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}\My 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.

Windows® 7/8/10 Location:

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

WD_FAM.DAT: Overrides the family tag code of the library symbols by mapping the codes to new values.

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.

PLCs

Windows® 7/8/10 Location:

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

ACE_PLC.MDB: This is the 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.

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*

AUTOCAD ELECTRICAL FAMILY CODES

AM: Ammeters
AN: Buzzers, horns, bells
CB: Circuit breakers
C0: 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

SCHEMATIC ATTRIBUTES FOR PARENT AND CHILD COMPONENTS

TAG1: (Parent only) required component tag name (64 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 (24 characters max)
LOC: component location code (16 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

H P B 1 2 M
(① H, ② PB, ③ 1, ④ 2, ⑤ M)

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

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)

REPLACEABLE PARAMETERS

%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

AUTODESK.

Tiffany Bachmeier, ACI
AutoCAD Electrical Implementation Consultant
email: tiffany.bachmeier@autodesk.com　website: www.autodesk.com