AUTODESK UNIVERSITY 2014

AutoCAD Electrical – Advanced Productivity

Scott Dibben – D3 Technologies
Thomas Smith – D3 Technologies

MD6865

Now that we have been using AutoCAD Electrical in industry for a while, let's see what we can do to squeeze more productivity out of the software. The ability to uncover tools that are not being used and identifying different workflows can help to improve efficiencies in our designs. This class covers unique ways to use the software's existing tools to become more productive.

Learning Objectives

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

- Creating nested projects
- Advanced block creation
- Using peer relationships in a unique way
- Customizing the environment file

About the Speaker

Technical Manager for D3 Technologies, Scott has taught and implemented AutoCAD Electrical in a variety of industries. 13 years of industry experience as a Senior Controls Design Engineer on a global scale. Interviewed and published in Controls Design Magazine on "Who's Responsible for Machine Safety", Scott has been considered an expert on machine design safety for North America and European Union Countries.

Introduction

Most Designers/Engineers utilize AutoCAD Electrical (ACADE) in different ways in order to get their desired output. This class will focus on four areas of interest in ACADE that will assist users in getting the most out of their software.

I have had the pleasure of helping several electrical design departments setup and implement their software. While each implementation has its challenges, the versatility of ACADE proves to help the transition.

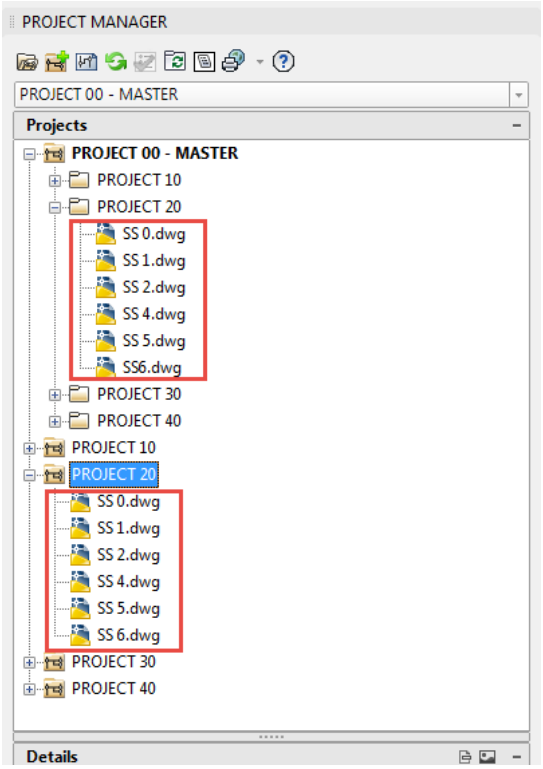
Nested Projects

What are Nested Projects?

This is a term used to identify a workflow for subdividing large design work into small projects that also have a top level or master project that contains the complete design.

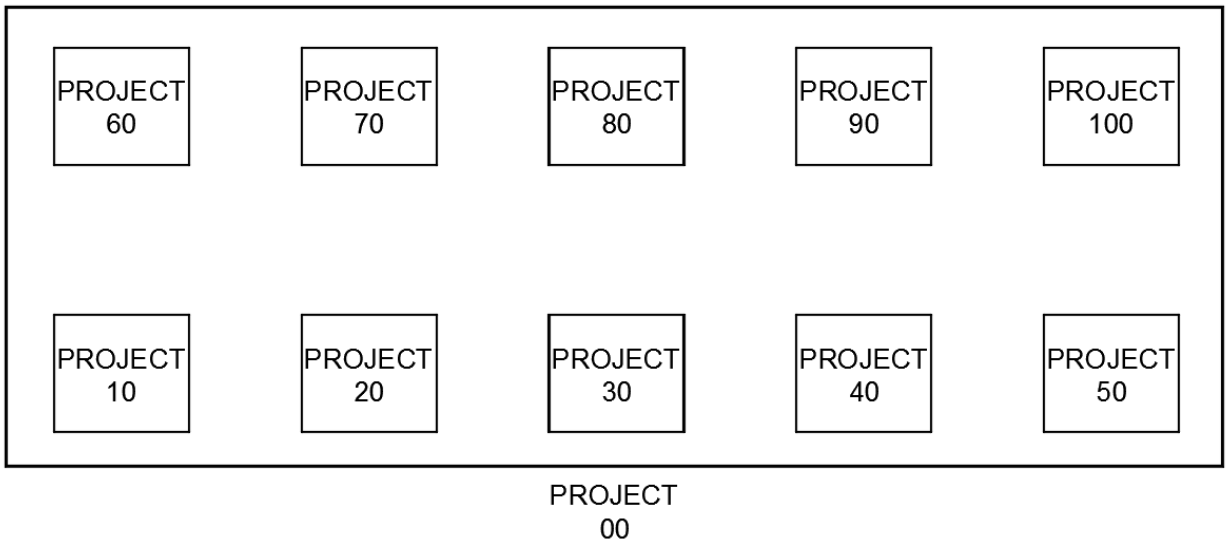
Why use Nested Projects?

In large designs it is common for more than one designer/engineer to be working to complete a project. Nested projects help to separate work into small more manageable designs.



While AutoCAD Electrical does not have the ability to have sub projects, we can still find unique ways to work around this.

The follow method will be discussed in class.



- Project 10 – 100 are setup as individual projects and worked on separately
- Project 00 (integration project) is setup with all of the drawings from Project 10 – 100 included. Power distribution is assigned in this project.

