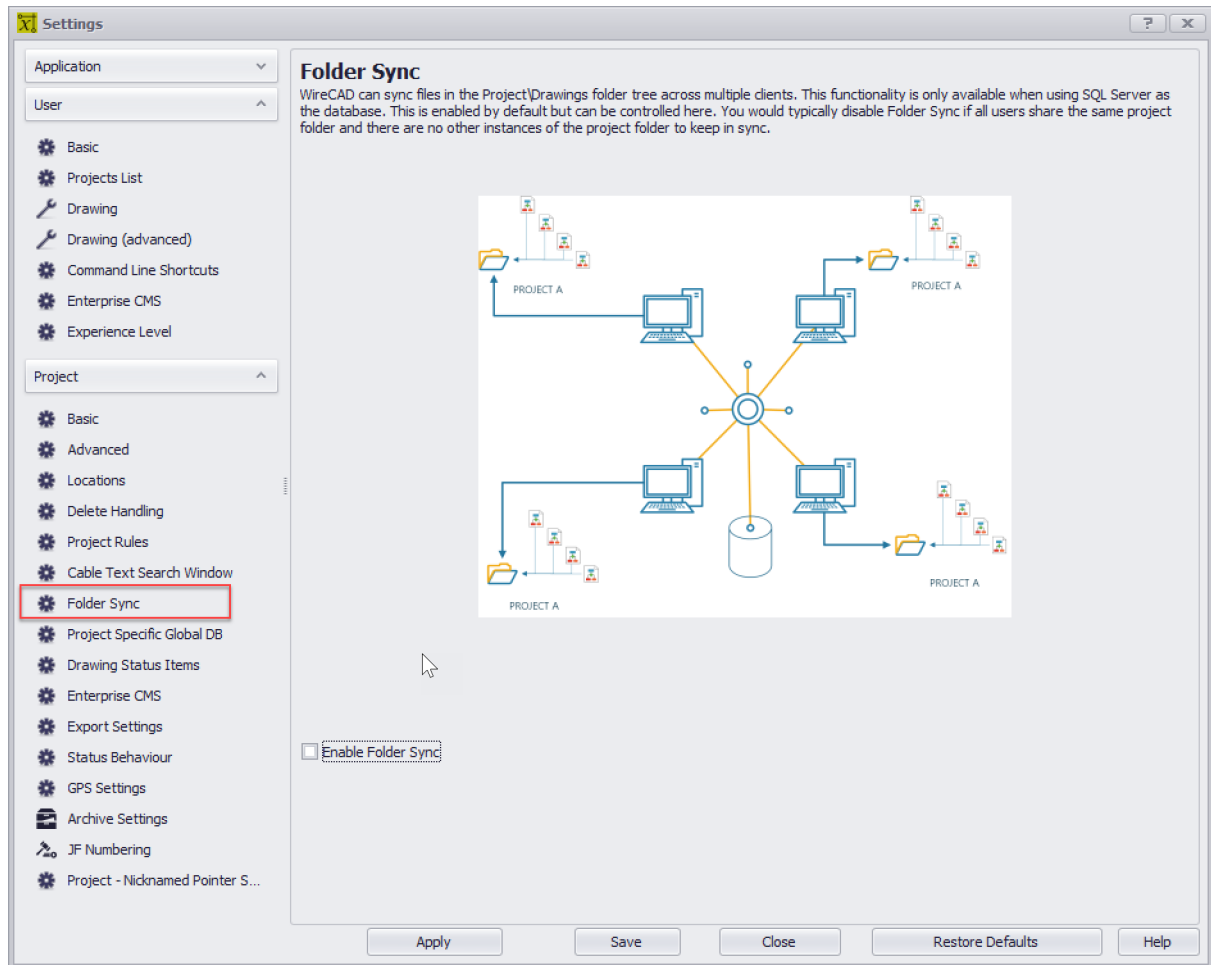
Sometimes it is desirable to have a different Global Database (GDB) for a project. This could be because you have different client requirements for a project from your normal setup.

If this is the case, you can copy the GDB (WireCADGlobalEquipment.vdb6) to another directory (probably the project folder) and enable this function. The point to the GDB in the project folder.

Alternately, you can hook up to a SQL Server database host.

All of the flexibility that is provided in the [GDB Location](#) settings panel is replicated here.

5.2.1.9.3.14 Folder Sync



Explanation

WireCAD provides a sync mechanism that can sync local project drawing folders across different locations.

This is similar to Dropbox, OneDrive, Google Drive, Jungle Drive, etc.

This can be used to share the project drawings across the planet.

Typically this function is used with Azure SQL databases in the cloud, where users may be in different cities.

Folders are synchronized by scanning the Project Drawings folder and comparing the file size, hash and timestamp with those in the Drawings table of the project database.

If the file timestamp is older than the server, the server version is downloaded to the local folder.

If the file timestamp is newer than the server, the local version is uploaded to the server.

Limitations

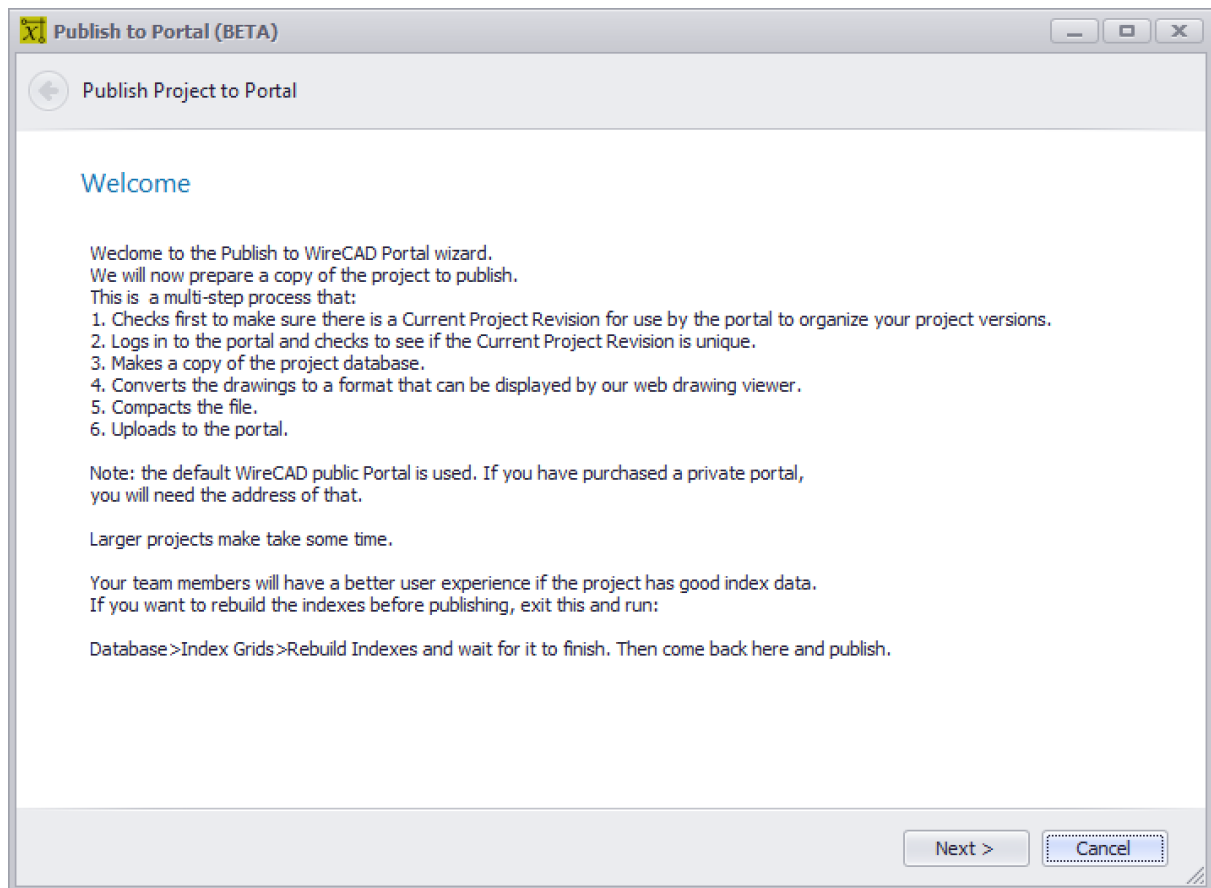
The sync tool only scans the project drawings folder tree.

The sync tool only works with SQL or SQL Azure databases.

5.2.2 Project Dialogs

Contained herein are explanations about the dialogs that appear on the Project Menu.

5.2.2.1 Publish Project to Portal



Explanation

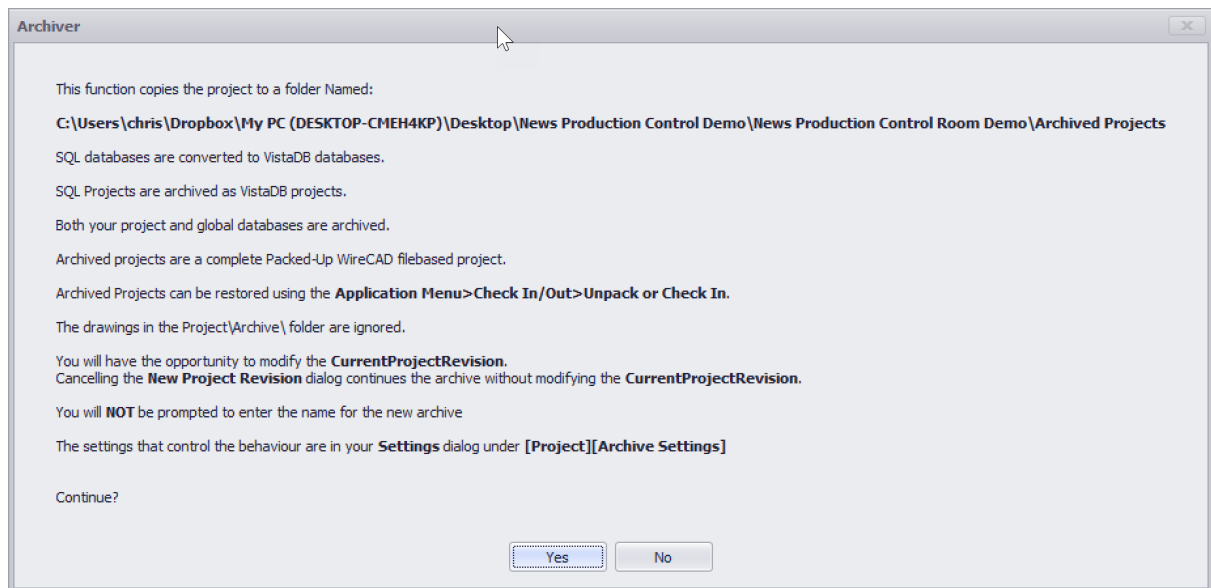
WireCAD can publish the project to the WireCAD teams portal at teams.wirecad.com.

This allows you to share the project to anyone that you invite to the team.

Team members can view and redline drawings as well as view data and message team members from any modern browser.

You must first create an account at: teams.wirecad.com.

You will need your WireCAD key to create an account.

5.2.2.2 Archive Project

Explanation

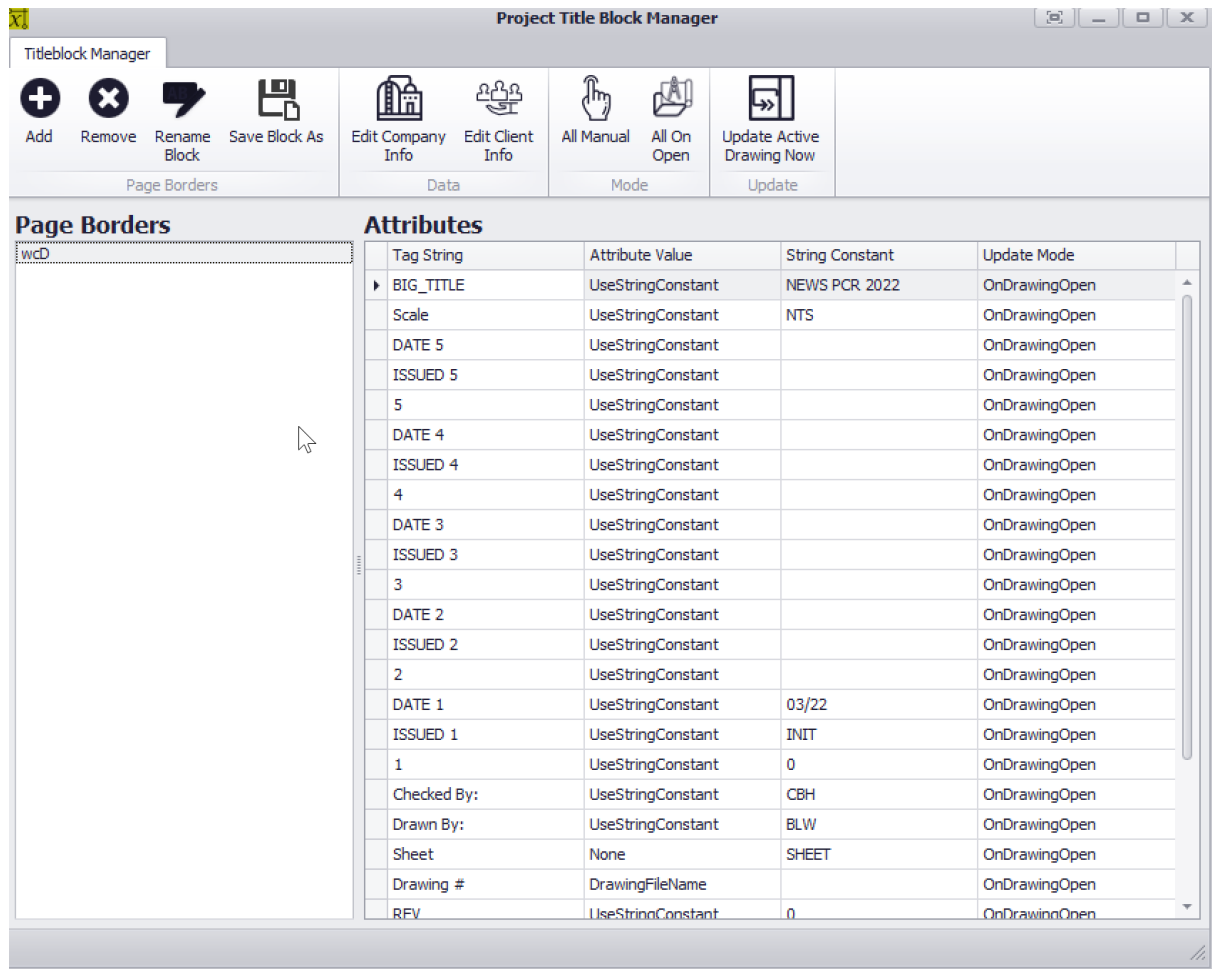
The Archive project tool which inherits the settings from the Project Settings[Archive] settings panel. The archive tool packs up the project and stores it the location determined in the settings.

Related Topics:

[Project Archive Tool Settings](#) ³⁷²

[Unpack Project](#) ³³⁸

[Check-In Project](#) ³³⁹

5.2.2.3 Titleblock Manager

Explanation

The Titleblock Manager tool associates the attributes of a titleblock with variables or string constants.

Menu Items

- [Add] - Add a titleblock to the list of Page Border blocks.
- [Remove] - Remove the selected block from the list of Page Border blocks.
- [Rename Block] - Rename the selected block in the list of Page Border blocks.
- [Save Block As] - Create a copy of the selected block with a new name in the list of Page Border blocks.
- [Edit Company Info] - Open the [Company Info](#)^[352] settings panel.
- [Edit Client Info] - Open the [Account Info](#)^[340] dialog.
- [All Manual] - Set the **Update Mode** for all attributes to Manual mode. In Manual mode the titleblock is only updated when you open the **TitleBlock Manager** dialog and click the **[Update Active Drawing Now]** button.
- [All on Open] - Set the **Update Mode** for all attributes to **OnDrawingOpen** mode. In **OnDrawingOpen** mode the attributes that are flagged as **OnDrawingOpen** will be updated when the drawing is opened.
- [Update Active Drawing Now] - Do it!

Variables

Most of these are self explanatory. Those that are not will be defined.

- None - Do nothing
- ProjectName
- ProjectDescription
- ProjectLeadPerson
- ProjectInfoUser1 - User1 field from the Application Menu>Project Info form.
- ProjectInfoUser2 - User2 field from the Application Menu>Project Info form.
- ProjectInfoUser3 - User3 field from the Application Menu>Project Info form.
- ProjectInfoUser4 - User4 field from the Application Menu>Project Info form.
- ProjectPath
- ProjectCreationDate
- ProjectModifiedDate
- CurrentProjectRevision

See this [form](#)^[540].

- DrawingName
- DrawingFileName - file name with extension.
- DrawingFileNameAndPath
- DrawingLayoutName - the layout where the Page Border lives.
- DrawingCreationDate
- DrawingModifiedDate
- DrawingUser1 - User1 from the Drawings table.
- DrawingUser2
- DrawingUser3
- DrawingUser4
- DrawingUser5
- DrawingUser6
- DrawingUser7
- DrawingUser8
- DrawingUser9
- DrawingUser10

- DrawingUser11
- DrawingUser12
- DrawingShort - the Short description from the Drawings table.
- DrawingLong - the Long description from the Drawings table.
- DrawingFSCM
- DrawingNumber

Date and Time

- DateTimeNow - The current date and time as right now.
- DateShort - The short date as of now.
- DateLong - The long date as of now.

Environment

- UserName - The environment User Name.

Strings

- UseStringConstant - Use the data from the **String Constant** field.

See this [settings panel](#)^[352].

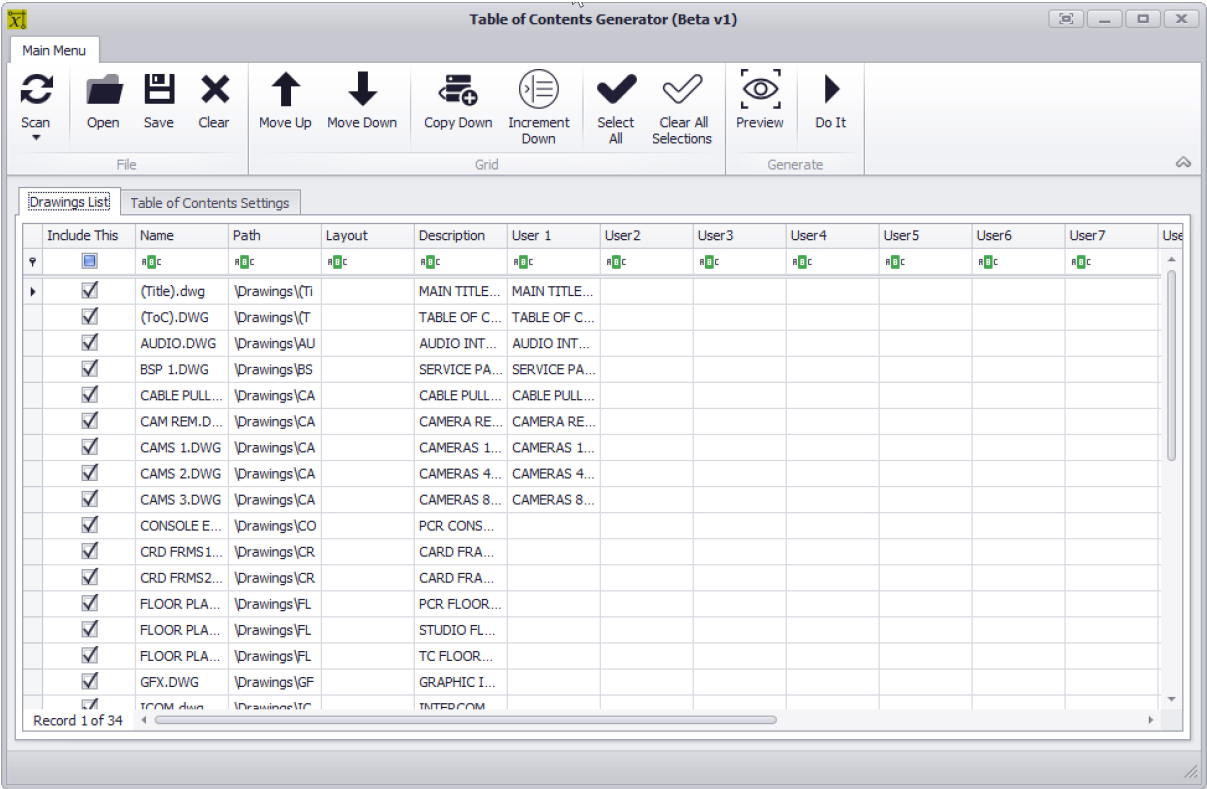
- CompanyName - Company Name from the Application Settings[Application]>Organization settings panel.
- CompanyAddress1 - Address1 from the Application Settings[Application]>Organization settings panel.
- CompanyAddress2 - Address2 from the Application Settings[Application]>Organization settings panel.
- CompanyCity - City from the Application Settings[Application]>Organization settings panel.
- CompanyState - State from the Application Settings[Application]>Organization settings panel.
- CompanyPostalCode - Zipcode from the Application Settings[Application]>Organization settings panel.
- CompanyPhone - Company Name from the Application Settings[Application]>Organization settings panel.
- CompanyWebsite - Company Name from the Application Settings[Application]>Organization settings panel.
- CompanyCountry - Company Name from the Application Settings[Application]>Organization settings panel.

See this [dialog](#)^[340].

- ClientAccountName - Account Name from the Application Menu>Account Info dialog.
- ClientContactName - Contact Name from the Application Menu>Account Info dialog.
- ClientAddress1 - Address1 from the Application Menu>Account Info dialog.
- ClientAddress2 - Address2 from the Application Menu>Account Info dialog.
- ClientCity - City from the Application Menu>Account Info dialog.
- ClientState - State from the Application Menu>Account Info dialog.
- ClientPostalCode - Zipcode from the Application Menu>Account Info dialog.
- ClientPhone - Phone from the Application Menu>Account Info dialog.
- ClientFax - Fax from the Application Menu>Account Info dialog.
- ClientWebsite - Website from the Application Menu>Account Info dialog.
- ClientEmail - Email from the Application Menu>Account Info dialog.
- ClientInfoUser1 - User1 from the Application Menu>Account Info dialog.
- ClientInfoUser2 - User2 from the Application Menu>Account Info dialog.
- ClientInfoUser3 - User3 from the Application Menu>Account Info dialog.
- ClientInfoUser4 - User4 from the Application Menu>Account Info dialog.

5.2.2.4 ToC

Table of Contents Generator



Explanation:

This tool generates a Table of Contents from the data in the Drawings table.

It is best to start with an empty drawing.

The ToC can have up to 7 display columns. The column caption and display data can be set based on the Table of Contents Settings [TAB] settings.

To limit the number of columns displayed is based on the ColumnCount property.

Page No	Drawing	Layout	Path	Description	User1	User Bool
1	CTITLE		\\Drawings\\CTitle.dwg	MAIN TITLE PAGE	MAIN TITLE PAGE	Unchecked
2	CTOC		\\Drawings\\CTOC.DWG	TABLE OF CONTENTS	TABLE OF CONTENTS	Unchecked
3	AUDIO		\\Drawings\\AUDIO.DWG	AUDIO INTERCONNECT	AUDIO INTERCONNECT	Unchecked
4	BSP 1		\\Drawings\\BSP 1.DWG	SERVICE PANEL INTERCONNECT	SERVICE PANEL INTERCONNECT	Unchecked
5	CABLE PULLS		\\Drawings\\CABLE PULLS.DWG	CABLEPULL LIDDER DIAGRAMS	CABLEPULL LIDDER DIAGRAMS	Unchecked
6	CAM REM		\\Drawings\\CAM REM.DWG	CAMERA REMOTE CONTROL INTERCONNECT	CAMERA REMOTE CONTROL	Unchecked
7	CAMS 1		\\Drawings\\CAMS 1.DWG	CAMERAS 1-3 INTERCONNECT	CAMERAS 1-3 INTERCONNECT	Unchecked
8	CAMS 2		\\Drawings\\CAMS 2.DWG	CAMERAS 4-7 INTERCONNECT	CAMERAS 4-7 INTERCONNECT	Unchecked
9	CAMS 3		\\Drawings\\CAMS 3.DWG	CAMERAS 8-10 INTERCONNECT	CAMERAS 8-10 INTERCONNECT	Unchecked
10	CONSOLE ELEVATIONS		\\Drawings\\CONSOLE ELEVATIONS.DWG	FOR CONSOLE ELEVATIONS		Unchecked
11	ORD FRMS1		\\Drawings\\ORD FRMS1.DWG	ORD FRAMES 1-4 ELEVATION		Unchecked
12	ORD FRMS2		\\Drawings\\ORD FRMS2.DWG	ORD FRAMES 5-8 ELEVATION		Unchecked
13	FLOOR PLAN POR		\\Drawings\\FLOOR PLAN POR.DWG	POR FLOOR PLAN		Unchecked
14	FLOOR PLAN STUDIO		\\Drawings\\FLOOR PLAN STUDIO.DWG	STUDIO FLOOR PLAN		Unchecked
15	FLOOR PLAN		\\Drawings\\FLOOR PLAN.DWG	TO FLOOR PLAN		Unchecked
16	GPX		\\Drawings\\GPX.DWG	GRAPHIC INTERCONNECT		Unchecked
17	IGDM		\\Drawings\\IGDM.dwg	INTERROOM INTERCONNECT		Unchecked
18	KVM		\\Drawings\\KVM.DWG	KVM INTERCONNECT		Unchecked
19	LAN		\\Drawings\\LAN.DWG	NETWORK INTERCONNECT		Unchecked
20	MVIEW		\\Drawings\\MVIEW.DWG	MULTIVIEW INTERCONNECT		Unchecked
21	PINOUTS		\\Drawings\\PINOUTS.dwg	PINOUTS		Unchecked
22	PROMPT		\\Drawings\\PROMPT.DWG	PROMPTER INTERCONNECT		Unchecked
23	QCD		\\Drawings\\QCD.DWG	QUALITY CONTROL INTERCONNECT		Unchecked
24	RACK ELEVATIONS		\\Drawings\\RACK ELEVATIONS.DWG	RACKS ELEVATION VIEW		Unchecked
25	REF		\\Drawings\\REF.dwg	REFERENCE GENERATION AND OUTPUT		Unchecked
26	revert test		\\Drawings\\revert test.DWG			Unchecked
27	ROUTER		\\Drawings\\ROUTER.dwg	ROUTER INTERCONNECT		Unchecked
28	RTR CTRL		\\Drawings\\RTR CTRL.DWG	ROUTER CONTROL		Unchecked
29	SROS1		\\Drawings\\SROS1.DWG	SOURCE BUS 1-34 INTERCONNECT		Unchecked
30	SROS2		\\Drawings\\SROS2.DWG	SOURCE BUS 35-60 INTERCONNECT		Unchecked
31	SROS3		\\Drawings\\SROS3.DWG	OTHER SOURCE BUSES AND GLUE		Unchecked
32	SWR		\\Drawings\\SWR.DWG	SWITCHER INTERCONNECT		Unchecked
titleblock						
33	WCD		\\Drawings\\titleblock0.dwg			Unchecked
\\Drawings						
34	UTL1		\\Drawings\\UTL1.dwg	UTILITY		Unchecked

Generated ToC**Dialog Options****Menu Items:**

Scan>Scan - Populates the list with all of the drawing in the project.

Scan>Layouts - Populates the list with all of the drawings and their layouts from the project.

Open - Open an existing ToC file.

Save - Save the current settings to the project root folder.

Clear - Clear the list contents.

Move Up - Move the selected drawing/layout up in the list.

Move Down - Move the selected drawing/layout down in the list.

Copy Down - Copy the selected row down.

Increment Down - Append a number sequentially to the selection from the top to bottom.

Select All - Selects all drawing records [Include This] field to be included in the Table of Contents.

Clear All Selections - Clear all check marks in the [Include This] field.

Preview - Builds the table as an overlay of the current drawing and dismisses the dialog so you can see it. Click the **[Esc]** key to return to the ToC Generator.

Click **[Esc]** or click in the drawing to return to the ToC Generator

Page No	Drawing	Layout	Path	Description	User1	User Bool
1	CTITLE		\\Drawings\\CTITLE.dwg	MAIN TITLE PAGE	MAIN TITLE PAGE	Unchecked
2	CTOC		\\Drawings\\CTOC.DWG	TABLE OF CONTENTS	TABLE OF CONTENTS	Unchecked
3	AUDIO		\\Drawings\\AUDIO.DWG	AUDIO INTERCONNECT	AUDIO INTERCONNECT	Unchecked
4	BSP 1		\\Drawings\\BSP 1.DWG	SERVICE PANELS INTERCONNECT	SERVICE PANELS INTERCONNECT	Unchecked
5	CABLE PULLS		\\Drawings\\CABLE PULLS.DWG	CABLE PULL LADDER DIAGRAMS	CABLE PULL LADDER DIAGRAMS	Unchecked
6	CAM REM		\\Drawings\\CAM REM.DWG	CAMERA REMOTE CONTROL INTERCONNECT	CAMERA REMOTE CONTROL	Unchecked
7	CAMS 1		\\Drawings\\CAMS 1.DWG	CAMERAS 1-3 INTERCONNECT	CAMERAS 1-3 INTERCONNECT	Unchecked
8	CAMS 2		\\Drawings\\CAMS 2.DWG	CAMERAS 4-7 INTERCONNECT	CAMERAS 4-7 INTERCONNECT	Unchecked
9	CAMS 3		\\Drawings\\CAMS 3.DWG	CAMERAS 8-10 INTERCONNECT	CAMERAS 8-10 INTERCONNECT	Unchecked
10	CONSOLE ELEVATIONS		\\Drawings\\CONSOLE ELEVATIONS.DWG	FOR CONSOLE ELEVATIONS		Unchecked
11	ORD FRMS1		\\Drawings\\ORD FRMS1.DWG	CARD FRAMES 1-4 ELEVATION		Unchecked
12	ORD FRMS2		\\Drawings\\ORD FRMS2.DWG	CARD FRAMES 5-8 ELEVATION		Unchecked
13	FLOOR PLAN FOR		\\Drawings\\FLOOR PLAN FOR.DWG	FOR FLOOR PLAN		Unchecked
14	FLOOR PLAN STUDIO		\\Drawings\\FLOOR PLAN STUDIO.dwg	STUDIO FLOOR PLAN		Unchecked
15	FLOOR PLAN		\\Drawings\\FLOOR PLAN.DWG	TO FLOOR PLAN		Unchecked
16	GFX		\\Drawings\\GFX.DWG	GRAPHIC INTERCONNECT		Unchecked
17	ICOM		\\Drawings\\ICOM.dwg	INTERCOM INTERCONNECT		Unchecked
18	KVM		\\Drawings\\KVM.DWG	KVM INTERCONNECT		Unchecked
19	LAN		\\Drawings\\LAN.DWG	NETWORK INTERCONNECT		Unchecked
20	MVIEW		\\Drawings\\MVIEW.DWG	MULTIVIEWERS INTERCONNECT		Unchecked
21	PINOUTS		\\Drawings\\PINOUTS.dwg	PINOUTS		Unchecked
22	PRMPT		\\Drawings\\PRMPT.DWG	PROMPTER INTERCONNECT		Unchecked
23	QC		\\Drawings\\QC.DWG	QUALITY CONTROL INTERCONNECT		Unchecked
24	RACK ELEVATIONS		\\Drawings\\RACK ELEVATIONS.DWG	RACKS ELEVATION VIEW		Unchecked
25	REF		\\Drawings\\REF.dwg	REFERENCE DISTRIBUTION		Unchecked

ToC Preview Mode

Do It - Does it. You will be presented with a Yes/No/Cancel dialog. Choosing **[Yes]** creates the ToC and allows you to save a file that you can open in the future.

Choosing **[No]** just creates the ToC.

Choosing **[Cancel]** cancels the Do It function and takes you back to the ToC generator dialog.

Drawings List [TAB]

Drawings Grid. This is the data from the project Drawings table.

The grid data can be assigned to columns in the ToC using the Settings tab as explained below.

Grid Columns:

[Include This] - Check to include this record in the ToC.

[Name] - Drawing Name.

[Path] - Relative path to the drawing in the project drawing folder tree.

[Layout] - The layout if the **Scan>Scan Layouts** option is chosen, otherwise empty.

- [Description] - The DrawingShort field.
- [User 1-12] - The User1-12 text string fields.
- [UserBool1] - The UserBool1 field Boolean value.

Table of Contents Settings [TAB]

Properties

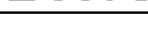
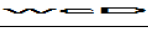
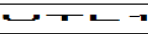
- Column 1-7 Caption - Sets the column header text.
- Column 1-7 Variable - Sets the data source for that column.

Position

- Insertion Point - Sets the coordinate of the upper left corner of the table. Clicking the [...] button temporarily dismisses the dialog so you can pick the insertion point from the current drawing space.

Table

- Column Count - Sets the total number of columns displayed.
- Line Width - Sets the LineWidth property of the lines that comprise the table.
- Max Rows - Sets the max number of rows displayed before the table is continued to the the right.
- Show Folder Names as Headers - If the drawing is in a folder that is a child of the Project\Drawings folder. The folder name will be shown as a header. If all drawings are contained in the root project Drawings folder, this property is ignored.

32			\\Drawings\SWR.DWG	SWITCHER INTERCONNECT		Unchecked
titleblock						
33			\\Drawings\titleblock\wcd.dwg			Unchecked
\\Drawings						
34			\\Drawings\UTL1.dwg	UTILITY		Unchecked

Folders As Headers

- Table Width - Sets the total width in DU of the table. The columns are all displayed within this distance.

Text

- Empty Cell Text - Sets the text string to display if the data is a null or empty string.
- False Value Display Text - Sets the text string to display if a boolean value evaluates to False.
- Heading Height - Sets the text height in DU of the Header text.
- Heading Style - Sets the TextStyle of the Header text.
- Height - Sets the TextHeight property of all text in the ToC in DU that is not part of a header.
- Horizontal Padding - Sets the distance from the left and right side of the cell that text cannot be displayed in.
- Style - Sets the TextStyle property of all text in the ToC that is not part of a header.
- Subheading Height - Sets the TextHeight property in DU of text that is displayed in sub heading. Applies only if the [Show Folder Names as Headers] property is set to true.
- Subheading Style - Sets the TextStyle property of text that is displayed in a sub heading.
- True Value Display Text - Sets the text string to display if a boolean value evaluates to True.
- Vertical Padding - Sets the distance from the top and bottom side of the cell that text cannot be displayed in.

5.2.3 Drawing Dialogs

Navigation: [Reference](#) [288] > [Dialogs](#) [326] >



The following is a set of dialogs associated with Drawing Form.

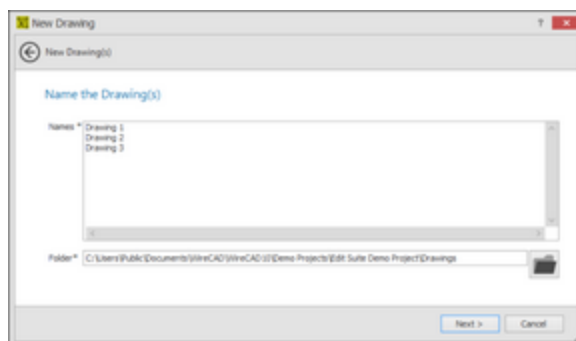
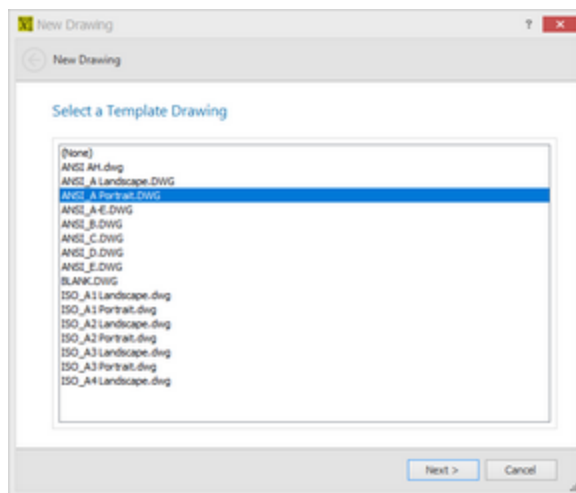
5.2.3.1 CAD Dialogs

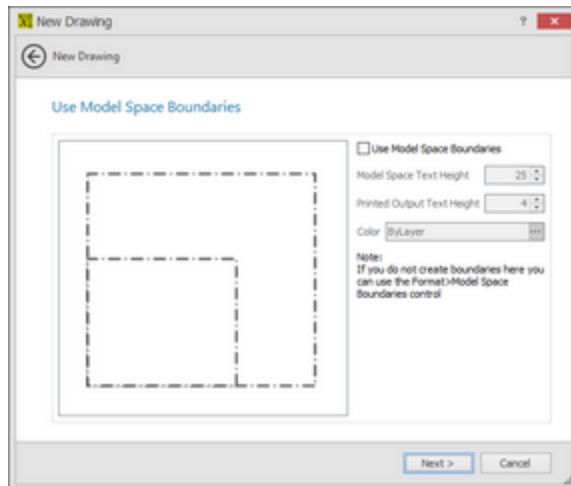
Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] >



5.2.3.1.1 New Drawing Wizard

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >





Drawing > File > New
Commandline: nd

Explanation

Create a new drawing for the project somewhere in the project\drawings\ folder tree.

- **Select a template drawing.** Template drawings are drawings that are placed in the Template Drawings [Support Path](#)^[351]. They typically contain layouts and page borders, but may also be populated with barebones systems.
- **Set whether to display Model Space Boundaries.** These are bounding rectangles based on a scale factor of the Model Space text height used and the desired output text height of the viewport.

Related Topics:

[How To: Create a New Drawing](#)^[78]
[Model Space Boundaries](#)^[400]

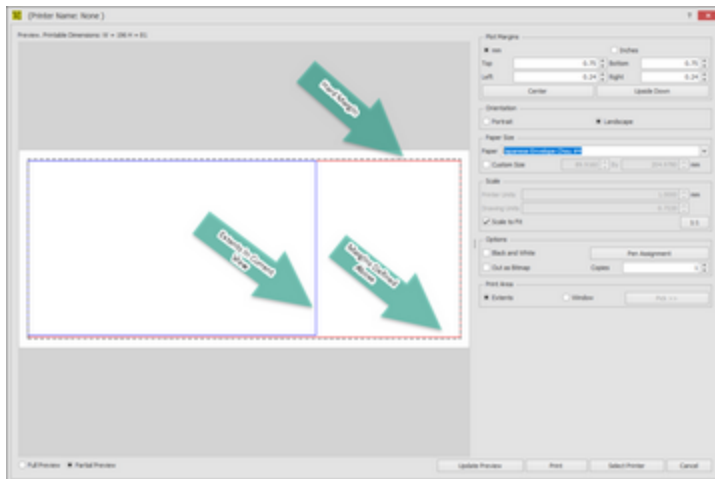
Dialog Options

- **Template Drawings List** - Drawings placed in the %TEMPLATEDRAWINGS% support path. Any drawing is a candidate for saving as a template.
- **Model Space Boundaries** - Your selected template may already have Model Space Boundaries in which case it is unnecessary to include them.

5.2.3.1.2 Print Preview

Navigation: [Reference](#)^[288] > [Dialogs](#)^[326] > [Drawing Dialogs](#)^[389] > [CAD Dialogs](#)^[389] >

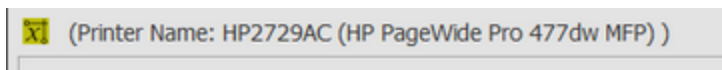




Drawing > File > Print Preview
[Ctrl]+[P]
Commandline: print

Dialog Options

- **Selected Printer** - Make sure that you select a printer first.

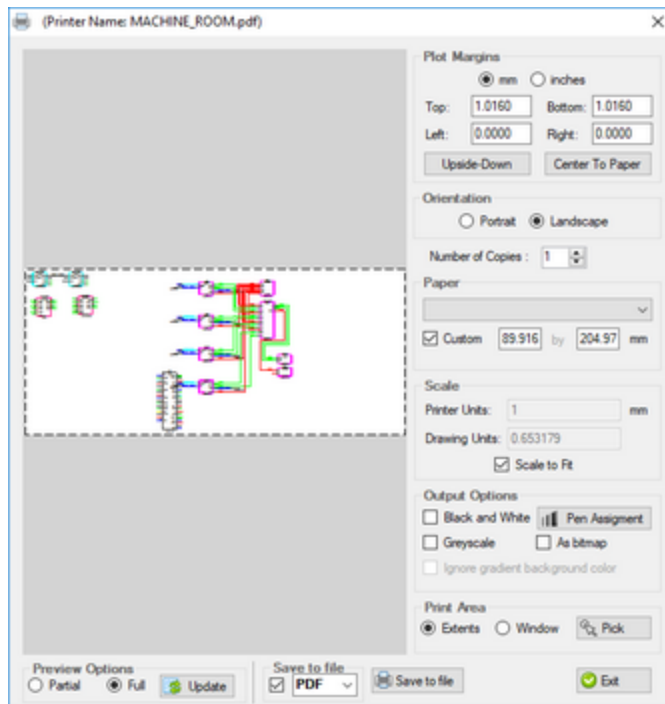


- **Plot Margins** - Set the unit of measure and the Top, Bottom, Left and Right margins. NOTE: the printer driver will return the hard margins of the printer. These are represented by the black dashed line in the Partial Preview mode. Setting the margins less than the hard margin values will have no effect.
- **Center** - Adjusts the left and top margins to center the entities in the given space.
- **Upside Down** - Rotate the output to the plotter.
- **Orientation** - Portrait/Landscape.
- **Paper Size**
- **Scale** - Numerical scaling of the entities to the paper.
- **Scale to Fit** - Fit the entities to the paper.
- **1:1** - Scale 1:1.
- **Black and White**
- **Out as Bitmap** - Render to a bitmap then output that. Has different effect on quality depending on printer/plotter.
- **Copies**
- **Pen Assignment** - Shows the Pen Assignment dialog. You can assign a different line thickness to each of the 255 indexed colors.
- **Print Area** - Pick a window or print the extents.
- **Preview Mode** - Full preview or partial view to show margins.
- **Update Preview** - Refresh the preview.
- **Print**
- **Select Printer**

- Cancel

5.2.3.1.3 PDF Export

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >

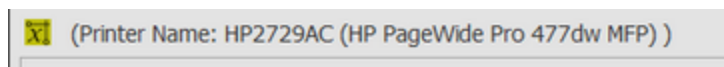


Drawing > File > PDF Export

Commandline: pdf

Dialog Options

- Selected Printer** - Make sure that you select a printer first.



•**Plot Margins** - Set the unit of measure and the Top, Bottom, Left and Right margins. NOTE: the printer driver will return the hard margins of the printer. These are represented by the black dashed line in the Partial Preview mode. Setting the margins less than the hard margin values will have no effect.

•**Center** - Adjusts the left and top margins to center the entities in the given space.

•**Upside Down** - Rotate the output to the plotter.

•**Orientation** - Portrait/Landscape.

•**Paper Size**

•**Scale** - Numerical scaling of the entities to the paper.

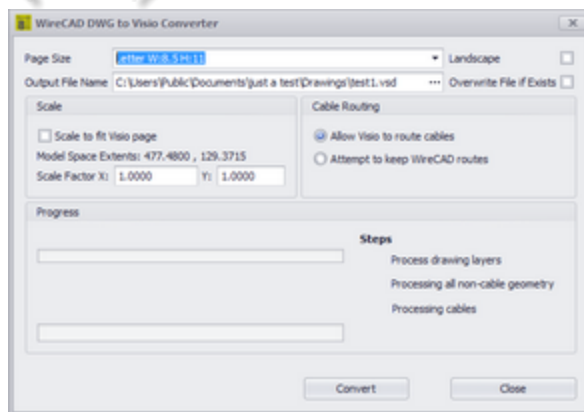
•**Scale to Fit** - Fit the entities to the paper.

•**Black and White**

- **Out as Bitmap** - Render to a bitmap then output that. Has different effect on quality depending on printer/plotter.
- **Copies**
- **Pen Assignment** - Shows the Pen Assignment dialog. You can assign a different line thickness to each of the 255 indexed colors.
- **Print Area** - Pick a window or print the extents.
- **Preview Mode** - Full preview or partial view to show margins.
- **Update Preview** - Refresh the preview.
- **Save to File**
- **Select Printer**
- **Cancel**

5.2.3.1.4 Export to Visio

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >



Drawing > File > Export to Visio
Commandline: visio

Explanation

The WireCAD to Visio converter is an intelligent converter. It creates active working Visio drawings from your WireCAD drawings. What this means is that it is not just a dumb geometric import but rather entities are examined and functioning Visio entities are created. Cables in WireCAD become Visio Connectors allowing movement of blocks while keeping the wires attached.

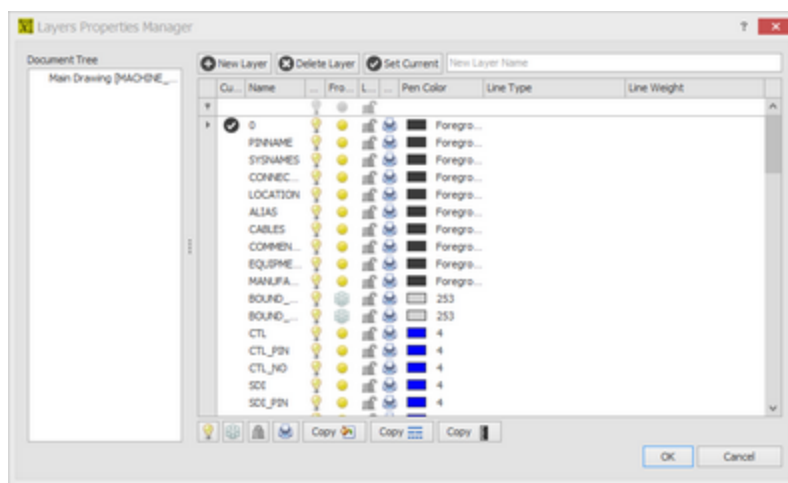
Dialog Options

- **Page Size** - Set the Visio page size.

- **Output File Name**
- **Landscape**
- **Overwrite File if Exists**
- **Scale to Fit** - Fit the WireCAD dwg entities onto the selected page size.
- **Scale Factor** - Manually scale the WireCAD entities to the Visio page size.
- **Cable Routing** - Here is the magic. You can either let Vision route the cables or you can attempt to keep the appearance of your WireCAD routes.
- **Convert** - Do IT!

5.2.3.1.5 Layers

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >



Drawing > Drawing > Layers
Commandline: lay

Explanation

The Layers dialog controls the document Layers collection.

A Layer is the equivalent of the overlay used in paper-based drafting. It is the primary organizational tool in the WireCAD CAD space, 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 the result is the complete drawing.

Having kindred objects on the same layer it is very helpful in order to organize the 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 the Layers editor 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.

Dialog Options

- **Document Tree** - Lists the document's external references. By selecting an item in the document tree you can control the layers of XREF drawings within the current drawing.

- **New Layer**

- **Delete Layer** - Deletes the selected layer as long as it is not the current layer or layer 0.

- **Set Current** - Set's the current layer. All entities added to the drawing are added to the current layer.

- **Status** - The one and only current layer.

- **Name**

- **On** - Is it visible.

- **Frozen** - Is it visible. The difference between Thawed/Frozen and On/Off is a very subtle distinction. Turning a layer off using the ON/OFF setting makes the objects on that layer hidden, but these objects will still be considered part of the drawing. For example, objects that have been turned off are still selectable in the drawing. Selecting it directly on screen of course still isn't possible, as you've nothing to click on. But other ways of selecting objects will still pick it up - try a SELECTALL for example, and your objects that are turned off will be selected. Frozen layers on the other hand are completely off. They are not considered part of the drawing at all, and are therefore not selectable.

- **Lock** - Make the layer unselectable.

- **Pen Color** - Set the pen color for the layer. Only entities that have their PenColor property set to ByLayer will receive this value.





- **Line Type** - Set the Line Type for the layer. Only entities that have their Line Type property set to ByLayer will receive this value.

- **Line Weight** - Set the Line Weight for the layer. Only entities that have their Line Weight property set to ByLayer will receive this value.

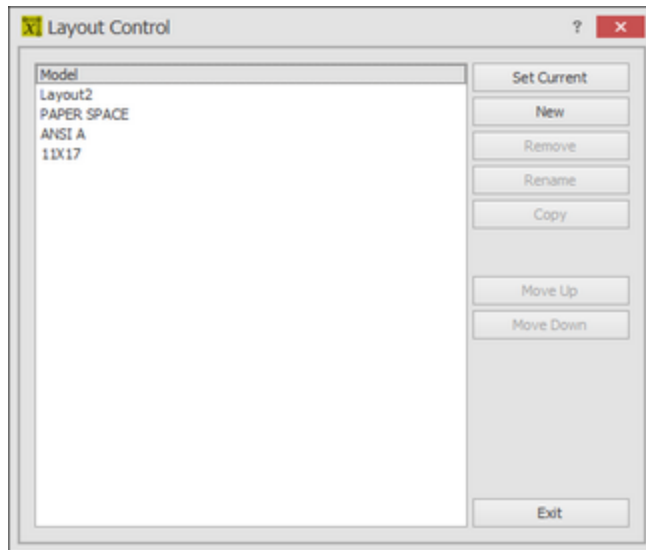
- **OK**

- **Cancel**

5.2.3.1.6 Layouts

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [CAD Dialogs](#)  >





Drawing > Drawing > Layouts
Commandline: layouts

Explanation

A Layout is used to compose or lay out your model drawing for printing. A layout may consist of a title block, one or more viewports, and annotations. As you create a layout, you can design floating Viewport configurations to visualize different details in your drawing.

A layout is a paper space environment that simulates a sheet of paper. In a layout, you can create and position viewport objects, and you can add a title block or other geometry. You can create multiple layouts in a drawing to display various views. Each layout displays the drawing as it will be printed on the sheet of paper.

Typically, when you begin designing a layout environment, you step through the following process:

- 1.Create a model drawing.
- 2.Activate or create a layout.
- 3.Insert a title block.
- 4.Create floating viewports and position them in the layout.
- 5.Set the view scale of the floating viewports.
- 6.Print your layout.

Dialog Options

- Layout List**

- Set Current** - Set the selected item to be the current view.

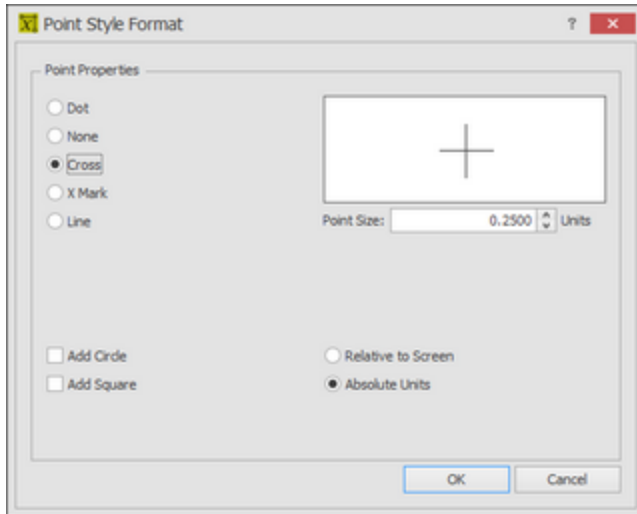
- New, Remove, Rename**

- Copy** - Copy all entities from the selected layout to a new layout with the name of your choosing.

- Move Up/Down** - Reorder the list.

5.2.3.1.7 Point Styles

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >



Drawing > Drawing > Point Styles

Commandline: pointstyles

Explanation

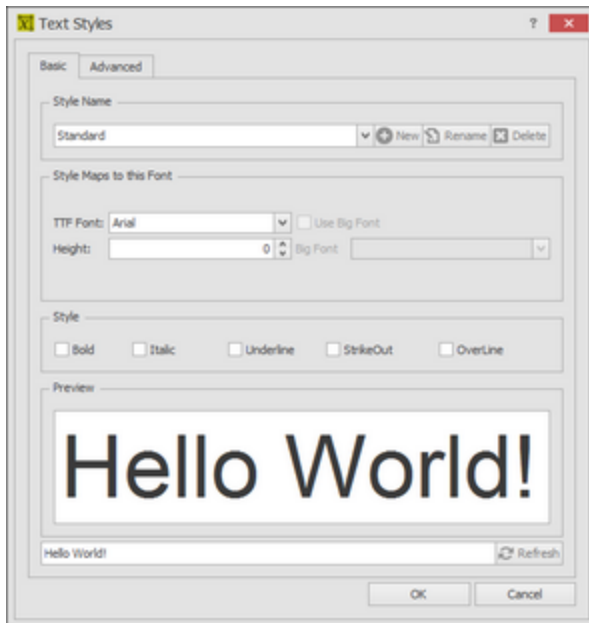
All point entities inserted into the drawings space will render based on the settings here.

NOTE: if you are using the Rack Builder tool WireCAD uses point entities to generate the positioning grids and sets this value programmatically.

5.2.3.1.8 Text Styles

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >





Drawing > Drawing > Text Styles
Commandline: ts

Explanation

TextStyle is a named, saved collection of settings that determines the appearance of text strings. You can create your own text styles which can have specific fonts and text height. You can also specify if the text will be underlined, bold etc.

There is no limit to the number of text styles you can create in your drawing.

The active text style determines the appearance of new text created in the drawing. StyleName of text object will get the value of ActiveTextStyle property.

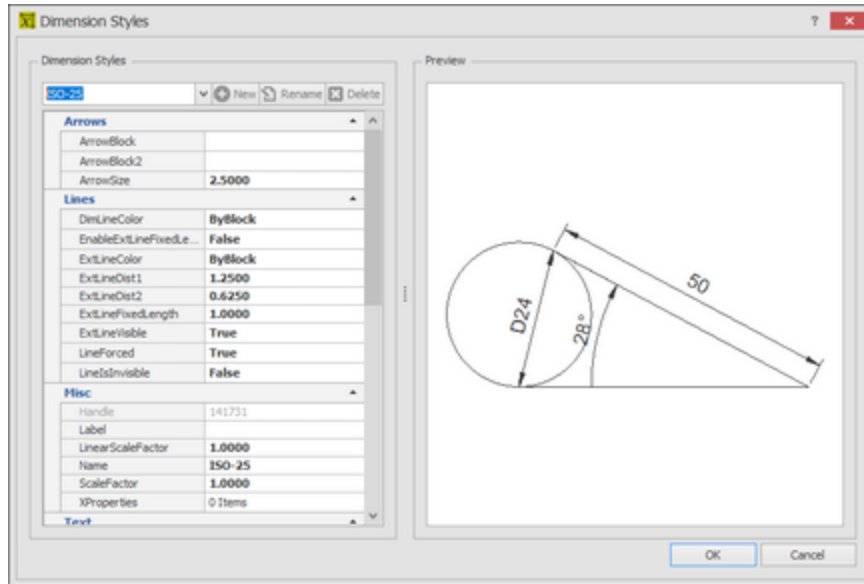
When you enter text, it uses the current text style, which sets the font, size, and other text characteristics. If you want to create text using a different text style, you can make another text style active.

Dialog Options

- **Style Name, Add, Rename, Delete**
- **Font** - Set the font family of the Text Style.
- **Style** - Style attributes.
- **Preview, Preview Text Edit** - Preview of Text Style applied to the Preview Text Edit.
- **Advance Settings** - The advance tab displays a property grid with all of the basic settings and a few more.
- **CodePage** - Sets the character set used to display text.
- **DrawOutline** -
- **Flag** - LeftToRight, Backwards, UpsideDown, etc.
- **ObliqueAngle** - in degrees.
- **WidthFactor** - Value used to stretch text by changing its width.

5.2.3.1.9 Dimension Styles

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >



Drawing > Drawing > Dimension Styles
Commandline: dimstyles

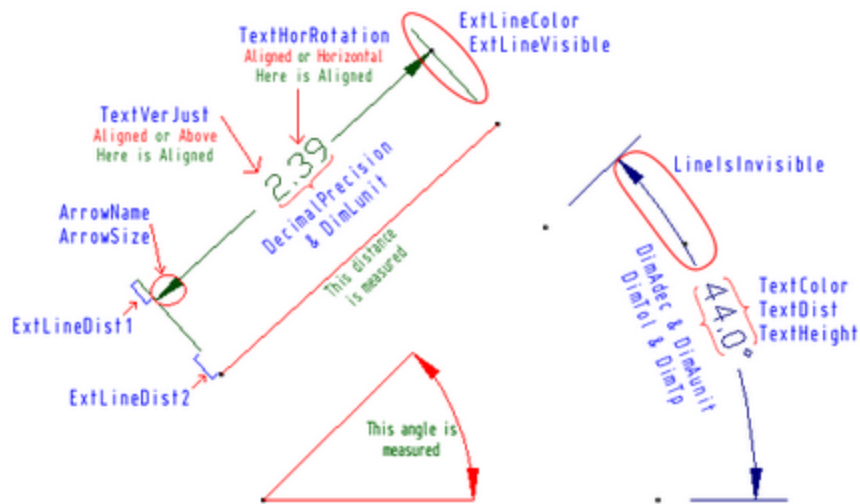
Explanation

A **Dimension Style** is a group of dimension settings that determines the appearance of a dimension .

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.

