

MFG12185

Guardians of the Electrical Galaxy

Robert Stein
Autodesk

Tiffany Bachmeier
Autodesk

Learning Objectives

- Understand the AutoCAD Electrical Environment
- Understand AutoCAD Electrical Methodologies
- Discover AutoCAD Electrical Best Practices
- Learn How to Deploy AutoCAD Electrical

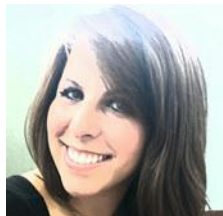
Description

Are you responsible for all things electrical in your galaxy? If so, this class is for you. We'll cover all aspects of guarding your AutoCAD Electrical implementation. We'll cover symbol and database management, environment configuration, templates, standards, and best practices. We'll discuss planning out an implementation and upgrades. We are Groot!!

Speaker(s)



Based in Novi, Michigan, **Robert Stein** manages a team of data management implementation consultants for Autodesk, Inc., focusing on data management. Prior to working for Autodesk, Robert worked as a designer, CAD manager, and consultant at a variety of companies. Robert has worked on a number of large-scale implementations and custom programming projects with Autodesk manufacturing products, including Inventor software, AutoCAD Electrical software, AutoCAD Mechanical software, and the Vault software family. Robert is in his 9th year as a presenter at Autodesk University



Tiffany Bachmeier has been an Autodesk Consultant for the last decade. Her primary focus is as a technical consultant/instructor for AutoCAD Electrical, but she also focuses on AutoCAD, Inventor, and a variety of other products in the Autodesk family. She is an Autodesk Certified Instructor and she (and team) has won awards for developing a full line of online, live, instructor-led training classes for the Autodesk manufacturing products. Before becoming a consultant she earned her bachelor's degree from Michigan State University (MSU) and she worked in many different industries gaining valuable CAD experience, including electrical engineering, interior design/architecture, mechanical engineering, software engineering, and she was part of MSU's CAD Development Team. She started on AutoCAD R10 and has carried a strong passion for Autodesk products ever since.

Learning Objective 1: Understand the AutoCAD Electrical Environment

Overview of the Environment File (wd.env)

The wd.env file establishes the environment or default configuration for AutoCAD® Electrical commands and symbol searches. To configure the software for optimum results in a custom environment, you need to know what the file is, where it is located, and how to customize it.

