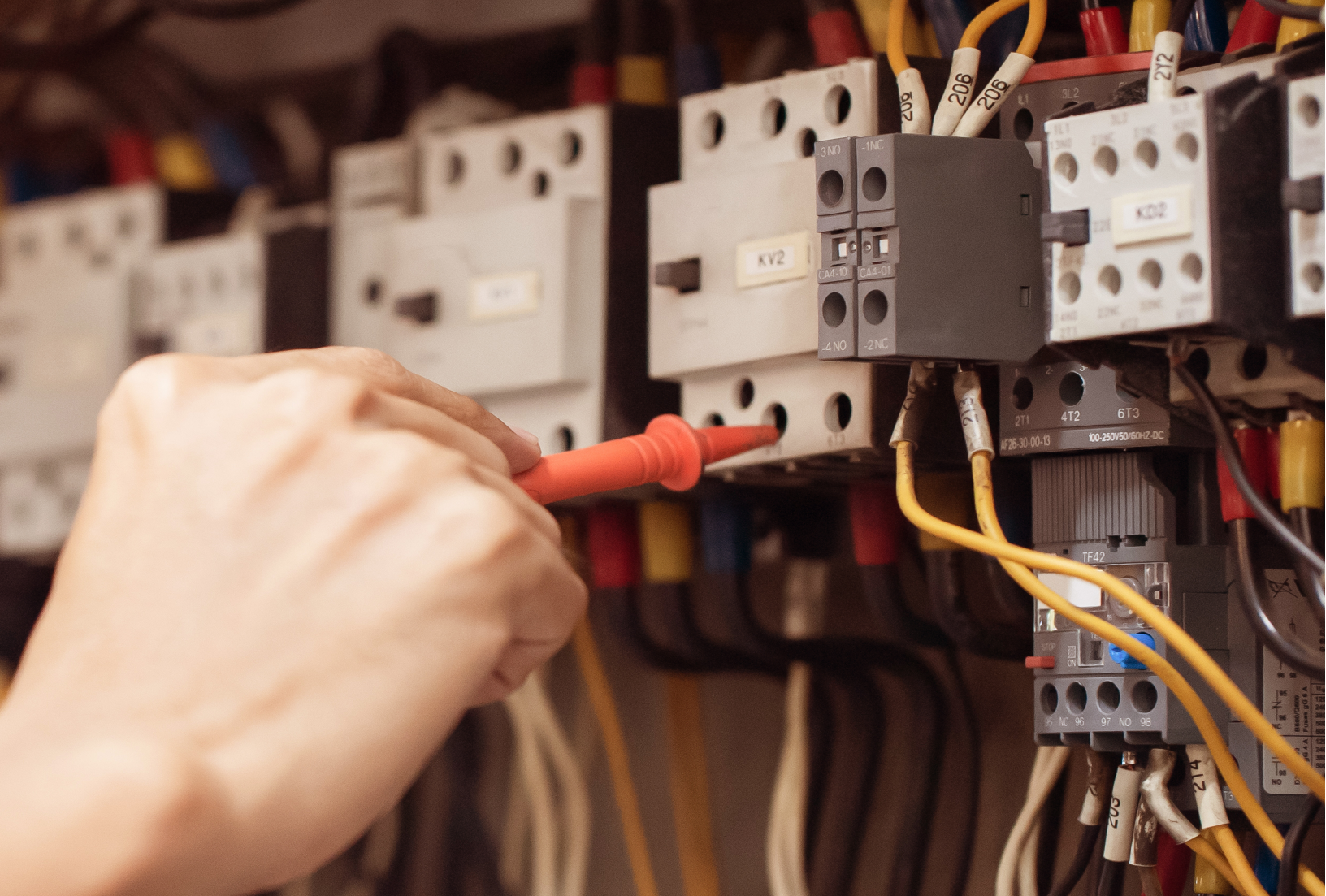




A TECHNICAL GUIDE TO AUTOCAD ELECTRICAL

TODD SCHMOOCK AND CAROL DUNN

2022



**Applied
Software**
GRAITEC Group



GRAITEC

TABLE OF CONTENTS

INTRODUCTION	2
SYSTEM REQUIREMENTS	3
AUTOCAD ELECTRICAL FEATURES	5
Electrical Symbols	5
Block Editor Environment	5
Attribute Template	6
Automation of Wires and Component Tags	6
Automatic Reports	6
Table Output	7
Support for Electrical Standards	8
Project Management	8
Circuit Design and Reuse	9
SQL Catalog Support	10
Coil and Contact Cross-Referencing	10
PLC I/O Drawings from Spreadsheets	10
TOP 5: HOW TO	11
Projects	11
Wire Numbers	14
PLC	16
Panel Layout	20
Reports	
6 MORE HELPFUL HOW TOS	35
AUTOCAD ELECTRICAL TIPS	59
MENUS	60
7 STEPS FOR CREATING A PROJECT	61
SETTING UP AUTOCAD ELECTRICAL FOR MULTIPLE USERS	62
ABOUT THE AUTHOR	63

INTRODUCTION

AutoCAD Electrical is a specialized toolset included with AutoCAD.

AutoCAD Electrical enables users to boost productivity by up to 95%* with electrical design features that help create, modify and document electrical controls systems. With AutoCAD Electrical, you can:

- **Access a library** of 65,000+ intelligent electrical symbols.
- **Automate numbering** of wires and **generation** of component tags.
- **Generate and update** multiple customized reports automatically.

** Productivity data is based on studies by an outside consultant commissioned by Autodesk. The studies compared experienced AutoCAD users performing common tasks using basic AutoCAD compared to using the specialized toolsets within AutoCAD.*

In the following document, the Top 5 How-To and 6 More Helpful How To's have been excerpted from the Autodesk AutoCAD Electrical User's Guide.

The entire [User's Guide](#) is available online.

SYSTEM REQUIREMENTS

System requirements for AutoCAD 2022 including Specialized Toolsets (Windows)	
Operating System	64-bit Microsoft® Windows® 11 and Windows 10. See Autodesk Product Support Lifecycle for support information.
Processor	Basic: 2.5–2.9 GHz processor Recommended: 3+ GHz processor
Memory	Basic: 8 GB Recommended: 16 GB
Display Resolution	Conventional Displays: 1920 x 1080 with True Color High Resolution & 4K Displays: Resolutions up to 3840 x 2160 supported on Windows 10 (with capable display card)
Display Card	Basic: 1 GB GPU with 29 GB/s Bandwidth and DirectX 11 compliant Recommended: 4 GB GPU with 106 GB/s Bandwidth and DirectX 12 compliant
Disk Space	10.0 GB
Network	See Autodesk Network License Manager for Windows
Pointing Device	MS-Mouse compliant
.NET Framework	.NET Framework version 4.8 or later

System requirements for AutoCAD 2022 for Mac	
Operating System	Apple® macOS® Monterey v12 (requires Update 2022.2) Apple macOS Big Sur v11 Apple macOS Catalina v10.15 Apple macOS Mojave v10.14
Model	Basic: Apple Mac Pro® 4.1, MacBook Pro 5.1, iMac® 8.1, Mac mini® 3.1, MacBook Air®, MacBook® 5.1 Recommended: Apple Mac® models supporting Metal Graphics Engine Apple Mac models with M series chip are supported under Rosetta 2 mode.
CPU Type	64-bit Intel CPU Recommended: Intel Core i7 or higher
Memory	Basic: 4GB Recommended: 8GB or higher
Display Resolution	Basic: 1280 x 800 display High Resolution: 2880 x 1800 with Retina Display

SYSTEM REQUIREMENTS

Disk Space	4 GB free disk space for download and installation
Pointing Device	Apple-compliant Mouse, Apple-compliant Trackpad, Micro-soft-compliant mouse
Display Card	Recommended: Mac native installed graphics cards
Disk Format	APFS, APFS (Encrypted), Mac OS Extended (Journaled), Mac OS Extended (Journaled, Encrypted)

Additional Requirements for large datasets, point clouds, and 3D modeling	
Memory	8 GB RAM or more
Disk Space	6 GB free hard disk available, not including installation requirements
Display Card	3840 x 2160 (4K) or greater True Color video display adapter; 4G VRAM or greater; Pixel Shader 3.0 or greater; DirectX-capable workstation class graphics card.

Specialized Toolset (Windows Only)

Toolset	Additional Requirements
AutoCAD Electrical	Disk Space: 20GB Microsoft Access Database Engine 2016 Redistributable (x64) (16.0.5044.1000) or later

Chart adapted from Autodesk Knowledge Network "[System requirements for AutoCAD 2022 including Specialized Toolsets](#)", May 6, 2022.

AUTOCAD ELECTRICAL TOOLSET FEATURES

ELECTRICAL SYMBOLS

A library containing over 65,000 easy-to-use, colorful, customizable electrical symbols is available for use in projects. If you need a symbol that isn't part of the library, you can convert symbols or create custom components on the fly using the Symbol Builder tool. [Learn more.](#)

Symbols created using Symbol Builder are fully compatible with AutoCAD Electrical. They will break wires upon insertion and will appear in the bill of material (BOM) and component and wire connection reports.

The Symbol Builder tool can be entered and exited at any time. Regular AutoCAD commands can be used to edit or finish a symbol you create, with the Wblock command writing it to disk. Each time you go back in to the Symbol Builder tool, you can select objects from the Select Symbol/Objects dialog box and allows the tool to track the standard attributes and wire connection points you have already inserted.

The new symbols you create can be inserted using AutoCAD Electrical "Insert Component" or "Insert Panel Component" command. The new symbol can be added to the icon menu. It can also be selected from the "Type it" or "Browse" options at the bottom left of the icon menu.

BLOCK EDITOR ENVIRONMENT

Initial selections for block editing can include symbol type, attribute template, existing objects, and insertion point. After making your initial selections, you enter the Block Editor environment. Besides Block Editor menus for adding and modifying symbol graphics, you can use the Symbol Builder attribute editor to insert and modify AutoCAD Electrical attributes.

AUTOCAD ELECTRICAL TOOLSET FEATURES

ATTRIBUTE TEMPLATE

AutoCAD Electrical expects certain attributes for each symbol type, schematic parent, schematic child, etc. Symbol Builder uses attribute templates to make it easier to add attributes to your symbol. Attribute template drawings are AutoCAD drawings with Electrical toolset attributes. There are different attribute templates for different types of symbols and for different family codes. The attribute templates are supplied in the symbol library folders; all attribute template drawing names begin with “AT_”.

After a symbol type is selected, the associated attribute template is used to create the list of attributes. The attribute template can contain attributes defined as required, with others optional. Required attributes are expected on the specific symbol type you’re building; optional attributes may not be necessary but are supported. You can insert attributes individually as needed or all the attributes from the template at one time.