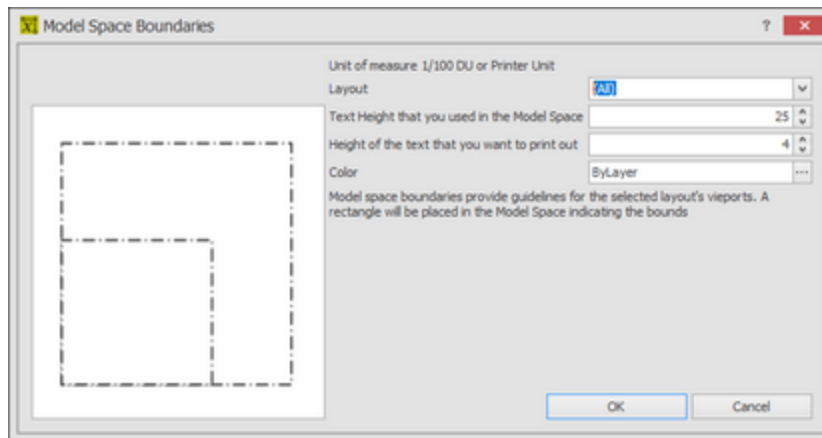
Dialog Options

- **Style Name, Add, Rename, Delete**
- **Preview Pane** - Displays the results of the current settings.
- **Settings** -



5.2.3.1.10 Model Space Boundaries

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >



Drawing > Drawing > Model Space Boundaries
Commandline: bo

Explanation

It's a big model space in there. We can, if we are not careful, create a drawing that is so big that it can't be effectively printed or plotted. In order to have some Idea of where the fences are WireCAD

can place boundaries in the model space. The boundary is created from the viewport. We use the text height as the terms for our equation. We do this because a drawing is considered readable if we can read the text. If we can't read the text the drawing becomes useless.

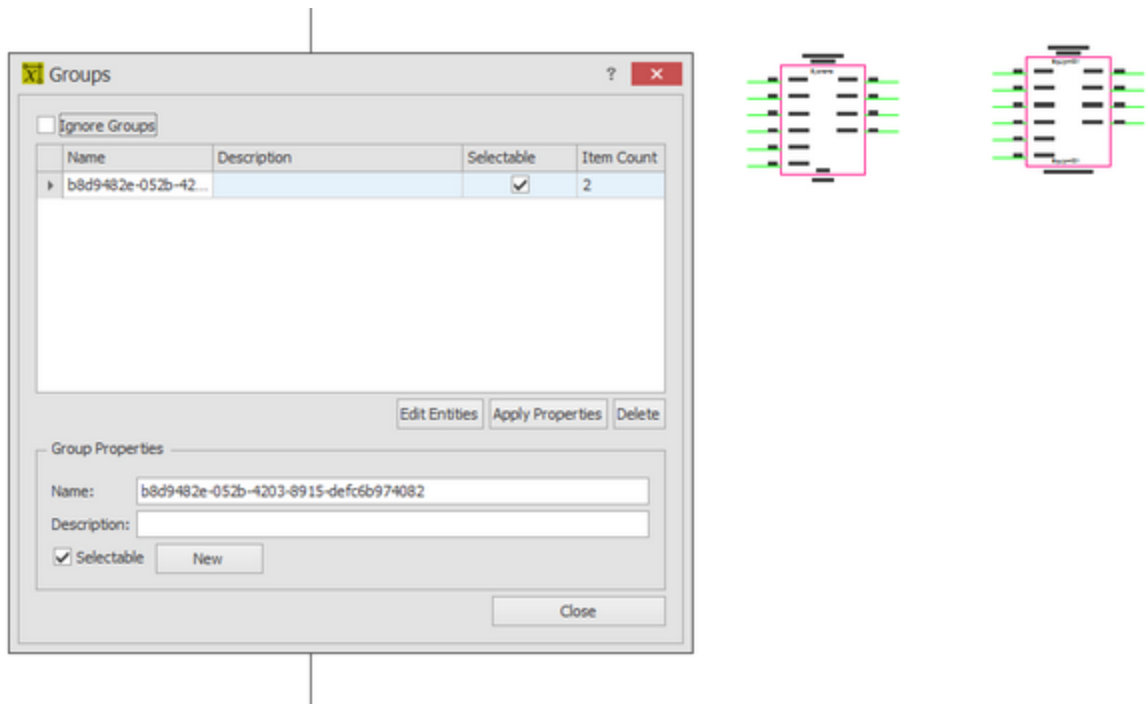
Each boundary will be placed on its own layer and named **bound_layout_name** where **layout_name** is replaced with the name of the layout.

Dialog Options

- **Layout** - Select the layout from which we will create the boundary.
- **Text Height that you used in Model Space** - The default is .25 DU or as given in 1/100th of a Drawing Unit: 25.
- **Height of the text that you want to print** - Here we want the printed output height in 1/100th of a printer unit.
- **Color**

5.2.3.1.11 Groups

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >

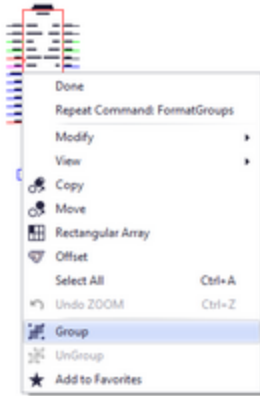


Drawing > Drawing > Groups
Commandline: formatgroups

Explanation

Create groups of entities in order to better organize your drawings. Groups can be named and described as well as disabled and edited later.

To create a Group select some entities and right-click then click Group



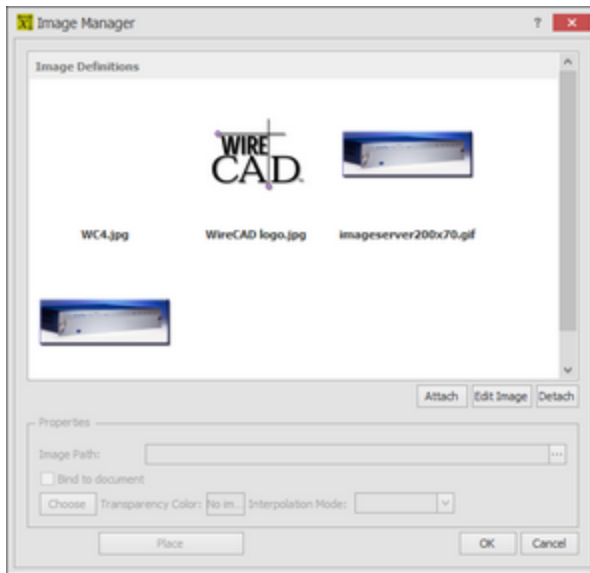
Dialog Options

- **Ignore Groups** - Disable grouping.
- **Name** - The default Group name will be a GUID. You can rename it if you like.
- **Description**
- **Selectable** - Is the group selectable as a group. If selectable = true then when one object in the group is selected all objects are selected.
- **Item Count**
- **Edit Entities** - Add or remove entities to the group.
- **Apply Properties** - Send the value of the Group Properties fields to the group.
- **Delete** - Deleting the Group does not delete the entities.
- **New** - New from the Group Properties.

5.2.3.1.12 Images

Navigation: [Reference](#) 288 > [Dialogs](#) 326 > [Drawing Dialogs](#) 389 > [CAD Dialogs](#) 389 >





Drawing > Format > Images
Commandline: formatimages

Explanation

Embed images in the drawing as well as to edit the properties of those images. The document now contains an **Images** collection from which images may be inherited throughout the drawing. Using the **Images** collection you may now either link or embed an image.

Dialog Options

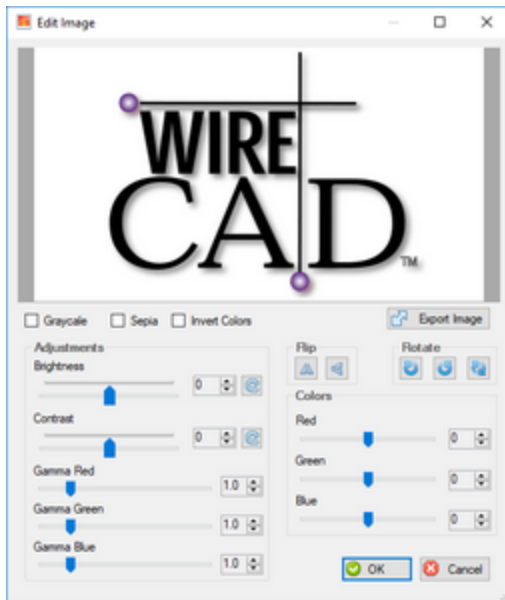
•Images Gallery

- Attach** - Browse to an image file to attach to the document.
- Edit Image** - Display the [Image Editor](#) ^[403] dialog for the selected image.
- Detach** - Remove the selected image if no instances exist in the drawing.
- Image Path** - Path to the linked file.
- Bind to Document** - Check this box to embed the image in the document.
- Choose Transparency Color**
- Place** - Place an instance of the image in the drawing space.

5.2.3.1.13 Image Editor

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >





Double-click an Image

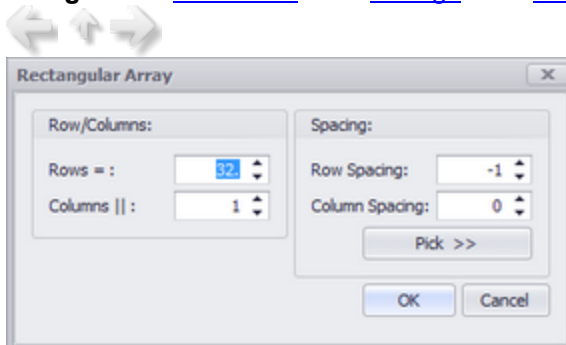
Commandline: none

Explanation

Image editor. You are editing the Image Definition of the document images collection not the instance. All instances will follow the definition.

5.2.3.1.14 Rectangular Array

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >



Drawing > Modify > Rectangular Array

Commandline: ar

Explanation

Creates multiple copies of objects in a pattern.

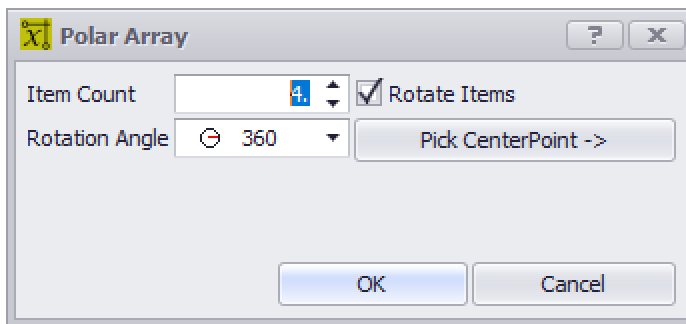
With the rectangular array you can create an array defined by a number of rows and columns of copies of the selected object.

First you have to select the objects. Then you have to define number of rows and number of columns of the rectangle, the distance between rows and the distance between columns.

Dialog Options

- **Rows** - The number of rows in the array including the source elements.
- **Columns** - The number of columns in the array including the source elements.
- **Row Spacing** - Row spacing in DU.
- **Column Spacing** - Column spacing in DU.
- **Pick >>** - Momentarily dismiss this dialog so you can pick the spacing out of the drawing.

5.2.3.1.15 Polar Array

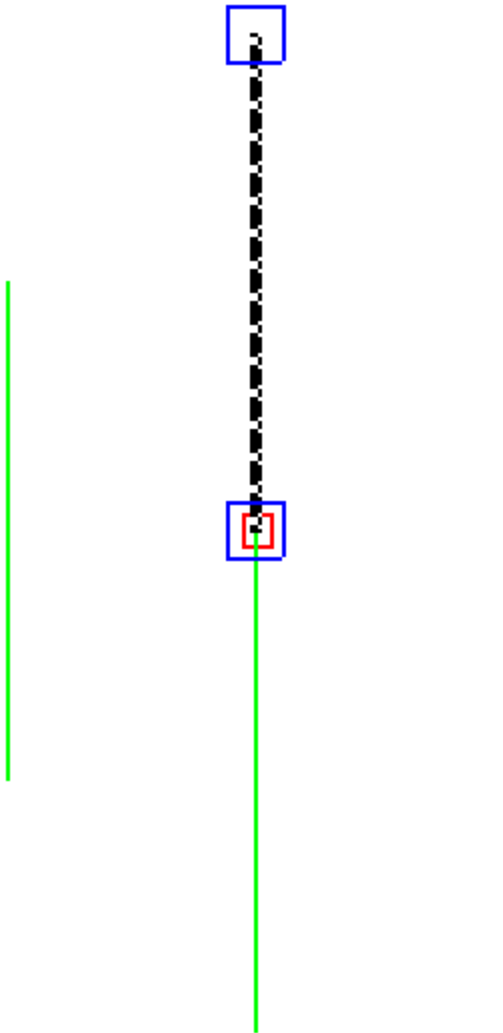


Drawing > Modify > Polar Array

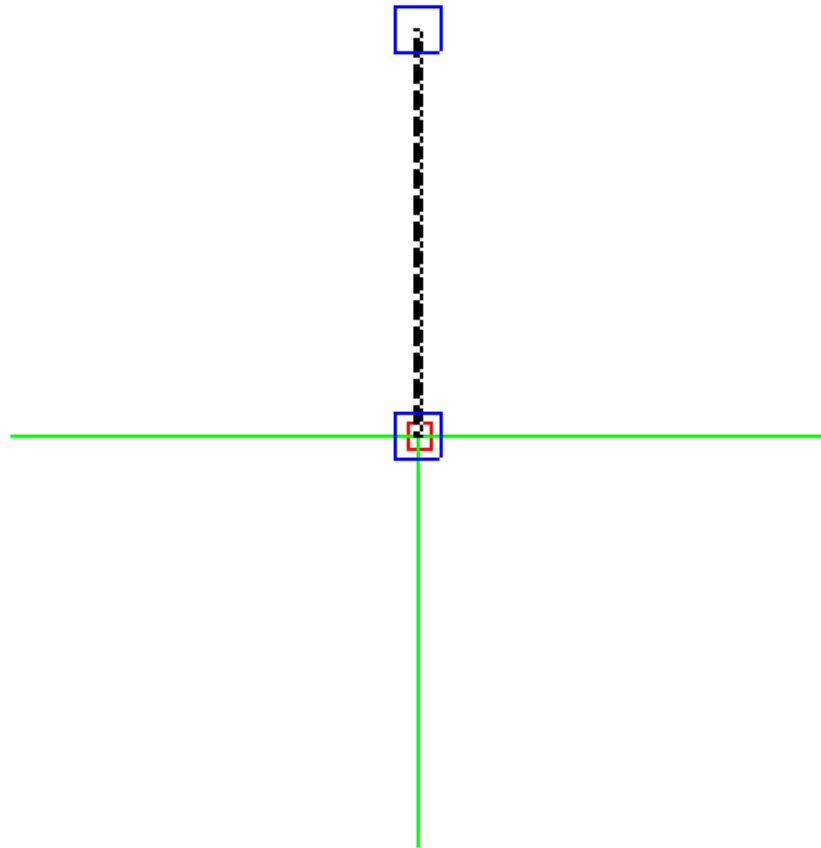
Commandline: parray

Explanation

Creates multiple copies of objects in a polar pattern based upon a center point.



Polar Array Without Rotation



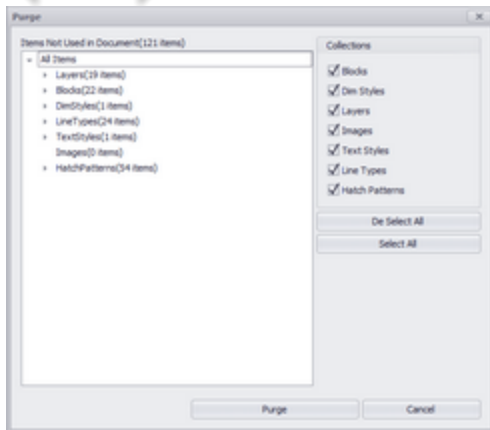
Polar Array With Rotation

Dialog Options

- **Item Count** - The number of elements in the array including the source elements.
- **Rotate Items** - Sets whether the items are rotated about the center point.
- **Rotation Angle** - Sets the rotation angle.
- **Pick Center Point** -> - Momentarily dismiss this dialog so you can pick the center point from of the drawing.

5.2.3.1.16 Purge

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >



Drawing > File > Purge
Commandline: purge

Explanation

Remove unused entities from the drawing. Only unused entities and object appear here. For an item to be purgable it must not appear in any drawing space or by used by another entity or object.

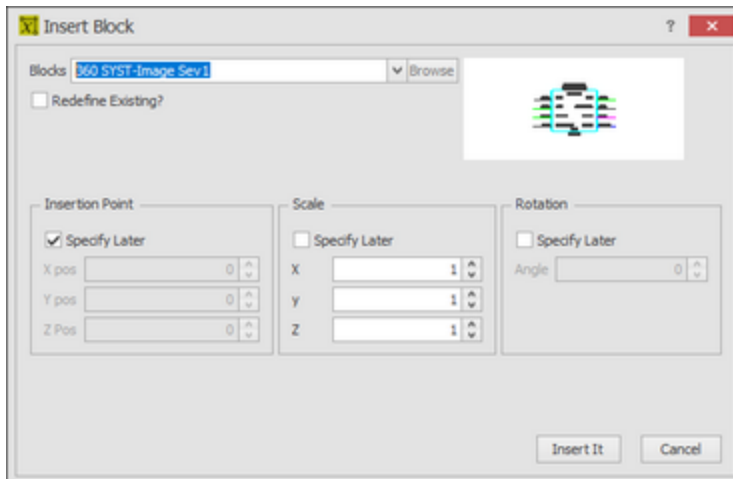
Dialog Options

- **Collection Tree** - The document collections.
- **Collection to Purge** - The collections to purge.

5.2.3.1.17 Inserts Dialog

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >





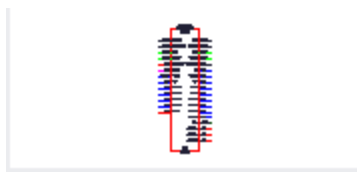
Drawing > CAD Tools > Insert Block into Drawing
Commandline: insert

Explanation

Insert blocks in to the drawing. This is the collection of Blocks in the drawing. You may also browse to any .dwg file and insert it into the file.

Dialog Options

- **Blocks** - Select an existing block or browse to a dwg file and add it to the drawing.
- **Preview** -



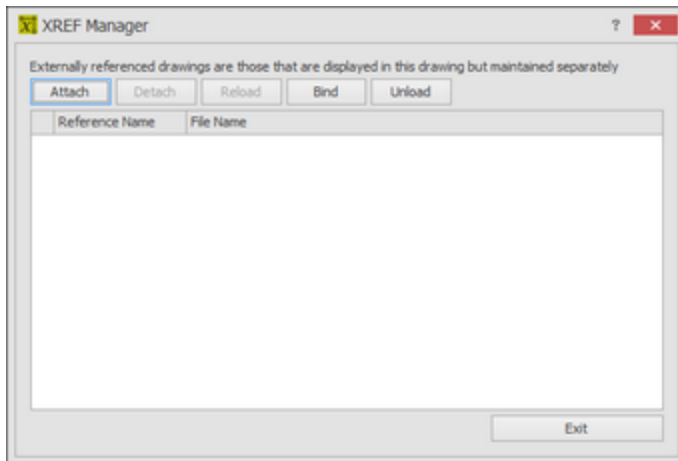
The selected block

- **Insertion Point, Scale, Rotation** - Pick here or on the drawing.

5.2.3.1.18 XREF Manager

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [CAD Dialogs](#) ³⁸⁹ >





Drawing > CAD Tools > XREF Manager

Commandline: xref

Explanation

Manage externally referenced drawings. These are drawings that are visible in the current Model space but maintained in separate files. They are linked here read only. This allows base drawings to be managed separately yet display in the current drawing. Because they are linked they are not editable.

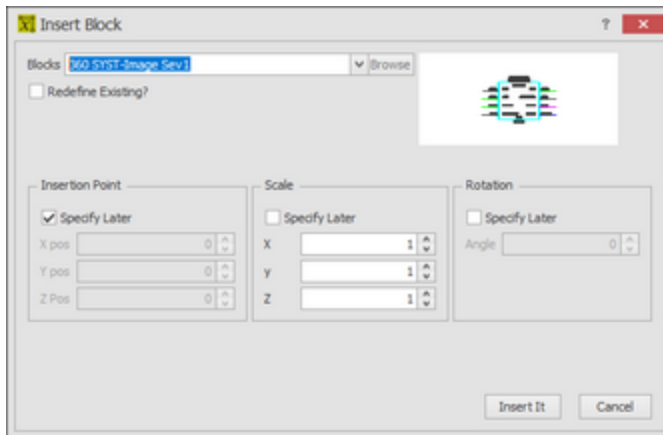
You can bind a previously Externally Referenced drawing to the current document if you desire that it be editable in the current view. You can control the visibility of XREF layers from the Layers dialog.

Related Topics:

[Layers Dialog](#) 394

Dialog Options

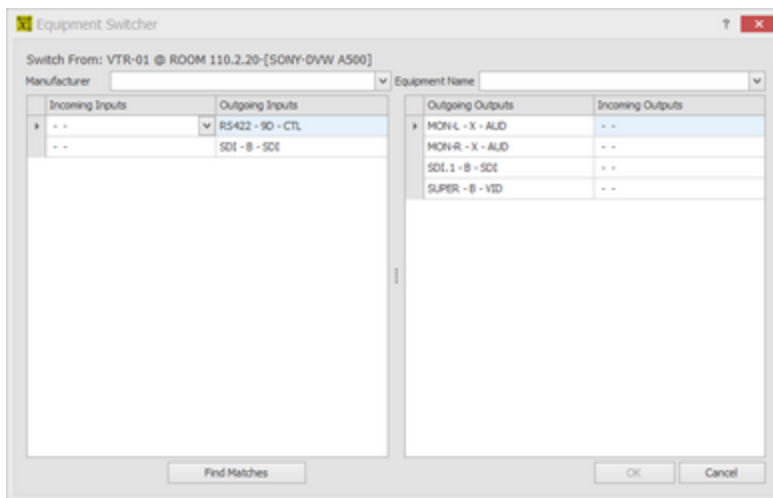
- **Attach** - Browse to the file. After which you will be presented with the standard Insert dialog to position the incoming file on the screen.



- Detach** - Remove the XREF.
- Reload** - Get any changes to the XREF file and display them.
- Bind** - Make it a Block in this drawing file. It is no longer linked and changes in the source file will no longer reflect in the current drawing.
- Unload** - Get rid of it.

5.2.3.1.19 Swap Equipment

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >



Context Menu with Equipment Insert Selected>Equipment Functions>Swap Equipment

Default command line shortcut: **none**

The Swap Equipment function allows you to swap Manufacturer, Model, and port data on an existing functional block or concept block.

Applies To:

Functional Blocks
Concept (High Detail) Blocks.

Used For:

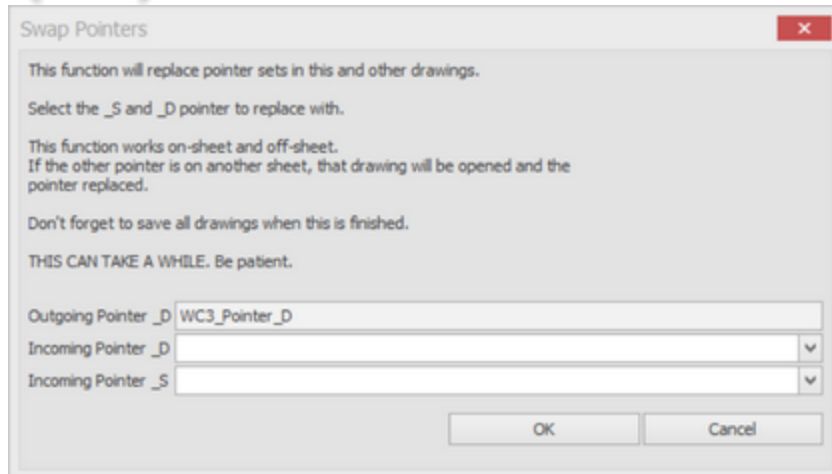
Switching from one make/model to another.

Dialog Options

- **Manufacturer** - Select the **Manufacturer** to replace the outgoing device with.
- **Equipment Name** - Select the **Equipment Name** / Model / PN to replace the outgoing device with.
- **Inputs** grid, **Outputs** grid - Select the ports to replace.
- **Find Matches** - Matches incoming ports to outgoing ports.
- **OK** - Do it!

5.2.3.1.20 Swap Pointers

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >



[Advanced Tools] > [Swap] > [Swap All Pointers]

Default Command Line Shortcut: SwapPointers

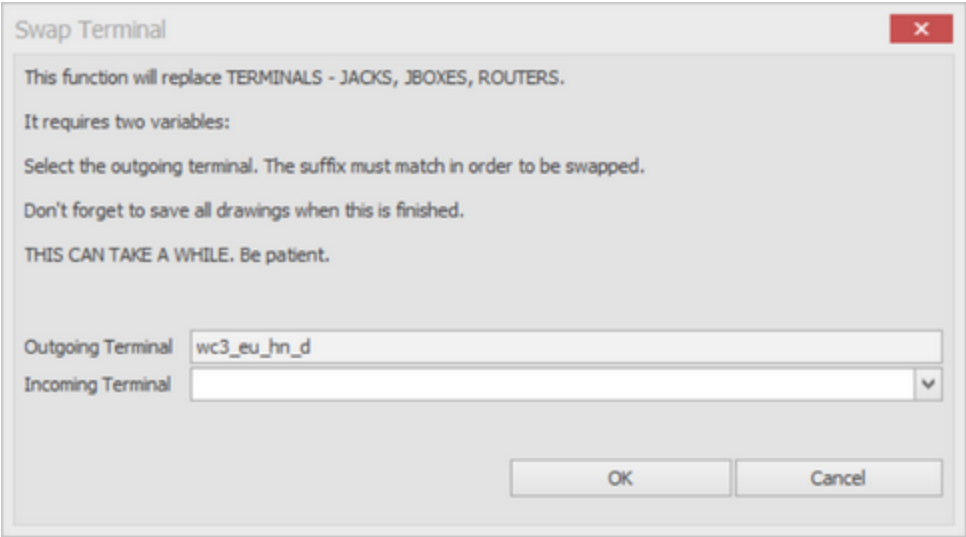
If a pointer is displeasing to the eye, it can be changed using this tool.

See also: [How To: Swap Pointers](#) ^[100]

5.2.3.1.21 Swap Terminals

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >





[Advanced Tools] > [Swap] > [Swap All Terminals]

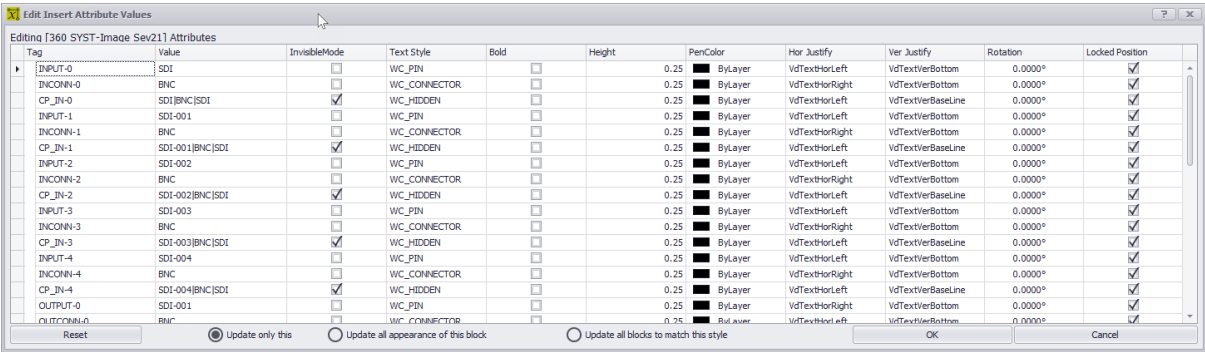
Default Command Line Shortcut: SwapTerminals

If a terminal is displeasing to the eye, it can be changed using this tool.

See also: [How To: Swap Terminals](#) ¹⁰⁷

5.2.3.1.22 Edit Attributes

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [CAD Dialogs](#) ³⁸⁹ >



CAD Tools > Edit Attributes

Default command line shortcut: EditAttributes

If an insert has Attributes they can be edited using this tool.

Be careful using this on WireCAD generated blocks as there is usually a tool that will update the attributes and the database concurrently.

Applies To:

CAD Blocks.

Used For:

Editing text in a Insert entity.

Dialog Options

Tag - Descriptive text.

Value - The data that renders as text in the insert.

InvisibleMode - Sets whether the attribute is invisible. True = invisible.

Text Style - Sets the TextStyle and therefore font of the attribute.

Bold - Sets the whether the Attribute text is bold.

Height - Sets the Text Height of the Attribute.

Pen Color - Sets the Pen Color of the Attribute.

Hor Justify - Sets the horizontal justification of the Attribute.

Ver Justify - Sets the Vertical justification of the Attribute.

Rotation - Sets the Rotation in degrees of the Attribute.

Locked Position - Sets whether the attribute can be moved when the document is in Drawing[Status]>**GripBlockAttributes** mode.

[Reset] - Resets the attributes back to their default values.

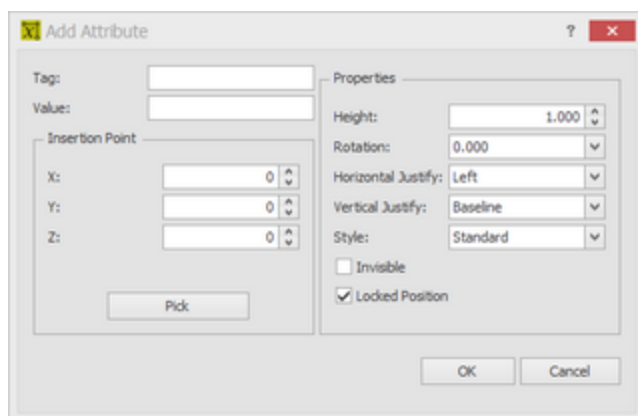
[Update Only This] - Updates this block instance only upon clicking [OK].

[Update all appearance of this block] - All inserts that bear the same Block Name in this drawing will be updated to match this.

[Update all blocks to match this style] - All inserts in the drawing that have matching attributes will be updated to match this.

5.2.3.1.23 Add Attribute Definition

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >

**CAD Tools > Create Attribute Definitions**

Default command line shortcut: **none**

Used to create an Attribute Definition. Attributes exist in two states.

1. Before a block is created an Attribute is referred to as an Attribute Definition or AttribDef.

- 2.After a block is created with an AttribDef it becomes an Attribute.
- 3.AttribDefs are editable and easily modified. Once added to a block they are harder to change.

Blocks do not limit the number of AttribDefs.

Applies To:
Standard CAD Blocks.

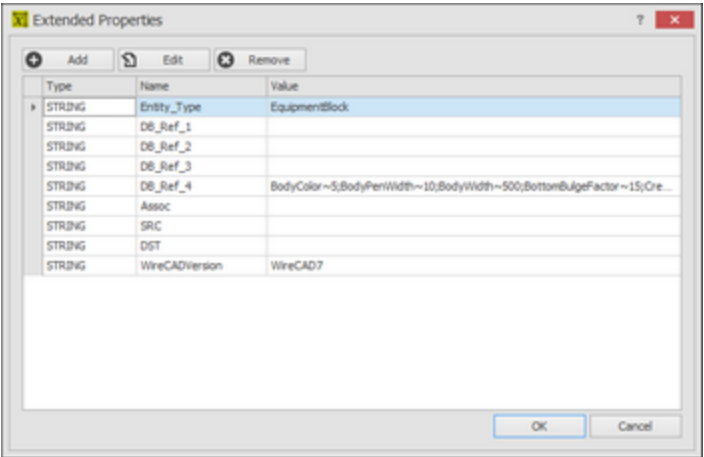
Used For:
Creating editable text in a Insert entity.

Dialog Options

- Tag** - Descriptive text. This is what renders when not included in a block.
- Value** - The data that renders as text in the insert.

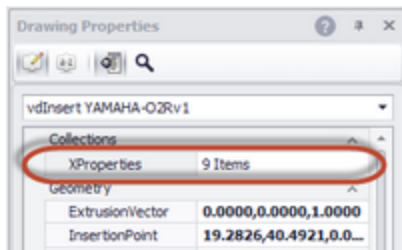
5.2.3.1.24 X Properties Grid

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [CAD Dialogs](#) [389] >



Not directly callable.
Default command line shortcut: **none**

This dialog allows you to add edit and delete extended properties on an entity. This is generally done in code.
This dialog can be shown by clicking the [...] button on the **XProperties** row of the drawing Properties Tool Panel.



We recommend that you not edit any of this data.

Applies To:

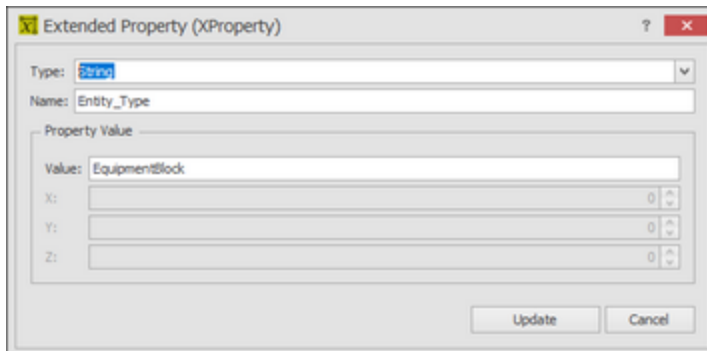
All entities.

Dialog Options

- **Type** - Variable type.
- **Name** - Descriptive text.
- **Value** - The data.
- **Add, Edit Delete** - obvious.

5.2.3.1.25 Edit X Properties

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >



Not directly callable.

Default command line shortcut: **none**

This dialog allows you to edit existing extended properties on an entity. This is generally done in code.

This dialog can be shown by clicking the **[Edit]** button on the [Extended Properties](#) ^[415] grid.

We recommend that you not edit any of this data.

Applies To:

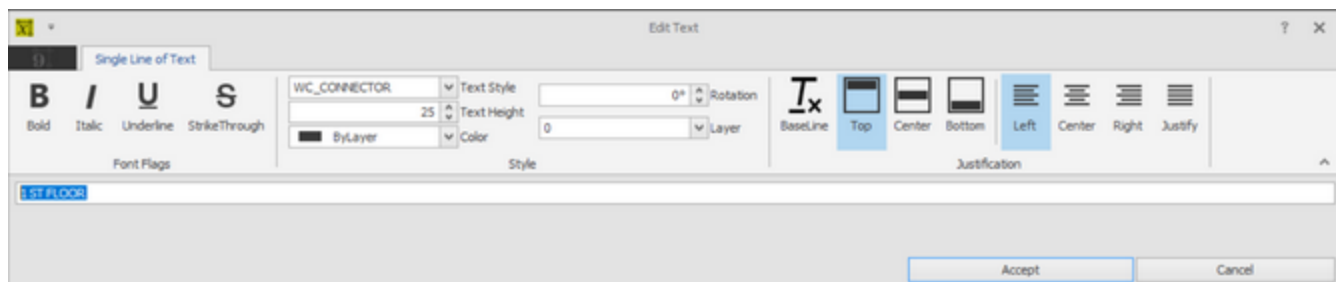
Select Extended Property of the selected item(s).

Dialog Options

- Type** - Variable type.
- Name** - Descriptive text.
- Value** - The data.
- X, Y, Z** - If the data one of the point types.

5.2.3.1.26 Singleline Text Editor

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >

**Double-click text entity**

Default command line shortcut: **none**

This dialog allows you to edit existing single line text entities.

Applies To:

Selected Single Line Text entity

Related Topics:

[Text Styles](#) ^[397]

[Layers](#) ^[394]

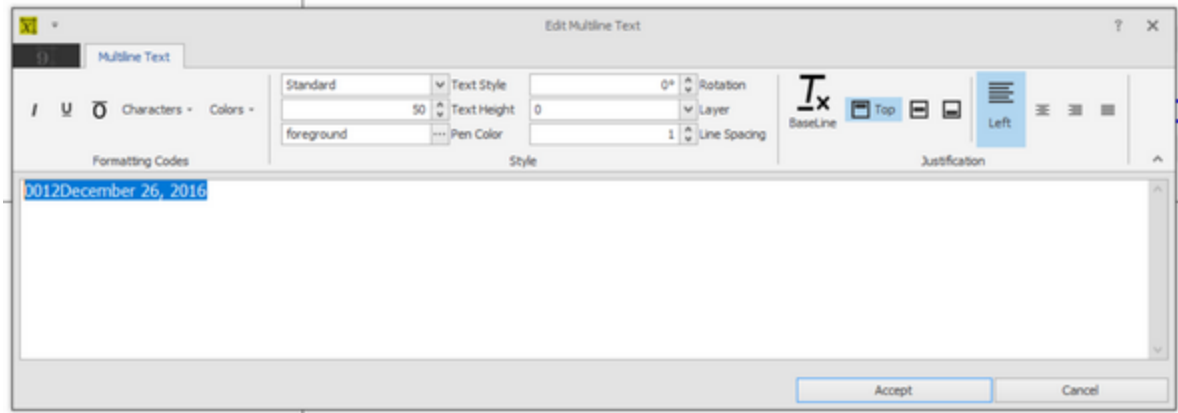
Dialog Options

- Bold, Italics, Underline, Strikethrough**
- Text Style** - Text entities do not have fonts but rather document based **TextStyles**. The **TextStyle** associates the font.
- Text Height** - In drawing units.
- Layer** - The Layer on which the entity resides.

- Color** - The pencolor of the text entity
- Rotation** - Angle in degrees.
- Justification** - Sets the horizontal and vertical justification.

5.2.3.1.27 Multi-line Text Editor

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [CAD Dialogs](#) ³⁸⁹ >



Double-click MText entity

Default command line shortcut: **none**

This dialog allows you to edit existing MText multi-line text entities.

MText entities display multiple lines of text with the ability to format selections within the MText entity.

This is a multi-color (MText) Entity
 This part of the text is 3x bigger than the selected TextHeight
 This part changes color mid line More text with a different color
 Then a fraction:
 $\frac{1}{4}$

MText Preview

MTEXT Control Codes

\0...\o	Turns overline on and off
\L...\l	Turns underline on and off
\\	Inserts a backslash
\{...\}	Inserts an opening and closing brace
\Cindex;	Changes to the specified color
\Hvalue;	Changes to the text height specified in drawing units
\Hvaluex;	Changes the text height to a multiple of the current text
height	
\Tvalue;	Adjusts the space between characters, from .75 to 4 times
\Qangle;	Changes obliquing angle
\Wvalue;	Changes width factor to produce wide text
\Ffile name;	Changes to the specified font file
\A	Sets the alignment value; valid values: 0, 1, 2
\P	Ends paragraph
\S...^...;	Stacks the subsequent text at the \, #, or ^ symbol

Applies To:

Selected MText text entity.

Related Topics:

[Text Styles](#) ³⁹⁷

[Layers](#) ³⁹⁴

Dialog Options

•**Italics, Underline, Overline**

•**Characters** - Insert control codes for the selected character.

•**Colors** - Insert control codes for the selected color. We have provided named colors but the control code \C can be followed by any of the index colors 0-127

•**Text Style** - Text entities do not have fonts but rather document based **TextStyles**. The **TextStyle** associates the font.

•**Text Height** - In drawing units.

•**Layer** - The Layer on which the entity resides.

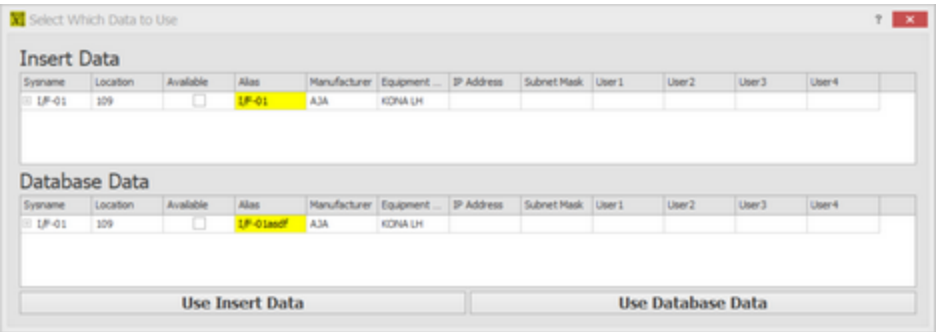
•**Color** - The pencolor of the text entity

•**Rotation** - Angle in degrees.

•**Justification** - Sets the horizontal and vertical justification.

5.2.3.1.28 Sync Insert Decision

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [CAD Dialogs](#) ³⁸⁹ >



Right-click a Functional Block then Equipment > Sync with Database

Default command line shortcut: **none**

If the block information does not match the database information you are presented with this choice. Yellow background cells represent fields that do not match. You must choose which data to use.

Applies To:

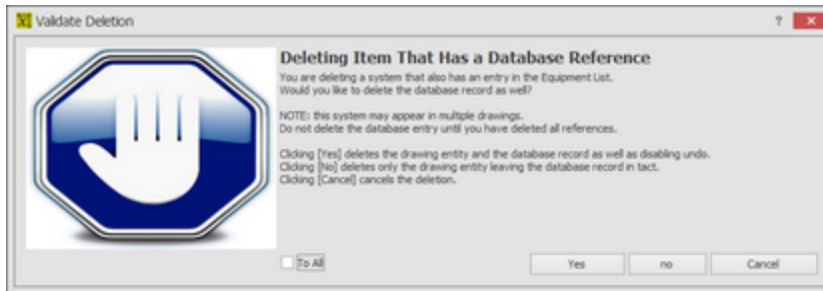
Selected Functional or Concept Block.

Related Topics:

None

5.2.3.1.29 Validate Deletion

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >



Delete or Erase

Default command line shortcut: **e**

When deleting an entity in the **Applies To** list you will be presented with this decision. This indicates that the entity you are deleting has an matching record in one of the databases. It is your choice as to whether to keep or delete the associated database record.

Example

Assume you have a device called SRVR-01. Further that SRVR-01 has appearances in the video drawing and the audio drawing.

When you decide to delete the instance that is in the video drawing, you will be shown this dialog and need to decide whether to delete the database entry as well.

The appropriate response in this case is not to delete the database record since we still have the instance in the audio drawing.

Applies To:

Blocks with SysNames
Terminals with SysNames
Cables with Cable Numbers.

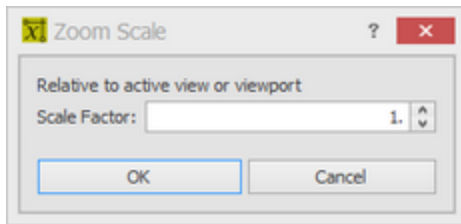
Related Topics:

[Delete Handling Setting](#) ^[366]

5.2.3.1.30 Zoom Scale

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [CAD Dialogs](#) ^[389] >





View > Zoom Scale

Default command line shortcut: **ZoomScale**

This function is used to numerically zoom the currently active viewport to the underlying Model Space.

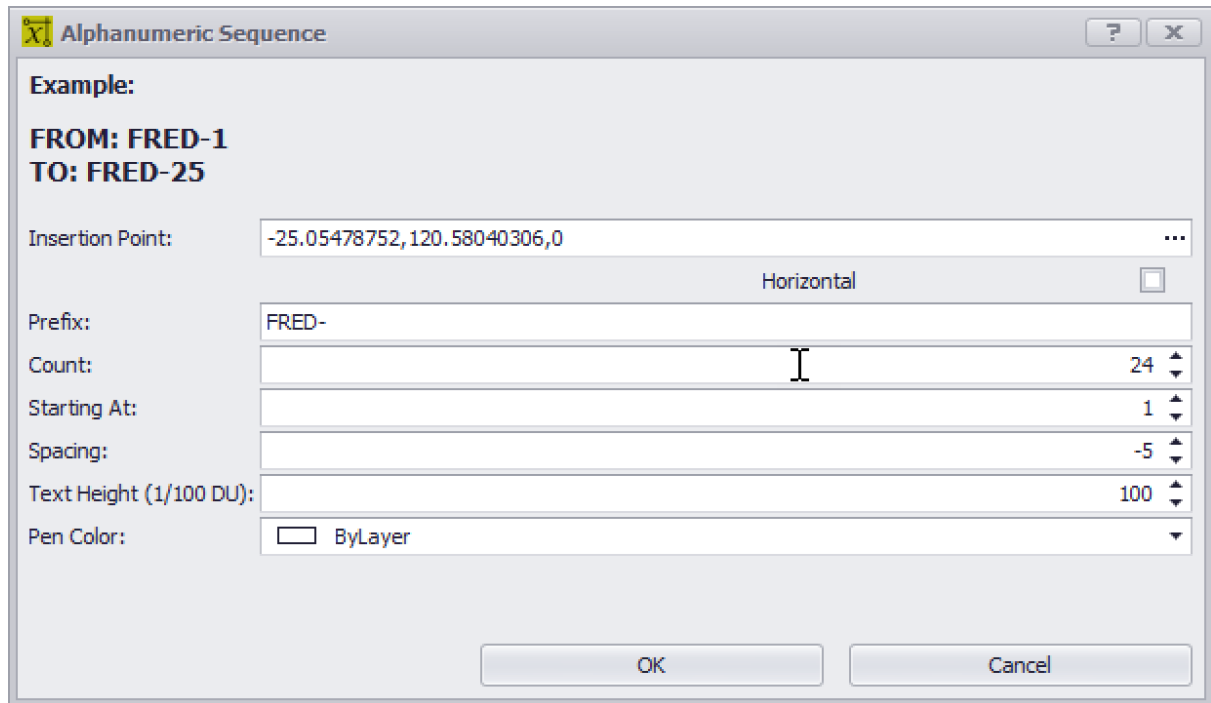
Usage

1. Open a drawing.
2. Switch to a layout with a viewport.
3. Double-click the viewport to activate it. You will see the crosshair is constrained to the viewport.
4. Click **View>Zoom Scale ...**
5. Enter a scale factor.
6. Click **[OK]**.

Applies To:

A layout with an activated Viewport.

5.2.3.1.31 Alphanumeric Sequence



The dialog box is titled "Alphanumeric Sequence" and contains the following fields and options:

- Example:**
FROM: FRED-1
TO: FRED-25
- Insertion Point:** -25.05478752, 120.58040306, 0
- Orientation:** Horizontal (selected)
- Prefix:** FRED-
- Count:** 24
- Starting At:** 1
- Spacing:** -5
- Text Height (1/100 DU):** 100
- Pen Color:** ByLayer

Buttons: OK, Cancel

CAD Tools > Entities > Text > Alphanumeric Sequence

Default command line shortcut: **AlphanumericSequence**

This function creates a sequence of text either vertically or horizontally in the drawing.

Example

FRED-1

FRED-2

FRED-3

FRED-4

Dialog Options

Insertion Point - Sets the insertion point of the first text item. Click the ellipsis button [...] to pick a point in the drawing.

Prefix - Sets the prefix of the text. The numbers will be appended to the value in the Prefix field.

Horizontal - Sets whether the sequence is created horizontally.

Count - Sets the total count of the sequence.


Starting At - Sets the starting number of sequence.

Spacing - Sets the spacing in Drawing Units (DU) between the text items in the sequence.

Text Height - Sets the Text Height of the text in 1/100ths of a DU.

Pen Color - Sets the Pen Color of the text in the sequence.

5.2.3.1.32 Text Increment

 Alphanumeric Sequence

?

x

Example:

FROM: FRED-1

TO: FRED-25

Insertion Point:

-25.05478752,120.58040306,0

...

Horizontal

☐

Prefix:

FRED-

Count:

I

24

▲▼

Starting At:

1

▲▼

Spacing:

-5

▲▼

Text Height (1/100 DU):

100

▲▼

Pen Color:

☐ ByLayer

▼

OK

Cancel

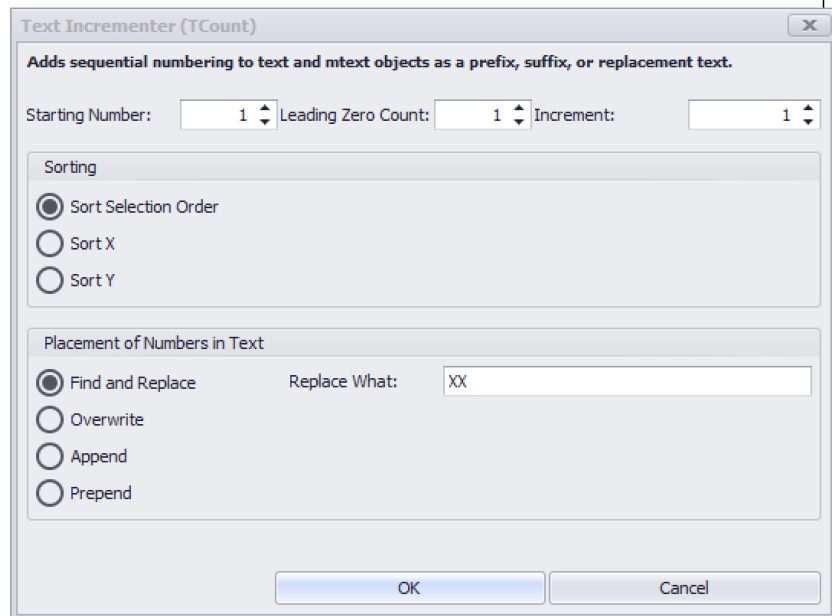
CAD Tools > Entities > Text > Text Increment

Default command line shortcut: **TCount**

This function numbers an existing array of text. It should not be used to number cables or equipment. Those functions have corresponding Assignment functions.

Example

FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX
FRED-XX



Text Increment - Before

FRED-01

FRED-02

FRED-03

FRED-04

FRED-05

FRED-06

FRED-07

FRED-08

FRED-09

FRED-10

FRED-11

FRED-12

Dialog Options

Starting Number- Sets the insertion point of the first text item. Click the ellipsis button [...] to pick a point in the drawing.

Leading Zero Count - Sets the prefix of the text. The numbers will be appended to the value in the Prefix field.

Increment- Sets whether the sequence is created horizontally.

Sorting - Sets the total count of the sequence.

Placement of Numbers in Text - Sets the starting number of sequence.

Find and Replace - Sets the spacing in Drawing Units (DU) between the text items in the sequence.

Overwrite - Sets the Text Height of the text in 1/100ths of a DU.

Append - Sets the Pen Color of the text in the sequence.

Prepend - Sets the Number sequence to be added to the text.

5.2.3.2 Advanced Tools Dialogs

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  >

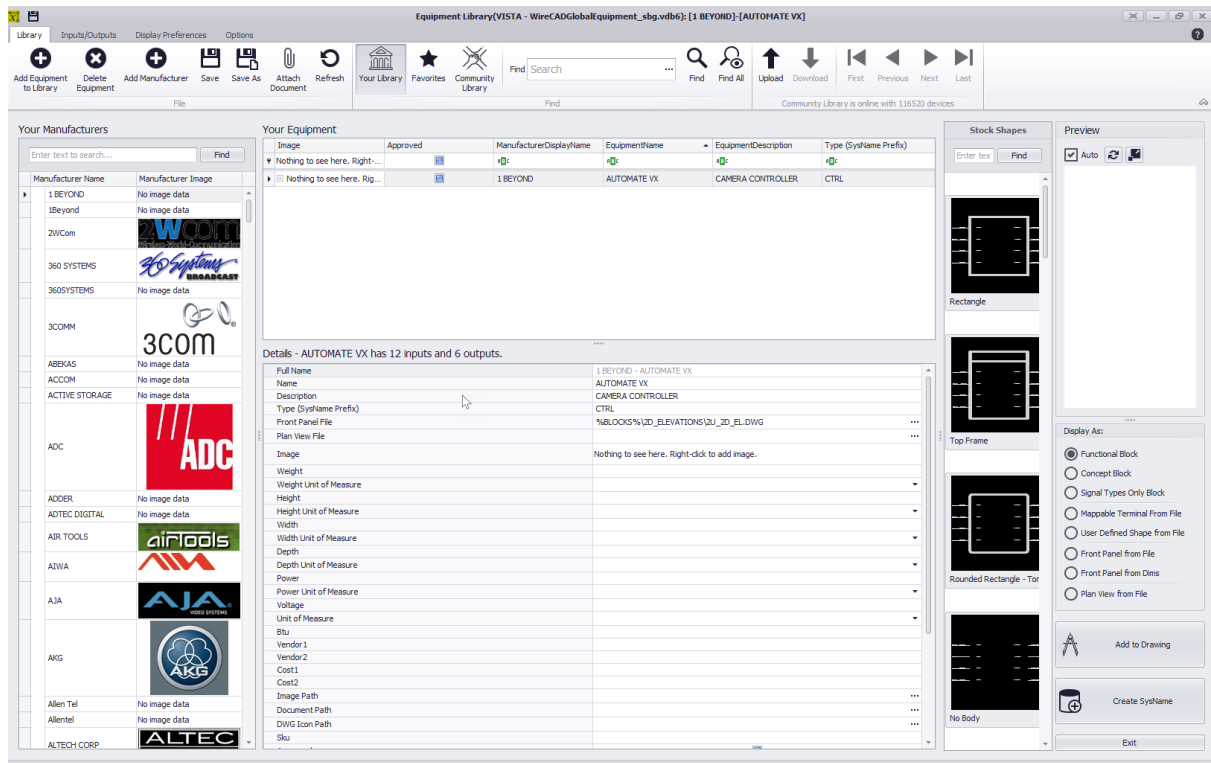


The following is a set of dialogs that may be presented while using the advance tools in the drawing environment.

5.2.3.2.1 Equipment Library

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [Advanced Tools Dialogs](#)  >





Database > Equipment Library
Drawing > Advanced Tools > Equipment Library
Commandline: le
Several Others

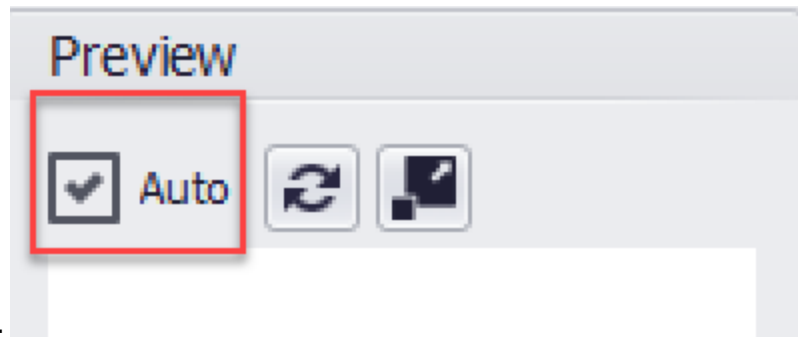
Explanation

The WireCAD **Equipment Library** is where you will spend a fair amount of time as you define equipment that you will use in your designs. The Equipment Library is a presentation of the global Manufacturers table and its hierarchy. The Equipment Library contains no CAD blocks just equipment definitions. These equipment definitions describe a piece of equipment its make, model, and IO.

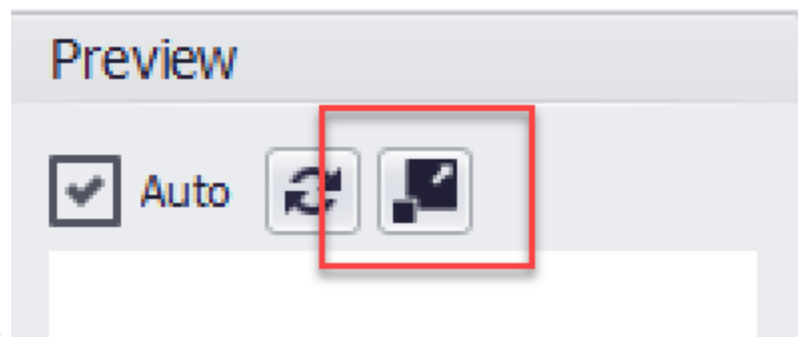
This is also where we come to create CAD blocks in our drawings. There are many settings here that let you customize appearance.

The interface is tabbed. The **[Find]** tab sets the current item. The current item will be enumerated in the **[Detail]** and **[I/O]** tabs. This topic covers the right-hand column of controls that is visible from all tabs.

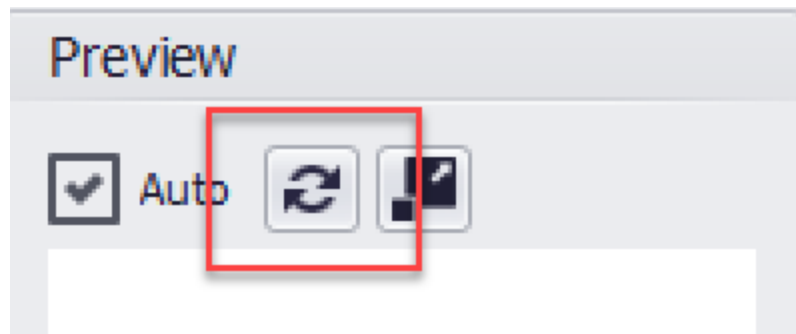
Dialog Options



- **Auto Preview** - Automatically generate the preview. This can be costly if you have an equipment definition with lots of Inputs and Outputs.



- **Big Preview** - Toggle between large preview mode.



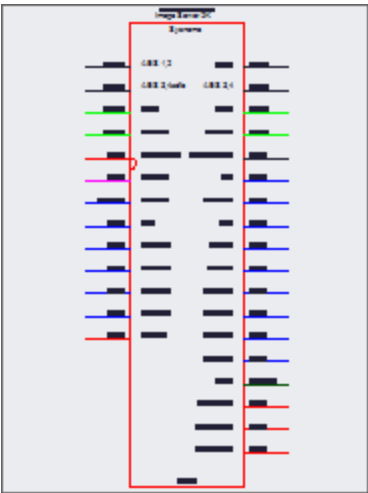
- **Refresh Preview** - Call for a refresh of the preview window. This will rebuild the image based on all of the data and settings.

- **Preview** - Displays the currently selected equipment definition based on the **Display As:** setting.

