



AUTODESK UNIVERSITY 2015

IM10423

AutoCAD Electrical 2016 What's New

Randy Brunette
Autodesk, Inc.

Gaurav Sachdeva
Autodesk, Inc.

Learning Objectives

- Discover possible new features of AutoCAD Electrical 2017
- Discover how new features will aid design workflow
- Contribute ideas for improving AutoCAD Electrical
- Share the excitement of the updates of AutoCAD Electrical

Description

Do you want to see what's new in the 2016 release? Join us in a demonstration and discussion about the new tools and features

Your AU Experts

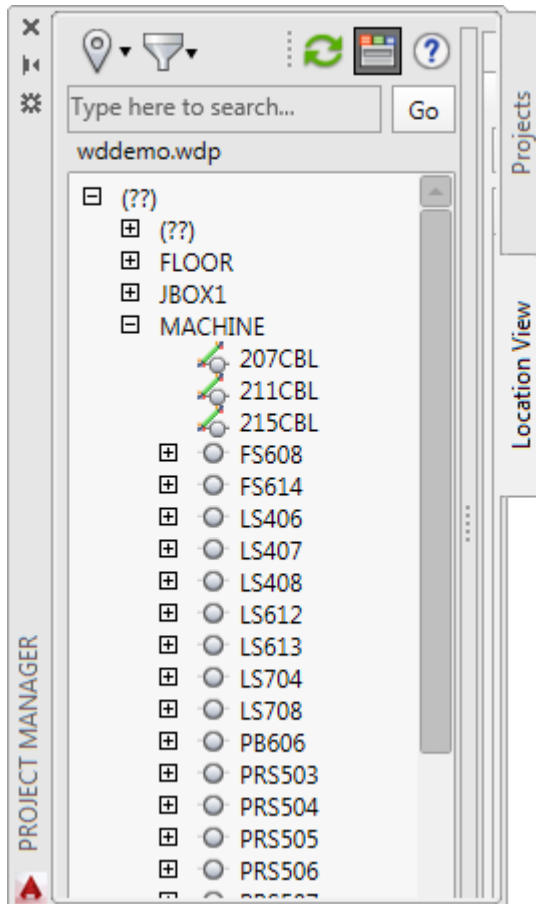
Randy Brunette joined Autodesk, Inc., as an electrical subject matter expert. Randy's duties include helping channel partners and customers through mentoring and understanding their business issues and finding solutions that solve their challenges. Randy has been in the design field using Autodesk products for over 30 years, with experience across multiple segments of the manufacturing industry. He has been in an application engineer role for 21 years. Prior to joining Autodesk, Randy was the sole proprietor of a consulting business specializing in AutoCAD Electrical software, traveling in North America and Europe providing consulting services. Randy is a top-rated speaker at Autodesk Universities, Technical Academies, and seminars. He has authored AutoCAD Electrical software training manuals, videos, and other materials.

Gaurav Sachdeva is product manager for AutoCAD Architecture software, AutoCAD MEP software, AutoCAD Electrical, and AutoCAD Mechanical software. An MPhil in computer science from University of Cambridge and an MBA from Indian School of Business (Hyderabad), Gaurav has regularly spoken at entrepreneurial summits. An artist and a musician on the right side of his brain, he is keenly interested in design and has driven the next-generation strategy and execution of an array of successful products.



What's New in AutoCAD Electrical 2016

Location View Tab on Project Manager

The new Location View tab on Project Manager provides a complete view of the devices and wiring contained in the active project. Components are organized by installation and location values.

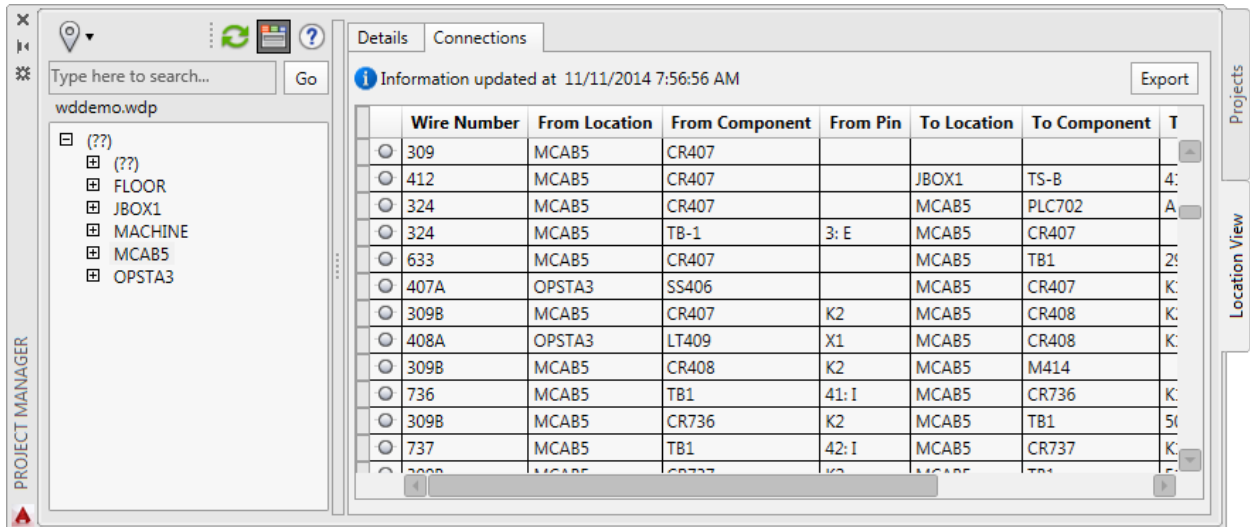


Use the Location View tab to:

- Hover over a device to view catalog, description, and rating information.
- Expand the device to view the component pin values.
- Filter the list based on installation and location values. 
- Refresh the view to reflect changes across the project. 
- Locate a device in the list by entering a search string.
- Right-click to surf to a selected component.



- Display the Details pane to view component details and the Connections pane to view wiring from\to information.



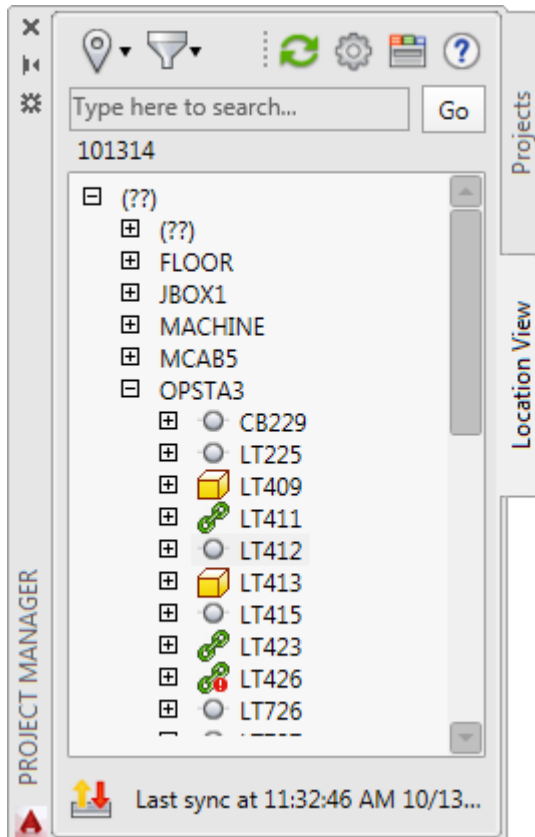
- Export the data shown in the Details or Connections pane to a .csv or .xls file.

	A	B	C	D	E	F	G	H	I
1	Wire Number	From Location	From Component	From Pin	To Location	To Component	To Pin	Cable Installation	Cable L
2	309	MCAB5	CR407						
3	412	MCAB5	CR407		JBOX1	TS-B	412: E		
4	324	MCAB5	CR407		MCAB5	PLC702	A		
5	324	MCAB5	TB-1	3: E	MCAB5	CR407			
6	633	MCAB5	CR407		MCAB5	TB1	29: I		
7	407A	OPSTA3	SS406		MCAB5	CR407	K1		
8	309B	MCAB5	CR407	K2	MCAB5	CR408	K2		
9	408A	OPSTA3	LT409	X1	MCAB5	CR408	K1		
10	309B	MCAB5	CR408	K2	MCAB5	M414			
11	736	MCAB5	TB1	41: I	MCAB5	CR736	K1		
12	309B	MCAB5	CR736	K2	MCAB5	TB1	50: I		
13	737	MCAB5	TB1	42: I	MCAB5	CR737	K1		
14	309B	MCAB5	CR737	K2	MCAB5	TB1	51: I		
15	202A	MCAB5	DS219		MCAB5	FU207			
16	220	MCAB5	DS219		MCAB5	FU221			
17	220A	MCAB5	DS219		MCAB5	FU222A			
18	203A	MCAB5	DS219		MCAB5	FU207			





AutoCAD Electrical and Inventor Interoperability

You can now link an AutoCAD Electrical project to an Inventor project.  Find








The link provides data exchange between your 2D and 3D electrical designs. Once your project is linked, use the Location View tab to view both the AutoCAD Electrical and Inventor components and data.

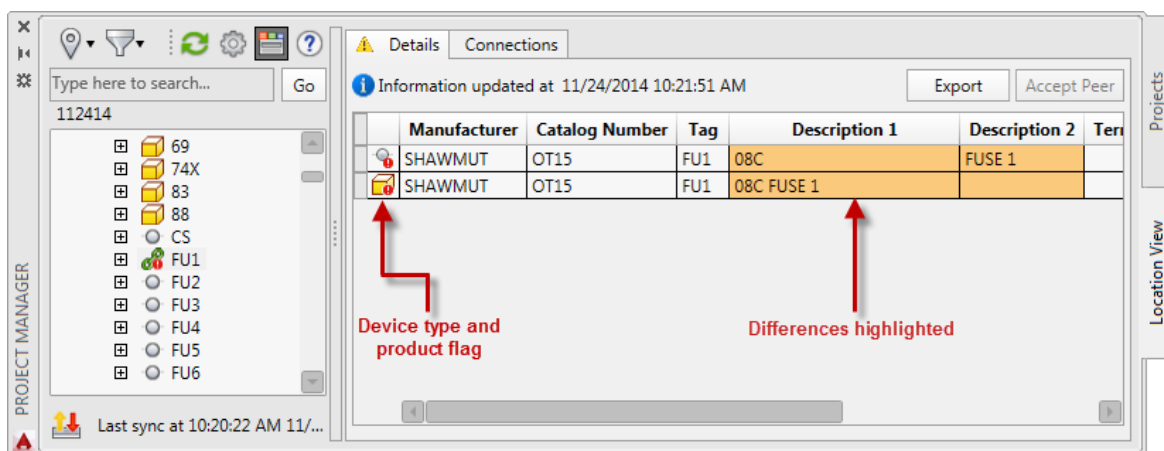


Once the projects are linked you can:

- Link a component in AutoCAD Electrical with a part in Inventor using one of the following methods:
 - **Insert from Catalog Browser.** Performs a catalog lookup, using the catalog information from the Inventor part, where you can insert a symbol to link to the Inventor part.
 - **Link to Existing in Drawing.** Select a symbol from the drawing to link to the Inventor part.
- View the link status of a device as indicated by the icon next to the device tag:
 -  An AutoCAD Electrical component not linked to an Inventor component
 -  An Inventor component not linked to an AutoCAD Electrical component
 -  Linked AutoCAD Electrical and Inventor components
 -  Linked, but there is a mismatch between the AutoCAD Electrical and Inventor components



-  An AutoCAD Electrical cable not linked to an Inventor cable
-  An Inventor cable not linked to an AutoCAD Electrical cable
-  Linked AutoCAD Electrical and Inventor cable
-  Linked, but there is a mismatch between the AutoCAD Electrical and Inventor cables
- Filter the view by link status. 
- Refresh the data from Inventor. 
- Use the Details and Connections pane to view differences between the linked components. 



