

IM125376

## AutoCAD Electrical: 60 Tips in 60 Minutes

Tiffany Bachmeier  
Autodesk, Inc.

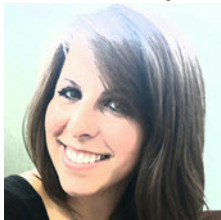
### Learning Objectives

- Discover Project Management Tips and Tricks
- Discover Drawing Creation Tips and Tricks
- Discover Customization Tips and Tricks
- Discover Template Tips and Tricks

### Description

On your mark...Get Set!...GO! Back by popular demand, this fast paced class will keep you on your toes with brand new rapid fire tips and tricks and all-time favorites to get the most out of your time in AutoCAD Electrical. Hold on tight, it's going to be a wild ride!

### Your AU Expert



**Tiffany Bachmeier** has been an Autodesk Consultant for over a 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.



### Tiffany Bachmeier, ACI

AutoCAD Electrical Implementation Consultant  
Global Services, Amer Project Delivery Team  
DESK 248.567.6520  
MOBILE 248.207.7518  
Autodesk, Inc.  
26200 Town Center Dr  
Suite 300  
Novi, MI 48375  
[www.autodesk.com](http://www.autodesk.com)

**Autodesk**  
Certified Instructor



## Overview

The complete list of all 60 Tips & Tricks from this class will be available after the class (have to build up the suspense ☺) In the meantime below are some foundational notes on AutoCAD Electrical.

## Learning Objective 1: Discover Project Management Tips and Tricks

### 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.

```

AU_2014_NFPA.wdp - Notepad
File Edit Format View Help
*[1]AutoCAD Electrical
*[2]Autodesk university
*[3]NFPA
+[1]%SL_DIR%/NFPA;%SL_DIR%/NFPA/1-;%SL_D
+[2]ACE_NFPA_MENU.DAT
+[3]%SL_DIR%panel/
+[4]ACE_PANEL_MENU.DAT
+[5]0

?[10]1
?[11]0.75
?[12]4.5
?[14]%N
?[15]1
?[16]0
?[17]0.5
?[18]
?[19]

?[75]1
?[76]0.0000,0.031250,0.000000
=Section
==Subsection
===Flow and
====Interconnection
=====Diagram and I/O list
=====SUB=SCHEMATICS
AU_2014_NFPA_01.dwg
===3-Phase
===Motor Control
===Control circuit
=====SUB=SCHEMATICS
AU_2014_NFPA_02.dwg
===Power supplies
===T/O module feeds

```

This is an example of a typical ACADE project file(Left):

- 1 Lines starting with "[n]" are project description lines.
- 2 Lines starting with "+[n]" are project-wide settings.
- 3 Lines starting with "?[n]" are drawing default settings.
- 4 Lines starting with "=" are drawing section labels.
- 5 Lines starting with "==" are drawing subsection labels.
- 6 Lines starting with "===" are drawing description lines.
- 7 Lines without a prefix are project drawing files.



- 8 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

## Learning Objective 2: Discover Drawing Creation Tips and Tricks

### *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.

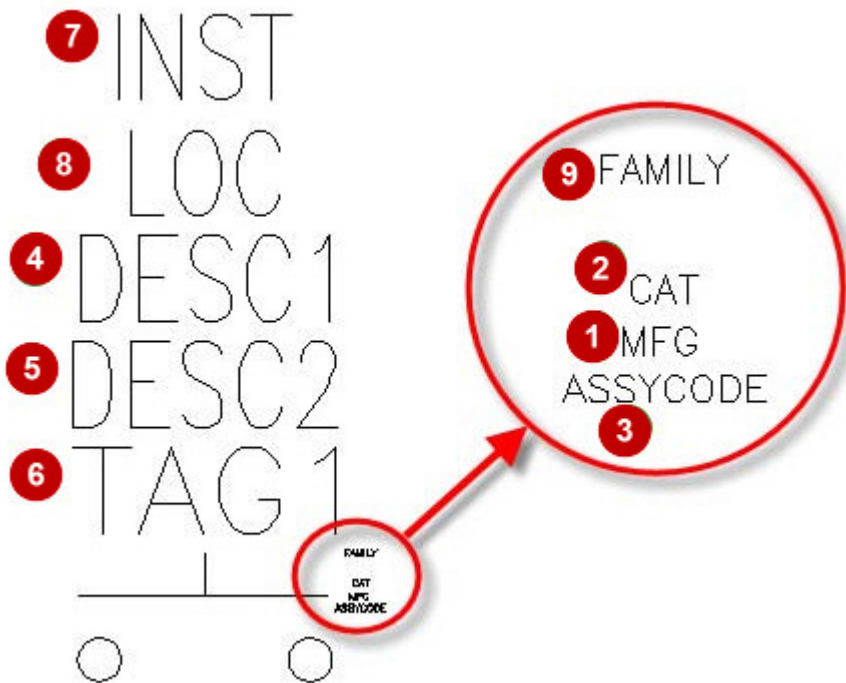
**Note:** You can override the family name at insertion, during a later edit, or automatically by using the `wd_fam.dat` mapping file. For more information, see AutoCAD Electrical Help. On the Index tab, type `wd_fam.dat`

