

Contents

Chapter 1	Locating Commands	1
	Ribbon Interface	1
	Project tab	1
	Project Tools panel	1
	Other Tools panel	2
	Troubleshooting panel	6
	Schematic tab	7
	Quick Pick panel	7
	Insert Components panel	8
	Edit Components panel	12
	Insert Wires/Wire Numbers panel	20
	Edit Wires/Wire Numbers panel	24
	Other Tools panel	30
	Power Check Tools panel	32
	Panel tab	32
	Insert Component Footprints panel	32
	Terminal Footprints panel	34
	Edit Footprints panel	35
	Other Tools panel	37
	Conduit Tools panel	39
	Reports tab	40
	Schematic panel	40
	Panel panel	40
	Miscellaneous panel	41

Import/Export Data tab	41
Import panel	41
Export panel	43
Conversion Tools tab	43
Tools panel	43
Schematic panel	46
Panel panel	48
Attributes panel	49
Symbol Builder tab	52
Edit panel	52
Help panel	52
New Commands	53




Locating Commands





1

Ribbon Interface


Project tab









Project Tools panel









	Command	Description
	Project Manager AEPROJECT	Lists the drawing files associated with each open project. Use this to add new drawings, reorder drawing files, and change project settings. You cannot have two projects open in the Project Manager with the same project name.
	Copy Project AECOPYPROJECT	Copies an existing project to a new name and creates renamed copies of the drawing files.
	Delete Project AEDELETEPROJECT	Deletes a project and provides the option to also delete the drawing files in the project. This is permanent and cannot be undone.








	Command	Description
	Zip Project AEZIPPROJECT	Creates a zip file of the .wdp file for the active project and one or more drawing files it references. The zip file can optionally include a copy of the temporary database file for the project.
	Project-Wide Update/Re-tag AEPROJUPDATE	Updates component tags, wire numbers, ladder references, and select drawing settings.
	Project-Wide Utilities AEUTILITIES	Updates wire numbers, component tags, and attribute text. Allows user-defined scripts to be applied project-wide.
	Mark/Verify DWGs AEMARKVERIFY	<p>Places an invisible mark on each component before sending the drawings to a client. When the drawings are returned, a list is generated that includes any components or wire numbers that have been modified, edited, or copied.</p> <p>NOTE This command writes information to the project database file that is used to check for deleted components. Your drawings must be named and part of the active project to use this command.</p>

Other Tools panel

	Command	Description
	Surfer AESURF	Moves from reference to reference across the project drawing set. A new window opens and the original window closes when Surf is selected unless you hold the Shift key while running the command.








Command	Description
	<p>Continue Surfer AESURFCONT</p> <p>Continues a previous surf session from the point where you left off.</p>
	<p>Previous DWG AEPREV</p> <p>Loads the drawing listed above the current drawing in the project explorer, and closes the current drawing.</p>
	<p>Next DWG AENEXT</p> <p>Loads the drawing listed below the current drawing in the project explorer, and closes the current drawing.</p>
	<p>Migration Utility AEMIGRATION</p> <p>Migrate database and support files from a previous version of AutoCAD Electrical to the current release.</p>
	<p>Language Conversion AELANG</p> <p>Translates component description text from one language to another. Description text and switch position text is processed on schematic and panel components.</p>
	<p>Edit Language Database AELANGDB</p> <p>Opens the current language table for review and modification. The default table is wd_lang1.mdb.</p>
	<p>Title Block Setup AESETUPTITLEBLOCK</p> <p>You can link some AutoCAD Electrical project description data entries and some of the drawing values to the attributes in the title blocks. There are two methods, an attribute mapping file or a mapping attribute embedded on the title block.</p>
	<p>Title Block Update AEUPDATETITLEBLOCK</p> <p>Automates updating title block information for the current drawing or the entire project drawing set. Project and drawing specific settings are linked to one or more attributes contained in the title block.</p>

	Command	Description
	IEC Tag Mode Update AEUPDATEIECTAG	Updates component tagging based on a change in the IEC tagging mode.
	Update to New WD_M Block, Values, Layers AESWAPWDM	Replaces the schematic wd_m.dwg block in the current drawing with a newer copy, and converts to the newer configuration values and layers.
	Update to New WD_M Block, No changes AESWAPWDM- NOCHANGE	Replaces the schematic wd_m.dwg block in the current drawing with a newer copy, but keeps existing configuration values and layer names.
	Update to New WD_PN- LM Block, Values, Layers AESWAPPNLM	Replaces the panel wd_pnlm.dwg block in the current drawing with a newer copy, and converts to the newer configuration values and layers.
	Update to New WD_PN- LM Block, No changes AESWAPPN- LMNOCHANGE	Replaces the panel wd_pnlm.dwg block in the current drawing with a newer copy, but keeps existing configuration values and layer names.
	Update Symbol Library WD_M Block AECOPY2SYMLIB	Writes the attribute settings for the wd_m block in the current drawing to the wd_m.dwg drawing file in the symbol library.
	Settings List Utility AEDWGCFG	Reports the settings of each drawing in the project, and provides the means to edit the report and update the drawing properties with the edited values.
	Xdata List AELISTXDATA	Lists extended entity data, xdata, on a selected object.

Command	Description
	<p>Xdata Editor AEXDATA</p> <p>Allows display and edit of an object's "1000" type extended entity data (Xdata).</p>
	<p>Right Click Menu Off AEOFFRIGHTCLICK- CONTEXTMENU</p> <p>Turns off the right-click menus in AutoCAD Electrical.</p>
	<p>Right Click Menu On AEONRIGHTCLICK- CONTEXTMENU</p> <p>Turns on the right-click menus in AutoCAD Electrical.</p>
	<p>Add Catalog Table AEADDCATALOG TABLE</p> <p>Adds a new, blank table to the catalog lookup database file.</p>
	<p>Create Project-specific Catalog Database AECREATEPROJCATA- LOG</p> <p>Creates a project-specific catalog database containing only the entries used in the project.</p>
	<p>Move Objects to Layer AEMOVE2LAYER</p> <p>Moves all objects on a layer in the active drawing to a different layer.</p>
	<p>PLC Database Migration Utility AEPLCMIGRATE</p> <p>Adds the Category field to the PLC database tables. This field is used by the Spreadsheet to PLC I/O utility to determine module placement.</p>

Troubleshooting panel







The Troubleshooting panel is off by default.

	Command	Description
	Clean DWG Utility AECLEANDWG	Inserts a project drawing as an exploded block into a new, blank drawing.
	Command Trace On AEONLISPDEBUG	Turns on the display of a real-time listing of internal calls in AutoCAD Electrical.
	Command Trace Off AEOFFLISPDEBUG	Turns off the display of a real-time listing of internal calls in AutoCAD Electrical.
	MDB Command Trace On AEONMDBDEBUG	Turns on the display of error messages generated during temporary MDB file rebuild or freshen.
	MDB Command Trace Off AEOFFMDBDEBUG	Turns off the display of error messages generated during temporary MDB file rebuild and freshen.
	Command Timer On AEONTIMER	Turns on the timer for command execution elapsed time.
	Command Timer Off AEOFFTIMER	Turns off the timer for command execution elapsed time.







Schematic tab









Quick Pick panel







The Quick Pick panel is off by default.






	Command	Description
	Relays AERELAYMENU	Activates the icon menu with the relay page displayed.
	Push Buttons AEPUSHBUTTONSMENU	Activates the icon menu with the push button page displayed.
	Selector Switches AESELECTORSWITCH-MENU	Activates the icon menu with the selector switch page displayed.
	Limit Switches AELIMITSWITCHMENU	Activates the icon menu with the limit switch page displayed.
	Pilot Lights AEPILOTLIGHTSMENU	Activates the icon menu with the pilot light page displayed.
	Insert Saved Circuit AESAVEDCIRCUIT	Inserts a user circuit selected from on-screen icon menu.

