Get Powered Up How to Implement AutoCAD Electrical

Robert Stein
Autodesk

Tiffany Bachmeier
Autodesk

Learning Objectives

- Discover AutoCAD Electrical Implementation Methodology
- Discover AutoCAD Electrical Settings & Paths
- Discover AutoCAD Electrical Templates
- Learn how to tie it all together

Description

Are you responsible for all things electrical in your universe? 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.

Your AU Experts

Based in the Novi, Michigan Autodesk office, Robert Stein is a Partner Success Manager focusing on the Autodesk Partner Services Enablement. Robert has a long Autodesk background working with customers, partners and students focusing mainly on manufacturing products. 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.

Tiffany Bachmeier is the Manager of the Manufacturing Team for the Americas within Autodesk Global Consulting Delivery. Tiffany leads an amazing team of brilliant consultants focused on manufacturing and advanced manufacturing solutions. Before management, she was an Autodesk Consultant and an Autodesk Certified Instructor for over fourteen years with a primary focus on AutoCAD Electrical, but she also focused on AutoCAD, Inventor, and a variety of other products in the Autodesk family. She started on AutoCAD R10 and has carried a strong passion for Autodesk products ever since.
Learning Objective 1: AutoCAD Electrical Implementation Methodology

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_wdttitle.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
Learning Objective: AutoCAD Electrical Settings and Paths

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.

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 3: Discover AutoCAD Electrical Templates

The AutoCAD Electrical Project

AutoCAD Electrical uses a project-based system to manage the multiple drawings and inter-drawing relationships contained in most electrical projects. Understanding how this system works is essential to increasing your efficiency and creating accurate electrical designs.

Definition of an ACADE Project File

A project file is an ASCII text file with a .wdp extension that stores information about a project. A project file contains some of the following information:

- Project description lines (most commonly used for automatically updating all title blocks)
- Project default settings (design standards)
- Project drawing list, including: Complete path information, Drawing description lines, Section and subsection assignments
- Other miscellaneous catalog and symbol library settings
- Folder structure of the project drawings

To ensure consistency throughout the project drawings, the project settings you store in the project file are referenced when you create or add new drawings to a project. A single project file can find an unlimited number of drawings located in many different directories (though this is not a best practice).

By default, project files are stored in the directory pointed to by the WD_PROJ setting in your environment file (defined during installation), but the project files can be stored in any subdirectory. The location of the project file is used early in the file search path. Custom drawing files, symbol libraries, and other reference files can be stored in the project directory so that you can easily change configurations for different project needs.

Relative Drawing File Paths

Relative path information is used to save the drawing file location. If the drawing is stored in the same directory as the project file, only the file name is stored in the project file. If the drawing is stored in a different directory than the project file, the drawing name information includes both the file name and complete relative path information.

Note: Absolute or fixed paths to drawing files can also be used. To use an absolute path to a drawing file, you must manually edit the project file using any text editor. You cannot enter a fixed path using the project manager.

Guidelines for Project Files

Follow these guidelines when working with project files:

- A single project file can have drawings located in many different directories. There is no limit to the number of drawings in a project.
  - The recommended location for the project file is in the same directory as the project drawing files. Although this is not required, it allows the project to be moved to different directories or entered into file management programs, such as Autodesk® Vault, with little or no management of file paths.
Although you can use any text editor to edit a project file, in most cases it is recommended that you use the Project Manager to make changes.

When archiving or backing up the project drawing files, it is important to include the project file.

This is an example of a typical ACADE project file:

<table>
<thead>
<tr>
<th>File</th>
<th>Edit</th>
<th>Format</th>
<th>View</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td>*[n]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Autodesk Electrical</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>+[n]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ACE_NFPA_MENU.DAT</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>+[n]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ACE_DIR%panel/</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>+[n]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ACE_PANEL_MENU.DAT</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>*[n]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10.75</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14.5</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.75</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>*[n]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.31250</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.00000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Section</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Subsection</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flow and</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interconnection</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Diagram and I/O list</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SUB=SCHEMATICS</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>AU_2014_NFPA_01.dwg</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3-Phase</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Motor Control</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Control Circuit</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SUB=SCHEMATICS</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>AU_2014_NFPA_02.dwg</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Power supplies</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>I/O module feeds</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Lines starting with "*[n]" are project description lines.

Lines starting with "+[n]" are project-wide settings.

Lines starting with "?[n]" are drawing default settings.

Lines starting with "==" are drawing section labels.

Lines starting with "== ==" are drawing subsection labels.

Lines starting with "== == =" are drawing description lines.

Lines without a prefix are project drawing files.

A project drawing file that is stored in the same directory as the project file. Only the drawing file name is listed.

💡 TIPS:

- A project file is not needed if the project consists of a single drawing.
- For more details on what is contained in a project file, go to AutoCAD Electrical Help > Projects and Drawings
Creating a New Project Workflow:

1. In the Project Manager, click New Project.

2. For Name, enter the name for the new project.

3. Select the Create Folder with Project Name check box to create a new folder for the project with the same name that you entered for the project. The folder is created in the path that is specified in Location. The location is also the path where the project file is saved. If left empty, AutoCAD Electrical uses the path to the wd.env file.