- Use the Accept Peer button to reconcile the differences between the linked components.

This same functionality is available in Inventor from the new Electromechanical tab on the Assembly ribbon.

Note: This was an idea from the [AutoCAD Electrical IdeaStation](#).

Default Wire Sequence

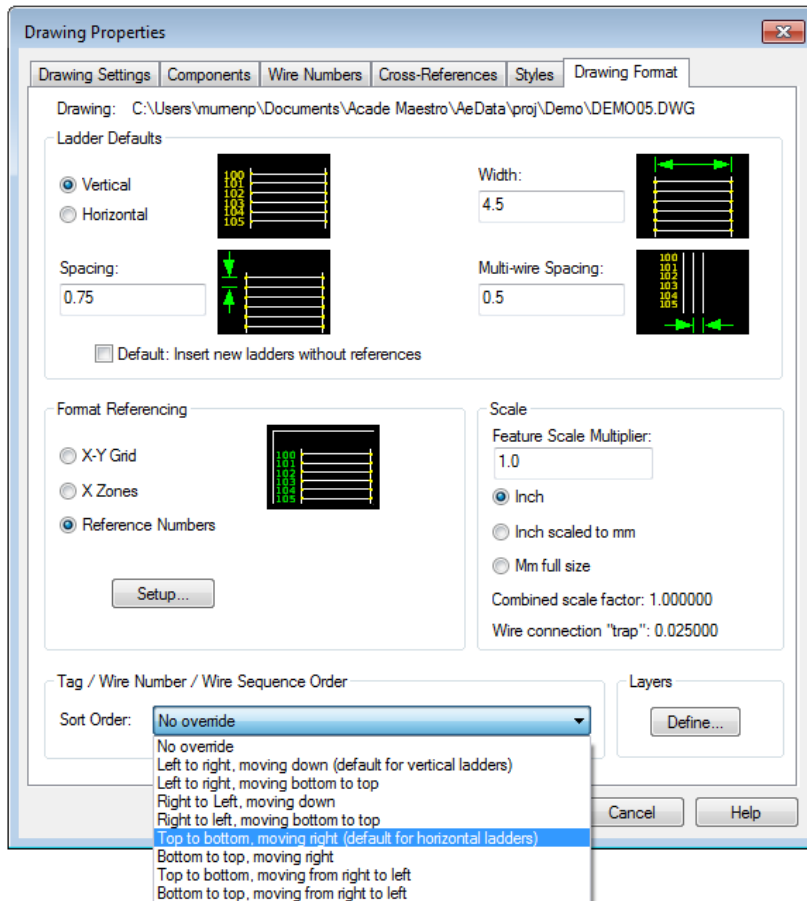
The drawing and project property for sort order now applies to wire sequence as well as tag and wire number order. This setting provides default wire sequencing for wire networks with multiple components.

AutoCAD Electrical performs the following steps when determining wire connection sequence:

1. Check for a wire sequence defined using the Edit Wire Sequence command, including any Direct-to-Terminal connections.
2. Check for angled tees on the wire network.
3. Check if any of the components have the same Location.



4. Use the Sort order drawing property to determine the connections for anything not accounted for in the previous checks.



Wire Number Attribute Support

Edit the wire number attributes, W01USER-W10USER, on the Edit Wire Number/Attributes dialog box. These values are now available fields in the following reports:

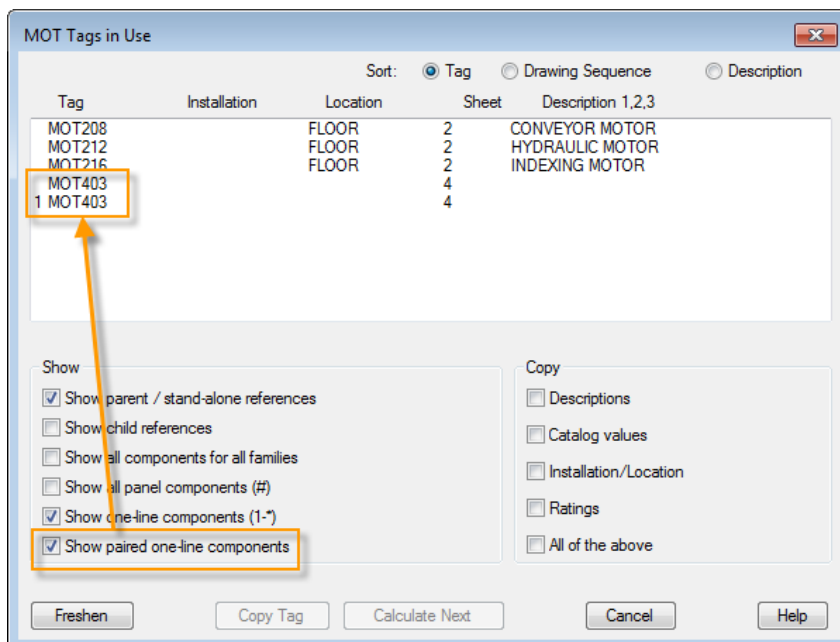
- Wire From\To
- Component Wire List
- Connector Plug
- Terminal Plan
- Cable From\To

Insert Connector

The Insert Connector dialog box settings are maintained across sessions of AutoCAD Electrical. Once you set your default settings, you do not have to set them again. The product supports two groups of settings, one for metric drawings and another for imperial drawings. The appropriate settings are used automatically based on the active drawing.

Annotate One-Line Components

When working on a one-line diagram you may want to annotate a one-line component based on a schematic or panel component already inserted. To make it easier, a new option is available to filter the list of components based on whether there is already a matching one-line symbol.



Select this check box to include all schematic, panel, and one-line references where there is a matching one-line component for either a schematic component or a panel footprint.



Note: The one-line reference is only included if the Show one-line components box is also checked.

Clear this check box to filter out all schematic, panel, and one-line references where there is a matching one-line component for either a schematic component or a panel footprint. The resulting list contains only tags that do not have matching one-line components.

MOT Tags in Use

Sort: ☒ Tag ☐ Drawing Sequence ☐ Description

Tag	Installation	Location	Sheet	Description 1,2,3
MOT208		FLOOR	2	CONVEYOR MOTOR
MOT212		FLOOR	2	HYDRAULIC MOTOR
MOT216		FLOOR	2	INDEXING MOTOR

Show

- ☒ Show parent / stand-alone references
- ☐ Show child references
- ☐ Show all components for all families
- ☐ Show all panel components (#)
- ☒ Show one-line components (1-*)
- ☐ Show paired one-line components

Copy

- ☐ Descriptions
- ☐ Catalog values
- ☐ Installation/Location
- ☐ Ratings
- ☐ All of the above

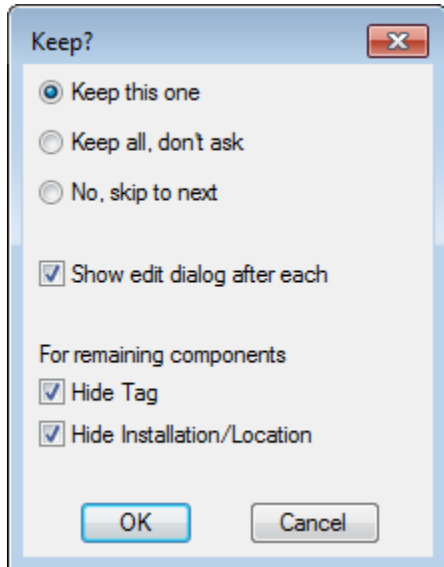
Freshen Copy Tag Calculate Next Cancel Help

Note: This was an idea from the [AutoCAD Electrical IdeaStation](#).



Schematic Terminals

If you insert schematic terminals using the Multiple Insert commands, there are now options to hide the tagstrip, installation, and location attribute values.



When using Multiple Insert (Icon Menu) the options are available after the first terminal is inserted. When using Multiple Insert (Pick Master), the options are available for each terminal inserted, including the first one.

Note: This was an idea from the [AutoCAD Electrical IdeaStation](#).

Miscellaneous Enhancements

- Child components linked directly to the parent with the dashed link line are filtered out of the cross-referencing on the parent component.
- Buttons for rating defaults added to the View / Edit Rating Values dialog box when editing a panel component. Each button opens the Ratings default file so you can select a value to assign to the attribute.
- The Insert\Edit Component dialog box opens when you insert a parametric connector so you can further annotate the connector.
- The Copy Circuit command defaults to selecting multiple insertion points for creating multiple copies on the same drawing.
- A new Circuit Builder API is available to link components within a circuit with a dashed link line.
- The Export to Spreadsheet now freshens the database automatically before performing the export.

Note: This was an idea from the [AutoCAD Electrical IdeaStation](#).

- A new check box on Terminal Strip Editor lets you save the internal and external device assignments.



What's New in AutoCAD Electrical 2015

Catalog Browser

The Catalog Browser is the new interface for accessing the catalog content. The Catalog Browser is search-based. Enter your search criteria and select from a list of catalog values that match your search criteria.

The browser can be used in the following ways:

- **Insertion mode** - Insert a component by selecting a catalog value. The Catalog Browser is a modeless palette which means it can remain on the screen while other commands are used. It can be docked, resized, and set to auto-hide.
- **Lookup mode** - Assign a catalog value to a component. When assigning a catalog value to a component, the Catalog Browser is a modal dialog and other commands can not be used.
- **Edit mode** - Edit the catalog database. You can edit the catalog database when in either the insertion mode or the lookup mode. Edit basic catalog information, and values associated with a catalog value such as pin lists, terminal properties, and conductor lists.