Insert Components panel









Command	Description
 Insert Component AECOMPONENT	Inserts selected components from the icon menu onto the drawing.
 Insert Component (Catalog List) AECOMPONENTCAT	Inserts schematic symbols by choosing a catalog number or a component description from a user-defined pick list. The data displayed in this pick list is stored in a database in generic Access format. The file name is wd_picklist.mdb and can be edited with Microsoft® Access or from Add/Edit/Delete along the bottom of the pick list's dialog box. The AutoCAD Electrical normal search path sequence is used to locate this file.
 Insert Component (Equipment List) AECOMPONENTEQ	This spreadsheet organizes the selected user-created equipment list and presents the list in a pick list. As you pick an item from the pick list, the appropriate schematic symbol is found and inserted in the drawing at your pick point. You can open a comma-delimited file, Microsoft® Excel® spreadsheet, or Microsoft Access database file for input.
 Insert Component (Panel List) AECOMPONENTPNL	Lists panel components extracted from your panel drawing, finds the appropriate schematic symbol, and inserts the schematic components at your pick point.
 Insert Terminal (Panel List) AETERMINALPNL	Lists panel terminals extracted from your panel drawing, finds the appropriate schematic symbol, and inserts the schematic terminals at your pick point.
 Circuit Builder AECIRCBUILDER	Build a motor control circuit dynamically.









Command	Description
	<p>Recalculate Wire Size AEEDITWS</p> <p>Displays Wire Size Lookup dialog box with previous calculated data for selected motor or power feed load representation.</p>
	<p>Multiple Insert (Icon Menu) AEMULTI</p> <p>Inserts a series of similar components at fence crossing points with underlying wires.</p>
	<p>Multiple Insert (Pick Master) AEMULTIPICK</p> <p>Inserts a copy of the selected component multiple times at each wire crossing and fence line intersection point.</p>
	<p>Insert WBlocked Circuit AEWBCIRCUIT</p> <p>Inserts WBlocked circuitry (external drawing file) with automatic component tag update.</p>
	<p>Insert Saved Circuit AESAVEDCIRCUIT</p> <p>Inserts a user circuit selected from on-screen icon menu.</p>
	<p>Insert PLC (Parametric) AEPLCP</p> <p>Generates PLC I/O modules on demand, in a variety of different graphical styles via a parametric generation technique. It is driven by a database file (ace_plc.mdb) and a handful of library symbol blocks.</p>
	<p>Insert PLC (Full Units) AEPLC</p> <p>Inserts PLC I/O modules that are fixed library symbol blocks.</p>
	<p>Location Box AELOCATIONBOX</p> <p>Draws a dashed box around selected components. A description can be assigned to the box, and components within the box can have their location and installation code(s) changed.</p>







	Command	Description
	Location Symbols AELOCATIONSYMBOL	Inserts location marks on symbols that are identified with location code in text form.
	Insert Connector AECONNECTOR	Generates a connector symbol from user-defined parameters. The symbol is created on the fly, and inserted as a block insert into your active drawing file. Since these are created on an as-needed basis, it eliminates the need for you to create and maintain a library of connector symbols.
	Insert Connector from List AECONNECTORLIST	Imports a connector wire list from another application, such as Autodesk® Inventor™ Professional Cable & Harness. NOTE If the AutoCAD Electrical drawing is missing one end of the connector or if a connection was not found, wiring information is displayed next to the pin and the information is written into a log file so you know AutoCAD Electrical was unable to resolve the wire connections in the drawing. The log file name is {drawing filename.LOG} and is found in the same folder as the drawing file.
	Insert Splice AESPLICE	Inserts a splice symbol selected from the on-screen icon menu.
	Link Components with Dashed Line AELINK	Draws a smart dashed line between stacked contacts of a multicontact component. When the dashed link line inserts, certain attributes automatically flip to invisible. Use the Attribute Hide command to turn the visibility of the selected attributes back on.
	Insert Reference Arrow - To AEREFARROWTO	Draws a dashed line from a component to a "To" arrow symbol.









Command	Description
	<p>Insert Reference Arrow - From AEREFARROWFROM</p> <p>Draws a dashed line from a component to a "From" arrow symbol.</p>
	<p>Insert Stand-Alone Cross-Reference AESAXREF</p> <p>Inserts standalone cross-reference symbol (not tied to a wire). You use standalone cross-reference symbols just as you would wire source/destination arrow symbols but without the wires. Insert a source reference symbol, and then tie one or more destination reference symbols to it. These can be on the same drawing or scattered across the project drawing set.</p>
	<p>Insert Pneumatic Components AEPNEUMATIC</p> <p>Inserts Pneumatic components from an on-screen icon menu. This icon menu can be modified, expanded, or replaced with your own custom menu. You can change the default icon menu using the Project Properties dialog box. Use the Icon Menu Wizard to easily modify the menu.</p>
	<p>Insert Hydraulic Component AEHYDRAULIC</p> <p>Inserts hydraulic components from an on-screen icon menu. This icon menu can be modified, expanded, or replaced with your own custom menu. You can change the default icon menu using the Project Properties dialog box. Use the Icon Menu Wizard to easily modify the menu.</p>
	<p>Insert P&ID Component AEPID</p> <p>Inserts P&ID components from an on-screen icon menu. This icon menu can be modified, expanded, or replaced with your own custom menu. You can change the default icon menu using the Project Properties dialog box. Use the Icon Menu Wizard to easily modify the menu.</p>








Edit Components panel








	Command	Description
	Edit Component AEEDITCOMPONENT	Edits components, PLC modules, terminal, wire numbers and signal arrows.
	Add/Edit Internal Jumper AEINTERNALJUMPER	Adds, changes, or deletes internal jumpers on a selected component. When wire numbers are inserted, these internal jumpers are read and wire numbers are assigned accordingly.
	Fix/UnFix Component Tag AEFIXTAG	Toggles selected component tag between fixed and normal.
	Copy Catalog Assignment AECOPYCAT	Inserts or edits catalog part numbers onto the currently selected component or footprint.
	Edit User Table Data AEUSERTABLE	Edits user-defined Xdata on component or wire numbers and populates the User table in project database file. You can add, edit, or remove free-form user data records attached to the selected block insert.
	Delete Component AEERASECOMP	Removes the selected component from the drawing. If you erase a parent schematic component, you have the option to search for related child components, surf to them, and delete them.
	Copy Component AECOPYCOMP	Inserts a copy of an existing component into the drawing and updates the component tags.
	Copy Circuit AECOPYCIRCUIT	Copies existing circuits and pastes the copied circuit to a specified location. The components are automatically retagged based on their new line reference locations.






Command	Description
 <div> Move Circuit AEMOVECIRCUIT </div>	Moves the selected circuit to a specified location. The components are automatically re-tagged based on their new line reference locations and cross-references are updated.
 <div> Save Circuit to Icon Menu AESAVECIRCUIT </div>	Saves windowed portions of circuitry for later reuse. Up to 24 circuits can be saved at any one time in this scratch menu.
 <div> Scoot AESCOOT </div>	Scoots selected components along their connected wires or scoots entire wires, including components, along the bus. A rectangle indicates the selected items.
 <div> Align AEALIGN </div>	Aligns selected components with a master component. All connected wires are adjusted, and wire numbers recentered if necessary. You can align vertically or horizontally by flipping the command by typing V or H at the command line.
 <div> Move Component AEMOVE </div>	Automatically moves the selected component to a new position.
 <div> Reverse/Flip Component AEFLIP </div>	Reverses or flips selected component graphics and its associated attributes NOTE This tool only operates on a component with 2-wire connections (ex: limit switch contact symbol).
 <div> Stretch PLC Module AESTRETCHPLC </div>	Stretches or compresses the windowed portion of PLC modules (or any block insert) while maintaining all of the original block information, including attributes.
 <div> Split PLC Module AESPLITPLC </div>	Splits selected PLC module into two separate block definitions (i.e. parent and a child or a child and another child).