Advantages

- Allows for Lead Designer/Engineer to verify all power distributions
- Allows for design approvals to be completed in small sections
- Modular design can be easier to reuse for future designs
- Can allow for more competitive bidding on projects.
- Updating can occur at sub-project level, or main project level
- Purchasing can begin sooner

Disadvantages

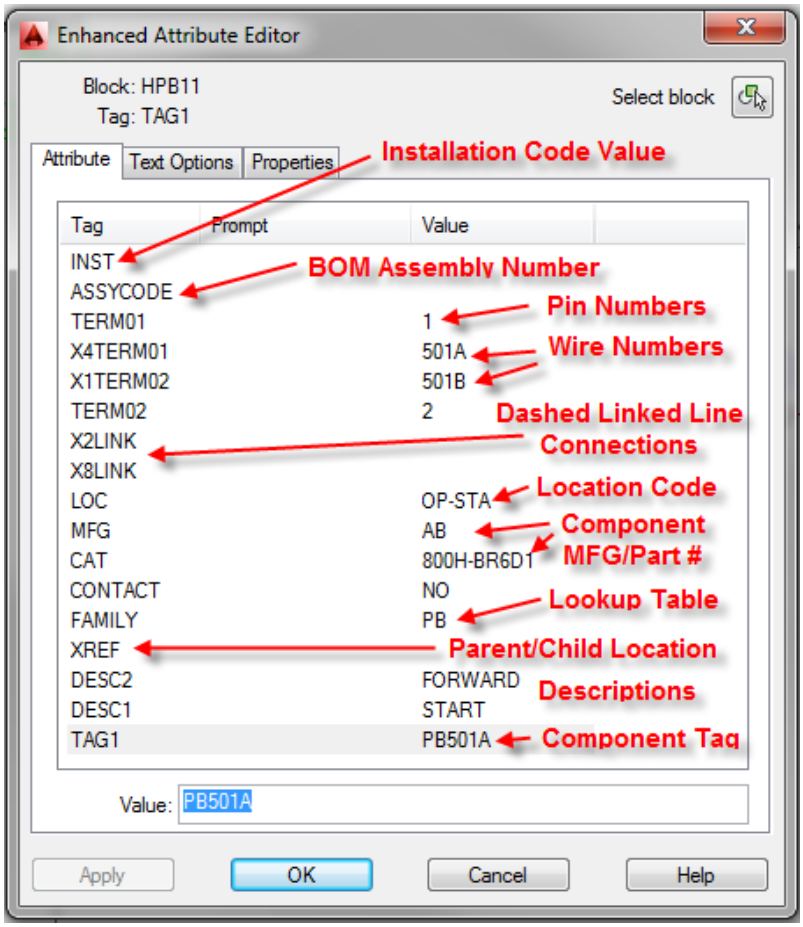
- Must have a clear vision for design layout
- Lead Designer may fill disassociated with projects

Advanced Block Creation

AutoCAD blocks are the foundation of the ACADE package. When we insert a component such as a pushbutton (HPB11) the block being used is intelligent. A block as simple as the HPB11 (NO pushbutton) holds data in attributes that are crucial for such things as reporting, children/parent location, and whether it is part of a BOM assembly.

When we create new symbols we need to understand what each attributes purpose is and if it is required for the symbol we are building.

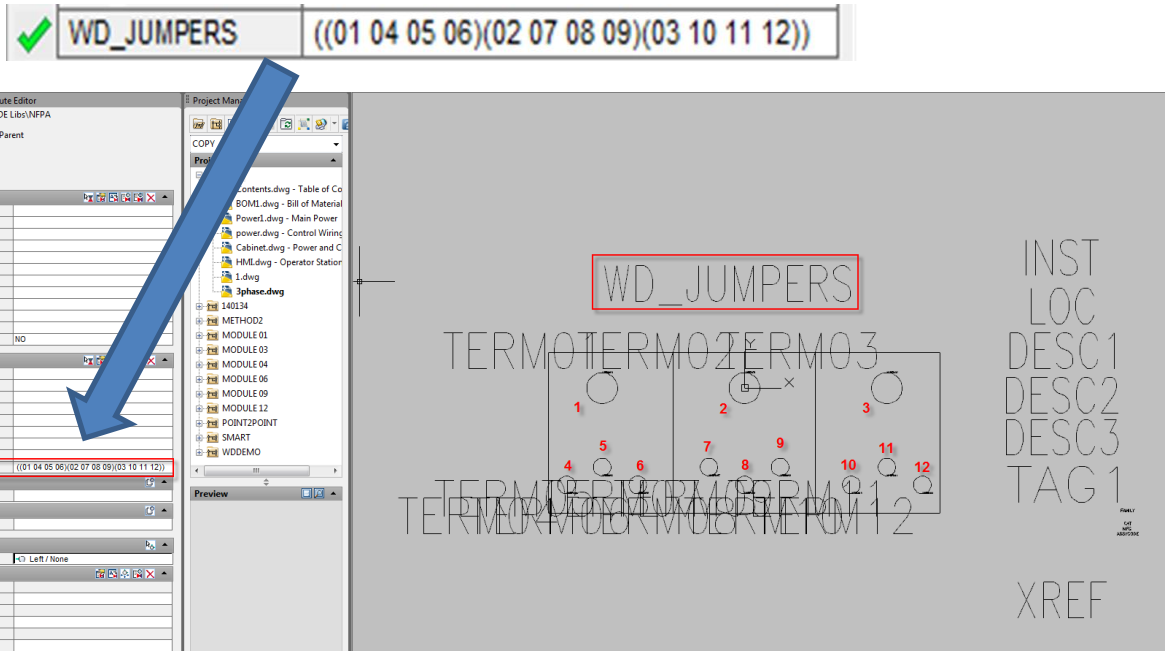
Along with the required attributes, there are optional attributes that can give advanced functionality to your blocks. These will be discussed later in the handout.