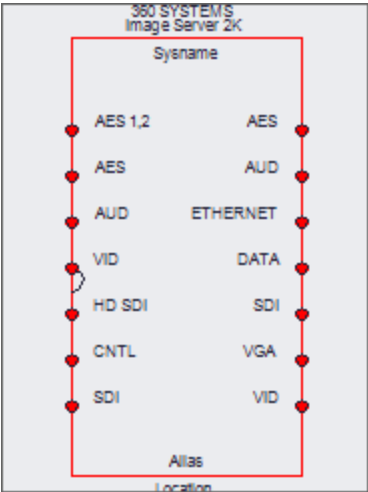
- **Display As:**

Setting	Preview
---------	---------

Functional I/O



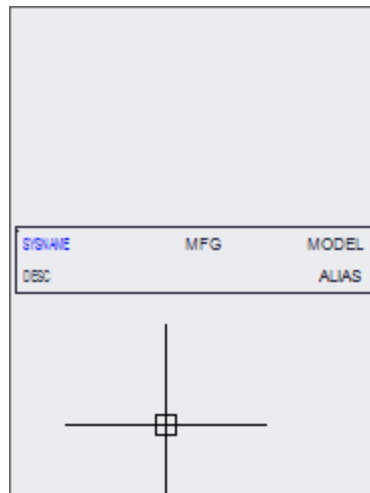
Conceptual I/O



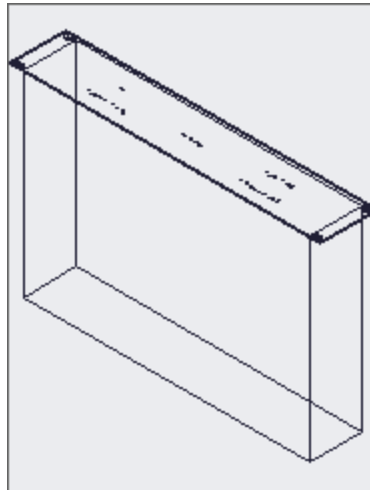
Conceptual I/O (high detail)



**Front
Panel
(File
Merge)**



**Front
Panel
(From
Dims)**



**Plan
View
(File
Merge)**



**Plan
View
(From
Dims)**

Not yet implemented.

- Add Manufacturer** - Shows the Add Manufacturer dialog.

- **Add Equipment** - Shows the Add Equipment Wizard.
- **Delete Equipment** - Deletes the selected record.
- **Add to Drawing** - Add the selected item to the drawing based on the current preview.
- **Add to Project Database Only (Assign SysName)** - Shows the SysName Assignment dialog.
- **Attach Document** - Attach any document to the current record. This is useful for storing data sheets or specification documents with the equipment definition.

5.2.3.2.1.1 Find

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[328] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] > [Equipment Library](#) ^[428] >



Database > Equipment Library

Drawing > Advanced Tools > Equipment Library [Find]

Commandline: le

Several Others

Explanation

The **[Find]** bar allows you to search the local global database as well as the **Community Server**. When searching locally all records are returned by default. When searching the **Community Server** you will need to enter a search term. The results will be returned in pages based on your **Results Page Size** value.

Search Term Hints:

When searching the databases for items, less is more. You want to enter a value that can be found in a single field.

For example: say we are looking for a Sony DVW A-500. The following is a list of terms and their results:

Search: **sony** - Result: All Sony products and any products from other manufacturers that have Sony in their description fields.

Search: **DVW** - Result: All products with DVW somewhere in the name or description.

Search: **SONY DVW** - Result: None. There are no products with the manufacturer name and the product name in a single field.

Find Bar Options

Find - Enter the search term here.

Find button - Do the search.

Find All button - Clear and find all records. (Local only).

Your Library - Display the data from the local global database as defined in the [Application Settings\[Global Database Location\]](#) settings panel.

Favorites - Show the Manufacturers that you have added to the **Favorites** list by right-clicking and clicking **Add to Favorites**. Conversely you can remove favorites from the list by activating the Favorites mode and right-click a Manufacturer and select either Remove or Clear. Clear removes all.

Community Server - Switch to the Community server.

Test Connection button - Test your connection to the Community. If this fails you may need to open your firewall to community.wirecad.com port: 1433.

Download From Community - Before any edits can be made to the device you will need to download it to your local database.

Upload to Community - You can upload manually by clicking this button.

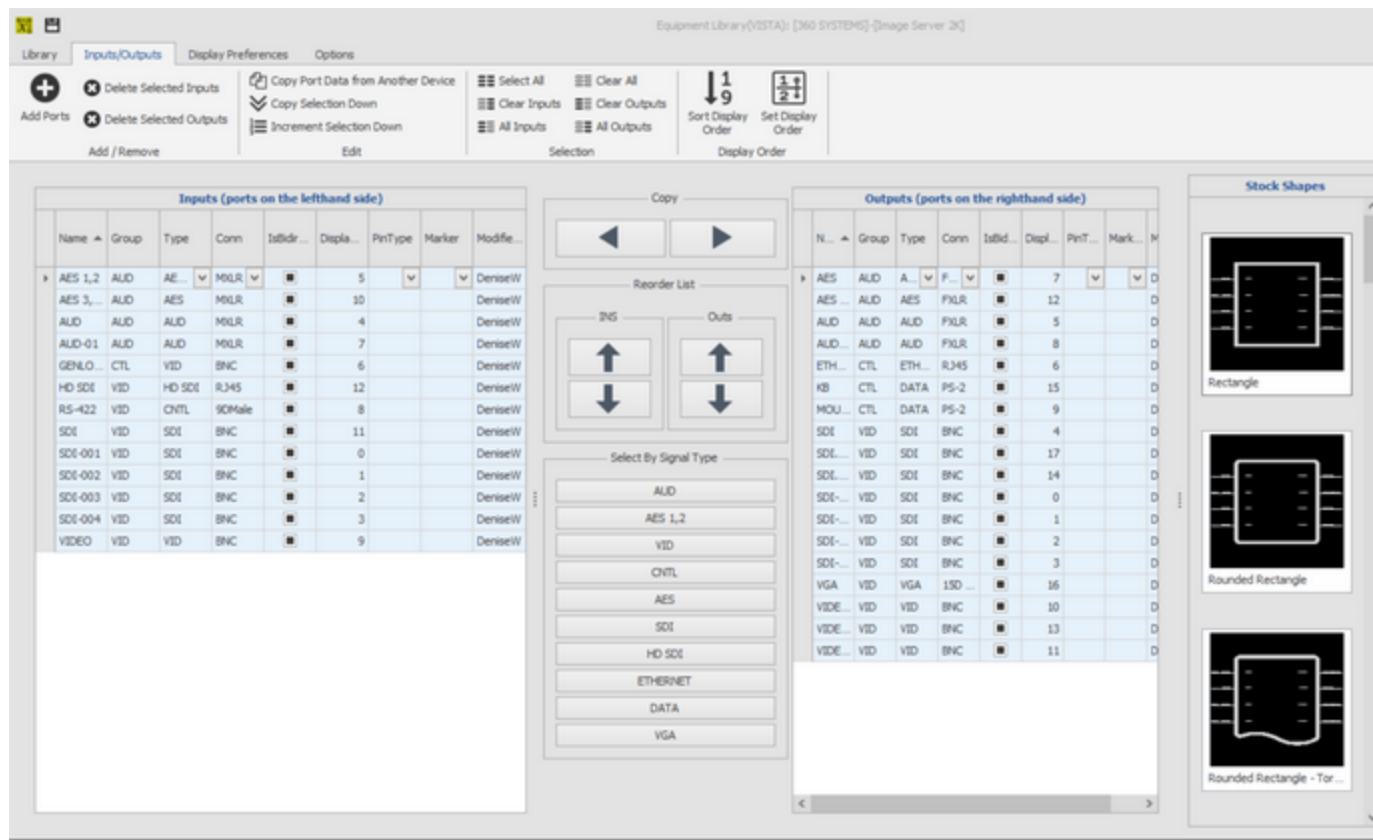
Grid - found items.

Results Paging - You can view the page results here:

Community Equipment - Records 1 to 10 of 244

5.2.3.2.1.2 Inputs/Outputs

Navigation: [Reference](#) > [Dialogs](#) > [Drawing Dialogs](#) > [Advanced Tools Dialogs](#) > [Equipment Library](#)



Database > Equipment Library [I/O]

Drawing > Advanced Tools > Equipment Library [I/O]

**Commandline: le
Several Others**

Explanation

This is the Inputs and Outputs page. It represents the I/O of the currently selected device. Inputs are displayed in the left-hand grid and outputs are displayed in the right-hand grid. This is by convention only. You are free to place inputs on the right-hand side and vice-versa. Each port record consists of a Name, Type, Connector and a Pin Type. Selected records will be included in the preview of any Function I/O and Concept blocks that you create.

If you are using connector genders be sure to enter the cable-end gender not the chassis side.

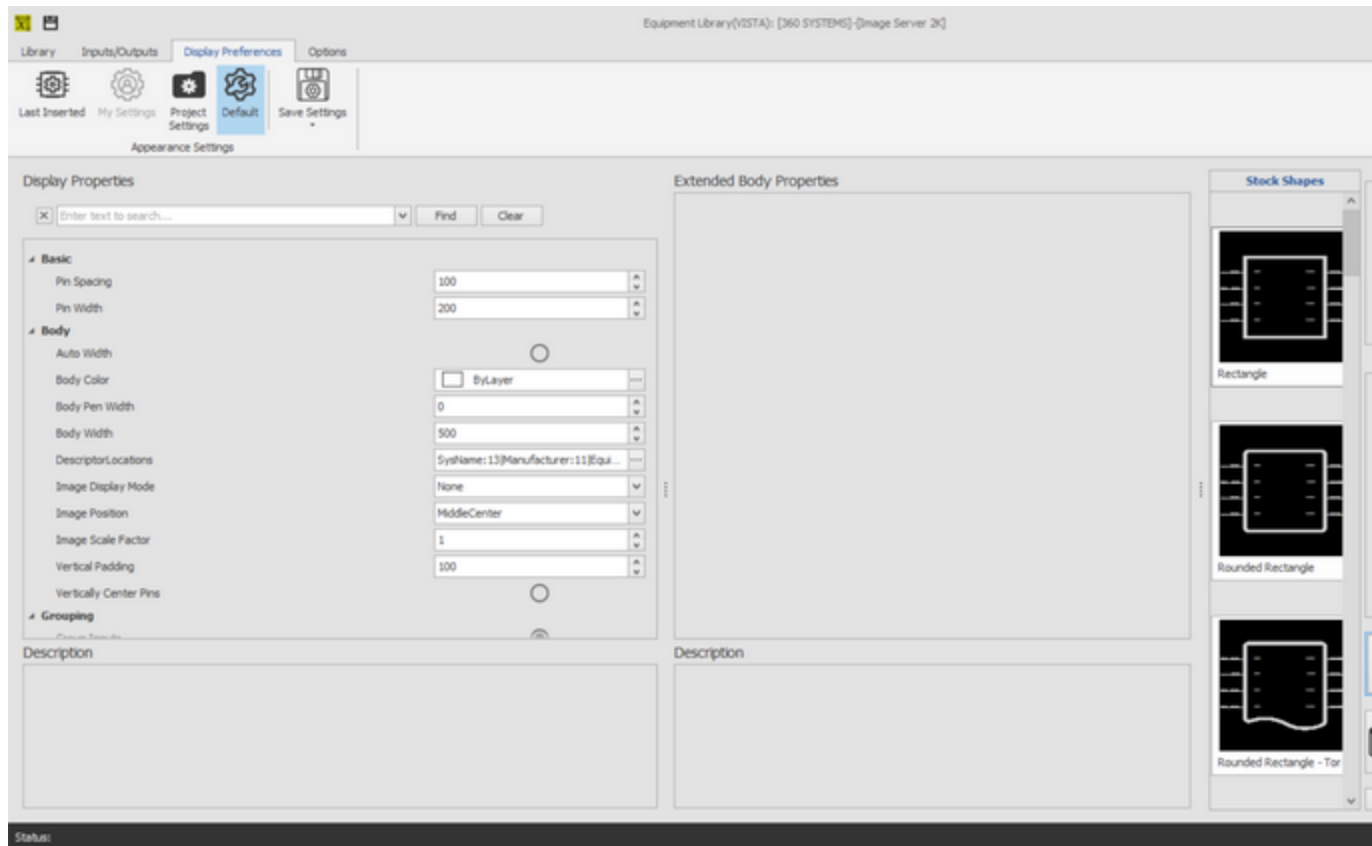
I/O Tab Options

- **Inputs Grid, Outputs Grid** - NOTE: Be sure to save your changes if you edit these grids manually by clicking **File>Save**.
- **Select All, Clear Selection, All Inputs, All Outputs** - Manipulate the selection.
- **Select by Signal Type** - Ten buttons in this frame will populate with the first ten signal types from the I/O records. Clicking the button will select all records in both grids of that signal type.
- **Reorder** - Move selection up/down.
- **Copy** - Copy side-to-side.
- **Add Ports** - [Show the Add Ports dialog](#) ^[442].
- **Delete Ports** - Delete the selected ports. You will be prompted for each grid. No save is necessary.
- **Refresh** - Refresh the preview.
- **Display Order** - Set and sort by the **Display Order** column. This allows you to rearrange the lists and be able to return later to your work.

5.2.3.2.1.3 Display Preferences

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] > [Equipment Library](#) ^[428] >





Database > Equipment Library

Drawing > Advanced Tools > Equipment Library

Commandline: le

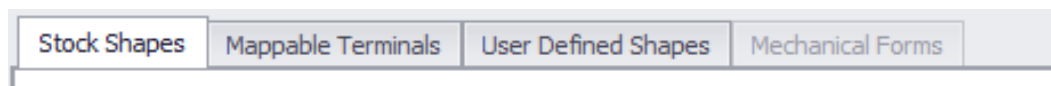
Several Others

Explanation

This is where we determine the look of the block we are about to add to the drawing.

There are four tabs in this view that allow you to customize the appearance of the block you are about to create.

1. [Stock Shapes](#) ⁴³⁵
2. [Mappable Terminals](#) ⁴³⁵
3. [User Defined Shapes](#) ⁴³⁵
4. [Mechanical Forms](#) ⁴³⁵



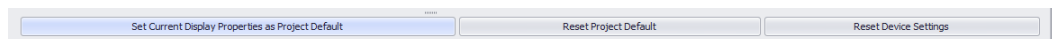
Understanding the Settings Mechanism

Settings store the appearance and display mode of the Equipment Library. These settings determine the look of the created block. You can tell which settings were used to create the block by looking at the **Equipment Library** status bar:

CDA5 has 1 inputs and 4 outputs. Preview created from Project settings.

WireCAD stores the settings used to create blocks in three different locations:

1. **Device** settings. If you create a block in a drawing we store the settings used to make the block with the device definition in the **Equipment Library**. These settings have priority.
2. **Project** settings. If you like the display of a block you can click the:



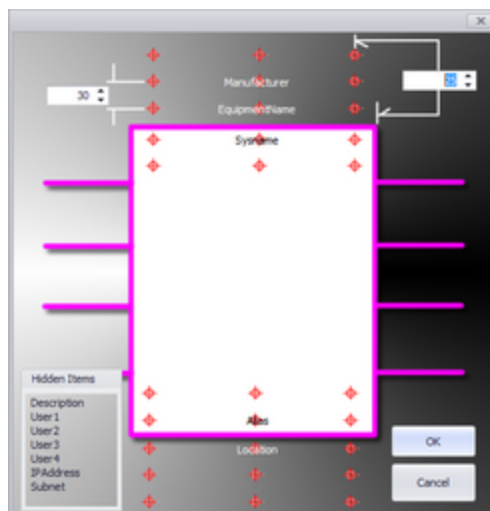
Doing so will set the current display settings as the **Project** default. All devices that do not currently have stored settings will then use the **Project** defaults.

3. **Default** settings. When no other settings are found we use the defaults.

Several controls are common to all tabs. We will go into those first.

Common Controls Options

- **Body Color** - Sets the color of Functional I/O and Concept block bodies. Has no effect on User Defined Shapes or Terminals. The geometry of the file determines the look.
- **Creation Mode** - Set by the UI. Leave it alone.
- **Descriptor Locations** - Shows the Descriptor Locations Map. Applies to Functional, Concept, and User Defined Shapes.



Drag to reposition

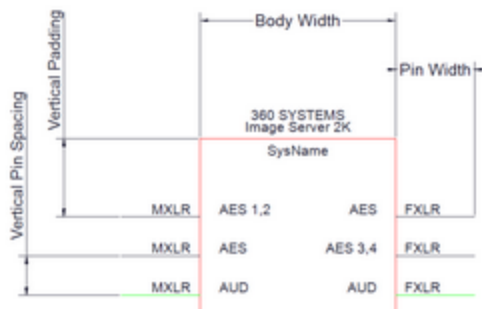
- **Image Display Mode - Applies to** Applies to Functional and Concept blocks.
None - no image or cad block is inserted in the created block.
Image - insert the image that is pointed to in the Image File Path of the Details tab for this device.
DWGIcon - insert the DWG Icon in the DWG Icon File Path of the Details tab for this device.
- **Image Position, Image Scale Factor**

**Image inserted and scaled at MiddleCenter**

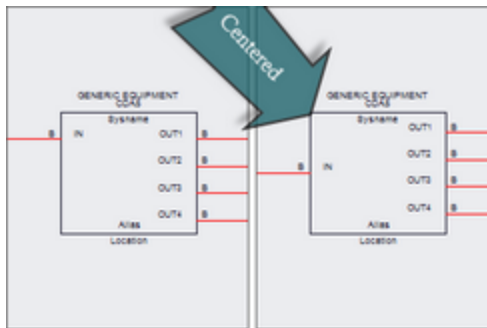
- **Merge and Center** - Centers identical text in the block. Applies to Functional and Concept blocks.



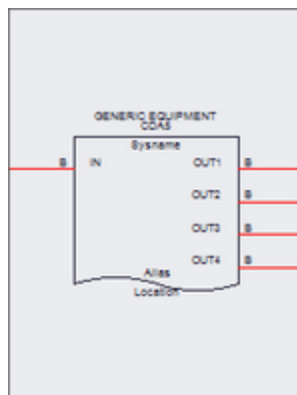
- **Text Height** - The height of all generated text.
- **Spacing and other dimensional info:**



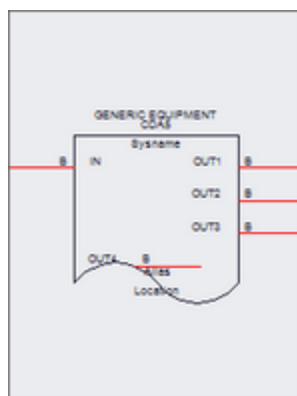
- **Vertically Center Pins**



- **Body Pen Width** - The thickness of the line that represents the block body.
- **Nudge Positions** - Use to nudge the various text around.
- **Tears and Bulge Factors** - Tears are visual elements that represent to the reader of the document that the device does not display all of its available I/O. Tears use bulges. The greater the bulge factor the more dramatic the tear.



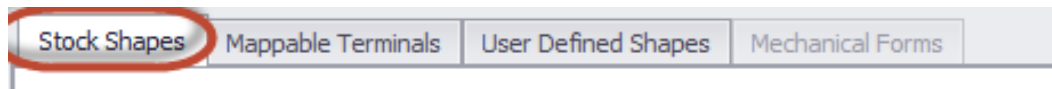
Bottom edge torn bulge factor .15 or 15/100



Bottom edge torn bulge factor .5 or 50/100

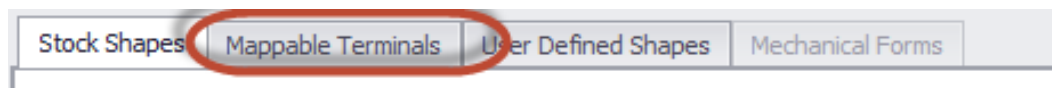
- **Set Current Display Properties as Project Default**
- **Reset Project Default**
- **Reset Device Settings**

Stock Shapes Options



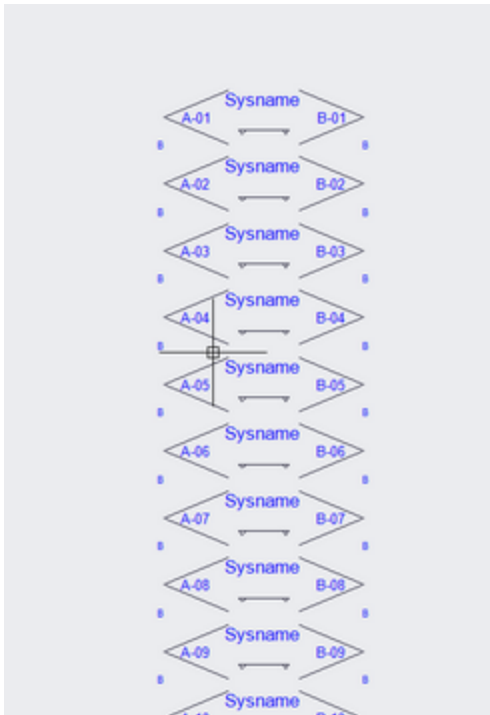
Here you choose from 18 stock shapes. We will not enumerate them here. Try clicking on them to see what they do.

Mappable Terminals Options



Here you can map the selected I/O on to terminals. Terminals are one or two port inline devices. If you select more than one record in the I/O tables we will continue to add terminals to accommodate the selection. If you select a terminal that has only an input port then any selected output records will be ignored.

Example showing multiple I/O selected and mapped on to a terminal that represents a full normal jack:



User Defined Shapes Options

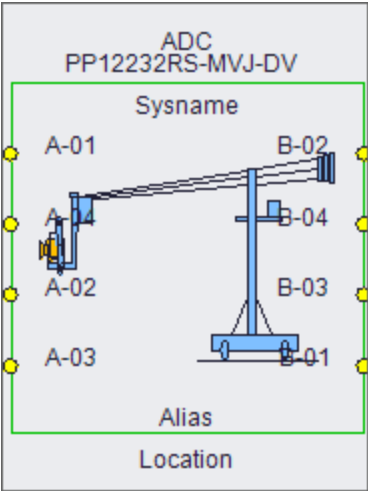


Any CAD file can be used as a **User Defined Shape**. It just needs to be placed in the **Shape Files** folder:

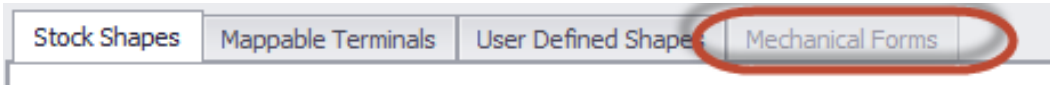
%BLOCKS%\Shapes Files

The **Display Properties** will then scale and stretch the object while applying port data.

Example showing **User Defined Shape** with four ports applied:



Mechanical Forms Options



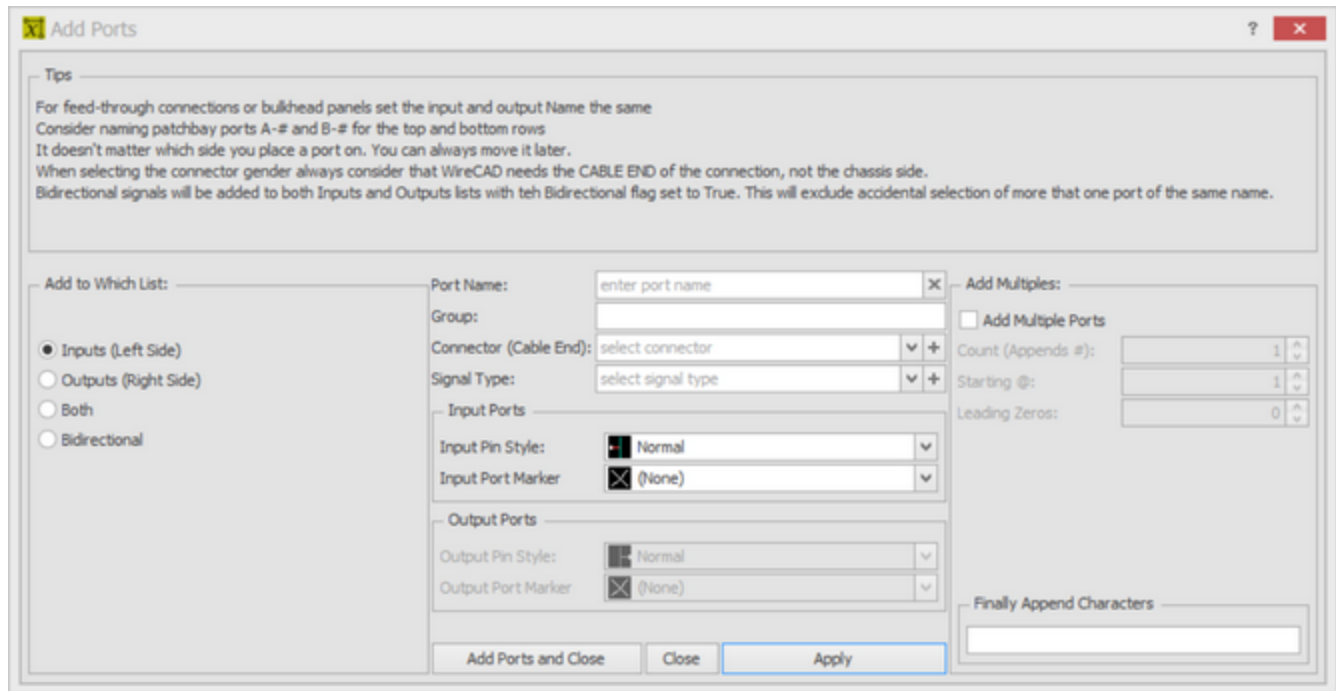
Mechanical Forms are hard-coded objects that get their dimensional data from the device definition and use it to render objects.

Item	Description
Rack Enclosure	
Rack Tray	
Cube	Select a form. Only available when Front Panel (from Dims) is selected.
Half Rack Left	
Half Rack Right	

5.2.3.2.1.4 Add Ports

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ > [Equipment Library](#) ⁴²⁸ >





Database > Equipment Library [I/O] [Add Ports]

Drawing > Advanced Tools > Equipment Library [I/O] [Add Ports]

Commandline: le

Several Others

Related Topics:

[How To: Add Some Inputs and Outputs](#) 

How To: Add Ports

Explanation

This is the Port Adder dialog. With it you add a single record, two records, or multiple records to the I/O grids. Here are some things to consider when naming your ports:

1. For feed-through connections or bulkhead panels set the input and output Name the same.
2. Consider naming patchbay ports A-# and B-# for the top and bottom rows.
3. It doesn't matter which side you place a port on. You can always move it later.
4. For bi-directional signals such as Ethernet or RS-422 consider your document flow when determining the list to which you will add these ports.
5. When selecting the connector gender always consider that WireCAD needs the CABLE END of the connection, not the chassis side.

Dialog Options

•Add to Which List

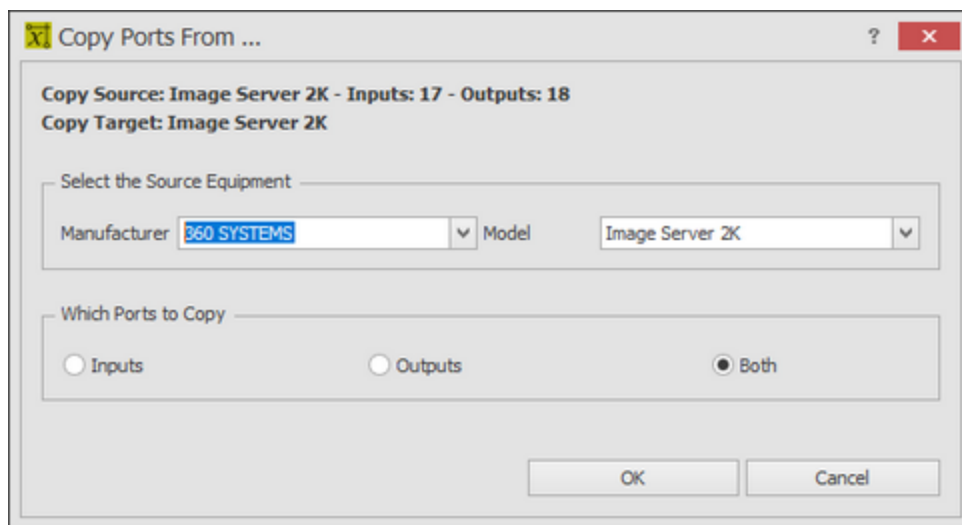
•**Port Name** - Enter the port name here. This may be a partial name if the Add I/O dialog is set to append either multiple records or the Finally Append field contains the remainder of the string.

•**Connector (Cable End)** - From the global Connectors table.

- **Signal Type** - From the Project Signal Types table.
- **Input Pin Style** - Straight pin (Normal) or looped (two connection points).
- **Output Pin Style** - Straight pin (Normal) or bridged (two connection points).
- **Add Multiples** - If checked a record will be created for each **Count**. The number will be appended to the **Port Name** info and formatted based on the **Leading Zeros** format.
- **Finally Append Characters** - Useful if you want to add characters after the multiple count number has been appended. See the How to Add Ports topic for more info.

5.2.3.2.1.5 Copy Port Data

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] > [Equipment Library](#) ^[428] >



Database > Equipment Library [Edit > Copy Port Data]

Commandline: `le`

Several Others

Related Topics:

None

Explanation

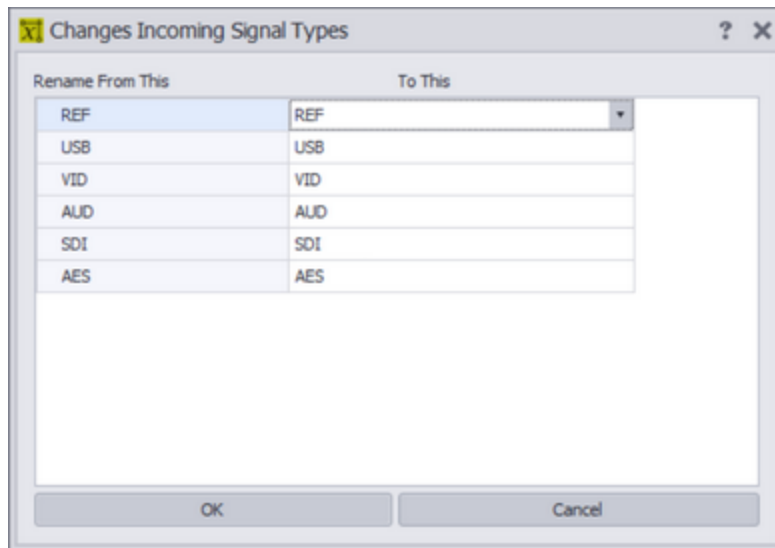
Allows you to copy port data from another device definition into the currently selected device definition.

Dialog Options

- **Select Manufacturer and Model** - to copy from.
- **Which Ports to Copy** - obvious.

5.2.3.2.1.6 Incoming Signal Types Map

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] > [Equipment Library](#) ^[428] >



Database > Equipment Library [Download From Community]

Commandline: le

Several Others

Related Topics:

None

Explanation

When downloading from the community server you will use this dialog to map from the Signal Types used in the incoming device definition to your Signal Types.

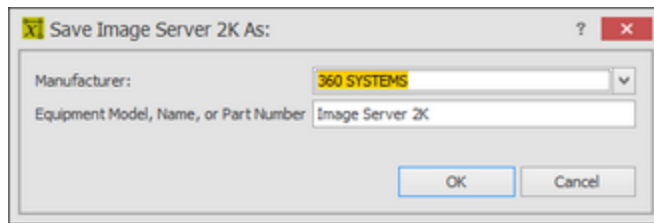
Dialog Options

- **Rename From This** - Incoming Signal Types.
- **To This** - Map to your Signal Types.

5.2.3.2.1.7 Save Equipment As

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] > [Equipment Library](#) ^[428] >





Database > Equipment Library [File > Save As]

Commandline: le

Several Others

Related Topics:

None

Explanation

Allows you to copy the current device definition as a new device definition with a new **Manufacturer** and **Model**.

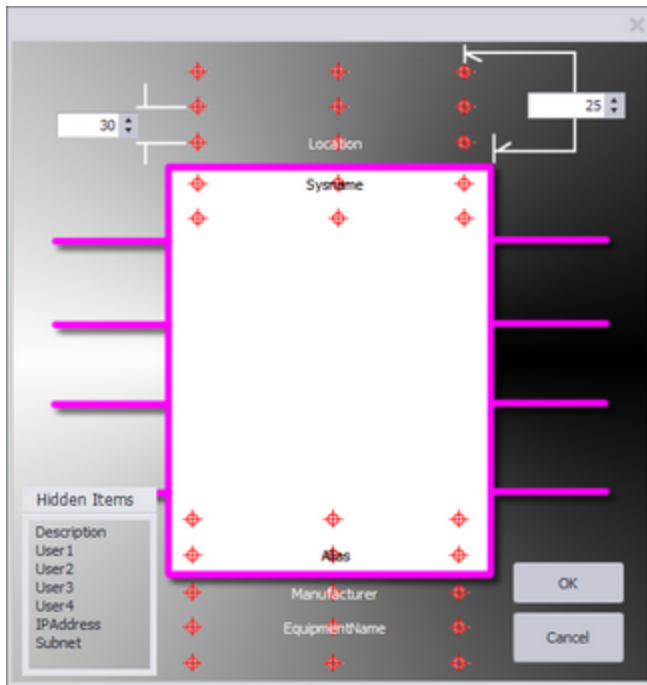
Dialog Options

- Manufacturer** - Select **Manufacturer**.
- Equipment Model** - The new name.

5.2.3.2.1.8 Descriptor Map

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁸ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ > [Equipment Library](#) ⁴²⁸ >





Database > Equipment Library [Display Preferences] Descriptor Locations

Commandline: **le**

Several Others

Related Topics:

[How To: Add Equipment to Drawings](#) 

Explanation

Change the locations and visibility of the various block descriptors:

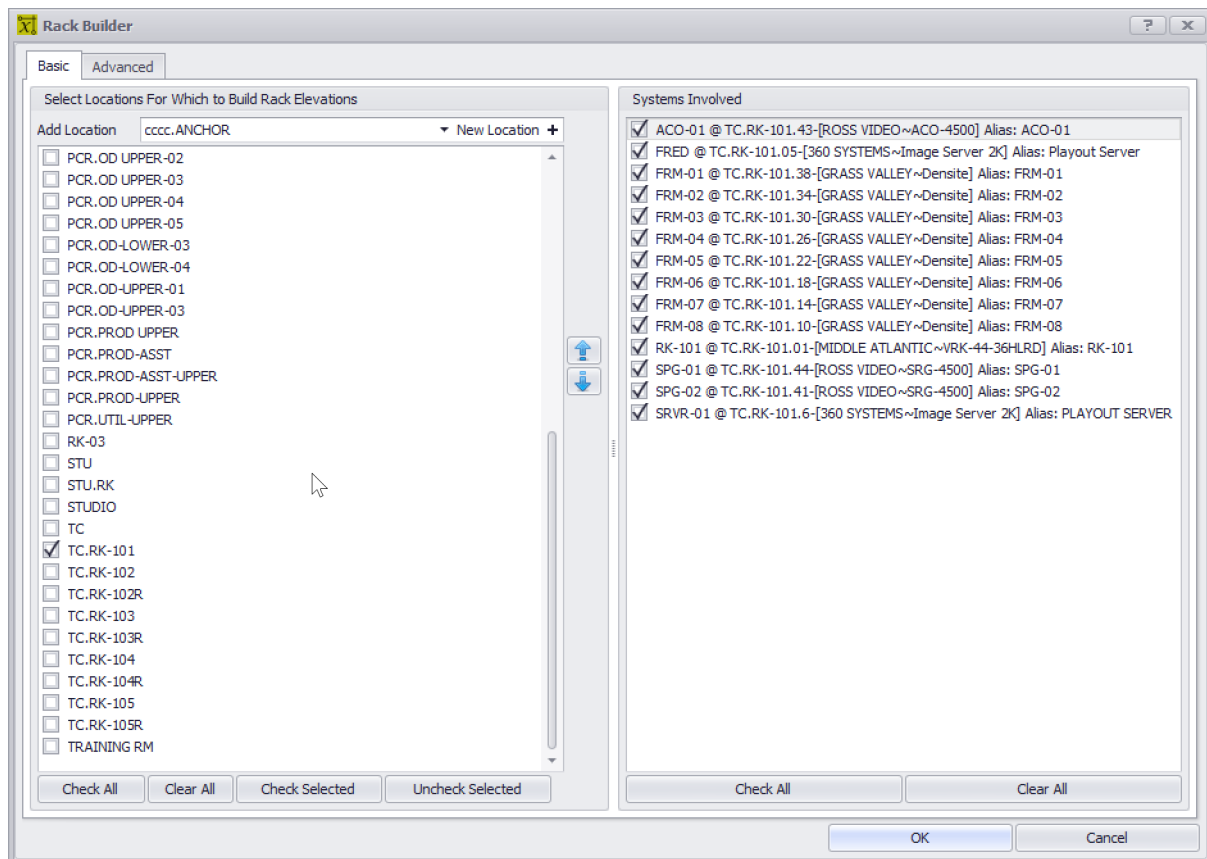
- **SysName**
- **Manufacturer**
- **EquipmentName**
- **Location**
- **Alias**
- **Description**
- **User1 - 4**
- **IPAddress**
- **Subnet Mask**

Operation

Drag the named descriptor to the target on the map.
To hide a descriptor drag it to the Hidden Items frame.

5.2.3.2.2 Rack Builder Tool

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ >



Drawing > Advanced Tools > Rack Builder

Commandline: rb

Explanation

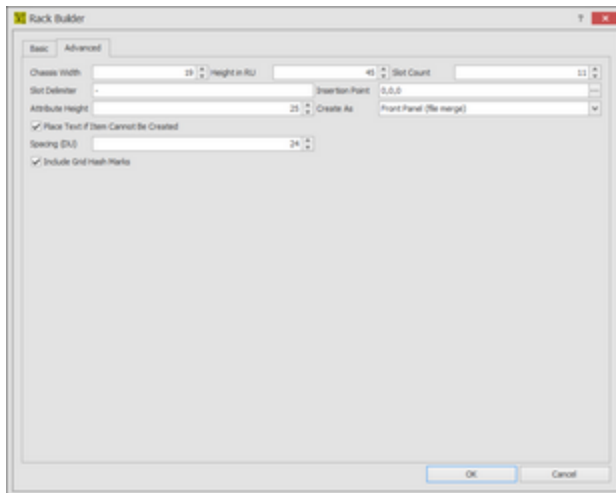
For more information see [How the Rack Builder Works](#) ¹¹².

Automatically generate rack layout. Technically speaking we are populating rack locator grids. They may or may not display an actual rack depending on whether you have assigned a SysName to a rack.

The **[Basic]** tab allows you to select the locations to include in the Rack Building function. As you select each location, the Systems Involved list will populate. You can further narrow your equipment list by changing the selection in the right-hand pane.

Dialog Options

- **Advanced** tab - exposes properties that control the behavior of the utility.

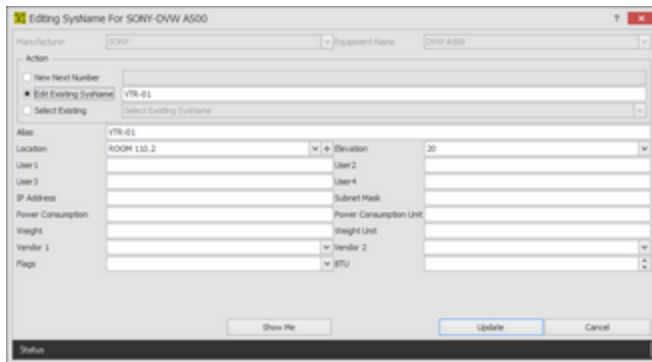


- **Chassis Width** - Sets the width of the chassis in DU.
- **Height in RU** - Sets the height of the locator grid in Rack Units (RU = 1.75 inches or 4.445cm).
- **Slot Count** - Sets the number of slots per locator grid. This is used to position items that may not be located at the insertion point of the rack unit.
- **Slot Delimiter** - WireCAD searches the Elevation field for numeric values first then for the slot delimiter if found it parses the data into two values: the elevation and the slot, or in other words how far up in the rack and how far over.
- **Insertion Point** - Where to start the whole process. Clicking the [...] ellipsis button will temporarily dismiss the dialog so you can pick a point from the active drawing.
- **Attribute Height** - If view rule is not **Front Panel (file merge)**, sets the attribute height of the displayed text.
- **Create As** -
 - Front Panel (file merge)** = use the dwg file found in the equipment definition **BlockRef (Front Panel File)**.
 - Front Panel (from dims)** = use the dimension data from the equipment definition to create a 3D rack block.
 - From Dims if file not found** = Use dimension data if the **BlockRef** is not found.
- **Place Text If Item Cannot Be Created** - If the item cannot be created due to lacking data, place a text marker in the drawing at the location.
- **Include Grid Hash Marks** - This will normally be checked unless you are rebuilding a drawing that already has the locator grids.
- **Spacing DU** - Sets the location grid spacing in Drawing Units. Measures from left edge to left edge.

5.2.3.2.3 Assign Sysname

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] >





Drawing > Advanced Tools > Equipment Library [Add to Project Database Only (Assign SysName)]
Double-click WireCAD Block
Commandline: le

Explanation

This dialog is presented when assigning a **SysName**. The **SysName** field is automatically generated based on the project **SysName Format** and the **Next Numbers** table.

Prerequisites

This dialog is only shown when **Verbose SysName Assignment** is checked.

Related Topics:

[How To: Assign a SysName](#) ¹⁰⁴

Dialog Options

- **Manufacturer and Model** - Filled automatically in most cases.
- **SysName** - This number is automatically generated based on the project **SysName Format** and the **Next Numbers** table. **WireCAD** will always present the next number though you do not have to use it. In fact, there are times when you should select the existing **SysName** from the dropdown list. For example say you have shown the video ports of VTR-01 in one drawing and the audio ports of VTR-01 in another drawing. When you assign the first instance you will get VTR-01 as the suggested SysName. The second instance will suggest VTR-02. You will then click the dropdown and select VTR-01 thus ensuring that the two representations of the same device have the same name.
- **Alias** - Friendly, functional, or descriptive name for the device.
- **Location** - of the device. For more information see the [Locations Grid](#) ⁵³⁷. Clicking the **[+]** button shows the [New Locations](#) ³⁹² dialog.
- **Elevation** - If numeric then WireCAD assumes RU. If not numeric it does not matter. There is one special circumstance where the data will be non-numeric but WireCAD will understand how to parse it. That is the situation when we want to locate something in a slot in a frame in a rack. We will need to use the form [elevation][slot delimiter][slot]. For example let's say that I have a DA that needs to go in slot 5 of a frame located at elevation 20 and my [project slot delimiter](#) ³⁶⁶ is a dash [-]. I would type into the Elevation field 20-5.

- **User Fields** - 50 characters max. You can define these captions in the **Translation Manager - Plugins > Translation Manager**. Search for SysNameUser in the key field.
- **IP Address Subnet Mask** - Masked to either IPv4 or IPv6 based on the [Project Settings](#) ³⁴⁶ dialog.
- **Power Consumption** - Pulled from the global Equipment definition or typed manually here.
- **Weight** - Pulled from the global Equipment definition or typed manually here.
- **Vendor 1 / Vendor 2** - Pulled from the global Equipment definition or typed manually here.
- **Flags** - Sort, query, filter flags. You can define these flags in the **Translation Manager - Plugins > Translation Manager**. Search for SysNameFlagItem in the key field.
- **BTU** - Heat load in British Thermal Units. Pulled from the global Equipment definition or typed manually here.

5.2.3.2.4 Assign Cable Number

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ >



Double-click WireCAD Cable
Commandline: ac

Explanation

This dialog is presented when assigning a Cable number if the **Verbose Cable Assignment** is checked. The **Cable Number** field is automatically generated based on the project **Cable Number Format** and the **Next Numbers** table.

Prerequisites

This dialog is show only when the **Verbose Cable Assignment** option is checked on the **Advanced Tools** ribbon tab.

Related Topics:

[How to Assign a Cable Number](#) ¹⁰⁹

[How to Assign Multiple Cable Numbers](#) ¹¹⁰

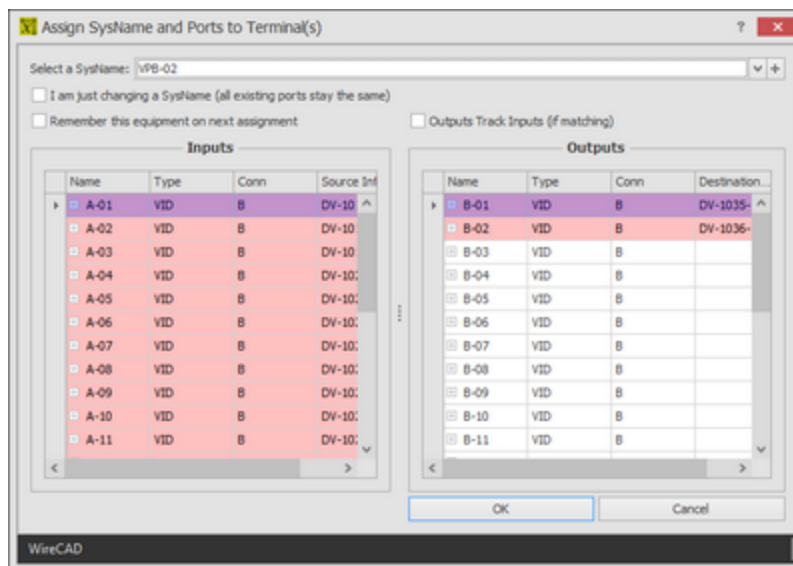
Dialog Options

- **CableNo** - This number is automatically generated based on the project **Cable Number Format** and the **Next Numbers** table. **WireCAD** will always present the next number though you do not have to use it.
- **Cable Type Manu, Cable Type*, Signal Type***
- **Named Path** - List of [Named Paths](#) ^[538]. See the documentation on the **Named Paths** table. Selecting a **Named Path** will cause the **Length** field to change.
- **Length** - Manually enter the cable length or select a Named Path.
- **Integrator** - Who is responsible for the installation of this cable.
- **User Fields** - You can define these captions in the **Translation Manager - Plugins > Translation Manager**. Search for CableUser in the key field.
- **Sheet** - Automatically filled with the current drawing name.
- **Replaced By** - A housekeeping field that you may choose to use.
- **Pinout** - If the Enable Pinouts setting is checked then select the Pinout to apply to this cable.
- **Vendor 1 / Vendor 2** - Pulled from the global Cable Type definition or typed manually here.

* changes to these fields may cause the number to recalculate.

5.2.3.2.5 Assign Terminals

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] >



Double-click Terminal or Select Terminal and Click Advanced Tools > Assign Terminal(s)
Commandline: ats

Explanation

This dialog is presented when assigning a terminal or a selection of terminals. The process is to first select the SysName to use, then select the Inputs and Outputs port.

Note that the **Inputs** or **Outputs** grid may be disabled if the terminal to which we are assigning this information does not have a CP_IN or CP_OUT to receive the data.

Record Highlighting

Red - Port assigned to a cable. The **Source Info** and **Destination Info** contains the **CableNo : SysName>Port info**.

Yellow - A cable has been attached to this port somewhere in the project. The **Source Info** and **Destination Info** contains the **SysName>Port info**.

Prerequisites

In order to assign a terminal you must first assign a **SysName** from which you will select ports.

Related Topics:

[Understanding Terminals](#) 

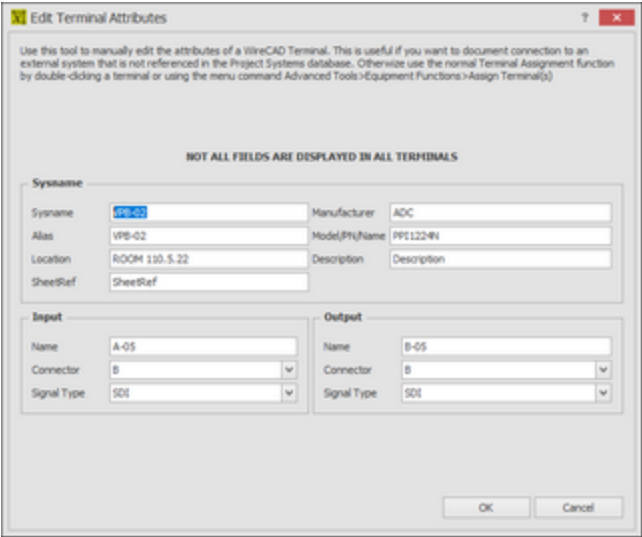
Dialog Options

- **Select a SysName** - The SysName from which we will select ports.
- **I am just changing a SysName (all existing ports stay the same)** - Use this if you have already assigned a **SysName** to a terminal but you want to switch with another **SysName**.
- **Remember this equipment on next assignment** - Persist the selected **SysName** on next Terminal Assignment.
- **Outputs Track Inputs (if matching)** - Select a matching record from the **Outputs** list if the string matches the record in the **Inputs** Name field. The Names must match exactly.

5.2.3.2.6 Assign Terminal No Database Reference

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [Advanced Tools Dialogs](#)  >





[Ctrl] + Double-click Terminal
Commandline: none

Explanation

Occasionally you will need to land a cable on a device that is not your responsibility. The cable is yours but the equipment is not.
You can use a **Terminal** to do this.
Using this method the data in the **Terminal** will not have a matching entry in the **Equipment List**.

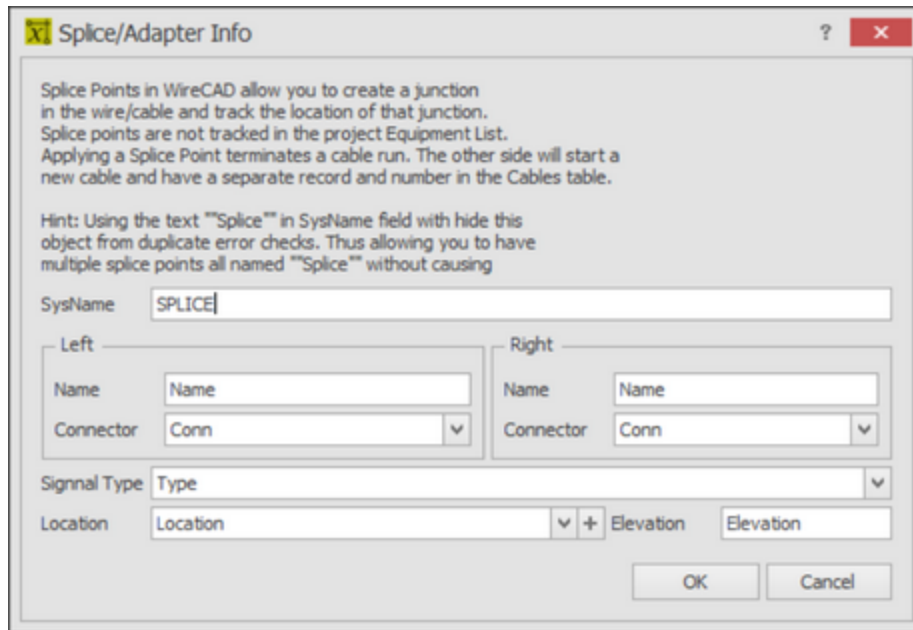
Note that the **Inputs** or **Outputs** section may be disabled if the terminal to which we are assigning this information does not have a CP_IN or CP_OUT to receive the data.

Prerequisites
None.

Related Topics:
[Understanding Terminals](#) ¹⁴⁹

5.2.3.2.7 Assign Splice

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ >



Double-click Splice Point Terminal Commandline: none

Explanation

Occasionally you will need to splice a cable through a connection point of some sort such as:

- Terminal blocks.
- Punch downs.
- Butt splices.

You can use a **Splice** to do this.

Using this method the data in the **Splice** will not have a matching entry in the **Equipment List**.


Note that the **Inputs** or **Outputs** section may be disabled if the terminal to which we are assigning this information does not have a CP_IN or CP_OUT to receive the data.

Using the string "SPLICE" in the SysName will cause the cable assignment function to ignore duplicate port checks.

Prerequisites

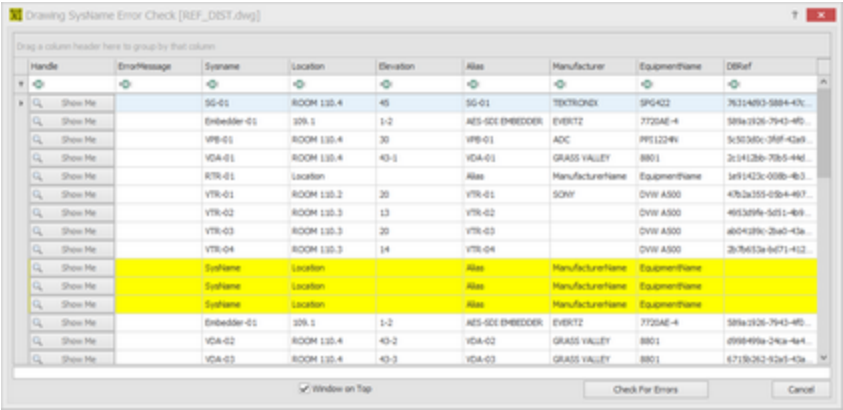
None.

Related Topics:

[Understanding Terminals](#) 

5.2.3.2.8 SysName Error Check

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [Advanced Tools Dialogs](#) [428] >



Drawing > Advanced Tools > Drawing SysName Error Check
Commandline: drawingsysnameerrorcheck

Explanation

Check the drawing against the database. Checks the following:

- Retrieves all equipment from the drawing.
- Checks the drawing SysName against the database.
- If matched the record is shown normally. If orphaned or not assigned the background color will be changed.

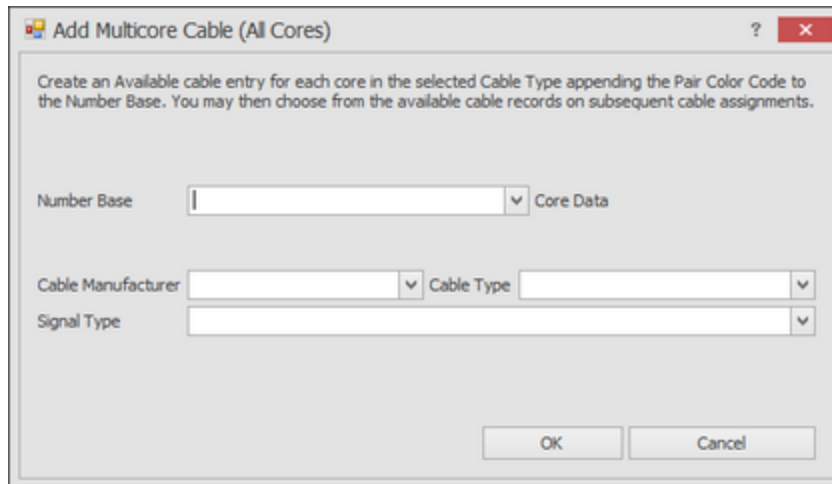
Dialog Options

- Show Me** - Zooms to the selected entity.
- Check for Errors** - Scans then fills the grid.

5.2.3.2.9 Add Multi-core Cable

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [Advanced Tools Dialogs](#) [428] >





Drawing > Advanced Tools > Add Multi-core Cable

Commandline: mc

Several Others

Explanation

Often times we install bundled cables - cables with multiple cores. The outside jacket gets a number and each inner core an extension identifier like: A-1001-RED. Where A-1001 is the number that goes on the outer jacket and A-1001-RED goes on the RED core. Use the **Add Multi-core Cable** tool to create a record in the database for each core in the **Cable Type**. The **CableNo** field will have the full number and the **CableNoPrefix** field will have the number base. All cores will be marked Available for use. You can then assign one or all or any of them during the normal **Cable Number** Assignment process. You must have **Verbose Cable Assignment** checked in order to assign cores in a multi-core cable. Otherwise you will get the next number default.

Dialog Options

- **Number Base** - This is the part that goes into the Cable No Prefix field.
- **Cable Type** - Only Cable Types that are flagged as Multi-core will show here.
- **Signal Type**

5.2.3.2.10 Cable Error Check

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Drawing Dialogs](#) [389] > [Advanced Tools Dialogs](#) [428] >



Drawing Cable Error Check [REF_DIST.dwg*]

Handle	ErrorMessage	CableNo	SrcSys	SRCPin	SRCLoc	SRCConn	DestSys	DestPin	DestLoc	DestCor
Sho	REF-1001-	REF-1001-	SG-01	BLK-01	ROOM 110.4	B	VDA-01	In	ROOM 110.4	B
Sho	DA-1015-	DA-1015-	SG-01	AES-01	ROOM 110.4	B	Embedder-01	AES 103	109.1	B
Sho	Not Assigned	Cable#	SG-01	D-BARS	ROOM 110.4	B	VPB-01	A-17	ROOM 110.4	B
Sho	REF-1002-	REF-1002-	VDA-01	OUT.1	ROOM 110.4	B	RTR-01	REF	Location	
Sho	V-1017-	V-1017-	VDA-01	OUT.2	ROOM 110.4	B	VTR-01	REF	ROOM 110.2	B
Sho	V-1018-	V-1018-	VDA-01	OUT.3	ROOM 110.4	B	VTR-02	REF	ROOM 110.3	B
Sho	V-1019-	V-1019-	VDA-01	OUT.4	ROOM 110.4	B	VTR-03	REF	ROOM 110.3	B
Sho	V-1020-	V-1020-	VDA-01	OUT.5	ROOM 110.4	B	VTR-04	REF	ROOM 110.3	B
Sho	Not Assigned	Cable#	VDA-01	OUT.6	ROOM 110.4	B	SysName	Name	Location	Cor
Sho	Not Assigned	Cable#	VDA-01	OUT.7	ROOM 110.4	B	SysName	Name	Location	Cor
Sho	Not Assigned	Cable#	VDA-01	OUT.8	ROOM 110.4	B	SysName	Name	Location	Cor
Sho	DA-1016-	DA-1016-	SG-01	AES-02	ROOM 110.4	B	Embedder-01	AES 104	109.1	B

Record 1 of 39

Source Pointer

Dest Pointer

Needs attention

Window on Top

Check Errors Now

Close

Multi Cable. This checks only the first cable in a multirelational cable. All others may show up as appearing in db only.