Command	Description
 Retag Components AERETAG	Retags components with contact updates. Run this when something changes on your drawing or project that affects the component tags. This can include revising the ladder line reference numbers or changing the tag format. Retag redoes each selected primary component tag, and then updates the related secondary components. You can select to update a single component, a group of components, a drawing, drawings within your project, or the entire project.
 Find/Edit/Replace Component Text AEFINDCOMPTXT	Finds and replaces component and terminal text values or find and replace substrings within those values. You can do this on the active drawing or across the project drawing set.
 Find/Replace Terminal Text AEFINDTERMTEXT	Finds and replaces terminal number text values or find and replace sub-strings within those values. You can do this on a selection from the active drawing, the entire active drawing, or across the project drawing set.
 Move/Show Attribute AEATTSHOW	Moves the selected attributes to a picked point. The attributes remain tied to the block inserts
 Edit Selected Attribute AEEDITATT	Edits an attribute's text by picking right on the attribute. A dialog box displays and you type in a new attribute value. This utility also works on invisible attributes. It finds and displays the closest attribute to your pick point on a block insert.
 Hide Attribute (Single Picks) AEHIDEATT	Hides selected attribute; to unhide pick on block graphics and un-toggle attribute name in the list. Select the graphic of a target block insert to display a listing of all attribute names and values. You can switch attributes between






Command	Description
	hidden and visible or you can edit individual attribute values.
	Hide Attributes (Window/Multiple) AEHIDEATTRIB
	Unhide Attributes (Window/Multiple) AESHOWATTRIB
	Add Attribute AEATTRIBUTE
	Rename Attribute AERENAMEATTRIB
	Squeeze Attribute/Text AEATTSQUEEZE
	Stretch Attribute/Text AEATTSTRETCH
	Change Attribute Size AEATTSIZE
	Rotate Attribute AEATTROTATE

	Command	Description
	Change Attribute Justification AEATTJUSTIFY	Changes the justification of wire number text, component description text, or attributes.
	Change Attribute Layer AEATTLAYER	Forces attribute text entities to a given layer. Select the target layer (type it in or select from the list), press OK and then select the attributes to change to the target layer.
	Toggle NO/NC AETOGGLENONC	Flips a contact from one state (open or closed) to the other. It looks at the picked contact, reads its block name, and checks the 5th character position for either 1 or 2. It then substitutes 1 or 2 for the found character.
	Swap/Update Block AESWAPBLOCK	Use to update or change blocks in place. Attribute values are retained during the swapping process. Wire connections are also maintained even if the new symbol is slightly wider or narrower than the original.
	Reverse Connector AEREVERSE	Reverses the orientation of the connector about its horizontal or vertical axis. None of the existing wire connections automatically reroute to the reverse side of the connector and you will have to resolve wiring using the wire editing tools.
	Rotate Connector AEROTATE	Rotates the connector about its insertion point in 90 degree increments. The wire connections do not reroute with each rotation of the connector. You must resolve wiring using the wire editing tools.
	Stretch Connector AESTRETCH	Increases or decreases the connector's overall shell length. You might do this to make room for new pins or to capture previously added pins that fell beyond the connector shell. You identify which end of the connector is to be altered and the measurement of displacement.



	Command	Description
	Split Connector AESPLIT	Splits the parametric connector into two separate block definitions (i.e. parent and a child or a child and another child).
	Add Connector Pins AECONNECTORPIN	Adds pins to an existing connector.
	Delete Connector Pins AEERASEPIN	Removes a pin from an existing connector and, if the connector has a defined pin list, frees this deleted pin to be re-inserted later on this connector or on a related child of this connector.
	Move Connector Pin AEMOVEPIN	Moves connector pin associated to selected connector.
	Swap Connector Pins AESWAPPINS	Exchanges one set of connector pin numbers for another on an existing connector or between connectors on the drawing. NOTE You cannot swap a combination connector with a single plug or receptacle connector. Additionally, you cannot use this tool to swap pins from one side of a connector to the other.
	Component Cross-Reference AEXREF	Collects and annotates groups of components that carry the same TAG text string value (such as "101CR"). Components do not have to be of the same family to be cross-referenced; they just need to have the same TAG1/TAG2/TAG_*/TAG attribute values.
	Hide/Unhide Cross-Reference AEHIDEXREF	Changes the visibility of cross-references. In most cases the cross-referencing should be visible but there are times when you may not want the cross-referencing displayed on parent symbols.









Command	Description
 Update Stand-Alone Cross-Reference AEUPDATESAXREF	Updates cross-reference information for two types of cross-reference symbols: wire number signal arrow symbols and standalone cross-reference symbols. It can update your source or destination signals singly, drawing-wide, or project-wide.
 Change Cross-Reference to Multiple Line Text AEXREF2TEXT	Converts a long string of relay coil or source/destination cross-reference text to a multiline text entity (MTEXT). The underlying attribute value is maintained, but flipped to visible. The MTEXT entity is created at the same XY location as the underlying attribute. The MTEXT entity updates, scoots, and behaves as if it is an attribute tied to the component block.
 Cross-Reference Check AEXREFCHECK	Displays all associated and parent components to the selected component. A complete list of components is extracted from the project drawing set. The component's tag is read, then all associated components are found and listed in the dialog box. A bill of material check can be performed to see if the item's description indicates that the quantity of contacts can be accommodated.
 Child Location/Description Update AECHILDLOCUPDATE	Updates child and panel components with installation, location, and description values carried by the associated parent schematic component.
 Copy/Add Component Override AECOPYOVERRIDE	Copies and/or adds cross-reference component overrides from another symbol. You can define components to have different cross-referencing styles. The settings specified using this tool override the drawing properties. Component overrides are copied when the component is copied; similarly they are applied to multiple inserts of the same component.







	Command	Description
	Remove Component Override AERMOVERRIDE	Removes the component overrides so the cross-referencing commands use the settings for the drawing or project.
	Cross-Reference Table AESHOWXREFTABLE	Displays a cross-reference table for all stand-alone PLC I/O points that carry the selected component tag.
	Copy Installation/Location Code Values AECOPYINSTLOC	Performs mass copies of location, installation, group, or mount codes to all of the components you select. You either type in the desired code, pick from an on-line list, or pick a similar master component.
	Associate Terminals AEASSOCTERMINAL	Associates two or more terminal symbols together. 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 properties.
	Break Apart Terminal Associations AEBREAKASSOC	Breaks one or more terminal symbols out of an existing association. Schematic terminals are removed from any multi-tier relationship and any schematic-panel relationships. Panel terminals are removed from any schematic-panel relationships.
	Show Terminal Associations AESHOWTERMASSOC	Displays terminals associated to a selected terminal.
	Edit Jumper AEJUMPER	Edits the jumper information, such as adding catalog data, or deletes the jumper.







	Command	Description
	Copy Terminal Block Properties AECOPYTERMINALPROP	Copies terminal properties from one terminal symbol to another. If the application of the terminal properties reduces the number of levels and the number of terminal symbols exceeds the total allowed, an alert displays and the properties are not copied.
	Terminal:Show Internal/External Connections AESHOWTERMCONN	Shows internal and external terminal block connections.
	Terminal:Mark Internal Connections AEMARKTERMINT	Marks internal terminal block connections. Controls which side of a terminal receives internal wire connections.
	Terminal:Mark External Connections AEMARKTEMEXT	Marks external terminal block connections. Controls which side of a terminal receives external wire connections.
	Terminal:Erase Internal/External Connections AEERASETEMCONN	Erases internal and external terminal block connections.




Insert Wires/Wire Numbers panel

	Command	Description
	Insert Wire AEWIRE	Inserts single line wire segments on a wire layer (the wire layer does not have to be the current layer).
	Insert 22.5 Degree Wire AE225WIRE	Inserts an angled (22.5 degree) line wire segment on a wire layer (the wire layer does not have to be the current layer).