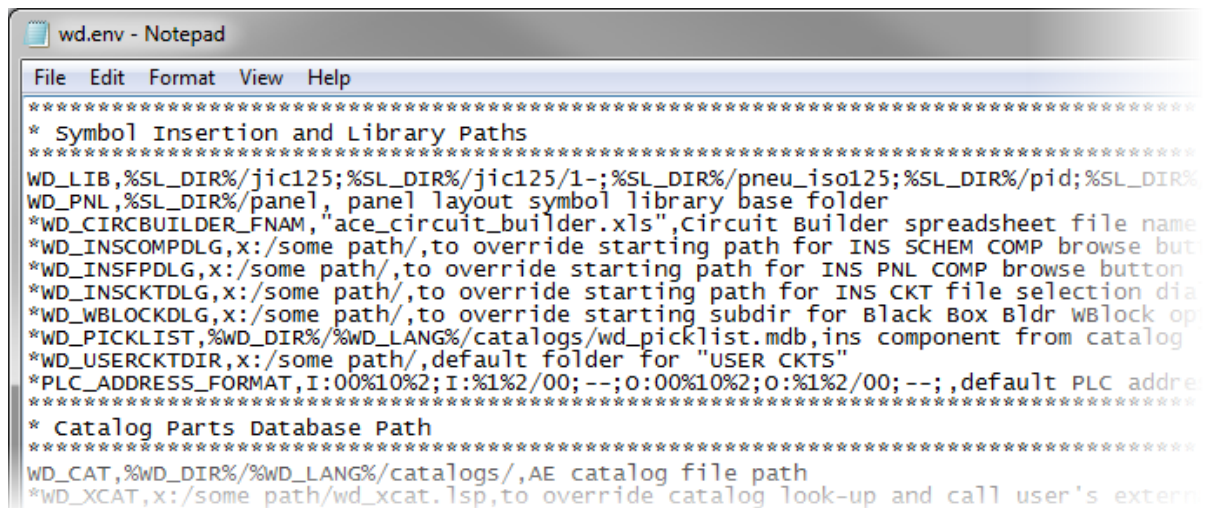
The Environment File Defined

The wd.env environment file for AutoCAD Electrical provides the default search paths when the software searches for reference files or directory locations.

The settings in the wd.env file can be overridden by settings within a project file. For example, changing the symbol library search path in the project properties overrides the settings in the wd.env file. However, clicking Default on the Project Settings dialog box returns the library search paths to the settings from the wd.env file.

You can create project-specific wd.env files. To create a project-specific file, copy the wd.env file and rename it to <projectname>.env. Then open the new file, make the desired project-specific changes, and save it.

A sample of the settings available in the wd.env file is shown below.



```
wd.env - Notepad
File Edit Format View Help
*****
* Symbol Insertion and Library Paths
*****
WD_LIB,%SL_DIR%/jic125;%SL_DIR%/jic125/1-;%SL_DIR%/pneu_iso125;%SL_DIR%/pid;%SL_DIR%/
WD_PNL,%SL_DIR%/panel, panel layout symbol library base folder
*WD_CIRCBUILDER_FNAM,"ace_circuit_builder.xls",Circuit builder spreadsheet file name
*WD_INSCOMPDLG,x:/some path/,to override starting path for INS SCHEM COMP browse but
*WD_INSFDDLG,x:/some path/,to override starting path for INS PNL COMP browse button
*WD_INSCKTDLG,x:/some path/,to override starting path for INS CKT file selection dia
*WD_WBLOCKDLG,x:/some path/,to override starting subdir for Black Box Bldr Wblock op
*WD_PICKLIST,%WD_DIR%/WD_LANG%/catalogs/wd_picklist.mdb,ins component from catalog
*WD_USERCKTDIR,x:/some path/,default folder for "USER CKTS"
*PLC_ADDRESS_FORMAT,I:00%10%;I:%1%2/00;--;O:00%10%;O:%1%2/00;--;default PLC addre
*****
* Catalog Parts Database Path
*****
WD_CAT,%WD_DIR%/WD_LANG%/catalogs/,AE catalog file path
*WD_XCAT,x:/some path/wd_xcat.lsp,to override catalog look-up and call user's extern
```

Note: To edit the wd.env file you must use an external ASCII editor such as Notepad. There are no tools or wizards for this purpose within AutoCAD Electrical.

Location of the Environment File

The default installation location for the wd.env file is:

Windows Vista/7/8: C:\Users\{username}\Documents\Acade {version}\AeData.