Note: Make sure to use the Migration Utility to migrate any custom catalog database files to support symbol insertion from the Catalog Browser.

Catalog Browser Add-in for Autodesk Inventor

The AutoCAD Electrical Catalog Browser is also available within Inventor. The Electrical Catalog Browser accesses the same catalog database as AutoCAD Electrical. Associate 3D parts to catalog values and insert those parts by selecting the catalog value from the Catalog Browser. The catalog database editing capabilities are also available using the Catalog Browser in Inventor.



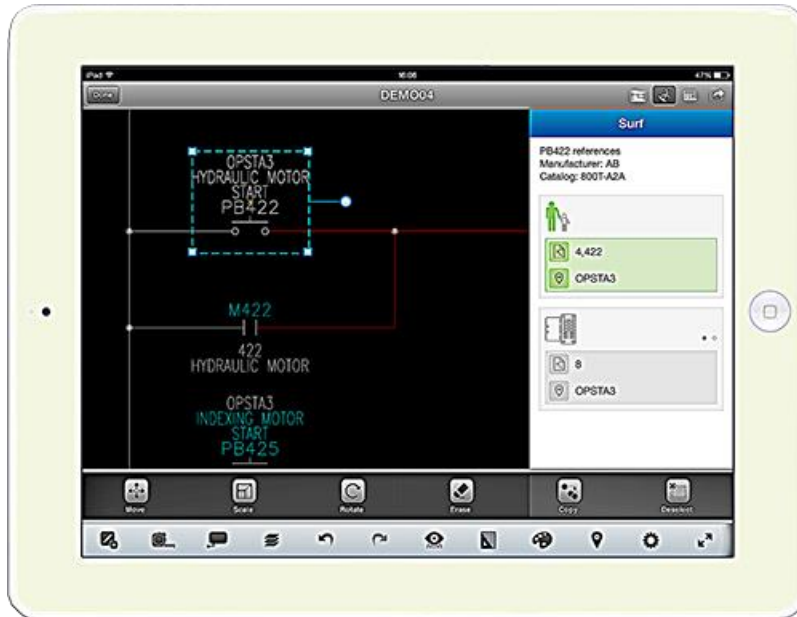
- Select an electrical part from the imported harness data, and insert the part that is associated to its catalog value from the Electrical Catalog Browser.
- Insert a part from the Electrical Catalog Browser and assign the imported cable and harness data to the part.



AUTODESK UNIVERSITY 2015

AutoCAD Electrical Mobile App

The new mobile companion app for AutoCAD Electrical allows you to connect to your AutoCAD Electrical projects while on the go.



- Integrate with Autodesk 360 or some other cloud storage for immediate access to your projects and drawings.
- Integrated with AutoCAD 360 mobile app.
- Use Design Feed for markup and collaboration.
- Navigate the project surfing on components, terminals, and signal arrows.

If the project and drawings are in the Autodesk 360 folder, they are seamlessly moved to the Autodesk 360 website. Upload projects not located in the Autodesk 360 folder to Autodesk 360, or some other cloud storage, to make them available to the mobile app.

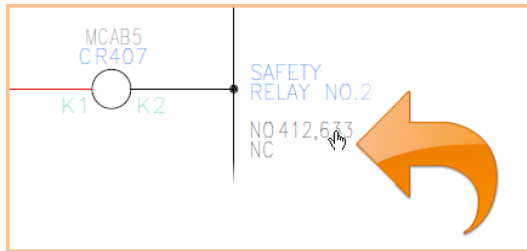
PDF Publish with Hyperlinks

Publish a set of drawings to a multipage PDF file with hyperlinks. Open the PDF and use the hyperlinks to:

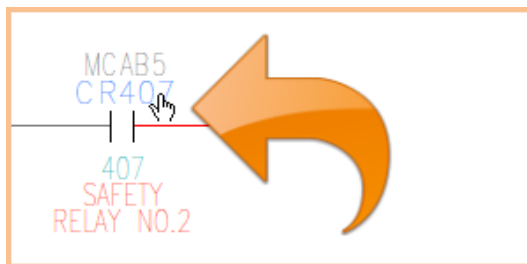
- Jump between related schematic parent and child components.
- Jump between related schematic parent and PLC components and panel footprints.
- Jump between related one-line and schematic components.
- Jump between related schematic and panel terminals.
- Follow a wire network across signal arrows.
- Follow stand-alone cross-references



Jump to Specific Child



Jump to Parent



Cut, Copy, Paste

New commands allow you to cut, copy, and paste electrical objects from one drawing to another. Select the objects to cut or copy to the circuit clipboard. Select a basepoint for the objects. Paste the objects on the same drawing or another drawing, using the basepoint to accurately place the objects.

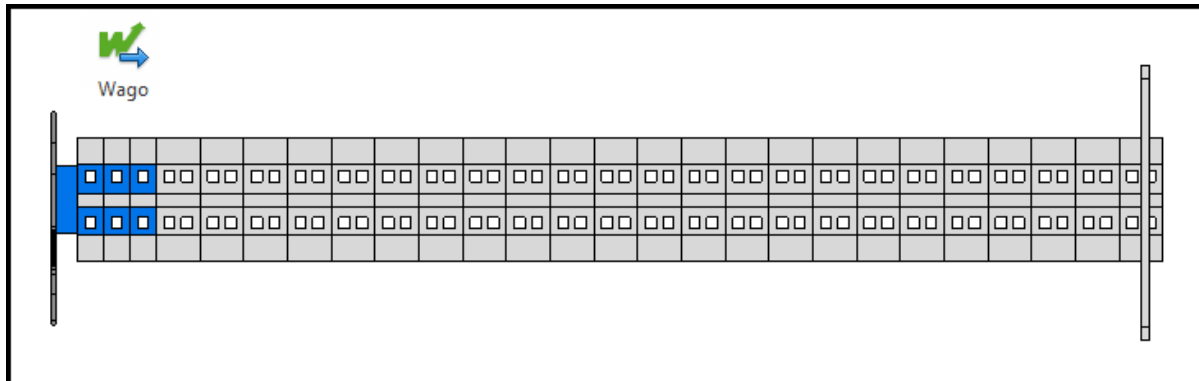
Note: Objects placed on the circuit clipboard can only be pasted on an AutoCAD Electrical drawing using the AutoCAD Electrical Paste command.

Export terminal strips to Wago® and Phoenix Contact applications

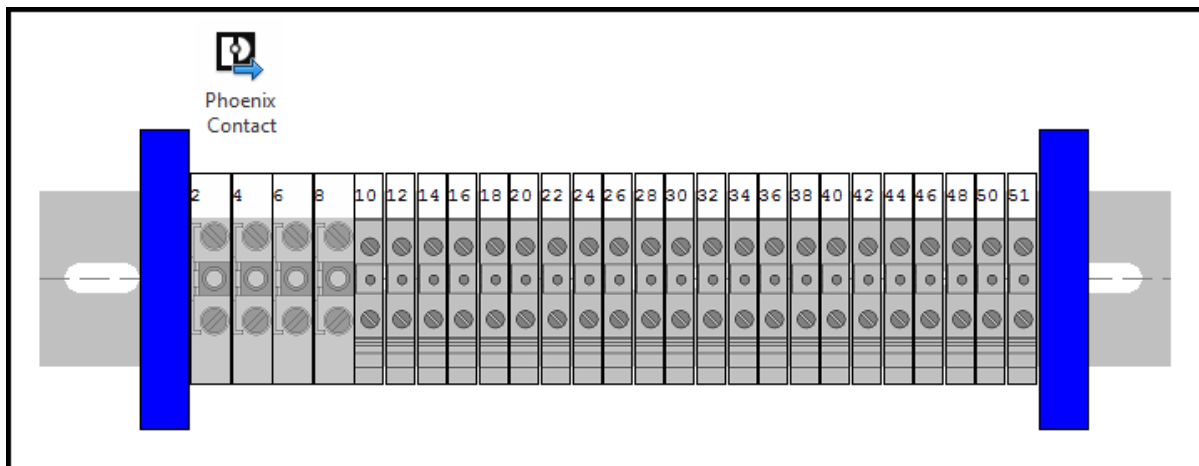
Export the terminal data such as catalog number, terminal number, installation code, location code, tag, quantities, spare, and accessories on a terminal strip to the Wago and Phoenix Contact applications. Assign catalog numbers from the Wago or Phoenix Contact content in a project. Use the Wago Export or Phoenix Contact Export command to export the desired terminal strips to an XML file for import to the vendor tools. The interoperability between AutoCAD Electrical and the vendor tools provides accurate data transfer to reduce errors in a design.



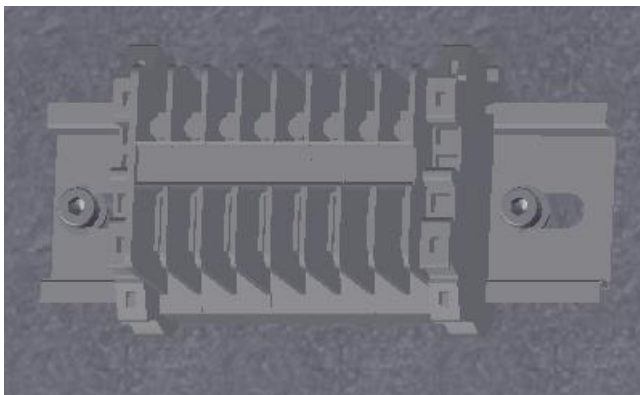
Wago® Proserve Smart Designer



Phoenix Contact Clip Project



From the vendor tools, save the terminal strip to place in an Inventor assembly.



Report Tables

The Report Format Setup now lets you select a table style for report tables. Previously, individual settings such as text height, border style, and title options controlled the look of a

report table. By using a table style for a report table, you have control over more settings, such as cell margins, title text height, header text height, and more.