Drawing > Advanced Tools > Drawing Error Check

Commandline: dec

Explanation

Check the drawing against the database. Checks the following:

- Retrieves all cables from the drawing.
- Checks the drawing Cable against the database.
- If matched the record is shown normally. If orphaned or not assigned the background color will be changed.

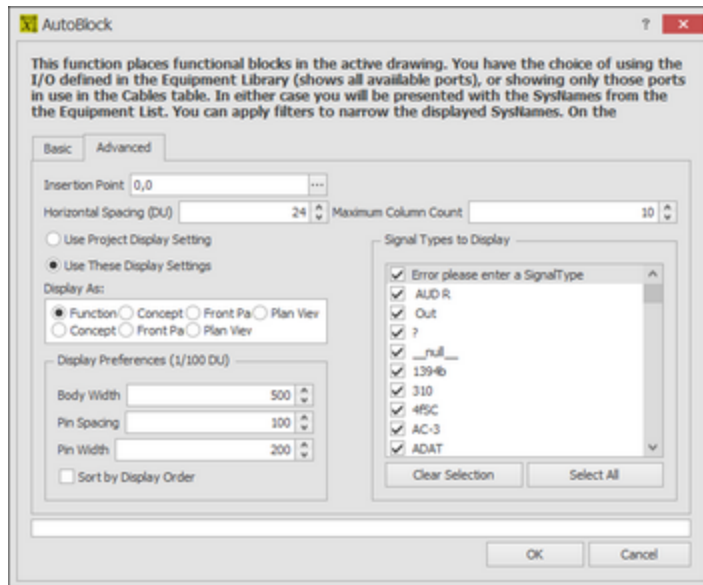
Dialog Options

Item	Description
Show Me	Zooms to the selected entity
Check for Errors	Scans then fills the grid.

5.2.3.2.11 Auto Block

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ >





Drawing > Advanced Tools > Auto Block

Commandline: ab

Explanation

The Auto Block tool automatically places functional blocks in the drawing. This tool requires that the Project Systems table be populated.

Possible Uses

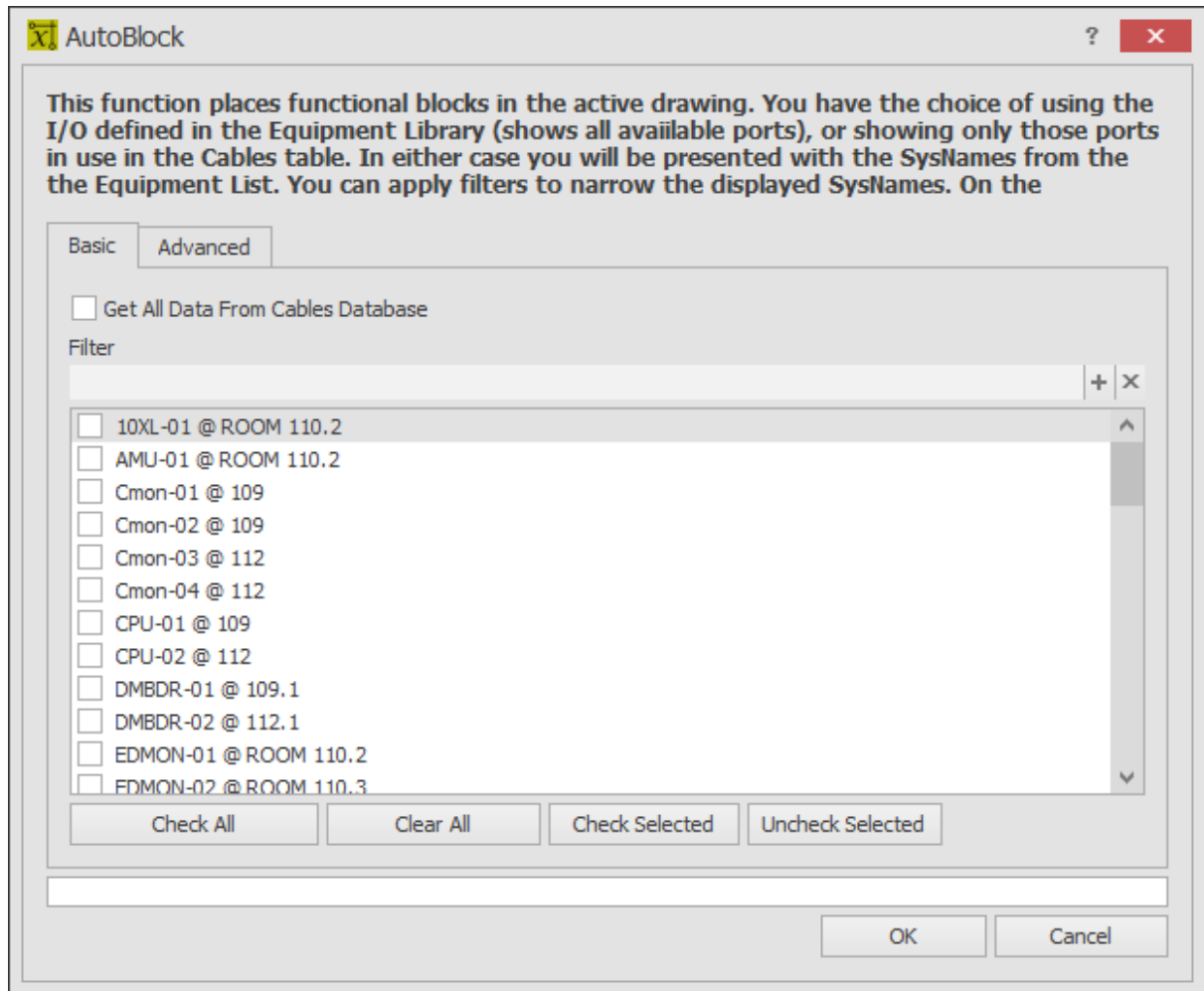
- Add functional blocks after creating SysNames from Rack Builder drawings.
- Create overall systems views.
- Create drawings from imported data.

Related Topics:

[Auto Block tool.](#) ¹²⁰

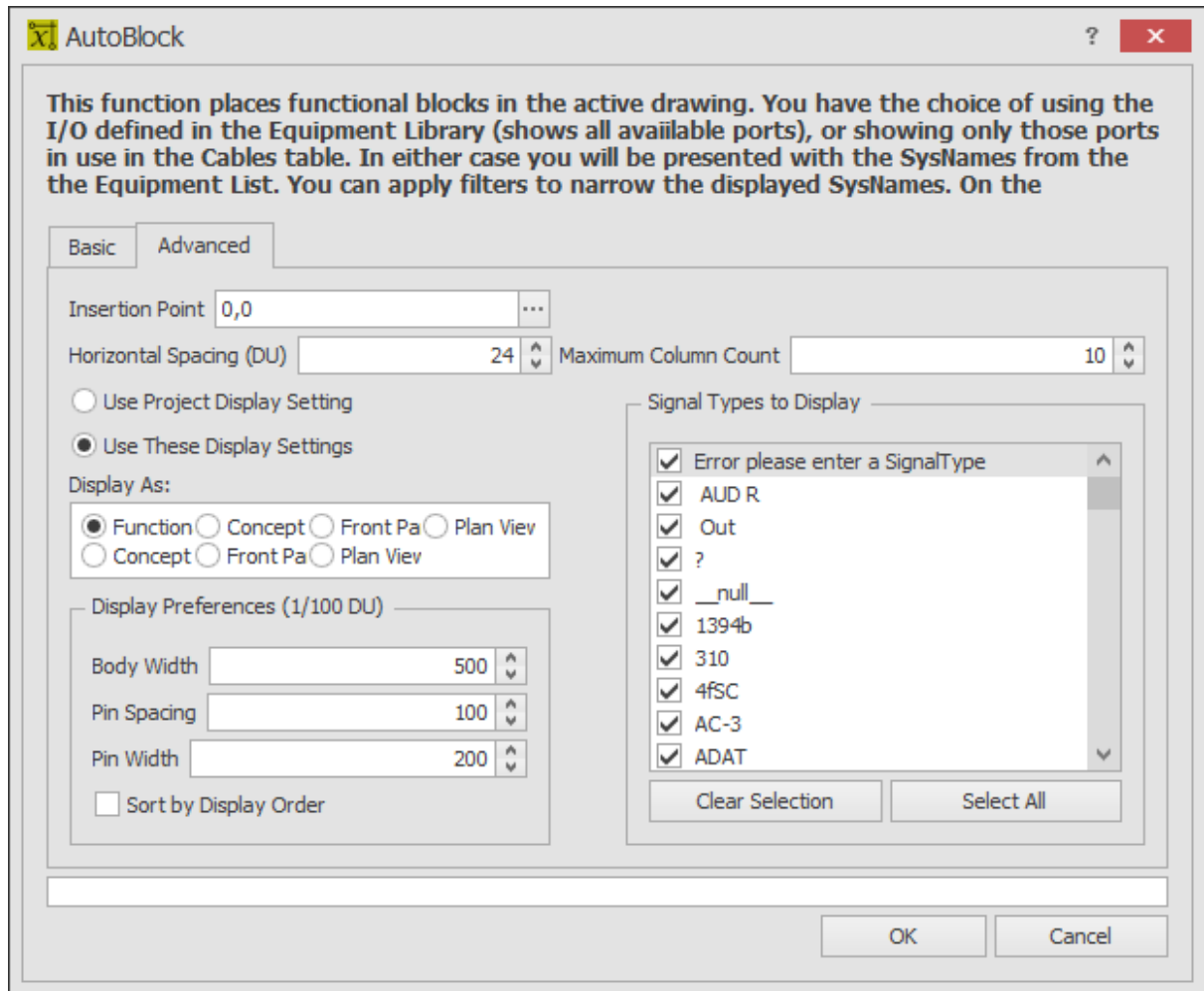
Dialog Options

- The [Basic] tab allows you to determine which systems to add to the drawing.



- **Filter** - Filters the list by your criteria.

The Advanced Tab

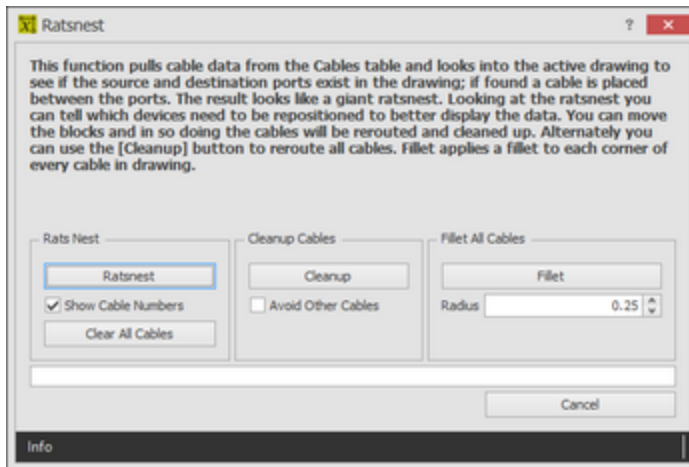


- **Insertion Point** - The point we start from.
- **Horizontal Spacing DU** - How far apart horizontally. The vertical spacing is defined by the height of the highest block in the row.
- **Maximum Column Count** - How many columns horizontally.
- **Get Port Data From Cables Database** - Select this option to search the cables database for port info instead of the global equipment database. This will effectively show only those ports to which we have attached cables.
- **Display As** - How to display the blocks.
- **Display Preferences** - If Functional Block or Concept block is selected then set basic display parameters.
- **Signal Types to Display** - Filter ports by the selected signal types.

5.2.3.2.12 Ratsnest

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Advanced Tools Dialogs](#) ^[428] >





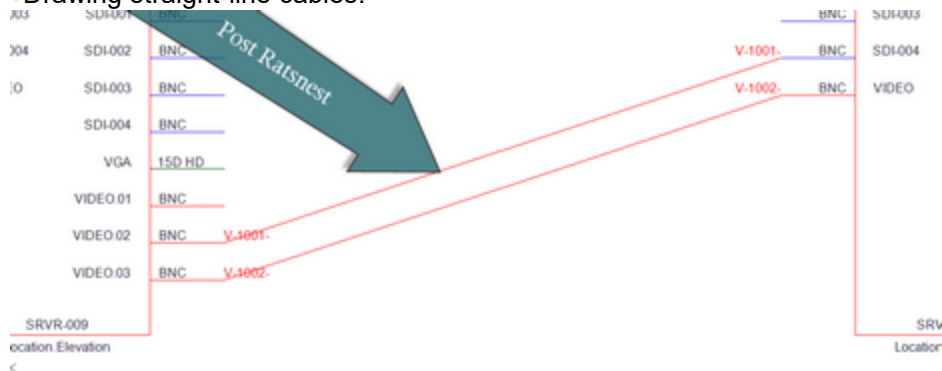
Drawing > Advanced Tools > Ratsnest

Commandline: **rn**

Explanation

The **Ratsnest** tool works in conjunction with the Auto Block tool. How it works:

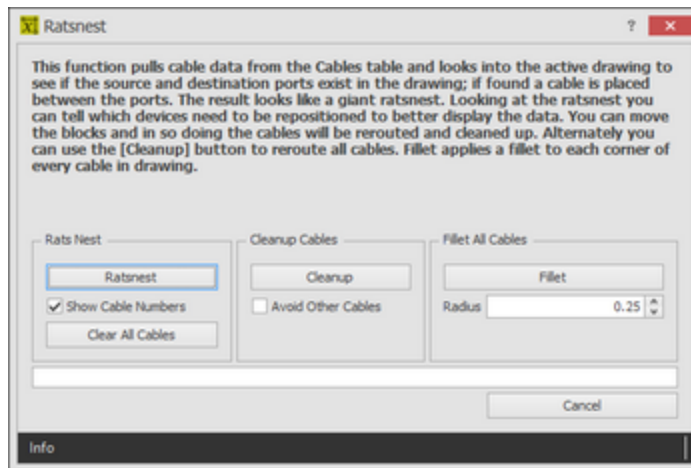
- Get the cables collection.
- Get the SysName>Port info from the drawing.
- Find matches.
- Drawing straight-line cables.



Possible Uses

- Create overall systems views.
- Create drawings from imported data.

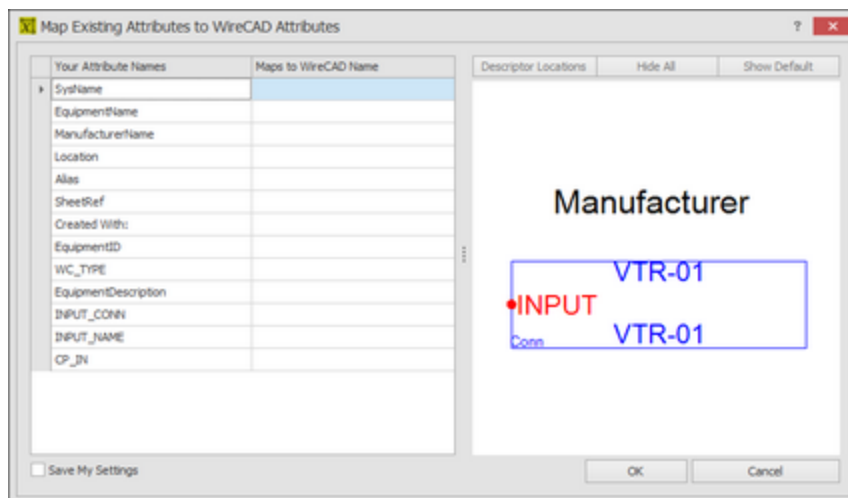
Dialog Options



- This tool has three sections. The Rats Nest section does the work of placing the cables in the drawing as defined in the Cables database.
- **[Ratsnest]** - Run the utility to place the cables.
- **Show Cable Numbers** - With or without cable numbers.
- **[Clear All Cables]** - Removes **ALL** cables from the drawing. Not just ratsnested cables. This will remove ALL.
- **[Cleanup]** - Applies the auto-router to all cables in the drawing.
- **Avoid Other Cables** - Auto-router avoids other cables on cleanup.
- **[Fillet]** - Applies fillets to **ALL** cables in the drawing.
- **Distance** - Fillet distance in 100/DU.

5.2.3.2.13 WireCADify Block

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ >



Drawing > Advanced Tools > WireCADify Block

Commandline: wirecadifyblock

Explanation

Occasionally you will want to use the geometry of a non-WireCAD generated CAD block. In order to do this you will need to give the necessary attribute set to the CAD block so that it can function in WireCAD as a working assignable entity.

Steps

- Start the command.
- Select a standard CAD block. It must be a block and not exploded entities.
- Follow the directions in the dialog to complete the process of adding the WireCAD attribute set to the CAD block.

Possible Uses

- Use existing CAD drawings and work with WireCAD to move the drawing forward.
- Create custom appearance.

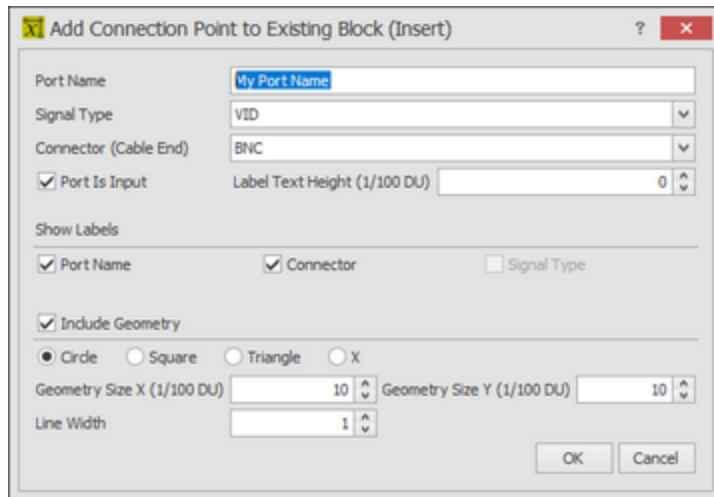
Dialog Options

- Attribute Map Grid** - Map your attribute to WireCAD's.
- Descriptor Locations** - Reposition the base WireCAD attributes using the Descriptor Locations map.
- Hide All** - Add all the WireCAD attributes but hide them from view.
- Show Default** - Show the default attribute set.
- Preview** - Preview of the merged block.

5.2.3.2.14 Add Connection Point

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [Advanced Tools Dialogs](#)  >





Drawing > Advanced Tools > Add Connection Point

Commandline: addconnectionpoint

Explanation

If you need to place a connection point (a point to which you can connect a WireCAD cable). You can use this tool.

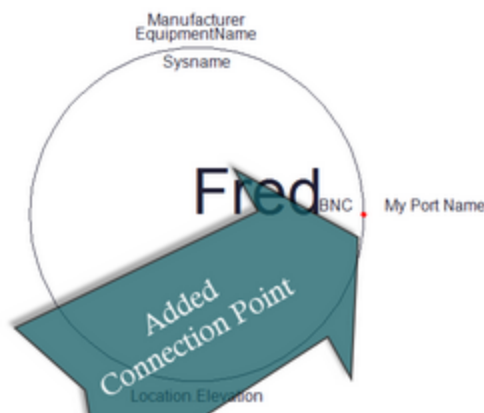
You must use this tool on blocks that have the WireCAD attribute set.

Steps

- Start the command.
- Select a WireCAD block or a block that you have run the WireCADify command on.
- Select the geometric point at which the connection point will appear. This should be something that is easy to snap a WireCAD wire to.
- Follow the directions in the dialog to complete the process of adding the WireCAD attribute set to the CAD block.

Possible Uses

- Use existing CAD drawings and work with WireCAD to move the drawing forward.
- Create custom appearance.



Dialog Options

- **Port Name, Signal Type, Connector (Cable End), Label Text Height, Show Labels**
- **Port is Input** - This will determine which side the label appears on.
- **Include Geometry** - Display something to snap to.

5.2.3.2.15 Cable Connection Exists

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Advanced Tools Dialogs](#) ⁴²⁸ >



Not directly callable
Commandline: none

Explanation

The standard cable number assignment process is a multi-step process. One of the checks that is performed is to see if the ports to which we have attached our cable are already assigned to some other cable in the database. If they are found to exist in the database this dialog is shown indicating whether it is the source or the destination that is in question. Bear in mind that multiple records might meet this criteria; hence the grid.

There are legitimate instances where this dialog will be shown and your choice requested:

- Y cable - The first assignment will proceed without error. The second will show this dialog indicating that the source exists. As it should. You have a choice regarding numbering in this case. Do you duplicate the existing number with a new record in the database or request the new number.
- One-To-Many - see above.
- Many-To-One - Inverse of above. You will see the destination exists.
- If you land multiple cable on the same port to parallel the connection.

Pay close attention if you are presented with this dialog. It is catching a common mistake before it gets to your run sheet.

Prerequisites

At least one cable record in the **Cables** database.

Related Topics:

[How To: Assign a Cable](#) 109

[How To: Assign Multiple Cables](#) 110

5.2.3.2.16 Add Spare Cable

Add Spare(s)

Cable Type: + Signal Type: +

Left Side

Location: +

Connector: +

Right Side

Location: +

Connector: +

Count: 1

Geometry

☒ Show Location ☒ Show Connector

☒ Show Cable Type ☒ Show Cable Number(s)

☒ Color by Signal Type

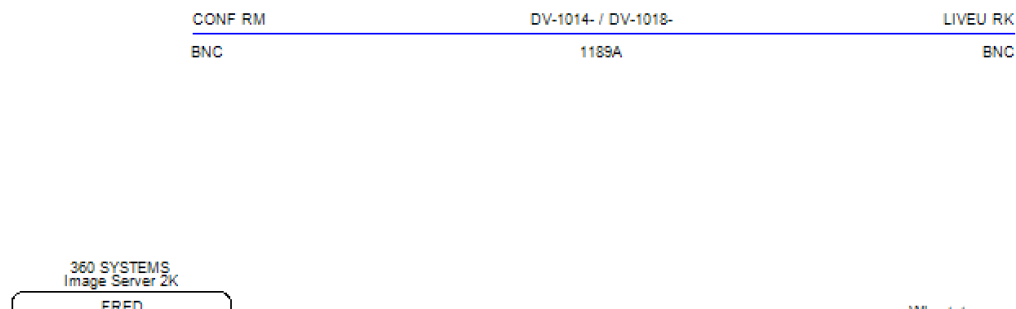
Drawing > Advanced Tools > Add Spare Cable(s)
Commandline: AddSpareCables

Explanation

Add a cable entry to the database and add a polyline to the drawing to represent the spare(s).

Steps

- Start the command.
- Enter the Cable Type and Signal Type.
- Enter the source and destination locations and connectors
- Enter the count of spare cables to add.



5.2.3.2.17 Error Check Pointers

Pointer Error Check [Paste Test.DWG]													
Src Handle	Message	Src Sysname	Src Alias	Src Location	Src Port	Src DWG	O...	Dest Handle	Dest Sysname	Dest Alias	Dest Location	Dest Port	Dest DWG
Show Me	OK	DCIOH:001	Digital Cinem...	2312:2M:Proj...	AUTOMATIO...	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	FIRESH:001	Shielded 4-W...	2312:2M:Proj...	RJ45 1	Paste Test.D...
Show Me	OK	BLUDAN:001	BSS AUDIO D...	2312:2M:Proj...	BLU link	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	CP950:652	DOLBY CP950A	2312:2M:Proj...	DOLBY ATMO...	Paste Test.D...
Show Me	OK	HDDA2:654	CRESTRON H...	2312:2M:Proj...	HDMI	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	HD70RX:655	CRESTRON HD...	2312:2M:Proj...	HDMI	Paste Test.D...
Show Me	OK	CP950:652	DOLBY CP950A	2312:2M:Proj...	HDMI IN	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	HDDA2:654	CRESTRON H...	2312:2M:Proj...	HDMI 1	Paste Test.D...
Show Me	OK	DCIOH:001	Digital Cinem...	2312:2M:Proj...	HDMI IN	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	HDDA2:654	CRESTRON H...	2312:2M:Proj...	HDMI 2	Paste Test.D...
Show Me	OK	DCIOH:001	Digital Cinem...	2312:2M:Proj...	LINE INPUT [...]	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	CP950:652	DOLBY CP950A	2312:2M:Proj...	HI	Paste Test.D...
Show Me	OK	ULXDQ4:002	Quad-Chann...	2312:2M:Proj...	Antenna A	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	ULXDQ4:001	Quad-Chann...	2312:2M:Proj...	Antenna B	Paste Test.D...
Show Me	OK	ULXDQ4:002	Quad-Chann...	2312:2M:Proj...	Antenna B	Paste Test.D...	<input checked="" type="checkbox"/>	Show Me	ULXDQ4:001	Quad-Chann...	2312:2M:Proj...	Antenna A	Paste Test.D...
Show Me	Unable to fin...	SysName	Alias	Location	Name	Paste Test.D...	<input type="checkbox"/>	Show Me					
Show Me	Unable to fin...	SysName	Alias	Location	Name	Paste Test.D...	<input type="checkbox"/>	Show Me					
Show Me	Unable to fin...	SysName	Alias	Location	Name	Paste Test.D...	<input type="checkbox"/>	Show Me					
Show Me	Unable to fin...	SysName	Alias	Location	Name	Paste Test.D...	<input type="checkbox"/>	Show Me					
Show Me	Unable to fin...	SysName	Alias	Location	Name	Paste Test.D...	<input type="checkbox"/>	Show Me					
Show Me	Unable to fin...	SysName	Alias	Location	Name	Paste Test.D...	<input type="checkbox"/>	Show Me					
Checking: Destination Side Pointers													
Show Me	OK	NSW1:002	CISCO NETW...	2312:2M:Proj...	Power	RACK 002.D...	<input type="checkbox"/>	Show Me	20PDU:002	POWER STRIP	2312:2M:Proj...	(Circuit A) Ou...	Paste Test.D...
Show Me	OK	NWS2:002	NETGEAR NE...	2312:2M:Proj...	POWER	RACK 002.D...	<input type="checkbox"/>	Show Me	20PDU:002	POWER STRIP	2312:2M:Proj...	(Circuit A) Ou...	Paste Test.D...
Show Me	OK	R-969A-002	TT PATCHBAY	2312:2M:Proj...	TOP 29	RACK 002.D...	<input type="checkbox"/>	Show Me	QIOL40:019	4 CH LINE OU...	2312:2M:Proj...	LINE OUT CH 1	Paste Test.D...
Show Me	OK	R-969A-002	TT PATCHBAY	2312:2M:Proj...	TOP 30	RACK 002.D...	<input type="checkbox"/>	Show Me	QIOL40:019	4 CH LINE OU...	2312:2M:Proj...	LINE OUT CH 2	Paste Test.D...
Show Me	OK	R-969A-002	TT PATCHBAY	2312:2M:Proj...	TOP 31	RACK 002.D...	<input type="checkbox"/>	Show Me	QIOL40:019	4 CH LINE OU...	2312:2M:Proj...	LINE OUT CH 3	Paste Test.D...
Show Me	OK	R-969A-002	TT PATCHBAY	2312:2M:Proj...	TOP 32	RACK 002.D...	<input type="checkbox"/>	Show Me	QIOL40:019	4 CH LINE OU...	2312:2M:Proj...	LINE OUT CH 4	Paste Test.D...
Show Me	OK	R-969A-002	TT PATCHBAY	2312:2M:Proj...	TOP 33	RACK 002.D...	<input type="checkbox"/>	Show Me	QIOL40:020	4 CH LINE OU...	2312:2M:Proj...	LINE OUT CH 1	Paste Test.D...
Show Me	OK	R-969A-002	TT PATCHBAY	2312:2M:Proj...	TOP 34	RACK 002.D...	<input type="checkbox"/>	Show Me	QIOL40:020	4 CH LINE OU...	2312:2M:Proj...	LINE OUT CH 2	Paste Test.D...
Show Me	OK	R-969A-002	TT PATCHBAY	2312:2M:Proj...	TOP 35	RACK 002.D...	<input type="checkbox"/>	Show Me	QIOL40:020	4 CH LINE OU...	2312:2M:Proj...	LINE OUT CH 3	Paste Test.D...
<input checked="" type="checkbox"/> Window On Top													
<input type="button" value="Check for Errors"/>										<input type="button" value="Cancel"/>			

Drawing > Advanced Tools > Pointer Operations > Error Check Pointers ...
Commandline: ecp

Explanation

Error check the linkage between pointers. Properly linked pointers are paired and pointed to each other. This tool checks only the linkage as follows:

- The source side pointer is linked to the destination side pointer.
- The destination side pointer is linked to the source side pointer.
- The pointer generations match.

The error check function does not test:

- Whether the source port is properly linked to the source pointer.
- Whether the destination port is properly linked to the destination pointer.
- Whether the source and destination is properly inserted in the project cables database.

Controls

Grid - Displays the results.

Columns

Show Me - Zoom the pointer into view on the source side.

Message - Displays the result. Possible results are:

- **OK** - It's all good. Both sides are linked properly.
- **The pointer is not properly linked in both directions** - The pointer is linked to one pointer in one direction and another in another direction.
- **Unable to find other end of the pointer set** - Can't find the other set, either because the pointer is not linked yet or the other end cannot be found.
- **Unable to get other side pointer connection point data** - Can't get the other side connection point data. This message will probably never show unless a custom pointer is not properly configured.
- **The pointer is self linked** - This condition is rarely found and difficult to create.
- **Pointer version generations do not match** - This condition occurs when pointers from different generations are linked. This condition is difficult to create.

OnSheet - Displays a check mark if the **Src Drawing** and **Dest Drawing** are the same.

Show Me - Zoom the pointer into view on the source side.

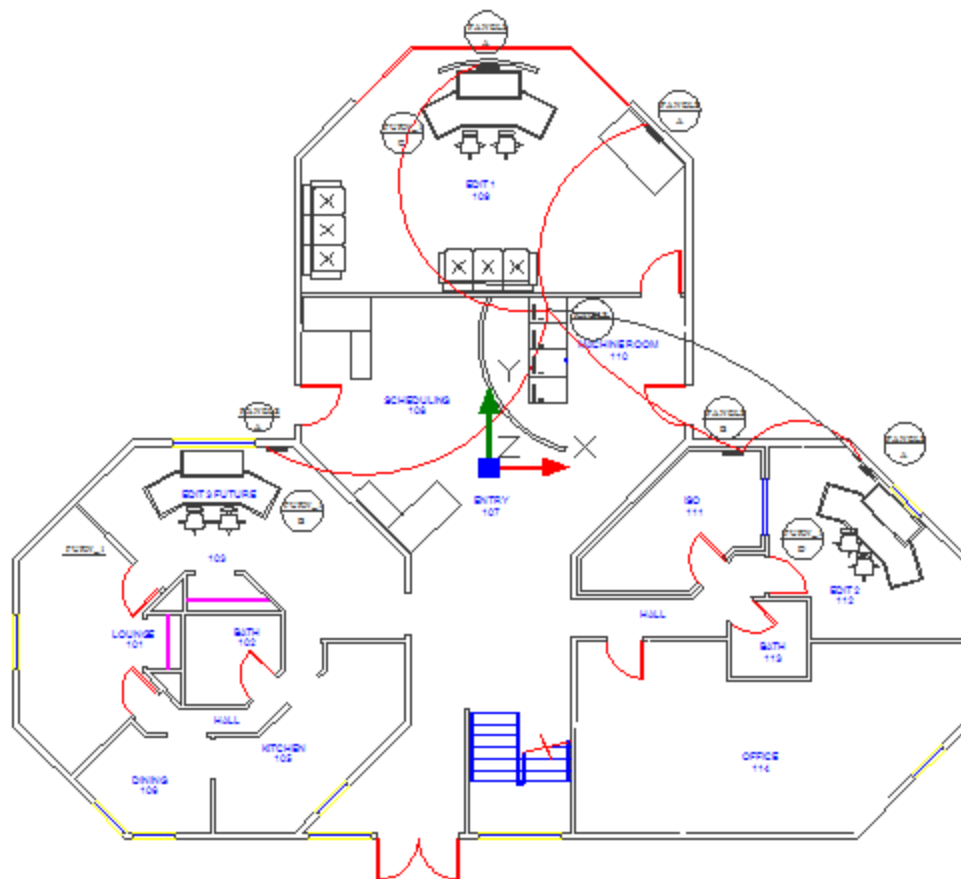
The remainder of the fields are self explanatory.

5.2.3.3 Plan View and Layout Tools Dialogs

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  >



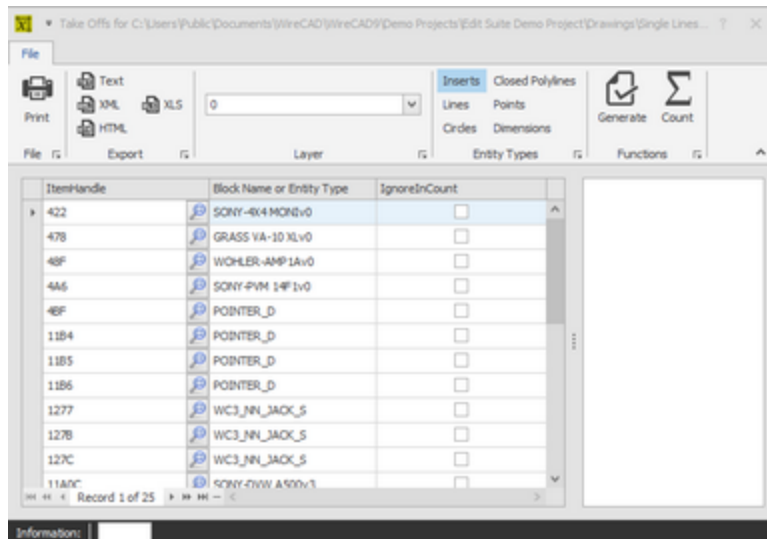
The following set of tools is targeted at the plan view space.



5.2.3.3.1 Take Offs

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Drawing Dialogs](#) ³⁸⁹ > [Plan View and Layout Tools](#)
[Dialogs](#) ⁴⁷⁰ >





Drawing > Plan View and Layout Tools > Take Offs
Commandline: showtakeoffs

Explanation

Often we need to count items in the plan view space for quoting or other purposes. The Take Offs tool facilitates this process by filtering the drawing for specific entity types on specific layers then counting those instances.

Possible Uses

- Quote preparation.
- Trouble-shooting.

Related Topics:

[How to Pull a Take-off](#) 

Dialog Options

- Print** - Print the grid.
- Export** - Export the grid.
- Layer** - What layer are we searching for the selected entity type(s).
- Entity Types** - What entity type(s) are we searching for.
- Generate** - Do It! The results will be displayed in the grid.
- Count** - Count the items in the list. The results will be displayed in the right-hand list.
- ItemHandle ShowMe** - Zooms to show the item in the drawing.
- IgnoreInCount** - Ignore me when counting.

5.2.3.3.1.1 Spare Cable

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [Plan View and Layout Tools Dialogs](#)  > [Take Offs](#) 

Prerequisites

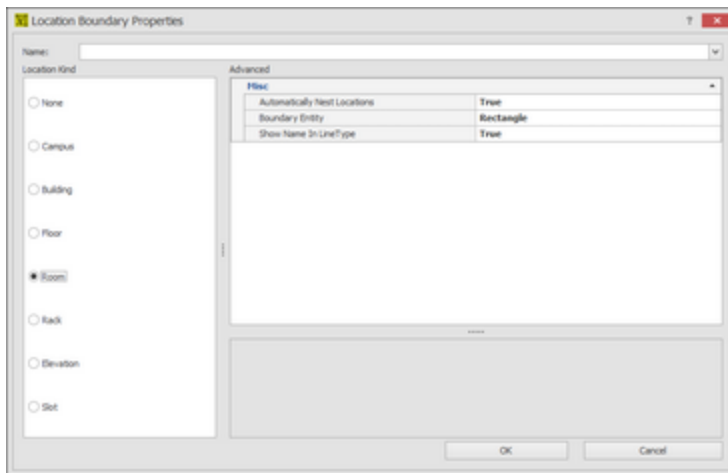
At least one drawing

Related Topics:

[How To: Add a Spare Cable](#) 

5.2.3.3.2 Location Boundary

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [Plan View and Layout Tools Dialogs](#)  >



Drawing > PlanView and Layout Tools > Location Boundary

Commandline: pw

Explanation

Define geometric boundaries for locations important to the design. For example we would want to know where on the drawing the Machine Room is but not necessarily the kitchen (unless part of our cable ends up in the kitchen).

Prerequisites

1.Source and destination Location Boundaries placed in the Plan View space.

Related Topics:

[How To: Place a Location Boundary](#) 

Operation

- 1.Start the tool.
- 2.Select a text entity with the name of the location to pre populate the dialog.
- 3.Set the location type and geometry type.
- 4.Draw the boundary
- 5.Done. Lather , rinse, repeat.

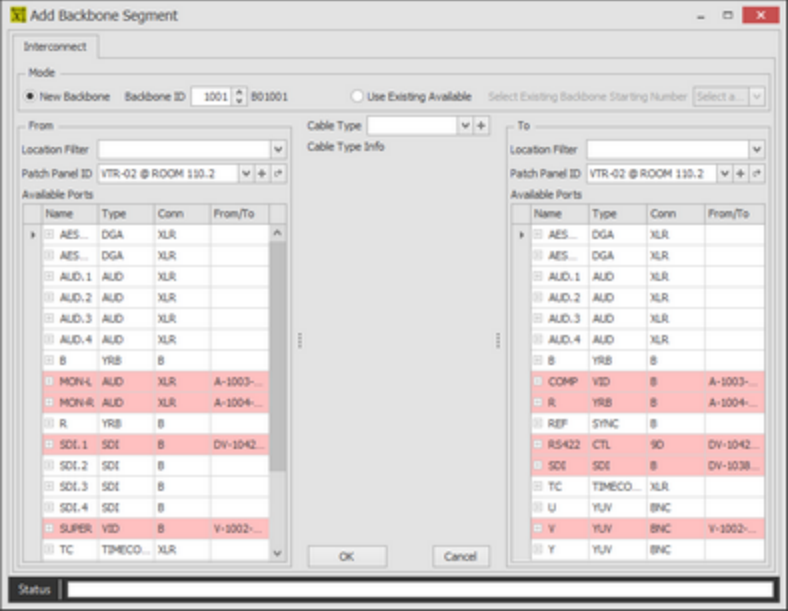
Related Topics:
None

Dialog Options

- Name** - The location name. Either from the selected entity or entered directly or selected.
- Location Kind** - Select the level of the location that you are drawing a boundary for.
- Automatically Nest Locations** - Searches the drawing for containing boundaries and formats a nested location string. The creates an entry in the **Locations** table if not found.
- Boundary Entity** - Boundaries must be a closed polyline of some sort.
- Show Name in LineType** - Creates a new **LineType** with the name of the location and renders boundary entity with the new **LineType**.

5.2.3.3.3 Draw Backbone (ENT Only)

Navigation: [Reference](#)^[288] > [Dialogs](#)^[326] > [Drawing Dialogs](#)^[389] > [Plan View and Layout Tools Dialogs](#)^[470] >



Drawing > PlanView and Layout Tools > Draw Backbone
Commandline: none

Explanation

Backbones are collections of cable/fiber that are contained in a single jacket that run from one location to another and are typically sized for growth. An example might be a fiber optic cable that contains 288 fiber cores that runs from building A to building B. We know that initially we will not use all 288 fibers and have planned for growth. As the facility needs change the usage of the backbone's fibers change.

WireCAD maintains backbones just like any multi-core cable with the exception that the Cables table record is flagged IsBackBone = true.

WireCAD will create a cable record for every core in the Cable Type used.

Prerequisites

- 1.SysNames assigned to the panels/equipment to which you will attach the ends of the Backbone.
- 2.Multi-core Cable Type representative of the Backbone.

Operation

This tool allows you to draw geometry in the Plan View space that represents the path of the Backbone. The process requires the following steps:

- 1.Draw the polyline that represents the backbone.
- 2.If locations boundaries are found in the drawing the source and destination locations will be used. Otherwise you will be prompted to define a location for each end.
- 3.Next you will be presented with the New Backbone tool where you will select the source and destination panels/ports and the cable type.
- 4.Clicking [OK] will build a record in the Cables table for each core in the multi-core cable type.
- 5.The polyline length populate the cable record Length field.



Related Topics:

[How To: Create a New Backbone](#) 

Dialog Options

- New Backbone** - Select whether we are creating a new number or assigning existing unused core.
- Backbone ID** - The Backbone number. This will be formatted based on the **Project Backbone Format** setting.
- Use Existing Available** - Pick a **Backbone** that has available cores.
- Location Filter** - Filter the list by location if the list is long and noisy.
- From Panel Info, To Panel Info** - Select the panel. The ports will be shown in the list.
- Cable Type** - Select the Cable Type.

5.2.3.3.4 Draw Cable

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [Plan View and Layout Tools Dialogs](#) 



Drawing > PlanView and Layout Tools > Draw Backbone

Commandline: **pwdc**

Explanation

This tool allows you to draw a cable in Plan View space that is assigned to the Cables table.

Prerequisites

1. SysNamed source and destination equipment placed in the Plan View space or:
2. Source and destination Location Boundaries placed in the Plan View space.

Operation

This tool allows you to draw geometry in the Plan View space that represents the path of the cable. The process requires the following steps:

1. Start the tool.
2. Select the source SysNamed equipment.
3. Select the destination SysNamed equipment.
4. Place the points in the polyline to finalize the appearance.

Related Topics:

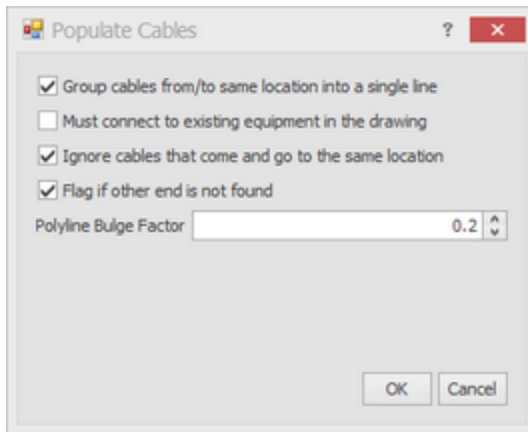
[Draw Cables](#) 

Dialog Options

- **Cable No** - Click the [...] button to generate a cable number.

5.2.3.3.5 Populate Cables

Navigation: [Reference](#)  > [Dialogs](#)  > [Drawing Dialogs](#)  > [Plan View and Layout Tools Dialogs](#)  >



Drawing > PlanView and Layout Tools > Populate Cables**Commandline:** popc***Explanation***

This tool pulls the cables from the Cables table into the drawing. The assumption with this tool is that you will use it after you have done your functional drawings and assigned cable numbers and SysNames.

Prerequisites

- 1.SysNamed source and destination equipment placed in the Plan View space or:
- 2.Source and destination Location Boundaries placed in the Plan View space.
- 3.Cable data in the Cables table that matches the placed equipment or locations.

Operation

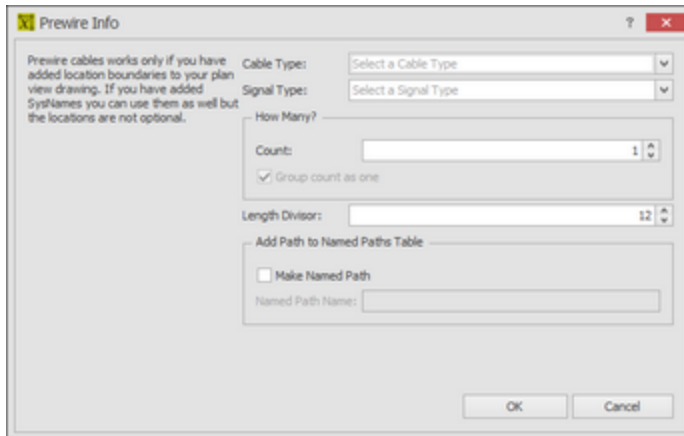
- 1.Start the tool.
- 2.Select options.
- 3.Click **[OK]** to run.

Related Topics:[How To: Populate Cables in Plan View](#)  159***Dialog Options***

- Group Cables** - Collapse cables that go from the same source and destination SysName or Location into a single polyline with hash marks to indicate how many cables are represented by the one polyline.
- Must connect to equipment in the drawing** - Only connect to plan view SysNamed equipment.
- Ignore cables that come and go to the same location**
- Flag if other is not found** - Draw a pointer out to the un-found equipment.
- Polyline Bulge Factor** - Non zero for arced cables.

5.2.3.3.6 Draw Prewire

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Plan View and Layout Tools Dialogs](#) ^[470] >



Drawing > PlanView and Layout Tools > Draw Prewire Cable

Commandline: pw

Explanation

This tool allows you to draw geometry in the drawing that represents cables that have not yet had their functions assigned. The only thing we know about these cables is the locations from which they start and end and the Cable Type. A record will be added to the Cables table based on the count defined. The record is marked Available and PreWIRE. Prewire cables can be automatically consumed later if the [Automatically Consume Prewire](#) ^[365] setting is checked.

Prerequisites

1. Source and destination Location Boundaries placed in the Plan View space.

Operation

1. Start the tool.
2. Fill in the form.
3. Draw the cable path.

Related Topics:

[How To: Draw a Prewire Cable in PlanView](#) ^[160]

Dialog Options

- **Cable Type** - Select the **Cable Type**.
- **Signal Type** - Select the **Signal Type**.

- How Many** - How many wires of this type take this path. A record will be created for each.
- Length Divisor** - We will divide the length of the polyline by this value and place that info in the **Length** field of the record. For example, say your Drawing Unit = 1 inch and you want the length in the **Length** field to be displayed in feet. Your **Length Divisor** would be 12.
- Create Named Path** - Create an entry in the **Named Paths** table with the name provided and length of the polyline.

5.2.3.3.7 Populate Equipment

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Drawing Dialogs](#) ^[389] > [Plan View and Layout Tools Dialogs](#) ^[470] >



There is no UI for this function

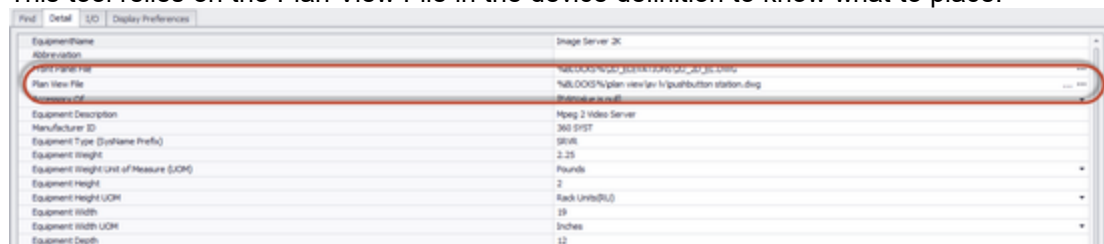
Drawing > PlanView and Layout Tools > Populate Equipment

Commandline: pope

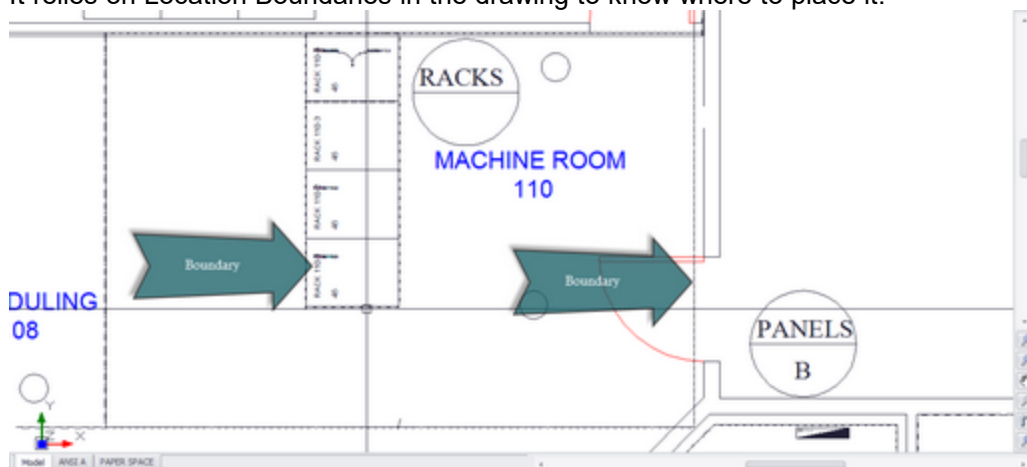
Explanation

This tool pulls the Plan View version of the equipment from the Equipment List. The assumption with this tool is that you will use it after you have done your functional drawings and assigned cable numbers and SysNames.

This tool relies on the Plan View File in the device definition to know what to place.

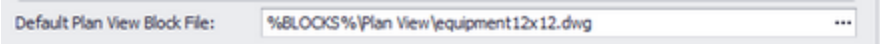


It relies on Location Boundaries in the drawing to know where to place it.



If the Plan View File is blank the Project Default Plan View File will be placed if the Location Boundary is found:

Application Menu > Settings [Project][Basic][Default Plan View Block File]



Prerequisites

- 1.SysNamed equipment preferably with an associated Plan View File in the global Equipment Library.
- 2.Location Boundaries placed in the drawing.

Operation

- 1.Start the tool.
- 2.A message box will explain the operation of the function. Click **[OK]**. The tool will run.
- 3.At completion the tool will report the number of placed items.

Related Topics:

None

5.2.4 Data Dialogs

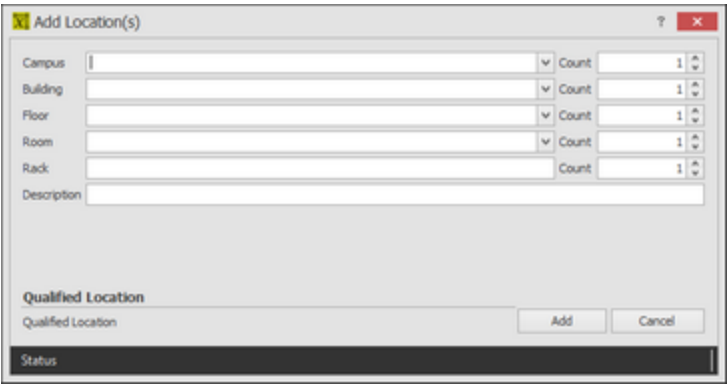
Navigation: [Reference](#)  > [Dialogs](#)  >



The following is a collection of dialogs that may be presented while in the Data or Grid environment.

5.2.4.1 New Location

Navigation: [Reference](#)  > [Dialogs](#)  > [Data Dialogs](#)  >



Database > Locations [New]

Commandline: none

Explanation

This tool is used to create entries in the project Locations look-up table.


Prerequisites

1. None

Operation

1. Start the tool.
2. Enter the data in the fields as desired.
3. Verify the **Qualified Location** looks acceptable (its the part that the rest of WireCAD uses).
4. Click **[Add]**.

Related Topics:

[Locations Look-up table](#)  537

[Locations form reference](#)  537

[Defining Locations](#)  101

Dialog Options

- **Campus** - Enter the **Campus** name if applicable. If there are multiple numbered campuses you can increment the **Count** field to append a number and create multiple records for each.
- **Building** - Enter the **Building** name if applicable. If there are multiple numbered buildings you can increment the **Count** field to append a number and create multiple records for each.
- **Floor** - Enter the **Floor** name if applicable. If there are multiple numbered **Floors** you can increment the **Count** field to append a number and create multiple records for each.
- **Room** - Enter the **Room** name if applicable. If there are multiple numbered **Rooms** you can increment the **Count** field to append a number and create multiple records for each.
- **Rack** - Enter the **Rack** name if applicable. If there are multiple numbered **Racks** you can increment the **Count** field to append a number and create multiple records for each.
- **Qualified Location** - This is the important part. The **Qualified Location** is a concatenation of all the used fields. The fields are merely and organizational construct for you. WireCAD will use the **Qualified Location** throughout the application.

Example

In the following example we will add two locations using different approaches to achieve the same result.

- **Enter the data across all fields:**

The screenshot shows the 'Add Location(s)' dialog box. It has a title bar with a yellow 'X' icon, a question mark, and a close button. The dialog contains several input fields: 'Campus' (LA), 'Building' (THE FACTORY), 'Floor' (MAIN), 'Room' (MR), and 'Rack' (RK-10). Each of these fields has a 'Count' field next to it, all set to 1. There is also a 'Description' field. Below these fields, the 'Qualified Location' field is highlighted with a red box and contains the text 'LA.THE FACTORY.MAIN.MR.RK-10'. To the right of this field are 'Add' and 'Cancel' buttons. At the bottom of the dialog is a 'Status' bar.

•Enter the data in a single field:

This screenshot is identical to the one above, showing the 'Add Location(s)' dialog box with the 'Qualified Location' field highlighted by a red box. The field contains the text 'LA.THE FACTORY.MAIN.MR.RK-10'.

As you can see from the above examples the Qualified Location is the same for both. It does not matter which approach you take.

5.2.4.2 New Cable

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁸ > [Data Dialogs](#) ⁴⁸¹ >



Database > Cables
Commandline: cg

Explanation

This tool allows you to manually create a Cable in the Cables table.

Prerequisites

1.SysNamed source and destination equipment in the **Equipment List**.

Operation

- 1.Start the tool.
- 2.Select the source SysNamed equipment.
- 3.Select the destination SysNamed equipment.
- 4.Click the [...] button on the Cable No field to generate a Cable Number.

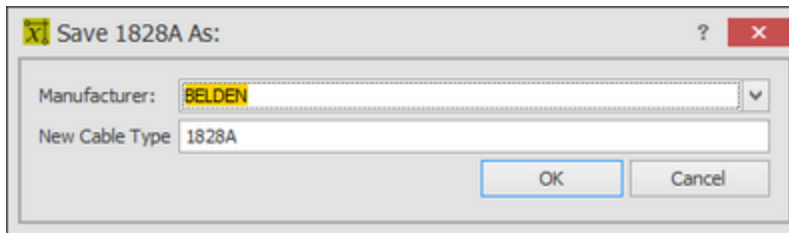
Dialog Options

- Cable Type**
- Source Information** - Set the source info.
- Destination Information** - Set the destination info.
- Cable No** - Click the [...] button to generate a cable number.

5.2.4.3 Save Cable Type As

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Data Dialogs](#) ⁴⁸¹ >





Database > Cable Type - File > Save As
Commandline: showcabletypesgrid

Explanation

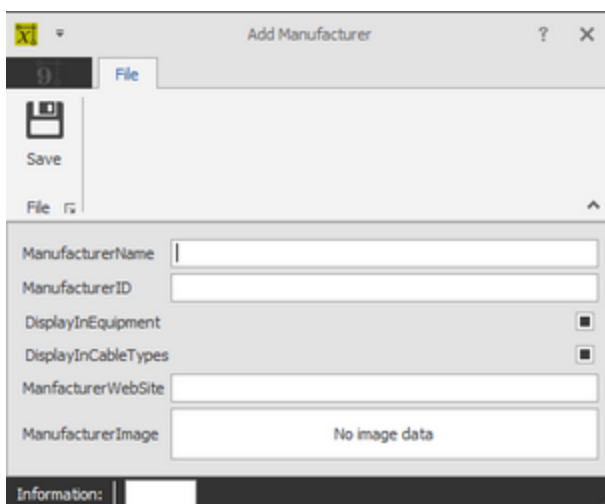
Create another entry in the **CableTypes** database record by copying the source record into a new destination record with a new **Manufacturer** and **Cable Type**.

Controls

- **Manufacturer** - Select a **Manufacturer**.
- **New Cable Type** - Name the new **Cable Type**.

5.2.4.4 New Manufacturer

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Data Dialogs](#) ^[481] >



Database > Manufacturers [File > New]
Commandline: none

Explanation

This tool allows you to create a new **Manufacturer** in the **Global Equipment** database.

Dialog Options

- **Manufacturer Name**
- **ManufacturerID** - Should match the **Manufacturer Name** field.
- **DisplayInEquipment** - Show this Manufacturer in the **Equipment Library**.
- **DisplayInCableTypes** - Show this Manufacturer in the **Cable Types** grid.
- **Website**
- **Image** - Right-click and select **Load** from the context menu to add a **Manufacturer** logo.

5.2.4.5 New Equipment Wizard

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[328] > [Data Dialogs](#) ^[481] >



Database > Equipment [File > New]

Commandline: none

Explanation

This tool allows you to create a new **Equipment** definition in the **Global Equipment** database.

Related Topics:

[How to Add Equipment](#) ^[65]

Dialog Options

- **Manufacturer** - Select the Manufacturer. Clicking the [...] button opens the [New Manufacturer](#) ⁴⁸⁵ dialog.
- **EquipmentName/Model/Part Number** - Name it.
- **Description** - Describe it.
- **SysName Prefix** - Be brief.
- **Front Panel File** - Pointer to the front panel file. Must reside in the %BLOCKS% support path.
- **Plan View File** - Pointer to the Plan View file. Must reside in the %BLOCKS% support path.
- **Image** - Right-click and select **Load** from the context menu to add a image.
- **Categories, Synonyms** - This is for you. Create categories and synonyms.
- **Abbreviation, Accessory Of** - Not really used yet.

