AUTODESK UNIVERSITY 2013

Ground Yourself with a Properly Implemented Electrical: Implementing Autodesk® AutoCAD® Electrical

Eric France – IMAGINiT Technologies
Stan Wile – IMAGINiT Technologies

MA1600 Do you work collaboratively or independently in AutoCAD Electrical software? If you have more than one user at your company creating drawings with AutoCAD Electrical, this class helps you understand all the ways you can reference common, shared files effectively. The class covers how to install, configure, and organize those files so that drawings easily follow established standards and appear consistent from project to project and user to user. The class digs into some key ways AutoCAD Electrical promotes standardization and shows you an effective way to implement AutoCAD Electrical that allows users to work productively and consistently from a common set of files.

Learning Objectives

At the end of this class, you will be able to:

- Share common libraries, databases, and other reference files effectively
- Configure the environment file to access shared locations first
- Organize your shared locations for convenient access and maintenance
- Use standard reference files to share data among users and between projects

About the Speaker

Eric France is a Solutions Consultant for the Manufacturing Solutions Division of IMAGINiT Technologies. He has worked in their Denver office since 1998 and has had the privilege of helping hundreds of Manufacturers maximize their software investment. His Manufacturing experience goes back to his childhood, when he would help his father at his machine shop in upstate New York.

His first exposure to CAD, in various forms, was in the late '80's. He has been using AutoCAD since 1992. He is proud to say his first encounter with AutoCAD Electrical was when it was still VIA WD. His career has allowed him the opportunity to learn, internalize, and promote the suite of Manufacturing tools provided by Autodesk from their release, as well as many complimentary products from third-party vendors.

With the notable exception of a 3 year paid vacation hauling Abrams tanks around Europe for the US Army... Computers, Manufacturing, and CAD have been a staple of his existence for over 25 years.

EFrance@Rand.com
@IMAGINiT_EricF

Share common libraries, databases, and other reference files effectively

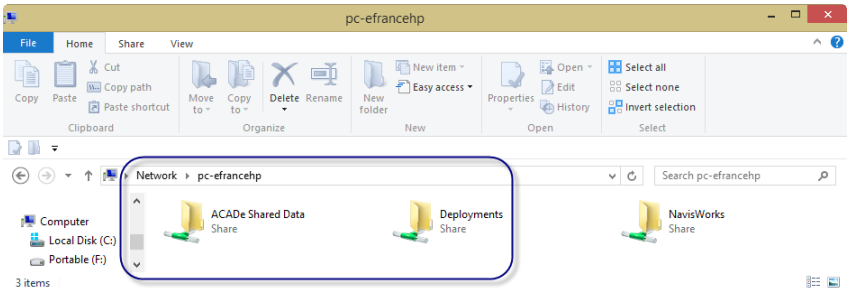
Let's begin at the beginning, shall we? The best and most reliable method of sharing AutoCAD® Electrical content between multiple users within the same company is by creating a deployment and sharing the databases and libraries on the network. The opportunity to path users by default to a network location for referencing AutoCAD® Electrical content is only available during the creation of a deployment and any other method of sharing that data will produce inconsistent results. The greatest benefits of having users share the AutoCAD® Electrical content is the time that is saved by reusing instead of creating and the consistency that results from accessing data from a single source.

Creating a Deployment

When creating a deployment for AutoCAD® Electrical, we should have two discrete network shares available. The first is the network location for the deployment itself and the second is a separate share for the AutoCAD® Electrical content. Most IT departments have a common location where software installers are located to aid in the deployment of software throughout the organization. This is a great place for the deployment to reside.

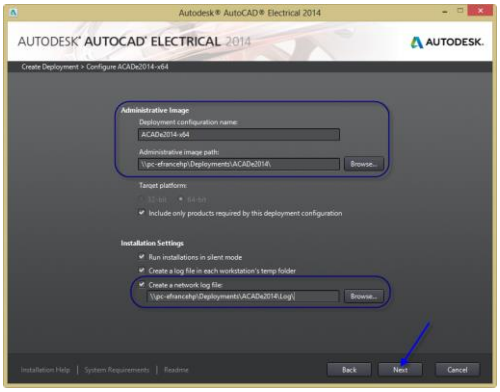
Sharing Content

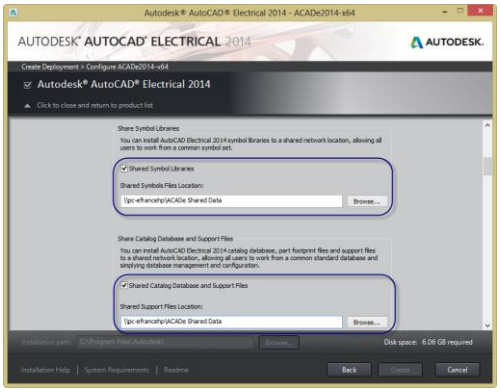
However, the share for the AutoCAD® Electrical content should be isolated from the deployment share and is typically located on the dedicated Engineering server, if available. The reason for this is because the permissions for the two shares are significantly different. AutoCAD® Electrical users will be adding to and adjusting the content that is on the content share, so its permissions must be more open for AutoCAD® Electrical users, but closed to most everyone else. Once these two shares are established, we can configure the deployment.