The attribute template naming convention is: AT_{symbol}_{type}. The {symbol} string is displayed in the Symbol list of the Select Symbol/ Objects dialog box. The {type} string is displayed in the Type list. You can also map abbreviations for the {type} in the _FAMILY_DESCRIPTION table of the catalog database, default_cat.mdb.

AUTOMATION OF WIRES AND COMPONENT TAGS

AutoCAD Electrical lets you save time with enhanced automation, allowing for automatic numbering of wires and generation of component tags. [Learn more.](#)

AUTOMATIC REPORTS

You can simplify your workflow and save time by automatically generating and updating multiple customized reports. [Learn more.](#)

The Automatic Reports tool enables you to run multiple reports at one time. You can save each report to a file and/or place each report as a table.

The Report Generator dialog box is not displayed for each report. No user input is required once it is launched. In order to use Automatic Reports, format files are required. They must define file output, table output or both, since the reports are not displayed on the screen. Use Format File Setup to define file and table output in a format file.

If you tend to run the same group of reports, you can save that group as a report grouping. To set one up, add all your format files as though you are going to run the reports, then save the report grouping. The information is saved in a Report Grouping file with an *.rgf* extension. The next time you want to run that group of reports, open the *.rgf* saved previously.

TABLE OUTPUT

If any of your selected format files contain table output, the program checks for existing tables for the selected report. If an existing table meets all of the following conditions, it will be considered a match to the current report and updates:

- The existing table is for the same report.
- The existing table is for the same scope.
- The existing table used the same format file.

If no matching tables are found, the report's tables insert on new drawings. If you are running multiple reports with multiple table output, each report inserts new tables on its own drawings. You can specify the first drawing name and the template name for any new drawings. Subsequent drawing names generate automatically by incrementing the previous drawing's name.

AUTOCAD ELECTRICAL TOOLSET FEATURES

SUPPORT FOR ELECTRICAL STANDARDS

AutoCAD Electrical supports the latest electrical standards, as well as providing legacy support for JIC and older IEC symbol libraries.

[Learn more.](#)

AutoCAD Electrical provides library symbols that comply with the standards:

- IEEE 315/315A (metric)
- IEC-60617 (metric)
- NFPA (imperial)

To scale the metric IEEE and IEC library symbols for imperial, use the Modify library symbols utility.

The JIC standard is no longer updated and was incorporated into the NFPA 79 standards. The NFPA standard states that the library symbols are in accordance with the IEEE 315/315A standard.

PROJECT MANAGEMENT

The time-saving features of Project Manager boosts productivity and simplifies projects. [Learn more.](#)

The Project Manager is where you create new projects and work with existing ones. You can retag components, sheets, wire numbers, drawings, title blocks, and more. You can specify what needs to be changed in a file and make those changes easily. Once you are ready to export, you can plot the project and then publish to the web or a preferred file format.

CIRCUIT DESIGN AND REUSE

The Circuit Builder tool enables simplified electrical design. It is also customizable. You can add new circuit definitions and edit existing ones.

The tool is prepopulated with data to build and annotate a sampling of motor control circuits and power feed circuits. Circuits include 3-phase, single-phase, and one-line representations. Each circuit is built dynamically, adjusting the power bus to match the wire bus for the drawing. Wiring is then added between components, and elements are annotated with suggested values based upon the selected load. Each time a circuit is configured, it's added to a history list of circuits. This list streamlines reinsertion at a later time.

Three items control this feature:

1. The spreadsheet defines the available circuits, circuit types and defaults for each option within a circuit.
2. The template (.dwg file) for a selected circuit defines the wiring and the placement position for the individual components on that wiring.
3. The electrical standards database provides suggested values used to annotate components of the circuit and the wire size and type of the power wiring.

Circuit Builder supplies and uses a one-line symbol library when building a one-line circuit. Each one-line symbol has a WDTYPE attribute with a value of "1-" or "1-1". The WDTYPE attribute value distinguishes the one-line symbol from a schematic symbol. A schematic symbol either has no WDTYPE attribute or a blank WDTYPE attribute value. One-line symbols follow the same symbol naming conventions and have the same attribute requirements as schematic symbols with a few attribute exceptions.

AUTOCAD ELECTRICAL TOOLSET FEATURES

SQL CATALOG SUPPORT

The 2022 Electrical toolset supports the SQL catalog database for the Bill of Materials in Autodesk Vault. This helps users save time when generating a Bill of Materials in Autodesk Vault for an AutoCAD Electrical project. The data from the AutoCAD Electrical drawings will automatically be added to the report, including the details in the catalog database. In addition to Microsoft Access database, the support is extended to the Microsoft SQL format of the catalog. [Learn more.](#)

COIL AND CONTACT CROSS-REFERENCING

Projects can be simplified because parent/child contacts are tracked in real time. [Learn more.](#)

As Circuit Builder dynamically builds a circuit, each component receives a component tag. A child contact must link to a parent component to receive the same component tag as the parent.

The parent and child components are automatically linked by Circuit Builder if they each have the same default tag value.

There can be more than one parent/child relationship within the overall circuit with the same default tag. The overall circuit includes the main circuit template and any branching or nested circuit templates.

PLC I/O DRAWINGS FROM SPREADSHEETS

AutoCAD Electrical can speed up your workflow by using a single data file to generate multiple drawings. Once a project's I/O assignments are defined, PLC drawings can be generated automatically from a spreadsheet, database or comma-delimited text file.

[Learn more.](#)

TOP 5: HOW TO

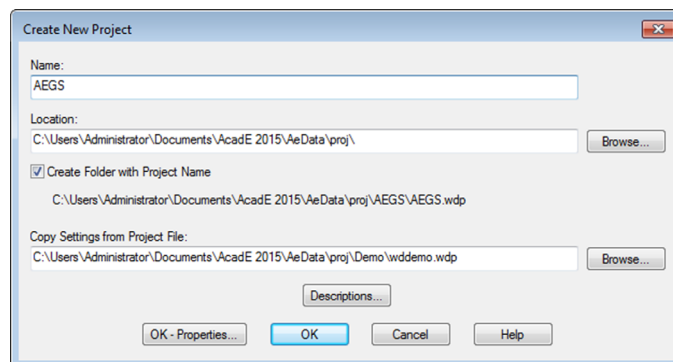
1. PROJECTS

AutoCAD Electrical is a project-based system. Each project is defined by an ASCII text with a **.wdp** extension. The project file contains the project information, default project settings, drawing properties, and drawing file names. You can have an unlimited number of projects, but only one project can be active at a time.

Use Project Manager to add new drawings, reorder drawing files, organize drawings in subfolders, and change project settings. You cannot have two projects open in Project Manager with the same project name. By default, Project Manager resides on the left side of your screen. You can change the location on the screen or hide it until you want to use the project tools. Right-click the properties icon to display options to move, size, close, dock, hide, or set the transparency for Project Manager.

Follow these instructions to create an AutoCAD Electrical project.

1. Click Project tab -> Project Tools panel -> Manager.
2. In Project Manager, click the New Project tool.
3. In the Create New Project dialog box, specify:
Name: AEGS
A name must be entered to define any of the project properties. The .wdp extension is not required in the edit box.
4. Make sure *wddemo.wdp* is specified in the Copy Settings from Project File edit box.



5. Click OK-Properties.

Your new project is added to the current projects list and automatically becomes the active project.

The Project Properties dialog box displays, where you can modify your project default settings. All information defined on these tabs are saved to the project definition file as project defaults and settings.

Set project properties

1. In the Project Properties dialog box, click the Components tab.
2. In the Component Tag Format section, verify that Line Reference is selected.

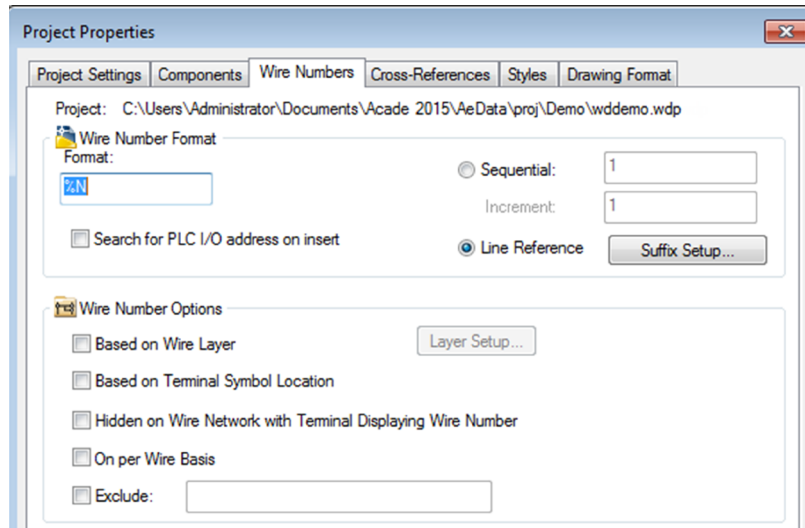
This creates unique reference-based tags when multiple components of the same family are located at the same reference location. When reference-based tagging is used, a suffix variable is required to keep components of the same family type unique.

For example, three push buttons on line reference 101 could be labeled PB101, PB101A and PB101B. Click Suffix Setup to change the suffix variable.

3. Click the Wire Numbers tab.
4. In the Wire Number Format section, verify that Line Reference is selected.

This creates unique reference-based wire number tags for

multiple wire networks beginning at the same reference location. When reference-based numbering is used, a suffix variable is required to keep wires on the same reference line or in the same reference zone unique. Click Suffix Setup to change the suffix variable.



5. Review the various options on the different tabs of the Project Properties dialog box.

Note: In the Project Properties dialog box, icons indicate whether the settings apply to project settings or drawing defaults. Project settings have the project icon next to them and are saved inside the project definition file (*.wdp). Settings saved in the project file as drawing defaults have the drawing icon next to them. Drawing related data to add to the project when running the Add Drawing command is saved as Drawing Custom Properties.

6. Click OK.

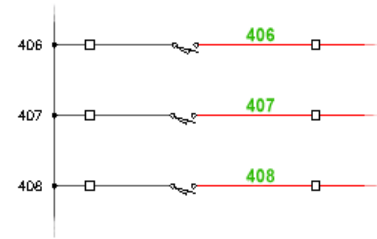
TOP 5: HOW TO

2. WIRE NUMBERS

AutoCAD Electrical assigns a unique wire number to each wire network. A wire network consists of one or more wires that are electrically connected.

Wire numbers can be assigned to:

- any existing wires on an individual selection
- an entire drawing
- selected drawings in a project
- an entire project



You can process and tag wires with sequential wire numbers or with wire numbers based on the line reference location start of the wire network. When wire numbers are automatically inserted into a drawing, the numbers are not duplicated.

AutoCAD Electrical works from left to right and top to bottom as it processes wire networks by default. To change the direction of wire numbering use the Project Properties -> Wire Numbers dialog box. [In Project Manager, right-click the project name and select Properties. In the Project Properties dialog box, click the Wire Numbers tab.]