Report for Drawing DEM004.DWG					
SH	TAGS	QTY	CATALOG	MFG	DESCRIPTION
4 4 4	CR406 CR407 CR408	3	700-P200A1	AB	P TYPE ELECTRICALLY HELD RELAY, 4 POLE, AC COIL, CONVERTIBLE CONTACTS TYPE P 120VAC 2 NO 115-120VAC 60Hz / 110VAC 50Hz COIL, RATING: 10A, OPEN TYPE RELAY RAIL MOUNT
4 4 4	LT415 LT423 LT426	3	800H-PR16G	AB	GREEN PILOT LIGHT - STANDARD, ROUND DILTIGHT 30.5mm 120VAC XFMR PLASTIC LENS CORROSION RESISTANT
4 4 4 4	LT409 LT411 LT412 LT413	4	800H-PR16R	AB	RED PILOT LIGHT - STANDARD, ROUND DILTIGHT 30.5mm 120VAC XFMR PLASTIC LENS CORROSION RESISTANT
4 4 4 4 4	M414 M422 M425 M433 M435	5	AN16DN0AB	EATON	FULL VOLTAGE MOTOR STARTER NEMA SIZE 1 NON-REVERSING 27AMPS, 110/120VAC 3-POLE, 110/120 AC, 50/60HZ COIL OPEN TYPE WITHOUT ENCLOSURE
4	PB414	1	800EP-E2	AB	PUSH BUTTON - MOMENTARY, IP66, NEMA 4/4X/13 22.5mm EXTENDED, IEC STYLE BLACK 2 NO PLASTIC OPERATOR w 2 ACROSS MTG
			800E-A2L	AB	800E MOUNTING LATCH - 2 ACROSS 22.5mm IEC STYLE 2 ACROSS MOUNTING LATCH 2 ACROSS MOUNTING LATCH
4 4	PB422 PB425	2	800T-A2A	AB	PUSH BUTTON - MOMENTARY, NEMA 4/13 30.5mm FLUSH BLACK 1 NO 1 NC
4 4 4	PB414A PB422A PB425A	3	800T-D1B	AB	PUSH BUTTON - MUSHROOM, NEMA 4/13 30.5mm GREEN 2 NO 2 NC
4 4	SS406 SS433	2	800MR-HH2B B	AB	SELECTOR SW - 2 POS MAINT, NEMA 13 22.5mm BLACK KNOB 2 NO 2 NC QUICK CONNECT TERMS

Library Path in Project Files

Standard paths to library symbols are now saved in the project file in such a way that they update automatically for a new release. Previously if the library path was "C:\Users\Public\Documents\Autodesk\Acade 2014\Libs\jic125" in the project file, you would need to update the project file to "C:\Users\Public\Documents\Autodesk\Acade 2015\Libs\jic125" for the new release.

The symbol library paths use a variable to indicate the default library path, for example %SL_DIR%\jic125/, %SL_DIR%\jic125/1-/.

User Post in Reports

You can now define a User Post option in your report format .SET file. Use the Report Format Setup command to add the options for a particular report. If a User Post option is defined in a report .SET file, it is run automatically before the report is displayed in the



Report Generator dialog box. You can also run Automatic Reports using report .SET files containing User Post options.

User Post .lsp and .dcl files are provided for each report type. Some of these contain samples of User Post options that you can use as examples to create your own User Post options.

Miscellaneous Enhancements

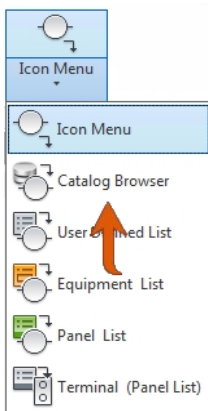
- Additional Allen-Bradley catalog content and library symbols.
- Hp/kW converter for the catalog database is available on Autodesk Exchange Apps.
- Tags in Use dialog box now includes Installation and Location columns to help identify components.
- Next and Previous Sheet options added to the Title Block Setup and Title Block Update.

Help Updates

- The Learn page of the New Tab replaces the Welcome Screen. All Essential Skills videos can be viewed from the Learn page.
- The table of contents is now focused on actions and tasks.
- Located at the end of each topic are related concept, task, and reference topics.
- In a Help topic, click the icon button or the find link next to it to find a command on the ribbon,

for example  [Find](#).

An arrow appears, pointing to the command on the ribbon.



Note: This link to locate a command only works if you are viewing the Help from within the program.



What's New in AutoCAD Electrical 2014

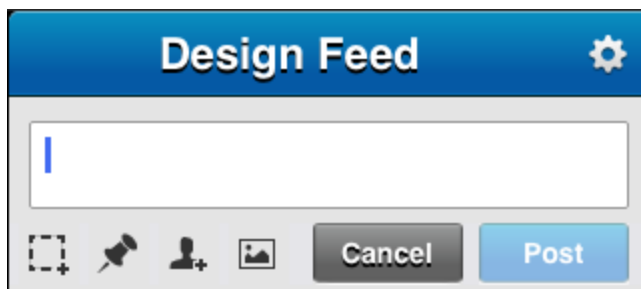
Autodesk 360 and AutoCAD Electrical

Store your projects or individual files in Autodesk 360 and access them from anywhere. Autodesk 360 offers secure access to design files so you can store, edit work, and share work confidently.

- Control access to your files
- Share files as view only, view and download, or full access
- View and add comments
- Edit drawings online in AutoCAD WS

Design Feed and AutoCAD Electrical

Design Feed provides a way of entering collaborative posts about a drawing in both real-time and asynchronous modes. Posts appear with the related drawings on the desktop, on the web, and across mobile devices.



When you create a post, you use the Design Feed palette to enter a text message and/or attach an image that conveys what you want to express about the drawing to which the post is attached. You can link the original post to a point, area, or object in the drawing. You can also tag colleagues and clients to notify them of your post. Notifications are sent to them through email and within AutoCAD Electrical.

Once you have committed a post, it is uploaded to Autodesk 360 with the drawing, along with any images you've attached. From there, colleagues and clients can access it and post their replies. Replies are listed as "children" below the original "parent" post. If there are no replies, the original post can stand by itself.

When the questions in a post and its replies are no longer active, you can resolve the thread to hide it from the Design Feed.



Metric Panel Libraries

Panel footprint symbols are now available in metric. The inch and metric panel libraries are both installed during installation. To use the metric version, set the Project Properties ► Project Settings ► Panel Footprint Libraries path to:

- **Windows XP:** C:\Documents and Settings\All Users\Documents\Autodesk\Acade {version}\Libs\panel_mm
- **Windows 7, 8:** C:\Users\Public\Documents\Autodesk\Acade {version}\Libs\panel_mm

Language Support

The language database, wd_lang1.mdb, now includes the following languages:

- Danish
- Swedish
- Norwegian
- Finnish

The language database is used by the Language Conversion tool which converts description and switch position component text.

Autodesk Inventor Integration

The Export to Inventor tool is updated as follows:

- Include all components
- Parent and child components treated as one device
- Option to export terminals as terminal strips
- No duplication of terminal numbers
- Level number appended to the terminal number for multi-level terminals
- Terminal numbers in numeric order

Startup Experience

The new Welcome Screen provides links to:

- Create or open a drawing
- Open recently used drawings



- View the Essentials Videos
- Autodesk Exchange Apps store
- Autodesk 360

Select Help drop-down ➤ Learning Tools ➤ Welcome Screen to relaunch the Welcome Screen after start up.

Help Updates

Five new videos were added to the Essentials Videos series. The Essentials Videos were moved from the Help to the Welcome Screen.



AutoCAD Electrical What's New 2005-2013

What's New AcadE 2010-2013 Chart

Feature	2010	2011	2012	2013
Autodesk Exchange Apps				X
Autodesk Inventor Professional Integration		X		
Catalog Lookup User-interface		X		
Catalog Database Creation		X		
Circuit Builder	X			
Component Update from Catalog			X	
Copy Circuit			X	
Electrical Audit	X		X	
Export/Import Drawing Settings			X	
Help Updates	X	X		X
Insert Balloon			X	
Insert Components from a Menu	X			
Item Numbering		X		X
Location Box		X		
One-Line Support	X		X	
Power Feed Support	X			
Project Manager			X	X
Reports			X	
Ribbon Interface	X			
Symbol Libraries	X		X	
Terminal Strip Editor			X	
User-Defined Attributes			X	
User Interface				X
Wire Collision Avoidance		X		
Wire Number Placement	X			
Wire Type Creation/Import		X		



What's New in AutoCAD Electrical 2013

Marking Menu

The marking menu replaces the right-click context menu for AutoCAD Electrical objects. You can perform a selection from the marking menu in either menu mode or marking mode.

- **Menu mode**

Right-click in the graphics window. Menu items surround the cursor. To dismiss the marking menu, click once again in the center of the menu display. Do not press the Esc key, this can cancel a command in progress.

- **Mark mode** (also called gesture behavior)

Draw a mark. To enter this mode, press and hold the right mouse button, and immediately move the cursor in the direction of the desired menu item. A trail follows the cursor. Release the mouse button to select and execute the command that corresponds to the direction of movement of the cursor.

The marking menus are built dynamically based on the context menus defined in the .cui file. Customize the marking menu by modifying the context menus using the Customize User Interface (cui) editor.

Project organization

Sub-divide a project into folders, grouping related drawings. Use the context menus to add or remove folders within the project.

- Unlimited folder levels
- Select drawings to process based on folders
- Easily revert to a flat list

Note: *The project folder structure does not change the actual drawing location.*

Drag and drop within Project Manager

Drag & drop to organize drawings and subfolders within the project.

- Reorder drawings
- Reorder subfolders



- Move drawings in or out of a subfolder
- Move drawings from one subfolder to another

Item Numbering

Item numbers between schematic and panel representations of the same component are kept in sync. Assign or edit an item number and the item number on each representation of that component is updated to match. The Item Resequence command updates panel and schematic components.

Autodesk Exchange Apps

Extend AutoCAD Electrical with Autodesk Exchange Apps. Discover and download apps that are powerful, fun and help you get your job done faster!



What's New in AutoCAD Electrical 2012

Library Standards

Two sets of library symbols are added that comply with the standards:

- IEEE 315/315A
- IEC-60617

The JIC standard is no longer updated, and is incorporated into the NFPA 79 standards. The NFPA standard states that the library symbols are to be in accordance with the IEEE 315/315A standard. The JIC and older IEC symbol libraries remain for legacy support.

Export/Import Drawing Settings

Export Drawing Settings properties modify the values in the output file and import the values back to the drawings and project file. These properties are exported:

- Section
- Sub-section
- Sheet (%S)
- Drawing (%D)
- IEC Project (%P)



- IEC Installation (%I)
- IEC Location (%L)
- Drawing Descriptions 1-3

Update components from catalog database

When you assign a catalog value to a component, the component updates with related values from the catalog database. If these related values change later in the database, you can update the components with the new values. This feature updates the following values:

- Schematic parent pin list
- Weblink
- Textvalue
- Terminal properties

A list displays so you can surf to related components that need updates based on the changed values, such as child pins.

Copy Circuit enhancements

In the Copy Circuit Options dialog box, options are added for updating terminal numbers. Choose from the following options for terminals within the copied circuit:

- Keep terminal numbers
- Blank terminal number values
- Increase terminal numbers

Use AutoCAD Content Explorer in AutoCAD Electrical

Content Explorer combines content from a variety of local folders for quick location and real-time access from within the AutoCAD design environment. You can index content for quick access, catalog the objects in each file, and perform searches on local folders. The following AutoCAD Electrical commands are available from Content Explorer:

- Insert schematic component
- Insert circuit
- Insert panel footprint, nameplate, or terminal

Add your AutoCAD electrical library folders to Content Explorer. Use the search function to find a library symbol or circuit, right-click and select the AutoCAD Electrical command from the menu.