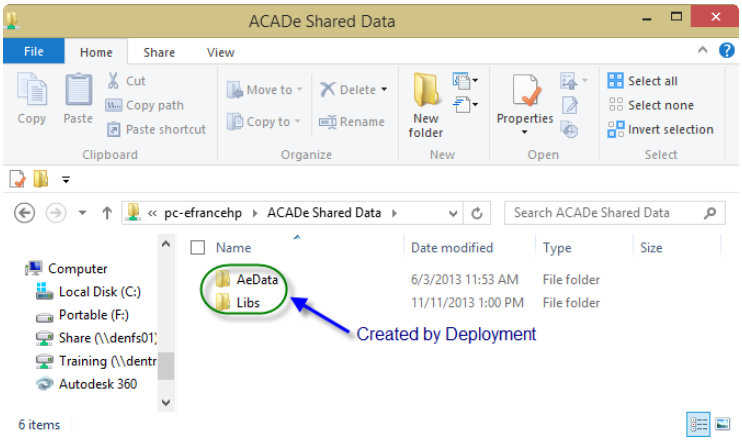
Configuring the Deployment

With the shares in place, we kick off the installer. Soon enough, we are presented with the option to *Install* or *Create Deployment*. To set up the users to use shared content, we **pick** *Create Deployment* and complete the deployment name and location information. Then, we accept the EULA, enter the licensing information, and expand the options for configuring Autodesk® AutoCAD® Electrical 2014. In the Shared Content section, we identify the share path for the "Shared Symbol Libraries" and the "Shared Catalog Database and Support Files" with the same shared path. Subfolders will automatically be created for each content type.





We complete the configuration of the deployment by selecting Manufacturers for the catalog, standard libraries, Express Tools, the latest Service Pack, and any other options we require. By collapsing the options for configuring Autodesk® AutoCAD® Electrical 2014, we enable the *Create* button for the deployment creation. **Pick** *Create* to initiate the creation of the deployment and the copying of the selected content to the content share.



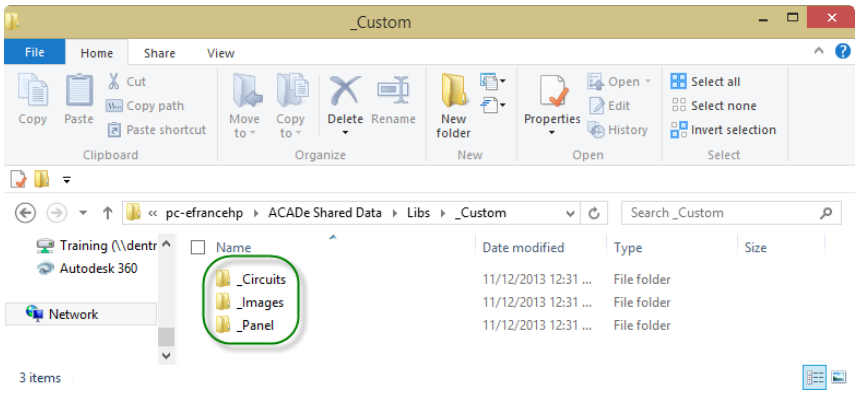
Adding Other Reference Files

Once the deployment has been created, we can start adding additional reference files and folders to the content share. Ideally, anything that users need to share while working with AutoCAD® Electrical should be located here. The only exception to consolidating our custom content is the AutoCAD® Electrical projects themselves. Those should be stored on the corporate file server with all the other project data or inside a data management system, like Autodesk® Vault. For now, we will focus on the locations needed for our symbol libraries.

Adding Custom Library Paths

Understandably, many organizations like to keep their custom symbols separated from the standard libraries. To facilitate this, we create a _Custom folder in Libs to store custom symbols. Also, we need somewhere to store the images used in the customization of the Icon Menu. We will place them in an _Images sub-folder in our _Custom folder to organize them. Since we will be creating and saving custom Circuits and Panel footprints, we will add sub-

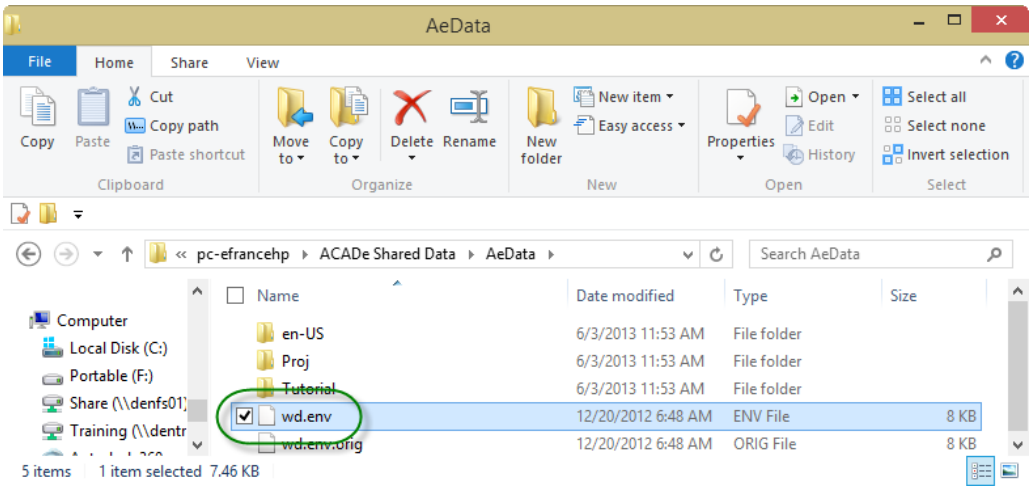
folders for them to _Custom too. There are certainly more files we can consolidate here, but we will discuss that additional content later on in the session.