5.2.4.6 New Cable Type

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Data Dialogs](#) ⁴⁸¹ >



Database > Cable Types [File > New]
Commandline: showcabletypesgrid

Explanation

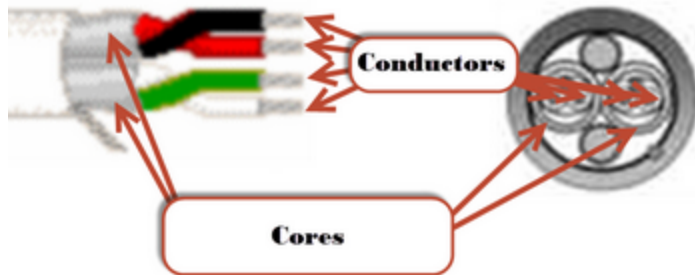
This tool adds a new **Cable Type** to the **Global Equipment** database. **Cable Types** may be either single or multi-core and have different shielding configurations. WireCAD projects use the multi-core core color code if any as the descriptor when identifying multi-core cables. The **Cable Type** Name and Manufacturer are used in the Cable record. All other fields are maintained for reference of the designing engineer.

More About Multi-core Cable Types

WireCAD can create multi-core cable types at two different levels of detail:

- Core level.
- Conductor level.

Cores have conductors. Conductors being the base unit. Most people using WireCAD will not document down to the conductor level but rather the Core level.



Prerequisites

- 1.None

Related Topics:

[Create a New Cable Type](#) 

Dialog Options

•Save

- ManufacturerID** - Select a manufacturer. Only **Manufacturers** that are flagged as **DisplayInCabletypes** appear in this list.
- Cable Type or P/N** - The name by which this cable type shall be known.
- Description** - A description for posterity.
- Cable Char Z** - Characteristic Impedance.
- Cable OD** - Outside Diameter.
- Cable Gauge** - Gauge.
- Cable Rating** - Rating.
- Cable Weight, Standard Length** - Standard put up length and weight.

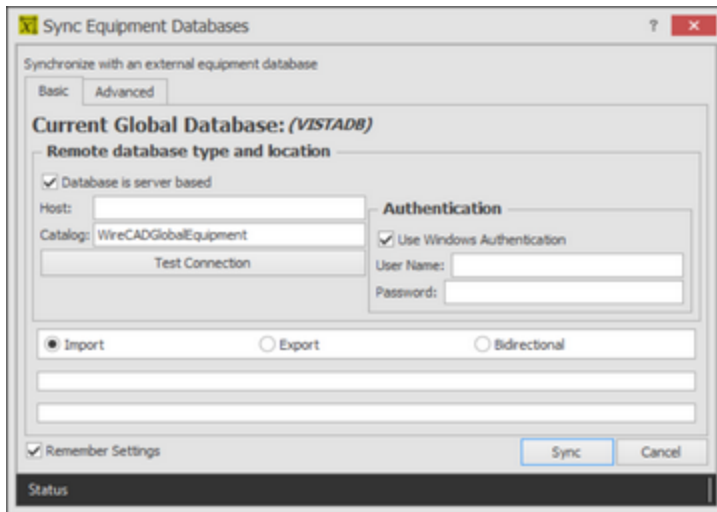
Core/Conductor Configuration

- Shielding** - Shielding determines how many conductors.
- **Is Multi-core** - Flag this Cable Type as multi-core.
- Core Count** - How many cores.
- Conductors per Core** - Usually one.
- Conductor Count Including Shield(s)** - Set automatically based on the core/conductor counts and the **Shielding** configuration.
- Default Core/Fiber Mode** - SM/MM.
- Color Code Applies To** - Apply a color code to the conductors, cores or not.

5.2.4.7 Synchronize Global Databases

Navigation: [Reference](#)  > [Dialogs](#)  > [Data Dialogs](#)  >





Database > Sync Global Database ..
Commandline: sync

Explanation

This tool allows you to sync two **Global Databases**. The current **Global Database** will be synchronized with the database you select in this dialog. The local and remote database may be either SQL Server or VISTADB. You may import, export or sync with respect to the **current Global Database**. An Import would collect data from the remote database and import it to the local Global Database.

Prerequisites

1. Two Global Databases. The currently active one and a remote one.

Related Topics:

[Sync Basics](#) ¹⁸⁴

Dialog Options

•External Database Location

•**Database is Server Based** - Enter the host information and login. Otherwise browse to the location of the WireCADGlobalEquipment.vdb3 file.

•**Import** - With respect to the current **Global Database**. This would import data into the current **Global Database**.

•**Export** - With respect to the current **Global Database**. This would export data into the external **Global Database**.

•**Bidirectional** - Sync the two.

Advanced Settings

•**Which collections to include** - By default you would want to sync all collections.

5.2.4.8 Combine Circuits (CMS Only)

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Data Dialogs](#) [481] >



Menu: **Database > Circuits**

Default command line shortcut: **cmscg**

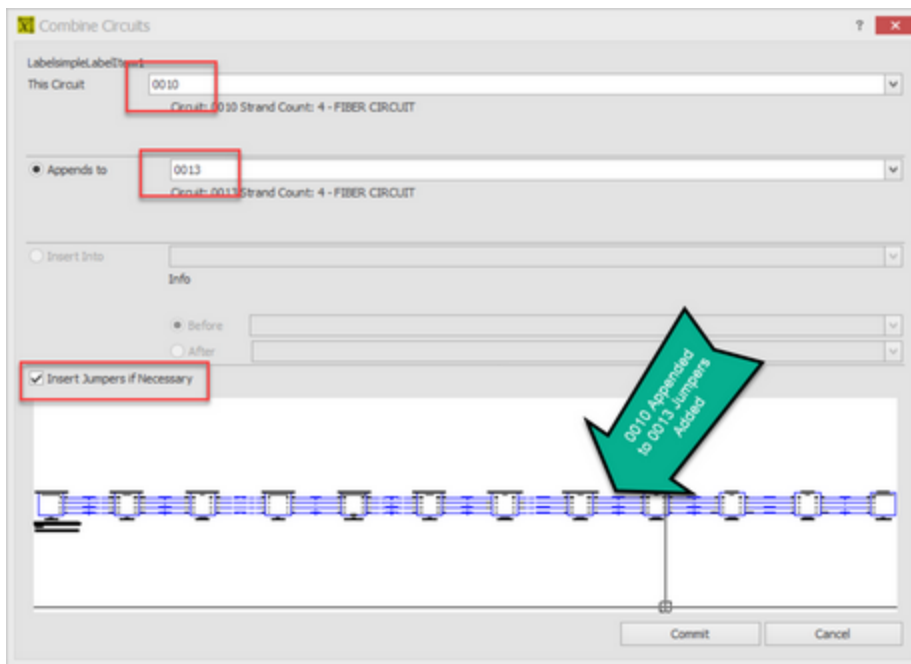
Related Project Settings:

Circuit Status Items

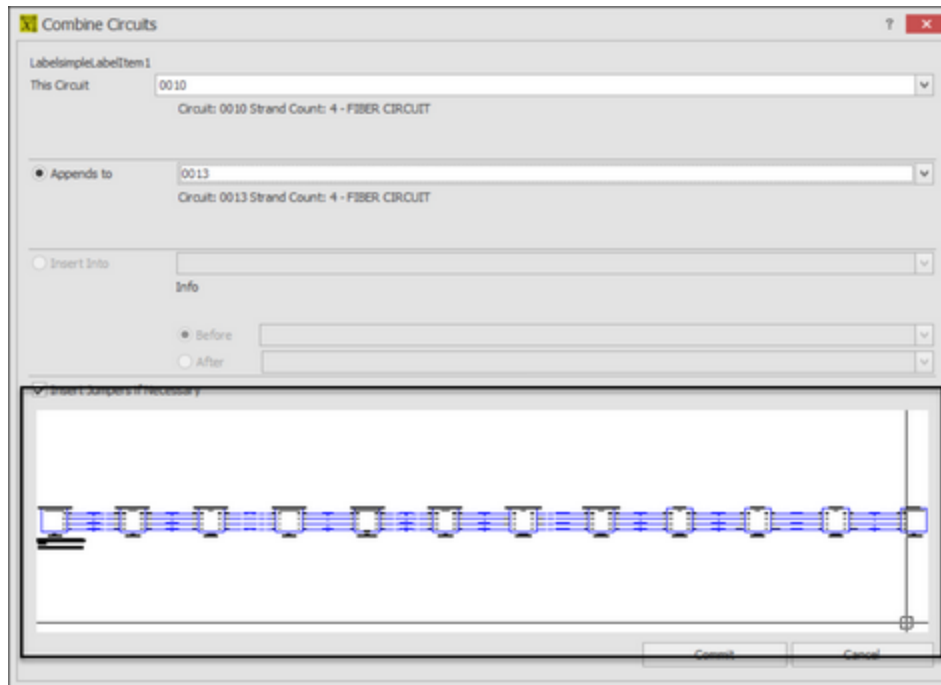
Product Level: **CMS**

Click **Tools > Combine Circuits**.

WireCAD CMS tools support the concept of combining circuits. Circuits to be combined must have the same Strand Count. The Circuit Name/Number of the Circuit that is being appended or inserted will be discarded.

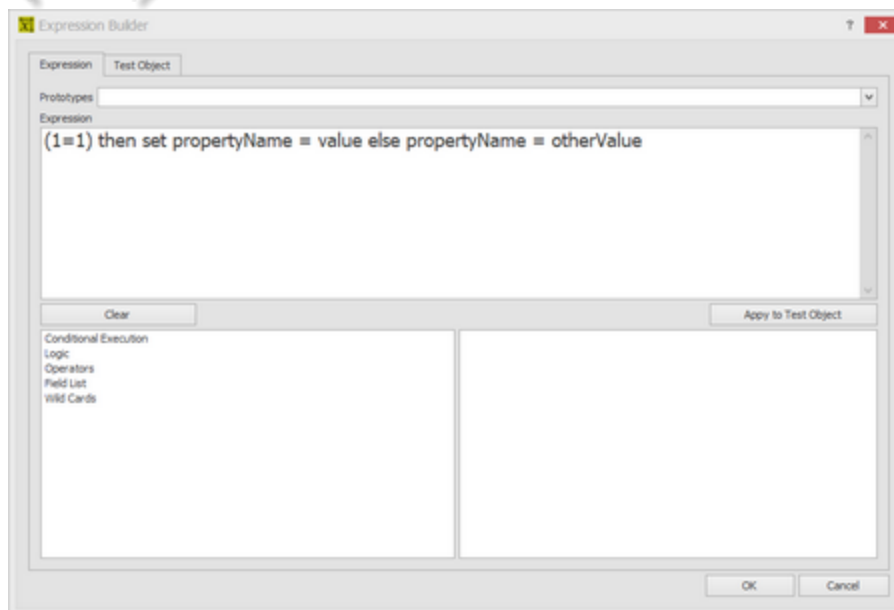


- **This Circuit** - The Circuit Name/Number of the circuit that will be appended or inserted into another circuit. This circuit Name/Number will be discarded if successfully appended or inserted.
- **Appends To** - The Circuit Name/Number of the circuit that will be appended to.
- **Inserts Into** - The Circuit Name/Number of the circuit that will be inserted into.
- **Before** - Before this SysName.
- **After** - After this SysName.
- **Insert Jumpers if Necessary** - It is possible to create collections of cables in a circuit that do not interconnect. If you do not select this option you will create collection of cables that do not interconnect.
- **Preview**



5.2.4.9 Expression Builder

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Data Dialogs](#) ⁴⁸¹ >



Database > Any
Commandline: varies

Explanation

This tool allows you to manipulate data using the Expression Engine.

For more information on [Expression Syntax](#)^[204].

Expression Tab

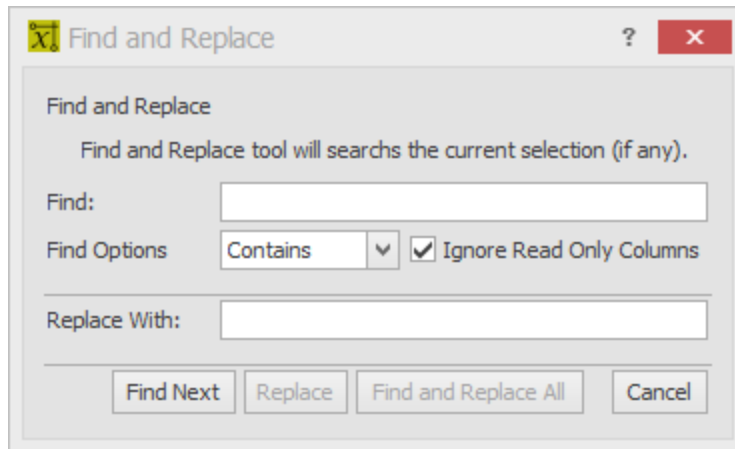
- Prototypes** - Select from a number of pre-formatted expressions. Then replace the variables as appropriate.
- Expression** - Enter your expression to evaluate here.
- [Clear]** - Clears the Expression.
- [Apply to Test Object]** - A prototype collection object is loaded into the property window of the **Test Object** tab. Clicking this button applies the **Expression** to the test object data. You will want to edit the **Test Object** data before executing this command so that you have a meaningful test.
- Left List** - Variables, operators and fields to apply to the expression.
- Right List** - Detail variables, operators and fields. Double-clicking a item here will insert it into the **Expression** window at the caret.
- [OK]** - Do it to the data in the grid.

Test Object Tab

- Property Window** - Displays a prototype of the grid collection item. Edit this to have test data on which to apply your **Expression** when the **[Apply to Test Object]** button is clicked.

5.2.4.10 Find and Replace

Navigation: [Reference](#)^[288] > [Dialogs](#)^[326] > [Data Dialogs](#)^[481] >



Database > Any
Commandline: varies

Explanation

This tool allows you to find and replace string within the open grid.

Searches the current selection if any. If not, searches the whole grid.

Related Topics:

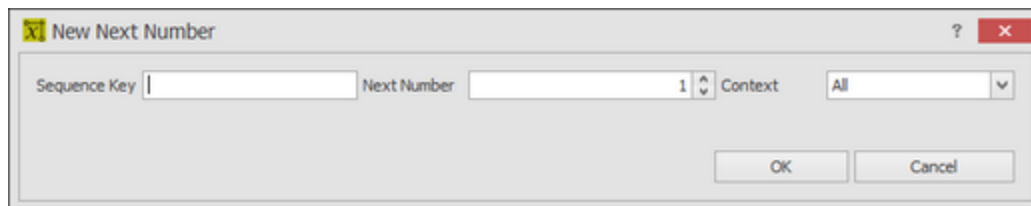
[Grid Basics](#) 49

Controls

- **Find** - Search Text.
- **Find Options** -
 - Contains** - Any or whole Find string exists.
 - MatchCase** - Any or whole Find string matches case sensitive.
 - Exact** - Whole string matches exactly.
 - **Ignore Read Only Columns** - Searches only columns that can be edited.
 - **Replace With** - Replace the found part with this.
 - **Find** - Search and highlight the first found instance.
 - **Replace** - Do the replace the found instance.
 - **Find and Replace All** - Do it to all found instances.

5.2.4.11 New Next Number

Navigation: [Reference](#) 288 > [Dialogs](#) 326 > [Data Dialogs](#) 481 >



Database > Next Numbers - File > New
Commandline: nn

Explanation

Add a new Next Number item. Next Numbers are normally added automatically but if you need to take control and add your own.

Next Numbers use a Sequence Key and an integer within a context.

Example

Say our cable number is V-1001. V is the **Sequence Key** and 1001 is the integer **Next Number** in the **Context** of Cables.

or:

Our SysName is SRVR-101. SRVR becomes the **Sequence Key** and 101 the **Next Number** in the **Context** of SysNames.

Related Topics:

[Grid Basics](#) 49

[Next Numbers](#) 114

Controls

- Sequence Key - What is the part we are sequencing on.
- Next Number - Integer next number.
- Context -
All
Backbones
Circuits
Cables
SysNames

5.2.4.12 Attach Document

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Data Dialogs](#) ⁴⁸¹ >



Database > Most
Commandline: varies

Explanation

Many of the grids in WireCAD support Attached Documents. The document can be of any file type. The document is stored in the database as a Binary Large Object (BLOB). Attached document can be launched from the grid by clicking the hyperlink. The document will be opened by the system reader of the stored file type.

AD-01		Location	Elevation	MSTCAM 1...	BLACKMAGIC	TERRANEX		
Documents Collection								
File Name	Description	DocumentSize	CreatedBy	CreatedOn	ModifiedOn			
winwin.jpg	Some Descriptive Text	112 KB	chris_000	12/16/2016	12/16/2016			

Related Topics:

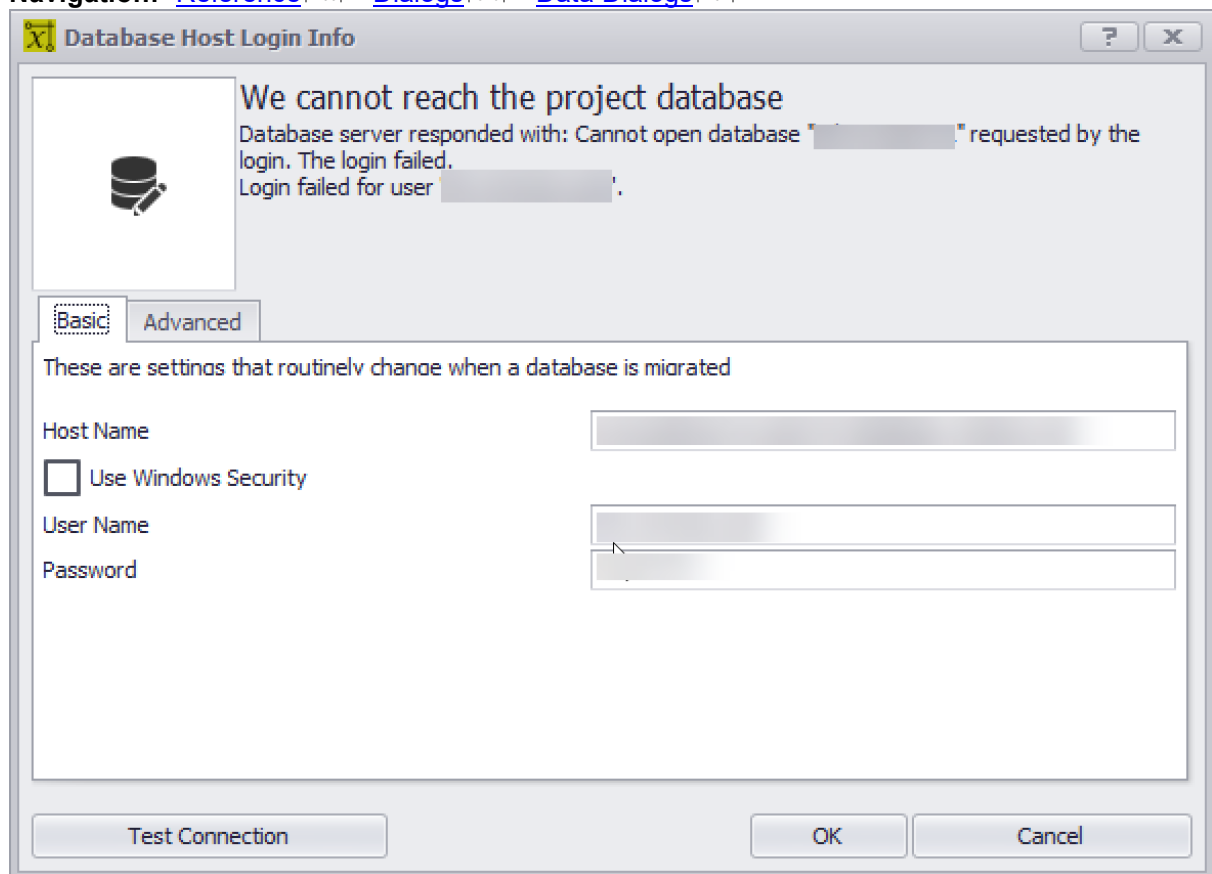
- [Grid Basics](#) ⁴⁹
- [Next Numbers](#) ¹¹⁴

Controls

- File** - Browse to the file to attach by clicking the [...] button.
- Description** - More words.

5.2.4.13 Database Host Settings

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Data Dialogs](#) [481] >



Project > Open Project Fails
Commandline: op

Explanation

The only reason you will see this dialog is that you have attempted to open a SQL project where the database server is unreachable or has actively refused our connection. Here you have a chance to fix **Host** and **User** login Info.

Basic Info [TAB]

Host Name - The fully qualified path to your SQL Server. In some cases your configuration will require you to append the instance name. This can be something like: MyHost\SQLEXPRESS where MyHost is the host name and SQLEXPRESS is the instance.

Use Windows Security - Sets whether you will connect with Windows Security. Doing so disables the User Name and Password fields as Windows takes over.

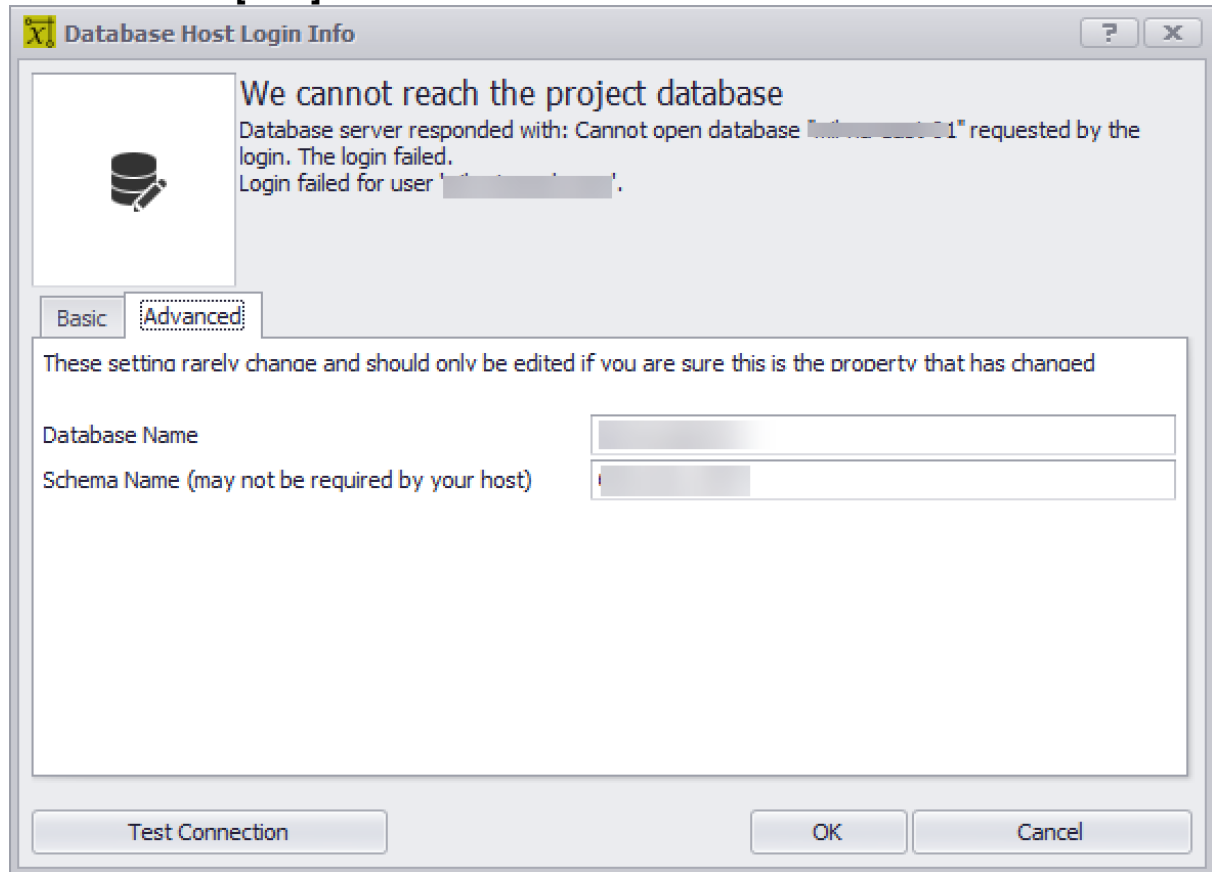
UserName - Determined by the database administrator and should have been provided to you.

Password - Determined by the database administrator and should have been provided to you.

If you are the database administrator you will need to create a Login and map that login to the various databases.

It is beyond the scope of this manual to discuss database administration.

Advanced Info [TAB]



Database Name - This is the name of the database to which you are connecting on the server. If you are using a cloud service it may be the same for all your projects. Normally it is the project name with spaces replaced by underscores.

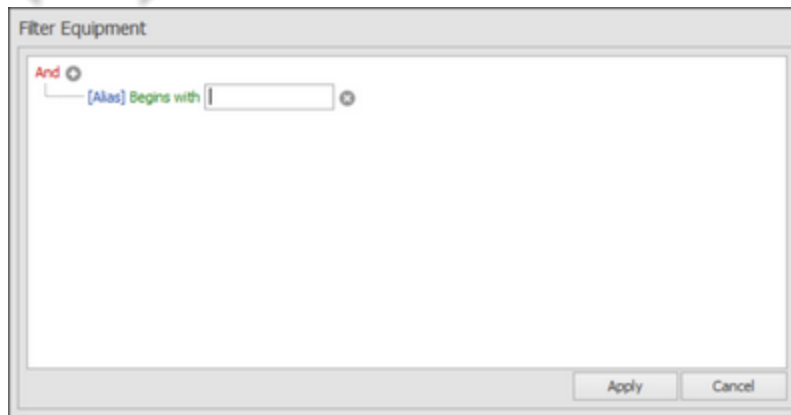
Schema Name - This is normally empty or dbo. If you are using a cloud service like Azure SQL it will be the project name with spaces replaced by underscores.

•**[Test Connection]** - make sure that this returns that you can successfully connect to the database before clicking **[OK]**.

Related Topics:

5.2.4.14 Filter Editor

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Data Dialogs](#) ⁴⁸¹ >



Various

Commandline: none

Explanation

The filter editor allows you to create filter criteria base on the collection fields combined with a logical operator and a value.

Click the [+] button to add another complex condition.

Related Topic

[BOM Generator](#) ⁵⁵⁶

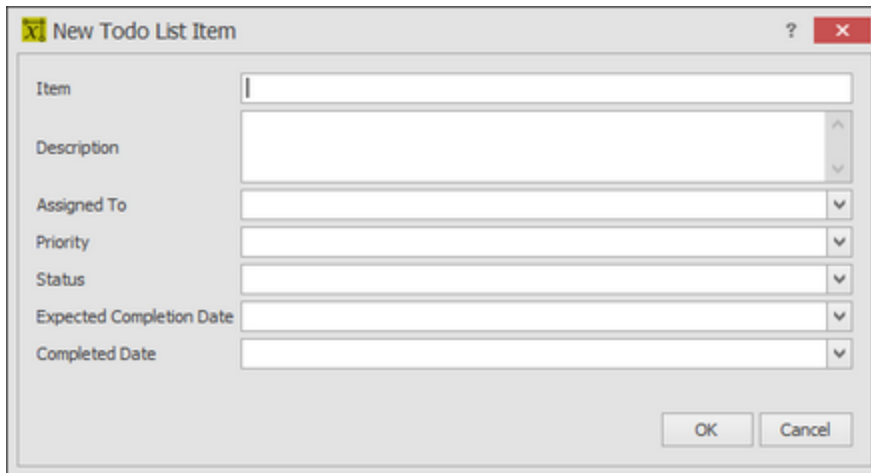
[AutoBlock](#) ¹²⁰

[How To: Filter Reports](#) ¹⁶⁵

5.2.4.15 New Todo

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Data Dialogs](#) ⁴⁸¹ >





Database > Project > Todo - File > New
Database > Project > Equipment List - Todo
Database > Project > Cables - Todo
Database > Project > Drawings - Todo
Commandline: varies

Explanation

Create a new Todo List item. When launched from the following grids it will associate the Todo List item with the selected record(s):

- Equipment List.
- Cables.
- Drawings.

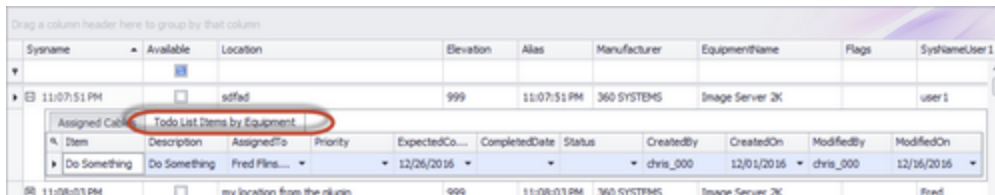
Related Topics:

[Todo List](#) 354 [Settings](#)
[Todo List Grid](#) 539
[Grid Basics](#) 49

Controls

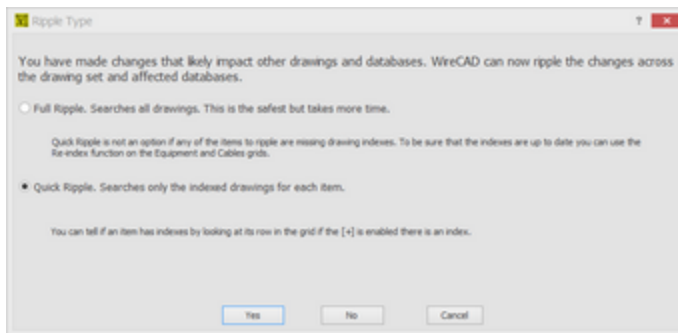
- Item** - Words.
- Description** - More words.
- Assigned To** - Who is to do it.
- Priority** - Set the urgency.
- Status** - Where are we at in the process.
- Expected Completion Date** - When are we expecting this done by.
- Completed Date** - When did we get it done.

You can see the Todo List items in the Todo List grid and in the associated grids:



5.2.4.16 Ripple Choice

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Data Dialogs](#) [481] >



Post Save - Rackbuilder Drawings, Equipment List, Cables table, Locations table
Commandline: varies

Explanation

Something has changed that potentially effects data in other drawings and databases. This is your chance to update those other entities in other places.

Controls

- **Full Ripple** - Search all drawings for the changed items and update.
- **Quick Ripple** - Search only indexed drawings. If the indexes are up to date this can significantly reduce ripple time.
- **Yes** - Do it!
- **No** - Don't Do It!
- **Cancel** - Do nothing.

Related Topics:

[Save and Ripple](#) [140]

5.2.4.17 Add/Edit Via (CMS Only)

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Data Dialogs](#) [481] >



Adding vias to 5 cable(s)

Idx

Name

Description

Type

Location

Elevation

Ports

OK

Cancel

Database > Backbones then Tools > New Via
Commandline: none

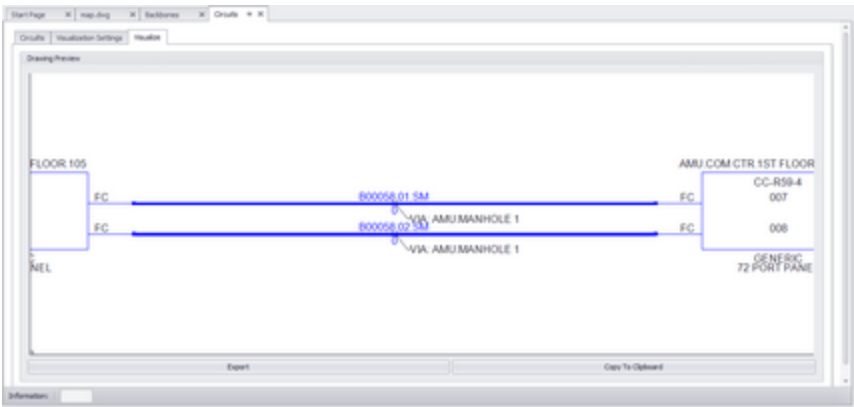
Explanation

WireCAD ENT supports the concept of **VIAS**. Vias are waypoints or appearances of **Backbones** as they make their way from starting point to ending point. **VIAS** can be used to list all of the pullboxes or manholes that a backbone appears in. They can also be used to indicate splice points.

VIAS are an ordered list that is attached to the **Backbone**. The **Idx** field determines list order.

CableInfoPrefix: B00065											
0	GENERIC	24 STRAND SM FIBER	B00065.01	P2B-09	ACM2-16		073	025	AMU,ANNE...	AMU,BLDG 122,ROOM 1060	SCA
1	GENERIC	24 STRAND SM FIBER	B00065.02	P2B-09	ACM2-16		074	026	AMU,ANNE...	AMU,BLDG 122,ROOM 1060	SCA
Vias Collection By FK Cables											
Idx	Name	Description	Type	GPS Lat	GPS Lon	GPSE	Location				
0							AMU,MANHOLE 1				
1							AMU,MANHOLE 2				
0	GENERIC	24 STRAND SM FIBER	B00065.03	P2B-09	ACM2-16		075	027	AMU,ANNE...	AMU,BLDG 122,ROOM 1060	SCA

VIAS can be shown in the the following visualizations:



VIAS shown in the Circuits visualizer

[Circuits Visualization Settings](#) ^[230]
[Backbone Display](#) ^[319] in Calibrated Drawing

Dialog Options

All of the following are optional except the Idx field.

- **Idx** - the position in the list of VIAS.
- **Name** - Optional name of the VIA.
- **Description** - Optional description of the VIA.
- **Type** - Optional type of the VIA.
- **Location** - Optional location of the VIA.
- **Elevation** - Optional elevation of the VIA.
- **Ports** - Optional ports of the VIA.

5.2.4.18 New Blind Patch

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Data Dialogs](#) ^[481] >



File > New from the Blind Patches Tool

Commandline: none

Explanation

Used to select the FromSys value in the Blind Patches tool. Select either a named piece of equipment or create a new CFE piece of equipment with a Manufacturer and Model name.

NOTE: CFE equipment will not appear in the Equipment List.

Prerequisites

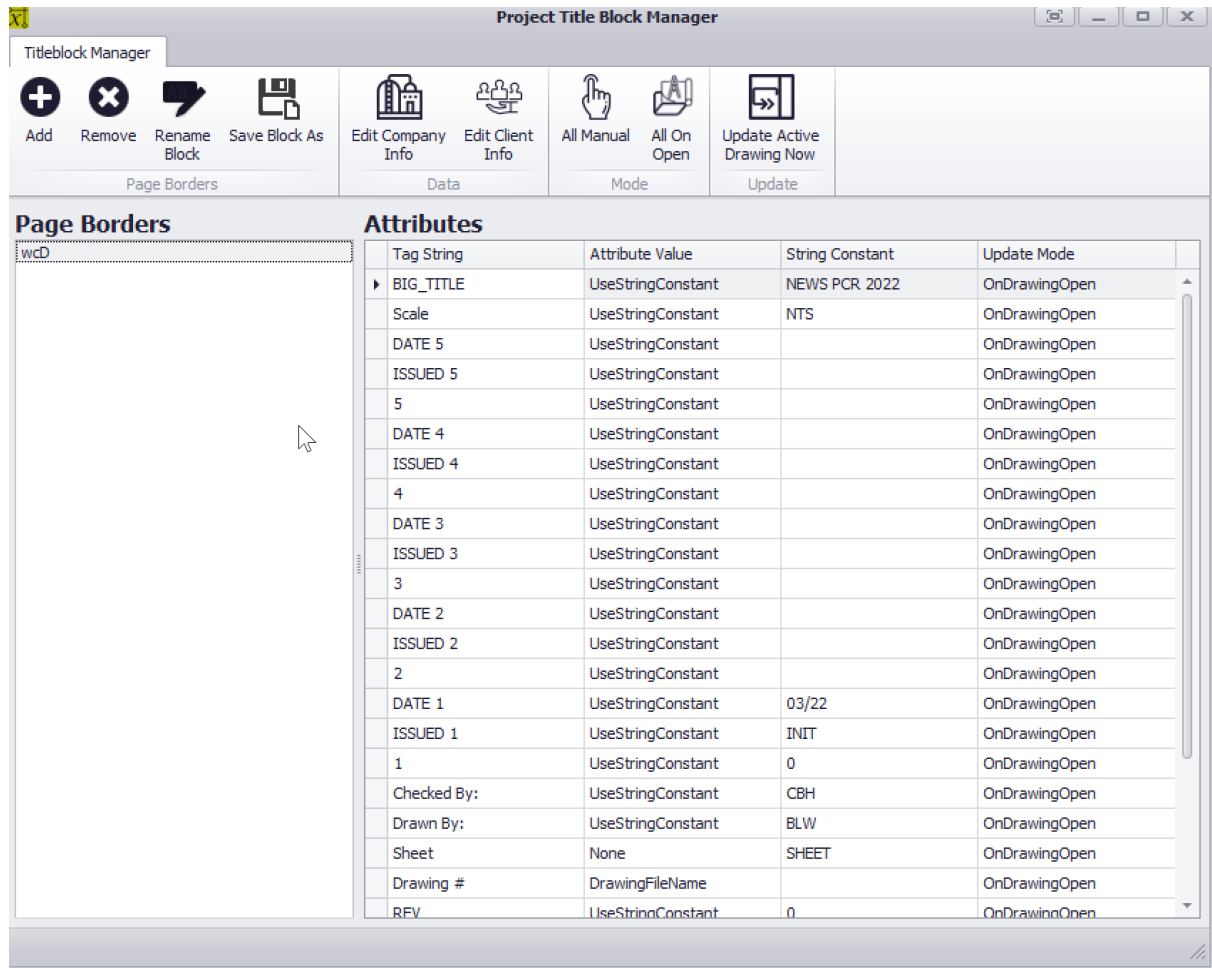
1. At least two Backbones in the database.
2. You must first select a ToSys from the dropdown.
3. Select the row to add a Blind Patch to or if **Duplex Jumper** is selected click the top row.

Related Topics:

[Blind Patch Tool Explanation](#) ^[226]
[How To: Create a Blind Patch](#) ^[227]
[How To: Discover Circuits from Blind Patches](#) ^[227]
[How To: Make a Real Circuit from Blind Patches](#) ^[228]

Blind Patch form

5.2.4.19 Titleblock Manager



Explanation

The Titleblock Manager tool associates the attributes of a titleblock with variables or string constants.

Menu Items

- [Add] - Add a titleblock to the list of Page Border blocks.
- [Remove] - Remove the selected block from the list of Page Border blocks.
- [Rename Block] - Rename the selected block in the list of Page Border blocks.
- [Save Block As] - Create a copy of the selected block with a new name in the list of Page Border blocks.
- [Edit Company Info] - Open the [Company Info](#)^[352] settings panel.
- [Edit Client Info] - Open the [Account Info](#)^[340] dialog.
- [All Manual] - Set the **Update Mode** for all attributes to Manual mode. In Manual mode the titleblock is only updated when you open the **TitleBlock Manager** dialog and click the **[Update Active Drawing Now]** button.
- [All on Open] - Set the **Update Mode** for all attributes to **OnDrawingOpen** mode. In **OnDrawingOpen** mode the attributes that are flagged as **OnDrawingOpen** will be updated when the drawing is opened.
- [Update Active Drawing Now] - Do it!

Variables

Most of these are self explanatory. Those that are not will be defined.

- None - Do nothing
- ProjectName
- ProjectDescription
- ProjectLeadPerson
- ProjectInfoUser1 - User1 field from the Application Menu>Project Info form.
- ProjectInfoUser2 - User2 field from the Application Menu>Project Info form.
- ProjectInfoUser3 - User3 field from the Application Menu>Project Info form.
- ProjectInfoUser4 - User4 field from the Application Menu>Project Info form.
- ProjectPath
- ProjectCreationDate
- ProjectModifiedDate
- CurrentProjectRevision

See this [form](#)^[540].

- DrawingName
- DrawingFileName - file name with extension.
- DrawingFileNameAndPath
- DrawingLayoutName - the layout where the Page Border lives.
- DrawingCreationDate
- DrawingModifiedDate
- DrawingUser1 - User1 from the Drawings table.
- DrawingUser2
- DrawingUser3
- DrawingUser4
- DrawingUser5
- DrawingUser6
- DrawingUser7
- DrawingUser8
- DrawingUser9
- DrawingUser10

- DrawingUser11
- DrawingUser12
- DrawingShort - the Short description from the Drawings table.
- DrawingLong - the Long description from the Drawings table.
- DrawingFSCM
- DrawingNumber

Date and Time

- DateTimeNow - The current date and time as right now.
- DateShort - The short date as of now.
- DateLong - The long date as of now.

Environment

- UserName - The environment User Name.

Strings

- UseStringConstant - Use the data from the **String Constant** field.

See this [settings panel](#) ^[352].

- CompanyName - Company Name from the Application Settings[Application]>Organization settings panel.
- CompanyAddress1 - Address1 from the Application Settings[Application]>Organization settings panel.
- CompanyAddress2 - Address2 from the Application Settings[Application]>Organization settings panel.
- CompanyCity - City from the Application Settings[Application]>Organization settings panel.
- CompanyState - State from the Application Settings[Application]>Organization settings panel.
- CompanyPostalCode - Zipcode from the Application Settings[Application]>Organization settings panel.
- CompanyPhone - Company Name from the Application Settings[Application]>Organization settings panel.
- CompanyWebsite - Company Name from the Application Settings[Application]>Organization settings panel.
- CompanyCountry - Company Name from the Application Settings[Application]>Organization settings panel.

See this [dialog](#) ^[340].

- ClientAccountName - Account Name from the Application Menu>Account Info dialog.
- ClientContactName - Contact Name from the Application Menu>Account Info dialog.
- ClientAddress1 - Address1 from the Application Menu>Account Info dialog.
- ClientAddress2 - Address2 from the Application Menu>Account Info dialog.
- ClientCity - City from the Application Menu>Account Info dialog.
- ClientState - State from the Application Menu>Account Info dialog.
- ClientPostalCode - Zipcode from the Application Menu>Account Info dialog.
- ClientPhone - Phone from the Application Menu>Account Info dialog.
- ClientFax - Fax from the Application Menu>Account Info dialog.
- ClientWebsite - Website from the Application Menu>Account Info dialog.
- ClientEmail - Email from the Application Menu>Account Info dialog.
- ClientInfoUser1 - User1 from the Application Menu>Account Info dialog.
- ClientInfoUser2 - User2 from the Application Menu>Account Info dialog.
- ClientInfoUser3 - User3 from the Application Menu>Account Info dialog.
- ClientInfoUser4 - User4 from the Application Menu>Account Info dialog.

5.2.5 Report Dialogs

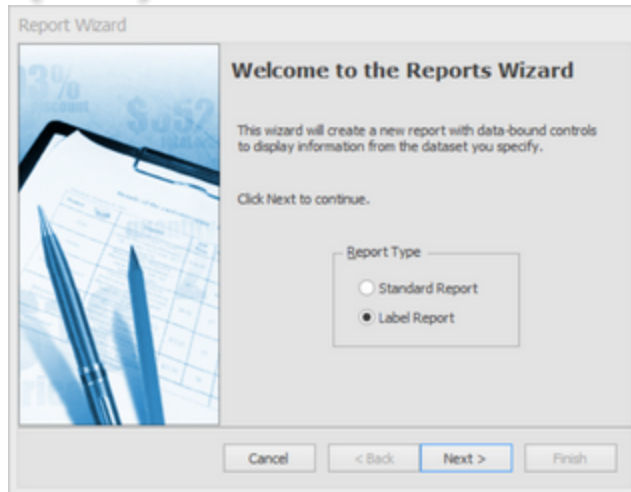
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[328] >



Dialogs associated with report designer.

5.2.5.1 New Report Wizard

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Report Dialogs](#) ⁵⁰⁵ >



Reports > New with Wizard ...

Commandline: rw

Explanation

Create a new report using the New Report Wizard.

NOTE: We recommend that you find an existing report that is close to what you are looking for and modify that after saving it with your name. It will save you time and effort.

Prerequisites

1. Open Project

Operation

1. Start the tool.
2. Step through the wizard
3. Finish the wizard.
4. Edit the report in the designer to finalize it.

Related Topics:

[Creating Reports](#) ¹⁷⁰

5.2.5.2 Scan for Discrepancies

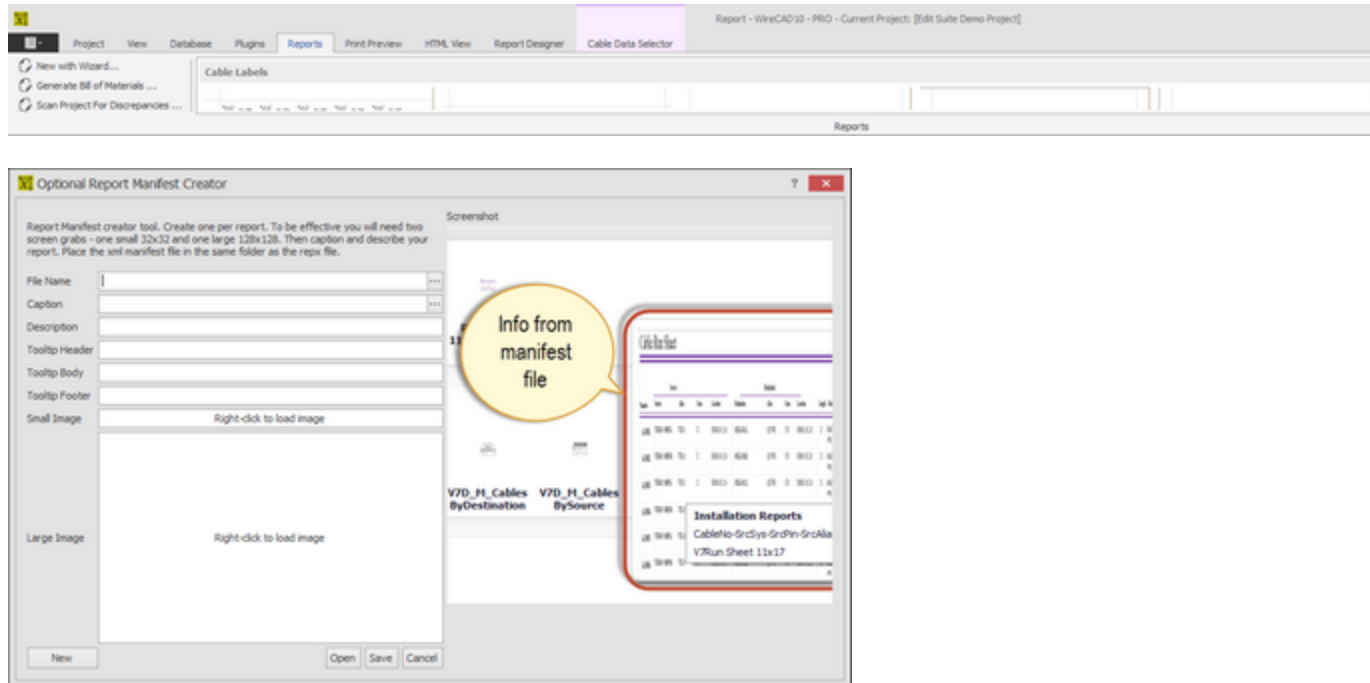
Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Report Dialogs](#) ⁵⁰⁵ >



See the section on [Index Grids](#) ⁵⁴¹

5.2.5.3 Report Manifest Editor

Navigation: [Reference](#) [288] > [Dialogs](#) [326] > [Report Dialogs](#) [505] >



Report > Smart Tag
Commandline: none

Explanation

If your Report Show Mode = Gallery the gallery items will display:

- Preview bitmap.
- Large bitmap when hovering over the gallery item.
- Caption text.
- Descriptive text.
- Tooltip text.

When the gallery loads it searches for a manifest file with the same name as the report file.

Example:

MyReport.repx - report file.

MyReport.xml - manifest file.

Make sure that the manifest file name matches the report name and that they are in the same directory.

If the manifest file does not exist the stock bitmap is shown and the file name is used as a caption.

Prerequisites

A custom report.

Related Topics:

[Report Show Mode = Gallery](#) ³⁵⁴

- **File Name** - the name of the report repx file.
- **Caption** - gallery item caption.
- **Description** - gallery item description.
- **Tooltip** - text when hovering over the gallery item.
- **Small Image** -
- **Large Image** - displayed when the gallery is expanded and the item is hovered.

**5.2.6 Plugins Dialogs**

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ >

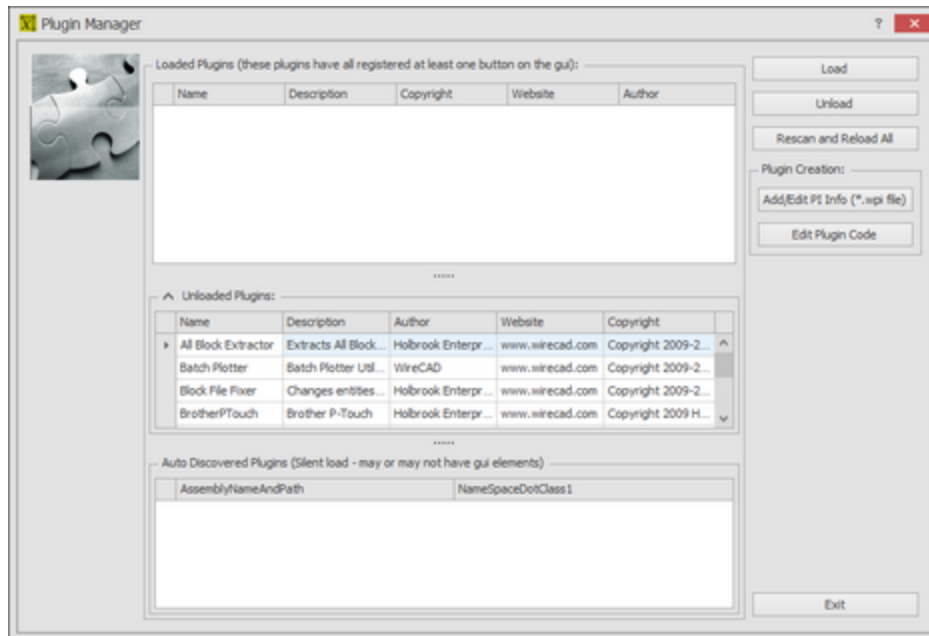


The following sections relate to plugins that register dialogs for editing. These are plugins included with WireCAD. Other plugins may register dialogs that do not appear here.

5.2.6.1 Plugin Manager

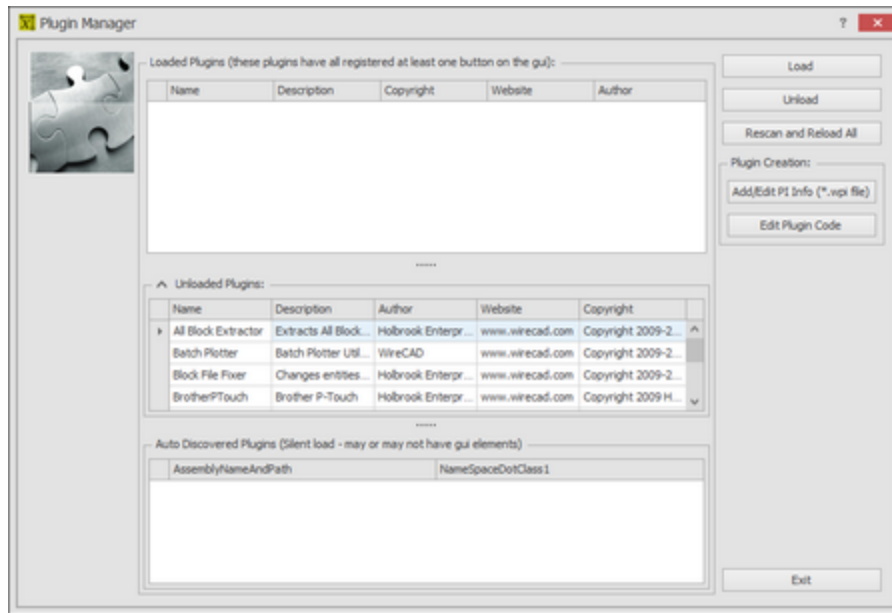
Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Plugins Dialogs](#) ⁵⁰⁸ >



**Plugins>Plugin Manager****Commandline: pim*****Explanation***

The WireCAD Plugin Manager shows:

- Loaded plugins.
- Unloaded plugins.
- Automatically Discovered plugins.



Only Enabled plugins are eligible to load/unload. Disabled plugins will not show here.

WireCAD Plugin Types

WireCAD has facility for two types of plugin:

1. Auto-discovered plugins. These plugins register themselves on application startup. They may or may not include a graphical user interface element.
2. Registered plugins. These plugins register themselves via a plugin manifest file located in the C:\users\public\documents\WireCAD\WireCAD9\Plugins\Active Plugins*.wpi. Registered plugins may place a button somewhere in the WireCAD workspace, register a corresponding Commandline shortcut, and may respond to WireCAD events.

Related Topics:

[Included Plugins](#) ¹⁹⁰

[WireCAD Software Development Kit \(SDK\)](#) ⁵⁷⁶

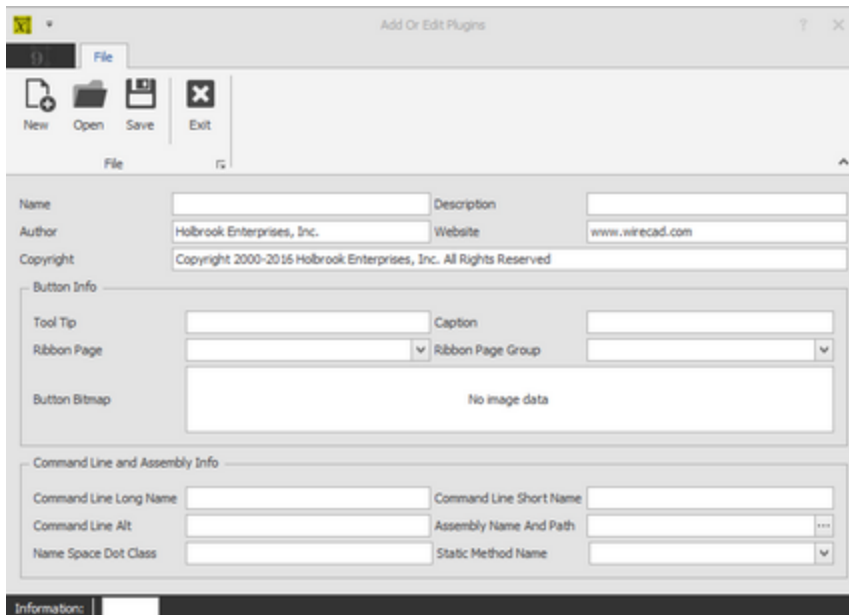
[Enable/Disable Plugins](#) ³⁵³

Dialog Options

- **Load** - Loads the selected plugin from the **Unloaded Plugins** grid. This function does not apply to the Automatically Discovered plugins.
- **Unload** - Unloads the selected plugin from the **Loaded Plugins** grid. This function does not apply to the Automatically Discovered plugins.
- **Rescan and Reload All** - Unloads then rescans and reloads all plugins. Applies to all plugins.

Plugin Creation

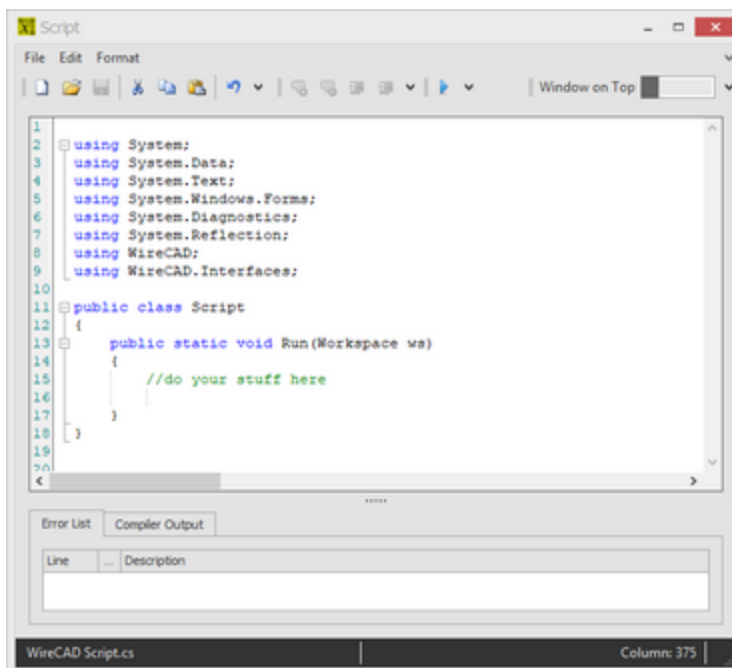
- **Add/Edit PI info (*.wpi)** - Show the Add or Edit Plugins dialog to work with WireCAD Plugin Info files (wpi).



•**Edit Plugin Code** - Open your IDE to edit code. You will be prompted to enter a path to the IDE exe file.

5.2.6.2 Script Editor/Runner

Navigation: [Reference](#) ²⁸⁸ > [Dialogs](#) ³²⁶ > [Plugins Dialogs](#) ⁵⁰⁸ >



Explanation

Often times there are operations that you find yourself repeating some thing or process endlessly. Scripts are a good way to automate those processes. There are many example scripts to browse through and see how they work. Example scripts can be found at:

c:\users\public\documents\WireCAD\WireCAD9\scripts*.cs

NOTE: Scripts that run in WireCAD MUST have the following method signature or they will not execute:

```
using System;
using System.Data;
using System.Text;
using System.Windows.Forms;
using System.Diagnostics;
using System.Reflection;
using WireCAD;
using WireCAD.Interfaces;
//You may add additional using statements as needed but the listed ones are the minimum.

public class Script
{
    public static void Run(Workspace ws)
    {
        //do your stuff here
    }
}
```

Possible Uses

- 1.Title block filling.
- 2.New Drawing creation.
- 3.Database cleanup.