#### ***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*

### Learning Objective 3: Discover Customization Tips and Tricks

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.

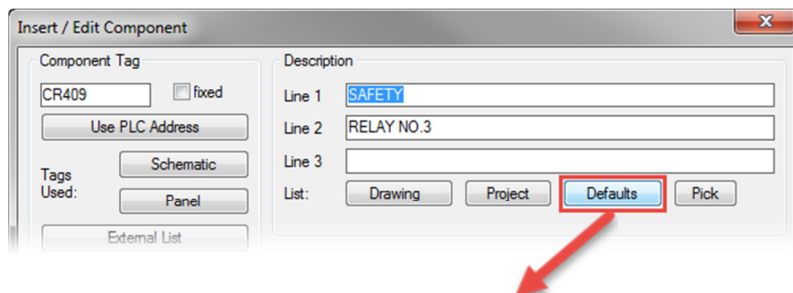
#### Component Reference Files

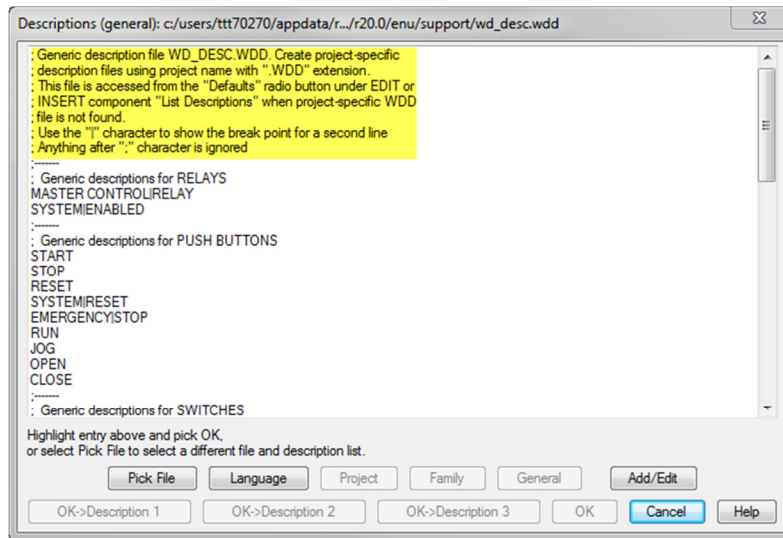
Description (wd\_desc.wdd), installation (default.inst), and location (default.loc) files are generic ASCII text files that contain either common values or your company's standard nomenclature for these fields. Instead of reentering values for each field, you can select the entry from a list.