4. If you want to copy project settings from an existing project, click Browse to select the existing project file.

5. Click Descriptions to enter project information that can be included in report headers and title blocks. (Described later in this document)
6. Click **OK - Properties** to create the new project, the **Project Properties** dialog box will open. **Note:** This is where you can make changes to the project settings if needed.

![](image)

7. Click **OK** to create the new project without making changes to the settings.
Customizing AutoCAD Electrical Support Files

Various reference files are supported by AutoCAD Electrical to help annotate your drawings. ASCII text files are used as reference files for many different purposes. Only a few of the more frequently used files are briefly explained here.

Knowledge of these files, how they are used, and how they can be made project-specific can help make tasks, such as changing drawing descriptions or mapping title block attributes, easier to understand and complete and when they are customized to your company’s needs, they set the foundation for everyone creating these drawings to follow the same standards.

Component Reference Files

Description (*wd_desc.wdd*), installation (*default.inst*), and location (*default.loc*) files are generic ASCII text files that contain either common values or your company’s standard nomenclature for these fields. Instead of reentering values for each field, you can select the entry from a list.

You can use wizards in the software, or any external text editor, such as Notepad, to edit these files.

- **Component Description file** (*wd_desc.wdd*)
  - Used for defining standard descriptions for components
  - Can be accessed and edited via the Insert/Edit Component dialog box or can be edited via external text editor

Out-of-the-box component reference file *wd_desc.wdd*
- **Location** (*default.loc*) and **Installation** (*default.inst*) files
  - Used for defining standard Location/Installation codes for components
  - Accessed via the **Insert/Edit Component** dialog box and checking marking **Include external list** in the **All Locations - Project** dialog box or **All Installations – Project** dialog box
  - Edited via external text editor

Multiple versions of these files can exist. You can make the files project-specific by replacing *wd_desc* or *default* with the project name. For example, *wd_desc.wdd* can also be labeled `<projectname>.wdd` or *default.loc* can be labeled `<projectname>.loc`. 
Multiple versions can exist because of the how ACADE searches for these files. First, the project directory, where the project's WDP file is stored, is searched for a file with the same name as the project. If a project-named file is not found, the software searches the project directory for the default file. If a project default file is not found, then the software searches for a default file in the support directory (defined at installation).

**Example Use of Project Specific Files**

You work for a company that completes schematic designs and builds panels for many different companies. Your clients use different nomenclature, and in some cases, different languages for the component descriptions and labels.

You create project-specific reference files for each client containing the data specified by the client. You store these files in the same directory as the project file. As you move between projects, the different reference files for each customer are automatically referenced.

**Setting Up Automated Title Block Updating**

- **Define Project Line Labels** (*default_wdtipline.wdl*)
  - Project line description label mappings are stored in a project reference WDL file. These values replace the generic Line 1, Line 2, and so forth, values used in the Project Description dialog box. An unlimited number of lines can be stored in the file.
  - Typically, these values are changed to match the attribute values of the drawing title block, making the title block mappings much easier. They can also be used for many other purposes, including revisions, drawing descriptions, and report information.

Generic line labels:                                    Custom line labels:

- Either a project-based mapping file or a default mapping file can be used for this purpose. You name these files `<projectname>_wdtipline.wdl` or `default_wdtipline.wdl`, respectively. The software searches first for a file that matches the current project name. If a file is not found, the default file is used.
- A wizard is not provided to edit this file; therefore, you must create the file manually, using any ASCII text editor. The entries do not have to be in order and line numbers may be skipped. The file should contain one line per label in the format `LINEx=label` as shown in the following examples:
  - `LINE1=Title 1:`

![Diagram of Project Description dialog box]
Before updating title blocks in your project, you must define how the project and drawing data is mapped to the matching title block attributes. To accomplish this task, you need to understand the formatting involved when using the internal attribute or the external ASCII file.

You can map project and drawing information to attributes in your title block in several ways. You can use the following:

- Any text editor to create an external ASCII-formatted mapping file with a WDT extension.
- The Title Block Setup wizard to create an external ASCII-formatted mapping file with a WDT extension.
- The Title Block Setup wizard to store the mapping on your title block in an invisible WD_TB attribute.
  - When you use the Title Block Setup wizard all mapping formats are maintained automatically whether you use the internal or external mapping methods.
- Typically all drawings in a project share the same title block that contains basically the same information. With the Title Block Update utility, you can automatically update title block attributes with mapped information at any time. You can update the current drawing or selected drawings project-wide.

**Mapping File Options**

- You have the option to store the mapping information in an external file or in an invisible attribute in the title block.
- Each method of storing mapping information has advantages and disadvantages:
  - External File
    - Advantage:
      - External files are easy to edit and change, especially when working with client title blocks, because no changes to the title block are necessary. You can edit these files at any time with any ASCII editor or the Title Block Setup utility. This method is used more frequently when you work with a variety of title blocks from different companies.
    - Disadvantage:
- External files must be in the project search path. Because the data is not contained in the drawing itself, it is not necessarily transferred when the drawing is moved.

