

AUTOCAD ELECTRICAL QUICK REFERENCE GUIDE

AutoCAD® Electrical File Descriptions & Directory Locations

Reference files should be located in a shared directory or folder when you want to share changes to files with all users for common projects.

ENVIRONMENT FILE

Windows® 7/8/10 Location:

C:\Users\{username}\Documents\Acade{version}\AeData\wd.env

SYMBOL BLOCK LIBRARIES

Windows® 7/8/10 Location:

C:\Users\Public\Public Documents\Autodesk\AcadE {version}\Libs

INSERT COMPONENT ICON MENUS

Windows® 7/8/10 Location:

C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support

ALL .DAT Files (i.e. ACE_NFPA_MENU.DAT): These are text files of all of the icon menus of all of the symbols per electrical standard that you can also customize to meet your needs.

DATABASES

Windows® 7/8/10 Location:

C:\Users\{username}\Documents\Acade{version}\AeData\{country code}\Catalogs

DEFAULT_CAT.MDB: This is the Part Catalog Database.

FOOTPRINT_LOOKUP.MDB: This is the database for mapping the graphical footprint assignments to catalog part number assignments.

SCHEMATIC_LOOKUP.MDB: This is the database for mapping panel footprints and terminal representations to the equivalent schematic component block names.

WD_LANG1.MDB: This is the default language table.

WD_PICKLIST.MDB: This is the user-defined pick list that schematic or panel symbols can be inserted from.

PLCs

Windows® 7/8/10 Location:

C:\Users\{username}\My Documents\Acade {version}\AeData\{country code}\Plc

ACE_PLC.MDB: database for creating and modifying PLC modules.

Windows® 7/8/10 Location:

C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support\User

<custom name>.WDI (default settings in WDIO.LSP): This is the PLC settings file for the Spreadsheet to PLC I/O Utility. (**Note:** If you do not create a WDI file, the default settings found in the WDIO.LSP file will be used.)

Windows® 7/8/10 Location:

C:\Program Files\Autodesk\AutoCAD {version}\Acade\Support\en-US\Shared

WDIO.LSP: The program file for the Spreadsheet to PLC I/O utility.

WDIO.DCL: The companion dialog definition file for the Spreadsheet to PLC I/O utility.

CIRCUIT BUILDER

Windows® 7/8/10 Location:

C:\Users\Public\Public Documents\Autodesk\AcadE {version}\Support\{country code}

ACE_CIRCUIT_BUILDER.XLS: This is the reference file that the Circuit Builder tool uses.

DIN RAILS

Windows® 7/8/10 Location:

C:\Users\{username}\My Documents\Acade {version}\AeData\{country code}\Catalogs

WDDINRL.XLS: This is the reference file that generates Din Rails.

SHARED REFERENCE FILES

Windows® 7/8/10 Location:

C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support

DEFAULT.INST: This file provides preset values for installation codes.*

DEFAULT.LOC: This file provides preset values for location codes.*

DEFAULT.WDA: This is the reference file for user-defined attributes on blocks.*

DEFAULT.WDT: This is the attribute mapping support file for the Title Block Update tool.*

DEFAULT_WDTITLE.WDL: Customizes the generic LINEx labels in the various title block and project information dialog boxes.*

WD_DESC.WDD: This contains preset component descriptions, accessible by clicking Defaults on the Insert/Edit Component dialog box. Family-specific versions of the file can also be created, such as *pb.wdd*.

WD_RATINGS.WDR: Generic Ratings file, accessible by clicking the Defaults button in the Show all Ratings subdialog box of the Insert/Edit Component db.

WDSRCDST.WDD: Generic list of descriptions for Source & Destination signals, accessible by clicking the Defaults button in the Insert Source/Destination dbs.

Windows® 7/8/10 Location:

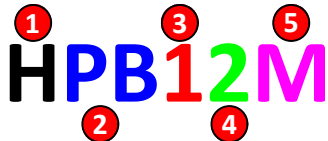
C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support\User

DEFAULT.WDU: This file saves the settings in the Title Block Update dialog box when the "Save" button is selected in said dialog box.

DEFAULT.WDW: Maps color and gauge wire descriptions based on wire layers.

NOTE: You can always override the shared reference files with by locating a copy of the reference file in the same directory as the project (WDP) file. You can also make a project-specific reference file in the same directory as the project (WDP) file. Just rename the copied reference file to the same name as the project, for example *projectname.wdt*

**These files do not exist in these locations by default. These locations are where you would create them as txt files if you want them located in shared directories.*