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

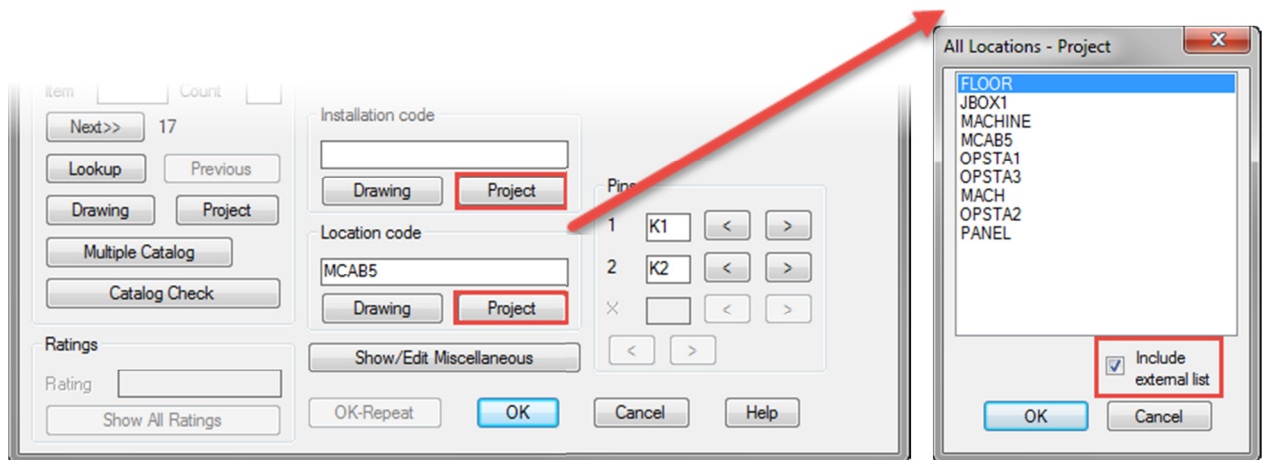




Out-of-the-box component reference file wd\_desc.wdd

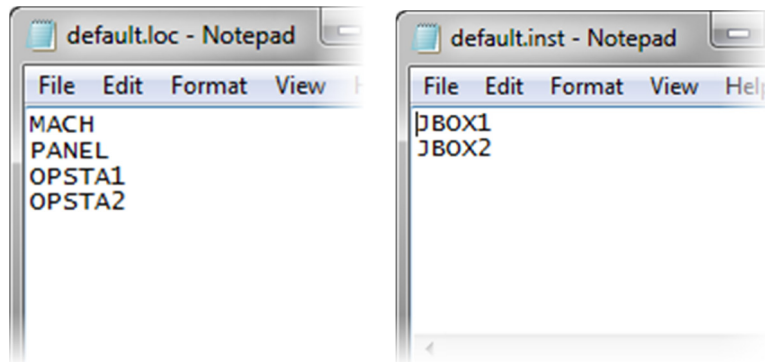
**Location (default.loc) and Installation (default.inst) files**

- 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



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).

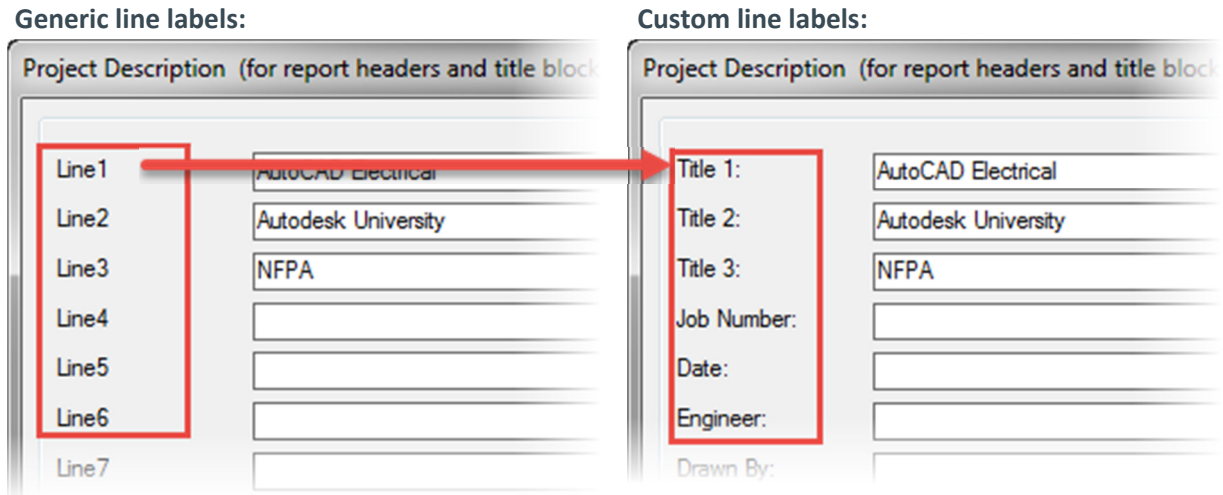
### Example Use of Project Specific Files

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\_wdtitle.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.