	Command	Description
	Insert 45 Degree Wire AE45WIRE	Inserts an angled (45 degree) line wire segments on a wire layer (the wire layer does not have to be the current layer).
	Insert 67.5 Degree Wire AE675WIRE	Inserts an angled (67.5 degree) line wire segments on a wire layer (the wire layer does not have to be the current layer).
	Interconnect Components AECONNECTCOMP	Inserts wires between aligned connection points on a pair of selected components.
	Insert Wire Gap AEWIREGAP	Inserts a gap/loop at the point of two crossing lines. Gaps are automatically inserted when a new wire crosses another.
	Multiple Wire Bus AEMULTIBUS	Inserts vertical or horizontal bus wiring. Bus spacing defaults to the default ladder rung spacing for horizontal bus. Multiple bus wiring automatically breaks and reconnects to any underlying components that it finds in its path. If it crosses any existing wiring, wire-crossing gaps automatically insert (if the drawing is so configured).
	Insert Wire Numbers AEWIRENO	Inserts or updates wire numbers associated with wire line entities.
	3 Phase Wire Numbers AE3PHASEWIRENO	Inserts special wire numbering generally associated with 3-phase bus and motor circuits.
	PLC I/O Wire Numbers AEPLCWIRENO	Inserts wire numbers based on the I/O address that each PLC connected wire touches. Wire numbers go in as FIXED which means that







Command	Description
	they will not change if a wire number retag is run later on.
	<p>Source Signal Arrow AESOURCE</p> <p>Copies wire number from a source-arrowed wire network to any/all associated destination-arrowed wire network.</p>
	<p>Destination Signal Arrow AEDESTINATION</p> <p>Retrieves the wire number for a destination-arrowed wire network from its associated source-arrowed wire network.</p> <p>NOTE A Destination signal arrow cannot be tied to a wire network that carries a pre-assigned fixed wire number.</p>
	<p>Fan In/Out Source AEFANINSRC</p> <p>Inserts in-line source marker symbols and changes the connected wire on the fan-in side to be on a non-wire layer.</p>
	<p>Fan In/Out Destination AEFANINDEST</p> <p>Changes the connected common wires on the fan-out side to non-wire layer but leaves the individual segments on the opposite side of marker on the original wire layer.</p>
	<p>Wire Arrows for Reference Only AEREFWIREARROWS</p> <p>Inserts non intelligent, reference-only arrows.</p>
	<p>Insert Ladder AELADDER</p> <p>Inserts ladders of a set width and length onto the drawing. There is no limit to the number of ladders that can be inserted into a drawing, but ladders may not overlap each other. Multiple ladder fragments in the same vertical column need to be vertically aligned along their left-hand side.</p> <p>NOTE These limitations do not apply when X-Y Grid or X-Zone referencing is selected.</p>







Command	Description
 <div> XY Grid Setup AEXYGRID </div>	Inserts the X-Y grid labels for drawings that use X-Y Grid for the Format Referencing. You can also change other settings from here (such as origin) instead of going back into the Drawing Properties dialog box. Your drawing must be configured for X-Y Grids. Set the Format Referencing in the Drawing Properties dialog box to X-Y Grid.
 <div> X Zones Setup AEXZONE </div>	Inserts the X grid labels for drawings that use X Zones for the Format Referencing. You can also change other settings from here (such as origin) instead of going back into the Drawing Properties dialog box. Your drawing must be configured for X Zones. Set the Format Referencing in the Drawing Properties dialog box to X Zones.
 <div> Wire Number Leader AEWIRENOLEADER </div>	Repositions the wire number text with an attached leader.
 <div> Wire Color/Gauge Labels AEWIRECOLORLABEL </div>	Inserts wire color gauge labels with a leader on your drawing's wiring.
 <div> In-Line Wire Labels AEINLINEWIRE </div>	Inserts a reference-only in-line wire label.
 <div> Cable Markers AECABLEMARKER </div>	Inserts cable markers onto the drawing. Cable markers carry a cable TAG value, just like any parent/child device combination. They can also carry a conductor color value (carried as a RATING1 attribute value on the marker block symbol).







	Command	Description
	Multiple Cable Markers AEMULTICABLE	Inserts all the markers for a particular cable. In addition, you can edit existing cable marker sets, or even delete cable markers from this dialog box.
	Insert Dot Tee Markers AEDOTTEE	Inserts a dot tee connection symbol at a manually drawn wire intersection. If present, this replaces an existing angled wire connection symbol with a dot connection symbol. You cannot insert a tee connection symbol into empty space. A valid line wire ending (not crossing) at a tee intersection somewhere along the length of another line wire is needed. This means that it will not insert a tee connection symbol at a 90-degree wire turn.
	Insert Angled Tee Markers AEANGLETEE	Inserts an angled tee connection symbol at a manually drawn wire intersection. You cannot insert a tee connection symbol into empty space. If present, this replaces an existing wire connection dot with a tee connection symbol.







Edit Wires/Wire Numbers panel








	Command	Description
	Edit Wire Number AEEDITWIRENO	Allows manual edit of an existing wire number or insert of a new one if none exists.
	Fix Wire Numbers AEFIXWIRENO	Fixes all or many wire numbers on a drawing at their current values. Fixing a wire number means that the wire number tag is left unchanged if later processed or reprocessed by the automatic wire numbering utility.
	Swap Wire Numbers AESWAPWIRENO	Swaps wire numbers between two wire networks.

	Command	Description
	Find/Replace Wire Numbers AEFINDWIRENO	Finds and replaces wire number text values or substrings within those values. You can do this on the active drawing or across the project drawing set.
	Hide Wire Numbers AEHIDEWIRENO	Moves the wire number to a special hide layer so that the number is no longer visible on the screen. The new hide layer is created from the wire number layer name with a “_HIDE” suffix. For example, if the wire number text layer is called WIRENO then the hide layer name is called “WIRENO_HIDE.” The layer is created automatically when needed and you are asked if you want to freeze this layer.
	Unhide Wire Numbers AESHOWWIRENO	Moves the wire number out of the hide layer so that the number is visible on the screen.
	Trim Wire AETRIM	Removes a wire segment and dots as required. You can select a single wire or draw a fence through multiple wires to trim.
	Delete Wire Numbers AEERASEWIRENUM	Deletes selected wire numbers. NOTE If you erase a wire number and select right on an extra wire number copy, AutoCAD Electrical erases just that copy but leaves the network’s main wire number and any other copies in place.
	Add Rung AERUNG	Finds the nearest line reference location and places a ladder rung at that reference position (both bus wires must be visible on the screen for this to work. If the new rung encounters a schematic device floating in space, it tries to break the wire across the device.






	Command	Description
	Revise Ladder AEREVISELADDER	Adjusts the line reference numbering along the side of the ladders; however it doesn't change existing ladder rung spacing.
	Renumber Ladder Reference AERENUMBERLADDER	Renumbers the ladder for the selected drawings from the active project.
	Create/Edit Wire Type AEWIRETYPE	Creates and edits wire types. Use the grid control to sort and select the wire types for easy modification.
	Change/Convert Wire Type AECONVERTWIRETYPE	Changes between wire types and converts lines to wires. Use the grid control to sort and select the wire types for easy modification.
	Copy Wire Number AECOPYWIRENO	Inserts extra wire numbers anywhere on a wire network. These copies follow the network's main wire number attribute. If AutoCAD Electrical modifies it, then any wire number copies on the network also update. Extra wire numbers go on their own layer that is defined in the Define Layers dialog box. If you assign a color to this layer that is different than the normal wire number and fixed wire number layers, then it is easy to tell them apart from the network's main wire number.
	Copy Wire Number (In-Line) AECOPYWIRENOIL	Inserts extra wire numbers such that they appear in-line with the wire rather than above or below the wire. These copies follow the network's main wire number attribute; if AutoCAD Electrical modifies it then any wire number copies on the network also update. Extra wire numbers go on their own layer that is defined in the Define Layers dialog box. If you assign a color to this layer that is different than the normal wire number and fixed wire







Command	Description
	number layers, then it is easy to tell them apart from the network's main wire number.
 Adjust In-Line Wire/Label Gap AEWIRELABELGAP	Adjusts the gap between the wire and the wire number text of wire numbers that are in-line with the wire.
 Move Wire Number AEMOVEWIRENO	Moves an existing wire number from one segment of the network to another.
 Stretch Wire AESTRETCTWIRE	Lengthens a wire until it meets another wire or an AutoCAD Electrical component.
 Bend Wire AEBENDWIRE	Bends a wire in a right angle and makes 3 right angle turns to avoid or add geometry. 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.
 Show Wires AESHOWWIRE	Highlights all wires and displays wire number to wire segment relationships.
 Check/Trace Wire AETRACEWIRE	Helps troubleshoot problems with unconnected or shorted wires and invalid wire crossing gap pointers by single stepping through and highlighting each connected wire of the selected wire network.