By entering a search string into the search field, Content Explorer goes through every file located in the selected folder and returns the files with a file name, author, keyword, comment, name, subject, or title that meets the search criteria.

You can also search for text contained in:

- MText
- Tables
- Fields
- Multileaders
- Dimensions
- Attributes

For more information see the documentation on Content Explorer in the AutoCAD documentation.

Project Manager

The Open Project button is added to the Project Manager to open a project. 

Multiple balloon placement

When a component has multiple catalog values, select the direction for placement of multiple balloons. Options include up, down, left, right, and angle of last segment.

Report enhancements

- User-Defined Attributes support xdata in AutoCAD Electrical. Only xdata names with the VIA_WD_ prefix are included. Do not include the VIA_WD_ prefix in the User-Defined Attributes list.
- Bill of Material reports include sheet and reference fields.

Electrical Audit enhancement

Components and wires highlight when you select the Go To function from Electrical Audit, making it easier to identify the object with the exception.

Footprint update enhancement

When you modify a catalog on a schematic component, a footprint symbol can update on the related panel footprint. If the footprint for the new catalog does not exist, choose from the options:



- Leave the footprint symbol unchanged while updating the catalog values.
- Cancel the catalog change.

Update a schematic component from a one-line component

Copy information from a one-line component to a schematic component. Select which types of values to update:

- Descriptions
- Catalog values including multiple catalog values
- Installation/Location
- Ratings
- All values

Duplicate tag check for panel footprints

On the Panel Layout - Component Insert/Edit dialog box, when you enter a tag value, the program checks for duplication.

Terminal Strip Editor

- Create a terminal strip with the Terminal Strip Editor, even if there are no terminals present in the active project.
- When creating a Terminal Strip table, table styles from the active drawing are available for selection.



What's New in AutoCAD Electrical 2011

Catalog Lookup

The catalog lookup user-interface enhancements include:

- Control which fields are displayed and the field order in the catalog lookup dialog box.
- Filter catalog records by selecting existing values from any of the catalog fields.
- Search for a catalog record based on a value you supply, including wildcards, for any one of the catalog fields.
- Sort catalog records by clicking the column header of any of the displayed fields.
- BOM details displayed on the main dialog box (formerly called Catalog Check).
- Ability to define a default filter value for any catalog field using the expanded _LISTBOX_DEF table in the catalog database.
- Dialog box tooltips describing the function of each control on the dialog box.

Import Wire Type

Wire Types are defined on a per-drawing basis. You can import wire types from an existing drawing or template to another drawing. Use the Import button on the Create/Edit Wire Type dialog box to import wire types to the active drawing. Use the Project-wide Utilities to import wire types to a set of project drawings.

Item Numbering enhancements

Item numbering now supports fixed item numbers on panel components. If Resequence Item Numbers is run later on, fixed item numbers do not change.

Resequence item numbers based on the manufacturer values of the components. Select the manufacturers from a list of values used in the project. Only item numbers on components with selected manufacturer values change.

Define the resequence order based on the manufacturer values of the components. Use the Move Up and Move Down buttons to change the order of the manufacturers for processing by the Resequence Item Numbers command.



Project-specific catalog database

Create a catalog database containing only the catalog values used in the project drawings. This command takes the default_cat.mdb database and removes all catalog values not used in the project drawings. Use this smaller catalog database to:

- Send to a client with the finished project.
- Limit future catalog selections to components used in the project.

Location Box enhancements

When you insert a Location Box, a check box on the Location Box dialog box indicates whether to update the location and installation values for the parent components that fall within the new location box.

AutoCAD Electrical commands are now location box aware. If you insert or move schematic parent components into an existing location box, location and installation values update to match the location box. If you move schematic parent components out of a location box, choose whether to update component values to match the drawing defaults.

Note: *Circuit commands do not update components within the circuit due to placement within an existing location box.*

Suppress wire collision check

Turn the collision checking off temporarily when inserting wires. Default behavior is to route around a component when routing a wire from one specified location to another. If you want the wire to break across any components encountered as you insert the wire, turn the collision checking off.

Learning Solutions

- Getting Started exercises added to the online Help.
- Title block tutorial added detailing how to create a template, create a title block, and define the title block mapping for a title block update.
- Cable Design Interoperability - AutoCAD Electrical and Inventor tutorial added with exercises in both environments.



What's New in AutoCAD Electrical 2010


Ribbon Interface

To provide easy access to AutoCAD Electrical commands, a ribbon interface is now available. The ribbon layout is based on workflow and function. See the Help topic, Ribbon Interface, for more information on the ribbon layout.

Workspaces

AutoCAD Electrical provides three predefined workspaces.

- ACADE & 2D Drafting & Annotation - ribbons that provide the AutoCAD Electrical tools, and the AutoCAD 2D Drafting and Annotation tools.
- ACADE & 3D Modeling - ribbons that provide the AutoCAD Electrical tools, and the AutoCAD 3D Modeling tools.
- AutoCAD Electrical Classic - toolbars and pull down menus that provide the AutoCAD Electrical tools and AutoCAD tools.

To switch to another workspace, you can select the Workspace icon  on the status bar.

Circuit Builder goes green

Circuit Builder now provides engineering analysis/green calculations in the area of power conductor size versus energy losses. Designing to meet minimum code requirements can conflict with green design.

During the code requirements analysis, Circuit Builder displays parallel energy loss calculations so you can make better green design decisions. For example, suppose you must oversize the conductors for a motor to reduce conductor heating losses. This action results in a higher initial cost. But this higher cost can potentially be recovered many times over in reduced energy losses in the wiring over the lifetime of the installation.

Circuit Builder - power feed support

Circuit Builder now provides power feed circuits for insertion.

- Option to add a source arrow symbols at end of the power feed bus.
- Option to add a generic load box representation at the end of the power feed bus.



- Supports defining a user-created load symbol, for example a variable speed drive symbol, for insertion at the end of a power feed bus.
- Support for adjusting the load representation based on the rung spacing.

Circuit Builder - additional features

- Wire conductor sizing based upon electrical code requirements.
- Support for split-parallel conductor sizing is available. You can choose to substitute multiple, smaller diameter conductors to meet the equivalent ampacity requirement of a single, large diameter conductor.
- Fuse, breaker, disconnect switch, and overload calculations based upon electrical code requirements.
- Circuit Builder can be set up to predefine motor description text, installation, location, and text description for individual components in the circuit.
- Insert a new circuit and reference an existing circuit. This option can transfer the values from the existing circuit to your new circuit.
- Electrical standards database editor to view, modify, and expand the ace_electrical_standards.mdb file.

Motor control one-line circuits

AutoCAD Electrical now provides library support and software support to create motor control one-line diagrams that link back to other drawing types in a project drawing set.

- New motor control one-line symbol library accessible from the icon menu.
- Circuit Builder helps you build motor control one-line circuits dynamically. You can design one-line circuits, with component values and wire sizes, to conform to a given electrical code.
- One-line component symbols can be related to parent/child counterparts on the schematic and panel layout drawings within a project. You can surf between one-line and related components, and all related components update when one is modified.
- Tagging of schematic or panel components using existing commands can reference a pick list that includes components pulled from the one-line diagrams.
- Certain schematic reports have a new category option. You select the category, for example One-Line, and the data is filtered based on that



category. It can also be used to filter a report for Hydraulic, P&ID, or Pneumatic components.



No wire numbering option for wire layers

Wire layers now have a “no wire numbering” option. These wires behave normally for inserting, breaking, and scooting components, and show up in the Wire From/To report.

The Insert Wire Numbers command follows these rules:

- If **all** wires in the network are on layers set “No” for Wire Numbering, no new wire number is inserted.
- If **any** wire in the network is on a layer set “Yes” for Wire Numbering, the existing non-fixed wire number is updated, or a new wire number is inserted.
- If a wire network already has a non-fixed wire number, it is updated regardless of the Wire Numbering setting. Use the Delete Wire Numbers command to remove the wire number.

Electrical Audit

The Electrical Audit can now display the results for the active drawing only. Run the Electrical Audit, click the Active Drawing button and quickly see the issues for this drawing only. Open another drawing and the dialog box updates to display the results for the newly opened drawing. The active drawing must be part of the active project.

Help Updates

Reference topics that show command access now include the following features:

- Command line access
- Ribbon access
- If the location of a command is changed, ribbon and menu access to the command are updated in the Help system to reflect the new location.

Note: *This dynamic update only works when the Help is used within AutoCAD Electrical.*



What's New AcadE 2006-2009

Feature	2006	2007	2008	2009
Circuit Builder				X
Symbol Builder				X
Migration Utility				X
Item Numbering				X
Surfing				X
Merge Utility	X			X
Autodesk Inventor Professional Integration		X		
Terminal Strip Editor			X	X
Terminals				X
Engineering Design Management	X	X		
Catalog Content Updates		X	X	
MDI Aware	X			
Insert Component (Equipment List) Tool	X			
AutoCAD Data Migration	X			
Convert promis.e Drawings	X			
Project Manager	X	X		X
PLC I/O Libraries	X		X	
Wire Sequence Updates	X		X	X
Help Updates	X		X	X
Connector Generation		X		
Splice Tool		X		
Wire Number Placement		X		X
Multiple Wire Bus		X		
Bend Wire Tool		X		
New Symbol Libraries		X		
Real-time Error Checking		X		
User Defined Attributes		X		
Wire Label Report		X		
New Project Tool		X		
New Drawing Tool		X		
Wire Type Selection		X		
Autodesk Productstream Integration		X		X



What's New AcadE 2006-2009, continued

Feature	2006	2007	2008	2009
Simplified Configuration Settings		X		
Wire Number Leaders		X		
Wire Connection Improvements		X		X
Wire Collision Avoidance		X		
Cross-reference Updates		X		
Table Style Cross-reference Updates		X		X
Stand-alone Cross-reference				X
Enhanced Drawing Audit Report		X		
Improved Performance		X	X	
Insert Components from a Menu		X	X	
Icon Menu Wizard Enhancements			X	
PLC I/O Import/Export			X	
Spreadsheet to PLC I/O Utility Enhancements			X	X
Surfable Reports			X	X
Direct Wire Sequencing			X	
Inserting Spare Terminals			X	
Multi-Level Terminals			X	X
Terminal Jumpers			X	X
Terminal Properties Database Editor			X	X
Pin List Data Management			X	
Installer Improvements			X	
64-bit AutoCAD Electrical			X	
DWG Product Recognition			X	
InfoCenter			X	
User's Guide			X	



What's New in AutoCAD Electrical 2009

Circuit Builder

The Circuit Builder feature comes with many predefined motor control circuits. Insert a circuit by picking from the list of motors and selecting the location on your drawing. AutoCAD Electrical builds the selected circuit on-the-fly, matching the rung spacing, adding wiring between components, and annotating the circuit based on motor horsepower and industry standards. Inserting a custom motor control circuit can also be as easy as a few mouse clicks. Select the options to define the circuit, such as breaker type, control circuitry and motor horsepower. Select the location for the circuit and a custom circuit is built based on the selected options.