- Either a project-based mapping file or a default mapping file can be used for this purpose. You name these files *<projectname>\_wdtitle.wdl* or *default\_wdtitle.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:
  - LINE2=Title 2:
  - LINE4=Job Number:

```

default_wdtitle - Notepad
File Edit Format View Help
LINE1=Title 1:
LINE2=Title 2:
LINE3=Title 3:
LINE4=Job Number:
LINE5=Date:
LINE6=Engineer:
LINE7=Drawn By:
LINE8=Checked By:
LINE9=Scale:
  
```

➤ **Title Block Mapping**

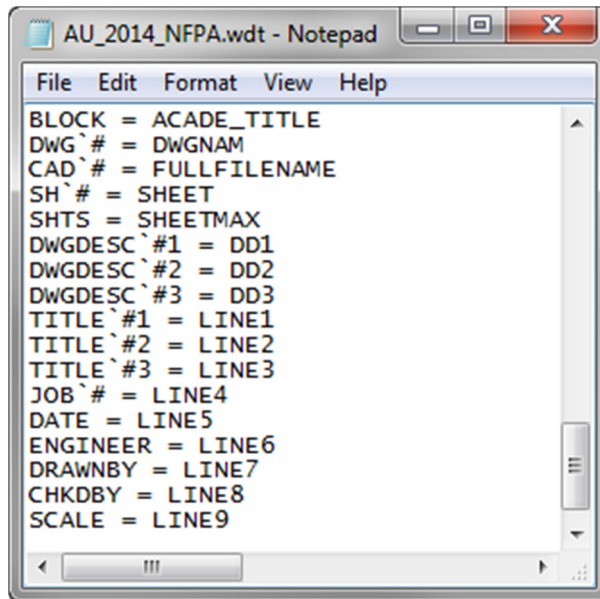
- Before updating title blocks in your project, you must define how the project and drawing data is mapped to the matching title block attributes. To accomplish this task, you need to understand the formatting involved when using the internal attribute or the external ASCII file.
- You can map project 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.



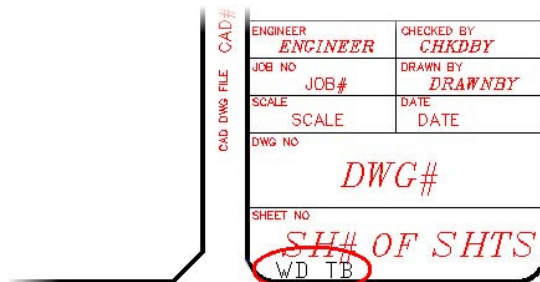
- Disadvantage:
  - Because internal attributes are stored in title block definitions, title blocks must be exploded to edit these attributes' mappings with the Title Block Setup utility. You can manually edit the mappings, without exploding the block, using an attribute editing command. This method 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.



Example of external project specific WDT file

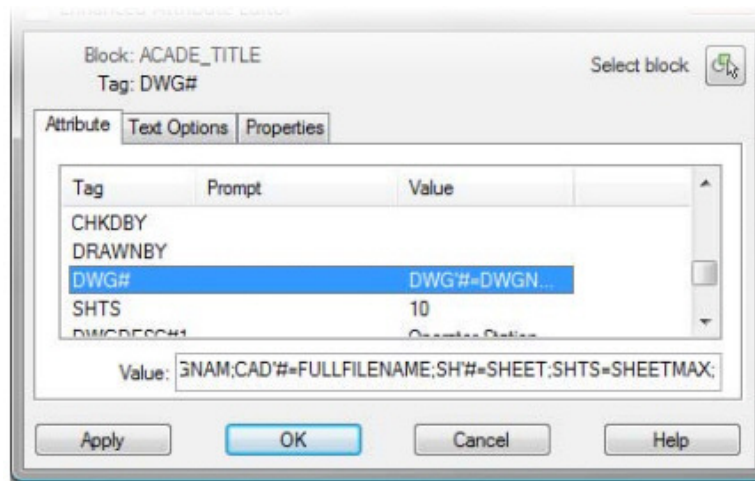
- **Internal Attribute Format**

- 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.