Command	Description
 <div> Flip Wire Number AEFLIPWIRENO </div>	Flips the wire number across its associated wire.
 <div> Toggle Wire Number In-line AETOGGLEWIRENO </div>	<p>Switches the wire number between above or below and in-line. If the selected wire number is in-line, it toggles to above or below based on the default Wire Number Placement setting in the Drawing Properties dialog box. If it starts as above or below, the selected wire number toggles to in-line.</p> <p>NOTE If there isn't room for a wire number to become an in-line wire number, it remains an above or below line wire number.</p>
 <div> Toggle Angled Tee Markers AETOGGLETEE </div>	<p>Toggles an existing angled tee connection symbol (or windowed symbols) through a total of 4 possible orientations. Right-click to toggle through the various tee connection orientations, and press ESC when the appropriate one displays. This replaces any dot tee symbols with angled tee symbols, and then toggles through the 4 possible orientations for each.</p>
 <div> Flip Wire Gap AEFLIPWIREGAP </div>	<p>Flips the gap to the other wire. AutoCAD Electrical makes the gapped wire solid and flips the gap/loop to the crossing wire(s).</p>
 <div> Delete Wire Gap AEERASEWIREGAP </div>	<p>Removes a gap/loop that is no longer needed in an existing wire.</p>
 <div> Check/Repair Gap Pointers AEGAPPOINTER </div>	<p>Verifies that the invisible Xdata pointers on both sides of a wire gap/loop are valid. If not, appropriate pointers are established.</p>

Command	Description
 Edit Wire Sequence AEEDITWIRESEQUENCE	Predefines a wire network's connection sequence, either in a single drawing or across multiple drawing files.
 Show Wire Sequence AESHOWWIRESEQ	Shows the wire sequence defined using the Define Wire Sequence tool. If the wire sequence crosses multiple drawings and you try to view the sequence as an animation, a dialog box listing the off-drawing wire connection information displays so that you can indicate to go to the other drawings to continue viewing the sequence.
 Update Signal References AEUPDATESIGREF	Updates cross-reference information for two types of cross-reference symbols. Wire number signal arrow symbols and standalone cross-reference symbols.
 Fan In/Out - Single Line Layer AEFANIN	Defines a special layer or set of layers for the wires going out of a Fan In/Out source marker and the wires coming into a destination marker.
 List Signal Code AELISTSIG	Follows a signal from a specific source or destination symbol and lists the signal code references.
 Show Signal Paths AESHOWSIG	Displays signal source and destination paths on the active drawing.
 Multiple Cable Markers Update AEUPDATECABLEMARKERS	Updates cable marker assignments defined or edited in a from/to listing.




Other Tools panel

Command	Description
 Symbol Builder AESYMBUILDER	Converts existing symbols or creates new, custom components on the fly. It works nicely for quickly building power supplies, filters, drives, controllers, and other custom devices or for converting existing non-AutoCAD Electrical symbols to make them “AutoCAD Electrical smart.” Schematic symbols created or converted using the Symbol Builder are fully compatible with AutoCAD Electrical, break wires upon insertion, and appear in the various BOM, component, and wire connection reports.
 Modify Symbol Library AEUPDATESYMLIB	Performs an update of all library symbol scaling and text heights in the folder.
 Icon Menu Wizard AEMENUWIZ	Launches the Icon Menu Wizard to easily modify or expand an icon menu, or replace an existing icon menu with your own custom menu. You can change the default icon menu using the Project Properties dialog box. The default icon menu can also be redefined in “wd.env.”
 Drawing Properties AEPROPERTIES	Defines defaults for component and wire tag formats, signal references, cross references, and layers.
 Rename Schematic Layers AERENAMELAYER	Renames layers one by one, or multiple layers at once by using the Find/Replace method. In addition to renaming the layer, this also updates the AutoCAD Electrical layer assignment information carried on the drawing's WD_M block. For example, if DEMO-WIRES is currently assigned as an AutoCAD Electrical wire layer, and you rename it using this utility, the new layer name is substituted for DEMO-

Command	Description
	WIRES in the AutoCAD Electrical wire layer name list.
	Settings Compare AESHEETCOMPARE Displays differences between drawing and project settings. Allows update.
	PLC Database File Editor AEPLCDB Creates and modifies PLC I/O module definitions. All editing and creation of PLC data is stored within the PLC database file (ACE_PLC.MDB).
	Electrical Standards Database Editor AEBDEDITOR Edits the electrical standards database file, ace_electrical_standards.mdb. The electrical standards database file sets default Circuit Builder values, defines calculations, and allows Circuit Builder to perform engineering analysis in the area of power conductor size versus energy losses
	Pin List Database Editor (AEPINLISTTABLE) Edits a pin list database table in the catalog database.
	Terminal Properties Database Editor AETERMDBEDITOR Edits a Terminal Properties table in the catalog database.
	Schematic Database File Editor AESCHEMATICDB Edits the records in the schematic_lookup.mdb file to use for mapping panel footprints and terminal representations to the equivalent schematic component block names.


Power Check Tools panel





The Power Check Tools panel is off by default.





	Command	Description
	Add/Edit Power Source/Load Levels AEPOWERLOADLEVELS	Marks a component with a power source and load value. A related routine, when invoked, then scans the wire interconnections and reports if there is too much load on a given power source
	Mark Component To Pass Power AEPASSPWR	Marks a component with a PASSPWR flag. The PASSPWR flag instructs the Power Report to pass through the marked component when calculating the load on a given source. If a component carries the PASSPWR flag the Power Report program will pass through the component and continue looking for load values on the network. NOTE Certain components don't need a PASSPWR flag (such as terminals and contacts) since they are automatically passed through.
	Power Load Check Report AEPOWERLOADREPORT	Scans the wire interconnections and reports if there is too much load on a given power source.

Panel tab



Insert Component Footprints panel



	Command	Description
	Insert Footprint (Icon Menu) AEFOOTPRINT	Inserts panel footprint selected from on-screen icon menu. This icon menu can be modified, expanded, or replaced with your own custom menu. You can change the default icon menu using the Project Properties dialog box. Use

Command		Description
		the Icon Menu Wizard to easily modify the menu.
	Insert Footprint (Schematic List) AEFOOTPRINTSCH	Inserts and annotates panel footprint by referencing the project's schematic component list. This report provides error checking between the schematics and the panel layout drawings. The program looks at the selected components, both schematic and panel, to find a match in the project. For each schematic component selected, the routine tries to find a matching panel component based on tag, location, and installation information. If a match is found, then it further compares catalog information looking for any discrepancies. The program looks at each selected panel component looking for a matching schematic component in the same way.
	Insert Footprint (Manual) AEFOOTPRINTMAN	Inserts panel footprint using a generic shape or by converting an existing non intelligent AutoCAD block.
	Insert Footprint (Manufacturer Menu) AEFOOTPRINTMFG	Inserts and annotates panel footprint using manufacturer-specific icon menu. This can save a lot of time if you frequently use the same vendor and panel components. You can even apply this method to create client-specific menus making it easier to use the vendor or components that each client prefers.
	Insert Footprint (Catalog List) AEFOOTPRINTCAT	Inserts and annotates panel footprint from user-defined list of components with catalog assignments. The data displayed in this pick list is stored in a database in generic Microsoft Access format. The file name is wd_pick-list.mdb and can be edited with Access or from Add/Edit/Delete along the bottom of the pick list's dialog box. The AutoCAD Electrical normal search path sequence is used to locate this file.