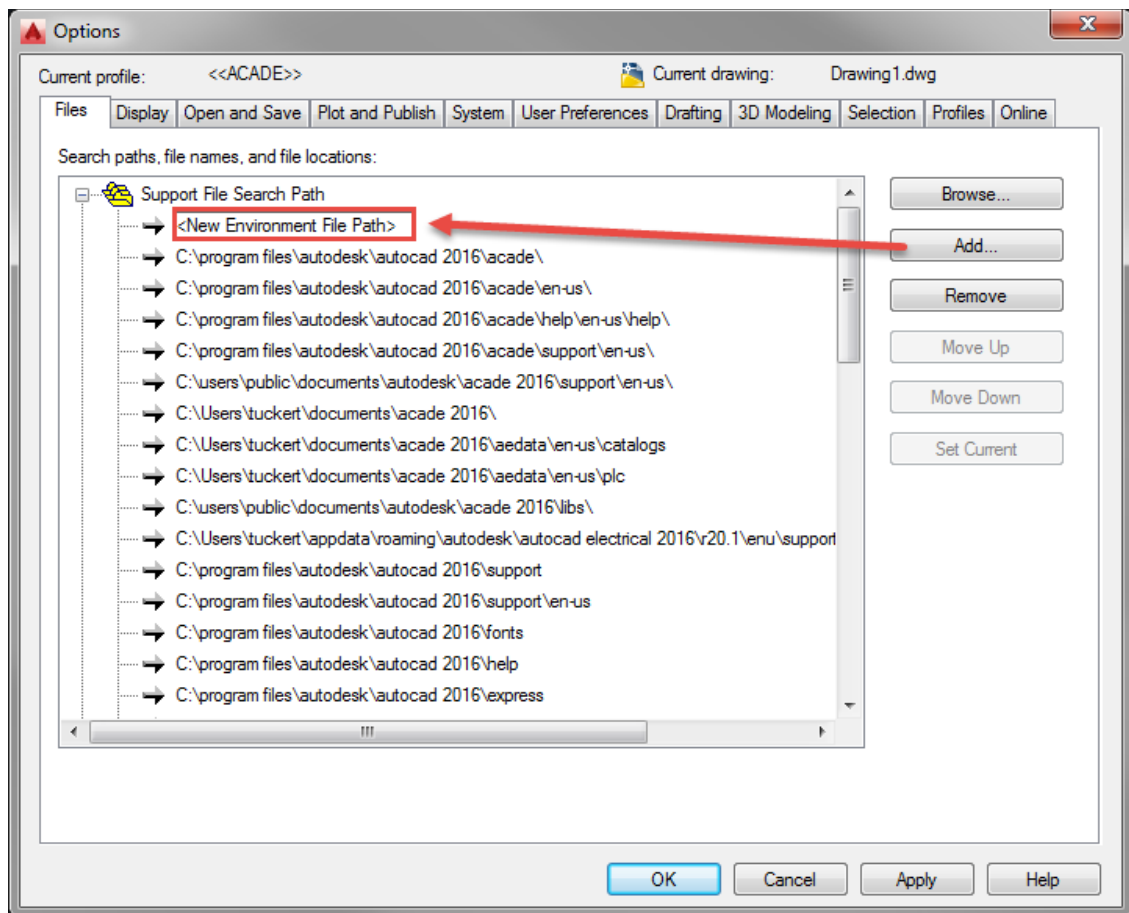
The software stores the path to the wd.env file in the wd_load.lsp file, which is located in the ACADE directory. In a default installation this directory is located at:

C:\Program Files\Autodesk\ AutoCAD {version}\Acade

The wd_load.lsp file is loaded every time AutoCAD Electrical is started. If you change the location of the wd.env file, you need to edit the wd_load.lsp file to point to the new location. All environment files, including project-specific files, must be located in the same directory as specified by the wd_load.lsp file.

The Environment File can also be moved to a different location by saving it in the new location with the same name (wd.env) and removing it from:

C:\Users\{username}\Documents\Acade {version}\AeData\wd.env, then adding the new location to the top of your Support File Search Paths in the Options dialog box > Files Tab:



Example of changing WD.ENV Setting

Your company has created a custom symbol library directory located at W:\Custom Symbols. When you click the Browse button in the Insert Component dialog box, the Browse dialog box should open in this directory.

Edit the wd.env file via any ASCII text editor (Notepad is one of the most commonly used programs), change the following line:

*WD_INSCOMPDLG,x:/some path/,to override starting path for INS SCHEM COMP browse button

The new line should read as follows:

WD_INSCOMPDLG,W:/Custom Symbols, to override starting path for INS SCHEM COMP browse button.

Be sure to remove the leading asterisk and be sure to use the forward slash separator in the path name.