To insert wire numbers automatically:

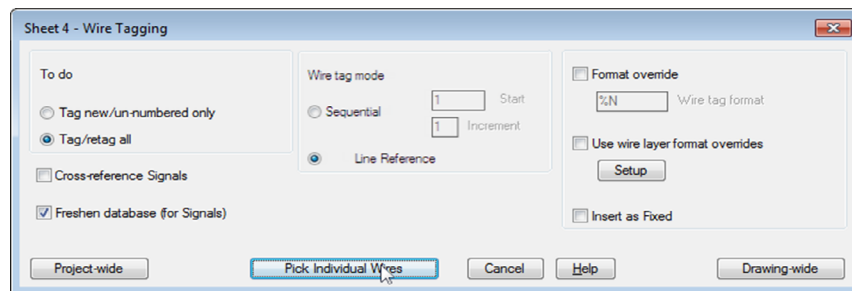
- A. If AEGS is not the active project, activate it in the Project Manager by right-clicking AEGS and selecting Activate.
- B. In Project Manager, double-click AEGS to expand the drawing list.
- C. In Project Manager, Project Drawing List, double-click

AEGS04.dwg.

D. Zoom in on the top portion of the wire network on the left side of the drawing.

E. Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wire Numbers drop-down -> Wire Numbers.

F. In the Sheet 4 - Wire Tagging dialog box, click Pick Individual Wires.



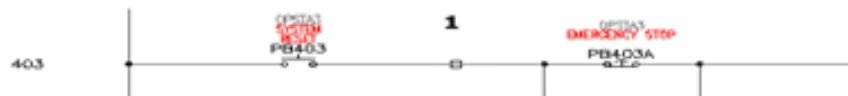
G. Respond to the prompts as follows.

Select objects:

Select the wire segment between the two push buttons on line reference 403 (1), right-click

To add wire numbers to the entire drawing:

A. Click Schematic tab -> Insert Wires/Wire Numbers panel ->



Insert Wire Numbers drop-down -> Wire Numbers.

B. In the Sheet 4 - Wire Tagging dialog box, click Drawing-wide.

Wire numbers are assigned to each segment in your drawing.

TOP 5: HOW TO

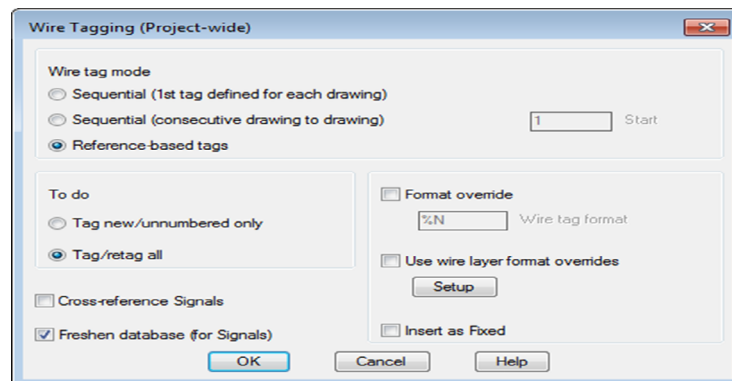
To add wire numbers project-wide:

- A. Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wire Numbers drop-down -> Wire Numbers.
- B. In the Sheet 4 - Wire Tagging dialog box, click Project-wide.
- C. In the Wire Tagging (Project-wide) dialog box, verify these settings:

Wire tag mode: Reference-based tags

To do: Tag/retag all

Freshen database (for Signals)



- D. Click OK.
 - E. In the Select Drawings to Process dialog box, Project Drawing List section, press Shift as you select AEGS03.dwg and AEGSO04.dwg. Click Process.
 - F. Verify AEGS03.dwg and AEGS04.dwg are listed as the drawings to process, then click OK.
 - G. If asked to save the drawing, click OK.
- Wire numbers are processed for the selected drawings.

3. PLC

AutoCAD Electrical can generate any number of different PLC I/O modules on demand and in various graphical styles. AutoCAD

Electrical generates the selected PLC I/O module through a parametric generation technique based on information in the PLC database (ACE_PLC.MDB).

The PLC database contains the stack sequence and text values to annotate onto each symbol in the stack. As AutoCAD Electrical builds the module, it reads the underlying ladder rung spacing and inserts the I/O points and terminals to match the rung spacing. During the insertion process, you can add spacing or split the module into multiple pieces.

Following is the process to parametrically generate a PLC module adapting to the underlying rung spacing. Modules are defined in the PLC database.

- A. Click Schematic tab -> Insert Components panel -> Insert PLC drop-down -> Insert PLC (Parametric).
- B. In the upper left of the PLC Parametric Selection dialog box, locate the PLC in the tree, expanding the manufacturer, series and type.
- C. Select the module.
- D. Select the graphical style of the PLC module as needed.
- E. Select its orientation as needed.
- F. Enter the scale for the PLC module if other than 1.0.
- G. Click OK to insert the PLC module.
- H. Use the preview of the module outline to pick an insertion point.
- I. (Optional) Select Allow Spacers/Breaks option on the Module Layout dialog box to break the module.
- J. (Optional) If the module has connections marked as Optional in the PLC database, select to include or exclude them.
- K. Click OK.

- L. Answer the prompts. The prompts vary depending on the module definition.
- M. Click OK.
- N. Enter or select a first address.
- O. If prompted, select an addressing style.

4. PANEL LAYOUT

Panel Layout tools create intelligent mechanical/panel layout drawings. Key features are:

- Layouts can be driven from information carried on AutoCAD Electrical schematic wiring diagram drawings, or they can be constructed independently of schematics.
- AutoCAD Electrical places no requirements on special naming or attribute requirements on mechanical footprint symbols. Vendor-supplied footprint symbols in AutoCAD format can be used as is with the Electrical toolset.
- Bi-directional update capabilities allow certain schematic wiring diagram edits to update the panel drawings automatically and vice versa.
- Wire number, wire color/gauge information and connection sequencing data can be extracted directly from the schematics and annotated onto the panel footprint representations.
- AutoCAD Electrical extracts various reports from these smart panel layout drawings, including panel BOM, panel component/item lists, nameplate reports, and schematic versus panel exception reports.

Panel drawing configuration settings are saved as attribute values on a nonvisible block named WD_PNLM. If your current drawing does not have this block present when any AutoCAD Electrical panel

layout command is invoked, AutoCAD Electrical pauses and asks for permission to insert this block. It inserts at “0,0”, but this location is not critical. The key point is that the nonvisible block must be present somewhere on the drawing.

To define panel layout drawing defaults:

- A. Click Panel tab -> Other Tools panel -> Panel Configuration drop-down -> Configuration.
- B. Enter a first item number. **Note:** This value controls item numbering on schematic and panel components.
- C. Click Setup in the Balloon section to open the Panel balloon setup dialog box where you pick a balloon type, text size and leader arrow type.
- D. If you change the balloon setup and want to update any existing balloons, check Apply to all Balloons in Active Drawing.
- E. Click Setup in the Footprint layers section to define the panel component layers, non-text graphic layers and nameplate layers.
- F. Enter the default spacing used when you insert multiple panel footprints from a schematic list. **Note:** This value can be overridden at insertion time.
- G. Enter the default footprint insertion scale.
- H. If you want to insert the attribute template when a footprint is inserted, check Enabled. Enter a scale factor if you want to scale the footprints during insertion.
- I. Enter the x-, y- or z-offset value used in the Panel Wire Connection report. If your footprint symbols carry the X?TERMxx attributes, their XYZ wire connection location is included in the report and offset by these values.
- J. Use the Setup button to define the default wire connection

TOP 5: HOW TO

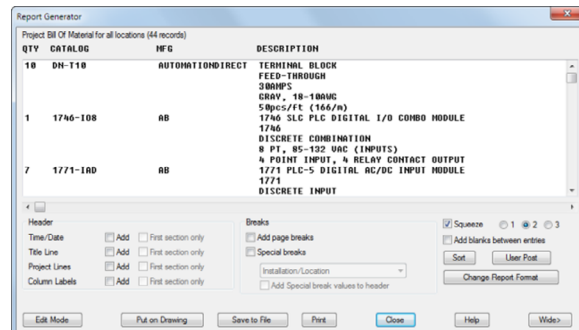
text format used by the Wire Annotation command.
K. Click OK.

5. REPORTS

Working with reports includes the process for generating schematic and panel reports, changing the report format, saving a report to a file, and placing a report on a drawing.

GENERATE SCHEMATIC OR PANEL REPORT

- A. Do one of the following:
- Click Reports tab -> Schematic panel -> Reports.
 - Click Reports tab -> Panel panel -> Reports.
- B. Select which report to generate from the report list.
- C. Select to process the project, current drawing or selected components.
- D. Specify any report options (if applicable).
- E. Select installation or location codes to extract (if applicable).
- **All:** Extracts all components regardless of value.
 - **Blank:** Extracts only those components that do not have a value.
 - **Named:** Extracts only those components that have a value matching the value entered in the box. Wild-card characters are supported.
- F. Indicate whether to update the project database or the wire connection table with out-of-date drawings.
- G. Click OK.
- H. Sort the report, change



the format or edit the data.

- I. Save the report to a file, place the report on the drawing as a table or print the report.

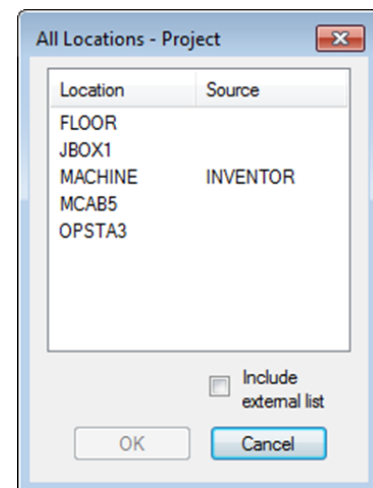
GENERATE AN ELECTROMECHANICAL BILL OF MATERIAL REPORT

Create a Bill of Material report that contains components from your electrical project and linked Inventor assembly.

- A. Do one of the following:
 - Click Reports tab -> Schematic panel -> Reports.
 - Click Reports tab -> Panel panel -> Reports.
- B. Select Bill of Material from the report list.
- C. Select to process the project.
- D. Check Include Inventor Parts.

Note: If your project is linked to an Inventor assembly, this check box is enabled.
- E. Specify any report options.
- F. Select installation or location codes to extract.
 - **All:** Extracts all components regardless of value.
 - **Blank:** Extracts only those components that do not have a value.
 - **Named:** Extracts only those components that have a value matching the value entered in the box. Wild-card characters are supported.

The Drawing button is disabled but you can select a value from the Project list. The project lists for Installation and Location include



TOP 5: HOW TO

values from the Inventor assembly. Values from Inventor are indicated in the Source column.