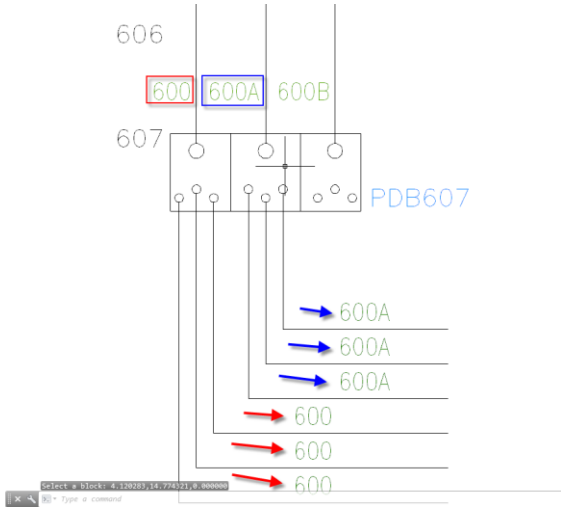


Schematic symbol that passes wire numbers

WD_Jumper Attribute is used to tie wire connections together. When tied together properly the wire number does not change as it passes through the connection point. In industry this attribute is used on such symbols as terminals and power distribution blocks.

- WD_Jumper Attribute
- Format: (01 04 05 06)

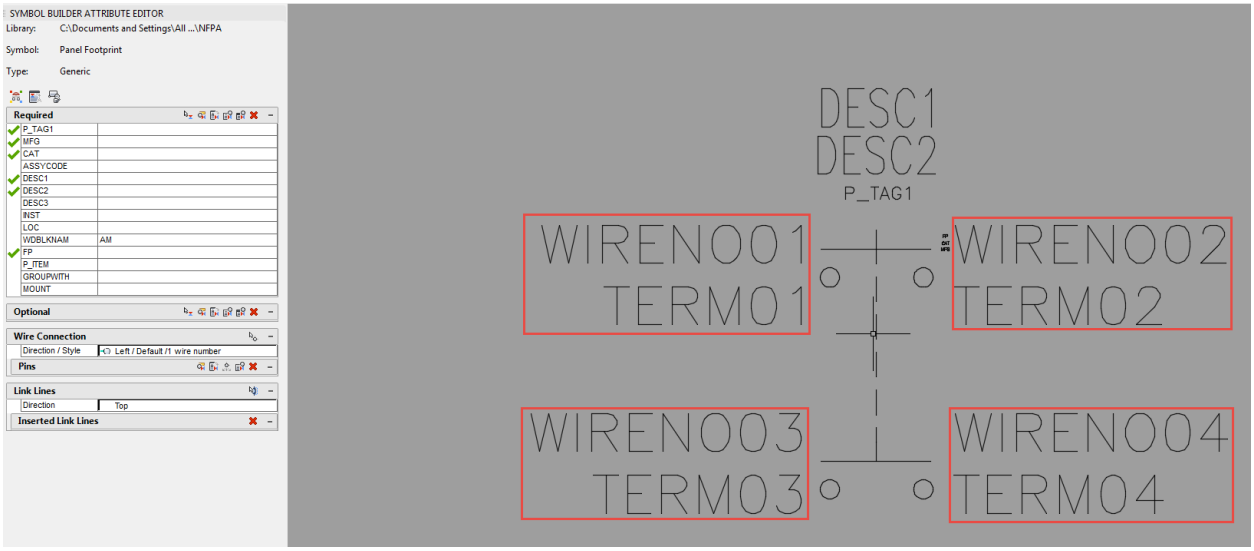




Footprint with Terminal and Wire Data

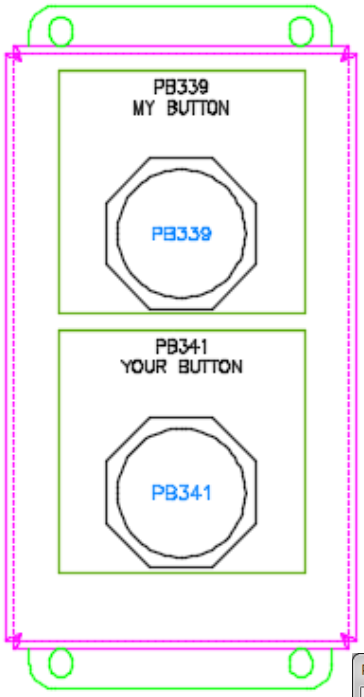
Some companies I work with require certain footprint blocks to show wiring data. While AutoCAD Electrical will give you wire data in an mtext format, we can get better results from simply building the symbol with the required attributes.

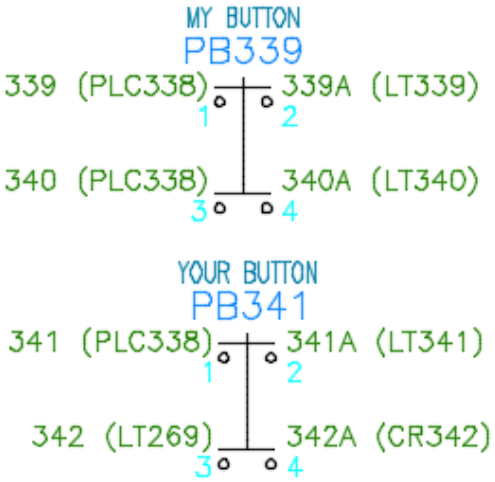
- WIRENOXX (Each Wire Number)
- TERMXX (Each Terminal Point)