AUTOCAD ELECTRICAL FAMILY CODES	SCHEMATIC ATTRIBUTES FOR PARENT AND CHILD COMPONENTS		
AM: Ammeters AN: Buzzers, horns, bells CB: Circuit breakers CO: Connectors/pins CR: Control relays DI: Din Rail DN: Device networks DO: Diodes DR: Drives DS: Disconnect switches EN: Enclosures/hardware FM: Frequency meters FS: Flow sensors FT: Foot switches FU: Fuses LR: Latching relays LS: Limit switches LT: Lights, pilot lights MISC: Miscellaneous MO: Motors MS: Motor starters/contactors NP: Nameplates OL: Overloads PB: Push buttons PE: Photo switches PLCIO: Programmable logic controllers PM: Power meters PNEU-ACT: Actuators PNEU-ALU: Lubricators PNEU-CYL: Cylinders PNEU-FLC: Flow Control PNEU-FLT: Filters PNEU-MET: Pressure Gauges PNEU-MFL: Silencers PNEU-MNF: Manifolds PNEU-MOT: Motors PNEU-NOZ: Nozzles PNEU-OPR: Push buttons PNEU-PMP: Pumps PNEU-TNK: Reservoirs PNEU-VAC: Suction PNEU-VLV: Valves PS: Pressure switches PW: Power supplies PX: Proximity switches RE: Resistors SS: Selector switches SU: Surge suppressors SW: Toggle switches TD: Timer relays TRMS: Terminal blocks TS: Temperature switches VM: Volt meters WO: Cables, multi-conductor cables WW: Wire ways XF: Transformers	TAG1: (Parent only) required component tag name (255 max) TAG1_PART1, TAG1_PART2, and TAG1_PARTX: Alternate to using a single TAG1 attribute in order to split the TAG into two pieces TAG2: (Child only) Copy of the parent component's tag TAG2_PART1, TAG2_PART2: same as above COPYTAG: a copy of the TAG value MFG: manufacturer name or code (24 characters max) CAT: catalog part number assignment (60 characters max) ASSYCODE: optional subassembly code (60 characters max) ITEM: the item or detail number for a component FAMILY: component's family type, used as a check at the time child components are linked to a parent DESC1, DESC2, DESC3: component's description (60 max) INST: component installation code (255 characters max) LOC: component location code (255 characters max) XREFNO, XREFNC: (Parent only) normally open and normally closed cross-reference annotation XREF: used to include non-NO/NC references COMMON: defines which TERMxx attribute receives the first PINLIST value		
	POSn: mark switch position text where "n" is the position number digit (POS1 - POS12; 36 characters max) CONTACT: only on contact symbols for NO or NC values STATE: Contact state text to denote relationship between switch positions and open/closed contact state RATINGn: Rating value where "n" is the rating number digit (RATING1 - RATING12; 60 characters max) X?LINK: allows you to tie in dashed link lines automatically between related components ("?"= 1,2,4, or 8) PINLIST: (Parent only) stores the contact pin list for the child contacts of the parent (no limit on characters) WDTAGALT: (Parent only) for a peer-to-peer relationship WDTYPE: Attribute used to define the component category WD_WEBLINK: (Parent Only) for Internet URL's, .pdf, .xls, or .doc links that can be surfed on X?TERMn: Wire connection attributes where an external wire connects to the origin point of the attribute X?TERMDESCn: Wire connection description attributes that match up with X?TERMn (128 characters max) TERMn: Terminal pin number (ten characters max) WD_JUMPERS: to internally jumper terminals together		
	SCHEMATIC SYMBOL NAMING CONVENTION QUICK REFERENCE		
	 1 The first character is either H or V for horizontal or vertical wire insertion, respectively. 2 The next two characters are reserved for family type (for example, PB for push buttons, CR for control relays, LS for limit switches). 3 The fourth character is generally a 2 for child contacts or a 1 for everything else (parent or stand-alone components). 4 If the symbol is a contact, then the fifth character is a 1 for normally open or 2 for normally closed. 5 Any additional characters are not specified in the naming convention. They are used to keep names unique. NOTE: Do not use "HH", "HV", "VH", or "VV" as the first two characters of a symbol block name. TERMINALS: The first two characters are "HT." The third character is "0" if the wire number does not change through the terminal, "1" if the terminal symbol should trigger a wire number change. The fourth character is an underscore (_) if the terminal carries no attributes for AutoCAD Electrical to process (such as a dumb terminal symbol). Otherwise, the fourth through eighth character positions of the symbol file name are user-defined.		
	AUTOCAD ELECTRICAL SUPPORT DIRECTORIES QUICK REFERENCE		
	Windows 7/8/10 Location: C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support\ C:\Program Files\Autodesk\ AutoCAD {version}\Acade\Support\{country code} C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support\ C:\Users\Public\Public Documents\Autodesk\AcadE {version}\Support\{country code}		
	INCLUDED SYMBOL LIBRARIES	REPLACEABLE PARAMETERS	
	NFPA (U.S./Inches) IEEE (U.S., International/Millimeters) JIC (U.S./Inches - Legacy library only) IEC-60617 (Europe, International/Millimeters) IEC Legacy (Millimeters - Legacy library only) JIS (Japan/Millimeters) GB (China/Millimeters) AS (Australia/Millimeters) PNEU (Pneumatic/Inches & Millimeters) HYD (Hydraulic/Inches & Millimeters) PID (Piping & Instrumentation/Inches & Millimeters)	%F: Component family code string (for example, "PB," "SS," "CR," "FLT," "MTR") %S: Sheet number of the drawing (for example, "01") %D: Drawing number %G: Wire layer name %N: Sequential or Reference-based number applied to the component %X: Suffix character position for reference-based tagging (not present = end of tag) %P: IEC-style project code (default for drawing) %I: IEC-style installation code (default for drawing) %L: IEC-style location code (default for drawing) %A: Project drawing list's SEC value for active drawing %B: Project drawing list's SUB-SEC value for active drawing	



Tiffany Bachmeier, ACI
Manager, Consulting Delivery - MFG
email: tiffany.bachmeier@autodesk.com website: www.autodesk.com



Tiffany Bachmeier, ACI

Manager, Consulting Delivery - MFG

email: tiffany.bachmeier@autodesk.com website: www.autodesk.com