- Internal Attribute
  - Advantage:
    - Because the invisible attribute WD_TB is embedded in the title block definition, the mapping information goes wherever the title block goes. This mapping information is seldom lost and is more difficult to change unintentionally.
  - Disadvantage:
    - Because internal attributes are stored in title block definitions, title blocks must be exploded to edit these attributes’ mappings with the Title Block Setup utility. You can manually edit the mappings, without exploding the block, using an attribute editing command. This method is used more frequently with internal title blocks that change less frequently.

- Tip: You can also use a combination of both methods. If available on a title block, an internal attribute is used first. If the internal attribute is not found, the default search path is used to locate an external mapping file.

- External File Options
  - When using an external title block update mapping file, you have three file options to choose from:
    - `<Projectname>.wdt`: Has the same name as the active project and is stored in the active project directory. Used only for the project title blocks.
    - Default.wdt: Stored in the current project directory. If a project-specific file (`<projectname>.wdt`) is not available, this file is used for any project in the same directory.
    - Default.wdt: Final option, located in the search path. Used if either the `<projectname>.wdt` or default.wdp file cannot be located in the active project directory.

- External File Mapping Format
  - The external WDT file has a single line that defines each attribute’s mapping. The first line defines the block name where the attributes are found as shown in the following example:
    - BLOCK=TITLE
    - PROJ_TITLE=LINE1
    - DRAW_TITLE=LINE2
  - As with most configuration files, a project-specific file can be used. The software first searches for a file extension matching the current project name, for example, `<PROJECTNAME>.wdt`. If the WDT file with the project name is not found, the `default.wdt` file is used.
• **Internal Attribute Format**
  
  ▪ If the internal attribute is used, it must be named WD_TB. The attribute must be a part of the block definition itself. If the title block consists of nested blocks, the attribute must be located on the first sublevel; it cannot be a part of a nested block definition.

  ![Attribute Example](image)

  **Note:** The location of the WD_TB attribute within the title block is not important for the function of the software, but it is recommended to keep the attribute within or very near to the title block border. This helps if the title block is exploded and you are looking for the attribute.

  ▪ When manually entering mapping information using an attribute editor, the following format is used:
    
    • **Attribute Name = Project or Drawing Variable**
Each mapping entry is separated by a semicolon, as shown in this example.

Title Block Setup Tool

- The Title Block Setup tool automates the formatting of the mapping data and makes the mapping process easier by listing the available project and drawing data as well as the available block attributes.

- Command Access:
  
  Title Block Setup

  Ribbon: Project tab > Other Tools panel > Title Block Setup

Enter Block Name

- After you select the mapping method in the Setup Title Block Update dialog box, the Enter Block Name dialog box is displayed. You use this dialog box to enter the title block names to search for attributes to map information to. You can select only a single drawing using the Pick Block button, but you can manually enter several names. For example, your company may use different title blocks for different-sized drawings, such as Title A, Title B, and Title C. As long as all three use the same attribute names, the same mapping can be used for all three title blocks.

- In the Block Name field, enter Title A, Title B, Title C. Each time the title block is updated, the drawing is searched for all entered title blocks. Any that are found are updated with the mapped values. You can also use this feature for other blocks that you want to update, such as revision blocks.
You use the Project Values, Drawing Values, and User Defined buttons to move between dialog boxes in the Title Block Setup tool. Each dialog box is specific to the type of data being mapped to the block attributes.

**Project Values**
- The **Title Block Setup - Project Values** dialog box is used to map project description lines to the title block attributes. The Attribute list displays all available attributes in the selected title blocks. Select an attribute from the list to map it to the project description value.
- You use **Pick** to select the attribute in the drawing on the title block itself.

**Drawing Values**
- Use the **Title Block Setup - Drawing Values** dialog box to map information from the individual drawings to the title block attributes. This information changes for each drawing. The title block is updated with information only from the same drawing that the title block is located in.
- Some information in the dialog box is generated automatically. For example, **Sheet Maximum** is the total number of drawings listed as part of the project in the **Project Manager**.
User-Defined Values

- You use the Title Block Setup - User-Defined dialog box to map custom information to title block attributes. You can enter a fixed value, such as your name, or you can enter LISP expressions to generate calculated values.
- In this example, you enter your name in the Text Constant box, mapping it to the DrawnBy attribute. Whenever you run the Title Block Update command your name is automatically mapped to the attribute.
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).
LOC: Optional component location code (16 characters maximum).

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.
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. Rename 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 7/8/10: C:\Users\Public\Public Documents\Autodesk\AcadE {version}\Libs
- Insert Component Icon Menus:
  Windows 7/8/10: C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support
- Icon Menu Slide Images:
  Windows 7/8/10: C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release}\{country code}\Support
- Catalog databases and PLC Database and Images:
  Windows 7/8/10: 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.

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.