**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:**

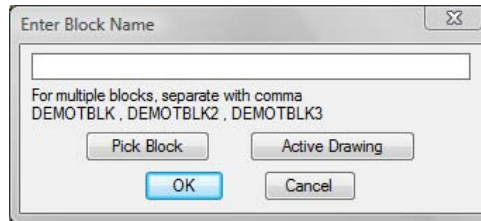


**Title Block Setup**

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

- **Enter Block Name**

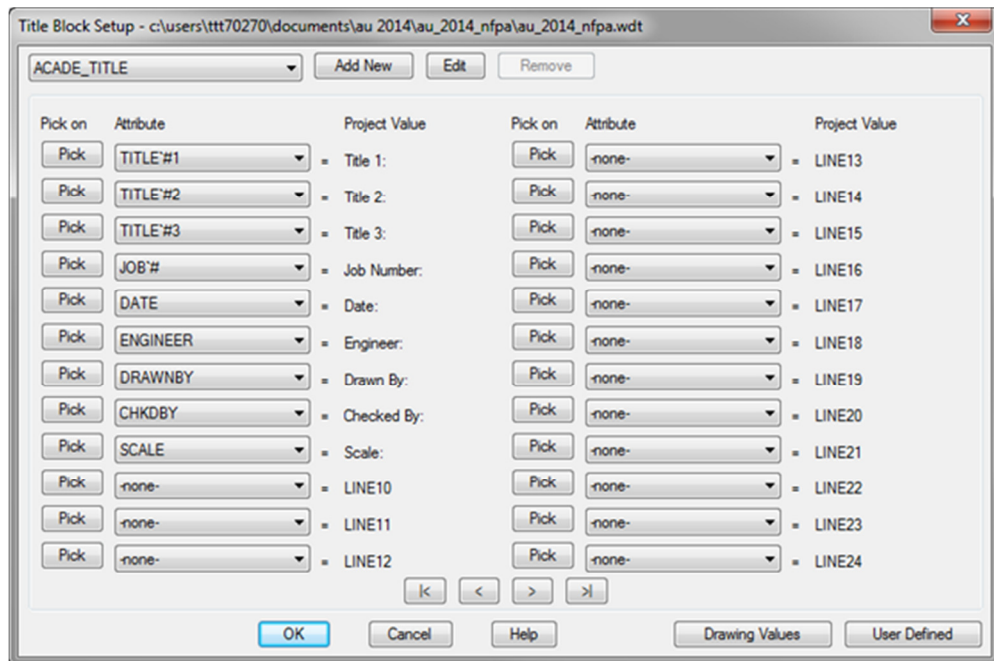
- After you select the mapping method in the **Setup Title Block Update** dialog box, the **Enter Block Name** dialog box is displayed. You use this dialog box to enter the title block names to search for attributes to map information to. You can select only a single drawing using the **Pick Block** 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.



- o 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**

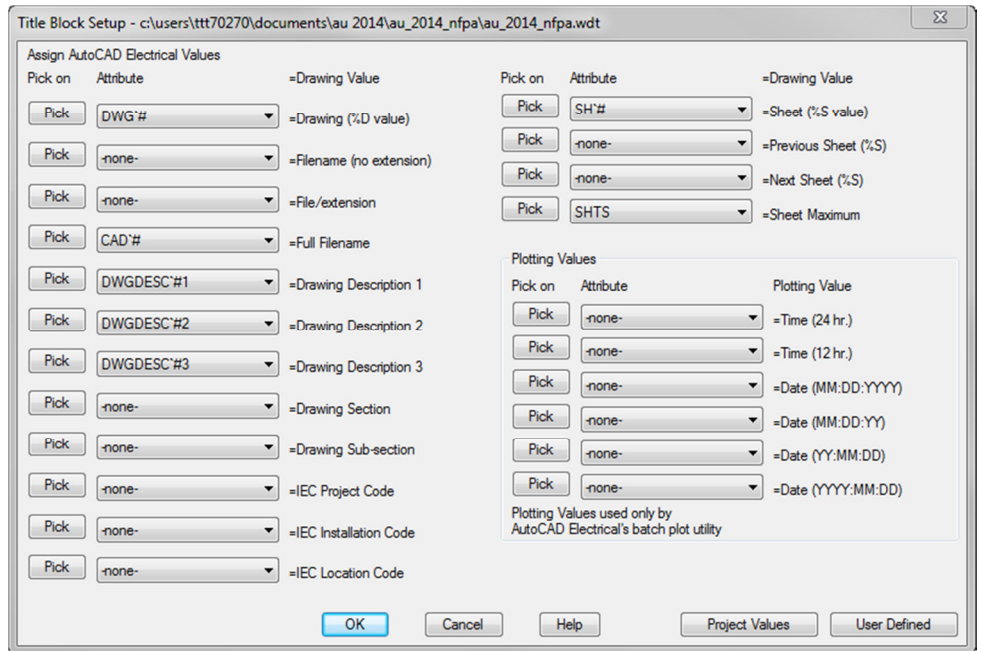
- o The **Title Block Setup - Project Values** dialog box is used to map project description lines to the title block attributes. The Attribute list displays all available attributes in the selected title blocks. Select an attribute from the list to map it to the project description value.
- o You use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