Now that we have established a folder structure for the custom symbols we will use, let's take a look at how we direct AutoCAD® Electrical to use them by default.

Configure the environment file to access shared locations first

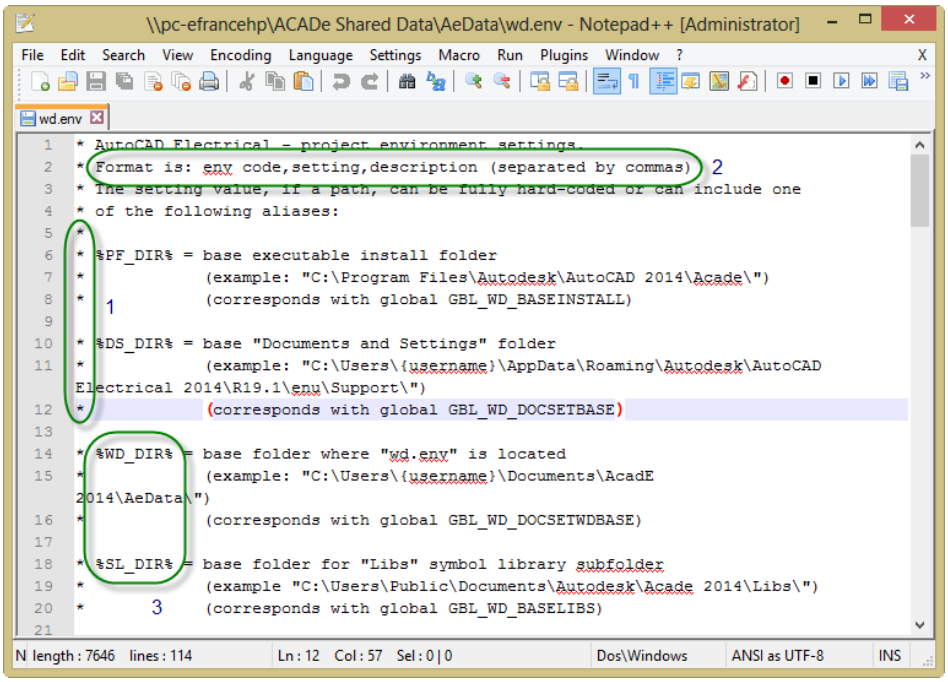
Having created all the paths that we will be storing our custom symbols in, we now need to adjust the environment file to look to those paths by default. First, we need to locate the wd.env that was created in the deployment. It should be found in the root of the AeData folder where the databases are stored. Once located, create a copy of the original called wd.env.orig and then open the wd.env in your favorite ASCII text editor, ex. Notepad++.



Configuring the wd.env file

Let's spend a little time orienting ourselves to the format and syntax used in the wd.env file. The first thing to know is that an "*" will prevent everything that follows it from being read by

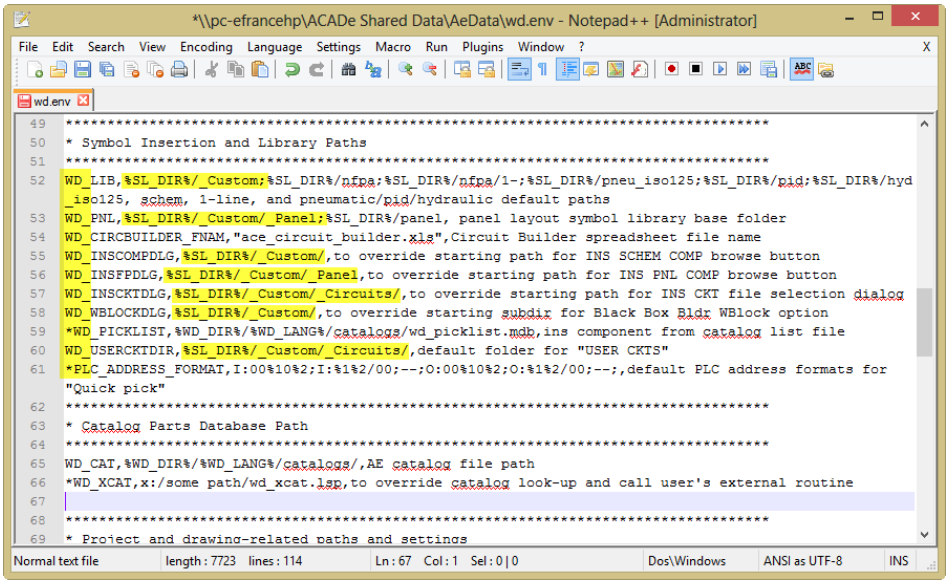
AutoCAD® Electrical. This can be seen in use quite effectively in the top portion of the environment file below (green vertical bubble marked “1”).