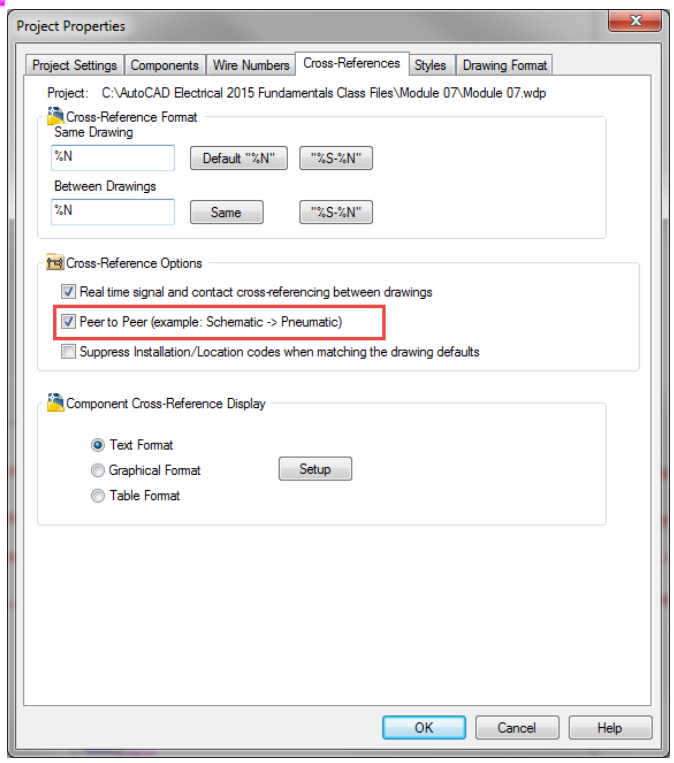


Using Peer relationships in unique ways

Peer relationships can be used for more than just linking pneumatic symbols to electrical symbols. This type of relationship can be used for creating two footprints linked to one schematic part. In the example below we have both a footprint showing wire data along with the typical front view of a the same component.





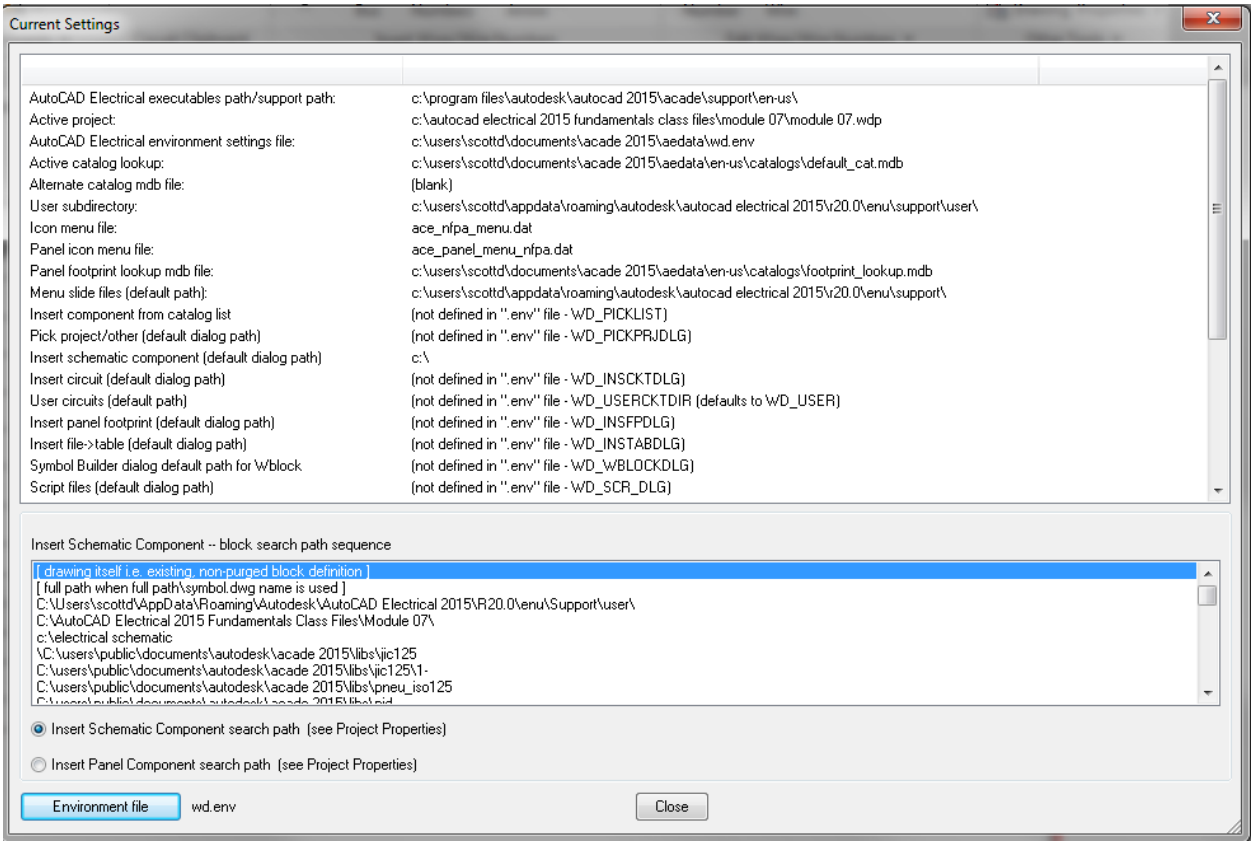


Customizing the environment file

The environment file controls things such as:

- Default Library Paths
- Database Paths
- Default Location for Save Project
- Default Symbol Builder Save Location
- Save Location for Save Circuits to Icon Menu

Customizing the environment file can lead to great efficiency gains in AutoCAD Electrical. Below are a few examples of items covered in the class