▪ **Drawing Values**

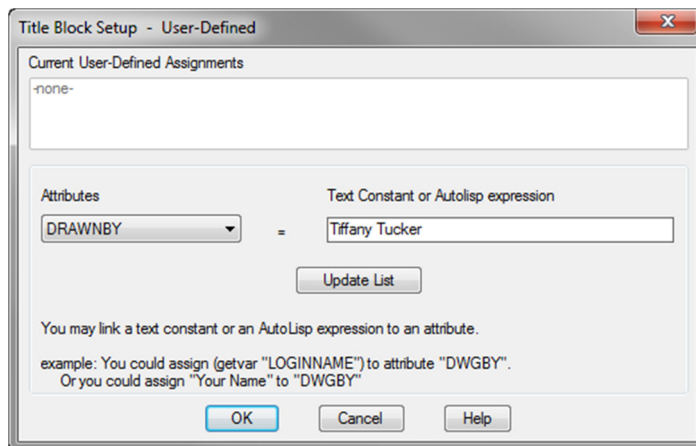
- o Use the **Title Block Setup - Drawing Values** dialog box to map information from the individual drawings to the title block attributes. This information changes for each drawing. The title block is updated with information only from the same drawing that the title block is located in.

- o 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**

- o 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.
- o 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.







## 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 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 4: Discover Template Tips and Tricks**

Drawing templates are extremely helpful in situations where you need to create your drawings with predefined drawing standards, such as layers and drawing properties. Using drawing templates enables you to save the time that you would have to otherwise spend in setting the required standards every time you begin a drawing. In organizations, CAD managers create template drawings and make them available for their team.

### **Definition of Drawing Templates (for AutoCAD and AutoCAD Electrical)**

A drawing template is a collection of standard predefined settings, such as units, title blocks, layers, text styles, and dimension styles, which you can use for creating many drawings. Drawing template files have a `.dwt` file extension.

### **Drawing Templates and CAD Standards (for AutoCAD and AutoCAD Electrical)**

When you work in a project in which many people are involved in creating a design, you must ensure that all team members consistently follow the same drawing settings. Therefore, to maintain consistency across drawings, you can establish CAD standards by sharing and using `DWT` files.

For creating a `DWT` file, you define the required drawing settings and save the file as a drawing template. You can also save a `DWT` file as a drawing standard (`DWS`) file. You can then use a `DWS` file to check and map a drawing with a drawing template for any violation of the set standards.

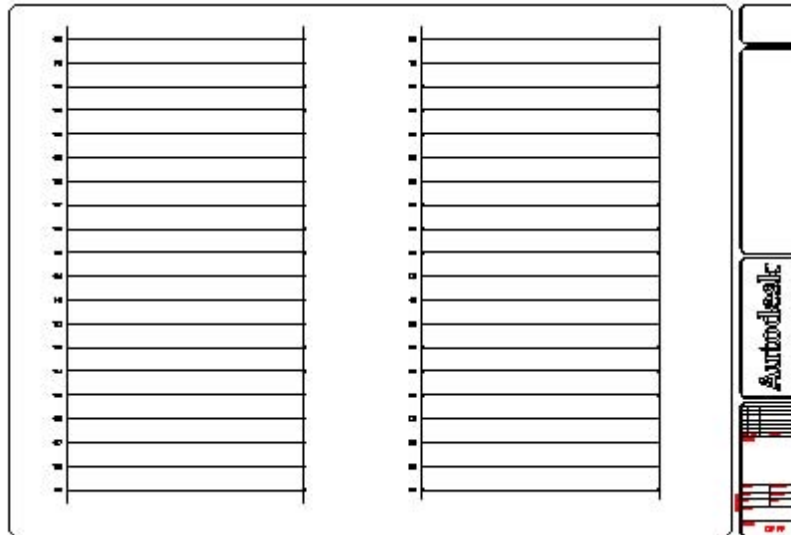
### **Preset Drawing Graphics**

Templates can also include partially completed or preset drawings. These are useful when a drawing or part of the drawing is a standard component that is frequently used in your company design projects.