You can customize the Circuit Builder feature to build your custom circuits. The feature is driven by a spreadsheet and drawing templates. The spreadsheet defines the available options for the circuit and the defaults for each option. The drawing template defines the placement for the individual components and the wiring.

Terminal Strip Editor

Enhancements to the Terminal Strip Editor make it a more comprehensive utility.

- Controls added to insert, edit, and delete jumpers inside Terminal Strip Editor.
- Internal jumper support for multi-level terminals based on the catalog assignment, as defined in the TERMPROPS table, or block properties.
- Columns added for the display of internal jumpers, shown on the left of the terminal number as squares, and add-on jumpers, shown on the right as circles.
- Option to launch Automatic Wire Numbering once terminal updates are complete if jumper changes are made.
- New Jumper Chart option to display graphically all jumpered terminals in a table object placed on a drawing. These jumper charts are updated automatically when the graphical terminal strip is updated.
- Additional control over the jumper circles in the table object and jumper chart.
- Options to split the table object into multiple sections are provided. Controls for number of rows per section, number of sections per drawing, section placement, offset distance, and offset direction are available. Use the Drawing to Preview slider to preview each drawing.
- The Table Preview now takes angle and scale into account.
- An additional Preview is available while defining table settings. This preview reflects all table settings include drawing template. The preview includes a list of drawings if the settings result in multiple drawings.



Symbol Builder

The Symbol Builder now takes advantage of the block editor environment. All the block editor features are available along with AutoCAD Electrical dialog boxes. It is easy to adjust the graphics of the component and add the necessary attributes for each type of AutoCAD Electrical component.

Start with a template supplied with attributes for the specific component type you want to create. All necessary attributes are immediately available for insertion along with any optional attributes. Drag them from the dialog box into your symbol.

When your symbol is ready, save it using the suggested symbol name following the AutoCAD Electrical naming standards, or enter your own symbol name. Audit your symbol to see any potential issues with your symbol.

Migration Utility

The Migration utility replaces the Merge utility which dealt exclusively with the catalog database. The Migration utility includes the option to merge the catalog database and supports other file types and settings, including the environment file, icon menu files, lookup database files, table styles drawing, user circuits, recent projects, and so on. Some file types allow you to merge the default AutoCAD Electrical file and your custom file, other file types allow you to keep your custom file or overwrite it with the default AutoCAD Electrical file.

Select to migrate from a specific AutoCAD Electrical release. The existing files are found and listed for migration selection. Select which files to migrate, define the migration options, and your files from a previous release are ready for the latest AutoCAD Electrical release. Save the migration settings to use at a later time or to repeat the migration for another user or computer.

Item Numbering

You can now assign an item number on a per part basis. Each multiple catalog assignment can receive its own item number. Set the item numbering options for a project:

1. Right-click on the project name in Project Manager.
2. Select Properties.
3. Select the Components tab.
4. Click Item Numbering.
5. Click Per-Part Number Basis (excluding ASSYCODE combinations).
6. Click OK.

Note: An item number can be assigned to the main catalog entry and any multiple catalog entries. It cannot be assigned to a catalog entry based on an assembly code.



The Insert Balloon feature supports multiple balloons per component. Multiple balloons are inserted automatically if a component carries multiple item number assignments. When an item number is modified or removed, the item balloon is updated or deleted.

Surfing

The ability to surf on an item number was added. Right-click an item balloon and select surf, or type the item number in the Surf dialog box. You can also surf on an item number in a report table. Run the Surf command and click the item number text in the report table.

Cross-referencing

The graphics used to represent each contact type in the table cross-reference style are now customizable. A graphic drawing (.dwg) file is assigned to a specific contact block file name through a new cross-reference mapping table in the catalog lookup database. The assigned graphic drawing file is inserted as a block in the table cell in the TYPE column.

Spreadsheet to PLC/IO

You can now direct the Spreadsheet to PLC/IO utility to start a new drawing before generating the next module. Enter the keyword, NEW_DWG, in the CODE column of the spreadsheet at the point where you want to start a new drawing. AutoCAD Electrical creates the drawing before generating the next module in the spreadsheet.

You can now predefine other attributes on the module like the inline device fields. For example, you want a module to have a Rack value of “2”, an Installation value of “MACH1”, and a Rating2 value of “Hazardous Duty”. In the spreadsheet, in the RACK column, enter “2;INST=MACH1;RATING2=HAZARDOUS DUTY”. When this module is generated these extra attribute values are assigned.

Productstream - Title Block Update

The project description values in Productstream can now be written back to AutoCAD Electrical drawings. Assign these values to the AutoCAD Electrical project description values. Project descriptions can be used in the AutoCAD Electrical title block update and the drawing list report. If the project descriptions are out of date when these features are used, you are prompted to import the Productstream values in to the project descriptions.

3-Phase Wiring

Connecting the 3-phase wiring to the motor symbol is now automatic. When you insert a motor symbol and 3-phase wiring is already present, place the symbol on or near the wires and the wires are angled and trimmed to meet the motor symbol.



Note: For pre-2009 symbols, you must add a default attribute prompt value, "XOSTRETCH", to the XOTERMxx attributes.

Flip Component

When you use the Flip Component feature, existing dashed link lines are recalculated and reinserted if necessary.

Project Manager

You can now right-click the active project and select Close. The active project is closed, removed from the list, and the next project becomes the active project.

Toggle Wire Number

Toggling a wire number from above or below the wire to an in-line wire number now keeps its original center point, rather than centering it on the wire.

Trim Wire

You can now use the dynamic pan and zoom while using the Trim Wire command. Start the command and select the Fence option. Select the first fence point and pan the drawing to select the next fence point. All wires are trimmed that cross the fence rather than just the wires on the screen.

Edit Wire Sequence

The Edit Wire Sequence command remains active until you choose to exit the command by not selecting a wire. This feature makes it more efficient to assign a wire sequence to multiple wire networks.

Cable Conductor Table

Adding a cable with many conductors is more efficient with the Alt+A hot key. Once you type in the conductor color, press Alt+A to add the conductor to the list.

COPYTAG for Terminals

Terminals now support the COPYTAG attribute. COPYTAG is the optional TAG copy attribute. When AutoCAD Electrical updates a TAGSTRIP attribute, it also looks for and updates any COPYTAG attributes present on the symbol with a copy of the TAGSTRIP text. A special replaceable parameter, "%T", can be encoded onto the prompt value of the COPYTAG attribute definition. This replaceable parameter allows for adding a suffix and/or prefix to the TAGSTRIP



text. If you need more than one extra TAGSTRIP copy on a symbol, name the attributes COPYTAG01, COPYTAG02, and so on.

Stand-alone Cross Reference

The DESC1 attribute is now supported on stand-alone cross-reference symbols. You can add a value to this attribute in the Insert/Edit Stand-alone Signal/Destination dialog boxes. The description field is an available field in the Stand-alone cross reference report.

Export to Spreadsheet

The option to export information from parent components only was added.

Catalog Content Updates

Pneumatic catalog content was added to the catalog database of over 350,000 components from the most popular vendors of the industry. Additional catalog content for terminal blocks, PLC, and limit switches, was added.

Help Updates

The New Features Workshop provides an outline and graphical view of new features this release.

A symbol library preview guide lists the library symbols supplied with AutoCAD Electrical.

Toolbar Tooltips

Some of the commonly used AutoCAD Electrical commands now provide an additional level of information in the tooltip. Pause the mouse over the command icon. The first level of information for that command is displayed. Continue to pause the mouse over the command icon and the next level of information is displayed in the tooltip. For more information about any command, press the F1 key while the tooltip is displayed.



What's New in AutoCAD Electrical 2008

PLC I/O Import/Export

You can now communicate your electrical designs between AutoCAD Electrical and Schneider Electric's Unity Pro. Employing the widely used XML language format, you can transfer design data back and forth while maintaining structure and organization.

Use the new Unity Pro Export to Spreadsheet tool to import Unity Pro XML export files to aid in the creation of PLC-style ladder drawings and panel layout drawings (in both vertical and horizontal format) in the active project. The Unity Pro export files also contain catalog information. You can reformat it to generate an equipment list to aid in the creation of a rack layout drawing used in panel layouts or separate rack layout drawings.

Use the new Unity Pro Export tool to create a Unity Pro I/O variable file (*.xsy) in the Unity Pro XML format from your AutoCAD Electrical drawings. The Unity Pro export file is generated from the PLC drawings and their respective PLC symbols.

PLC I/O Libraries Enhancements

You can quickly create PLC I/O drawings by selecting from a library of over 3,000 intelligent PLC I/O modules from the industry's most popular manufacturers.

Spreadsheet to PLC I/O Utility Enhancements

You no longer have to create and save the starting drawing for the Spreadsheet to PLC I/O Utility tool. You can now define a starting drawing file name or start with the active drawing. Additional enhancements include:

- Default settings are now read automatically the first time you run the tool.
- You can select a settings file and make it the default.
- You can have the newly created drawings added automatically to the end of the active project.

Surfable Reports

When reports are placed into a drawing as a table, you can click on various report cells to quickly find the corresponding devices within the schematic or panel layout drawings in the active project.

When surfing on a table inserted by the Terminal Strip Editor, you can click the title cell to surf on the Tagstrip value, even if the Tagstrip is not included in the title. If you select a cell that is not surfable (such as the Tag, Catalog, or Wire Number cell) the Tagstrip value is surfed for the terminal strip.



Insert Component and Insert Footprint Enhancements

The Insert Component and Insert Footprint dialog boxes are updated to improve ease of use when selecting components to insert into your drawing. Enhanced dialog box controls include:

- *Menu tree structure*

Displays the main menu and submenus from which you can freely navigate. Clicking the menus displays the corresponding menu icons in the Symbol Preview window. The menu is created by reading the *.dat file defined in the Project Properties dialog box.

- *Symbol Preview window*

Displays the symbol icons and submenu icons corresponding to the selected menu. Clicking an icon performs one of the following functions based on the icon properties as defined in the *.dat file: insert a component or circuit, display a submenu, or execute a command.

- *Recently Used*

Displays the last components inserted during the current editing session. The most recently used icon displays at the top. This list follows the view options setting in the symbol preview window and the total number of icons displayed depends on the value specified in the Display edit box.