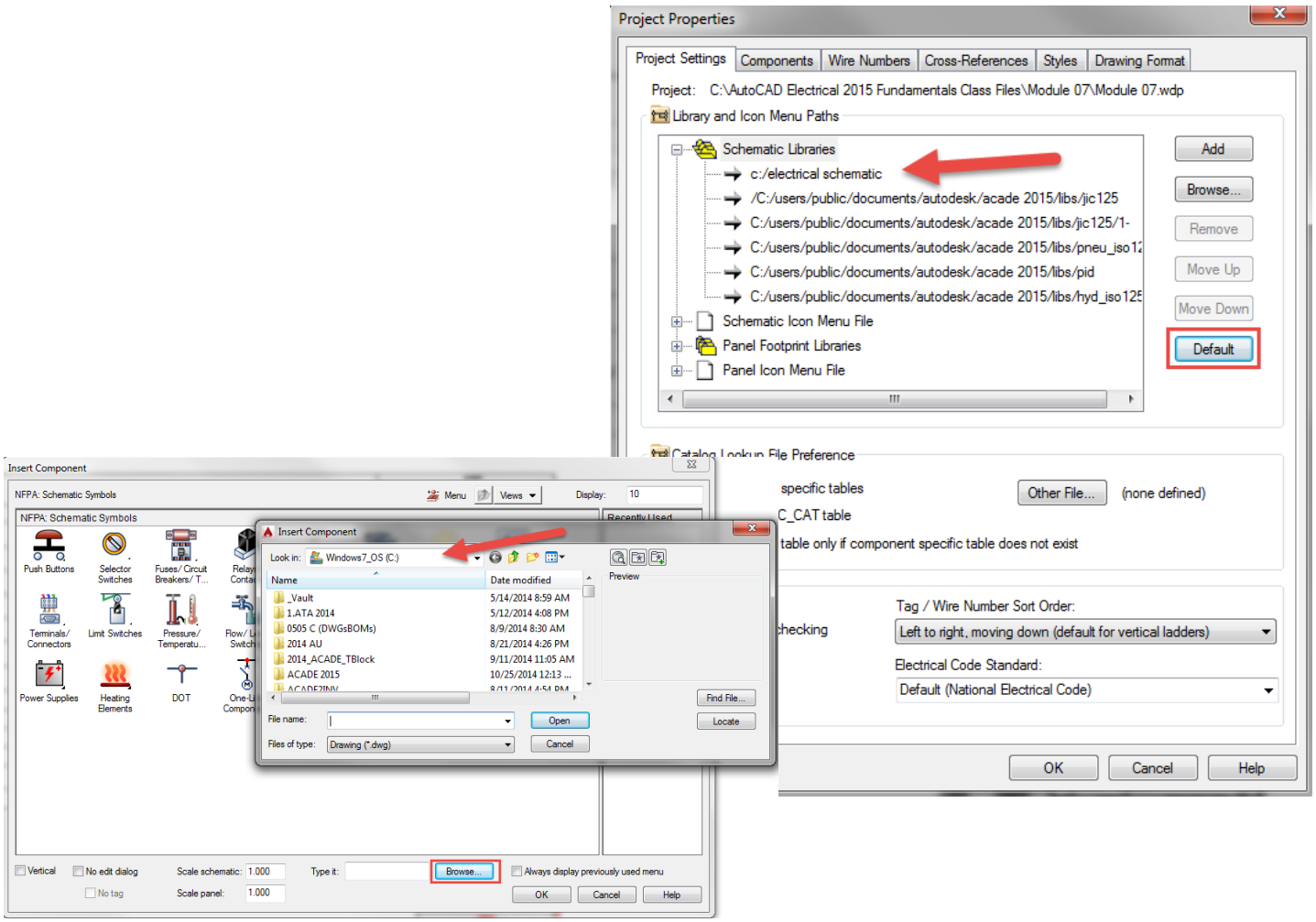


Default Library Path – Default Insert Component

WD_LIB line identifies the paths included in the project when selecting the Default button. This can be setup for both the Schematic Libraries and Panel Footprint Libraries.

WD_INSCOMPDLG & WD_INSFPDLG lines identify the default path when selecting browse from the Schematic or Footprint icon men