Simply activate a project to force the rereading of the wd.env file. Now, when the Browse button is selected, the W:/Custom Symbols directory is opened in the Browse window.

Learning Objective 2: Understand AutoCAD Electrical Methodologies

Overview of ACADE Intelligent Blocks

Core AutoCAD® technology manages the symbol library system. Schematic and panel symbols are standard AutoCAD blocks that use attributes or xdata to store component information. The icon menu system for organizing, presenting, and retrieving the library symbols is easy to use.

Because standard AutoCAD objects are used, there are no proprietary objects contained within the drawing. AutoCAD Electrical drawings are fully portable. They can be opened, viewed, and edited with no loss of functionality or integrity by any program that can recognize AutoCAD drawings.

Schematic Symbols

A schematic symbol is a standard AutoCAD® block with specifically named attributes. These blocks are generic AutoCAD entities and you can use standard AutoCAD block and attribute creation commands to create them. Any schematic symbol that follows the naming convention and contain expected attributes are fully compatible with AutoCAD® Electrical.

Symbols can be any size and width. You can use any type of AutoCAD object, such as polylines and circles, for the graphical representation of the block. Custom-created blocks function the same way as the supplied symbols. They break wires upon insertion and appear in the Bill of Material, Component, and Wire Connection reports.

One of the easiest ways to create a new symbol is to start with an existing library symbol. Insert a copy of a similar type, explode it, and then modify it to meet your requirements. Avoid deleting the existing attribute definitions. Reposition and edit their default values as required. Then use Symbol Builder or AutoCAD to write a block to an external file (Wblock) to create your new symbol.

TIPS

- Keep all custom symbols that you create in a separate directory. Add the path to your custom symbol directory to the library search path.
- Keep all objects on Layer 0 when creating a new block. During insertion, individual objects in the block are automatically moved to the layers set in the drawing configuration.

Symbol Naming Convention

A specific naming convention is used to enable some automation features. For example, the horizontal and vertical orientation of the symbol is specified by the first character, and the NO and NC state of the symbol is specified by the fifth character.



- 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.

TIP

- The fourth character of a block name has another special property. If the character is a 0 (zero), the wire number will not change as the wire passes through the component. This is true for any type of component or electrical block name.

Symbol Attributes

Attributes are objects that are included in a block definition to store alphanumeric data and are the primary data storage on schematic and panel symbols.

Attributes are especially useful on schematic symbols because they are very consistent. One schematic symbol can represent many different manufacturers and part numbers.

To be treated as an electrical symbol, a schematic symbol must have an attribute named either TAG1 for a parent symbol, or TAG2 for a child symbol, associated with it. Though they are very important for Bill of Material data, wire connections, and other electrical functions and data storage, all other attributes are optional.

TAG1 Attribute

On a parent or stand-alone component, the TAG1 attribute is used for the component tag name or unique identifier, for example, PB101 or CR-55. The default value assigned to this attribute definition is used as the Family Code (%F) portion of the tag format code as set in the drawing configuration. If the TAG1 attribute carries no default value, then the symbol's FAMILY attribute value is used.