G. Click OK.

Note: All drawings in the project are processed when you include Inventor.

CHANGE A REPORT FORMAT

Define which fields to include in the report, the field order, the field labels, and field justification.

- A. Generate a report.
- B. In the Report Generator dialog box, select Change Report Format.
- C. Click a field in the Available Fields list to add it to the Fields to Report list.
- D. Select a field in the Fields to Report list.
 - Use the Move Up and Move Down buttons to change the field order.
 - Edit the field name.
 - Select a justification for the field.
- E. (Bill of Material report only) Select fields to include in the multi-line description field.

Note: If a field is listed in the Fields to Report list, it is not included in the multi-line description even if checked in Lines for Description list.
- F. Click OK to save the changed format to the default file name, or Click OK+Save As to save to a different file name.
- G. Click Close.

SAVE REPORT TO A FILE

- A. Generate a report.
- B. In the Report Generator dialog box, select Save to File.
- C. Select the file format.
- D. Choose any options within the selected file type.
 - **LINEx values:** Project description lines marked as in reports.
 - **Labels:** Labels for the fields included in the report.
 - **First section only:** If the report contains special breaks, select whether to include on the first section only.
- E. Click OK.
- F. Enter a file name, define the path and click Save.
- G. The Optional Script File dialog box displays where you can run an optional script on your report data.
- H. Click Close on the Report Generator dialog box.

INSERT A REPORT AS A TABLE

- A. Generate a report.
- B. In the Report Generator dialog box, select Put on Drawing.
- C. Select the table type:
 - **Insert New:** If the same report is run again later, the table can be updated.
 - **Insert New (non-updatable):** If the same report is run again later, the table is not updated.
 - **Update Existing:** If there is an existing table that matches the report, it updates. This option is not available if there are no matching tables for the report.
- D. Select a table style from the list. You can browse to another drawing and add any tables style on that drawing to the

selection list.

- E. Specify whether to include any title information.
- F. Specify the method to use for calculating the width of the columns.
- G. Specify the XY coordinates for the upper left corner of the table. If left blank, you are prompted to select a location after you click OK.
- H. Enter a start and end line value. The default is to include all rows of report data.
- I. Specify the options for the rows.
 - **Apply Special Breaks:** If you selected any special breaks on the Report Generator box, you can break the table into separate table objects based on those values.
 - **Rows for Each Section:** Specifies to break the report into separate table objects based on the value in the Rows box.
- J. If table sections are defined:
 - Specify the maximum number of table sections placed on each drawing. A blank value indicates an unlimited number of sections on one drawing.
 - Specify the X and Y distance from the end of one table section to the beginning of the next. These values are used when there are multiple sections on the same drawing.
- K. Click OK.

If you did not specify the XY coordinates, your cursor is a box approximating the size of the table when it is generated. To use object snap mode, enter an **S** at the command line.

EDIT A REPORT

If you want, you can modify the report before you insert it into your drawing or save it to a file. You can move data up or down in the report, add lines from a catalog and delete lines.

- A. Generate a report.
- B. In the Report Generator dialog box, select Edit Mode.
- C. Select a row within the report.
- D. Click Edit to change individual values in that row.

- E. Select a row or group of rows within the report.
 - Click Delete to remove the selected rows.
 - Use the available move options to change the order of the selected rows.
- F. Use the available options to add extra rows with data to the report. Options differ depending on the report type.
- G. If the report contains wiring information, each line item in the report can include two connected components, referred to as the From and To components. The From field labels end in a 1, and the To field labels end in a 2.
Click Swap to swap the From and To field values.
- H. Click OK-Return to Report.

POST-PROCESS A REPORT

Further customize the report data with a custom lisp function.

- A. Generate a report.
- B. In the Report Generator dialog box, select User Post.
- C. Select from the available post-processing options.
- D. Click Close.

TOP 5: HOW TO

The LISP routine processes the data and returns to the Report Generator Window.

Sample .lsp and .dcl files for each report are installed in the C:\Program Files\Autodesk\AutoCAD {version}\Acade\Support\{language code}\Shared\ folder. A few of the reports have sample options already defined. Otherwise, these options are user-created. **Note:** See the Report Generator Dialog Box help topic for a list of user post file names.

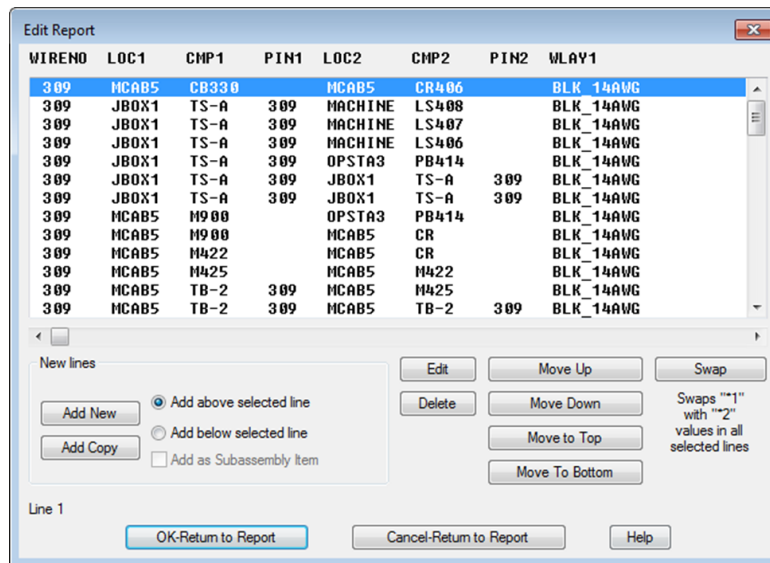
5. REPORTS

Working with reports includes the process for generating schematic and panel reports, changing the report format, saving a report to a file, and placing a report on a drawing.

GENERATE SCHEMATIC OR PANEL REPORT

- A. Do one of the following:
 - Click Reports tab -> Schematic panel -> Reports.
 - Click Reports tab -> Panel panel -> Reports.
- B. Select which report to generate from the report list.
- C. Select to process the project, current drawing or selected components.
- D. Specify any report options (if applicable).
- E. Select installation or location codes to extract (if applicable).
 - **All:** Extracts all components regardless of value.
 - **Blank:** Extracts only those components that do not have a value.
 - **Named:** Extracts only those components that have a value matching the value entered in the box. Wild-card characters are supported.
- F. Indicate whether to update the project database or the wire

connection table with out-of-date drawings.



- G. Click OK.
- H. Sort the report, change the format or edit the data.
- I. Save the report to a file, place the report on the drawing as a table or print the report.

GENERATE AN ELECTROMECHANICAL BILL OF MATERIAL REPORT

Create a Bill of Material report that contains components from your electrical project and linked Inventor assembly.

- A. Do one of the following:
 - Click Reports tab -> Schematic panel -> Reports.
 - Click Reports tab -> Panel panel -> Reports.
- B. Select Bill of Material from the report list.
- C. Select to process the project.
- D. Check Include Inventor Parts.

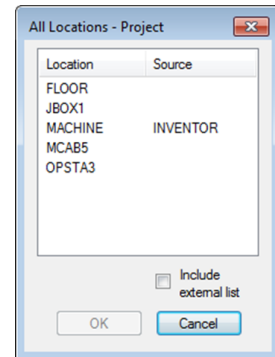
Note: If your project is linked to an Inventor assembly, this

TOP 5: HOW TO

check box is enabled.

- E. Specify any report options.
- F. Select installation or location codes to extract.
 - **All:** Extracts all components regardless of value.
 - **Blank:** Extracts only those components that do not have a value.
 - **Named:** Extracts only those components that have a value matching the value entered in the box. Wild-card characters are supported.

The Drawing button is disabled but you can select a value from the Project list. The project lists for Installation and Location include values from the Inventor assembly. Values from Inventor are indicated in the Source column.



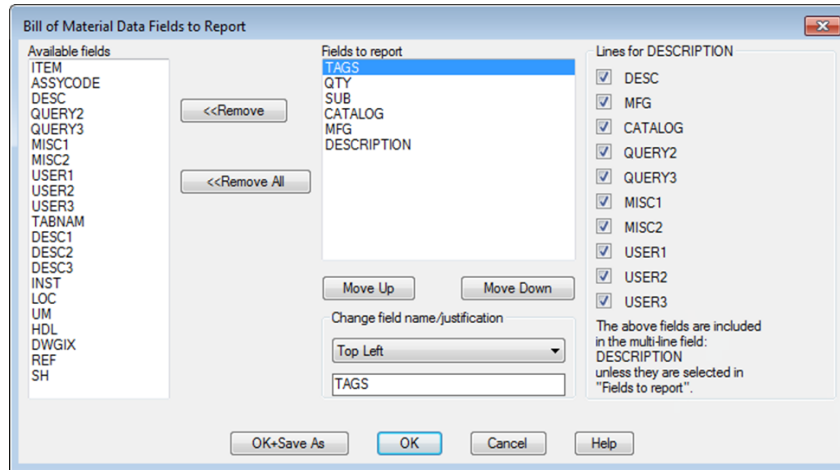
- G. Click OK.

Note: All drawings in the project are processed when you include Inventor.

CHANGE A REPORT FORMAT

Define which fields to include in the report, the field order, the field labels, and field justification.

- A. Generate a report.
- B. In the Report Generator dialog box, select Change Report



Format.

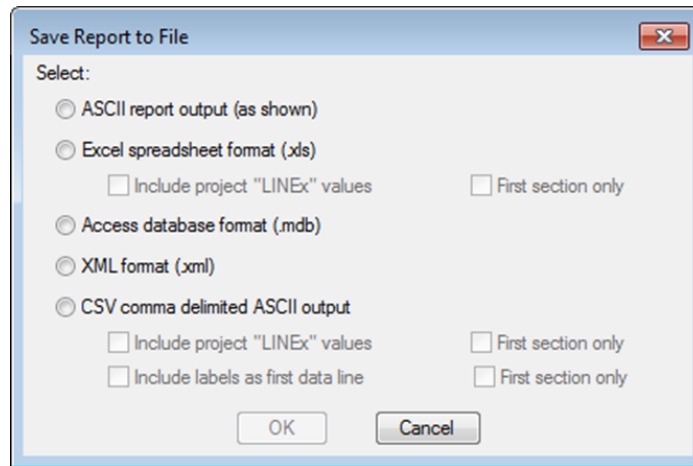
- C. Click a field in the Available Fields list to add it to the Fields to Report list.
- D. Select a field in the Fields to Report list.
 - Use the Move Up and Move Down buttons to change the field order.
 - Edit the field name.
 - Select a justification for the field.
- E. (Bill of Material report only) Select fields to include in the multi-line description field.

Note: If a field is listed in the Fields to Report list, it is not included in the multi-line description even if checked in Lines for Description list.