	Command	Description
	Insert Footprint (Equipment List) AEFOOTPRINTEQ	Inserts and annotates panel footprint from user-defined list of equipment.
	Insert Balloon AEBALLOON	Inserts item number balloon.
	Wire Annotation of Panel Footprint AEWIREANNOTATION	Annotates panel footprint symbols with wire connection information extracted from selected schematics.
	Insert Panel Assembly AEPANELASM	Inserts WBlocked panel footprint assembly. Use the Insert Panel Assembly utility instead of the AutoCAD Insert/Explode command when you want to insert a WBlocked group of panel component footprints with balloons or nameplates. Since AutoCAD Electrical establishes invisible Xdata pointers when these are tied to a footprint, they are properly updated when copied using this utility.






Terminal Footprints panel




	Command	Description
	Terminal Strip Editor AETSE	Displays terminal strips inside of the active project database. The combination of Function, Location, and Terminal Strip values make a complete unique record for selection in the Terminal Strip Selection dialog box.
	Terminal Strip Table Generator AETSEGENERATOR	Controls the Tabular Terminal layout format automatically. This creates a new drawing file with each section break and automatically adds them to the project listing. The terminal strip's function (installation) code, location

Command	Description
	<p>code, and tag are written to the Page Description Field inside of the Project Listing (*.WDP).</p> <hr/> <p>Insert Terminal (Schematic List) AEPANELTERMINALSCH</p> <p>Inserts and annotates panel terminals by referencing the project's schematic terminal list. This report provides error checking between the schematic terminals and panel layout terminals. The program looks at the selected terminals, both schematic and panel, looking for a match in the project. For each schematic terminal selected, it tries to find a matching panel terminal based on tag, location, and installation information. If a match is found, then it compares catalog information, and description information, looking for any discrepancies. The program then looks at each selected panel terminal looking for a matching schematic terminal in the same way.</p>
	<p>Insert Terminal (Manual) AEPANELTERMINAL</p> <p>Inserts and annotates panel terminal footprint using a generic shape or by converting an existing non intelligent AutoCAD block. Some schematic components may not carry manufacturer/catalog information or have a part number assigned that is not listed in the footprint lookup file. In such a case, AutoCAD Electrical cannot determine what footprint block needs to be used so you have to select to make catalog assignments, select or create a footprint, or create a lookup entry on the fly.</p>



Edit Footprints panel






Command	Description
	<p>Edit Footprint AEEDITFOOTPRINT</p> <p>Edits panel footprint or terminal. Converts selected block if it is not AutoCAD Electrical compatible. In some cases, a footprint update may be required due to manufacturer, catalog</p>

Command	Description
	<p>or assembly value changes. When asked whether to manually force a footprint change, click No to leave the existing footprint block as is or click Yes to set up a footprint lookup database file or manually draw a simple footprint representation.</p>
 <div> Copy Footprint AECOPYFOOTPRINT </div>	<p>Copies selected panel footprint on active drawing. Use the Copy Footprint tool instead of AutoCAD Copy when a panel component footprint has a balloon or a nameplate associated to it. Since AutoCAD Electrical establishes invisible Xdata pointers when these are tied to a footprint, they are properly updated when copied using this utility.</p>
 <div> Resequence Item Numbers AERESEQUENCE </div>	<p>Assigns or resequences item number assignments on a drawing or project. This extracts all panel components and nameplates and resequences their item numbers starting at the value you provide. Resequencing is based on the main MFG/CAT/ASSYCODE value combination. Additional multi-catalog numbers on a specific component are ignored. Only the main part number combination is used to group similar components together under a common item number.</p>
 <div> Delete Footprint AEERASECOMP </div>	<p>Removes the selected footprint from the drawing. You have the option to search for related components, surf to them, and delete them.</p>
 <div> Copy Installation Code AECOPYINST </div>	<p>Copies Installation Code to one or more selected panel footprints.</p>
 <div> Copy Location Code AECOPYLOC </div>	<p>Copies Location Code to one or more selected panel footprints.</p>

	Command	Description
	Copy Mount Code AECOPYMOUNTCODE	Copies Mount Code to one or more selected panel footprints.
	Copy Group Code AECOPYGROUPCODE	Copies Group Code to one or more selected panel footprints.
	Copy Assembly AECOPYASM	Copies one or more selected panel footprints.







Other Tools panel

	Command	Description
	Symbol Builder AESYMBUILDER	Converts existing symbols or creates new, custom components on the fly. It works nicely for quickly building power supplies, filters, drives, controllers, and other custom devices or for converting existing non-AutoCAD Electrical symbols to make them "AutoCAD Electrical smart." Schematic symbols created or converted using the Symbol Builder are fully compatible with AutoCAD Electrical, break wires upon insertion, and appear in the various BOM, component, and wire connection reports.
	Icon Menu Wizard AEMENUWIZ	Launches the Icon Menu Wizard to easily modify or expand an icon menu, or replace an existing icon menu with your own custom menu. You can change the default icon menu using the Project Properties dialog box. The default icon menu can also be redefined in "wd.env."

Command	Description
 <div> Panel Configuration AEPANELCONFIG </div>	Sets panel footprint drawing defaults such as text sizes and layer assignments. Configuration settings are saved as attribute values on a non-visible block named WD_PNLM (that inserts at 0,0). If your current drawing does not have this block present when any AutoCAD Electrical panel layout command is invoked, AutoCAD Electrical pauses and asks you for permission to insert this block.
 <div> Rename Panel Layers AERENAMEPANLELLAYER </div>	Renames panel-related layers and updates panel drawing settings. The Panel Layer Rename utility makes it easy to rename layers one by one, or multiple layers at once by using the Find/Replace method. In addition to renaming the layer, this also updates the AutoCAD Electrical layer assignment information carried on the drawing's WD_M block. For example, if DEMO-WIRES is currently assigned as an AutoCAD Electrical wire layer, and you rename it using this utility, the new layer name is substituted for DEMO-WIRES in the AutoCAD Electrical wire layer name list.
 <div> Update Footprint Layers AEFPLAYERS </div>	Updates selected footprint layer assignments to match panel drawing settings layer assignments.
 <div> Make Xdata Visible AESHOWXDATA </div>	Converts any piece of non-visible extended entity data (Xdata) into a visible attribute tied directly to the footprint block.
 <div> Footprint Database File Editor AEFOOTPRINTDB </div>	Edits the catalog number and footprint block name lookup file. The footprint lookup database links a manufacturer's catalog part numbers to appropriate footprint block .dwg files. This information is in a multitable Access database file (<i>footprint_lookup.mdb</i>).






Conduit Tools panel

The Conduit Tools Panel is off by default.


	Command	Description
	Conduit Marker (Pick) AECONDUITMARKER	Formats an inter-wiring list from a selection of interconnected components. Inserts as a conduit tag.
	Conduit Marker (From/To List) AECONDUITMARKERLIST	Formats an inter-wiring list from a subset of a component from/to report. Inserts as a conduit tag.
	Edit Conduit Marker AEEDITCONDUITMARKER	Edits conduit marker tag, descriptions, and wire assignments.
	Conduit Marker Report AECONDUITMARKERRPT	Extracts conduit marker information into a report. Extractable conduit marker symbols are named "WWAY*." A conduit can be represented by a line or a polyline and by itself does not carry any intelligence. However, you can insert a conduit marker symbol and associate it to a conduit. The conduit marker symbol then carries wire information intelligence pulled from the AutoCAD Electrical drawings.
	Wire/Conduit Routing Report AEROUTINGREPORT	Reports a list of conduit tag assignments that a given wire or cable passes through.
	Extract Wire Data AEEXTRACTWIREDATA	Extracts schematic wiring information prior to conduit assignment.