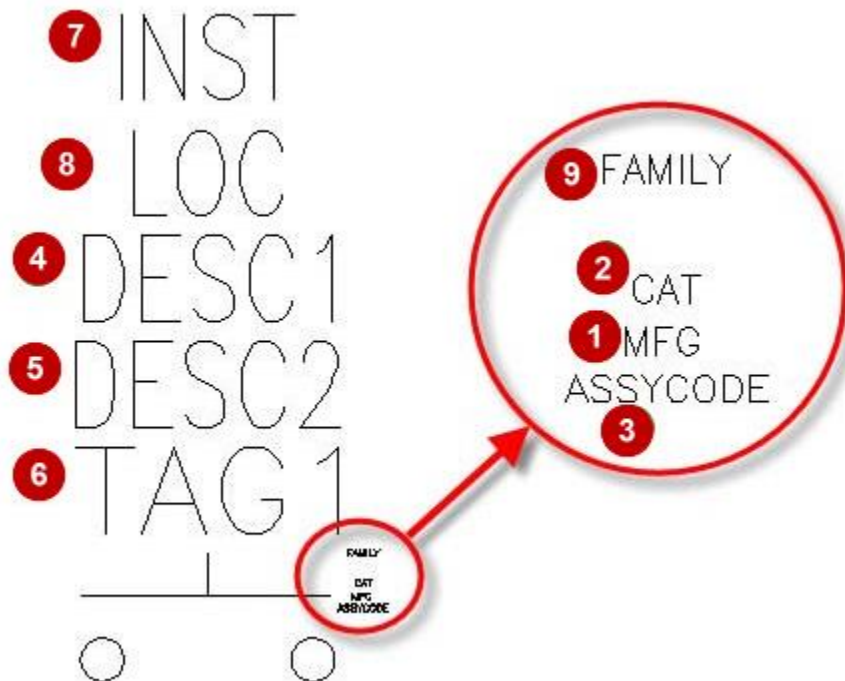
Note: You can override the family name at insertion, during a later edit, or automatically by using the wd_fam.dat mapping file. For more information, see AutoCAD Electrical Help. On the Index tab, type *wd_fam.dat*

TAG2 Attribute

On a child component, the TAG2 attribute is used for the component tag name. When a child component is linked to its parent, the value of the TAG2 attribute is copied from the TAG1 attribute of the parent component. If the child component is not linked to a parent, the default value of TAG2 is displayed.

Other Standard ACADE Attributes

The attributes displayed below are standard attributes and they are typical for any schematic symbols. These attributes are optional, but they are commonly used in both parent and child components.



1 **MFG:** Normally invisible attribute that carries the component manufacturer code.

2 **CAT**: Normally invisible attribute that carries the component catalog number.

3 **ASSYCODE**: Normally invisible attribute that carries the component subassembly code.

4 **DESC1**: Carries the first line of description text (60 characters

maximum). **5** **DESC2**: Second line of description text (60 characters maximum).

6 **DESC3**: (Not Shown) Third line of description text (60 characters maximum).

7 **INST**: Optional component installation code (24 characters maximum).

8 **LOC**: Optional component location code (16 characters maximum).

9 **FAMILY**: Invisible attribute that carries the component family type, for example, CR, PB, or LT. Generally, the default value is the same as the default value for the component's TAG1 or TAG2 attribute.

Note: Many other optional attributes, such as contact, rating, and linking attributes, are used in component symbols. For more information, see AutoCAD Electrical Help. On the Index tab, type *attributes, for schematics (all)*

TIP

- If you need to separate the tag value into two separate lines, you use split TAG attributes: TAG1_PART1 and TAG1_PART2 for the parent, and TAG2_PART1 and TAG2_PART2 for the child components. For more information, see AutoCAD Electrical Help. On the Index tab, type *split tags*

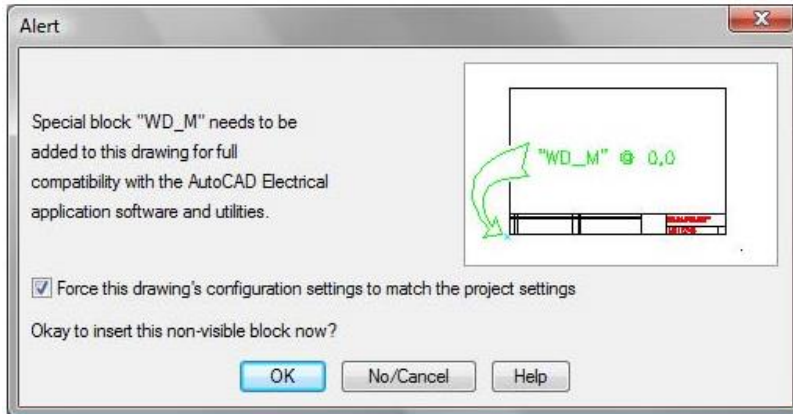
Other Special ACADE Intelligent Blocks

WD_M Block

The wd_m.dwg block file is inserted at the 0,0 location of every AutoCAD Electrical drawing. This block is the only thing that differentiates an AutoCAD Electrical drawing from an AutoCAD® drawing. The wd_m.dwg block file is invisible and contains no graphical information.