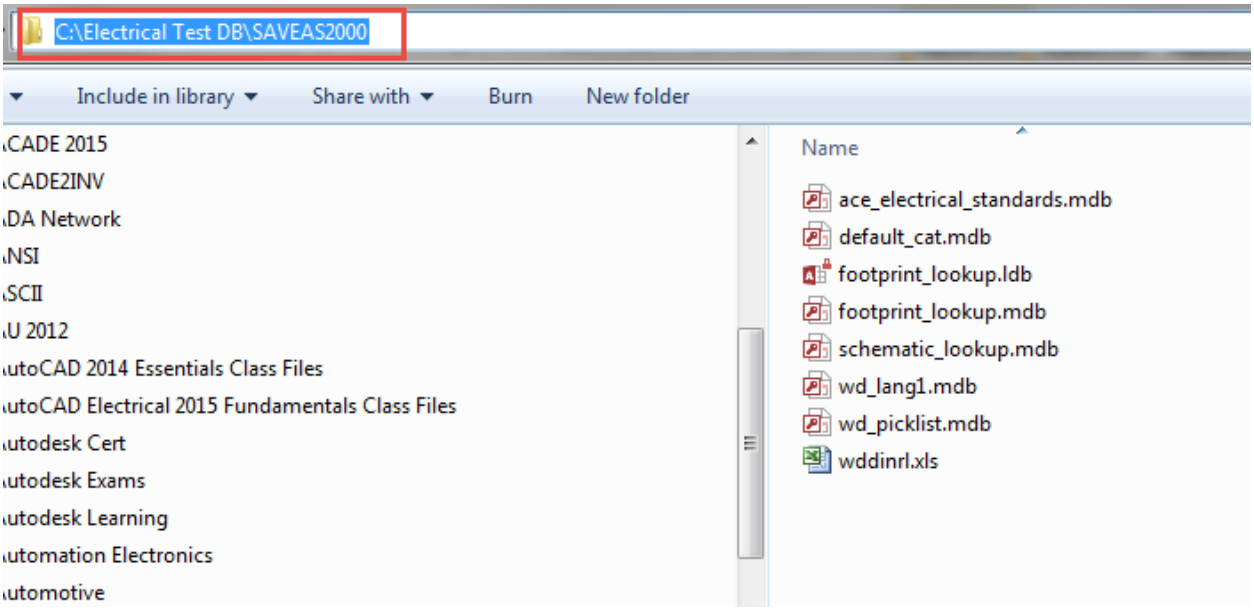
```
*************************************************************************************
* Symbol Insertion and Library Paths
*************************************************************************************
WD_LIB,C:/Electrical Schematic;\%SL_DIR%/jic125;%SL_DIR%/jic125/1-;%SL_DIR%/pneu_iso125;%SL_DIR%/pid;%SL_DIR%/hyd_iso125, schem, 1-l
WD_PNL,%SL_DIR%/panel, panel layout symbol library base folder
*WD_CIRCBUILDER_FNAM,"ace_circuit_builder.xls",Circuit Builder spreadsheet file name
WD_INSCOMPDLG,C:/,to override starting path for INS SCHEM COMP browse button
*WD_INSCOMPDLG,X:/some path/,to override starting path for INS SCHEM COMP browse button
*WD_INSFPDLG,x:/some path/,to override starting path for INS PNL COMP browse button
*WD_INSCKTDLG,x:/some path/,to override starting path for INS CKT file selection dialog
*WD_WBLOCKDLG,x:/some path/,to override starting subdir for Black Box Bldr WBlock option
*WD_PICKLIST,%WD_DIR%/%WD_LANG%/catalogs/wd_picklist.mdb,ins component from catalog list file
*WD_USERCKTDIR,x:/some path/,default folder for "USER CKTS"
*PLC_ADDRESS_FORMAT,I:00%10%2;I:%1%2/00;--;O:00%10%2;O:%1%2/00;--;,default PLC address formats for "Quick pick"
```



Catalog Database Location

WD_CAT line identifies the path to the electrical databases. The path should be pointed to the folder where the databases exist. This can be on a network drive.

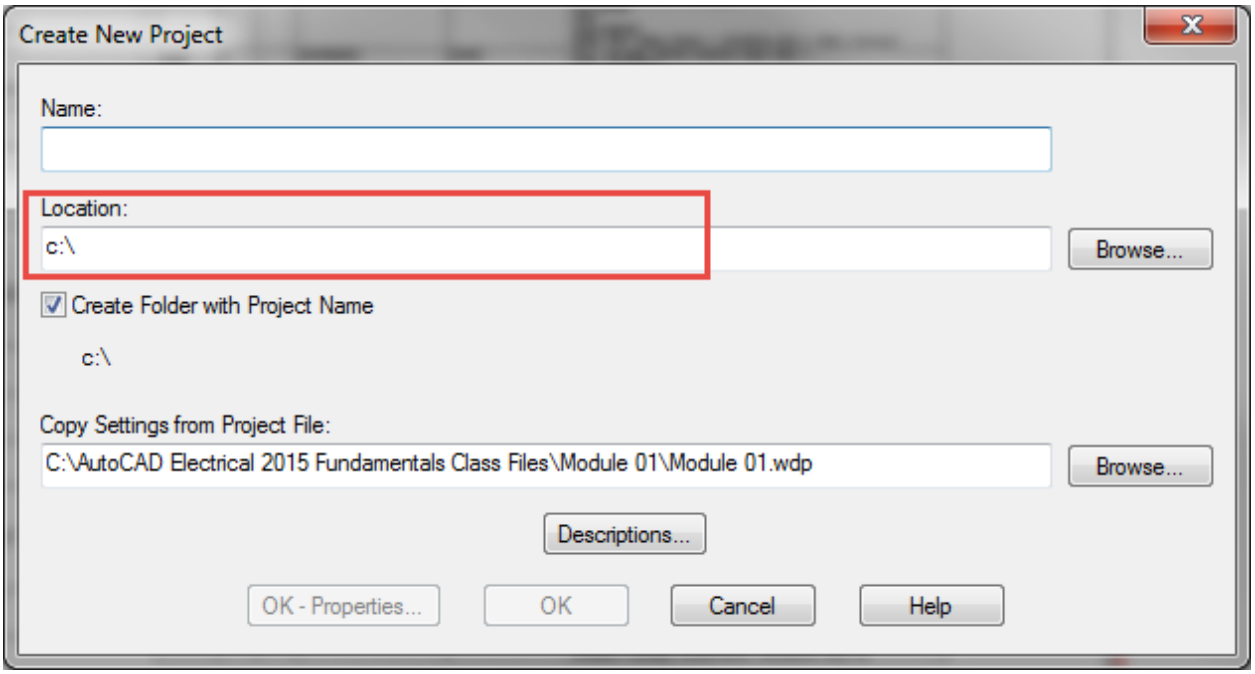
```
* Catalog Parts Database Path
*****************************************************************************************
*WD_CAT,C:/Electrical Test DB/SAVEAS2000
WD_CAT,%WD_DIR%/%WD_LANG%/catalogs/,AE catalog file path
*WD_XCAT,x:/some path/wd_xcat.lsp,to override catalog look-up and call user's external routine

*****************************************************************************************
```