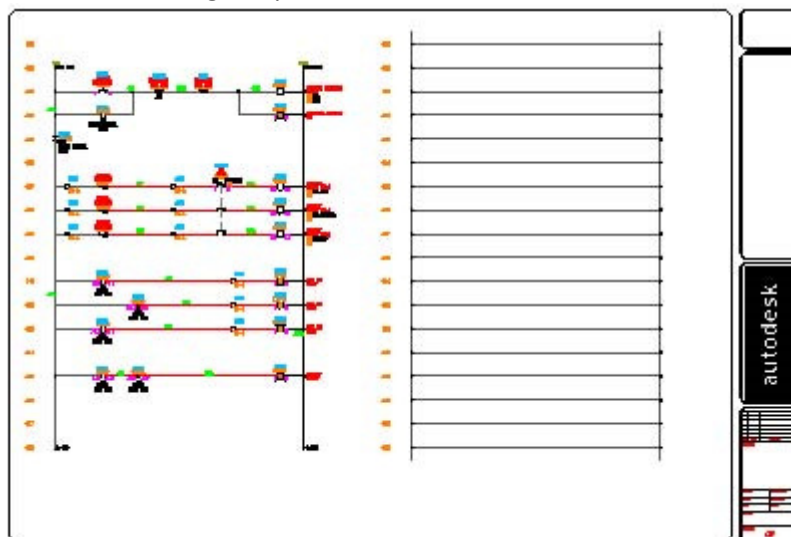
**Note:** After creating a drawing that is based on a *DWT* file, if you modify the new drawing, the changes do not affect the *DWT* file.

### Example of Drawing Templates

The following images show various examples of electrical drawing templates.



An electrical drawing template that includes title block and two ladders.



An electrical drawing template with a title block, basic circuits, and a ladder.

When creating drawing templates, you can save all or some of the template properties and settings based on the type of drawings that you can create with a new template. You can modify these properties later, if required.



## Template Properties and Settings

You use drawing templates to provide a starting point for all the new drawings that you create. In most design environments, your drawings share some common properties and settings. When you save a drawing template, you can save all the drawing commonalities, thereby eliminating the need to create or adjust properties and settings each time you create a new drawing.

### AutoCAD Electrical Templates

For templates created for use with AutoCAD® Electrical, it is recommended that you have the *wd\_m.dwg* block inserted and the drawing properties set to match the template purpose.

You can include wire layers, ladders, partial circuits, symbols, and other graphical information to provide a preset starting drawing that matches company standards or commonly used designs.

The following are some of the properties and settings that you should save in a drawing template:

- Drawing properties settings for electrical configuration
- WD\_M Block
- Wire Layers, colors, and names
- Snap and grid mode settings
- Dimension, text, and table styles
- Title blocks and borders

The following are some of the other items that you can save in a drawing template:

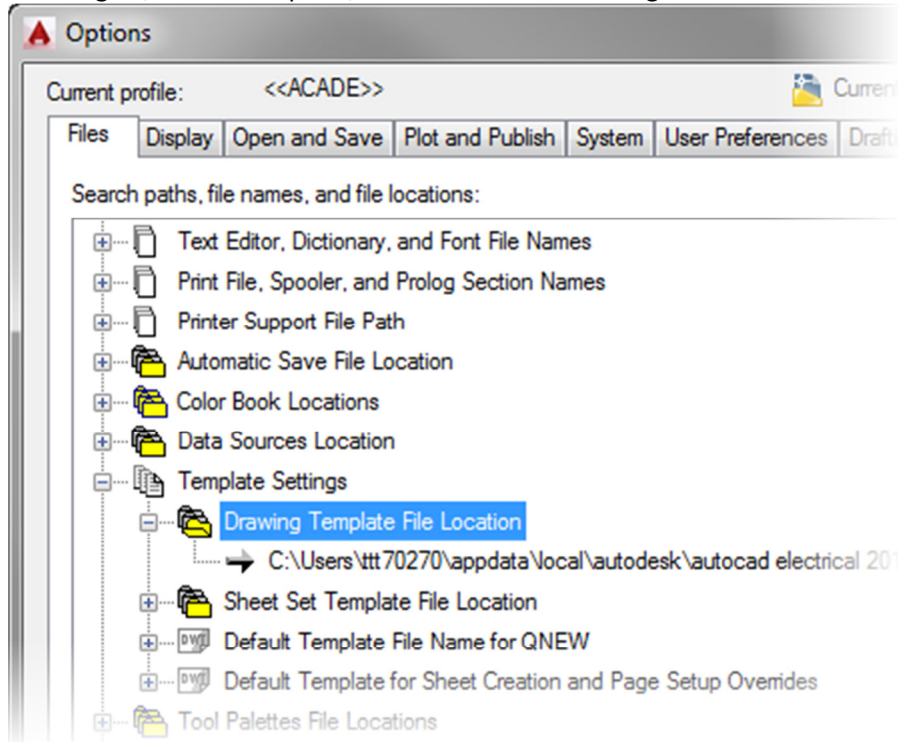
- Blocks, such as symbols or other objects that you commonly use in your drawings
- Ladders, circuits, and other graphical entities

### Storage Location of Drawing Templates

Before you create your drawing templates, you need to specify their storage location.

You specify the path to the *DWT* files on the Files tab of the Options dialog box. A path on the local hard drive may work if you are working in a single user environment. However, if you are working as a part of a design team, you should set the path to a network location where all project drawing templates are consolidated.

The path that you specify as the file location of drawing templates controls the default location that appears when you select the Drawing Template (\*.dwt) format in the Files of Type list in the Save Drawing As, Select Template, and the Select File dialog boxes.

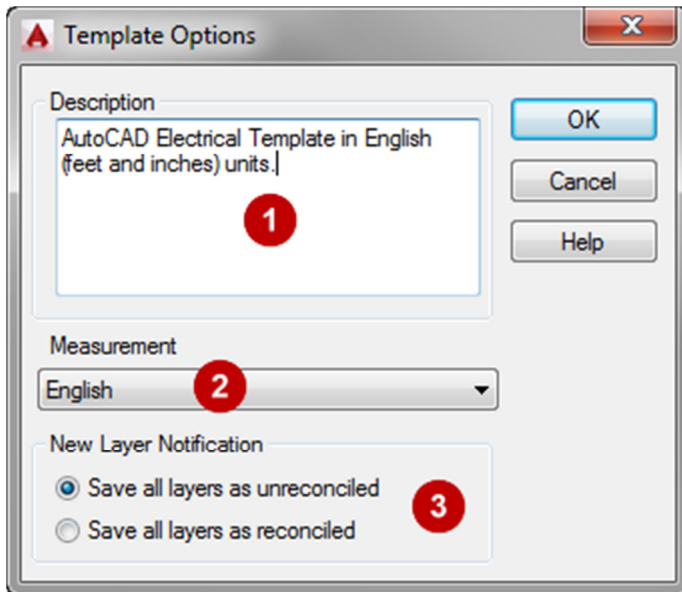


### Template Options Dialog Box

By using the **Template Options** dialog box, you can set the drawing units to either imperial or metric, provide a description for the template, and control new layer notification.

To access the **Template Options** dialog box, you select the **AutoCAD Drawing Template (\*.dwt)** option from the **Files of Type** list in the **Save As** dialog box.

The following image shows the **Template Options** dialog box.



- 1** **Description:** Specifies a description for the DWT file.
- 2** **Measurement:** Determines whether drawings based on this template use English or Metric units.
- 3** **New Layer Notification:** Saves all layers as unreconciled or reconciled. When you save a *DWT* file with unreconciled layers, the layer baseline is not created; therefore, the new layer notification is not displayed. When you save a template with reconciled layers, a layer baseline is created; therefore the software notifies you of any new layers in the drawing.

**Note:** All the layers in a *DWT* file are saved as unreconciled by default.