Reports tab




Schematic panel

	Command	Description
	Schematic Reports AESCHEMATICREPORT	Generates schematic reports such as Bill of Material, Component lists, Wire From/To, PLC descriptions.
	Show Missing Catalog Assignment AEMISSINGCATREPORT	Displays components that do not carry a catalog number assignment. The components are marked with diamond-shaped temporary graphics.
	Electrical Audit AEAUDIT	Displays a report of detected problems or potential problems. You can save this file for reference or surf the file to view and correct the errors.
	Drawing Audit AEAUDITDWG	Displays a report of detected problems or potential problems. You can save this file for reference or surf the file to view and correct the errors.
	Signal Error/List Report AESIGNALERRORREPORT	Displays a signal list and exception report.

Panel panel


	Command	Description
	Panel Reports AEPANELREPORT	Generates panel reports such as Bill of Material, Component lists, Nameplates.






Miscellaneous panel

	Command	Description
	Automatic Report Selection AEAUTOREPORT	Defines a list of reports and their format files to run automatically.
	Report Format File Setup AEFORMATFILE	Creates and maintains report formatting files.
	User Defined Attribute List AEUDA	Creates an attribute text file (*.wda) of user-defined attributes defined on AutoCAD Electrical block files. The User Defined Attribute List is used by report tools to determine which additional attributes are listed in a report. The list file name can be the same as the active project or named Default to be used by the entire system. The Default .wda file is saved in the base project folder, while the <project_name>.wda file is saved in the same folder as the project definition file (*.wdp).




Import/Export Data tab

Import panel

	Command	Description
	Unity Pro Export to Spreadsheet AEUNITYPROSS	Imports Unity Pro hardware (.xhw) and I/O variable (.xsy) files into AutoCAD Electrical to reformat the data into a PLC import spreadsheet. After the spreadsheet file is created use the Spreadsheet to PLC I/O Utility tool to automatically create PLC style drawing files.



Command	Description
 Spreadsheet to PLC I/O Utility AESS2PLC	<p>Creates a set of PLC I/O drawings from spreadsheet data. A project's PLC I/O requirements, in spreadsheet or database format, can drive automatic generation of the I/O schematic drawings. Your information can be read directly in Excel format (".XLS"), as a table in an Access Database file (.MDB), or you can save your information out to a comma-delimited format ("CSV") and then let AutoCAD Electrical construct a set of PLC I/O wiring diagrams directly from your data. Ladders and modules insert automatically, breaking at the bottom of one ladder and continuing on the next (or on to the next drawing).</p>
 Update from Spreadsheet AEIMPORTSS	<p>Imports data from an edited spreadsheet, and retags or updates components, wire numbers, terminal text, or PLC I/O.</p>
 Update from Project Scratch Database AEIMPORTDB	<p>Updates project drawings; attribute text only, from edits to the project's scratch database file.</p>
 Insert Spreadsheet Data to Table AEINSERTSTABLE	<p>Inserts comma-delimited spreadsheet data into a drawing as a table.</p>
 RSLogix 500 Export to Spreadsheet AERSLOGIX	<p>Prepares RSLogix 500 exported data to be processed by the Spreadsheet to PLC I/O Utility.</p>








Export panel








Command	Description
 Autodesk Inventor Professional Export AEAIPEXPORT	Extracts wire list information into an XML export file to be used exclusively with Autodesk Inventor Professional Cable and Harness. NOTE You must first configure wire numbering to be <i>On per Wire Basis</i> for export and set up the appropriate variables before running the report.
 Unity Pro Export AEUNITYPRO	Creates the Unity Pro I/O variable file (.xsy) in the Unity Pro XML format. The XML file contains the PLC I/O addresses and descriptions for import into the Unity Pro software.
 Export to Spreadsheet AEEXPORT2SS	Exports the selected data category to a comma-delimited, Excel XLS, or Access MDB file format for editing.




Conversion Tools tab

Tools panel



Command	Description
 Promis-e Conversion AEP2E	Converts drawing files from promis-e to AutoCAD Electrical. It examines the current symbol attributes on the drawing and maps them to the equivalent AutoCAD Electrical attribute to make them AutoCAD Electrical-smart.
 Add Geometry AEGEOMETRY	Adds AutoCAD geometry to a template block file to be created as part of a unique block instance. It creates a new block definition with the newly added geometry. You can sub-







Command	Description
	sequently create a new block file if the block is exploded.
	Add Wire Connections AEWIRECONN Adds wire connection attributes to the existing tagged block file. Select line endpoints or geometry to add the appropriate wire connection attributes to. A new block definition is created with the newly added wire connections. You can subsequently create a new block file if the block is exploded.
	Special Explode AEEXPLODE Explodes attributes and blocks to geometry and text entities while maintaining the value previously defined in the attributes. You can take advantage of the tagging tools to modify the text entities to attributes and the linking tools to make various blocks.
	Convert Ladder AE2LADDER Converts the upper-most line reference number on a non-intelligent ladder to be AutoCAD Electrical-aware.
	Change/Convert Wire Type AECONVERTWIRETYPE Changes between wire types and converts lines to wires. Use the grid control to sort and select the wire types for easy modification.
	Check/Repair Gap Pointers AEGAPPOINTER Verifies that the invisible Xdata pointers on both sides of a wire gap/loop are valid. If not, appropriate pointers are established.
	Change Attribute Size AEATTSIZE Changes attribute text size when components or wire numbers have already been inserted onto your drawings.
	Xdata Editor AEXDATA Allows display and edit of an object's "1000" type extended entity data (Xdata).


	Command	Description
	Convert to Schematic Component AEBLK2SCH	Takes non-AutoCAD Electrical blocks or graphics representing a symbol and replaces it with an AutoCAD Electrical block and transfers the attribute or text values to this new AutoCAD Electrical block.
	Convert Block to Source Arrow AEBLK2SRC	Replace a non-AutoCAD Electrical source arrow with a smart AutoCAD Electrical source arrow and maps the information to the new AutoCAD Electrical source.
	Convert Block to Destination Arrow AEBLK2DEST	Replaces a non-AutoCAD Electrical destination arrow with a smart AutoCAD Electrical destination arrow.
	Block Replacement AEBLOCKREPLACE	Performs drawing-wide and project-wide block replacements using a user-defined spreadsheet. This automatically maps the unconverted drawing's non-AutoCAD Electrical block inserts and attributes to appropriate AutoCAD Electrical-smart component symbols drawn from a symbol library.
	Swap/Update Block AESWAPBLOCK	Use to update or change blocks in place. Attribute values are retained during the swapping process. Wire connections are also maintained even if the new symbol is slightly wider or narrower than the original.
	Convert Text to Wire Numbers AETEXT2WIRENO	Converts a text object to an AutoCAD Electrical compatible wire number.
	Convert Text to Attribute Definition AETEXT2ATT	Converts a text object into an attribute definition object. This is not an attribute associated to an already-inserted block. This is an attribute definition possibly on a library symbol that becomes an attribute when the symbol drawing is inserted as a block into another drawing.

	Command	Description
	Add Attribute AEATTRIBUTE	Adds a new attribute to an existing instance of a block insert.
	Map Attributes from Old to New AEMAPATT	Reassigns attributes from a converted block to those expected by AutoCAD Electrical. This allows you to continue what you started with Convert to Schematic Component. Use this if you did not finish mapping values from your non-AutoCAD Electrical block.
	Stretch Wire AESTRETCTWIRE	Lengthens a wire until it meets another wire or an AutoCAD Electrical component.



Schematic panel



	Command	Description
	Tag Schematic Component AETAGSCH	Makes selected text entities an attributed block file with the TAG1 attribute visible. The template block file (HDV1_CONVERT.DWG or VDV1_CONVERT.DWG depending on the drawing properties) contains attributes for a schematic component.
	Tag PLC AETAGPLC	Makes selected text entities an attributed PLC address associated to a PLC tag. The template block file (PLCIO_ADDR_CONVERT.DWG, PLCIO_CONVERT.DWG, PLCIO_V_ADDR_CONVERT.DWG, or PLCIO_V_CONVERT.DWG depending on the drawing properties) contains attributes found useful for PLC addressing. After the addressing is defined on the block, select a PLC Tag or place one into the symbol definition for use with AutoCAD Electrical.