New Project Path

WD_PROJ line identifies the default path a new project would be saved to. This can be on a network drive

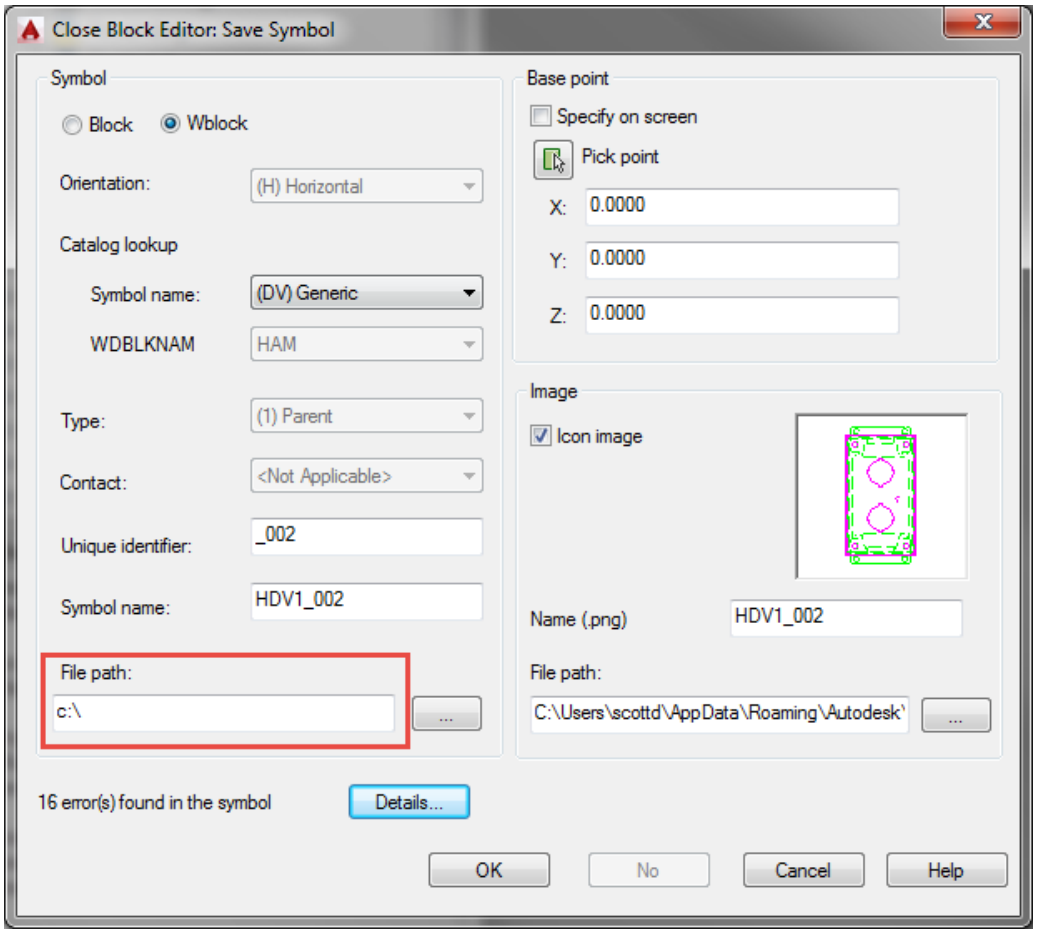
```
;*****************************************************************************
; Project and drawing-related paths and settings
;*****************************************************************************
WD_PROJ,C:/,AutoCAD Electrical default project data folder
*WD_PROJ,%WD_DIR%/proj/,AutoCAD Electrical default project data folder
*WD_PICKPRJDLG,x:/some_path/,to override starting subdir for "PICK PROJ" button
*WD_OPEN_DWG,(command "_.ZOOM" "_E"),AutoLISP prog or expression to autoexecute when dwg opened from PROJ dlg
*WD_ZIP,c:/program files/winzip/winzip32.exe,full path/name of zipping utility executable
*WD_ZIP,c:/program files/WinRAR/rar.exe, full path/name for alternate zipping utility
*WD_PROMPT_SAVE,1,set to 1 to trigger "Save changes" prompt when using open dwg next/prev arrows buttons
```



Symbol Builder Default Save Location

WD_WBLOCKDLG line identifies the default path a new block is saved to. This can be on a network drive