The format of the environment variables in the wd.env file is env code,setting,description (separated by commas) as is seen by the green horizontal bubble marked “2”. For the “env code” to be active and read, any preceding “*” must be removed and then the “setting” needs to be set to the path we want to use. The “description” is there to help us understand what that variable controls. There are several aliases listed in the top portion of the wd.env file as well. All can be used in defining the path for the variable, but two in particular are useful in the way we are setting up our shared content. They are %WD_DIR% (default path to the AeData folder) and %SL_DIR% (default path to the Libs folder) as is seen by the green bubble marked “3”. Let’s look at using them to define our Custom Library search paths.

Configuring Library Paths

As we scroll through the wd.env file, we reach the “Symbol Insertion and Library Path” section. Here we have the ability to redirect AutoCAD® Electrical from the default locations provided in the install to the custom paths we actually need it to use. As you can see below, we need to adjust the lines that we want to use by first removing the “*” that precedes the line. Then, we replace the default path with the specific location we want that aspect of the software to use. For the portion of the path that references the default Symbol Library directory structure defined in the Deployment, we are able to use the %SL_DIR%, like in the WD_INSCOMPDLG line. We can also use “;” to list multiple library paths in the order we want them to be searched, as can be seen in the WD_LIB and WD_PNL lines.



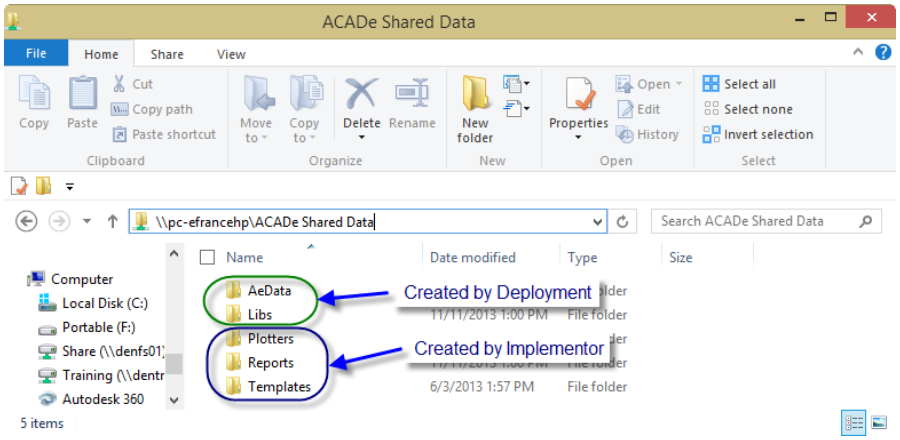
Now that we have directed AutoCAD® Electrical to use our custom library paths, let's look at some additional content that we can place in the content share.

Organize your shared locations for convenient access and maintenance

Beyond library paths, we can also use the content share as a location to consolidate the company data that we use frequently and the support data that AutoCAD® Electrical uses. This is really just an opportunity to maximize the paths created by the deployment for the shared content as a central repository for the rest of our AutoCAD® Electrical-related data. By consolidating the data here, we have an easier time getting new users up-to-speed and managing the data stored there. This is because there is only one location new users have to become familiar with for all things AutoCAD® Electrical. Finally, it streamlines administration because we only have one location to backup and to control the permissions on.

Adding Company Content Folders

Some typical additions to the content share are Templates and Plotters. By storing our template files on the network and defaulting to them, we maximize the likelihood that everyone starts new drawings with a consistent format. By storing our edited and saved plotter configurations on the network, we significantly increase the probability that plots to the office plotters are generated with consistent settings. An additional folder to consider adding to the network share is a Reports folder. This one is used to hold all the report-related files that get produced when leveraging Automatic Reports, like the Report Format and Report Grouping files, as well as the output report files themselves. Having a central location where we go to get all the data extracted from our project drawings can be very beneficial.



Combining Support Folders

Finally, combining the multiple Support folders that AutoCAD® Electrical uses by copying them to the content share can also provide a benefit for everyone. Having all the standard .ddl, .dcl, .lsp, and .slb files in a single shared location provides consistency in appearance and behavior of the application. Similarly, having all the Icon Menu files (.dat) stored and referenced from a single location means edits to the Icon Menus can be streamlined and always up-to-date on every user's machine. There are also a large number of files that AutoCAD® Electrical references to speed the project drawing creation process for users. Those will be discussed later in the session.

These Support paths can be found by default on a Windows 7 or Windows 8 system at:

C:\Program Files\Autodesk\AutoCAD 2014\Acade\Support\en-US,

C:\Users\Public\Documents\Autodesk\Acade 2014\en-US\Support,

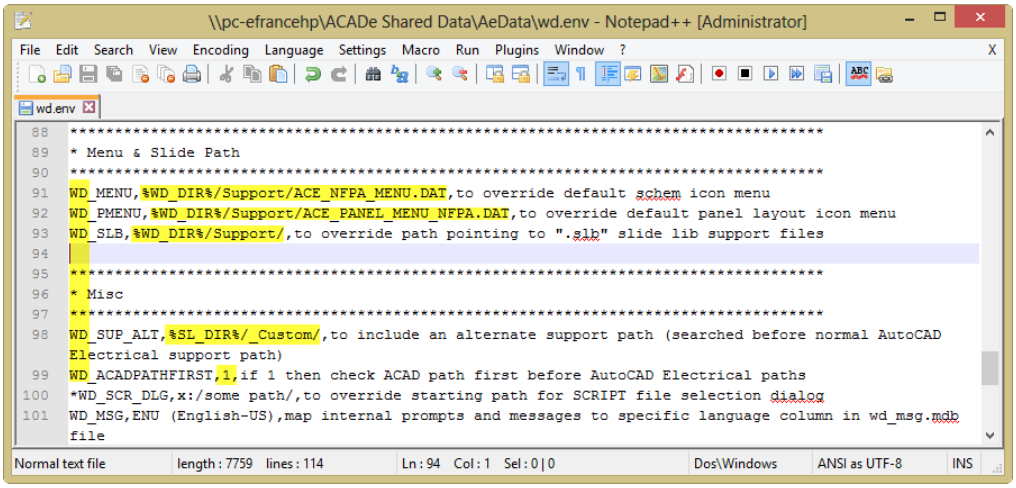
and

C:\Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical 2014\R19.1\enu\Support

Now that we have consolidated the common files we will be referencing in AutoCAD® Electrical, let's take a look at how we point AutoCAD® Electrical to use them in their shared location.

Configuring Support File Usage

With our support files all combined into our shared content path, we again need to look at directing AutoCAD® Electrical to use the files and paths in our shared content directory structure. We again scroll through the wd.env file until we reach the "Menu and Slide Path" section. First, we remove the preceding "*" for all the lines we want to redefine. Then we need to specify the common Schematic and Panel Icon Menus that we want AutoCAD® Electrical to use. We want everyone using these specified Icon Menus, because we will edit these menus to include our custom symbols and footprints that we use regularly. Next, we specify the path where all the .slb and .dll files have been copied, so the interface displays the correct images in the configuration and command dialogs.



Finally, in the “Misc” section, we can prioritize the path or order in which AutoCAD® Electrical searches for content. The WD_SUP_ALT path is used to force AutoCAD® Electrical to search for content in the specified location before searching in the normal search paths. Alternatively, the WD_ACADPATHFIRST variable allows us to have AutoCAD® Electrical run through the paths we specify in the Support Files Search Paths in the Options dialog before the normal search paths it would use.

Having reviewed the edits to the environment file, let’s look at all the standard reference files AutoCAD® Electrical provides to standardize the entry of data into our drawings.

Use standard reference files to share data among users and between projects

Our final topic is large, at least in quantity. These are files that can be used in AutoCAD® Electrical to store common data to be shared between users and across projects. For the most part, these files are either databases or they are ASCII text files. Most of the databases are an integral part of the software and come loaded with a large amount of content. The ASCII text files are typically much lighter weight and we will need to add our custom data to them. Let’s take a deeper look at where to find these files and how to access them within AutoCAD® Electrical to make our edits.

Company Reference Databases

The databases are Access .mdb files and are mostly stored in the …\AeData\en-US\catalogs folder. The major exception to that is the PLC database stored in the …\AeData\en-US\plc folder. The most commonly accessed one is the default_cat.mdb or the Catalog Lookup Database. This database also stores the Terminal Block Property tables and the Pin List Database tables. The following table identifies the different databases that AutoCAD® Electrical references, their filename, and where in the interface they can be accessed from for editing their contents.

Database Description	Filename	Access From…
Catalog Lookup Database	default_cat.mdb	Lookup button in Insert/Edit Component dialog
Footprint Lookup Database	footprint_lookup.mdb	Panel tab > Other Tools panel (expanded)
Schematic Lookup Database	schematic_lookup.mdb	Schematic tab > Other Tools panel (expanded)
PLC Database	ace_plc.mdb	Schematic tab > Other Tools panel (expanded)
Pin List Database	default_cat.mdb	Schematic tab > Other Tools panel (expanded)
Terminal Properties Database	default_cat.mdb	Schematic tab > Other Tools panel (expanded)
Electrical Standards Database	ace_electrical_standards.mdb	Schematic tab > Other Tools panel (expanded)
Language Conversion Database	wd_lang1.mdb	Project tab > Other Tools panel

The contents of most of these databases are determined at the time the Deployment is created based on the Manufacturers that we select. While there is a large amount of content included, it is likely that we will have to add our own content based on the specialized devices used in our industry and we may even want to remove content we will never use.