Usages

- 1.Launch the tool.
- 2.**File>Open** an example script. (c:\users\public\documents\WireCAD\WireCADx\Scripts\).
- 3.Read the comments (they are the ones proceeded by //).
- 4.Tweak the script to suit your needs.
- 5.Run it by clicking the **[RUN]** button



- 6.Fix any compiler errors. Lather, rinse, repeat.

Dialog Options

- New** - Creates a new script with the necessary method signature.
- Open** - Opens a .cs file to run.
- Comment** - Comment out your selection. Useful for hiding code from the compiler without deleting it.
- Un Comment** - Un comment your selection.
- Indent, Out dent**

•**Run** - If the Error List is empty then Do It. If it fails it will tell you why in the Compiler Output window.



•**Window On Top** - Make this the topmost window always.

•**Error List** - Design time scripting syntax errors.

•**Compiler Output** - Run time compiler and script execution errors, as well as Trace messages.

5.2.7 PDF Viewer Dialogs

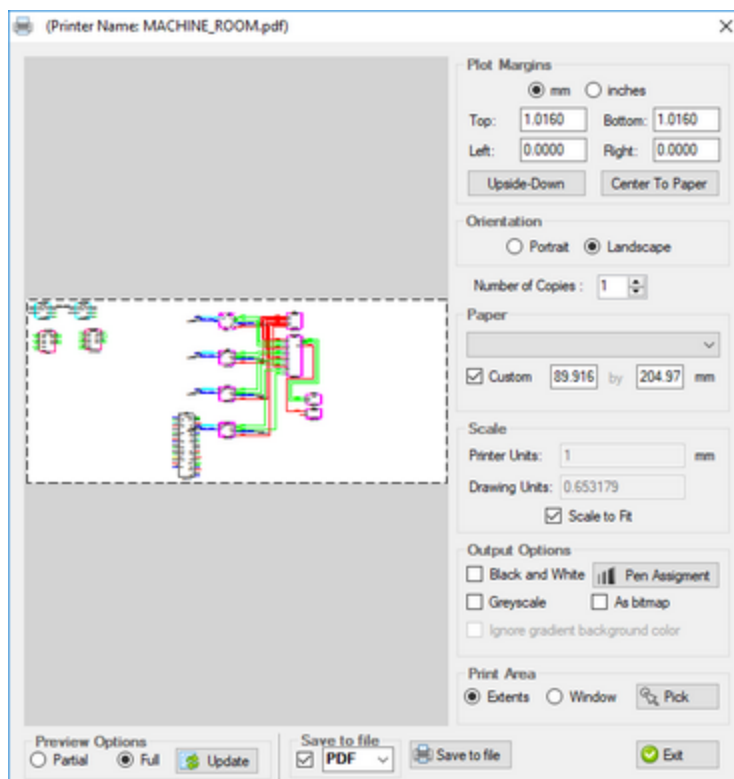
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] >



The following is dialogs associated with PDF Viewer.

5.2.7.1 PDF Print Preview

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [PDF Viewer Dialogs](#) ^[513] >



Project Explorer > Double-click PDF File [Print]
Commandline: None

Explanation

This is the **Print Preview Dialog** for the **PDF Viewer**. It's pretty standard so we won't enumerate all the controls here.

Prerequisites

- 1.At least one pdf file saved in the project drawings path.

5.2.8 Map Tool Dialogs

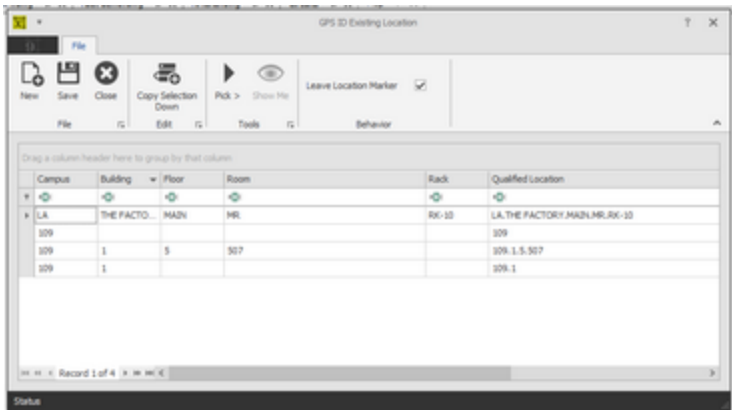
Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] >



For information on the Map tool see this [topic](#) ^[560].

5.2.8.1 ID GPS Locations

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Map Tool Dialogs](#) ^[514] >



Mapping Tools > Tools > ID GPS Locations
Commandline: none

Alternately from the **Map** tool:
Tools > ID GPS Locations

Explanation

This form is used to pick points from a drawing or map and associate them with the select Location record.
This tool only works with DWG drawings that have been calibrated to the GPS coordinate system or with the [Map tool](#) ^[560].

Controls

New: Open the [New Locations](#) ^[481] dialog.

Save: Commit your changes to the [Locations](#) ^[537] table of the project database.

Close: Close this form without saving.

Copy Selected Down: Copy the data from the top cell(s) down overwriting any data in the cell(s) below.

Pick >: Dismiss this dialog and select a point from the drawing or map. The GPS coordinate will be entered in each of the selected rows.

Zoom Show Selected Text: (drawings only) text entities in the drawing that match the text of the currently focused cell.

Show Me: Finds the GPS coordinate in the drawing and places a Location Marker. Then zooms the marker into view.

Leave Location Marker: Sets whether a location marker as determined by the settings is placed at the pick point.

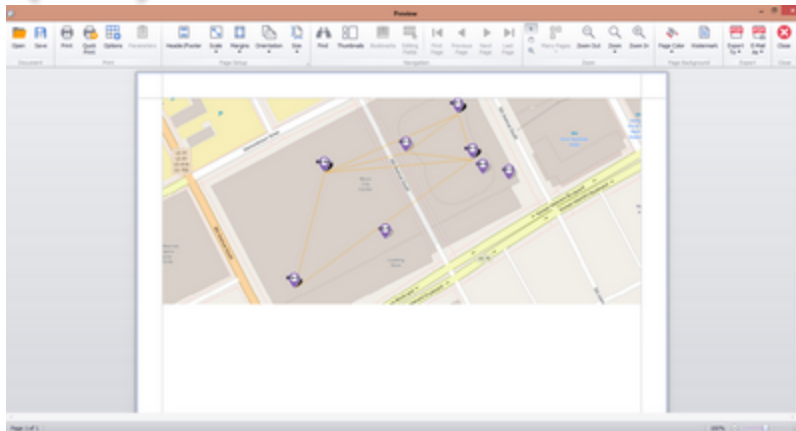
Related Topics:

[How To: Show Location Markers in the Map Tool](#) ^[266]

[How To: Associate GPS LAT, LON with your Locations](#) ^[255]

5.2.8.2 Print Preview

Navigation: [Reference](#) ^[288] > [Dialogs](#) ^[326] > [Map Tool Dialogs](#) ^[514] >



Menu: **File > Print**

Explanation

The print preview dialog can be used to send the current map view to the printer or exported to several file formats.

Controls

Document Section

Open - Open a saved *.prnx file for printing.

Save - Save a *.prnx file for printing later.

Print Section

Print - Open the Printer dialog to set the current printer, page size, number of copies, etc.

Quick Print - Do it now.

Options - Open a dialog to set the image fit properties and what visual elements to include in the printed output.

Header / Footer - Open a dialog to set the document header and footer.

Page Setup Section

Scale - Set the scale of the document or fit it to a number of pages.

Margins - Set the document margins. Alternately you can drag a margin indicator line to resize it.

Orientation - Set portrait or landscape mode.

Size - Set the page size. Sizes are based on the current printer.

Navigation Section

Find - Find text within the document.

Thumbnails - Show the Thumbnails tool panel.

Page Navigation - First, Previous, Next, Last. Move around the pages.

Zoom Section

Mouse Pointer Tool - Enable the mouse pointer.

Pan Hand Tool - Enable the panning tool.

Magnifier Tool - Enable the magnifier tool.

Many Page Selector - Set the number of pages in the viewer.

Zoom Out - Zoom the display out.

Zoom Numeric - Zoom to a specific numeric percentage value.

Zoom In - Zoom the display in.

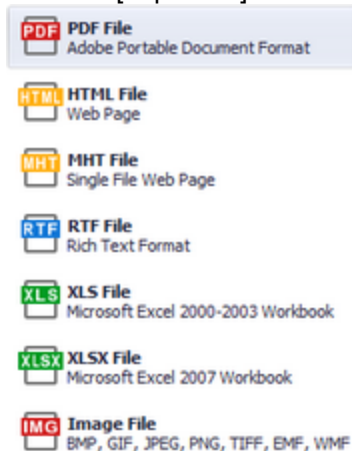
Page Background Section

Page Color - Set the color of the background.

Watermark - Open a dialog to set a watermark for the document. The watermark can be text, image, or both.

Export Section

Both the [Export To] function and the [Email As] functions support the following formats:



5.3 Forms

Navigation: [Reference](#)  >



Forms differ from dialogs in that they occupy the full client docking area of the main window. Only one form may be active at a time. That form may host other dialogs.

5.3.1 Application Menu Forms

Navigation: [Reference](#)  > [Forms](#)  >



The forms in this section are specific to the **Application Menu**.

5.3.1.1 Project Information

Navigation: [Reference](#)  > [Forms](#)  > [Application Menu Forms](#)  >



Application Menu > Project Information
Commandline: Project Info

Explanation

This form contains the basic project information and status. Project name and description here will show up in the reporting engine and can be used elsewhere. The CurrentProjectRevision field populates the Cables and SysNames grid when an edit is made or a new record added the value here will populate there.

The Project Checked-out status can be shown modified here.

Project File Info Tab

Read only look at the PROJECT_NAME.wc6plf file.

Related Projects Tab

Here you can Add/Edit/Remove related projects. Related projects show in the Project Explorer (read-only) and can warn if duplicate SysNames and Cable Numbers are about to be used.

5.3.1.2 Permissions Grid

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Application Menu Forms](#) ^[517] >



Database > Equipment
Commandline: eg

Explanation

This tool provides access to the **Equipment** table of the **Global Equipment** database. The grid is hierarchical. The **[+]** buttons may be expanded to show related **Inputs** and **Outputs**. Deletes here cascade, deleting any **I/O** associated with the **Equipment**.

Prerequisites

1. There is at least one **Manufacturer** added to the **Global Equipment** database.
2. There is at least one **Equipment** definition added to the **Equipment** table.

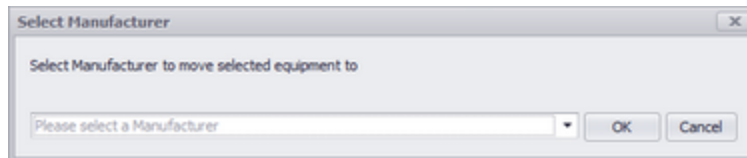
Related Topics:

[Grid Basics](#) ⁴⁹

[How To: Add Equipment](#) ⁶⁵

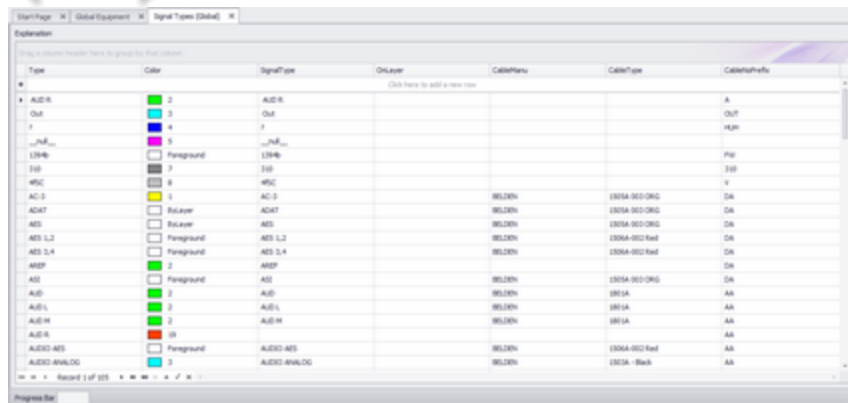
Form Options

- **New** - Show the [New Equipment Wizard](#) ⁴⁸⁶.
- **Attach Document ...** - Any document may be attached to any record. This is a handy place to store manufacturer data sheets, images and such.
- **Clean** - This function scans the table for Equipment with no I/O. If found the following occurs:
 The Equipment definition is deleted.
 The collection is saved.
- **Transfer to Different Manufacturer** - Allows movement of the selected device to a different Manufacturer.



5.3.2.3 Default Signal Types Grid

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Global Data Forms](#) ⁵¹⁹ >



Database > Default Signal Types**Commandline:** st***Explanation***

This tool provides access to the Default Signal Types table of the Global Equipment database. Many of the default display behaviors originate here, for instance:

- The color of pins on a block.
- The color of cables.
- The Cable Type associated with the Signal Type.
- The Cable Number prefix.

These values are preempted by the [Project Signal Types](#) ⁵³¹ grid which has priority. If signal type information is required by the application it first looks to the project, then to the default.

Prerequisites

1.None

Related Topics:[Signal Types](#) ⁵⁹[Grid Basics](#) ⁴⁹***Form Options***

- New** - There is no **New** button on this grid. The New Record Row appears at the top of the grid:

Type	Color	SignalType	OnLayer	CableName	CableType	CablePrefix
Click here to add a new row						

- Rename I/O Signal Types** - Rename the Signal Type in this grid and all I/O in the global database.

- Rebuild Signal Types from I/O** - This function scans the Inputs and Output for Signal Types then:

Checks existing in this grid.

Add if not found.

Collection is saved.

5.3.2.4 Connectors Grid**Navigation:** [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Global Data Forms](#) ⁵¹⁹ >

Connector	Description	DefaultPinOut	Manufacturer	PartNumber	TerminatorPin	Connector1	Connector2	ConnCost1	ConnCost2	ModifiedBy	ModifiedOn	CreatedOn
1	Not Sure	[1] To [2] - 204...						0	0		11/13/2008 10:11...	11/13/2008 10:11...
11	Audio Connector							0	0		11/13/2008 10:11...	11/13/2008 10:11...
12	Unbalanced										11/13/2008 10:11...	11/13/2008 10:11...
13	Balanced										11/13/2008 10:11...	11/13/2008 10:11...
14	MCMD										11/13/2008 10:11...	11/13/2008 10:11...
15	BALANCED										11/13/2008 10:11...	11/13/2008 10:11...
16	Stereo or Sub...										11/13/2008 10:11...	11/13/2008 10:11...
17	Instrument Panel										11/13/2008 10:11...	11/13/2008 10:11...
18	Mono 1/4" Phone							0	0		11/13/2008 10:11...	11/13/2008 10:11...
19	Stereo 1/4" Ph...							0	0		11/13/2008 10:11...	11/13/2008 10:11...
20	1/8" mini							0	0		11/13/2008 10:11...	11/13/2008 10:11...
21	Audio Connector							0	0		11/13/2008 10:11...	11/13/2008 10:11...
22	Prewire							0	0		11/13/2008 10:11...	11/13/2008 10:11...
23	25 Pin D SUB							0	0		11/13/2008 10:11...	11/13/2008 10:11...
24	25 Pin D SUB H...							0	0		11/13/2008 10:11...	11/13/2008 10:11...
25	25 Pin D Sub							0	0		11/13/2008 10:11...	11/13/2008 10:11...
26	3.5mm Standard 3.5m...										11/13/2008 10:11...	11/13/2008 10:11...
27	36 Pin D Sub							0	0		11/13/2008 10:11...	11/13/2008 10:11...
28	36 Pin D Sub							0	0		11/13/2008 10:11...	11/13/2008 10:11...
29	37 Pin D Sub C...										11/13/2008 10:11...	11/13/2008 10:11...
30	37 Pin D Sub C...							0	0		11/13/2008 10:11...	11/13/2008 10:11...

Database > Connectors
Commandline: cn

Explanation

This tool provides access to the **Connectors** table of the **Global Equipment** database.

Prerequisites

- 1.None

Related Topics:

[Grid Basics](#) ⁴⁹

Form Options

•**New** - There is no New button on this grid. The New Record Row appears at the top of the grid:

Type	Color	SignalType	OnLayer	CableMenu	CableType	CablePrefix
New Record Row						

•**Rename I/O Connector Types** - Rename the Connector in this grid and all I/O in the database.

5.3.2.5 Cable Types Grid

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Global Data Forms](#) ⁵¹⁹ >



Manufacturer	CableType	CableDesc	CablePart	CableCore	CableFusing	CableCond	CableStand	CableHeight	CableID	CableChar1	CableUsage	CableVendor1	CableVendor2	CableCost1	CableCost2
1	GENERIC	48 STRAND...	48 STRAND...			Numbered		48							

Conductor Color	Conductor Number	Pair Number	Pair Color Code	Pair Mode
17	17		17 17	MM
18	18		18 18	MM
19	19		19 19	MM
20	20		20 20	MM
21	21		21 21	MM
22	22		22 22	MM
23	23		23 23	MM
24	24		24 24	MM
25	25		25 25	MM
26	26		26 26	MM
27	27		27 27	MM
28	28		28 28	MM
29	29		29 29	MM
30	30		30 30	MM
31	31		31 31	MM
32	32		32 32	MM
33	33		33 33	MM
34	34		34 34	MM
35	35		35 35	MM
36	36		36 36	MM
37	37		37 37	MM
38	38		38 38	MM
39	39		39 39	MM
40	40		40 40	MM
41	41		41 41	MM
42	42		42 42	MM
43	43		43 43	MM
44	44		44 44	MM
45	45		45 45	MM
46	46		46 46	MM
47	47		47 47	MM
48	48		48 48	MM

Database > Cable Types

Commandline: ShowCableTypesGrid

Explanation

This tool provides access to the **Cable Types** table of the **Global Equipment** database. The grid is hierarchical. Clicking the **[+]** button will display a child grid of the **ConductorsCollection** for that record if any.

Prerequisites

- 1.None

Related Topics:

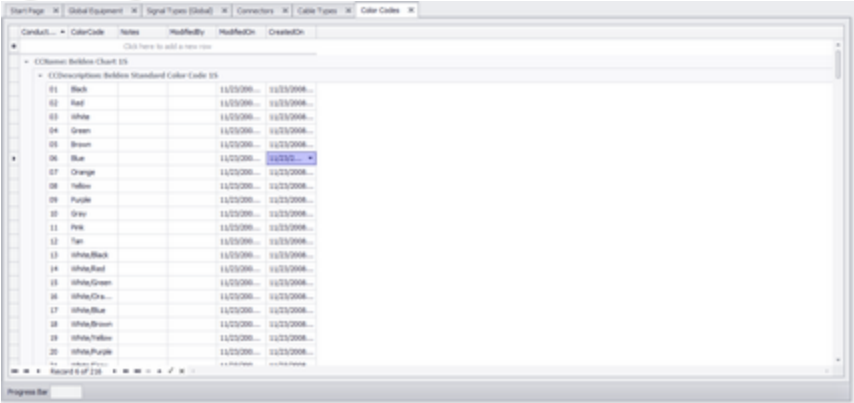
- [Grid Basics](#) ⁴⁹
- [How to Add a New Cable Type](#) ⁶¹

Form Options

- New - Shows the [New Cable Type](#) ⁴⁸⁷ dialog.

5.3.2.6 Color Codes Grid

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Global Data Forms](#) ⁵¹⁹ >



Project Explorer > Color Codes
Commandline: showcolorcodesgrid

Explanation

This tool provides access to the **Color Codes** table of the **Global Equipment** database.

Prerequisites

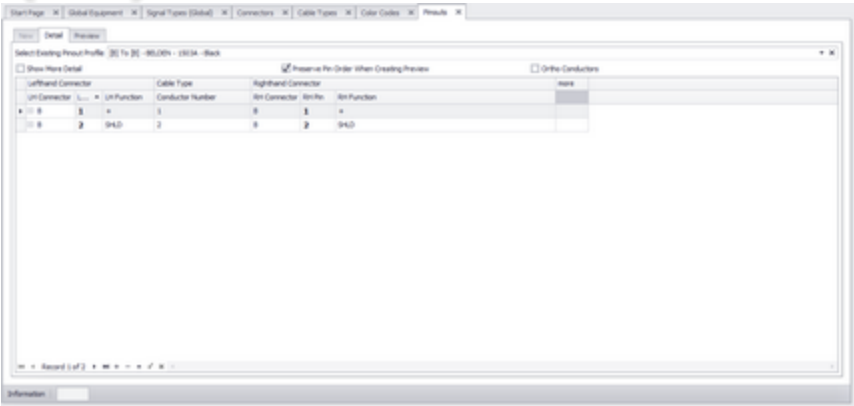
- 1.None

Related Topics:

- [Grid Basics](#) ⁴⁹

5.3.2.7 Pinouts

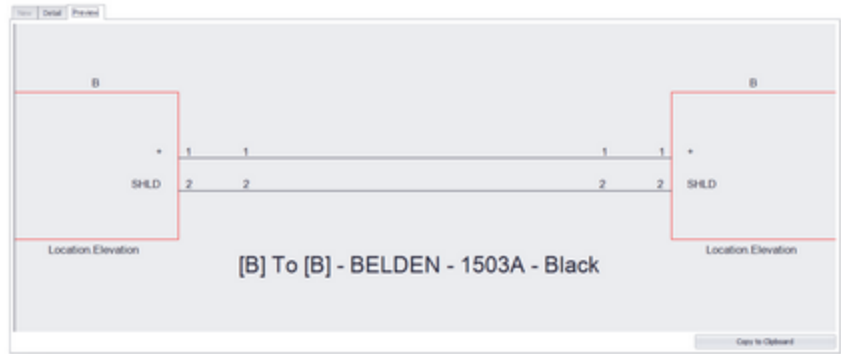
Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Global Data Forms](#) ^[519] >



Database > Pinouts
Commandline: po

Explanation

This tool provides access to the **Pinouts** table of the **Global Equipment** database. The concept of the **Pinouts** tool is to create a data set that describes two connectors and the conductors between them. The **Cable Type** data needs to have conductor level data. The data *may* be attached at the project level to a cable. The data can also be churned into a preview.

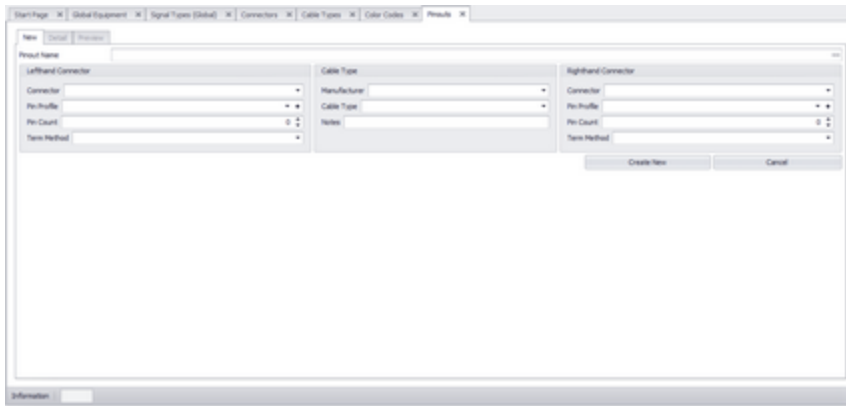


Prerequisites
1.None

Related Topics:
[Grid Basics](#) ^[49]

Form Detail Tab Options

- **Select Existing Pinout Profile** - Select a named pinout to fill the grid and subsequent preview.
- **Show More Detail**
- **Preserve Pin Order When Creating Preview, Ortho Conductors** - How is the **Preview** created.
- **Pinout Grid**



Clicking the **File > New** button enables the **New** tab. From here you enter the overall data for the pinout. The Left-hand and Right-hand connectors conductor counts, etc. Once the pinout is created you can edit the detail in the **Detail** tab.

Form New Tab Options

- **Left hand Connector Info** - Enter the connector type, pin profile (if any) and count.
- **Cable Type** - Select the **Cable Type**. Remember this type must have conductor level data. **Core Level** data alone will not suffice here.
- **Right hand Connector Info** - Enter the connector type, pin profile (if any) and count.

Click **File > Save** to commit your changes and switch back to the grid view.

5.3.3 Project Data Forms

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ >

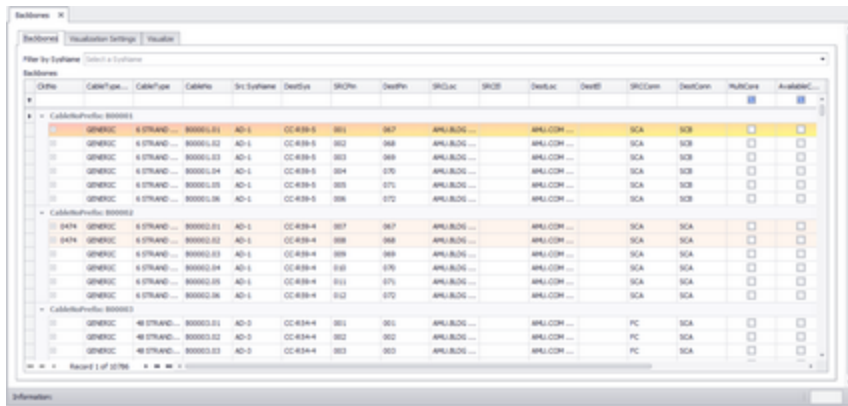


The forms in this section are specific to the **Project Database**.

5.3.3.1 Backbone Grid (CMS ONLY)

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >





Databases > Backbones

Commandline: bbg

Explanation

This is the Backbones view and includes a visualizer that can produce both ladder diagrams and riser diagrams.

Here you can view and search Backbone data. You can also set the Status of a Backbone.

Prerequisites

Backbones created in the project database.

Related Topics:

New Backbone  214

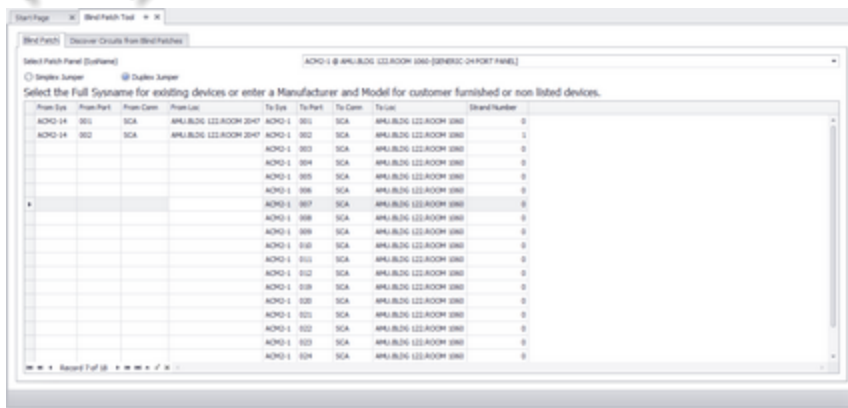
Backbone Preview Settings

Backbones 210

Grid Basics

5.3.3.2 Blind Patch Form (CMS ONLY)

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >



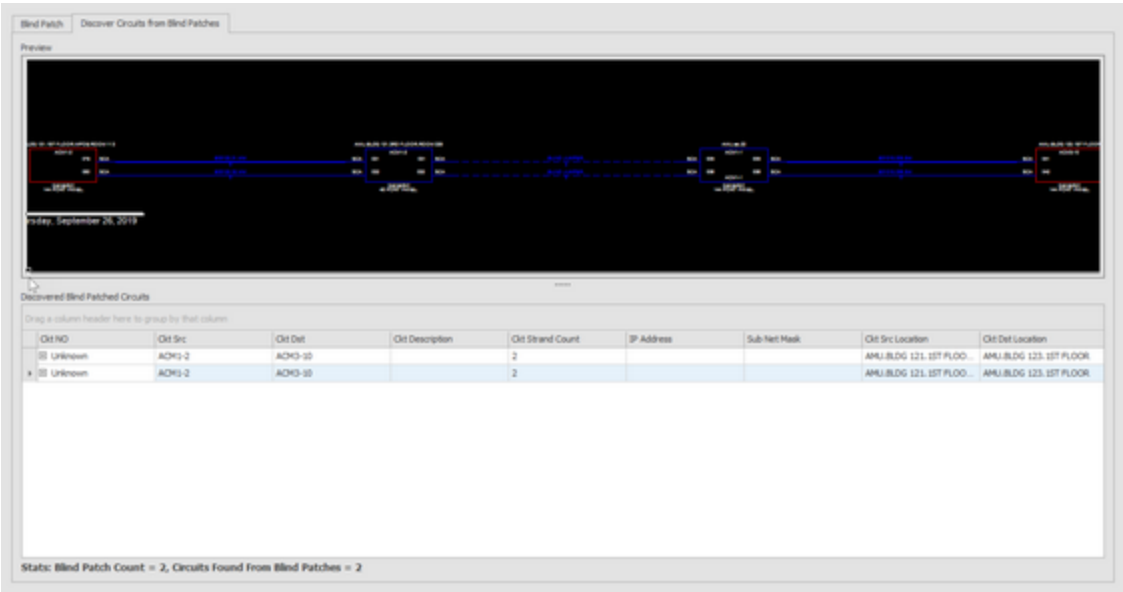
Project Explorer > Project Databases > Blind Patches

Commandline: ShowBlindPatchTool

Explanation

This is the **Blind Patch** tool that allows you to document your patches without knowing the whole circuit. Once the patches (jumpers) are documented you can use the **Discover** tool to find what is patched to what. If you find a discovered circuit to be complete you can use the **Make Circuit** command to create a real circuit and remove the **Blind Patches** from this list.

The form is tabbed. The **[Blind Patch]** tab (shown above) provides a data entry grid where you can add/edit/delete **Blind Patches**. The second tab (shown below) is used to Discover/Preview/Make Circuit(s) from your Blind Patches.



Prerequisites

At least two Backbones

Related Topics:

- [Blind Patch Tool Explanation](#) ²²⁶
- [How To: Create a Blind Patch](#) ²²⁷
- [How To: Discover Circuits from Blind Patches](#) ²²⁷
- [How To: Make a Real Circuit from Blind Patches](#) ²²⁸

Form Options:

Menus

New: Create a new **From SysName** for the selected record based on the **Simplex/Duplex** option and the selected ToSys.

Save: Commit the changes to the database.

Delete: Delete the selected record(s). This function will delete single records for Simplex patches and two records for Duplex patches.

Discover Circuits: Search the blind patches and find all possible connections to make a circuit.

[Circuits Preview Settings](#) ²³⁰
[Combine Circuits](#) ²⁴³
[Grid Basics](#) ⁴⁹

5.3.3.4 New Circuit (CMS ONLY)

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >

Database > Circuits [File > New]
 Commandline: none

Explanation

This tool provides a means by which to create a new circuit. **Circuits** can either be assembled manually or automatically. Creating circuits manually involves selecting each jumper and backbone segment. To create a circuit automatically you define the first and last jumper and hand it off to the **Path Finder** tool which will find and present candidate routes.

Prerequisites

1. Backbones added.

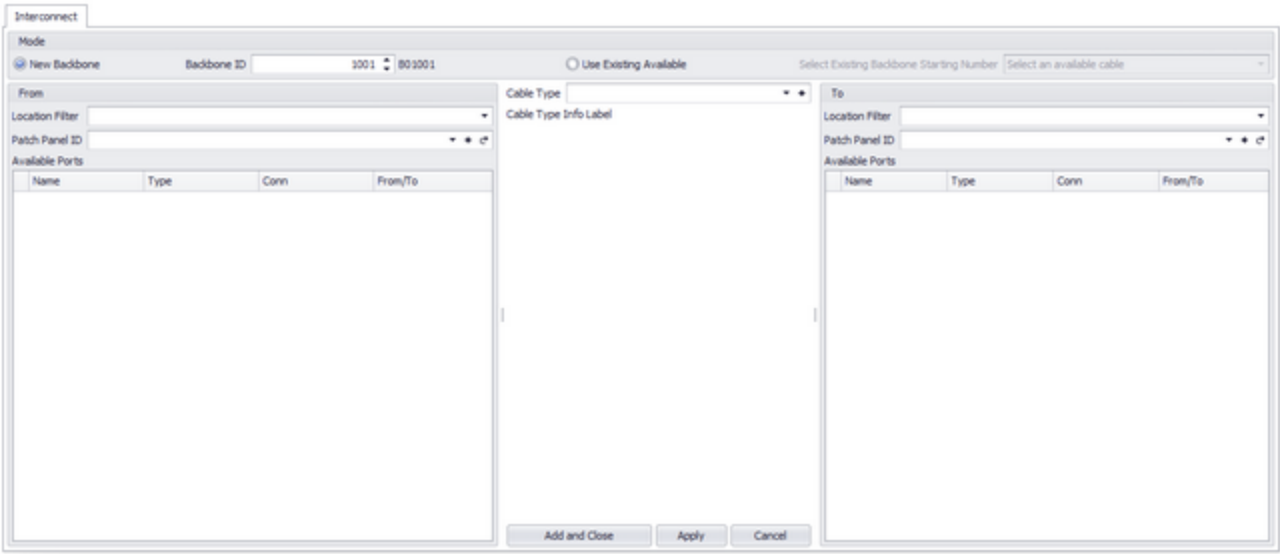
Related Topics:

[New Circuit](#) ²³⁶

5.3.3.5 New Backbone (CMS ONLY)

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >





Database > Backbones [File > New]
Commandline: none

Explanation

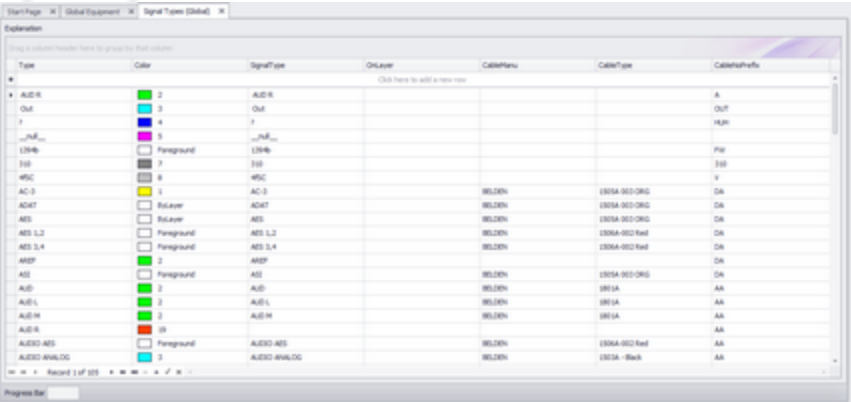
Create a new Backbone.

Prerequisites
1.None

Related Topics:
[Backbones Grid](#) ⁵²⁶
[Backbones](#) ²¹⁰
[Grid Basics](#) ⁴⁹

5.3.3.6 Project Signal Types Grid

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >



Database > Project Signal Types
Commandline: st

Explanation

This tool provides access to the Project Signal Types table of the Project database. Many of the default display behaviors originate here, for instance:

- The color of pins on a block.
- The color of cables.
- The Cable Type associated with the Signal Type.
- The Cable Number prefix.

These values preempt the [Default Signal Types](#)⁵²¹ grid. This grid has priority. If signal type information is required by the application it first looks to the project, then to the default.

Prerequisites

- 1.None

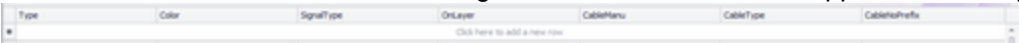
Related Topics:

[Signal Types](#)⁵⁹

[Grid Basics](#)⁴⁹

Form Options

- New** - There is no **New** button on this grid. The New Record Row appears at the top of the grid:



- Rename I/O Signal Types** - Rename the Signal Type in this grid and all I/O in the global database.
- Rebuild Signal Types from I/O** - This function scans the Inputs and Output for Signal Types then:
 - Checks existing in this grid.
 - Add if not found.
 - Collection is saved.

5.3.3.7 Equipment List

Navigation: [Reference](#)²⁸⁸ > [Forms](#)⁵¹⁷ > [Project Data Forms](#)⁵²⁶ >



Visualization Settings Visualization

Columns with colored background must be edited from the Equipment List table or Locations table and ripple here.

Search Find All

Drag a column header here to group by that column

Sheet	Category	Src SysName	DestLoc	SrcPin	DestPin	SrcLoc	DestLoc	SrcConn	DestConn
1001_1_Auto.dwg	A-1011	PA-01	SP-02	R	2N	SRN-1-12	SRN-ASHT	1	San
1001_1_Auto.dwg	A-1012	PA-01	SP-01	L	2N	SRN-1	SRN	1	San
1001_1_Auto.dwg	A-1013	PA-02	SP-04	R	2N	112-1-12	112-ASHT	1	San
1001_1_Auto.dwg	A-1014	PA-02	SP-03	L	2N	112-1-12	112-UPPT	1	San
1001_1_CTL.dwg	CT-1001	UP-01	SP-01	R1-422	R1-422-0	SRN-203K	ESTT-1-WALL	19	NO
1001_1_CTL.dwg	CT-1002	UP-02	SP-02	R1-422	R1-422-0	112-203K	ESTT-2-WALL	19	NO
1001_1_WB.dwg	DI-1040	WB-01	SP-01	R-01	102-N7N-0	ROOM-110-A-30	ESTT-1-WALL	8	8
1001_1_WB.dwg	DI-1040	WB-01	SP-02	R-02	102-N7N-0	ROOM-110-A-30	ESTT-2-WALL	8	8
1001_1_WB.dwg	DI-1040	Endblocker-01	SP-01	102-N7N-0	102-N7N-1	SRN-1-1-0	ESTT-1-WALL	8	8
1001_1_WB.dwg	DI-1040	Endblocker-02	SP-02	102-N7N-0	102-N7N-1	112-1-1-0	ESTT-2-WALL	8	8

Record 1 of 10

Starts with (DestLoc, 'w')

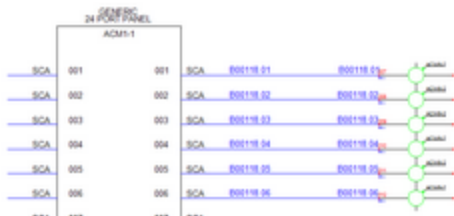
Sub Filter

Databases > Equipment List
Commandline: sys

Explanation

This is the main **Equipment List** of all **SysNames** in the project. You can edit and ripple your changes from here.

This grid contains a **Visualizer** that lets you see the data in graphical form. The **Visualizer** associated with this grid will create a system snapshot of the selected **SysName**.



Prerequisites
Assigned SysNames

Related Topics:
[Assign SysName](#) 449
[Grid Basics](#) 49
[How To: Use the Equipment List Visualizer](#) 116

Data Tab Options

- Search** - Search the collection for the search term.
- Find All** - Clear the search field and return all records.
- Ripple List** - Changes you have made that may need to be rippled across the drawing set. The ripple will occur after you save the grid to the database.

Tools

- Attach Document** - Any document can be attached to any record. This is useful for storing data sheets, test data, and images.
- Slurp Locations** - Collect all location data from the **Equipment List** and populate the **Locations** table with it.

Visualization Tab Options

- Terminal** - Select the terminal to which we will attach the cables.
- Vertical Pin Spacing 1/100 DU** - How far apart vertically are the ports.
- Use Last Display Order if Set**
- Use Last Saved Appearance** - Where is the SysName label positioned with reference to the end points of the polyline that represents the cable.
- Body Width** -
- Port Data Source** - Pull the port data from the **Equipment Library** which will show all available ports as defined in the **Equipment Library** or pull port data from the **Cable** table which will show only ports which have cables attached.



5.3.3.8 Cables

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Project Data Forms](#) ^[526] >



•**Ripple List** - Changes you have made that may need to be rippled across the drawing set. The ripple will occur after you save the grid to the database.

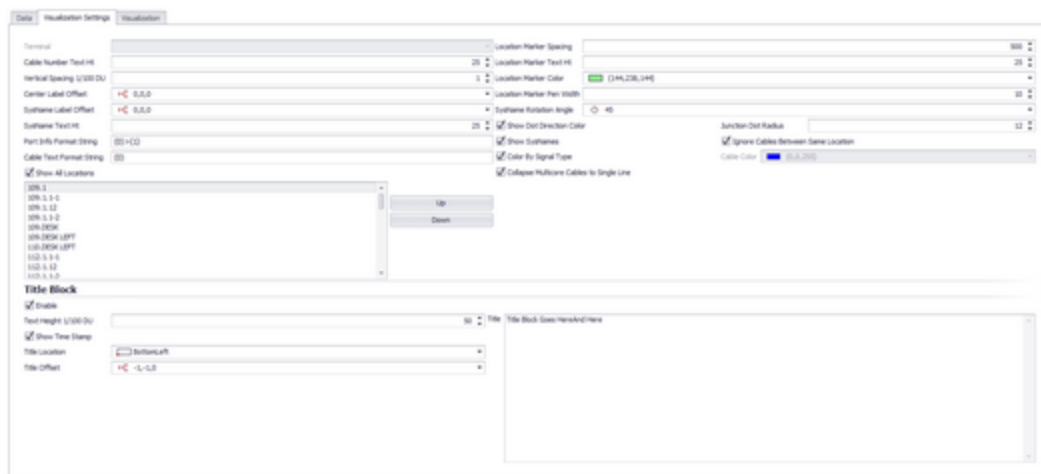
•**Expert Mode** - Removes the read-only flags and puts the grid in a completely editable state. Not for beginners.

•**Add Many** - Only available in **Expert Mode** this tool adds blank records to the table that you can edit as a spreadsheet.

Tools

•**Repair Equipment List** - Scans the **Cables** table for **SysNames** then compares with the **Equipment List**. Missing **SysNames** are then added to the **Equipment List**.

•**Attach Document** - Any document can be attached to any record. This is useful for storing data sheets, test data, and images.



Visualization Tab Options

•**Terminal** - Not yet implemented.

•**Cable Number Text Height**

•**Vertical Spacing 1/100 DU** - How far apart vertically are the cables.

•**Center Label Offset** - Where is the center label positioned with reference to the center of the polyline that represents the cable.

•**SysName Label Offset** - Where is the **SysName** label positioned with reference to the end points of the polyline that represents the cable.

•**SysName Text Height** - The text height of the **SysName** label.

•**Port Info Format String** - The following variables can be used along with the /n new line character to format the port info:

{0}SysName

{1}Port

{2}Location

{3}Alias

{4}Connector

{5}Cable Number

{6}Cable Manufacturer

{7}Cable Type

Example:

Assume that the SysName is VTR-01. The PortName is VID-OUT and the Location is RK-10 The following Port Info Format string:

{0}>{1}@{2}

Will produce the result:

VTR-01>VID-OUT@RK-10

•**Cable Text Format String** - The following variables can be used along with the /n new line character to format the Cable Text:

- {0}Cable Number
- {1}Cable Manufacturer
- {2}Cable Type
- {3}Length
- {4}Sheet

•**Location Marker Spacing** - How far apart horizontally the vertical location markers are spaced.

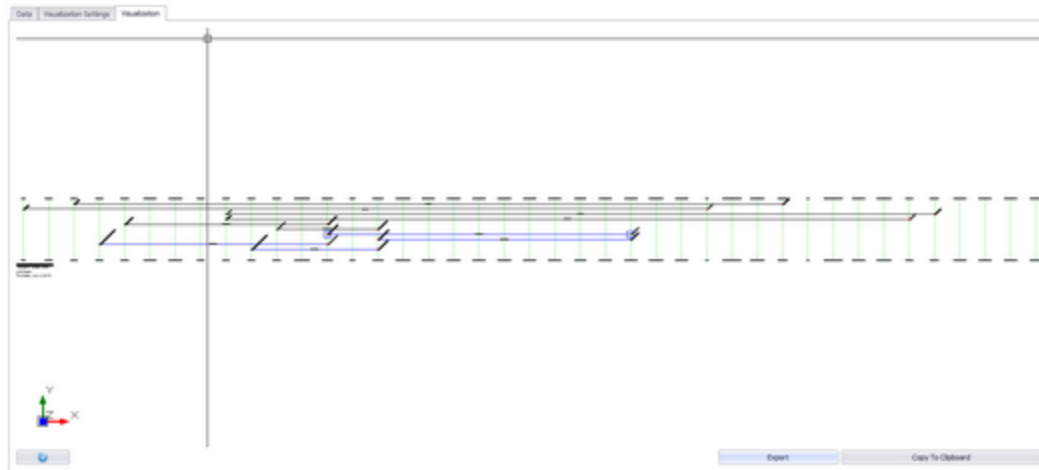
•**Location Marker Text Hit, Location Marker Color, Location Marker Pen Width, SysName Rotation Angle**

•**Show Dot Direction Color** - Display a green dot on the source side and a red dot on the destination side of the cable.

•**Show SysNames**

•**Color By Signal Type** - Inherit cable color by Signal Type or set a single color for all cables.

•**Collapse Multi-core Cables to Single Line** - Multi-core cables are represented by many records in the Cables table. You can choose to display a cable for each or collapse to a single cable.



5.3.3.9 Locations

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Project Data Forms](#) ^[526] >



Locations

You can use all or none of these fields to identify a location. The Qualified Location field is the one that will be used throughout the rest of the application.

Drag a column header here to group by that column

Campus	Building	Floor	Room	Rack	Description	Qualified Location (...)	Figurehandle
				ab		ab	
AMU				ab		AMU.ab	
AMU		BLDG 121	ROOM 312			AMU.BLDG 121.RO...	
AMU	ANNEX 1	1ST FLOOR	101			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	102			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	102			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	103			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	111			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	121			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	122			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	123			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	124			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	135			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 1	1ST FLOOR	150			AMU.ANNEX 1.1ST ...	
AMU	ANNEX 2	1ST FLOOR	104			AMU.ANNEX 2.1ST ...	
AMU	ANNEX 2	1ST FLOOR	105			AMU.ANNEX 2.1ST ...	
AMU	ANNEX 2A	1ST FLOOR	111			AMU.ANNEX 2A.1S...	
AMU	ANNEX 2A	1ST FLOOR	112			AMU.ANNEX 2A.1S...	
AMU	ANNEX 2A	1ST FLOOR	113			AMU.ANNEX 2A.1S...	
AMU	ANNEX 2A	1ST FLOOR	114			AMU.ANNEX 2A.1S...	

Record 139 of 319

Changes to Ripple Across Project

Databases > Locations
Commandline: showprojectlocationsgrid

Explanation

This is the main Locations table view. You can print, export and edit this view. Changes to this grid can be rippled throughout the project. Remember that the only bit of data WireCAD actually uses from this grid is the **QualifiedLocation**.

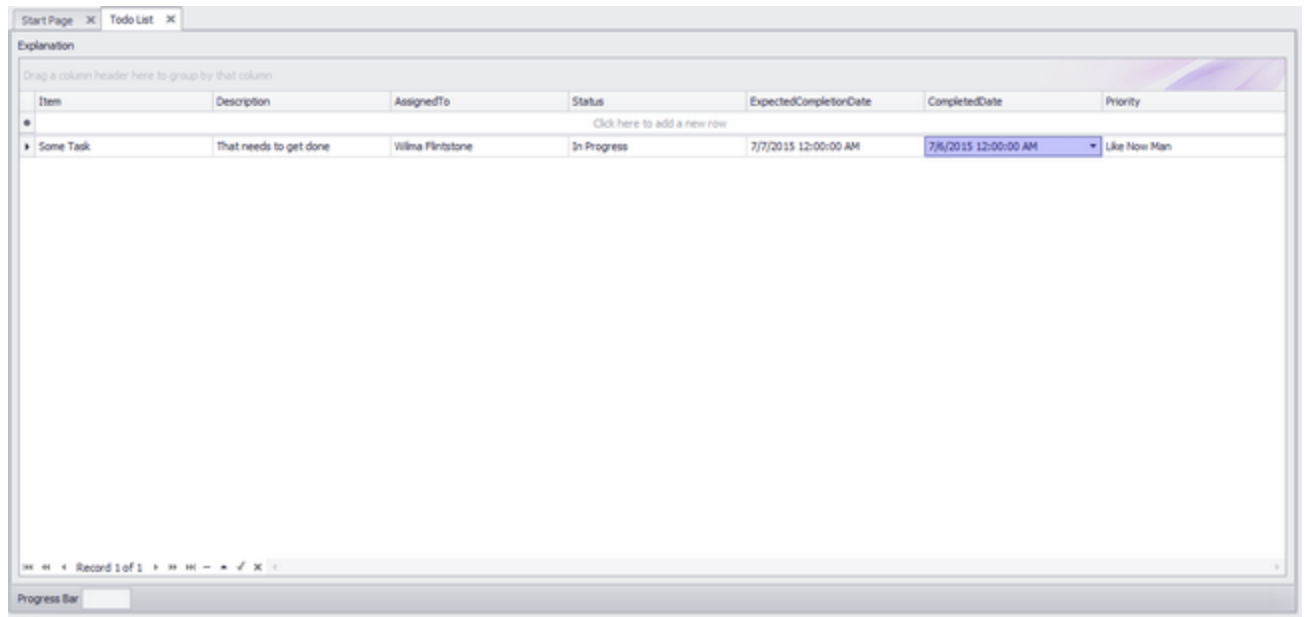
Prerequisites
None

Related Topics:
[New Location Dialog](#)
[Grid Basics](#)

5.3.3.10 Named Paths

Navigation: [Reference](#) > [Forms](#) > [Project Data Forms](#) >





Databases > Todo List
Commandline: tdl

Explanation

A To Do list. You can define the data that fills the following drop downs in the [Todo List Settings](#) ³⁵⁴ panel:

- **Assignees**
- **Statuses**
- **Priorities**

Prerequisites

An active project.

Related Topics:

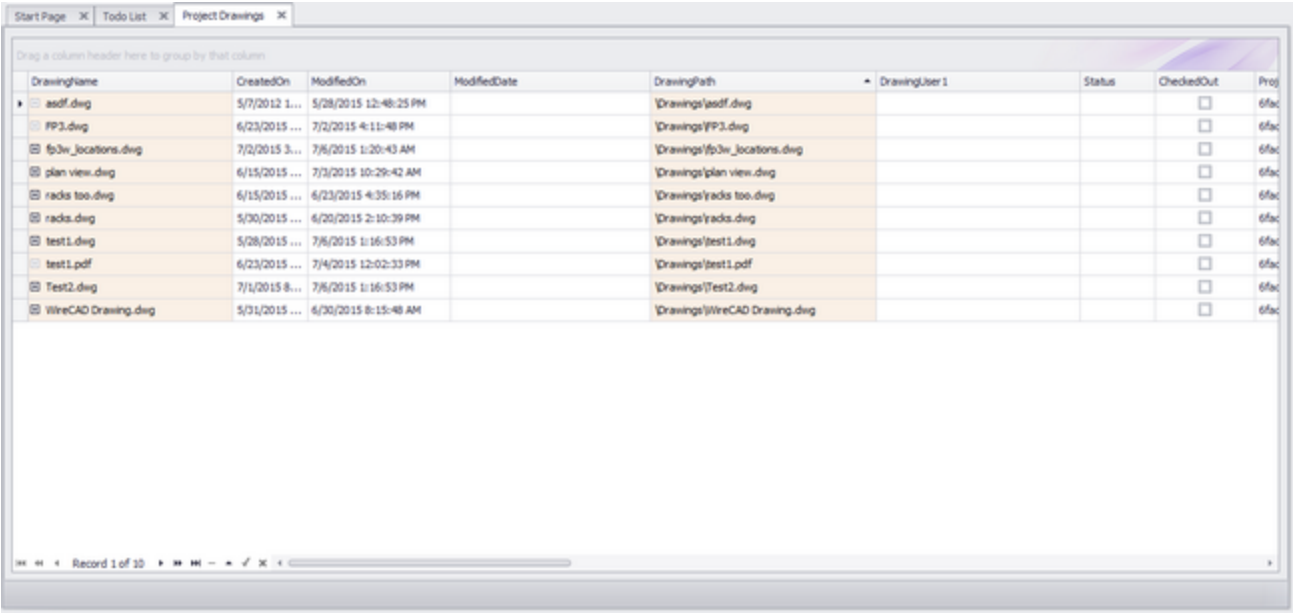
[Todo List Settings](#) ³⁵⁴

[Grid Basics](#) ⁴⁹

5.3.3.12 Drawings

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >





DrawingName	CreatedOn	ModifiedOn	ModifiedDate	DrawingPath	DrawingUser	Status	CheckedOut	Proj
asdf.dwg	5/7/2012 1...	5/28/2015 12:48:25 PM		Drawings\asdf.dwg			<input type="checkbox"/>	6fac
FP3.dwg	6/23/2015 ...	7/2/2015 4:11:48 PM		Drawings\FP3.dwg			<input type="checkbox"/>	6fac
fp3w_locations.dwg	7/2/2015 3...	7/6/2015 1:20:43 AM		Drawings\fp3w_locations.dwg			<input type="checkbox"/>	6fac
plan view.dwg	6/15/2015 ...	7/3/2015 10:29:42 AM		Drawings\plan view.dwg			<input type="checkbox"/>	6fac
racks too.dwg	6/15/2015 ...	6/23/2015 4:35:16 PM		Drawings\racks too.dwg			<input type="checkbox"/>	6fac
racks.dwg	5/30/2015 ...	6/20/2015 2:10:39 PM		Drawings\racks.dwg			<input type="checkbox"/>	6fac
test1.dwg	5/28/2015 ...	7/6/2015 1:16:53 PM		Drawings\test1.dwg			<input type="checkbox"/>	6fac
test1.pdf	6/23/2015 ...	7/4/2015 12:02:33 PM		Drawings\test1.pdf			<input type="checkbox"/>	6fac
Test2.dwg	7/1/2015 8...	7/6/2015 1:16:53 PM		Drawings\Test2.dwg			<input type="checkbox"/>	6fac
WireCAD Drawing.dwg	5/31/2015 ...	6/30/2015 8:15:48 AM		Drawings\WireCAD Drawing.dwg			<input type="checkbox"/>	6fac

Project Explorer> Project Databases > Drawings
Commandline: sdg

Explanation

This is the list of drawings in the Project\Drawings folder tree. The following changes in the Drawings table have occurred in WireCAD v8 and forward:

- The DrawingsPath has changed from absolute to relative.
- The tool now scans for and lists pdf files.
- If the project database is SQLAzure the dwg and pdf files are embedded in the database for portability if the project settings [Folder Sync](#)^[378] is enabled.

NOTE: to rename a drawing use the function in the Project Explorer. Right-click a closed drawing and select the context menu item Rename Drawing.

Prerequisites
An active project.

Related Topics:
[Grid Basics](#)^[49]

5.3.3.13 Index Grids

Navigation: [Reference](#)^[288] > [Forms](#)^[517] > [Project Data Forms](#)^[526] >



More Information
[Understanding Indexes](#)^[133]

5.3.3.13.1 Unassigned Cables

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Project Data Forms](#) ^[526] > [Index Grids](#) ^[541] >



Drawing Name	SRC Sys	Src Sys Loc	SRC Pin	SRC Conn	Dest Sys	Dest Sys Loc	Dest Pin	Dest Conn	Signal Type
Drawing Name: B03.DWG, 013.013R.dwg									
Show Me 1, 23...	APJ.BLDG	126	046-034	SCA	GPS-6	APJ.BLDG 013.013R...	001-006	SCA	FIBER
Show Me 1, 23...	AD-1	APJ.BLDG 013.013R...	001-006	SCA	CC.BLDG-5	APJ.COM CTR, 027 P...	007-072	SCB	FIBER
Show Me 1, 23...	AD-1	APJ.BLDG 013.013R...	013-018	SCA	AD-17	APJ.BLDG 013.013R...	001-006	SCA	FIBER
Show Me 1, 23...	AD-1	APJ.BLDG 013.013R...	013-024	SCA	AD-17	APJ.BLDG 013.013R...	007-072	SCA	FIBER
Show Me 1, 23...	AD-1	APJ.BLDG 013.013R...	007-012	SCA	CC.BLDG-4	APJ.COM CTR, 027 P...	007-072	SCA	FIBER
Show Me 1, 23...	AD-4	APJ.BLDG 013	029-036	ST	AD-5	APJ.BLDG 013.013R...	029-036	ST	FIBER
Show Me 1, 23...	AD-4	APJ.BLDG 013	037-048	ST	AD-5	APJ.BLDG 013.013R...	037-048	ST	FIBER
Show Me 1, 23...	AD-4	APJ.BLDG 013	013-024	ST	AD-5	APJ.BLDG 013.013R...	013-024	ST	FIBER
Show Me 1, 23...	AD-4	APJ.BLDG 013	001-012	ST	AD-5	APJ.BLDG 013.013R...	001-012	ST	FIBER
Show Me 1, 23...	AD-5	APJ.BLDG 013.013R...	046-060	ST	AD-6	APJ.BLDG 013.013R...	001-012	ST	FIBER
Show Me 1, 23...	AD-5	APJ.BLDG 013.013R...	001-012	ST	AD-6	APJ.BLDG 013.013R...	013-024	ST	FIBER
Show Me 1, 23...	AD-12	APJ.BLDG 013.013R...	115-128	SCA	AD-24	APJ.BLDG 013.013R...	013-036	SCA	FIBER
Show Me 1, 23...	AD-12	APJ.BLDG 013.013R...	009-114	SCA	AD-24	APJ.BLDG 013.013R...	029-036	SCA	FIBER
Show Me 1, 23...	AD-15	APJ.BLDG 013.013R...	001-048	SC	AD-18	APJ.BLDG 013.013R...	001-048	SC	FIBER
Show Me 1, 23...	AD-18	APJ.BLDG 013	046-072	SCA	AD-24	APJ.BLDG 013.013R...	001-024	SCA	FIBER
Show Me 1, 23...	AD-18	APJ.BLDG 013	001-144	SC	AD-21	APJ.BLDG 013.013R...	001-144	SC	FIBER
Show Me 1, 23...	AD-22	APJ.BLDG 013.013R...	001-048	SC	AD-23	APJ.BLDG 013.013R...	001-048	SC	FIBER

Database > Index Grids > Unassigned Cables
Commandline: ShowUnassignedCablesGrid

Explanation

Any **Cable** that does not yet have a number to this index by drawing.
As you approach the end of a project this grid should be empty.