Command	Description
 <div> Tag Child AETAGCHILD </div>	<p>Makes selected text entities an attributed block file with the TAG2 attribute visible. The template block file (HDV2_CONVERT.DWG or VDV2_CONVERT.DWG depending on the drawing properties) contains attributes used for a child component.</p>
 <div> Tag Child - N.O. AETAGNO </div>	<p>Makes selected text entities an attributed block file with the TAG2 attribute visible. The template block file (HDV21_CONVERT.DWG or VDV21_CONVERT.DWG depending on the drawing properties) contains attributes used for a child normally open contact component.</p>
 <div> Tag Child - N.C. AETAGNC </div>	<p>Makes the selected text entities an attributed block file with the TAG2 attribute visible. The template block file (HDV22_CONVERT.DWG or VDV22_CONVERT.DWG depending on the drawing properties) contains attributes used for a child normally closed contact component.</p>
 <div> Tag Child - Form C AETAGFORMC </div>	<p>Makes the selected text entities an attributed block file with the TAG2 attribute visible. The template block file (HDV23_CONVERT.DWG or VDV23_CONVERT.DWG depending on the drawing properties) contains attributes used for a child Form C contact component.</p>
 <div> Tag Schematic Terminal - Terminal Number AETAGTERMINAL </div>	<p>Makes the selected text entities an attributed block file with the TAGSTRIP and TERM01 attribute visible. The template block file (HT0T_CONVERT.DWG or VT0T_CONVERT.DWG depending on the drawing properties) contains attributes used for a terminal block component containing a terminal number.</p>
 <div> Tag Schematic Terminal - Wire Number </div>	<p>Makes the selected text entities an attributed block file with the TAGSTRIP and WIRENO attribute visible. The template block file</p>




Command	Description
AETAGWIRENO	(HT0W_CONVERT.DWG or VT0W_CONVERT.DWG depending on the drawing properties) contains attributes used for a terminal block component containing a wire number as the terminal number.
	<p>Tag Schematic Terminal - Wire Number Change AETAGWIRENOCHANGE</p> <p>Makes the selected text entities an attributed block file with the TAGSTRIP and TERM01 attribute visible. The template block file (HT1T_CONVERT.DWG or VT1T_CONVERT.DWG depending on the drawing properties) contains attributes used for a terminal block component that changes the wire number. This creates a terminal number block that has a different wire number for each wire connected to it.</p>






Panel panel






Command	Description
	<p>Tag Panel Component AETAGPANEL</p> <p>Makes selected text entities an attributed block file with the P_TAG1 attribute visible. The template block file (ACE_P_TAG1_CONVERT.DWG) contains attributes for a panel component.</p>
	<p>Tag Nameplate AETAGNAMEPLATE</p> <p>Makes selected text entities an attributed block file with the DESC1-3 attributes visible. The template block file (ACE_NP_CONVERT.DWG) contains attributes used in nameplate symbols. If the description text strings were previously defined as attributes on an AutoCAD Electrical panel component block definition, the attribute values on the panel component are hidden and the nameplate attributes DESC1-3 are added and made visible.</p>

Command	Description
 Tag Panel Terminal - Terminal Number AETAGPANELTERMINAL	Makes selected text entities an attributed block file with the TERM01 terminal number attribute visible. The template block file (ACE_TERM_CONVERT.DWG) contains attributes for terminal numbers.
 Tag Panel Terminal - Wire Number AETAGWIRENO	Makes selected text entities an attributed block file with the WIRENO wire number attribute visible. The template block file (ACE_TERM_CONVERT.DWG) contains attributes for panel terminal symbols.

Attributes panel

Command	Description
 Link Descriptions AELINKDESC	Links simple text as Description 1-3 attributes on an AutoCAD Electrical block file. You can link them as description attributes to one or more existing template block definitions. During the conversion process, the text entity is removed and replaced with the next available description attribute, up to 3.
 Link PLC Address Descriptions AELINKPLC	Links simple text to a PLC address attribute as PLC I/O address description attributes. During the conversion process, the text entity is removed and replaced with the next available PLC address description attribute, up to 5.
 Link Terminal Number AELINKTERMINAL	Links simple text to a TAGSTRIP attribute as a terminal number attribute on an AutoCAD Electrical terminal block symbol. During the conversion process, the text entity is removed and replaced with the TERM01 or WIRENO attribute.




Command	Description
 <div> Link Manufacturer AELINKMFG </div>	Links simple text as manufacturer attributes on an AutoCAD Electrical block file. The entity value is used as the Manufacturer value for one or more template block definitions. If only one template block is selected for the link, the text entity is removed and replaced with the Manufacturer attribute.
 <div> Link Catalog Number AELINKCAT </div>	Links simple text as Catalog Number attributes on an AutoCAD Electrical block file. The entity value is used as the Catalog Number value for one or more template block definitions. If only one template block is selected for the link, the text entity is removed and replaced with the Catalog Number attribute.
 <div> Link Location Code AELINKLOC </div>	Links simple text as Location attributes on an AutoCAD Electrical block file. The entity value is used as the Location value for one or more template block definitions. If only one template block is selected for the link, the text entity is removed and replaced with the Location attribute.
 <div> Link Installation Code AELINKINST </div>	Links simple text as Installation attributes on an AutoCAD Electrical block file. The entity value is used as the Installation value for one or more template block definitions. If only one template block is selected for the link, the text entity is removed and replaced with the Installation attribute.
 <div> Link Split Tag AELINKSPLITTAG </div>	Links another string of text to a tag attribute, creating a split tag. Create the device Tag using the TAG1, TAG, or P_TAG1 attributes, and then use this tool to select the existing TAG attribute on the drawing and link another string of text, creating a split tag situation. The first TAG becomes the Part1 of the split tag while the linked portion becomes the Part2 of the split tag.

	Command	Description
	Link User AELINKUSER	Links simple text (that is not an attribute definition or part of geometry) as User (01-99) attributes on an AutoCAD Electrical block file. The entity value is used as the user value for one or more template block definitions. If only one template block is selected for the link, the text entity is removed and replaced with the user attribute, up to 99. Window selection is allowed.
	Link Rating AELINKRATING	Links simple text as Rating 1-12 attributes on an AutoCAD Electrical block file. The entity value is used as the rating value for one or more template block definitions. If only one template block is selected for the link, the text entity is removed and replaced with the rating attribute, up to 12.
	Link Item Number AELINKITEM	Links simple text as an Item Number attribute on an AutoCAD Electrical Panel block file. During the conversion process, the text entity is removed and replaced with the Item Number attribute (P_ITEM).
	Show Links AESHOWLINK	Selects the tagged template block file and displays everything (such as description, location, manufacturer, and catalog number codes) that has been linked to it.
	Un Link AEUNLINK	Selects an existing linked attribute and unlinks the attribute from the symbol, changing the attribute to AutoCAD text.

Symbol Builder tab


Edit panel

The Symbol Builder tab is displayed automatically when you use Symbol Builder.

	Command	Description
	Save Symbol Definition AESAVESYM	Displays the Save Symbol Definition dialog box.
	Symbol Audit AESYMAUDIT	Displays the Symbol Audit dialog box.
	Show Hide Symbol Block Editor Palette AEPALLETESHOW	Switches on and off the visibility of the symbol builder attribute editor window.

Help panel

The Symbol Builder tab is displayed automatically when you use Symbol Builder.

	Command	Description
	Symbol Builder Help	Displays the Symbol builder Help.

New Commands

The following commands are added to AutoCAD Electrical this release:

- Create Project-Specific Catalog Lookup - Creates a catalog database of items used in the active project.