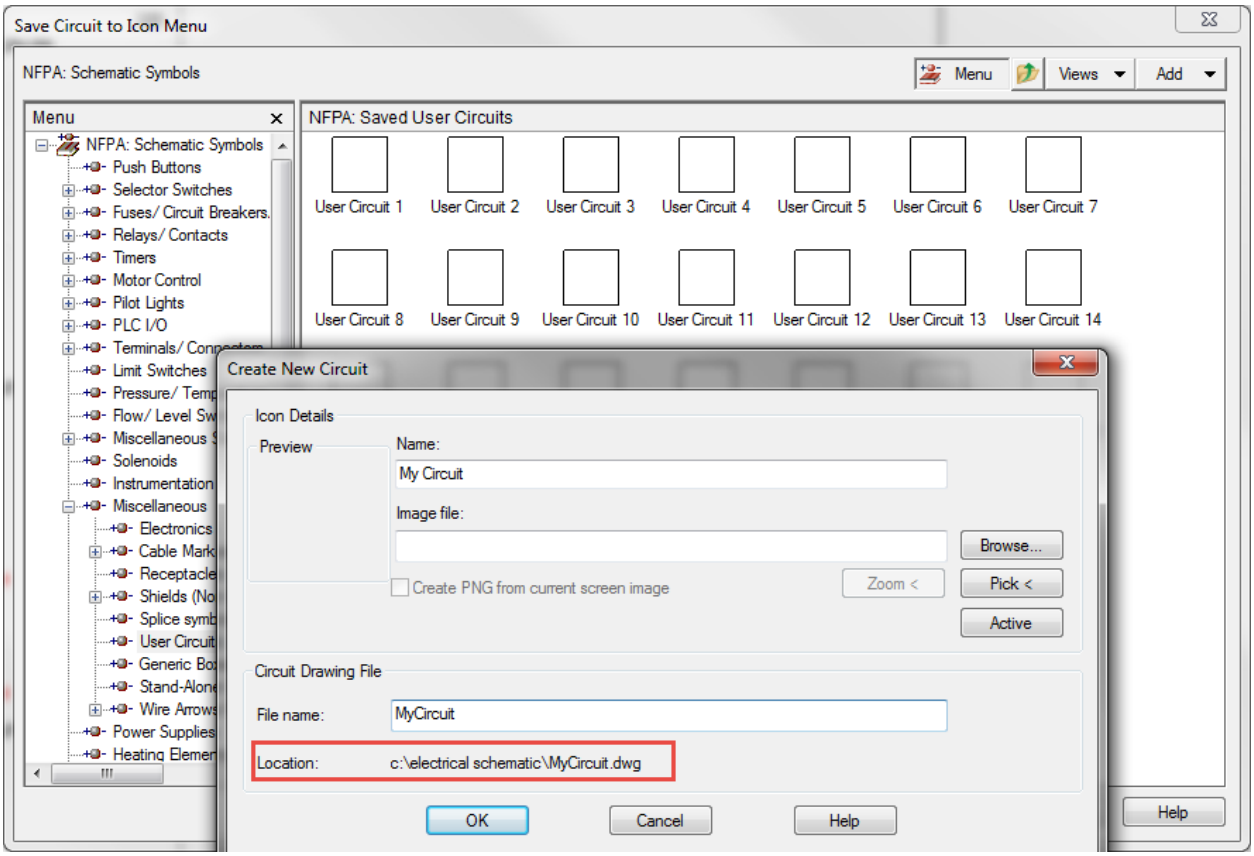
```
*************************************************************************************
* Symbol Insertion and Library Paths
*************************************************************************************
WD_LIB,C:/Electrical Schematic;\%SL_DIR%/jic125;%SL_DIR%/jic125/1-;%SL_DIR%/pneu_iso125;%SL_DIR%/pid;%SL_DIR%/h
WD_PNL,%SL_DIR%/panel, panel layout symbol library base folder
*WD_CIRCBUILDER_FNAM,"ace_circuit_builder.xls",Circuit Builder spreadsheet file name
WD_INSCOMPDLG,C:/,to override starting path for INS SCHEM COMP browse button
*WD_INSCOMPDLG,x:/some path/,to override starting path for INS SCHEM COMP browse button
*WD_INSFPDLG,x:/some path/,to override starting path for INS PNL COMP browse button
*WD_INSCKTDLG,x:/some path/,to override starting path for INS CKT file selection dialog
WD_WBLOCKDLG,c:/,to override starting subdir for Black Box Bldr WBlock option
*WD_WBLOCKDLG,x:/some path/,to override starting subdir for Black Box Bldr WBlock option
*WD_PICKLIST,%WD_DIR%/%WD_LANG%/catalogs/wd_picklist.mdb,ins component from catalog list file
*WD_USERCKTDIR,x:/some path/,default folder for "USER CKTS"
*PLC_ADDRESS_FORMAT,I:00%10%2;I:%1%2/00;--;O:00%10%2;O:%1%2/00;--;,default PLC address formats for "Quick pick"
```



Saved Circuit to Icon Menu Default Save Location

WD_WBLOCKDLG line identifies the default path a new block is saved to. This can be on a network drive

```
*************************************************************************************
* Symbol Insertion and Library Paths
*************************************************************************************
WD_LIB,C:/Electrical Schematic;\%SL_DIR%/jic125;%SL_DIR%/jic125/1-;%SL_DIR%/pneu_iso125;%SL_DIR%/pid;%SL_DIR%/hy
WD_PNL,%SL_DIR%/panel, panel layout symbol library base folder
*WD_CIRCBUILDER_FNAM,"ace_circuit_builder.xls",Circuit Builder spreadsheet file name
WD_INSCOMPDLG,C:/,to override starting path for INS SCHEM COMP browse button
*WD_INSCOMPDLG,x:/some path/,to override starting path for INS SCHEM COMP browse button
*WD_INSFPDLG,x:/some path/,to override starting path for INS PNL COMP browse button
*WD_INSCKTDLG,x:/some path/,to override starting path for INS CKT file selection dialog
*WD_WBLOCKDLG,x:/some path/,to override starting subdir for Black Box Bldr WBlock option
*WD_PICKLIST,%WD_DIR%/%WD_LANG%/catalogs/wd_picklist.mdb,ins component from catalog list file
WD_USERCKTDIR,C:/Electrical Schematic/,default folder for "USER CKTS"
*WD_USERCKTDIR,x:/some path/,default folder for "USER CKTS"
*PLC_ADDRESS_FORMAT,I:00%10%2;I:%1%2/00;--;O:00%10%2;O:%1%2/00;--;,default PLC address formats for "Quick pick"
```



Summary

The goals of all Electrical Engineers/Designers are to be more efficient and make the designs look more professional. Hopefully the discussed topics are found to do found to be of use in your work environment.

We can be reached at:

Scott.dibben@d3tech.net
Thomas.smith@d3tech.net