- *View*

Changes the view display for the Symbol Preview window and Recently Used window. The current view option is indicated with a check mark. Options include: Icon with text, Icon only, or List view.

- *Tooltips*

When you move the cursor over an icon, the icon name and block/circuit/command names display as tooltip information.

Icon Menu Wizard Enhancements

The Icon Menu Wizard allows you to easily customize the icon menus. You can now copy and paste icons from one submenu into another, drag and drop icons to place those that are commonly used at the top of the Symbol Preview window and those that are used less frequently at the bottom of the window, and create new icons to use when inserting components.



You can also easily modify the existing icon or menu properties like changing the name, image or block name. Right-click the menu or icon on the Icon Menu Wizard dialog box and select Properties. The existing data is overwritten in the *.dat file with the new changes.

Direct Wire Sequencing

You can now use the Edit Wire Sequence tool to define additional direct-to-terminal wire connection sequences in schematic networks. For example, 1 side of a schematic terminal might be connected to 3 field devices. A specific wire connection sequence can be defined to force the connection reporting, but it is limited to reporting the terminal as a common connection point for only 2 of the 3 field devices. The third has to be reported as jumpered to 1 of the other 2 devices. Now, with the support for secondary direct-to-terminal sequences, the third field device can be sequenced directly to the terminal and the Wire From/To report shows all 3 field devices tied directly to the terminal.

Visual Wiring Sequence Indicators

Once you define additional wire connection sequences, use the Show Wire Sequence tool to graphically show the new sequencing. When any changes are made to a wire sequence, the updated information is accurately reflected in the from/to wire list report.

Terminal Strip Editor Enhancements

The Terminal Strip Editor provides an easy way to manage and edit terminals used throughout a project. You can now start designs with a terminal strip layout drawing representing the terminal strip. In the modified Terminal Strip Selection dialog box, you can either select a terminal strip for editing, or create a new terminal strip definition in the project and maintain its properties in the graphical terminal strip layout drawing.

The Terminal Strip Editor dialog box now has an enhanced grid control with bolder grid lines that provide better visual definition for the terminal strip. Other enhancements to the dialog box include:

- The Terminal Pin (TPin) column is now “T.”
- The TERM column is now “Number” to indicate the terminal numbering, whether it is a wire number or user-defined number.
- The Function column is now “Installation.”
- A new column (on the far left side of the grid) indicates the level definition.
- Tooltip instructions display once you move your cursor over 1 of the tool buttons in the dialog box.
- There is better context menu support that is based on individual cells.
- The Preview tab is now “Layout Preview.”
- The Cable Preview tab is now “Cable Information.”



New tools are available on the Terminal Strip Editor dialog box to create associations, separate levels from a multiple level terminal block into separate terminal blocks, reverse the left and right wiring information for a terminal, and edit terminal block properties such as the number of levels and number of wires per connection.

The Layout Preview tab of the Terminal Strip Editor dialog box has been enhanced to allow AutoCAD's table objects to be inserted as a terminal strip. This allows for more accurate representations of what is in the Terminal Strip Editor, more flexibility with the style, and provides a means for automatic updating.

Inserting Spare Terminals

Extra terminal block definitions and accessory information is now maintained and saved on the graphical terminal strip layout. You can insert spare terminals and have them accurately update the Bill of Materials as well as various terminal reports.

Multi-Level Terminals

Multi-level terminal blocks are quickly becoming an industry standard. Using AutoCAD Electrical, you can define and manage the terminal numbers and levels as well as all connectivity information with no added complexity.

You can now associate schematic terminals to build a multi-level terminal block that is limited to the number of levels defined in the block properties. Use the new Add/Modify Associations tool to search project terminal strips for existing multi-level terminal blocks so that you can define and maintain terminal associations. Terminals must be in the same terminal strip and be in the same project to be associated together. You can also remove a terminal from any multi-level relationship and copy terminal properties from 1 terminal symbol to another.

Associating schematic terminals combines the terminals into a single terminal block property definition. The number of schematic terminals that can be combined is limited to the number of levels defined for the block property. Terminal associations can also tie together a set of schematic terminal block symbols to one panel representation of a terminal footprint.

Terminal Properties Database Editor

Terminal properties data is now managed based on manufacturer. Use the new Terminal Properties Database Editor tool to select the manufacturer table to edit or create a new one in the catalog database for the active project.



Terminal Jumper Support

Use the new Edit Jumper tool to add, edit, or remove jumpers between terminals that share the same potential in a schematic drawing. You can display temporary line graphics between the primary terminal and secondary terminals within the same drawing.

Jumpers now display on the panel drawing so you have a visual representation of jumpers that appear on tabular terminal strips. Cells of a table row are joined with a graphical jumper that looks like 2 circles connected by a solid thick line. Three columns of jumpers are supported within a single jumper column in the table.

Pin List Data Management

Pin list data is now managed based on manufacturer. Use the Pin List Database Editor tool to select the pin list table to edit or create a new table.

The _PINLIST table in the default_cat.mdb file now uses a single PINLIST column and a single PEER_PINLIST column. The continuation columns have been removed.

Installer Improvements for Manufacturer Content

You can now selectively install content based on manufacturer, reducing the size of the content databases and data redundancy. If you later decide you want to install content from another manufacturer, open the Add or Remove Programs tool in your Control Panel, select AutoCAD Electrical 2008, and click Change/Remove. Click Add/Remove Features, click Next on the first screen, and then select the manufacturers to install on the Manufacturer Contents Selection screen.

Catalog Content Updates

AutoCAD Electrical ships with a manufacturer's catalog database that contains over 350,000 components from the industry's most popular vendors. These components provide a full spectrum of input and output devices including switches, sensors, lights, and numerous panel devices, such as wireway and panel enclosures.

Improved Performance

Significant improvements in running commands that affect other drawings have dramatically improved the performance of AutoCAD Electrical 2008. Most notably, the Project Database Service (PDS) now only monitors the active project.



64-bit AutoCAD Electrical

AutoCAD Electrical now ships in 64-bit and 32-bit versions. The 64-bit version supports the same functionality as the 32-bit version.

DWG product recognition

Easily identify which Autodesk product created a DWG file and open the file with the application that owns the DWG file. For example, if the DWG file is owned by AutoCAD, double-clicking the file in Windows Explorer automatically opens the file in AutoCAD. When you move the cursor over a DWG icon, the tooltip identifies which Autodesk product and version was used to create the DWG.

Parametric Twisted Pair Symbol Enhancements

The icon menus are enhanced to include three new parametric twisted pair symbols. To insert a twisted pair symbol, click Components > Insert Component. On the Insert Component icon menu, click Cable Markers.

User's Guide

A User's Guide for AutoCAD Electrical is now available in PDF format. This is accessible from the Launchpad and the homepage of the Help.

InfoCenter

A new search engine, InfoCenter, is included on the title bar of the main AutoCAD Electrical window. It searches AutoCAD and AutoCAD Electrical Help systems to give you the most relevant information for any query you enter. You can filter content and add frequently used content to the "Favorites" section.

InfoCenter replaces the Communication Center. It provides notifications of software and content updates through a balloon notification mechanism. You can also publish internal content within your team, support RSS feeds, and easily provide feedback to Autodesk.



What's New in AutoCAD Electrical 2007

Connector Generation

Automatically generate a multi-pin parametric connector on the fly with the new Insert Connector command. The parametric connector build process allows you to select number of pins, spacing, and orientation to quickly create connector definitions in active drawing files without having to build or maintain a connector library of symbols. When you click Insert an outline of the connector displays for placement on the drawing. The rounded corners are the plug side of the connector, the x' indicates the connector insertion point and the arrow indicates the plug side wire direction. You can change the connector orientation before insertion using the Tab, V key, or X key on your keyboard.

New connector editing commands add to the features versatility:

- Scoot - (existing Scoot feature) moves the parametric connector horizontally or vertically, relative to the wires that are connected to the connector. Scoot also moves the wires and pins along the connector axis.
- Reverse, Rotate, Stretch, Split Connector - allow you to reverse a connector about its horizontal or vertical axis, rotate a connector about its insertion point at increments of 90 degrees, increase or decrease the connector's overall length or width, and split the connector into two separate block definitions.
- Add, Delete, Move, Swap Connector Pins - allow you to add, remove, or move the pins found inside of the connector.

Parametric Twisted Pair Symbols

The icon menus have been enhanced to include parametric twisted pair symbols. To insert a twisted pair symbol, select Components Insert Component. On the Insert Component icon menu, click Miscellaneous Shields Twisted Pair.

Wire Collision Avoidance

Instead of drawing each line segment between components on your point to point drawings, simply select the two connection points and let AutoCAD Electrical do the rest. Using the existing the Insert Wire command, select a connection point on each component and your wire is automatically routed, without running through your existing geometry.

Splices

The new Splice tool allows you create up to two wire to wire connections per side while maintaining connectivity throughout your drawing and project.



Wire Number Placement

AutoCAD Electrical supports the automatic placement of new wire numbers above, below, or directly in-line with the wire. You can set the wire number placement for all new wires inserted.

You can use the new Toggle Wire Number In-Line tool to switch the wire number between in-line and the drawing default (above or below the wire). If the selected wire number is in-line, it toggles to above or below the wire based on the default Wire Number Placement setting in the Drawing Properties > Wire Numbers dialog box. If it starts as above or below, the selected wire number toggles to in-line.

Wire Connection Improvements

Enhancements were made to the Insert Wire command to make generating point to point drawings easier. These include:

- Temporary wire graphics change color to indicate when an electrical connection can be made.
- Wire connection points display as a green 'x' at the wire connection point attributes' insertion point.
- Wires are drawn with an angled wire connection if a wire is already connected to the selected wire connection point.

Bend Wires

Bend a wire into a right angle turn to avoid or add geometry using the new Bend Wire tool. When a wire is defined at a right angle, you can modify the wire and create a new right angle bend while maintaining the original wire connections to the components.

Reposition Wire Number Leaders

When defining wire number leaders you can type "C" at the command prompt to go into a wire leader collapse mode to collapse the wire leader back to the wire number block. You can do this immediately after inserting a leader if you determine that you don't want the leader or you can re-run the Wire Number Leader command if you want to remove the leader from existing wire numbers.

Link to Autodesk Inventor Professional - Cable & Harness

You can now communicate your electrical designs bi-directionally between AutoCAD Electrical and Autodesk Inventor Professional Cable & Harness. AutoCAD Electrical users can pass electrical intent information for cables and conductors to Autodesk Inventor Professional for the automated creation of a 3D harness design. Autodesk Inventor Professional users can now pass



wire connectivity information to AutoCAD Electrical for the automatic creation of the corresponding 2D schematics. Employing the widely used XML language format, you can transfer design data back and forth while maintaining structure and organization.