5.3.3.13.2 Unassigned SysNames

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Project Data Forms](#) ^[526] > [Index Grids](#) ^[541] >



Figure Handle	Manufacturer	Equipment Name	Entry Type
Drawing Name: B03.DWG, 013.013R.dwg			
Show Me 1, 13448	360 0157095	Image Server (S)	EquipmentBlock
Show Me 1, 13448	360 0157095	Image Server (S)	EquipmentBlock
Show Me 1, 13448	CUSTOM PANEL	SSP	EquipmentBlock
Show Me 1, 13448	CUSTOM PANEL	SSP	EquipmentBlock
Show Me 1, 13448	CUSTOM PANEL	SSP	EquipmentBlock
Show Me 1, 13448	ManufacturerName	EquipmentName	Printer
Show Me 1, 13448	ManufacturerName	EquipmentName	Printer

Database > Index Grids > Unassigned SysNames
Commandline: ShowUnassignedSysNamedGrid

Explanation

Any equipment that does not yet have a **SysName** and add it to this index by drawing. As you approach the end of a project this grid should be empty.

5.3.3.13.3 Cables by Drawing

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Project Data Forms](#) ^[526] > [Index Grids](#) ^[541] >



Start Page | Cables by Drawing

These represent Cables found in drawings that match entries in the Cables database.

Figure Handle	Full Cable Type	Cable No.	Sys Sys	Qualified Sys Loc...	Sys Pin	SBC Conn	Dest Sys	Qualified Dest Loc...	Dest Pin	Dest Conn	Signal Type
+ Drawing Notes: 0007_1_A01.dwg											
Show Me	A-0001	CSR-01	SIN DESK LEFT	CR 1	S/A	PA-01	SIN S 12	1	X		ALD
Show Me	A-0010	CSR-01	SIN DESK LEFT	CR 6	S/A	PA-01	SIN S 12	6	X		ALD
Show Me	A-0011	PA-01	SIN S 12	6	X	SPN-01	SIN RSGHT	24	San	BAH	
Show Me	A-0012	PA-01	SIN S 12	1	X	SPN-01	SIN LEFT	24	San	BAH	
Show Me	075-0001	LP-01	SIN DESK	AS 402	28	SP-01	0077 S HALL	AS 402-2	90		R402
Show Me	0-0001	CPU-01	SIN DESK	PMW000	000	LP-01	SIN DESK	1304	1304		DATN
Show Me	0-0002	CPU-01	SIN DESK	MON-01	HE05	Chen-01	SIN DESK RSGHT	VSA	130-HD		VSA
Show Me	0-0003	CPU-01	SIN DESK	MON-02	HE05	Chen-01	SIN DESK RSGHT	VSA	130-HD		VSA
Show Me	DA-0001	LP-01	SIN DESK	CH12	N/A	CSR-01	SIN DESK LEFT	AS 5.6	X		AS5
Show Me	DA-0004	CSR-01	SIN DESK LEFT	AS 5.6	X	LP-01	SIN DESK	CH12	N/A		AS5
Show Me	DA-0005	CHMR-01	SIN S 1-0	AS01	8	CSR-01	SIN DESK LEFT	AS 5.2	X		AS5
Show Me	DA-0006	CHMR-01	SIN S 1-0	AS02	8	CSR-01	SIN DESK LEFT	AS 5.4	X		AS5
Show Me	DA-0007	CSR-01	SIN DESK LEFT	AS 5.2	X	Embedder-01	SIN S 1-2	AS5 01	8		AS5
Show Me	DA-0008	CSR-01	SIN DESK LEFT	AS 5.4	X	Embedder-01	SIN S 1-2	AS5 02	8		AS5
Show Me	DA-0009	SP-01	0077 S HALL	AS 479-0	8	Embedder-01	SIN S 1-1	AS5 Input	8		AS5
Show Me	DA-0010	LP-01	SIN DESK	001 OUT	SHC	Embedder-01	SIN S 1-2	AS5	8		AS5
Show Me	DA-0040	CHMR-01	SIN S 1-0	AS5 01	8	LP-01	SIN DESK	AS5 24	SHC		AS5

Record 1 of 200

Database > Index Grids > Cables by Drawing
 Commandline: ShowCablesbyDrawingGrid

Explanation

Any **Cable** that has a number and corresponding entry in the **Cables** database is added to this index by drawing.

5.3.3.13.4 SysNames by Drawing

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Project Data Forms](#) ^[526] > [Index Grids](#) ^[541] >



Start Page | Cables by Equipment | SysNames by Drawing

These represent SysNames found in drawings that match entries in the Equipment list.

Figure Handle	Sys Name	Alias	Qualified Loca...	Manufacturer	Equipment Name	IP Address	Subnet Mask	User 1	User 2	User 3	User 4
+ Drawing Notes: 0007_1_A01.dwg											
Show Me	Chen-01										
Show Me	Chen-02										
Show Me	CPU-01										
Show Me	CHMR-01										
Show Me	Embedder-01										
Show Me	LP-01										
Show Me	MON-01										
Show Me	CSR-01										
Show Me	PA-01										
Show Me	SP-01										
Show Me	SPN-01										
Show Me	SPN-02										
+ Drawing Notes: 0007_1_A01.dwg											
Show Me	CHMR-01										
Show Me	Embedder-01										
Show Me	LP-01										
Show Me	CSR-01										

Record 1 of 204

Database > Index Grids > SysNames by Drawing

Commandline: ShowSysNamesbyDrawingGrid

Explanation

Any equipment that has a **SysName** and corresponding entry in the **Equipment List** is added to this index by drawing.

5.3.3.13.5 Cables by SysName

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ > [Index Grids](#) ⁵⁴¹ >



Start Page | Cables by Equipment | Cables by SysName

These represent cables found in drawings that match entries in the Equipment List and Cables database.

The Sys Name

Figure Handle	Cable Number	Full Cable Type	Src Sys	Qualified Src Loc...	Src Pin	Src Conn	Dest Sys	Qualified Dest...	Dest Pin	Dest Conn	Signal Type	Drawing Name
+ Other Sys Names:												
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0218			026-01		R	WPU-01			CH-B			MACHINE_ROOM...
Show Me [L] A-0217			026-01		L	WPU-01			CH-A			MACHINE_ROOM...
Record 1 of 3214												

Progress Bar

Database > Index Grids > Cables by SysName
Commandline: ShowCablesbySysNameGrid

Explanation

Any **Cable** that has a number and corresponding entry in the **Cables** database and where both ends of the **Cable** have valid entries in the **Equipment List** will be added to this index.

5.3.3.13.6 Orphaned Cables

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ > [Index Grids](#) ⁵⁴¹ >



Start Page | Cables by Equipment | Cables by SysName | Orphaned Cables Drawing

These represent cables that exist in a drawing but do not match cables in the database.

Drawing Name

Handle	Cable No.	Src Sys	Qualified Src...	Src Pin	Src Conn	Dest Sys	Qualified Dest...	Dest Pin	Dest Conn	Signal Type	Created By	Orphaned At	Error Message
+ Drawing Name:													
Show Me [L] 21081	WPU-004	UP-02	EST-01A1	WPU	B	UP-02	EST-01	WPU-01	BNC	WPU	shw_000	DRAWING	
Show Me [L] 21089	WPU-005	UP-01	EST-01	WPU	B	UP-02	EST-01	WPU-01	B	WPU	shw_000	DRAWING	

Record 1 of 2

Progress Bar

Database > Index Grids > Orphaned Cables
Commandline: ShowOrphanedCablesGrid

Explanation

Any **Cable** with a number but no corresponding entry in the **Cables** database will be added to this index by drawing.
As you approach the end of a project this grid should be empty.

5.3.3.13.7 Orphaned SysNames

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ > [Index Grids](#) ⁵⁴¹ >



Drawing Name	Manufacturer	Equipment Name	Entry Type	Sys Name	Qualified Location	Orphaned At	Error Message
Show Me % 50 KCP	360 D170M	Instant Reply	EquipmentBlock	36-01	ROOM 105.5.08	DRAWING	
Show Me % 4702A	ADC	AP12246	EquipmentBlock	470-02	ROOM 105.4.26	DRAWING	
Show Me % 50 05C	ADC	AP12230-4M2	EquipmentBlock	47-05	ROOM 105.5.38	DRAWING	
Show Me % 50 07D	ADC	AP12230-4M2	EquipmentBlock	47-06	ROOM 105.5.37	DRAWING	
Show Me % 4703J	ENK12	HE1004	EquipmentBlock	CC-01	ROOM 105.4.39	DRAWING	
Show Me % 4704J	GENERIC EQUIPMENT	Rack 45	EquipmentBlock	RCK-02	ROOM 105.2.1	DRAWING	
Show Me % 4703J	GENERIC EQUIPMENT	Rack 45	EquipmentBlock	RCK-03	ROOM 105.3.1	DRAWING	
Show Me % 4703B	GENERIC EQUIPMENT	Rack 45	EquipmentBlock	RCK-04	ROOM 105.4.1	DRAWING	
Show Me % 50 01J	GENERIC EQUIPMENT	Rack 45	EquipmentBlock	RCK-05	ROOM 105.5.1	DRAWING	
Show Me % 50 03E	GENERIC EQUIPMENT	Traffic Computer	EquipmentBlock	TC-01	ROOM 105.6.20	DRAWING	
Show Me % 4703D	GRASS VALLEY	9000 Frame	EquipmentBlock	PRM-01	ROOM 105.4.40	DRAWING	
Show Me % 4703F	GRASS VALLEY	9000 Frame	EquipmentBlock	PRM-02	ROOM 105.4.40	DRAWING	
Show Me % 4704E	GRASS VALLEY	9040	EquipmentBlock	MDA-01	ROOM 105.4.40.1	DRAWING	
Show Me % 4703E	4546R	1000	EquipmentBlock	WDR-01	ROOM 105.3.25	DRAWING	
Show Me % 48P	ManufacturerName	EquipmentName	Printer	470-02	Location	DRAWING	
Show Me % 1284	ManufacturerName	EquipmentName	Printer	470-02	Location	DRAWING	
Show Me % 1285	ManufacturerName	EquipmentName	Printer	470-02	Location	DRAWING	

Database > Index Grids > Orphaned SysNames
Commandline: ShowOrphanedSysNamesGrid

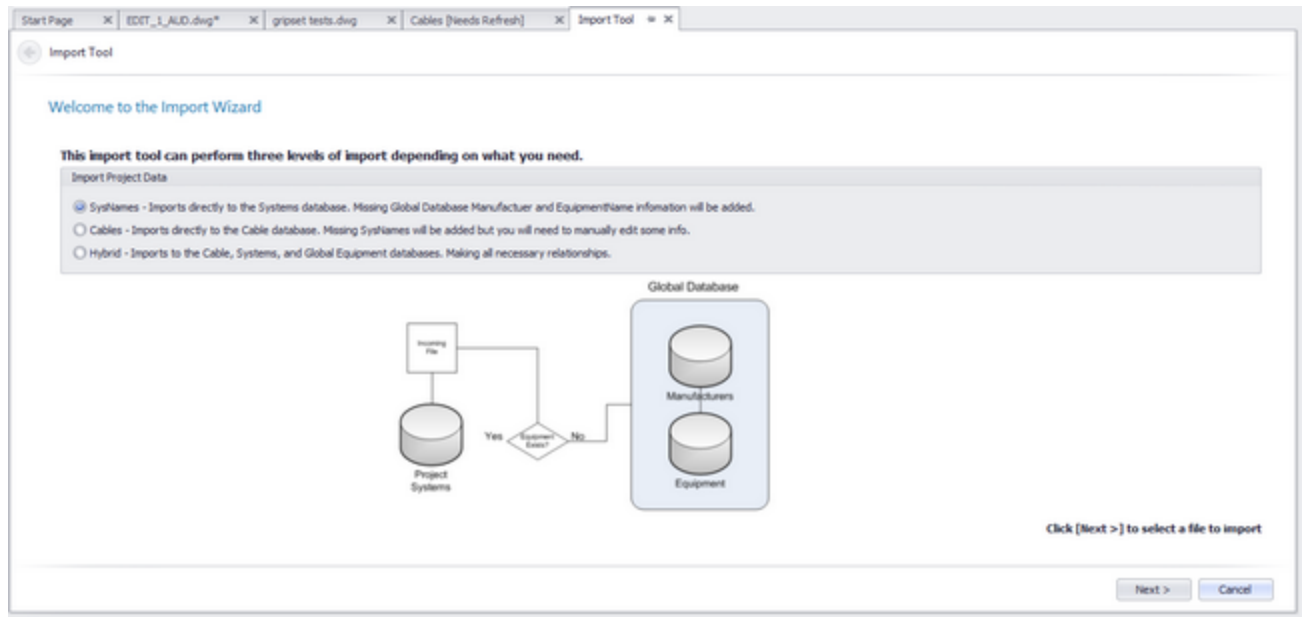
Explanation

Any equipment with a **SysName** but no corresponding entry in the **Equipment List** will be added to this index by drawing.
As you approach the end of a project this grid should be empty.

5.3.3.14 Import Project Data

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >





Database > Import Tool Commandline: it

Explanation

This tool imports data from a text delimited file into the various databases based on your initial selection.

There are three modes available for selection in this form:

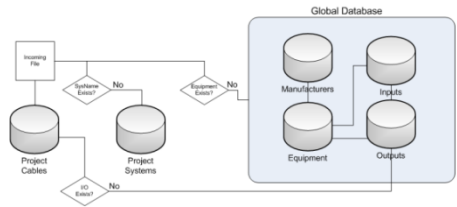
1.Import SysName data. This tool attempts to import data into the project Equipment List and the Global Equipment Library.



2.Import Cables data. This tool attempts to import data into the Cables table and the Equipment List.



3.Hybrid import. This tool attempts to import data into the Cables, Equipment List and Global Equipment Library.



The process leads you through file selection
Column Mapping
Data validation
Import.

Prerequisites
One of the import templates with your incoming data.

Related Topics:
None

5.3.3.15 Discrepancies

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Project Data Forms](#) ⁵²⁶ >



Start Page X Todo List X Project Drawings X								
Drag a column header here to group by that column								
DrawingName	CreatedOn	ModifiedOn	ModifiedDate	DrawingPath	DrawingUser 1	Status	ChededOut	Proj
asdf.dwg	5/7/2012 1...	5/28/2015 12:48:25 PM		\\Drawings\\asdf.dwg			<input type="checkbox"/>	6fac
FP3.dwg	6/23/2015 ...	7/2/2015 4:11:48 PM		\\Drawings\\FP3.dwg			<input type="checkbox"/>	6fac
fp3w_locations.dwg	7/2/2015 3...	7/6/2015 1:20:43 AM		\\Drawings\\fp3w_locations.dwg			<input type="checkbox"/>	6fac
plan view.dwg	6/15/2015 ...	7/3/2015 10:29:42 AM		\\Drawings\\plan view.dwg			<input type="checkbox"/>	6fac
racks too.dwg	6/15/2015 ...	6/23/2015 4:35:16 PM		\\Drawings\\racks too.dwg			<input type="checkbox"/>	6fac
racks.dwg	5/30/2015 ...	6/20/2015 2:10:39 PM		\\Drawings\\racks.dwg			<input type="checkbox"/>	6fac
test1.dwg	5/28/2015 ...	7/6/2015 1:16:53 PM		\\Drawings\\test1.dwg			<input type="checkbox"/>	6fac
test1.pdf	6/23/2015 ...	7/4/2015 12:02:33 PM		\\Drawings\\test1.pdf			<input type="checkbox"/>	6fac
Test2.dwg	7/1/2015 8...	7/6/2015 1:16:53 PM		\\Drawings\\Test2.dwg			<input type="checkbox"/>	6fac
WireCAD Drawing.dwg	5/31/2015 ...	6/30/2015 8:15:48 AM		\\Drawings\\WireCAD Drawing.dwg			<input type="checkbox"/>	6fac

Project Explorer > Project > Discrepancies
Commandline: dr

Explanation

This function is provided for legacy purposes. These functions have been replaced by the index grids.

There is one instance where the discrepancy report is used.

If a ripple is initiated in a multi-user install and another user has the drawing open. The ripple cannot update the drawing.

A record is added to the discrepancy report. Use the **Sync Equipment** function.

Prerequisites

Related Topics:

[Sync Equipment with Database](#) 

5.3.3.16 Next Numbers Grid

Navigation: [Reference](#)  > [Forms](#)  > [Project Data Forms](#)  >




Menu: **Database > Next Numbers**

Default command line shortcut: **NN**

Related Project Settings:

None

Product Level: **ALL**

The **Next Numbers** Grid will show you the next available Cable number based on your [Cable Number Format](#) . To access the **Next Numbers Tool**, type **NN** in the command line prompt.

This grid will show you both **Backbone Cable Numbers** as well as **Standard Cable Numbers**. As you progress through your project, these numbers will change.

To edit these numbers (such as in the event of deleted Backbones) simply click in the Next Number box and fill in the desired number. Be sure that you are not entering a number that is already in use.

Menu: Project Explorer - Project **Databases > Rules**

Default command line shortcut: **ShowProjectRules**

Related Project Settings:

Project>Settings[Project Rules] Enable Project Rules

Product Level: **ALL**

WireCAD 10 allows you to define project specific rules.

There are four contexts in which rules will be evaluated. Rules are contained in the **Project Rules** grid.

A **Rule** consists of:

- 1.The **Expression** to evaluate.
- 2.The **Context** in which to evaluate the expression.
- 3.The **ExecutionOrder**. If multiple rules are defined to evaluate within the same **Context** they will be evaluated in the **ExecutionOrder** order.
- 4.A **Description** of the rule.

Rule Contexts:

- 1.**AssignCableNoAssignBeforeEdit** - executed during a new Cable Number Assignment or existing Cable Number Edit before the dialog is shown.
- 2.**AssignCableNoAssignAfterEdit** - as above but after the user has clicked Save on the dialog or if **Verbose Cable No Assignment** = false then during the Cable save process.
- 3.**AssignSysNameAssignBeforeEdit** - executed during a new SysName Assignment or existing SysName Edit before the dialog is shown.
- 4.**AssignSysNameAssignAfterEdit** - as above but after the user has clicked Save on the dialog or if **Verbose SysName Assignment** = false then during the **SysName** save process.

Related Topics:

[Project Rules](#) ²⁰³

[Rule Syntax](#) ²⁰⁴

5.3.4 Report Forms

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ >



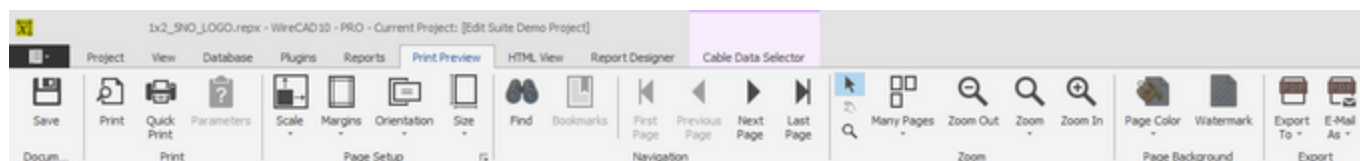
The forms in this section are specific to the **Reporting Engine**.

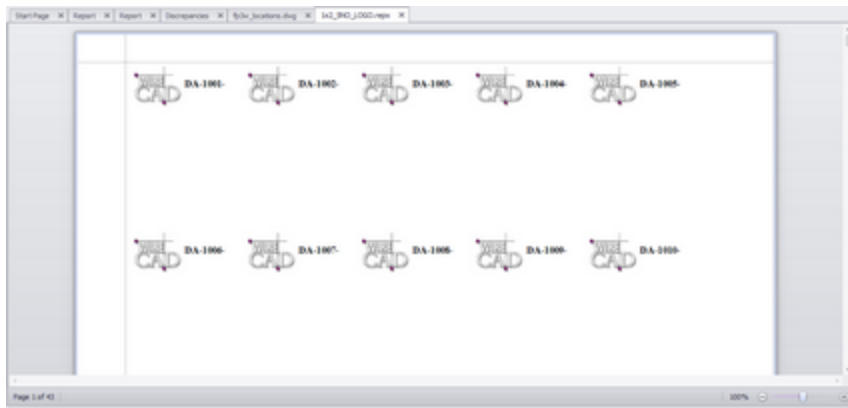
5.3.4.1 Print Preview

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Report Forms](#) ⁵⁵⁰ >



Visible only when the active environment is a report.





Explanation

This is the default view that loads when a report is loaded. From here you can determine the number of pages to print and to which device.

Form Options

•**Save** - Save the rendered document to the native print document format (.prnx). NOTE: this is the rendered document not the report design. The report design can only be saved from the **Report Designer** tab.

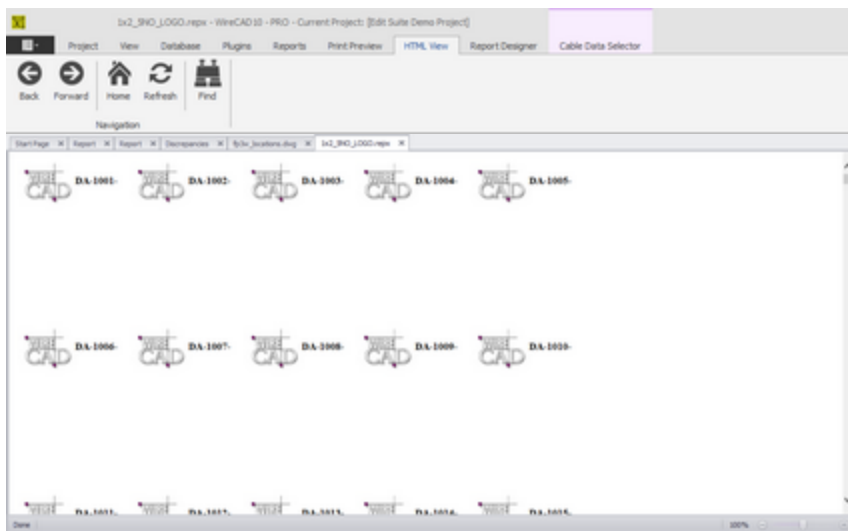
The remainder of the controls are obvious.

5.3.4.2 HTML View

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Report Forms](#) ⁵⁵⁰ >



Visible only when the active environment is a report.



Explanation

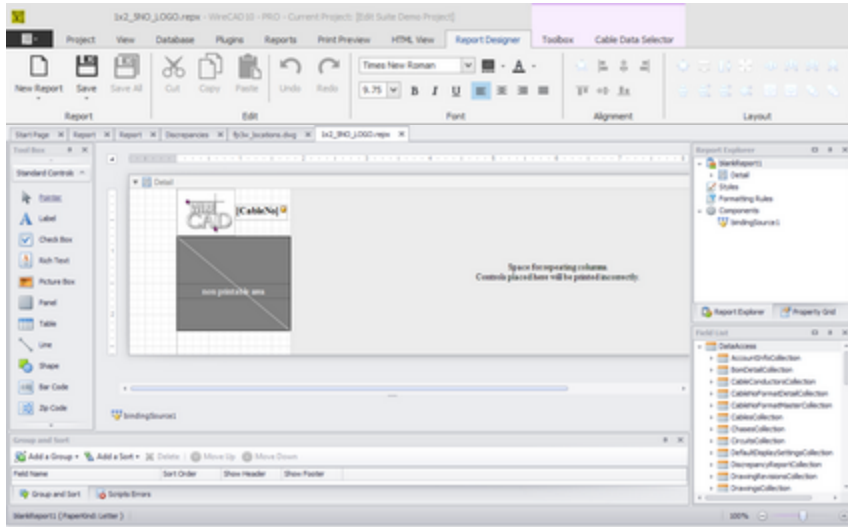
This is the report rendered to html. The controls are self-explanatory.

5.3.4.3 Report Designer

Navigation: [Reference](#) ^[288] > [Forms](#) ^[517] > [Report Forms](#) ^[550] >

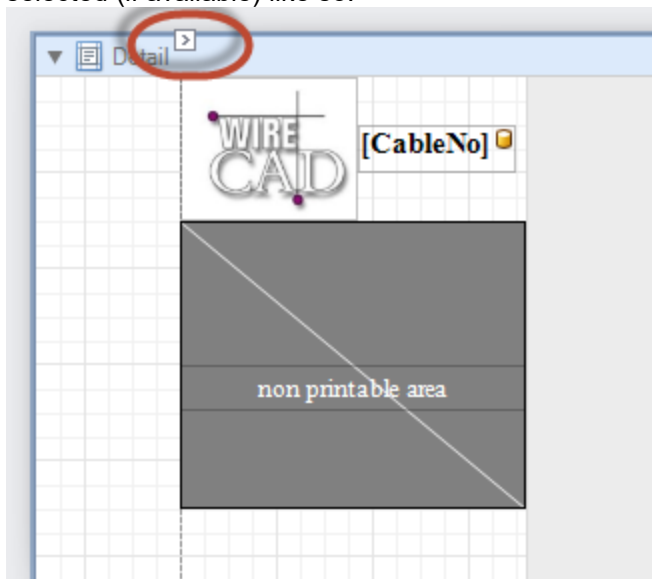


Visible only when the active environment is a report.

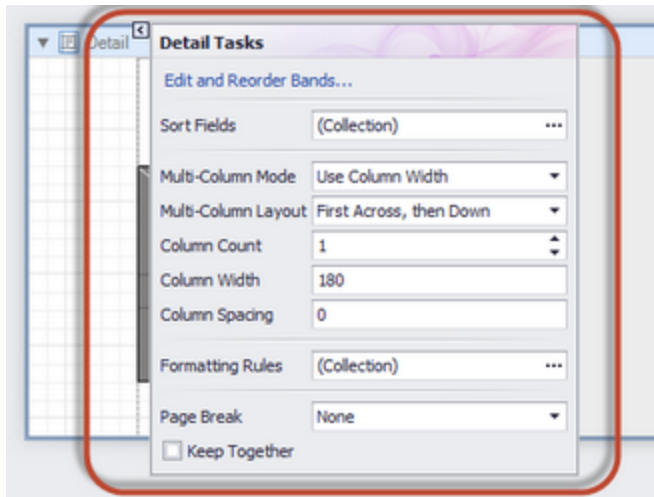


Explanation

Here we design the report. The menus are obvious so we won't bother. What may not be obvious is the Banded Report design and the Smart Tag. Smart Tags will be displayed on entities that are selected (if available) like so:



Smart tags can be expanded to show entity specific settings. Here we see properties specific to the Detail Band:



Related Topics:

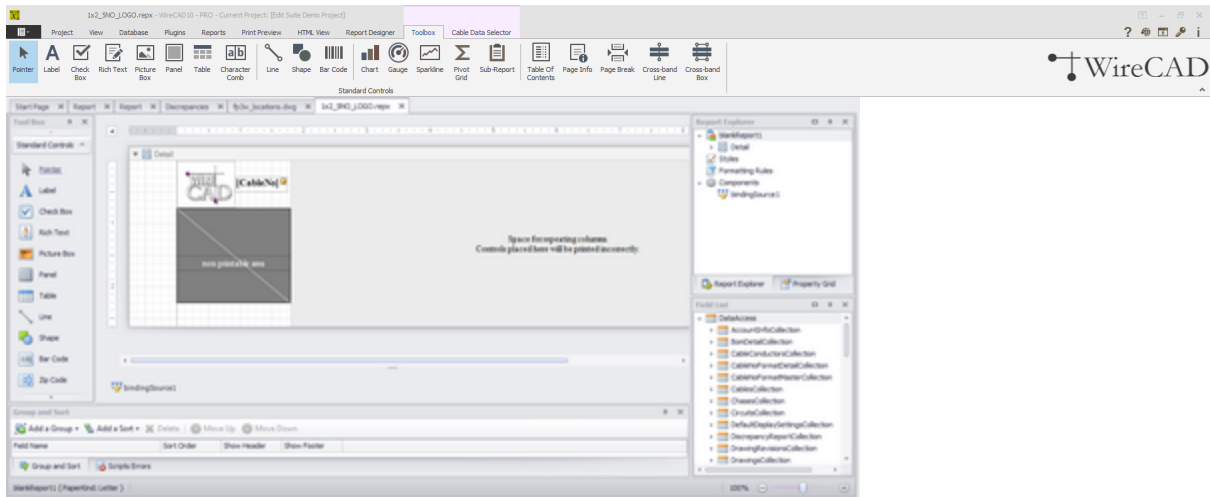
[A discussion on report design can be found here](#) ¹⁷⁰.

5.3.4.4 Toolbox

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ > [Report Forms](#) ⁵⁵⁰ >



Visible only when the Report Designer Tab is active.



Any of these tools can be added to the report under design.

Controls

•**Pointer** - Stop drawing and revert the mouse to a Selection tool.

•**Label** - The most common tool. Can be used to display static text or if the text is held in square braces[]. For example: a label with the caption S:[SRCSys]>[SRCPin] will render S:SomeSrcSys>SomeSrcPin once for every record in the collection.

•**Check Box** - A **Check Box** is used to display True/False or Checked/Unchecked/Indeterminate values in a report.

•**Rich Text** - The **Rich Text** control allows you to display formatted text in your report. It can represent static or dynamic text, or both. In addition to the capability to embed plain text into your report (using the Label control), you may need to display RTF or HTML content as well. WireCAD does not use RTF or HTML natively but there is no reason you could not put formatted text in a user field.

•**Picture Box** - **Picture Box** is used to embed static (stored along with the report definition) or dynamic (obtained from a data source) images into a report. It can display images of various file formats: BMP, JPG, JPEG, GIF, TIF, TIFF, PNG, ICO, DIB, RLE, JPE, JFIF, EMF, WMF.

•**Panel** - The **Panel** control is a container that frames separate report controls to allow them to be easily moved, copied and pasted, and visually unites them in the report preview (with borders or a uniform color background).

•**Table** - The **Table** is used to display tabular information in a report. Note: Table reports should not be confused with hierarchical master-detail reports or cross-tab reports. You can create two tables simultaneously, e.g., one for showing column titles in the Page Header, and the other for showing regular information in the Detail band.

•**Line** - The **Line** control draws a line of a specified direction, style, width and color. It can be used for both decoration and visual separation of a report's sections. Note that the Line cannot cross bands.

•**Shape** - The reporting engine provides you with the capability to embed shapes into your reports. You can simply add any **Shape** control to a report, choose one of the available shape types, and then print or export this report. The following shapes are available:

Arrow
Brace
Bracket
Cross
Ellipse
Line
Polygon
Rectangle
Star

•**Bar Code** - A **Bar Code** object. The supported bar code types are:

1D Barcode Types

Codabar
Code 11 (USD-8)
Code 39 (USD-3)
Code 39 Extended
Code 93
Code 93 Extended
Code 128
EAN 8
EAN 13
GS1-128 - EAN-128 (UCC)
GS1 - DataBar
Industrial 2 of 5
Interleaved 2 of 5
Matrix 2 of 5

2D Barcode Types

ECC200 - Data Matrix
GS1- Data Matrix
Intelligent Mail
PDF417
QR Code

MSI - Plessey
 PostNet
 UPC Shipping Container
 Symbol (ITF-14)
 UPC Supplemental 2
 UPC Supplemental 5
 UPC-A
 UPC-E0
 UPC-E1

•**Zip Code** - The **Zip Code** control transforms its content into a zip code. To specify the text for the control, use the `ZipCode` Text property. Note that the Zip Code control can only display numeric characters and dashes. Other characters are displayed as empty zip boxes.

•**Chart** - The **Chart** control visualizes the series of points using the available 2D chart types or 3D chart types. The chart type is defined via the View property of a series. And, a single chart can display multiple series, if their view types are compatible. A **Chart** contains multiple visual elements (diagram, axes, titles, labels, strips, constant lines, etc.) and, when any of these elements is selected, its properties are shown in the **Property Grid**.

•**Gauge** - The **Gauge** control provides you with the capability to embed graphical gauges into your report.

•**Sparkline** - The **Sparkline** control displays a compact chart that is commonly used to reflect the flow of data for every row in a report.

•**Pivot Grid** - The **Pivot Grid** allows you to create a pivot table, an Excel-inspired data visualization application for multi-dimensional data analysis. Using the **Pivot Grid**, large amounts of data can be summarized and represented in a cross-tabular format that can be sorted and filtered. Also, since the **Pivot Grid** provides customization you can freely change the layout of the report based on your analysis requirements, using simple drag-and-drop operations. It also supports drill-down (to view the underlying data for calculated cells).

•**Sub-Report** - The **Sub-Report** control is used to embed reports into each other; this allows you to create Master/Detail reports (reports with hierarchically linked data).

•**Table of Contents** - The **Table of Contents** control allows you to provide your report with a table of contents that reflects the hierarchical structure of the report bookmarks.

•**Page Info** - The **Page Info** is used to display auxiliary information on report pages, such as date, time, page numbers or user name.

•**Page Break** - The **Page Break** control's sole purpose is to insert a page delimiter at any point within a report. This control is visually represented by a short line, attached to the report's left margin. The **Page Break** control is useful when you need to insert a page break between controls within a report band - for example, to divide subreports, so that the second subreport starts printing on a new page. You can also insert a **Page Break** before or after a specific report band using the `Band.PageBreak` property.

•**Cross-band Line, Cross-band Box** - Cross-band controls are used to draw lines and rectangles through several bands, as opposed to **Line** and **Shape** controls that can be used only within a single band. The following two cross-band controls are available:

Cross-band Line allows for the drawing of vertical lines, which are not restricted to a particular band. For example, it can be used to emphasize a report section consisting of multiple band areas.

Cross-band Box allows for the drawing of rectangles through several bands. It can be used to encompass a report section consisting of multiple band areas.

5.3.4.5 Cable Data Selector

Navigation: [Reference](#)  > [Forms](#)  > [Report Forms](#)  >

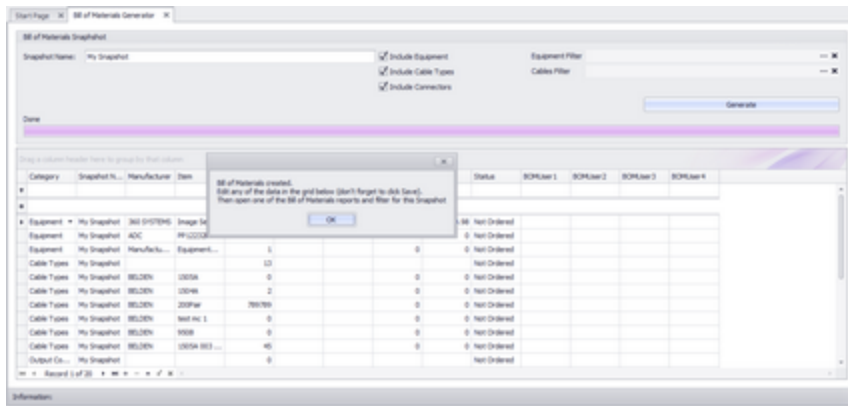


Visible only when the active environment is a report.



Controls

- #### 5.3.4.6 Bill of Materials Generator



Reports > Generate Bill of Materials

Commandline: bom

Explanation

Before a **Bill of Materials (BOM)** report can display data, that data must first be generated. We generate **Bill of Materials** data with a **Snapshot Name**. The report that you run will group by **Snapshot Name**.

You can choose which items to count in the **BOM** and refine your counts by creating filters.

One of the most common misunderstandings with the **BOM** generator is how it deals with cable lengths. The generator sums all lengths of a given **Cable Type**. But what if you have specified a pre made cable of 2 meter length? In this case we don't want to total the length but rather the quantity. To handle pre-made cables take the following steps:

1. Create a new [Cable Type](#)^[487] with the cable length in the name. For example: an off the shelf 2 meter HDMI cable might be named: OTS HDMI 2m. This will show up in the **Cable Type** field of the **Cables** table and further in your **BOM**.
2. When entering the cable **Length** for pre made cables enter the **Length** as 1.

In the **BOM** you will then see a count of your OTS HDMI 2m cables.

Prerequisites

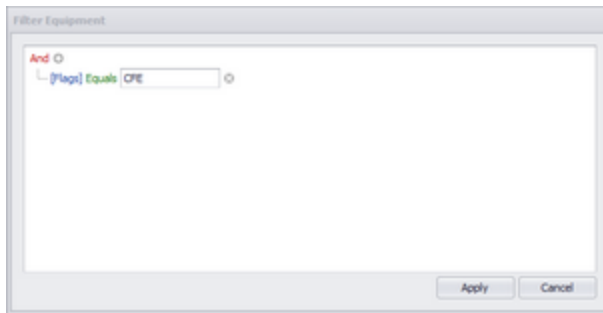
1. You will need an active project with **SysNames** and **Cable** data.

Related Topics:

[Grid Basics](#)^[49]

Form Options

- **Snapshot Name** - Name is something unique that you will remember. Preset by the Default BOM Snapshot Name in the Project Settings panel.
- **Include Equipment, Include Cable Types, Include Connector Types** - What to include.
- **Filter Equipment** - Apply a filter to the set. Click the [...]

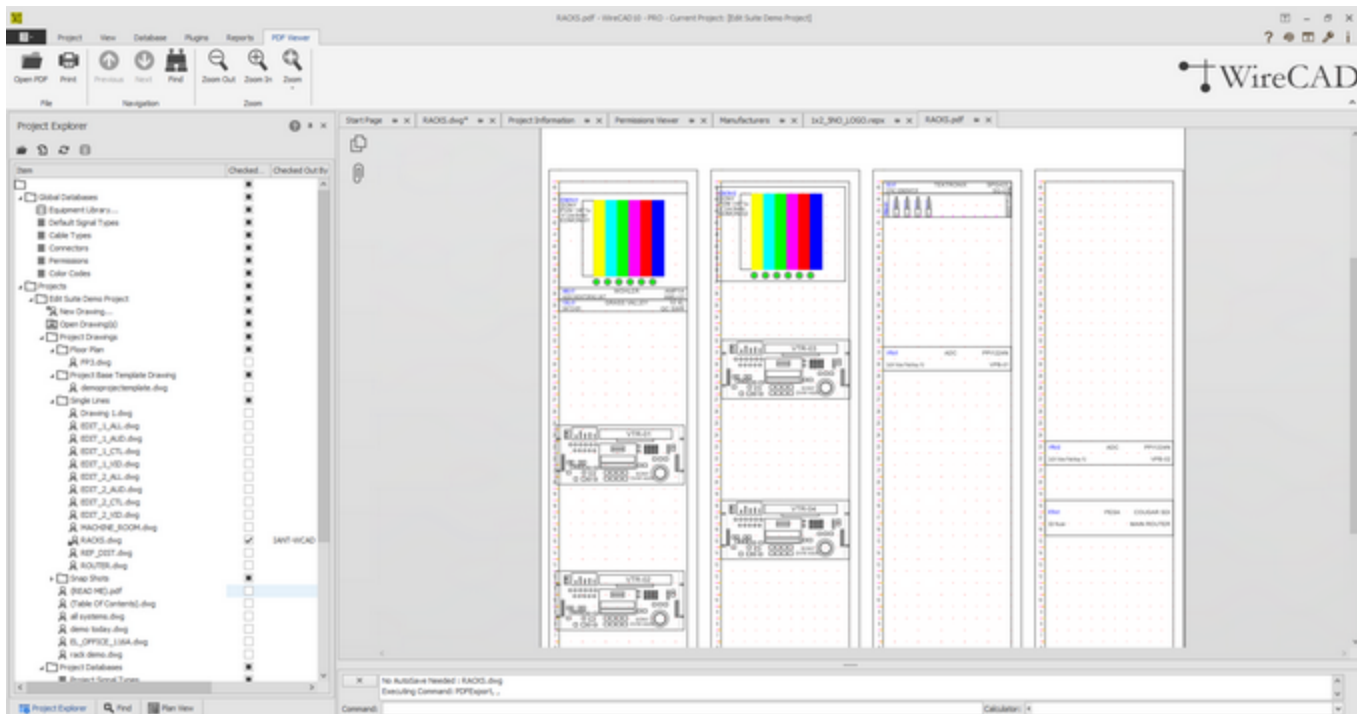


5.3.5 PDF Viewer

Navigation: [Reference](#) ²⁸⁸ > [Forms](#) ⁵¹⁷ >



Visible only when the active environment is a pdf document.



Explanation

This is the built in PDF viewer. Any pdf file placed in the project\Drawings\ folder tree can easily be previewed by double-clicking it in the **Project Explorer**.

Controls

- Open PDF - Open another pdf.
- Print - Shows the [PDF preview dialog.](#)
- Page Navigation
- Find - Shows the find text panel.

5.3.6 Plugins Forms

Navigation: [Reference](#) > [Forms](#) >



The following sections relate to plugins that register forms for editing. These are plugins included with WireCAD. Other plugins may register forms that do not appear here.

5.3.6.1 Translation Manager

Navigation: [Reference](#) > [Forms](#) > [Plugins Forms](#) >



Drag a column header here to group by that column

KeyString	en	fr	CalledBy
WireCAD	WireCAD		
UserSettings.xml	UserSettings.xml		
SupportPaths	SupportPaths		
Group	Group		WireCAD.Translation.wcLayoutLocalizer
Checked	Checked		WireCAD.Translation.wcEditorLocalizer
Unchecked	Unchecked		WireCAD.Translation.wcEditorLocalizer
Loaded Plugins:	Loaded Plugins:		WireCAD.frmSplash.labelControl1.Text
PRO	PRO		WireCAD.frmSplash.lblMode.Text
Copyright 2000-2009 Holbrook Enterprises, Inc.	Copyright 2000-2015 Holbrook Enterprises, Inc.		WireCAD.frmSplash.lblCopyright.Text
Serial Number: - Trial	Serial Number: - Trial		WireCAD.frmSplash.lblSN.Text
Info	Info		WireCAD.frmSplash.lblInfo.Text
Lubing up the engine room	Lubing up the engine room		WireCAD.frmSplash
Lubing up the engine room	Lubing up the engine room		WireCAD.Workspace
AutoSave In: {0}	AutoSave In: {0}		WireCAD.frmMain
(Unassigned Items)	(Unassigned Items)		WireCAD.Translation.wcBarLocalizer
(All Items)	(All Items)		WireCAD.Translation.wcBarLocalizer
Custom {0}	Custom {0}		WireCAD.Translation.wcBarLocalizer
layoutControl1	layoutControl1		WireCAD.ucDrawCables.lcJunctions.Text
Junctions	Junctions		WireCAD.ucDrawCables.layoutControlGroup16.Text
Start Cable	Start Cable		WireCAD.ucDrawCables.simpleButton5.Text
Cable Drawing Properties	Cable Drawing Properties		WireCAD.ucDrawCables.gcPrefs.Text
Preview:	Preview:		WireCAD.ucDrawCables.layoutControlGroup17.Text
True	True		WireCAD.ucDrawCables.rgManyToManyDb.Text
layoutControl3	layoutControl3		WireCAD.ucDrawCables.layoutControl3.Text
X Feeder Distance:	X Feeder Distance:		WireCAD.ucDrawCables.layoutControlItem31.Text
Y Feed Distance:	Y Feed Distance:		WireCAD.ucDrawCables.layoutControlItem22.Text
Hor Extension:	Hor Extension:		WireCAD.ucDrawCables.layoutControlItem24.Text
Output (Source)	Output (Source)		WireCAD.ucDrawCables.gcSrc.Text

Explanation

All visible text strings in WireCAD are contained in dictionary that is editable via the Translation Manager. The current UI culture is queried for a string. If not found the English (en) version is returned.


You can right-click a column header and select the KeyString column to show the base en string that the program searches from.

NOTE: Some forms and dialogs only get their text strings on program start so changes here will not be visible to all areas of the application until you restart WireCAD.

Possible Uses

- 1.Translate WireCAD into a different language.
- 2.Change user column names and labels in the application to aid your process.

See Also

[How to Translate a Form Caption](#)  195

Form Options

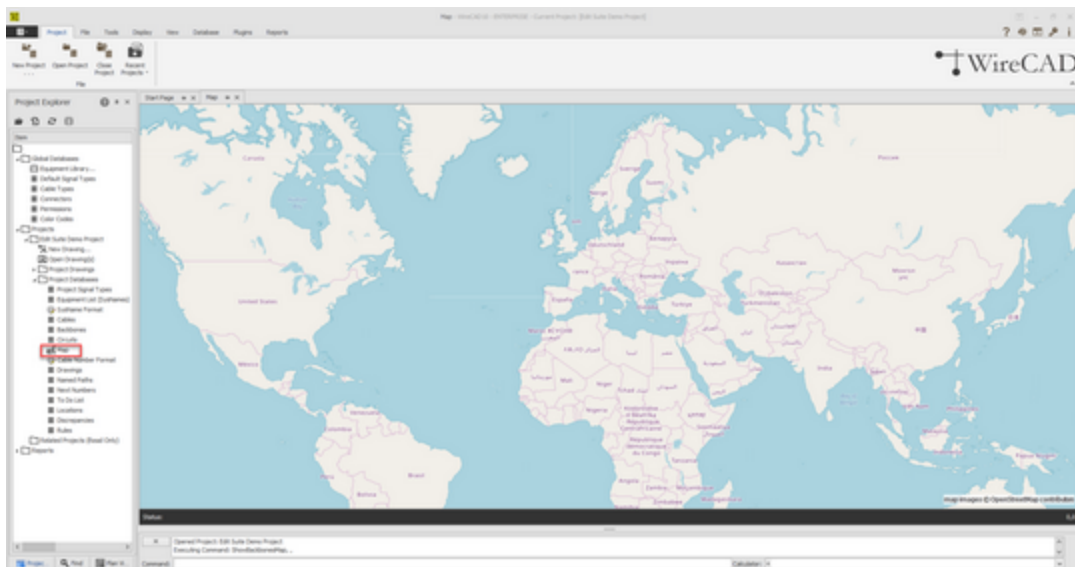
- Current Culture is** - The culture of your machine. If no translation exists, WireCAD defaults to the en (English) language.
- Select Language To Translate** - Selects the language to edit in the right-hand column.
- Show Context** - Select to display a column showing the primary context in which the string or message appears.

5.3.7 Map Tool

Navigation: [Reference](#)  288 > [Forms](#)  517 >



WireCAD Enterprise provides a world map tool.



Explanation

The Map tool within WireCAD can be used to associate your Locations table entries to specific GPS coordinates.

Once the Locations table entries have been associated with corresponding GPS coordinates we can automatically place:

- Location Markers
- Panel Markers

•Backbone Markers

To associate the Location records with specific points on the map use the Tools > [ID_GPS Locations](#) ⁵¹⁴ tool.

5.4 Tool Panels

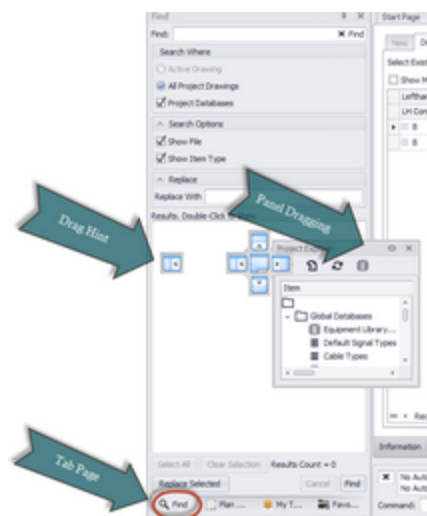
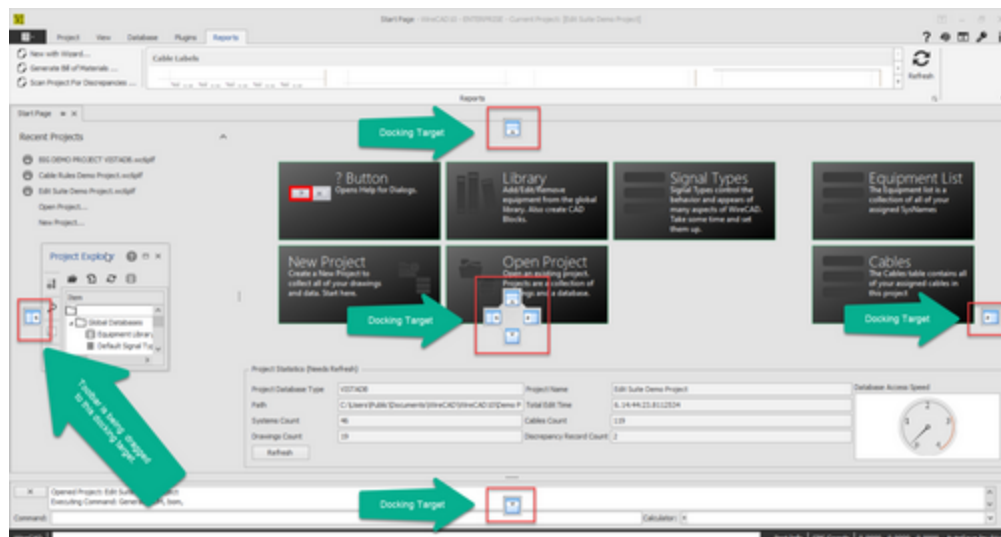
Navigation: [Reference](#) ²⁸⁸ >



Tool Panels differ from Forms and Dialogs in that they are windows that can be docked to the edges of the main application window.

Tool panels are used to present logical groups of controls. Tool panels can be shown/hidden from the **View>Tool Panels>Pane Name** menu.

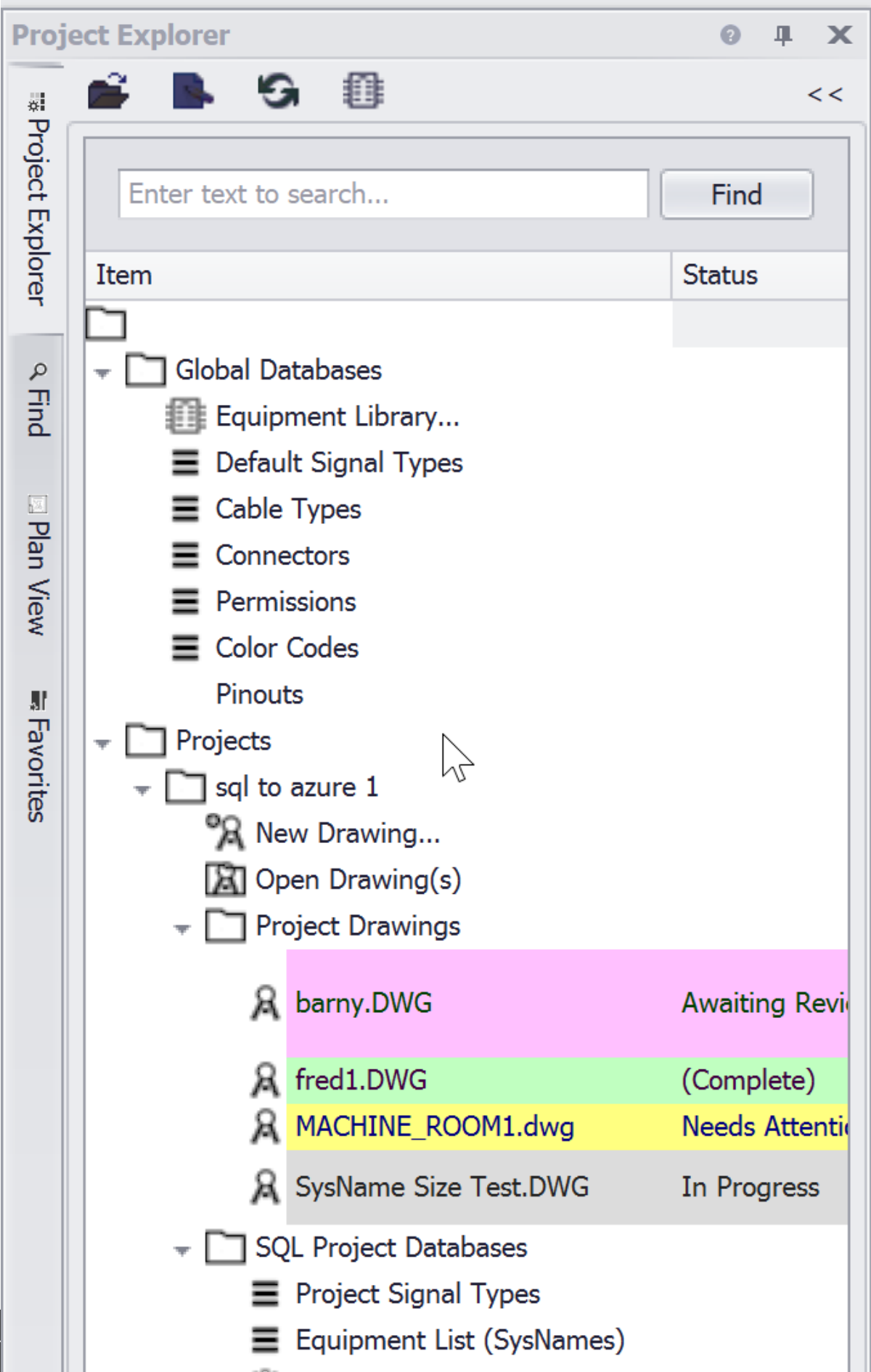
Tool panels can be docked together or undocked. To Undock a tool panel drag its tab page. As you move near a docking target a hint will be shown. Tool Panels will dock based on the hint you click.

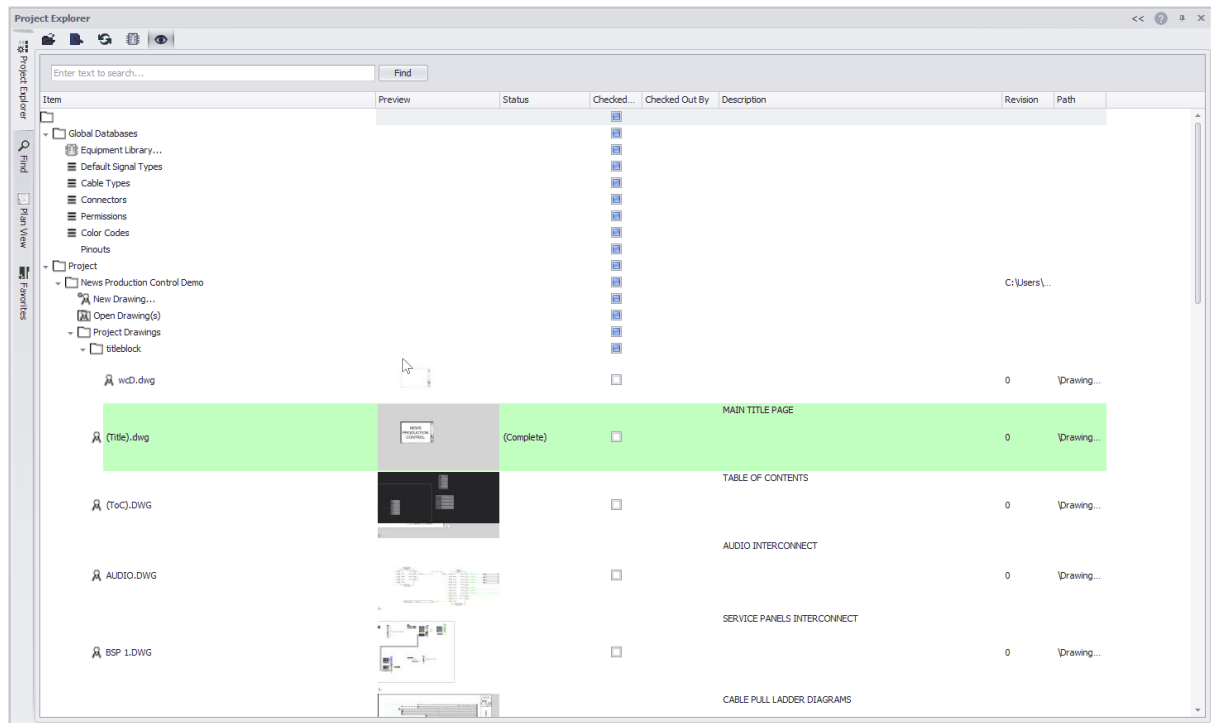


5.4.1 Project Explorer

Navigation: [Reference](#)  > [Tool Panels](#)  >







Project Explorer (Expanded)

View > Tool Panels > Project Explorer**Commandline:** none**Explanation**

This is the main access point to the project. From here you can:

- Open global database grids.
- Open drawing(s).
- Open Project Database grids.
- Open the Map Tool. (ENT only).
- Rename drawings.

Related Topics:
[New Project](#)
[Show Report Mode](#)
Prerequisites

In order to see project related information you will first have to open a project.

Tool Panel Options

Open Project Folder - Show the current project folder in a windows folder browser.