- F. Click OK to save the changed format to the default file name, or Click OK+Save As to save to a different file name.
- G. Click Close.

SAVE REPORT TO A FILE

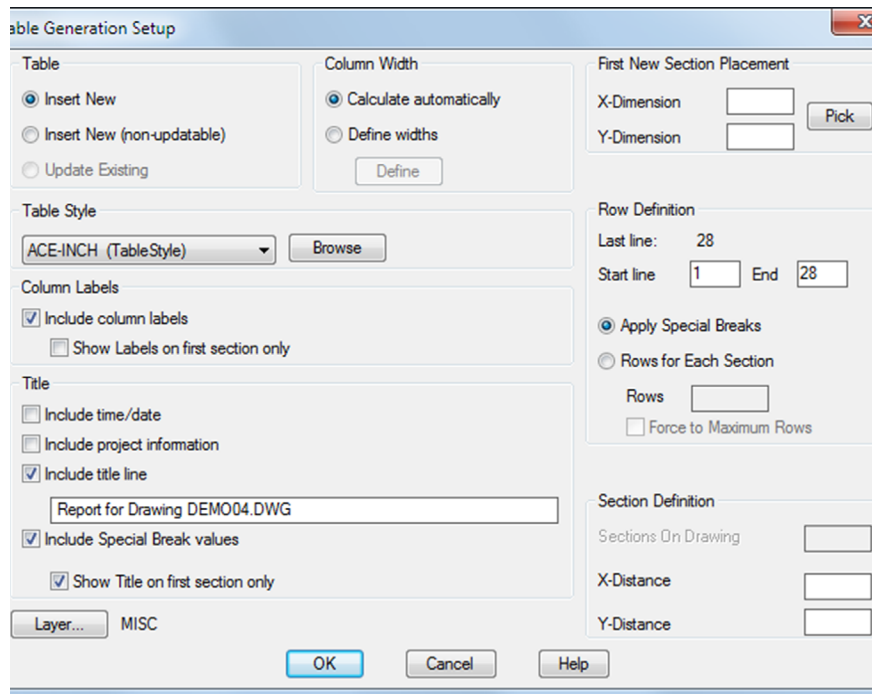
TOP 5: HOW TO



- A. Generate a report.
- B. In the Report Generator dialog box, select Save to File.
- C. Select the file format.
- D. Choose any options within the selected file type.
 - **LINEx values:** Project description lines marked as in reports.
 - **Labels:** Labels for the fields included in the report.
 - **First section only:** If the report contains special breaks, select whether to include on the first section only.
- E. Click OK.
- F. Enter a file name, define the path and click Save.
- G. The Optional Script File dialog box displays where you can run an optional script on your report data.
- H. Click Close on the Report Generator dialog box.

INSERT A REPORT AS A TABLE

Define which fields to include in the report, the field order, the field labels, and field justification.



- A. Generate a report.
- B. In the Report Generator dialog box, select Put on Drawing.
- C. Select the table type:
 - Insert New: If the same report is run again later, the table can be updated.
 - Insert New (non-updatable): If the same report is run again later, the table is not updated.
 - Update Existing: If there is an existing table that matches the report, it updates. This option is not available if there are no matching tables for the report.
- D. Select a table style from the list. You can browse to another drawing and add any tables style on that drawing to the selection list.
- E. Specify whether to include any title information.
- F. Specify the method to use for calculating the width of the columns.
- G. Specify the XY coordinates for the upper left corner of the

TOP 5: HOW TO

table. If left blank, you are prompted to select a location after you click OK.

H. Enter a start and end line value. The default is to include all rows of report data.

I. Specify the options for the rows.

- **Apply Special Breaks:** If you selected any special breaks on the Report Generator box, you can break the table into separate table objects based on those values.
- **Rows for Each Section:** Specifies to break the report into separate table objects based on the value in the Rows box.

J. If table sections are defined:

- Specify the maximum number of table sections placed on each drawing. A blank value indicates an unlimited number of sections on one drawing.
- Specify the X and Y distance from the end of one table section to the beginning of the next. These values are used when there are multiple sections on the same drawing.

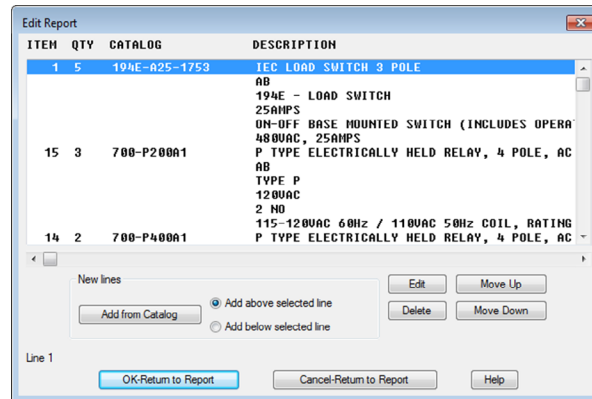
K. Click OK.

If you did not specify the XY coordinates, your cursor is a box approximating the size of the table when it is generated. To use object snap mode, enter an S at the command line.

EDIT A REPORT

If you want, you can modify the report before you insert it into your drawing or save it to a file. You can move data up or down in the report, add lines from a catalog and delete lines.

A. Generate a report.



B. In the Report Generator dialog box, select Edit Mode.

C. Select a row within the report.

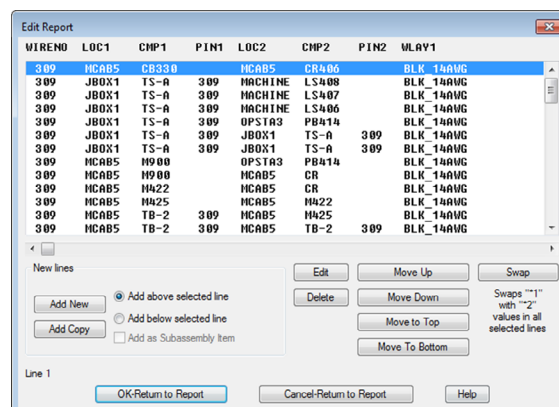
D. Click Edit to change individual values in that row.

E. Select a row or group of rows within the report.

- Click Delete to remove the selected rows.
- Use the available move options to change the order of the selected rows.

F. Use the available options to add extra rows with data to the report. Options differ depending on the report type.

G. If the report contains wiring information, each line item in the report can include two connected components, referred to



as the From and To components. The From field labels end in a

TOP 5: HOW TO

1, and the To field labels end in a 2.

Click Swap to swap the From and To field values.

H. Click OK-Return to Report.

POST-PROCESS A REPORT

Further customize the report data with a custom lisp function.

A. Generate a report.

B. In the Report Generator dialog box, select User Post.

C. Select from the available post-processing options.

D. Click Close.

The LISP routine processes the data and returns to the Report Generator Window.

Sample .lsp and .dcl files for each report are installed in the C:\Program Files\Autodesk\AutoCAD {version}\Acade\Support\{language code}\Shared\ folder. A few of the reports have sample options already defined. Otherwise, these options are user-created.

Note: See the Report Generator Dialog Box help topic for a list of user post file names.

6 MORE HELPFUL HOW-TOS

In order of importance, following are six helpful instructions for better utility of AutoCAD Electrical.

1. TITLE BLOCK SETUP

The title block is a border drawing inserted as an AutoCAD block on another drawing. The title block utility can update attributes on the title block.

- A. Start a blank new drawing.
- B. Draw your drawing border using standard AutoCAD commands and objects.
- C. Enter ATTDEF at the command prompt to insert attribute definition objects.
 - Note:** When the border drawing is inserted as a block on another drawing, attribute definition objects become attributes.
- D. Enter the Tag name, for example DESC1, DESC2, SHEET, SHEET_TOTAL.
- E. Set any other attribute definition properties and values.
- F. Select OK.
- G. Specify the insertion point.
- H. Repeat for each attribute definition for the title block.
- I. Save the border drawing as a DWG file.

This drawing is inserted as a block on another drawing. It can also be inserted when creating a drawing template file, DWT. A drawing template file is used to provide consistency in the drawings that you create by providing standard style, settings, layers, and border.

Note: Run Title Block setup with this border drawing file active if you are using the WD_TB attribute method to link AutoCAD Electrical values to the title block.

6 MORE HELPFUL HOW-TOS

2. SCHEMATIC COMPONENTS

Insert schematic components using any of the following methods:

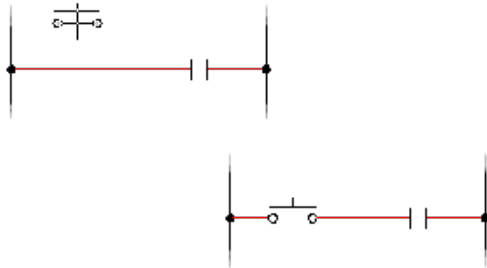
- **Catalog Browser**
Insert the component by selecting a catalog value from the catalog database.
- **Icon Menu**
Browse through the menu of available symbols organized by symbol type and parent and child symbols.
- **Copy a selected component.**
Includes an automatic update of the component tag.
- **User-defined pick list.**
Insert and annotate the schematic component with the catalog number or a component description.
- **Equipment list.**
This tool lists BOM data extracted from your equipment list and finds the appropriate schematic symbol by querying the schematic lookup database.
- **List of panel components.**

Let your project set of panel layout drawings help drive the schematic wiring diagrams. AutoCAD Electrical finds a match for the panel footprint in the schematic lookup database to determine the correct schematic symbol to insert.

After the schematic component is selected and inserted in the drawing, all panel-related information is copied to the schematic. Use the Insert/Edit Component dialog box to make any additional changes to the new schematic component.

6 MORE HELPFUL HOW-TOS

Pick an insertion point on the drawing. The orientation of the symbol tries to match the underlying wire. The wire breaks automatically if the symbol lands on it or very near it.



3. SCHEMATIC TERMINALS

Terminal symbols on the schematic are a representation of wire connection points. The terminal symbol representation on the schematic can have associations with the physical terminal block on the panel drawing. To insert a terminal, select the Insert Component command to display the icon menu, then select Terminals/Connectors.

There are four types of schematic terminal behavior that you can select from and five main terminal styles (square, round, hexagon, diamond, and triangle). Each type of terminal behavior is controlled by the terminal block name.



Non-intelligent terminals – do not show up in reports.

100



Terminals that take on a number – the number matches the wire number passing through or connected to the terminal.

6 MORE HELPFUL HOW-TOS

1

Terminals with a user-defined number.

99 1 10

Terminals that force a new number – the number is generated as a wire passes through the terminal.

4. WIRING

AutoCAD Electrical treats line entities as wires when the lines are found on an Electrical toolset-defined wire layer. You can have many wire layers set up on your drawing. Each wire layer has a descriptive name like "RED_16" or "BLK_14_THW" and is assigned a screen color to mimic the wire color visually. Wires do not have to begin or end at snap points, and they do not have to be orthogonal. They can be skewed at any angle.