From the XML import from Autodesk Inventor Professional into AutoCAD Electrical, you can select from a list of connectors defined in the export and then place the connectors onto a 2D drawing file. Once the connectors have been inserted onto the drawing, you can place all wire connections to all components on the drawing file. AutoCAD Electrical parses through the file data to determine all wire From and To connections. Once the wiring information is determined the wires are routed making sure to miss existing geometry on the drawing. The wire insertion tool finds the best possible route with the least amount of wire loops in between the connection and the wires are connected in the appropriate position on the connector representation.

Multi-discipline Symbol Libraries

AutoCAD Electrical now includes comprehensive symbol libraries for creating pneumatic, hydraulic, and P & ID drawings.

- *Hydraulic Symbol Library:* AutoCAD Electrical's hydraulic symbol library includes filters, valves, cylinders, pressure switches, motors, pumps, meters, restrictors, quick disconnects, flow arrows and more, all adhering to the NFPA/T3.10.4R1-1990 and AS1101.1-1993 standards.
- *Pneumatic Symbol Library:* AutoCAD Electrical's pneumatic symbol library includes operators, valves, flow paths, filters, regulators, cylinders, meters, motors, quick disconnects, mufflers, manifolds, flow arrows and more.
- *P & ID Symbol Library:* AutoCAD Electrical's P&ID symbol library includes equipment, tanks, nozzles, pumps, fittings, valves, actuators, logic functions, instrumentation, flow, and flow arrows, all strictly adhering to the ANSI/ISA's S5.1 Instrumentation Standard

Real-time Error Checking

AutoCAD Electrical monitors and alerts users to potential design errors as they occur. You can locate the problem component automatically using the Surf command.

Identify and clean up problems that might affect an AutoCAD Electrical drawing using the improved Electrical Audit tool. This tool displays a report of detected problems for the active project. You can save this file for reference or surf the file to view and correct the errors.

Multiple Wire Bus

With a single command you can configure a new multiple wire bus that automatically routes from an existing multi-contact component or bus or in empty space. While you are defining the wires, temporary display graphics appear on your cursor to indicate the direction and number of wires that will be placed on the drawing file.



User Defined Attributes

Add and define your own attributes for existing AutoCAD Electrical symbols. The newly defined metadata is easily customized and can be extracted for various reports. The new User Defined Attribute List tool allows you to selectively determine which non-AutoCAD Electrical attributes are allowed in the AutoCAD Electrical report generators; otherwise only those attributes defined inside of AutoCAD Electrical for each component category are processed in the project database and subsequent reports.

Table style cross-referencing updates

Tabular cross-referencing styles now function at the same level as graphical and text styles. Create customizable tables, updated your drawing in real-time and benefit from increased flexibility with the way you display cross-referencing information.

Cross-reference updates

Cross-reference settings are now supported at the project, drawing, and component level. During normal operation of cross-referencing commands, AutoCAD Electrical looks to the component for its settings information prior to using the drawing settings. If the component has settings defined, those are used. In the event that there are both component and drawing cross-reference settings on the same drawing, the component settings are used where applied and the drawing settings are used for the rest of the components. Use the new Copy/Add Component Override tool to set display settings for a specific component that are different than the drawing or use the new Remove Component Override tool to remove component overrides so the cross-referencing commands use the settings for the drawing.

Use the new Hide/Unhide Cross-Reference tool to change the visibility of cross-references. In most cases the cross-referencing should be visible but there are times when you may not want cross-referencing displayed on parent symbols.

Wire Label Report

Use the new Wire Label report to list wire and cable labels that exist in your drawing or project. The new preformatted wire label report is ready for export and can be printed on any ASCII, Microsoft Excel, Access, CSV or XML-compatible wire label printer. After the report is generated, you can still edit the format or make changes to the data before exporting to your desired file format.

Enhanced Drawing Audit Report

The Drawing Audit utility can be used to clean up certain problems that might affect your design connectivity. The audit checks for wire gaps, bad wire numbers or colors, zero length wires, wire



number floaters, and visually verifies all wires in your display. This report can be exported as a script file for post processing or sent directly to a printer.

Catalog Content Updates

AutoCAD Electrical ships with a manufacturer's catalog database that contains over 45,000 components from the industry's most popular vendors. These components provide a full spectrum of input and output devices including switches, sensors, lights and numerous panel devices, such as wire way and panel enclosures. In order to support our worldwide user base, the catalog database now includes a greater number of Asia Pacific and European vendors.

Improved Performance

Improved memory management and script-based command reallocation have dramatically improved the performance of AutoCAD Electrical 2007. Drawings open more quickly, with enhanced editing and referencing speed.

New Project Command

Creating new projects and applying project properties is now easier using the New Project tool. In a single dialog box you can define the minimum requirements to create an AutoCAD Electrical project definition file (WDP), the folder in which the project will be maintained, and the settings and options defined within the project. The new project automatically becomes the active project.

New Drawing Command

When you are faced with multiple customers or many one-off designs, the New Drawing tool helps reduce the hassle of configuring new drawings to specific standards. In a single dialog box you can apply a template, add drawing name, border, drawing type, and descriptions, which are then stored and available for future use. The new drawing then automatically becomes part of the active project.

Simplified wire type selection

Managing wire properties from the Layer Manager is no longer necessary. During wire insertion, the current wire type displays at the command prompt. Now you can simply type in the hotkey "T" for immediate access to the Set Wire Type dialog box where you can quickly assign the wire type. You can use this hotkey with the following commands:

- Wires > Insert Wire
- Wires > Angle Wires > Insert 22.5 Degree Wire (also 45 or 67.5)
- Wires > Multiple Wire Bus



- Wires > Add Rung
- Wires > Ladders > Insert Ladder

Use the new Create/Edit Wire Type tool to create new or edit existing wire types or use the new Change/Convert Wire Type tool to convert lines to wires.

Simplified configuration settings

Configuration settings have been condensed into a centralized Properties dialog box, where you can view and edit project settings, format styles and select default drawing settings for the entire project or a single drawing.

Autodesk Productstream Integration

You can now use Autodesk Productstream to manage AutoCAD Electrical bill of materials (BOMs) by controlling the release and change of a design using the Change Order environment. Additional enhancements include:

Productstream now supports AutoCAD Electrical components, quantities, catalog numbers and balloon numbers.

Productstream Explorer supports all Productstream and AutoCAD Electrical data.

New controls in Productstream make it easier to navigate to and find AutoCAD Electrical data.

Autodesk Vault Integration

Autodesk Vault integration provides tools for running Vault operations on the entire project or individual drawing files listed within the project. It supports a single-user environment where the Vault working folder is local to the customer or a multiple-user environment where the Vault working folder is shared by many users on a shared network resource. Additional enhancements include:

You can now check out individual files as they are needed rather than having to check out the entire project at once while maintaining drawing file versioning.

The multi-user environment in AutoCAD Electrical now provides drawing status indicators and better control of project-wide commands when you are logged into Autodesk Vault.

You can now get previous versions of the drawing or project file.



What's New in AutoCAD Electrical 2006

Merge Utility

Use the new Merge Utility tool to merge your existing manufacturer catalog databases, PLC I/O libraries, footprint lookup databases, and corresponding footprint symbols with your existing content in one simple operation. This maintains custom entries and modifications that you have made to your databases while adding the new database and library information.

Improved Page Management

Use the new Project Manager Enhanced Secondary Window (ESW) to manage all drawing files in a project. You can manage entire projects or make modifications to single drawings using the Project Manager ESW.

Vault Integration

Autodesk Vault Explorer and Vault Server now better handle AutoCAD Electrical projects and support work group environments where multiple users can be working on the same project at the same.

Starting designs with a panel layout drawing

You can now create a panel layout drawing, and then create the corresponding logical control schematics. Once the panel creation phase is complete, AutoCAD Electrical extracts a list of schematic components for placement into schematic drawings. You choose the component location and a physical schematic representation of each device to be inserted into the layout and a "link" is automatically created between the devices. Any changes to the either schematic or panel representation updates the other.

MDI Aware

You can now have multiple drawings open at any one time. This allows you to cut and paste design information between two open drawings without closing one of them.

AutoCAD Data Migration

Use the new conversion tools to migrate existing electrical designs from AutoCAD or AutoCAD LT into AutoCAD Electrical for further modification. The existing tools have been greatly enhanced, making it even quicker and easier to migrate from AutoCAD drawings to intelligent AutoCAD Electrical designs.



promis.e Data Migration

Use the new promis.e data migration tools to easily migrate electrical designs from promis.e into AutoCAD Electrical.

Additional PLC I/O Libraries

The PLC I/O libraries have been expanded to include more than 2,000 new PLC I/O modules from industry's most popular manufacturers. Additional connection devices and panel footprint symbols have also been created and added to the lookup database.

Visual Sequence Indicators

A new option has been added to the configuration that allows you to graphically indicate the proper wiring sequence of a circuit directly on the schematic.

Printable Help

The AutoCAD Electrical Help system has been enhanced to allow you to print all or portions of the Help for reference. In the Help system, click the Contents tab, right-click on the section you would like to print (or click Print AutoCAD Electrical Help to print the entire Help system), and Select Print. In the Print Topics dialog box, select Print the selected heading and all subtopics and press OK.

AutoCAD Electrical Launchpad

Use the new AutoCAD Electrical Launchpad to quickly access the Getting Started Manual, AutoCAD Electrical newsgroup, white papers, and more



What's New in AutoCAD Electrical 2005

Terminal Strip Editor

The Terminal Strip Editor tool automatically creates accurate terminal strip drawings in either a graphical or table format. You can now manage and edit terminals used throughout an entire project through a simple interface.

PLC Module Builder

Use the new PLC Module Builder tool to graphically add new PLC modules as they are introduced by manufacturers. This eliminates the need to ever manually add any module information or wait for a new version of AutoCAD Electrical to ship.

Expanded Catalog Content

The standard parts catalog has been expanded to include more than 43,000 components from multiple vendors.

Merge Catalog Database Utility

The Merge Catalog Database utility merges the new catalog database with the customized database already on the system. This maintains any custom entries and modifications that you have made to your database while adding the new catalog information.

Automatic Report Generation

Use the new Automatic Report Generation tool to generate multiple reports with one command.

Vault Integration

The Vault Explorer and Vault Server are now included with AutoCAD Electrical to help manage your engineering data.

Enhanced Link to Autodesk Inventor Professional

The link to Autodesk Inventor Professional has been enhanced to support cable information instead of simple discrete wires.

Enhanced Drawing Standards Support

Drawing standards have been expanded to cover the JIC, IEC, JIS, and GB standards.