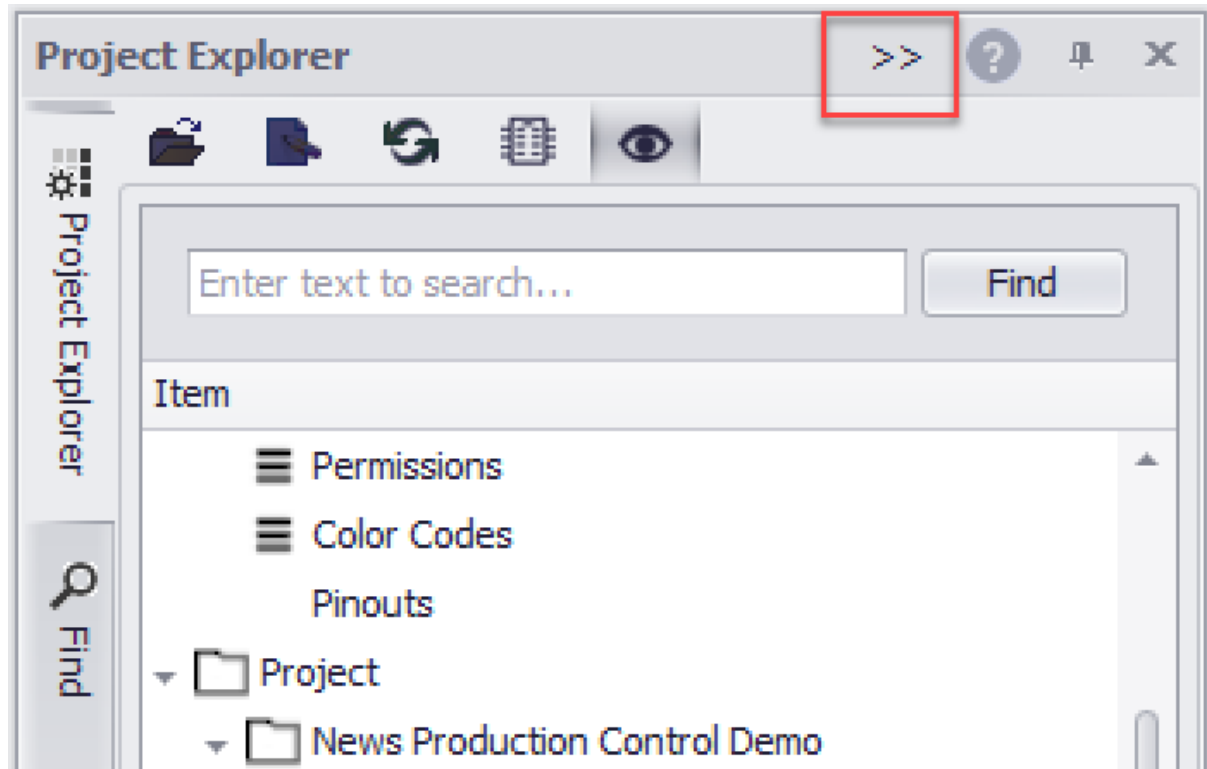
Project Properties - Show the [Project Info](#) form.

Refresh - Refresh the **Project Explorer**.

Equipment Library - Show the [Equipment Library](#) ⁴²⁸.

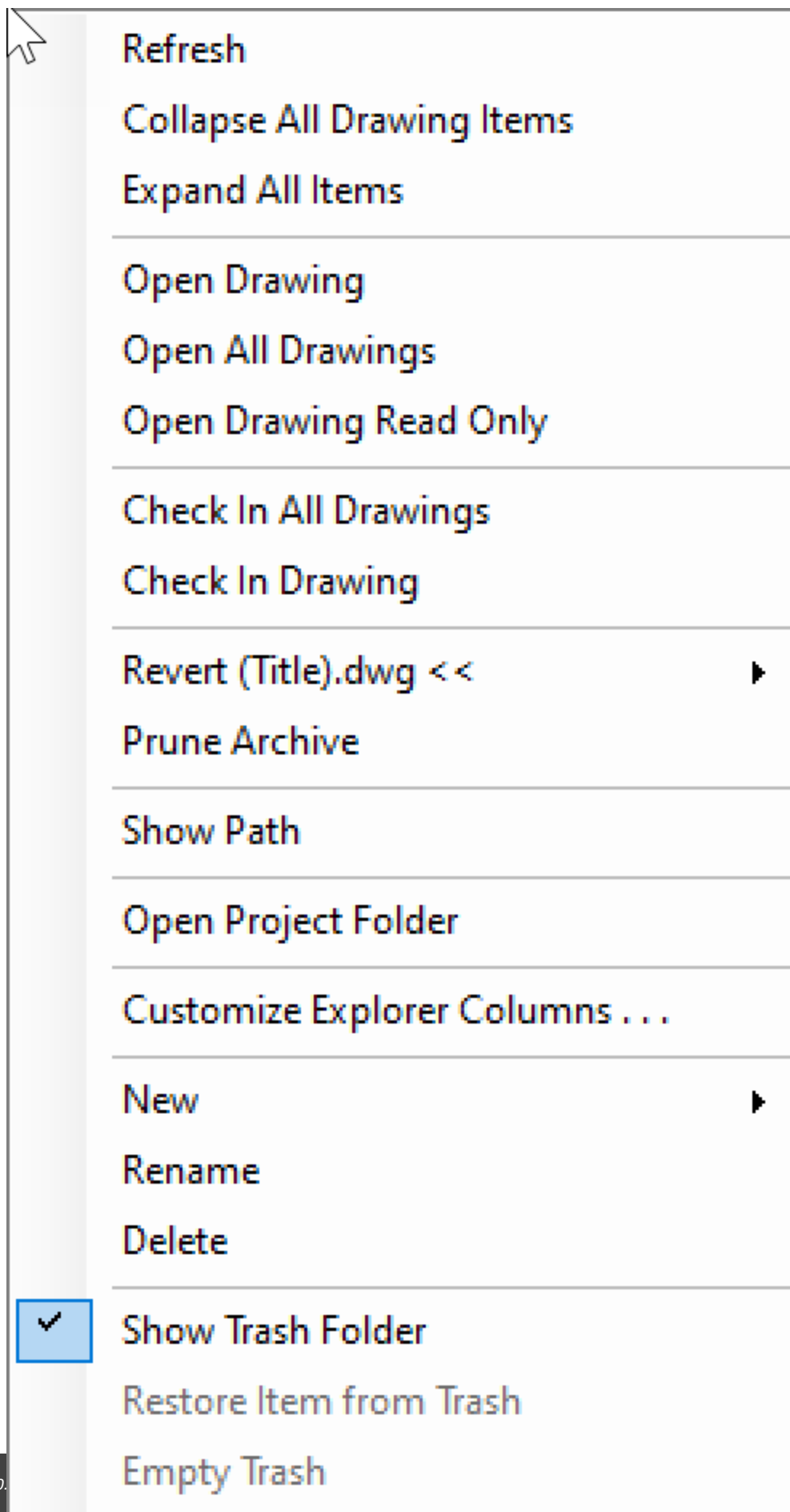
Show Drawing Preview Image - Shows the preview image in the Preview column. Hiding loads the Project Explorer faster.

Expand the Project Explorer tool panel to the right to see additional columns that you can view and edit.



Project Explorer Expand Button

Tool Panel Context Menu Options



Menu Items

Refresh - Reload the Project Explorer

Collapse All Drawing Items - Collapse all the drawing nodes to display one drawings folder.

Expand All Items - Show all items.

Open Drawing - Open the focused node (if it is a drawing).

Open All Drawing - Open all project drawings.

Open Drawing Read Only - Open the focused node (if it is a drawing) - read only.

Check In All Drawings -

Check In Drawing -

Revert (Title).dwg << > -

Prune Archive -

Show Path -

Open Project Folder -

Custom Explorer Columns... -

New>Folder -

New>Drawing ... -

New>Report -

Rename -

Delete -

Show Trash Folder -

Restore Item from Trash -

Empty Trash -

Remove All Empty Folders -

5.4.2 Draw Cables

Navigation: [Reference](#) [288] > [Tool Panels](#) [561] >



View > Tool Panels > Draw Cables**Commandline:** none***Explanation***

The Draw Cables tool panel is only active when the current environment is a drawing. This tool allows you draw several different types of cables:

- [One-to-One](#) ⁸⁶ - One output to one input.
- [One-to-Many](#) ⁹⁰ - One outputs to many inputs.
- [Many-to-One](#) ⁹³ - Many outputs to one input.
- [Many-to-Many](#) ⁹⁴ - Many outputs to many inputs.
- [Terminal on the Left](#) ⁹⁵ - Drag a Terminal into the drawing and drop it on a connection point in the drawing. This will place the terminal on the left hand side of the block.
- [Terminal on the Right](#) ⁹⁶ - Drag a Terminal into the drawing and drop it on a connection point in the drawing. This will place the terminal on the right hand side of the block.
- [Working with Pointers](#) ⁹⁸.

Prerequisites

Requires and open drawing.

Related Topics:

[Draw Cables](#) ⁸²

5.4.3 Drawing Properties

Navigation: [Reference](#) ²⁸⁸ > [Tool Panels](#) ⁵⁶¹ >



Provides access to the drawing properties. If no selection set exists in the drawing the general document properties are presented. If a selection exists then the common properties of all items in the selection set are displayed for edit. You may also select a single item from the selection set and edit all of its properties.

Prerequisites

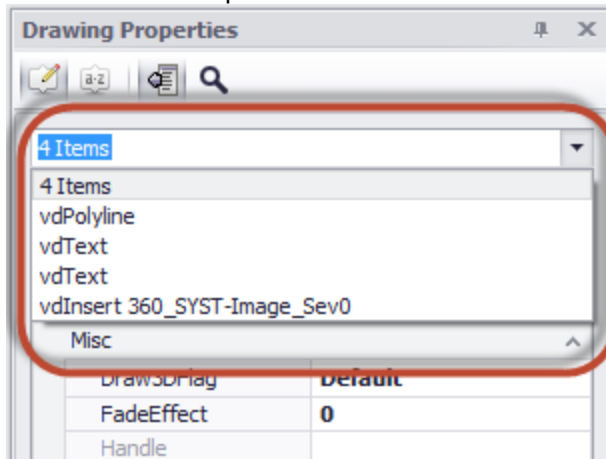
An open drawing.

Related Topics:

None

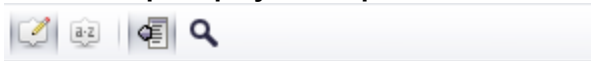
Tool Panel Options

•**Selected Object** - This combo box displays the selected object and allows you to select individual items from the drop down.



•**Property grid** - The properties displayed here will all have property editors that are specific to the entity type.

•**Sort/Group/Display Descriptions/Search toolbar buttons**



5.4.4 Find

Navigation: [Reference](#) ²⁸⁸ > [Tool Panels](#) ⁵⁶¹ >





View > Tool Panels > Find and Replace
Commandline: none

Explanation

Find and replace text found in any drawing and in editable fields of the project databases. The found list will tell you the context in which the searched text appears.

NOTE: found text may occur in an invisible attribute.

Changing a port name on a device using the Find and Replace tool requires that you change the text in the following contexts:

- The visible attribute.
- The invisible CP_IN or CP_OUT attribute string. This is a pipe delimited string that contains the name|conn|type. You must change the name portion of this.
- Any records that exist in the Project Cables table that reference that port.

Prerequisites

An active project

Related Topics:
None

Tool Panel Options

- Find** - The text to search for.
- [X]** - Clear the Find text.
- Search Active Drawing** - Only look in the active drawing.
- Search All Project Drawings** - Search every project drawing.
- Show File** - Show file name if any of the context in which the item is found.
- Show Item Type** - Is it a text entity in a drawing or a field in a database.
- Results** -



- [Select All] - Selects all items in the results window.
- [Clear Selection] - De-selects all items in the results window.
- Replace With - The text to replace the found text with.
- [Replace Selected] - Initiate the replacement.
- [Cancel] - Cancels the current search.
- [Find] - Initiate the search.

5.4.5 Plan View

Navigation: [Reference](#) [288] > [Tool Panels](#) [561] >



View > Tool Panels > Plan View

Commandline: none

Explanation

This tool panel provides three tabs. Each tab houses a gallery of **Plan View** blocks. The tabs are:

- All Plan View Files** - Enumerates the %BLOCKS%\Plan View Files\ folder tree presenting a preview of each dwg file. Each sub folder will be added as a gallery group and items in that folder added to the group.

- **Project Plan View Items** - Provides a gallery item for each **SysName** in the **Equipment List**. If the SysName's equipment definition has an empty **Plan View File**, the project **Default Plan View File** is presented.
- **Equipment Library Plan View Files** - Shows all **Equipment Library** items that have data in the **Plan View File** field.

Prerequisites

An open project.
An open drawing.

Tool Panel Options

- **Place with Rotation and Scale** - If checked then when placing the selected item in the drawing you will be prompted to scale the X, Y, Z and set the Rotation angle.
- **Plan View Galleries** - Click an item to add it to the drawing. Then place (and scale/rotate).

5.4.6 Favorites

Navigation: [Reference](#) [288] > [Tool Panels](#) [561] >



View > Tool Panels > Favorites

Commandline: none

Explanation

Often times we find ourselves creating the same block or circuit configuration over and over. These times are a great candidate for a Favorite. Creating a Favorite stores the item(s) as a block in the OS in one of three places:

- **Global** - Saves in the %BLOCKS%\Favorites path and may be visible to other user with the same support path.
- **User** - Saves to your user profile. Visible only to you.
- **Project** - Saves in the Project\Favorites folder for use by anyone with access to the project.

Prerequisites

A drawing with the block or configuration you want to save as a Favorite.

A drawing into which you wish to place a Favorite.

Tool Panel Options

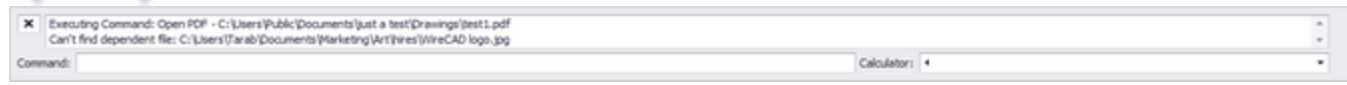
•**Explode** - If you have grouped a system together that includes multiple blocks and cables into a single block. It will need to be exploded back one level to expose the individual WireCAD objects.

•**Scale**

•**List** - Double-click to add to the current drawing.

5.4.7 Command Line

Navigation: [Reference](#) ²⁸⁸ > [Tool Panels](#) ⁵⁶¹ >



View > Tool Panels > Command Line

Commandline: none

Explanation

The **Command Line** interface to the application. Here you can view a history of commands and enter commands directly. All commands entered must be executed by clicking the **[Enter]** key.

Prerequisites

None

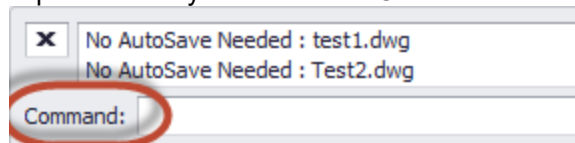
Related Topics:

[My Space Bar is my Enter Key](#) ³⁵⁴

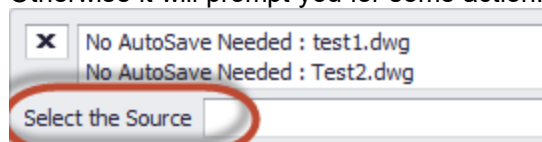
Dialog Options

•**Command History** - What went before.

•**Command Prompt** - What the application is expecting of you. If no job is running or no input is expected from you it will read: Command:



Otherwise it will prompt you for some action:



•**Command Line** - Enter the command here.

•Calculator

Writing Plugins

6 Writing Plugins



INTRODUCTION The examples can be found at: C:\users\public\documents\WireCAD\WireCAD9\WireCAD SDK\Examples Welcome to the WireCAD SDK examples. These examples show you how t

INTRODUCTION

The examples can be found at:

C:\users\public\documents\WireCAD\WireCAD9\WireCAD SDK\Examples

Welcome to the WireCAD SDK examples. These examples show you how to interact, via your own code, with WireCAD. These examples assume that you understand the basics of C# programming syntax and structure (the examples are in C# but you may use any .NET programming language that you are comfortable with).

If you do not understand it don't let that deter you. It is really very easy to understand. Microsoft has tons of examples. Just get on the web.

If you have WireCAD SDK specific questions email support@wirecad.com along with as much code as you can and an explanation of what you are trying to do. We will try to help. If you need more help or if you are developing a commercial plugin for WireCAD you may want to consider purchasing custom programming services from us. Please call the office for more information.

Getting Started

You will want to download a .net ready integrated development environment(IDE) like Visual Studio for c# or SharpDevelop. Visual Studio has free versions. SharpDevelop is an open source.

We recommend the Visual Studio for the ability to attach a debugger to your running process. Also use the **WireCADv9TemplateInstaller.vsix** to install several starter projects in Visual Studio that you can use from the **File>New Project** wizard.

The first place to go with any programming effort is the ubiquitous **"Hello World"** example in the Basic folder.

NOTE: there are two types of WireCAD plugin.

1. The automatically discovered (AD) plugin that loads silently and may or may not interact with the user.
2. Plugins that require a WireCAD .wpi manifest file that describes where the plugin can be found, how to call it, what icon to display and on which toolbar or menu, etc. Both plugins implement the WireCAD.IPlugin interface.

AD plugins are named YourPluginName.**Plugin.Dll** and are placed in the C:\users\public\documents\WireCAD\WireCAD9\bin folder.

Standard WireCAD plugins have no naming requirements but must be accompanied by a **.wpi** file in the C:\users\public\documents\WireCAD\WireCAD9\Plugins\Active Plugins folder.

To create a **wpi** file you can use the utility in **Plugins>Plugin Manager [New PI Info]**

Take some time to read through the code. The interface is very simple, yet you can access most of the WireCAD object model including drawing, data access, grids, GUI, reports, etc.

You can also create your own forms and functions to interact with the WireCAD objects.

THIRD PARTY LIBRARIES

WIRECAD USES THIRD PARTY LIBRARIES THAT MAY REQUIRE YOU TO PURCHASE DEVELOPER COPIES. PLEASE DO NOT IGNORE THIS STATEMENT. THESE ARE GREAT LIBRARIES AND IF YOU ARE PLANNING TO DEVELOP A COMMERCIAL PLUGIN FOR WIRECAD YOU WILL WANT

TO HAVE THEM. If you are developing a small project or line of business function for your own use you may not need to purchase licenses. Here is an example:

If you want your plugin to have the same look and feel, or create a custom form with the same gridview or treeview as WireCAD you will need to purchase the Developer Express library. If you are developing a custom form

that needs to show its own drawing engine you will need to purchase the VectorDraw developer framework. If, on the other hand, you are able to use the WireCAD baseforms object, are happy with the WinForms controls and can develop using the WireCAD drawing engine then you can get started without spending a dime.

BUILDING YOUR FIRST PLUGIN

STEPS:

1. Open your IDE
2. Click **File>Open Project** or Solution and browse to the basic hello world.csproj project file in the examples\basic\hello world folder
3. Click **Build>Build Solution** (Ctrl+Shift+B) to build the project.
4. Check that the file C:\Users\Public\Documents\WireCAD\WireCAD9\bin\helloworld.plugin.dll exists. If it does not then you need to check the project build path in the project properties section and point it to the WireCAD bin folder above.

If you get any reference errors check **Project>Hello World Properties... [Reference Paths]** section

points to C:\Users\Public\Documents\WireCAD\WireCAD9\bin\. If it does not then modify it and make sure that your reference errors disappear.

5. You now need to force WireCAD to load the shiny new plugin.

Either relaunch WireCAD (tedious) or click **Plugins>Plugin Manager ...** then click the **[Rescan and Reload All Plugins]** button. Make sure that the helloworld.plugin appears in the Auto Discovered Plugins list.

6. Now enter **hw** in the commandline and click **[Enter]**. This will launch the command that the plugin registered in its Load() method.

7. Make some changes and click **Build>Build Solution** (Ctrl+Shift+B) to build the project again.

If your changes are outside the load() method you will not need to force a reload.

Just call the command again by typing **hw** in the commandline and click **[Enter]**.

There you have it. You have just built your first WireCAD plugin. Now go forth and do more!

DEBUGGING YOUR PLUGIN

Invariably you will need to stop the execution of your plugin and examine the variables and logic structure to figure out why it is not working as you expect.

The following instructions assume you are using the Visual Studio IDE for C#. Further, that you have built your project and made sure it is loading in the WireCAD plugin framework by looking at the loaded plugins list in **Plugins>Plugin Manager...**

NOTE: The following will work for anything other than the initial Load() method. To debug that read on.

Steps:

1. Open your project in the Visual Studio IDE.
2. Run WireCAD.
3. In VS click **Debug>Attach to Process...**
4. Select WireCAD9.exe.
5. Place a breakpoint in the code where you want to stop execution. Do this by clicking in the gutter of the text editor(very left hand edge. A red dot will appear).
6. Execute your command. The debugger will pause at the line and highlight the paused line. In the interface you will find two very helpful tool windows - Locals and Autos. These windows list the variables in play at the time of the breakpoint.
7. On the IDE Debug menu you will find commands to continue execution:
 1. **Continue** - runs from the break point until it hits the next breakpoint if any.
 2. **Step Into** - continues execution of the next line. If the next line calls another function you will be stepped into that function.
 3. **Step Over** - continues execution of the next line. If the next line calls another function that function will execute in its entirety and you will step to the next line in the current code block.

While you cannot edit and continue you will be able to understand the changes you need to make to fix the issue you are having.

8. Stop the debugger. Fix your code. Lather, rinse, repeat.

DEBUGGING THE LOAD() METHOD OF THE IPLUGINCORE INTERFACE

The load method or the HasPermissionToRun methods of the IPluginCore interface require some special consideration if we are to debug them. The reason for this is that the WireCAD loading process will call these methods before we have a chance to attach the debugger to the process.

This section will outline two approaches:

1 - PAUSE EXECUTION USING MESSAGEBOX

Place a `MessageBox.Show("Pause");` command in the **Load** or the **HasPermissionToRun** method.

This modal dialog will pause execution for you to attach the debugger as shown above.

like this:

```

    /// <summary>
    /// This is called when the plugin is loaded at application start
    /// </summary>
    /// <param name="ws">Singleton WireCAD Workspace object</param>
    public void Load(IWorkspace ws)
    {
        MessageBox.Show("Pausing plugin load so you can attach the debugger");
        ..... Your code here
    }

```

2 - START WIRECAD IN THE DEBUGGER PROCESS

The other approach is to start WireCAD in the debugger from your project. While this seems like a good approach it suffers from a major drawback. When you want to fix your code you must stop the debugging session. Since WireCAD is running in the same process it will stop as well. Thus requiring you to start WireCAD every time you want to make a fix.

Steps:

1. Open your project in the IDE.
2. Close WireCAD.
3. Open the Project Properties window - **Project>YourProjectName Properties...**
4. Select the **Debug** section.
5. Select the **Start External Program** radio button.
6. Browse to the WireCAD9.exe at: C:\Users\Public\Documents\WireCAD\WireCAD9\bin\WireCAD9.exe
7. Place your breakpoint.
8. Click **Debug>Start Debugging** (F5).
This will start the debugging session by launching WireCAD first. As WireCAD loads it will eventually come to your **Load()** method and hit your breakpoint.

When you are finished with your examination close WireCAD and let it shutdown. Ending the debugging session with the **Debug>Stop Debugging** (Shift+F5) acts like Kill Process and will not allow WireCAD to store session state.

ADDITIONAL RESOURCES

C# is the language of choice for developing in WireCAD. There are tons of resources online to learn this language.

For WireCAD specific issues:

www.wirecad.com/wiki

support@wirecad.com

Please don't submit support tickets for SDK issues. Use the support email.

Developer Express is the library we use for many of the UI elements. They have a brilliant web site with documentation and support. If you are going to go deep on WireCAD development you will probably want to purchase a license.
www.developerexpress.com

VectorDraw is the drawing engine we use. While their site is minimal they do offer registered developers lots of sample code.
www.vdraw.com

for a full api listing visit:
 for the main WireCAD9.exe:
<https://www.wirecad.com/api/90/exe/index.html>

for the global data access layer:
<http://www.wirecad.com/api/90/globaldal/index.html>

for the project data access layer:
<http://www.wirecad.com/api/90/projectdal/index.html>

6.1 Hello World

Navigation: [Writing Plugins](#)  >

Every programming effort starts with the ubiquitous "Hello World" example. This example is meant to provide just enough information to show the framework and produce an output.

Please note that in C# comments are preceded by //.

```

////////////////////////////////////
//WireCAD Plugin
//Contents:
//    Basic WireCAD Plugin Framework
//
//Instructions:
//    1.This project assumes that you installed WireCAD in the default
//       location. If you did not, you will need to change the reference
//       path and the build path to that of your install path
//       ..\WireCADx\bin folder. You can do that by clicking:
//       Application Menu > Project Options [Reference Paths] Reference path
//       Application Menu > Project Options [Compiling Tab] Output Path
//    3.Add your code and build
//    4.Create a wpi file (from within WireCAD click Plugins>Plugin Manager [New]) that
//       points to your assembly and method and
//       place it in the ..WireCADx\bin\plugins folder
//
////////////////////////////////////
//Explanation:
//This heloworld example demonstrates a number of different WireCAD SDK
//concepts. This is WireCAD Automatically Discovered (AD) plugin and
//therefore does not require a .wpi file in the /plugins folder. As such,
//it will load silently and can only be executed from the commandline since
//it does not add any other GUI elements.
//
//TESTING:
//First familiarize yourself with the code below and try to understand
//what will happen before testing.
//1. Build the project and ensure that the helloworld.plugin.dll is located
//    in the ..WireCADx\bin folder.
//2. if WireCAD is running click Plugins>Plugin Manager[Rescan and Load Plugins]
//3. Type hw into the WireCAD commandline.

using System;
using System.Collections.Generic;
using System.Linq;
using System.Text;
using System.Windows.Forms;

```

```

using VectorDraw.Professional.vdFigures;
using VectorDraw.Professional.vdObjects;
using WireCAD;
using WireCAD.Interfaces;
using WireCAD.Translation;

namespace hello_world
{
    public class MyPlugin: IPluginCore
    {
        #region Fields
        //Place your field level variables here
        CommandInfo ci = null;
        #endregion

        #region Properties
        //Place your Properties Here

        #endregion

        #region IPluginCore Members

        /// <summary>
        /// Called before plugin is loaded to make sure that this plugin
        /// has permission to run at this application mode level and
        /// for this person(Identity)
        /// </summary>
        /// <param name="ws">The Singleton Workspace object</param>
        /// <param name="id">Current user identity</param>
        /// <returns>should return true if the plug can load</returns>
        public bool HasPermissionToRun(IWorkspace ws, Identity id)
        {
            return true;
        }

        /// <summary>
        /// This is called when the plugin is loaded at application start
        /// </summary>
        /// <param name="ws">Singleton WireCAD Workspace object</param>
        public void Load(IWorkspace ws)
        {
            //We pass an IWorkspace object exposing most if the WireCAD
            //object model

            //on load we will register a commandInfo object with our
            //commands so that our static method can be invoked from
            //the command line in WireCAD
            ci = new CommandInfo();
            //The name of our dll
            ci.Assembly = "helloworld.plugin.dll";
            //The NameSpace and Class of our function
            ci.NameSpaceAndClass = "hello_world.MyPlugin";
            //Our function's name
            ci.MethodName = "HelloWorld";
            //The long descriptive name of our function
            ci.CommandLongName = "Hello World Demo";
            //An alternative name for our function. You can type this text into the WireCAD
            command line
            //to call the function
            ci.CommandAlt = "hello";
            //The shortcut
            ci.ShortCut = "hw";
            //this registers the command so that the commandline knows how to parse the
            information
            //and call our function
            ws.Commands.RegisterCommand(ci);
        }
        /// <summary>

```

```

    /// Unload code for your plugin
    /// </summary>
    /// <param name="ws"></param>
    public void Unload(IWorkspace ws)
    {
        //here we place any code to unload our plugin.

        //Unregistering the commandInfo prohibits the command from
        //being persisted. This is more a development function.
        //once you are ready to distribute your plugin you will
        //probably want your user to be able to save his own shortcuts
        //and therefore not unregister the command.
        ws.Commands.UnRegisterCommand(ci);
    }

#endregion

#region Static Methods
    /// <summary>
    /// Static method that can be called from the WireCAD command line
    /// It's a good idea to rename this to something meaningful
    /// </summary>
    /// <param name="ws">WireCAD is expecting to find this parameter</param>
    public static void HelloWorld(Workspace ws)
    {
        //This hello world function demonstrates a number of different
        //areas of the WireCAD SDK

        //this is a winForms messagebox
        MessageBox.Show("Hello World");

        //now we'll show an instance of form1(defined elsewhere in this project);
        Form1 f = new Form1();
        f.ShowDialog();

        //now let's send a message to the Command Line History
        ws.MainForm.CommandLine.AppendHistory("Hello World");

        //let's check to make sure that we have an open drawing
        if(!Commands.IsActiveDrawing(ws,true))
        {
            //no active drawing so return
            return;
        }

        //Get some user input on the next step
        if(DialogResult.No == MessageBox.Show(
            "Would you like us to add some text to the active drawing?",
            "WireCAD SDK",MessageBoxButtons.YesNo)) return;

        //First we create a vdText object
        vdText text = new vdText();
        //register it with the active document
        text.SetUnRegisterDocument(ws.ActiveDrawing);
        //give it the document defaults
        text.setDocumentDefaults();
        //Set the string
        text.TextString = "hello world";
        //locate it in the coordinate space
        text.InsertionPoint = new VectorDraw.Geometry.gPoint(0,0,0);
        //Set the textHeight
        text.Height = .25;
        //Set the color
        vdColor colorRed = new vdColor();
        colorRed.SetUnRegisterDocument(ws.ActiveDrawing);
        colorRed.Palette = ws.ActiveDrawing.Palette;
        colorRed.FromRGB(255,0,0);

        text.PenColor = colorRed;

        //alternately you could do this
        //text.PenColor.ColorIndex = 1;

        //our text will be added to the drawing on the ActiveLayer and with

```

```

//the ActiveTextStyle if you want to change those you can by setting
//those properties on the text object

ws.ActiveDrawing.ActiveLayout.Entities.AddItem(text);

//now refresh the drawing
ws.ActiveDrawing.Redraw(true);
//alternately you can just invalidate the text object(less expensive)
//like this:
//text.Invalidate();

}

#endregion
}
}

```

6.2 Getting Started

Navigation: [Writing Plugins](#) 576 >



The following assumes that you have installed either Visual Studio 2017 or 2019.

Run the **WireCADv10.0TemplatesInstaller.vsix** template installer from C:

\Users\Public\Documents\WireCAD\WireCAD10\WireCAD SDK\Visual Studio Templates
Installer\WireCADv10.0ProjectTemplatesInstaller.vsix.

Launch Visual Studio

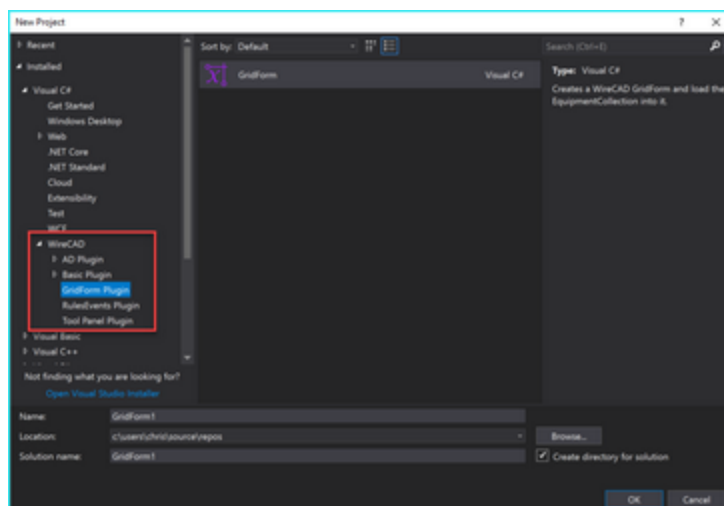
Study the examples by loading the All Project solution:

C:\Users\Public\Documents\WireCAD\WireCAD10\WireCAD SDK\Examples\All Projects.sln

When you want to create a new project to start your own plugin follow these steps:

Your First Plugin Project

1.Click [File] > [New Project]



2.Select one of the C# WireCAD Project Templates

3.Name it something meaningful.

4. Click **[OK]** and a new solution will be created for you with all the necessary references and interface files.

5. Now start coding!

6. Click **F5** to build your project and make sure that it builds and copies to c:\users\public\documents\WireCAD\WireCAD10\bin\YourPluginName.dll. If it does not build check to make sure that the **Project Properties - Reference Paths** section has a pointer to the above path.

7. If you are using a wpi file make sure that it is edited to point it to your dll and method and that you have copied it to c:\users\public\documents\WireCAD\WireCAD10\Plugins\Active Plugins folder. Relaunch WireCAD or click **[Rescan and Reload]** from the WireCAD Plugin Manager to make WireCAD load your plugin.

6.3 Registering Your Plugin

Navigation: [Writing Plugins](#)  576 >



Menu: **Plugins>Plugin Manager[Add/Edit PI Info]**

Default command line shortcut: none

Assuming your plugin requires user input to launch it, ie. it does not respond to an event. You will want to register your command with the application.

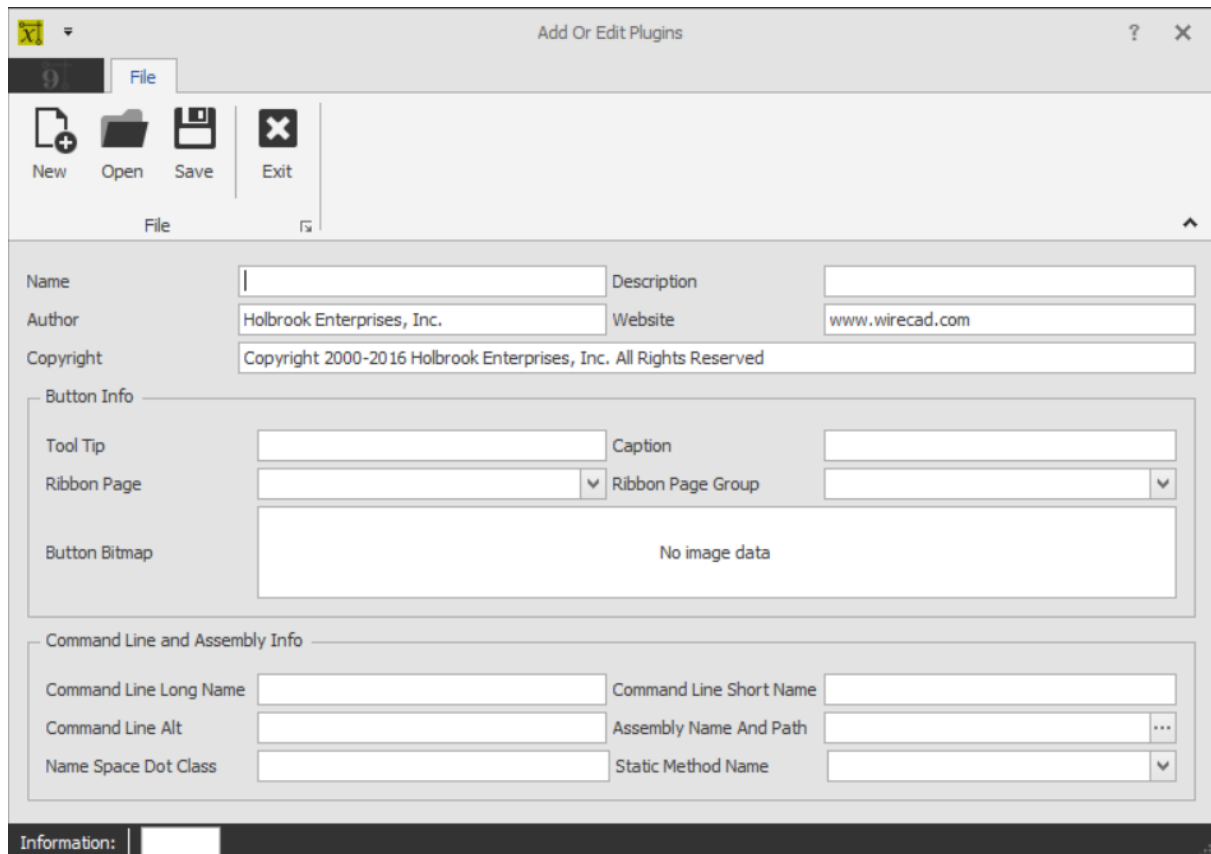
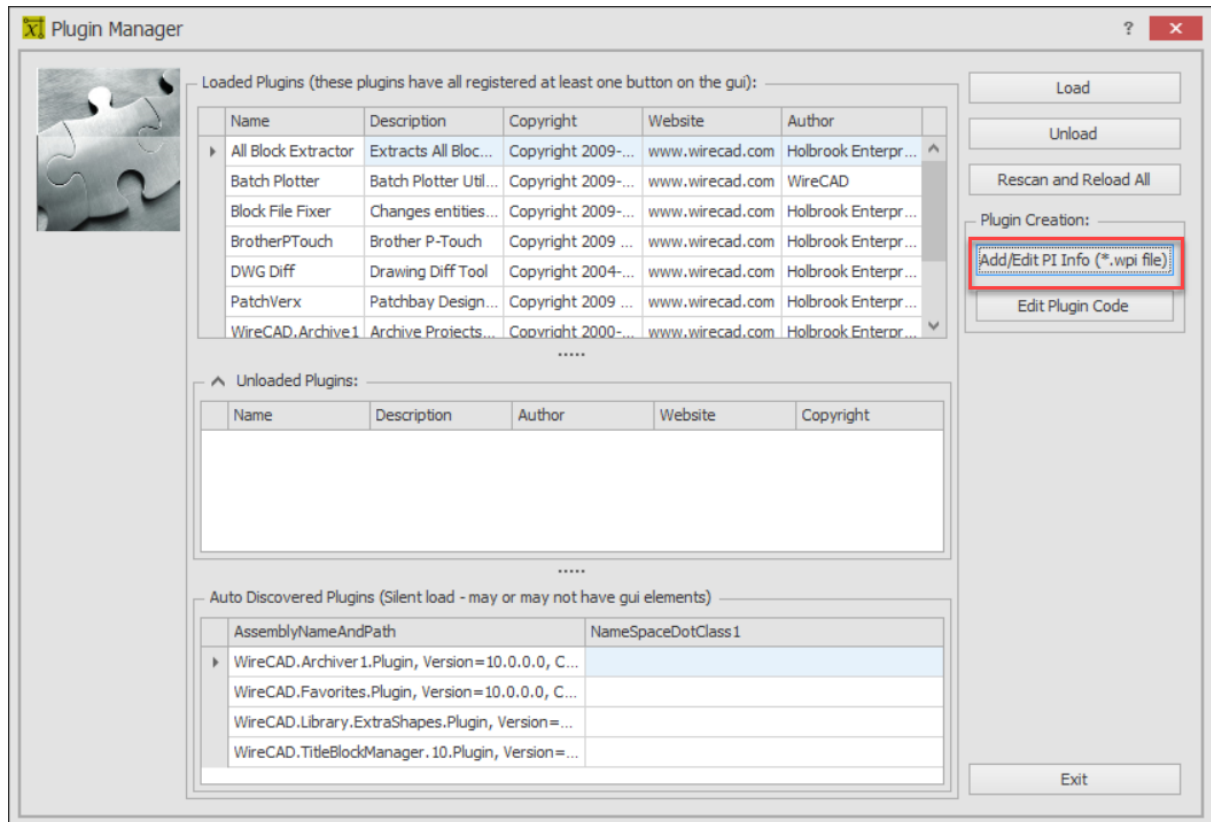
WireCAD uses an information file (wpi - **WireCAD Plugin Info**) to describe a command and to tell the application how to execute the command.

WireCAD looks for these files in the c:\users\public\documents\WireCAD\WireCAD10\plugins\Active Plugins

Commands can be executed directly from the Commandline or from a button on a menu bar.

In order to register your command you must, at minimum, set up the Commandline arguments.

Toolbar buttons are optional. To edit a wpi file use the editor.



Controls

•**Name, Description, Author, Website, Copyright** - Self explanatory.

Button Info


- Tool Tip** - Text displayed when mouse hovers over the button.
- Caption** - The button caption.
- Ribbon Page** - Select a page or create your own.
- Ribbon Page Group** - Select a page group or create your own.
- Button Bitmap** - bitmap image. 16x16 for a small button 32x32 for a large one.

Command Line and Assembly Info

- Long Name** - The display name of your command.
- Short Name** - The shortcut.
- Alt** - Shorter less descriptive.
- Assembly Name** - The name and fully qualified path of your dll. If no path we will search the \WireCAD\bin directory.
- Namespace Dot Class** - `YourNameSpace.YourClass` where your function is located.
- Static Method Name** - The static void method name of your function. This must have the proper signature as follows:

```
public static void YourFunctionName(Workspace ws)
{
    .....
}
```

Using the above example the Static Method Name would be: **YourFunctionName**

NOTE: if you use the WireCAD Plugin [template](#)  a static method with the proper signature will be created for you. You will need to rename it to be something meaningful.

This page is intentionally left blank.
Remove this text from the manual
template if you want it completely blank.

- [-

[Add Ports] 65
 [Add to Drawing] 79
 [Cleanup] 122
 [Clear All Cables] 122
 [Column Chooser] 49
 [Ctrl]+[D] 49
 [Ctrl]+[I] 49
 [Fillet] 122
 [Preview] 163
 [Ratsnest] 122
 [Register by Phone] 31
 [Register By Web] 31
 [Remove this Column] 49
 [Submit] 163
 [X] 49

- _ -

_D.DWG 82
 _S.DWG 82
 _SD.DWG 82

- A -

ac 108
 academic discounts 38
 ACID Compliant 188
 Activate WireCAD 31
 Activation 31
 Adapters 97
 Add All Cores 108
 Add Backbone Segment 237
 Add Backbones 219
 Add Connection Point 464
 Add Equipment to Drawings 79
 Add Jumper 237
 Add Multi-core Cable 456
 Adding Equipment to Drawings 48, 79
 Adding Equipment to the Drawing 79
 Advanced Tools 307
 Advanced Tools Dialogs 428
 Advanced Tools>Equipment Library 58
 Alias 103
 Appearance 219, 230

Application Menu Dialogs 326
 Application Menu Forms 517
 Application Setup Wizard 341
 Apply Aux Text to Existing Cable 145
 Assign Cable Number 108, 451
 assign multiple terminals 105
 Assign Sysname 103, 449
 Assign Terminal(s) Dialog 105
 Assign Terminals 105
 Assigning Cable Numbers 48
 Assigning Terminals 48, 105
 Assigning Unique IDs (SysNames) 48
 Assurance Subscription 38
 Attribute Height 110
 Authorization 31
 Auto Block 458
 Auto Preview 79
 AutoScheme 119
 Aux Text 86, 145
 Aux Text Enable 82
 Aux Text Height 82
 Auxiliary Text 145
 Available Cable Number Variables 125
 Available SysName Variables 123
 Avoid Other Cables 82, 122
 Avoid Other Calbles 82

- B -

Backbone 114, 210, 211, 214, 237, 249, 279, 475, 548
 Backbone Grid 211, 526
 Backbone Segment 214
 Backbone Segments 211, 237
 backbone status 211
 Backbone structures 237
 Backbones 114, 208, 210, 211, 214, 236, 237, 249, 279, 548
 backbones grid 211
 Backbones Section 319
 Backbones Visualize 219
 batch plot 17
 bidirectional 184
 Bill of Materials Generator 556
 BlockRef 110
 Body Width and Pin Spacing 115
 Brady 174

- C -

Cable 61, 114, 249, 548
 cable core data 184
 Cable Cores 58
 Cable Data Selector 317, 555
 Cable Error Check 457
 Cable Management Specific Features and Functions 61
 Cable Numbers 208
 Cable Router X Offset 86
 Cable Terminology 82
 Cable Text Height 82
 cable type 61, 214
 Cable Types 58, 184, 209, 210, 214, 237
 Cable Types database 61, 237
 Cable Types Database. 61
 Cable Types Grid 523
 CableNo 108, 174
 Cables 214, 237, 534
 cables database 211, 214
 Cables table 115
 Cables Visualization tool 117
 CablesCollection 174
 CableType 108
 CableTypeManu 108
 CAD Dialogs 389
 CAD Tools 302
 cg 117
 Chassis Width 110
 Check In 181
 Check in Project 339
 Check-out 181
 Choosing a Database Format 188
 Circuit 235, 237, 278, 279
 Circuit Grid 233, 235
 Circuit Recursion 115, 117
 Circuits Grid 233, 529
 CktNo 108
 Cleanup 119
 Clearing Filter Criteria 49, 50
 CMS 233, 278
 CMS module 214, 278, 279
 CMS tools 208
 collections 170
 Color Codes 58
 Color Codes Grid 524
 Column filtering 49, 50
 Column grouping 49, 50
 column header 49
 Column hiding/showing 49, 50
 Column re-ordering 49
 Column re-ordering/resizing 50
 Column resizing 49
 Combining Circuits 243
 Command Line 211, 233, 236, 573
 Concept 208
 Connection Point 464
 connectors 58, 184
 Connectors Grid 522
 Copy Selection Down 49, 50
 core 61, 211
 cores 211
 Count 82
 Create a New Label Report 174
 Create a New Project 70
 Create a Standard Report 170
 Create Model Space Boundaries 77
 Create New Circuit 237
 CreateFromDimensions 110
 CreateFromDimensionsIfNotFound 110
 Creating a New Drawing 48
 Creating a New Equipment Definition 48, 65
 Creating a New Project 48
 Current Record 219
 Customizing WireCAD 576

- D -

Data Dialogs 481
 Data environment 288
 Data Member 176
 Data Source 219
 Database 312
 Database Format 188
 Database Location 184
 Database menu 58
 Database>Equipment Library 58
 Database>Sync v6-v7 Equipment Libraries 188
 Default Pointer 82, 86
 Deleting Cell Data 49, 50
 Deleting Selected Rows 49, 50
 Demystifying Synchronization 184
 design mode 165
 Destination 108
 Detail Tab 63

- Dialogs 288, 326
- Dimension Styles 399
- Discrepancies 506
- Display As 120
- Display Hierarchical Detail Data 49, 50
- Display mode 79
- Display Preferences 63, 79, 120
- Distance 122
- drag 176
- drag fields 176
- Draw Backbone 214, 475
- Draw Cable 156, 476
- Draw Cable Toolbar 82
- Draw Cables 567
- Draw Cables Control Descriptions 82
- Draw Cables Toolbar 82
- Draw Prewire 159, 479
- Drawing 299
- Drawing Cables 48, 82
- Drawing Dialogs 389
- Drawing environment 288
- Drawing Properties 568
- Drawings 540

- E -

- Edit Cable Numbers Dialog 108
- Edit SysNames Dialog 103
- Elevation 103
- Enabling v7 Locations 70
- Engineering Data tab 103
- equipment 58, 184, 210, 214, 237
- Equipment Definitions 209
- Equipment Grid 520
- Equipment Library 48, 58, 63, 65, 428
- Equipment Library Access 58
- Equipment Library. 115, 214
- Equipment List 17, 532
- Equipment List (SysNames) 115
- Example 283
- Export 50, 184, 233, 235, 283
- Export Settings 280
- Export to PDF
 - EXCEL, TXT, HTML, XML and more 49
- Export to PDF, EXCEL, TXT, HTML, XML and more 49
- Export to Visio 393

- F -

- Favorites 572
- feature additions and changes 17
- Feeder distances in 100/DU 82
- Feeder Selection 82, 90
- Fiber Mode 61
- Field List 176
- File 298
- File>Export>[File Type] 49
- filter 49
- filter dialog 165
- filter pane 49
- Filtering Reports 165
- Find 569
- Find Tab 63
- Flags 103
- Flipped Pointers 98
- Floating License Lease 31
- floating licenses 31, 38
- Format 82
- Front Panel File 65
- Functional Block Diagram 237

- G -

- Get Port Data From Cables Database 120
- Getting Started 584
- global database 58
- Global Equipment Database 184
- grid 114, 249, 548
- Grid Searching 50
- GroupBy panel 49
- grouping 49, 170
- Groups 401

- H -

- Height in RU 110
- Hellerman Tyton 174
- Hello World 581
- hide a column 49
- hierarchical 170
- Hierarchical display 49
- home page 233
- Hor Extension in 100/DU 82

- Horizontal Spacing DU 120
- host 70
- How Rack Builder Works 112
- How SysName Formatting Works 123
- How SysName Formattion Works 125
- How to Place Custom Titleblocks into Your Drawing 179
- How To: Access the Equipment Library 62
- How To: Add a New Location 101
- How To: Add Equipment to Drawings 79, 80
- How To: Add Equipment to the Library 65
- How To: Add Multiple Locations 102
- How To: Add Some Inputs and Outputs 68
- How To: Assign a Cable Number 109
- How To: Assign a SysName 104
- How To: Assign Multiple Terminals 106
- How To: Assign Terminals 106
- How To: Create a Label Report 174
- How To: Create a New Backbone 214
- How To: Create a New Cable Type 61
- How To: Create a New Circuit 237
- How To: Create a New Drawing 78
- How To: Create a new drawing using the wizard 77
- How To: Create a New Label Report 174
- How To: Create a New Project 70
- How To: Create a New Report 170
- How To: Create a Standard Report 170
- How To: Create Your Own Custom Titleblocks 180
- How To: Customize Your CAD Blocks to Work with WireCAD 128
- How To: Draw a Prewire Cable in Planview 160
- How To: Filter a Report Using Parameters 167
- How To: Filter Reports 165
- How To: Find and Replace a Port Name 192
- How To: Grid Functions 50
- How To: Output a Circuit to CAD 233
- How To: Output Many Circuits to CAD 235
- How To: Place a Terminal as a Source 96
- How To: Place Custom Titleblocks (Page Borders) 179
- How To: Populate Cables in Planveiw 159
- How To: Print a Report 163
- How To: Print Reports 163
- How To: Pull a Take-Off from a Drawing 152
- How To: Setup a SQL Server 189
- How To: Translate a Form Caption 195
- How To: Use a Manufacturer Front Panel 199
- How To: Use the Cables Visualizer 119
- How To: Use the Equipment Visualizer 116
- HTML View 316, 551
- I -
 - I/O 214
 - Image Editor 403
 - Images 402
 - import 184
 - Include hashes 110
 - Increment Selection Down 49, 50
 - inputs 58, 105, 184
 - Inputs and Outputs 65
 - Insert 179
 - Insert Block Into Drawing 179
 - Insertion Point 110, 120
 - Inserts Dialog 408
 - IO 79
 - IO Tab 63
 - IP Address 103
 - IsSequential 103
- J -
 - Jumper 236, 237
 - Jumpers 237
 - Justified 170
- L -
 - Label Report 174
 - layered digraph 117
 - Layers 394
 - Layout 170
 - Layout Tools 150, 470
 - Layouts 395
 - Leading Zeros 103
 - Lease Period 31
 - License Agreement 36
 - license expire 38
 - Licensing FAQ 38
 - Location 82, 103
 - Location Boundary 154, 474
 - location change 110
 - Location Filter 120
 - Location Marker Section 319
 - Locations 70, 101, 537

- M -

Machine ID 31
 Manage Security 335
 Manual Draw 82
 Manual Draw Cables 86
 Manufacturer 65
 manufacturers 58, 184
 Manufacturers Grid 519
 Many Cable Database Entries 82
 Many to Many Cables 94
 Many to One Cables 93
 Many-to-Many Cable 82, 94
 Many-to-One Cable 82, 93
 masked 108
 Maximum Column Count 120
 Misc. 219, 230
 model 65
 Model Space Boundaries 400
 Model Space Text Height 77
 More about Regular Expressions 123, 125
 Moving Projects 181
 MText 357
 Multi Core 61
 multicore 108, 214
 Multi-core 456
 Multicore Data 108

- N -

Named Paths 538
 New + 108
 New Backbone 530
 New Cable 483
 New Cable Type 487
 New Circuit 530
 New Circuit tool 236
 New Drawing 77
 New Drawing Wizard 77, 389
 New Equipment 65
 new equipment block 79
 New Equipment Wizard 486
 New Location 481
 New Manufacturer 485
 New Plugin Template 584
 New Project 70

New Project Wizard 326
 New Report with Wizard 170, 174
 New Report Wizard 506
 New Viewport 179
 Next Numbers 70, 114, 249, 548
 Next Numbers Database 103
 Next Numbers Grid 114, 249, 548
 Next Numbers table 70

- O -

One Cable Database Entry 82
 One to Many Cables 90
 One-to-Many Cable 82, 90
 One-to-One Cable 82
 Ordinal 229
 Orientation 170
 OSnaps 357
 outputs 58, 105, 184
 Outputs track inputs 105
 Outputting Circuits to CAD 233

- P -

Pack Up/Check Out 188
 Packup 181
 Packup / Checkout 337
 Packup / Check-out 181
 Panduit 174
 Panel Marker Section 319
 parameters 163, 165
 Path Finder 241
 PDF Export 392
 PDF Print Preview 513
 PDF Viewer Dialogs 513
 PDF viewer environment 288
 Persist this equipment 105
 Pinouts 58, 191, 525
 Place Text If Item Cannot Be Created 110
 Plan View 150, 470, 571
 Plan View and Layout Tools 315
 Plan View and Layout Tools Dialogs 150, 470
 Plugin Manager 508
 Plugins 314
 Plugins Dialogs 508
 pn 65
 Point Styles 397

- Populate Cables 157, 477
- Populate Equipment 161, 480
- Port Data Source 115
- port selection dialog 105
- ports 214, 237
- Post Process Scripting 283
- Power Consumption 103
- Preview 163
- Pre-wires 117
- Print Preview 316, 390, 550
- Printed Output Text Height 77
- Printing Reports 163
- Printing Reports with Parameters 164
- PRO 31
- project 114, 237, 249, 279, 297, 548
- Project Cables Grid 117
- Project Database 214
- Project Explorer 61, 163, 165, 562
- Project Save As 330
- Project Settings 211, 277
- Project>New Project 70
- Project>Open Project 188
- Project>Utilities>Pack Up/Check Out 188
- projects 209
- Purge 408

- Q -

- Quick Access Toolbar 288

- R -

- Rack Builder 110
- Rack Builder Controls 110
- Rack Builder Tool 48, 110, 448
- Ratsnest 119, 461
- rb 110
- Rectangular Array 404
- Remember Settings 184
- Replace Cable with Pointers 82
- Report Design Basics 176
- Report Designer 317, 552
- Report filtering 165
- Report Style 170
- Report>New Report with Wizard 170
- Reporting 163
- Reporting environment 288

- Reports 163, 315
- Reports>New Report with Wizard 174
- Ribbon Tabs 288

- S -

- Scan for Discrepancies 506
- Script Editor/Runner 511
- Search Depth 219
- Search Width 219
- Searching 49
- Security 187
- Selected Format 123
- Setting Up On a Network 187
- Setting Up the WireCAD Global Databases on SQL Server 189
- Setting Up Your Global Data 48, 58
- settings menu 233, 277
- Show All 219
- Show Cable Numbers 122
- ShowDWGInPath 110
- signal types 58, 184
- Signal Types Grid 521, 531
- Signal Types to Display 120
- SignalType 108
- Slot Count 110
- Slot Delimiter 110
- Software Activation 31
- Software Key 31
- Sort 50
- Sort I/O By Last Display Order 115, 117
- Sorting 49
- Source 108
- Source and Destination info 108
- Spacing DU 110
- Splice Points 97
- Split Cable 144
- Split Cables 144
- SQL 209
- SQL databases 70
- SQL Server 188, 189
- Starting Cable Number 70
- Status 211
- Status Bar 184
- Strand Count 229
- Strand Count and Ordinal Explanation 229
- Submit 163
- Subnet Mask 103

suggest and test 70
 Swap 100, 107
 Swap All Pointers 100
 Swap All Terminals 107
 sync 184
 Sync Equipment Libraries... 184
 Synchronize Global Databases 488
 sys 115
 SysName 103, 105, 110
 SysName Error Check 456
 SysName Format 103
 SysName Format Controls 125
 SysName Prefix 65
 SysNames 103, 108
 System Snapshot 115, 117
 Systems Involved 110
 Systems Involved list 110

- T -

Tabs 296
 Take Offs 151, 471
 Template drawings 77
 Terminal 115, 117
 Terminal as a Destination Cable 96
 Terminal as a Source Cable 95
 Terminal as Destination Window 82
 Terminal as Source 95
 Terminal as Source Window 82
 Terminal Offset 82
 Terminals 82, 105
 Text Styles 397
 The Application Button 288
 The Application Menu 288
 The Command Line Interface 288
 The Draw Cables Tool Panel 288
 The Drawing Properties Tool Panel 288
 The Favorites Tool Panel 288
 The Find and Replace Tool Panel 288
 The Plan View Tool Panel 288
 The Project Explorer 58
 The Project Explorer Tool Panel 288
 The Quick Access Toolbar 288
 The Ribbon Toolbar 288
 The Settings Dialog 346
 Tidy Cables 82
 Title and Comment Block 219, 230
 Title Block 115

Title Block Manager 146
 ToDo 539
 Tool Panels 288, 561
 Toolbox 317, 553
 Translation Manager 559
 Troubleshooting Activation 31

- U -

UNC 187
 un-group 49
 Unlock 108
 Unpack 181
 Unpack Project 338
 upgrade a v5 or earlier project 188
 Upgrading from v6 188
 Use Last Display Order if Set 115
 Use Last Saved Appearance 115
 Use Location Lookup 101
 User 1 - 4 103
 User 1 - 4 103
 User Data tab 103
 User Settings 277
 Using The Backbone Grid 209
 Using The New Circuit Tool 209

- V -

Variable 82
 Variable List 123
 View 310
 View Permissions 187
 View Rule 110
 View>Windows>Property Grid 165
 VistaDB 188
 Visualization tab 115
 Visualization Tools 49

- W -

Weight 103
 Wiki Links 61
 WireCAD Cable Terminology 82
 WireCAD Community Server 58
 WireCAD Nomenclature 209
 WireCAD Project 70
 WireCADify Block 463

work flow 208
Work Flow Diagram 48
Workflow Diagram 48
Working with Pointers (on-sheet/off-sheet reference)
98
Workspace 288

- X -

X Offset 82
XL FREE 31
XLT 31
XREF Manager 409

- Y -

Y Offset 82
Y Spacing 82

- Z -

Zero Administration 188