Project Information Capture and Dissemination Files

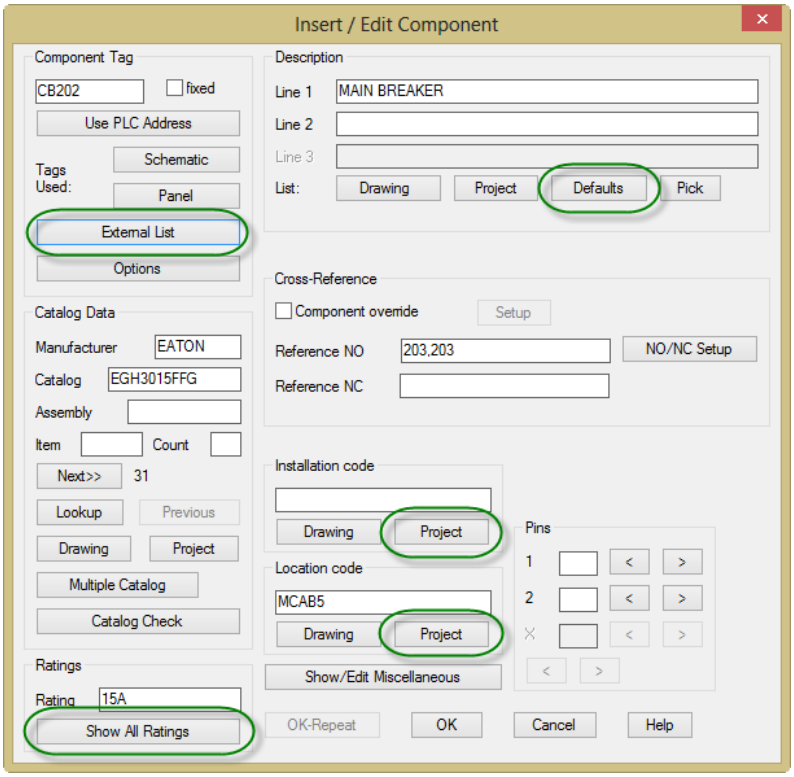
Having covered the databases, the first set of ASCII text files we will discuss are the ones that assist us with our project information. The two files that do this are the .wdl and the .wdt files. A .wdl file is a Project Line Label file. It changes the generic LINE1, LINE2, etc. labels to something we can relate to in the Project Descriptions and Title Block Setup dialogs. The .wdt file is one of two ways to define our Title Block Update mappings. A .wdt file provides maximum flexibility in defining our title block mapping, because it is an external file and can store the mappings for multiple blocks. It is very useful when we use an unlimited variety of title blocks in our project drawing sets. Placing both of these files in our …\AeData\Support folder allows them to be referenced by every project unless there is a copy in the project folder.

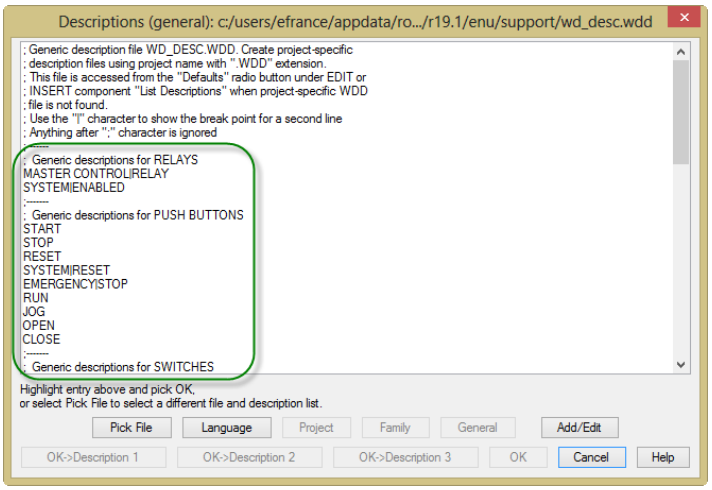
Standard Component Content Files

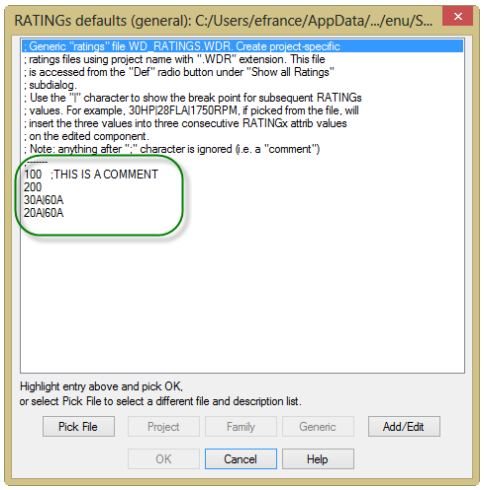
The next set of ASCII text files is a big one. These ten reference files can all be used in the standardization of content that gets applied to or is used to describe components of our electrical designs. Many of them are accessible from the Insert/Edit dialog box or a similar dialog for non-component entities. The table that follows outlines the files of this set.

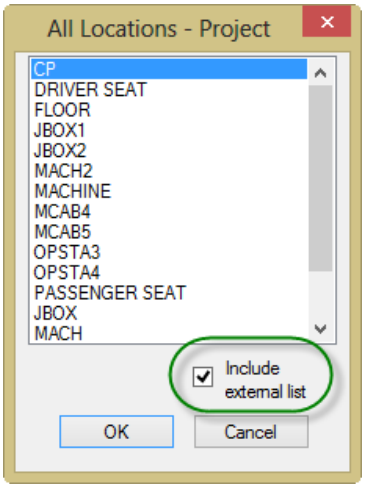
Content File Description	Filename	Access From…
Default Descriptions	wd_desc.wdd	"Defaults" button on Insert / Edit Component dialog
Default Ratings	wd_ratings.wdr	From "Show All Ratings" button > "Defaults" button in View/Edit Rating dialog
Default Location Codes	default.loc	From Location code "Project" button, check the "Include external list" checkbox
Default Installation Codes	default.inst	From Installation code "Project" button, check the "Include external list" checkbox
Default Component Tags	default.wdx	From Component Tag > "External List" button
Source/Destination Arrow Default Descriptions	wdsrcdst.wdd	"Defaults" button in Source or Destination Arrow dialogs
Generic Wire Color/ID Abbreviations	cblcolor.dat	"Generic" button on Insert / Edit Cable Marker dialog
Default Wire Color/Gauge Labels	default.wdw	"Setup" button on Insert Wire Color/Gauge Labels dialog
Default Terminal Number Audit Filter	default.wdn	Lists Terminal Numbers to be ignored in an Electrical Audit
User-Defined Attribute List	default.wda	Used by report tools to determine which additional attributes are listed in reports