When a command is started, AutoCAD Electrical checks for the existence of this block in the drawing and reads the configuration settings for the command from the attributes. If

the block file does not exist in the drawing, AutoCAD Electrical prompts you to automatically insert it before the command is executed.



R
%N
WIRES
WIRENO
WIRECOPY
WIREFIXED
R
%F%N
TAGS
XREF
DESC
TERMS
LOC
POS
MISC
XREFCHILD
DESCCHILD

Whenever you are prompted to insert the wd_m.dwg block file in a drawing, you also have the option to overwrite the default drawing properties with the active project properties. Clear the Force This Drawing check box to turn this option off. A partial listing of attributes contained within the WD_M block is shown to the right.

WD_PNLM Block

In addition to the wd_m.dwg block, panel layout drawings also have another invisible block inserted at the 0,0 location. This block is named wd_pnlm.dwg, and it stores the drawing configuration settings specifically for the panel drawing properties.

AutoCAD Electrical Wire Types

Wires are a very important aspect of AutoCAD Electrical. So what makes a wire, a wire? A wire is any line entity on a valid wire type. Wire types should be thought out as to what you want to get out of the software. AutoCAD Electrical will allow you to function with only one wire type, however your downstream benefits are limited. For example wire to from reports would be missing any information about the detail of what the wire color and gauge are.

Reporting

Reporting is probably one of the most important reasons people use a design package like AutoCAD Electrical. Sure, it helps you to draw faster and more accurate but it does so much more. All of that data you are putting into your design package allows for some very rich reporting on the back end.

Shared Files and Directories

AutoCAD® Electrical uses many types of reference files. In a multiple user environment, many of these files need to be shared with other coworkers to provide design efficiency and avoid redundant work efforts. Below is a list of some of the most commonly shared files and a brief description of their uses.

Reference File Descriptions

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.

- DEFAULT_CAT.MDB: This is the part catalog database.
- WD_DESC.WDD: This contains preset component descriptions. Family-specific versions of the file can also be created, such as pb.wdd.
- WD_FAM.DAT: This overrides the family tag code of the library symbols. For example, changing a push-button family code of "PB" to "K".
- FOOTPRINT_LOOKUP.MDB: This is the database for the graphical footprint assignments based on the catalog part number assignments.
- DEFAULT.WDT: This is the attribute mapping support file for the Title Block Update tool.
- DEFAULT.WDA: This is the reference file for user-defined attributes defined on blocks.
- DEFAULT.INST: This file provides preset values for installation codes.
- DEFAULT.LOC: This file provides preset values for location codes.
- ACE_CIRCUIT_BUILDER.XLS: This is the reference file that the Circuit Builder tool uses.

Note: You can always override the shared reference files with project-specific files by locating a copy of the reference file in the same directory as the project WDP file. Re name the copied reference file to the same name as the project, for example <projectname>_cat.mdb.

Reference Directory Descriptions

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

- Symbol Libraries:
Windows Vista/7/8: C:\Users\Public\Public Documents\Autodesk\AcadE
{version}\Libs
- Insert Component Icon Menus:

Windows Vista/7/8: `C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support`

- Icon Menu Slide Images:

Windows Vista/7/8: `C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support`

- Catalog databases and PLC Database and Images:

Windows Vista: `C:\Users\{username}\Documents\AcadE {version}\`

Windows 7/8: `C:\Users\{username}\Documents\Acade {version}\AeData\en-US\`

TIPS

- For the detailed listings of all the project-related files, see AutoCAD Electrical Help. The topic is located at the Index tab, type *Project Related Files*
- Only some of the more commonly shared folders are listed here. For more detailed listings of shared files, including recommended edits to the `wd.env` file, see AutoCAD Electrical Help. The topic is located at the Contents tab > Advanced Productivity > Set Up AutoCAD Electrical for Multiple Users.