A wire network is one or more wire line segments and optional branches that interconnect and form an electrically unbroken conductor. Wire segments of the network may contain inline terminals and wire crossing gaps. All segments of a wire network receive the same wire number unless you select On per Wire Basis in the Wire Number Options section of the Project Properties -> Wire Numbers dialog box (on the Project Manager, right-click the project name and select Properties). When multiple wires are tied to a common wire connection point, each wire is treated as an independent wire network and receives its own unique wire number assignment by AutoCAD Electrical.

Note: A wire connection point should only have up to three wire connections tied to it. Adding more wires to a single point prevents the angled wire connection to tie uniquely to the wire connection point.

INSERT WIRES

You can Insert wires individually, insert multiple wires, and interconnect components. Wires insert with automatic connection and wire crossing gaps or loops.

A. Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wires drop-down -> Wire.

Click the Insert Wires drop-down to access the Insert 22.5, 45 or 67.5 Degree Wire tools.

B. Select the starting point of the wire. You can start a wire segment in empty space, from an existing wire segment or from an existing component. If you start from a component, the wire segment is started at the nearest wire connection attribute to your pick point on that symbol.

During wire insertion, the current wire type displays at the command prompt.

- T = override the current wire type. Select a new wire type from the Set/Edit Wire Type dialog box. The new wire type becomes the current wire type, and the command continues with the wire insertion.
- X = show the wire connection points when the wire approaches a component.
- TAB = temporarily disable or enable collision checking during wire insertion.

This setting is remembered only for the current session. Collision checking is on when AutoCAD Electrical restarts.

- V = force a vertical direction for the wire segment.
- H = force a horizontal direction for the wire segment.
- C = insert the next wire segment at the current cursor location and continue wire insertion.

6 MORE HELPFUL HOW-TOS

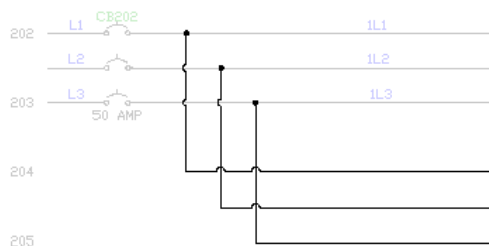
C. Select the ending point of the wire. You can end the wire segment or angled segment in empty space, from an existing wire segment or from an existing component. If it ends at a wire segment, a dot (wddot.dwg) is applied if appropriate. If it ends at another component, the nearest wire connection attribute is found and connected to your pick point on that symbol.

Note: If the distance between two horizontal wires is relatively small, the vertical wire crossing them avoids inserting loop gaps. The wire trap distance setting is used to see whether wire loop gaps are possible or not. Reduce the scale factor or increase the distance between two wires to insert gap loops.

If the wire was an angled wire, press “N” to switch to normal 90-degree wire mode. The wire picks up at the end of the angled wire segment and defaults to horizontal or vertical.

Note: A wire connection point has up to three wire connections tied to it. Adding more wires to a single point prevents the angled wire connection to tie uniquely to the wire connection point.

INSERT MULTIPLE WIRES



A. Click Schematic tab -> Insert Wires/ Wire Numbers panel -> Multiple Bus.

B. Set the horizontal and vertical spacing for the wires.

C. Specify where to start the wires.

D. Set the number of wires to 3, and click OK.

During wire insertion, the current

6 MORE HELPFUL HOW-TOS

wire type displays at the command prompt. The current wire type indicates the layer name in which the new wires are drawn. You can override it by typing "T" and selecting a new wire type from the Set/Edit Wire Type dialog box. The new wire type becomes the current wire type, and the command continues with the wire insertion.

E. If starting at a component or bus, select the component. Drag the cursor slowly to the right as the second and third phases latch on to their appropriate vertical wires. You can see the three wires stretch straight across the screen.

As you pull the 3-phase wire out, you can turn a corner by moving your cursor out of line with the bus. To reverse the phase sequence of the turn, press F.

F. Right-click to terminate the wires. The wires and wire connection dots insert, and loops are automatically inserted at wire crossing points.

Note: If the distance between two horizontal wires is relatively small, the vertical wire crossing them avoids inserting loop gaps. The wire trap distance setting is used to see whether wire loop gaps are possible or not. Reduce the scale factor or increase the distance between two wires to insert gap loops.

To tie the new 3-phase wire to an existing bus, but with reversed sequence, start the new 3-phase wire connected at the last wire on the existing bus. Move the cursor backward across the other wires until the connections are made, then move the cursor forward again. This results in a reversed sequence connection.

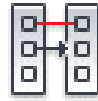
Tip: If you have trouble connecting a new 3-phase wire to an existing bus, start the command and select the starting point on the existing bus. Move the cursor slowly across the other wires of the bus. AutoCAD Electrical has a better chance of finding them and correctly connecting

6 MORE HELPFUL HOW-TOS

the new wiring.

INTERCONNECT COMPONENTS

- A. Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wires drop-down -> Interconnect Components.
- B. Select the first component.
- C. Select the second component.



Note: Wires are added only if the wire connection points are aligned.

5. SYMBOL BUILDER

Defines new AutoCAD Electrical component, terminal, and panel layout library symbols.

You can convert symbols or create custom components on the fly. Symbols created or converted using Symbol Builder are fully compatible with AutoCAD Electrical. They break wires upon insertion, and appear in the bill of material and various component and wire connection reports.

You can exit the Symbol Builder command and re-enter it at any time. You can also exit the command and use regular AutoCAD commands to edit or finish the symbol you are creating. The AutoCAD Wblock command writes it to disk. Each time you re-enter the Symbol Builder tool, select objects from within the Select Symbol/Objects dialog box. Selecting the objects allows the tool to track what standard attributes and wire connection points you already inserted.

New symbols you create are inserted with the AutoCAD Electrical Insert Component or Insert Panel Component commands. You can add your

6 MORE HELPFUL HOW-TOS

new symbol to the icon menu. You can also select it from the Type it or Browse options in the bottom left-hand corner of the icon menu.

CREATE AN ELECTRICAL TOOLSET SYMBOL USING SYMBOL BUILDER

Creating an AutoCAD Electrical symbol includes adding the necessary attributes based on the symbol type. For a schematic symbol, it involves selecting an appropriate file name. Symbol builder uses attributes templates to make it easier to add the attributes. If you select a schematic symbol type, symbol builder suggests a file name based on the AutoCAD Electrical naming conventions.

TO START SYMBOL BUILDER

1. Click Schematic tab -> Other Tools panel -> Symbol Builder drop-down -> Symbol Builder.
2. Select options on the Select Symbol/Objects dialog box, making sure to select the attribute template library path, symbol and type.
3. Click OK to enter the block editor environment.

CREATE A SYMBOL

- A. Insert attributes using the Symbol Builder Attribute Editor.
- B. For schematic symbol: Insert wire connection attributes.
- C. For schematic symbol: Insert link line attributes.
- D. Optional: Audit to see any potential issues with the symbol.
- E. Save the symbol.

INSERT ATTRIBUTES

6 MORE HELPFUL HOW-TOS

- A. Select options on the Select Symbol/Objects dialog box.
- B. Click OK to enter the block editor environment.
- C. If the Symbol Builder Attribute Editor is not visible, click Symbol Builder tab -> Edit panel -> Palette Visibility Toggle.
- D. Select the attribute you want to insert.
- E. Click the Insert Attribute tool.

Note: You can also right-click and select Insert Attribute or drag the attribute to insert it.

- F. Select an insertion point for the attribute.

ADD AN ATTRIBUTE TO THE LIST

- A. Select the Add Attribute tool.
- B. Define the attribute tag and properties on the Insert/Edit Attributes dialog box.
- C. Click Insert to insert the new attribute, or click OK to add it to the list.

INSERT WIRE CONNECTIONS

Symbol Builder inserts a wire connection template drawing when adding a wire connection to your symbol. The list of wire connection options is built dynamically based on the template drawings found in the symbol library path.

- A. If the Symbol Builder Attribute Editor is not visible, click Symbol Builder tab -> Edit panel -> Palette Visibility Toggle.
- B. Click the arrow on the Wire Connection section of the Symbol Builder Attribute Editor to expand the wire connection section.
- C. Click the Direction/Style list to expand the list of wire

connection options.

D. Select a wire connection.

Note: Only the options for the default style are shown in the list. To see other styles or change the default style, select Others.

Others

Selecting Others on the Wire Connection Direction/Style list opens the Insert Wire Connection dialog box. On the Insert Wire Connection dialog box you can insert multiple wire connection attributes, select from a style other than the default, or change the default style.

- i. Define the style and direction.
- ii. Enter the number of wire connections.
- iii. Optional: Select an attribute in the Pin Information section and click Convert. Select the text for conversion as prompted.
- iv. Optional: Select an attribute in the Pin Information section and click Delete. The attribute is removed from the list and is not inserted with the wire connection attribute.
- v. Optional: Select an attribute in the Pin Information section and click Properties to define the properties for the attribute using the Insert/Edit dialog box.

Note: Pin attributes added with each wire connection may differ based on symbol type.

- vi. Click Insert.

6 MORE HELPFUL HOW-TOS

- vii. Select the wire connection attribute insertion points.
The related pin attributes are inserted relative to the wire connection attribute based on the wire connection template.

- E. Click the Insert Wire Connection tool.
- F. Select an insertion point for the attribute.

Note: If the wire connection template contains the optional TERMn and TERMDESCn attributes, they are inserted with the wire connection attribute and added to the Pins section.

INSERT LINK LINE ATTRIBUTES

- A. If the Symbol Builder Attribute Editor is not visible, click Symbol Builder tab -> Edit panel -> Palette Visibility Toggle.
- B. Click the arrow on the Link Lines section of the Symbol Builder Attribute Editor to expand the section.
- C. Click the Direction list to expand the list of link line options.
- D. Select a direction.
- E. Click the Insert Link Line tool.
- F. Select an insertion point for the attribute.

SAVE THE SYMBOL

- A. Click Symbol Builder tab -> Edit panel -> Done.
The Save Symbol dialog box is displayed.
- B. Select WBlock in the Destination section.
- C. Modify the block name as needed.
- D. Modify the WBlock File path as needed.
- E. Specify the base point for symbol insertion.

6 MORE HELPFUL HOW-TOS

- F. Check Icon image to create a .png file. This image file can be used if you add the symbol to the icon menu.

Note: The symbol is not automatically added to the icon menu but can be added using the Icon Menu Wizard.

- G. Modify the image name as needed.
- H. Modify the image File path as needed. The default path is the image folder for the current icon menu.
- I. Click Details to examine any errors found in the symbol audit.
- J. Click OK to save the symbol.

Note: If you close the block editor without saving, the dialog box opens automatically. Select the No button to close the block editor without saving the symbol changes.