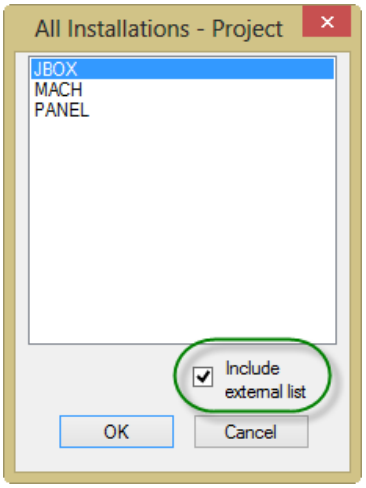
As mentioned before, most of these files contain sample or “filler” information that should be edited to contain the required content for our specific needs. Regardless of the ones we determine are the most useful for our needs, all of these files can help us use consistent information throughout the sheets of a project and from project to project. Placing all of these files in our …\AeData\Support folder allows them to be referenced by every project unless there is a copy of a specific file included in a particular project’s folder. A sample of screen captures identifying how to access the different reference files is displayed over the next couple of pages.

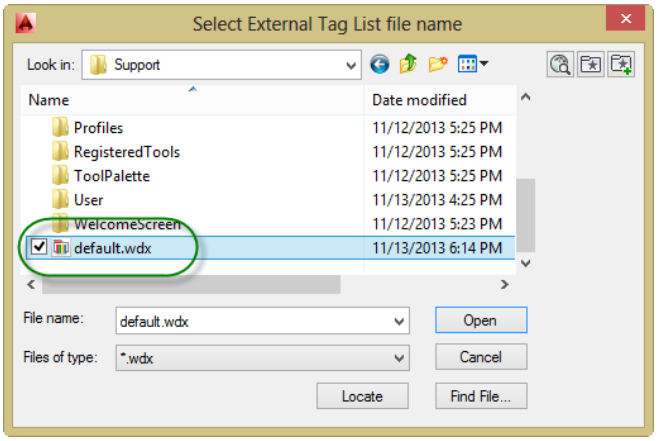


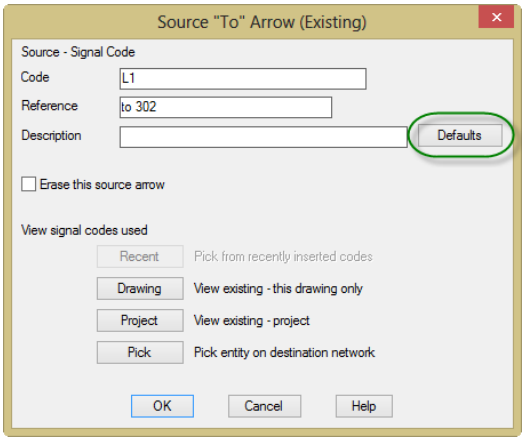


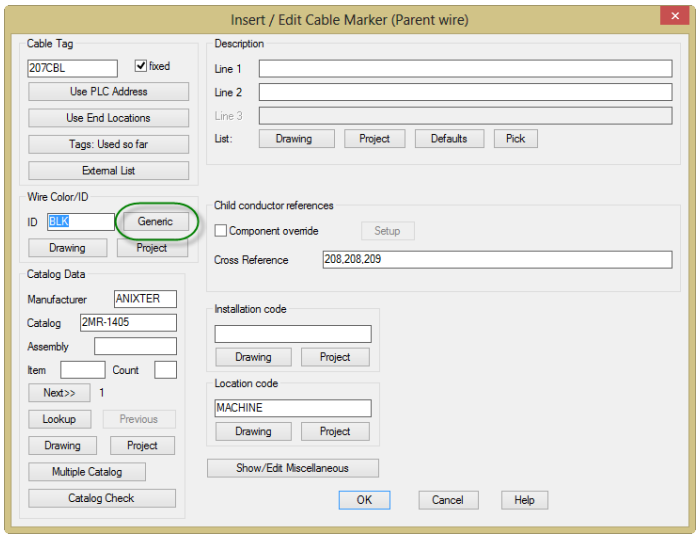


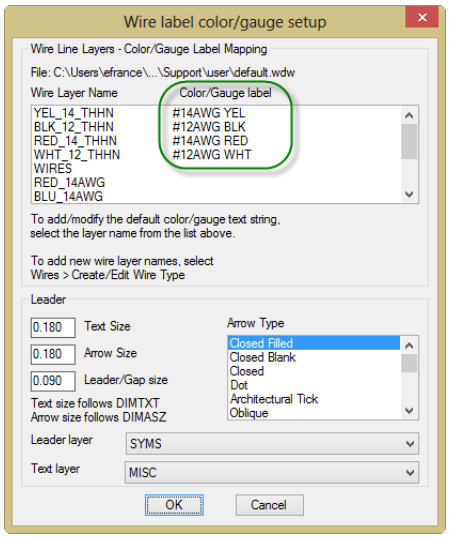

All Locations - Project
CP
DRIVER SEAT
FLOOR
JBOX1
JBOX2
MACH2
MACHINE
MCAB4
MCAB5
OPSTA3
OPSTA4
PASSENGER SEAT
JBOX
MACH
Include external list
OK
Cancel


All Installations - Project
JBOX
MACH
PANEL
Include external list
OK
Cancel


Select External Tag List file name
Look in: Support
Name
Date modified
Profiles 11/12/2013 5:25 PM
RegisteredTools 11/12/2013 5:25 PM
ToolPalette 11/12/2013 5:25 PM
User 11/13/2013 4:25 PM
WelcomeScreen 11/12/2013 5:23 PM
default.wdx 11/13/2013 6:14 PM
File name: default.wdx
Files of type: *.wdx
Open
Cancel
Locate
Find File...