Learning Objective 3: Discover AutoCAD Electrical Best Practices

Sure, you might think of AutoCAD Electrical like it's AutoCAD with some really powerful tools but it's a little more than that. Here are some best practices that we have found to be very useful when coming from an AutoCAD background.

Delete Components Not Erase Blocks

When trying to get rid of a component, think of it as a component since that's what it is. The underlying technology is an AutoCAD Block but within AutoCAD Electrical it's really a component. It has intelligence around it. When you use the AutoCAD Erase command to remove an AutoCAD Electrical component a few things are not going to happen causing you extra work.

- Scratch Database is not updated
- Any wires are not automatically healed.
- Bill of Materials Updates

Always use the Delete Component Command.

Copying a Component

When trying to copy a component in AutoCAD Electrical use the copy component command. Just like with Deleting components, creating new components will need extra work if you simply use AutoCAD commands. For example:

- Scratch Database is not updated
- Any wires are not automatically trimmed.
- Bill of Materials Updates

Always use the Copy Component Command.

Insert Wires Not Draw Lines

When creating wires use the Insert Wires command to help automate the process.

- Lines must be on a valid wire layer
- Wires can only be line entities

Always use the Insert Wire Command.

Learning Objective 4: Learn How to Deploy AutoCAD Electrical

AutoCAD Electrical Implementation Guide

This is an overview of how to setup a deployment for AutoCAD Electrical and how to implement it after it is installed.

Deployment & Implementation Prerequisites

For this implementation to be successful, you must have:

- Working knowledge of AutoCAD®.
- A background in electrical design.
- Have taken an AutoCAD® Electrical Essentials course at an Autodesk Authorized Training Center

Deployment & Implementation Steps

Deployment (see images below)

- Configure Deployment
 - Decide on which Symbol Libraries and Manufacturers to include from the lists provided
 - Decide which Symbol Library will be the Default
 - Decide on Installation and Search Paths for your ACADE Support Files (Network location vs. Local)
- Create Deployment
- Install on users' machines

Implementation

- Create Drawing Template(s)
 - Create/Define Wire Types
 - Define Standard Drawing Properties
 - Make sure your title block is in Paper Space
- Create Project Template(s)
 - Define Standard Project Properties
- Decide what ACADE Support Files to utilize and create
 - Examples
 - Project Line Descriptions: default_wdtitle.wdl
 - Title Block Setup: default.wdt
- Create a complete Standard Project(s) to be used as a template for all future projects
 - Creation of custom schematic symbols, as needed
 - Add symbols to the Icon Menu, as needed
 - Creation of custom panel footprints, as needed
 - Update the Footprint Database, as needed
 - Create new part numbers in the Part Catalog Database, as needed
 - If you use your own company part numbers in association with Manufacturer catalog numbers, fill in the User Fields in the Part Catalog Database

TIPS:

- *Try to work as “out of the box” as you can with ACADE. It will make your implementation much smoother. (Even if it means, changing the way some of your symbols look, etc.)*
- *Although your project files can access drawings from many different folders, it is a best practice to keep the project file, all of its drawings, and project-specific support files in the same place.*

Implementation Steps for a Vaulted Environment

Step 1: AutoCAD Electrical Essentials Training

Step 2: AutoCAD Electrical Implementation

Step 3: Vault Implementation

Step 4: Vault Basic Training

Step 5: Vault inside of AutoCAD Electrical Training