START WITH A NON-AUTOCAD ELECTRICAL BLOCK

- A. Click Schematic tab -> Other Tools panel -> Symbol Builder drop-down -> Symbol Builder.
- B. Browse to the existing block to select the symbol to create or edit.
- C. In the Select Symbol/Objects dialog box, Attribute template section: Browse to the Library path, for example C:\Users\Public\Documents\Autodesk\Acade {version}\Libs\jic125 .
- D. In the Attribute template section: Choose Symbol: Horizontal Parent, for example.
- E. In the Attribute template section: Choose Type: Generic, for example.
- F. Select OK.
- G. Convert existing attribute or text objects to AutoCAD Electrical attributes.

6 MORE HELPFUL HOW-TOS

H. Add wire connections as needed.

I. Click Symbol Builder tab -> Edit panel -> Done.

A default symbol name is supplied, which you can keep or change as needed depending on the symbol type and symbol naming conventions.

If the existing symbol contains attribute or text objects you can convert these to the expected attributes for the symbol type.

- A. If the Symbol Builder Attribute Editor is not visible, Click Symbol Builder tab -> Edit panel -> Palette Visibility Toggle.
- B. Select the Convert Text to Attribute tool to open the dialog box. All the attributes and text objects contained in your non-AutoCAD Electrical block are in the left-hand list. The AutoCAD Electrical attribute names are in the right-hand list.
- C. Select an existing attribute/text from the left-hand list. Click the arrow next to the attribute in the right-hand list.
- D. Repeat for each non-AutoCAD Electrical attribute or text object you want to convert.
- E. Select Done.

CONVERT TEXT TO ATTRIBUTE DEFINITIONS

- A. Convert a single text object
 - i. If the Symbol Builder Attribute Editor is not visible, click Symbol Builder tab -> Edit panel -> Palette Visibility Toggle.
 - ii. Select the attribute from the list.
 - iii. Click the Convert Text tool at the top of the section for the attribute.
 - iv. Select the text object.
- B. Convert multiple text objects to attribute definitions

6 MORE HELPFUL HOW-TOS

- i. Select the Text Convert tool at the top of the Symbol Builder Attribute Editor to launch the Convert Text to Attribute dialog box.
- ii. Select the text within the Text list.
- iii. Select the arrow next to the attribute name in the Attribute list.
- iv. Repeat for each text object you want to convert.
- v. Click OK.

Note: The text value becomes the default value for the attribute when the block is inserted on a drawing.

6. WIRE LAYERS

Use the Create/Edit Wire Type tool to create new or edit existing wire types. Use the Change/Convert Wire Type tool to convert lines to wires. The wire layer name and the associated wire properties (such as wire color, size, and whether the wire layer is processed for wire numbers) are saved in the drawing file.

The following rules determine the wire layer for a new wire:

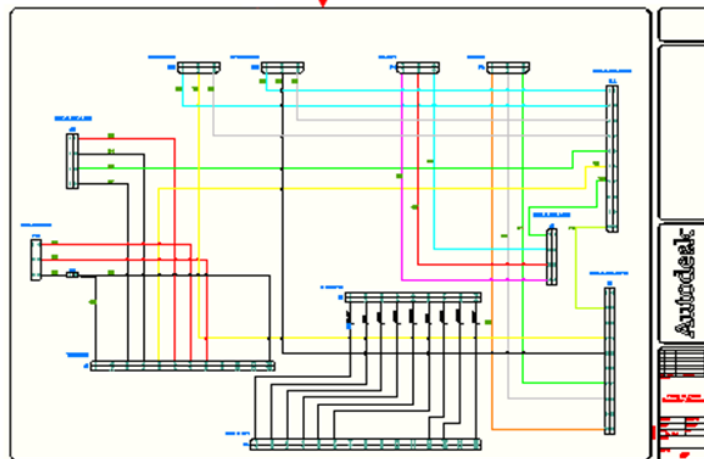
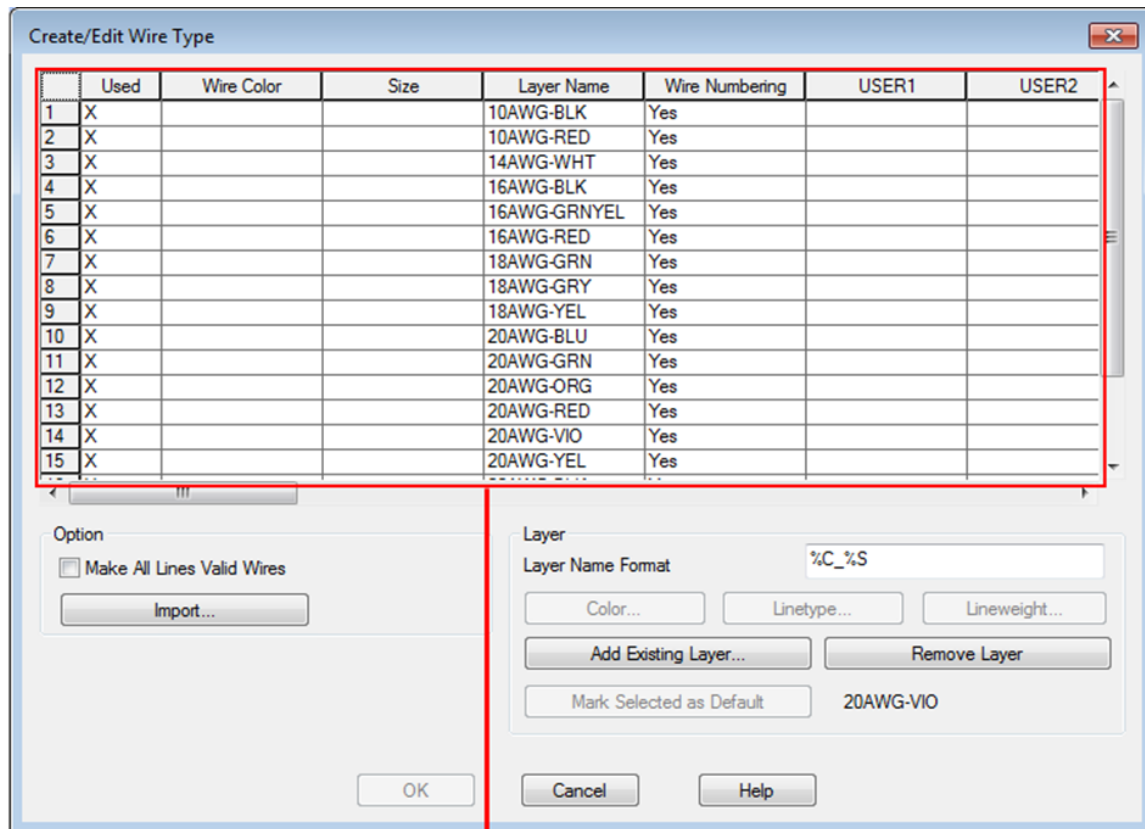
- When a wire is created from an existing wire, the new wire takes on the same layer as the existing wire. It ignores the current layer and the current wire type.
- When the new wire is started in empty space but ends at an existing wire, the new wire takes on the wire layer of the ending wire. The current layer and current wire type are ignored.
- When a new wire is started at an existing wire and ends at another existing wire, the new wire takes on the layer of the beginning wire.
- If there are no wire layers in the drawing, the new wire is

6 MORE HELPFUL HOW-TOS

drawn in the WIRES layer.

- When a wire starts in empty space and ends at the component wire connection point, the new wire is drawn on the current wire type. The layer of the wires already tied to the same component connection points are ignored. The same is true for a wire that starts at the component wire connection point and ends in empty space.

6 MORE HELPFUL HOW-TOS



6 MORE HELPFUL HOW-TOS

CREATE WIRE TYPES

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Create/Edit Wire Type.
 - B. In the Create/Edit Wire Type dialog box, click inside the Wire Color column for a blank row and specify a value for the new wire layer.
 - C. Click inside the Size column and specify a value for the size. The Layer Name is automatically created. If you specified Wire Color: Red and Size: 20, the name RED_20 is assigned to the wire layer you are creating.
 - D. If you do not want wires on this layer processed for wire numbers, select No for the Wire Numbering option.
 - E. Click Color, Linetype or Lineweight to assign values for the new layer.
- Note: If you want the new wire layer to be the default, click Mark Selected as Default.
- F. Click OK.

CREATE A WIRE TYPE BY ADDING AN EXISTING LAYER

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Create/Edit Wire Type.
 - B. In the Create/Edit Wire Type dialog box, click Add Existing Layer.
 - C. In the Layers for Line "Wires" dialog box, define the layer name and click OK. You can either enter a name in the edit box or click Pick to select a name from the existing layer list.
- The layer displays in the wire type grid. If you selected the wrong wire layer, highlight the layer in the dialog box and click Remove Layer. You can then go back into the Layers for Line "Wires" dialog box and select another layer to add.

6 MORE HELPFUL HOW-TOS

- D. In the Create/Edit dialog box, click Color, Linetype or Lineweight to assign new values for the layer.
- E. If you do not want wires on this layer processed for wire numbers, select No for the Wire Numbering option.
- F. Click OK.
- D. If you do not want wires on this layer processed for wire numbers, select No for the Wire Numbering option.
- E. Click Color, Linetype or Lineweight to assign

CREATE A WIRE TYPE FOR EXTERNAL JUMPER WIRES

Wires on jumper layers do not display in Wire From/To reports.

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Create/Edit Wire Type.
- B. In the Create/Edit Wire Type dialog box, enter a layer name that includes the word "JUMPER".
- C. If you do not want wires on this layer processed for wire numbers, select No for the Wire Numbering option.
- D. Click Color, Linetype or Lineweight to assign values for the new layer.
- E. Click OK.

RENAME A WIRE TYPE

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Create/Edit Wire Type.
- B. Right-click on a Layer Name cell and select Rename Layer.
- C. Enter the layer name.
- D. Click OK.

6 MORE HELPFUL HOW-TOS

SET THE DEFAULT WIRE TYPE

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Create/Edit Wire Type.
- B. Select a wire layer row in the grid.
- C. Click Mark Selected as Default.
- D. Click OK.

DELETE A WIRE TYPE

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Create/Edit Wire Type.
- B. Select a wire layer row in the grid.
- C. Click Remove Layer. **Note:** The default wire layer cannot be removed.
- D. Click OK.

Note: The layer is not deleted from the drawing. The layer is removed from the wire layer list, so any lines on that layer are no longer considered wires.

IMPORT WIRE TYPES

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Create/Edit Wire Type.
- B. Click Import. The Wire Type Import - Select Master Drawing dialog box displays.
- C. Select the drawing or drawing template containing the wire types for import.
- D. Click Open. The Import Wire Types dialog box displays.
- E. Select the wire types for import.