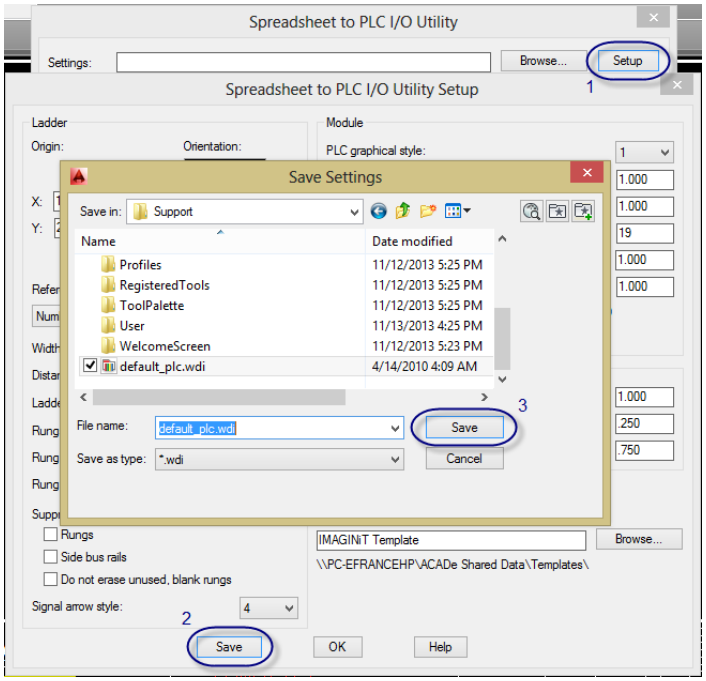

Source "To" Arrow (Existing)
Source - Signal Code
Code L1
Reference to 302
Description
Defaults
Erase this source arrow
View signal codes used
Recent Pick from recently inserted codes
Drawing View existing - this drawing only
Project View existing - project
Pick Pick entity on destination network
OK
Cancel
Help


Insert / Edit Cable Marker (Parent wire)
Cable Tag
207CBL
fixed
Use PLC Address
Use End Locations
Tags: Used so far
External List
Wire Color/ID
ID BLK
Generic
Drawing
Project
Catalog Data
Manufacturer ANIXTER
Catalog 2MR-1405
Assembly
Item
Count
Next>>
Lookup
Previous
Drawing
Project
Multiple Catalog
Catalog Check
Description
Line 1
Line 2
Line 3
List:
Drawing
Project
Defaults
Pick
Child conductor references
Component override
Setup
Cross Reference 208,208,209
Installation code
Drawing
Project
Location code
MACHINE
Drawing
Project
Show/Edit Miscellaneous
OK
Cancel
Help


Wire label color/gauge setup
Wire Line Layers - Color/Gauge Label Mapping
File: C:\Users\efrance\...\Support\user\default.wdw
Wire Layer Name
Color/Gauge label
YEL_14_THHN #14AWG YEL
BLK_12_THHN #12AWG BLK
RED_14_THHN #14AWG RED
WHT_12_THHN #12AWG WHT
WIRES
RED_14AWG
BLU_14AWG
To add/modify the default color/gauge text string, select the layer name from the list above.
To add new wire layer names, select Wires > Create/Edit Wire Type
Leader
0.180 Text Size
0.180 Arrow Size
0.090 Leader/Gap size
Text size follows DIMTXT
Arrow size follows DIMASZ
Arrow Type
Closed Filled
Closed Blank
Closed
Dot
Architectural Tick
Oblique
Leader layer SYMS
Text layer MISC
OK
Cancel

Standard Settings Files

Our last set of ASCII text reference files are our standard settings files. The two files in this group are the .wdi and the .wdf files. A .wdi file is a settings file for use with the PLC I/O Utility. It stores the setup and column mapping information for when we generate PLC drawings from a spreadsheet. The .wdf file defines the RSLogix mappings for an RSLogix file import. Placing both of these files in our …\AeData\Support folder allows them to be referenced whenever we perform these specific operations.



Class Summary

In this class, you learned how to:

- Share common libraries, databases, and other reference files effectively
- Configure the environment file to access shared locations first
- Organize your shared locations for convenient access and maintenance
- Use standard reference files to share data among users and between projects

Implementing AutoCAD® Electrical effectively can significantly increase the benefits that the software provides. From deploying the software correctly through setting up a shared symbol library to utilizing a multitude of reference files; a properly implemented AutoCAD® Electrical allows users to consistently follow established standards while working more productively.

Thank you for your time and attention.