6 MORE HELPFUL HOW-TOS

- F. Define how the import function behaves if a wire type exists on the active drawing.
 - Overwrite any Wire Numbering and USERn differences – If checked, this changes the wire numbering setting and all the USER values for the existing wire type to match the imported wire type.
 - Update any layer color and linetype differences – If checked, this changes the color and linetype settings for the existing wire layer to match the imported wire layer.
- G. Click OK. The selected wire types and settings display in the Create/Edit Wire Type dialog box.
- H. Continue adding, importing and editing wire types in the Create/Edit Wire Type dialog box.
- I. Click OK.

IMPORT WIRE TYPES PROJECT-WIDE

1. Click Project tab -> Project Tools panel -> Utilities.
2. Select the Import from specified drawing check box.
3. Browse to or enter the name of the drawing or drawing template containing the wire type definitions for import.
4. Click Setup to display the Import Wire Types dialog box.
5. Select the wire types for import.
6. Define how the import function behaves if a wire type exists on the drawing being processed.
 - **Overwrite any Wire Numbering and USERn differences** – If checked, this changes the wire numbering setting and all the USER values for the existing wire type to match the imported wire type.
 - **Update any layer color and linetype differences** – If checked, this changes the color and linetype settings for the existing wire layer to match the imported wire layer.

6 MORE HELPFUL HOW-TOS

7. Click OK and return to the Project-Wide Utilities dialog box.
8. Click OK. The Select Drawings to Process dialog box displays.
9. Select the drawings you want to import the selected wire types into.
10. Click OK.

SELECT WIRE TYPE WHILE INSERTING WIRES

- A. Select one of the following commands:
 - Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wires drop-down -> Wire.
 - Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wires drop-down -> 22.5 Degree.
 - Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wires drop-down -> 45 Degree.
 - Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Wires drop-down -> 67.5 Degree.
 - Click Schematic tab -> Insert Wires/Wire Numbers panel -> Multiple Bus.
 - Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Ladder drop-down -> Add Rung.
 - Click Schematic tab -> Insert Wires/Wire Numbers panel -> Insert Ladder drop-down -> Insert Ladder.
- B. Enter **T** on the command line.
- C. Select a wire type from the Set Wire Type dialog box.

The new wire type becomes the current wire type, and the command continues with the wire insertion.

Note: If you select Another Bus (Multi-Wire) in the Multiple Wire Bus dialog box, the wires are drawn on the same wire layer as the existing wire bus. You cannot type T to change the wire type during wire insertion.

6 MORE HELPFUL HOW-TOS

CHANGE WIRE TYPES

- A. Click Schematic tab -> Edit Wires/Wire Numbers panel -> Modify Wire Type drop-down -> Change/Convert Wire Type.

Optionally, you can right-click on an existing wire and select Change/Convert Wire Type.

- B. In the Change/Convert Wire Type dialog box, select a wire type record in the wire type list, or click Pick to select a wire type record from the drawing.

If you right-clicked on a wire and selected Change/Convert Wire Type, the wire type corresponding to the selected wire layer is highlighted in the list.

- C. Make any selections in the dialog box.

If Change all wires in the wire network is selected, all wires in the wire network are changed to the new wire type. If unselected, only the selected wire is changed.

If Convert Lines to Wires is selected, the selected lines are changed to the new wire type. If unselected, the lines are ignored.

- D. Click OK.
E. Select the wires or lines in the drawing to change and press Enter.

RENAME THE USER1- USER20 COLUMN HEADERS

You can rename the User1-User20 column headers on the Create/Edit

6 MORE HELPFUL HOW-TOS

Wire Type dialog box.

- A. Right-click the project name in the Project Manager and select Properties.
- B. In the Project Properties -> Wire Numbers dialog box, Wire Type section, click Rename User Columns.
- C. In the Rename User Columns dialog box, enter new column titles.
- D. Click OK.

Note: Renaming of user-defined columns is project-specific. You cannot rename the Color, Size or Layer Name columns.

AUTOCAD ELECTRICAL TIPS

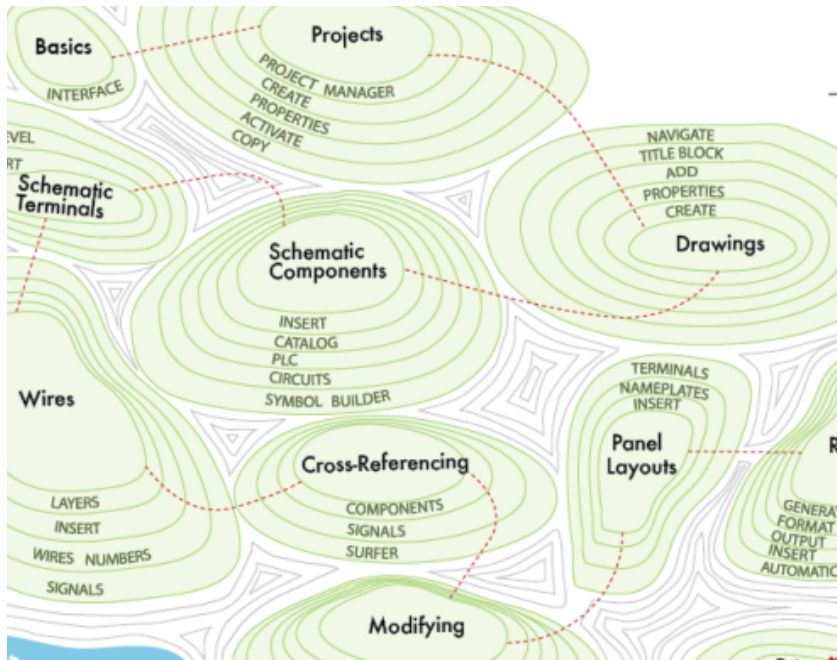


Image: Autodesk

AutoCAD Electrical includes a standard tabbed ribbon across the top of the drawing area. Electrical toolset-specific tabs are included along with the AutoCAD ribbon tabs.

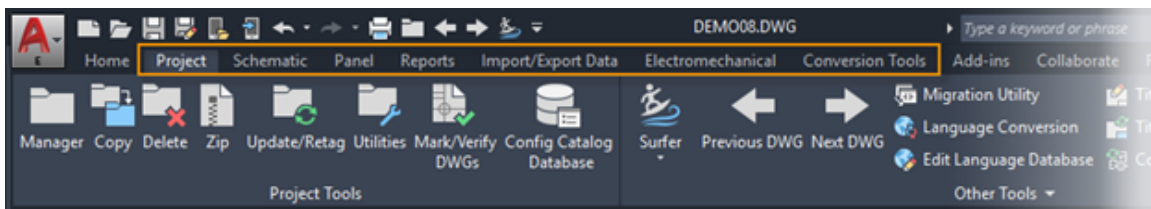


Image: Autodesk

You can access all functionality shown in these tips from the ribbon. A Quick Access toolbar also includes some navigation commands used in AutoCAD Electrical.

MARKING

The marking menu is in a radial context. When you place your mouse over an electrical object and right-click, the options on the menu will depend on the type of object under the cursor.

PALETTES

A palette can be used for managing projects and for working with the catalog database. Palettes can remain on the screen while other commands are used, remaining available when needed.

A palette can be docked, resized and set to auto-hide, controlling the amount of screen area it uses. If set to auto-hide, the palette will expand automatically when the cursor is hovered over it.

PROJECT MANAGER

Use Project Manager to create, activate, copy, and delete a project or modify information associated with a project.

To launch Project Manager: Click Project tab Project Tools panel Manager.

7 STEPS FOR CREATING A PROJECT

1. Start with one of the following:
 - On Project Manager click New Project.
 - Right-click at the bottom of the tree inside the Project Manager and select New Project.
 - Click the arrow on the Project drop-down menu and select New Project.
2. Enter the project's name. (The .wdp extension is not required.)
3. Select or create the directory for the project. If this field is left blank, the project WDP file will be created at the location defined in the WD.ENV file.
4. If you want, you can copy the project settings over from an existing project file (WDP). If you leave the field blank, the properties will automatically be copied from the active project.
5. As an option, you can click Descriptions to enter an unlimited number of descriptions for the project. Use the check boxes to include description lines in reports.
6. Also optional, you can click OK-Properties to modify your default project settings for libraries, icon menus, components, wire numbering, cross-references, styles, and drawing formats.
7. Finally, click OK.

SETTING UP AUTOCAD ELECTRICAL FOR MULTIPLE USERS

You can manually move any shared files to a new central location after installation by using normal Microsoft Windows operations to cut or copy and paste from their local location to a central shared location.

These shared files are located by AutoCAD Electrical as long as they are placed in:

- the AutoCAD Electrical defined path (such as in the project's subdirectory);
- the path given by the AutoCAD Electrical environment variable; or
- AutoCAD search paths.

To ensure that the data is being located correctly, back up your information in another location and remove the shared data from your local drive.

SHARED FILES

The following shared files can be pasted from your local machine to a shared location. The table lists the file names, default location and any WD.ENV file lines that need to be modified.

The main executables and static support files are located under C:\Program Files [(x86)]\Autodesk\AutoCAD {version}\Acade\.

The user-modifiable support files and database content are found under C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\.

Although it is not required that you share these files, sharing makes it easier for multiple users to work with projects in AutoCAD Electrical.

ABOUT THE AUTHOR



Todd Schmooch

Navigate CAD with a new perspective working alongside Todd Schmooch, Senior Application Specialist at Applied Software, Graitec Group.

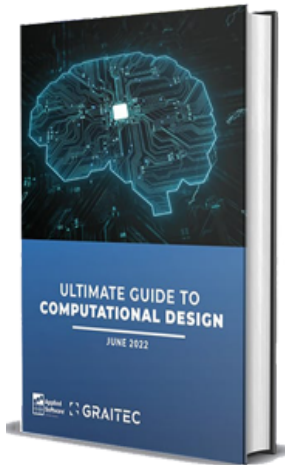
You can benefit from Todd's real-world experience when you work with him on training, consulting, helpdesk support, and pre-sales and post-sales support. Your company can harness innovation while drawing on Todd's decade of experience with government contracts and strict standards for drafting and design documentation.

Todd's industry experience includes:

- Documentation specialist/designer, Honeywell, Inc.
- CAD management for technical ceramic, elevator and specialty gas industries
- Degree from Delaware Valley University, Doylestown, PA

DID YOU LIKE A TECHNICAL GUIDE TO AUTOCAD ELECTRICAL

THEN CHECK OUT OUR OTHER GUIDES BELOW



THE ULTIMATE GUIDE TO COMPUTATIONAL DESIGN

World-renowned experts created an all-encompassing guide for everything computational design.

[CLICK TO READ](#)



THE ULTIMATE GUIDE TO AUTODESK INVENTOR

A practical guide to ramping up your knowledge and abilities in Inventor and its add-ons and integrations.

[CLICK TO READ](#)



©Applied Software Technology, Inc. 2022