AUTODESK **UNIVERSITY** 2014

Feeling the *Spark* for AutoCAD Electrical (For AutoCAD Users)

Tiffany Tucker – TATA Technologies

MD6033 If you are currently using AutoCAD® to develop controls drawings or if you have AutoCAD® Electrical but you are still primarily using it as vanilla AutoCAD to complete your controls drawings, this is the class for you. Learn key tips and tricks to get you quickly up and running with AutoCAD Electrical, so that you can start utilizing all of its amazing automation! Hot Topics will include template development, support file step up, and user friendly automated commands. This class will be electrifying and have you feeling the spark of love for ACADE!

Learning Objectives

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

- Understand the benefits of moving to AutoCAD Electrical
- Understand the basic concepts of starting an AutoCAD Electrical Project
- Understand the basic support files that can be made company-specific
- Understand the key items for converting AutoCAD templates to ACADE templates

About the Speaker

Tiffany has been an Autodesk Consultant for Tata Technologies for the last eight years. Her primary focus is as a technical instructor/consultant for AutoCAD, AutoCAD Electrical, and Inventor. She, along with two other consultants, 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 and she has worked in many different industries gaining valuable CAD experience, including Electrical Engineering, Interior Design/Architecture, Mechanical Engineering, Software Engineering, and 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.tucker@tatatechnologies.com

TATA TECHNOLOGIES

Tiffany Tucker, ACI | Autodesk Consultant
41050 W. Eleven Mile Road | Novi, MI 48375-1302

p: +1 248.699.1988 | f: +1 248.426.8498
email: tiffany.tucker@tatatechnologies.com
website: www.tatatechnologies.com

Autodesk Certified Instructor

Table of Contents

Learning Objective 1: Benefits of Moving from AutoCAD to AutoCAD Electrical

1. **AutoCAD® Electrical** was specifically designed for electrical engineers who design industrial control systems, but it can be used for many different electrical applications.

2. It is built right into the **AutoCAD®** environment, which makes for a very easy transition, with a task based user interface of Electrical-specific commands that gives the user access to many industry specific tools that automate the electrical design process, including Electrical symbol libraries for creating schematics, layouts, and related reports faster and more accurately than doing it manually in AutoCAD software. A recent study showed up to an 80 percent increase in productivity when moving to AutoCAD Electrical from AutoCAD.[1]

3. Many different design workflows are supported in ACADE. For example, you can draw your schematics and then extract a list of all of those components to create your layouts and the associations are automatically made between the components, so you only have to edit in one spot. You could also start with a layout (perhaps to choose and order components that require a long lead time) and then extract a footprint list from the layout to design the schematics.

4. You can do ladder style schematics, or you can design schematics in a point-to-point style (direct connect), placing components in empty areas of the drawing and then connecting the components with wires, or even create drawings that are a composite of both styles.

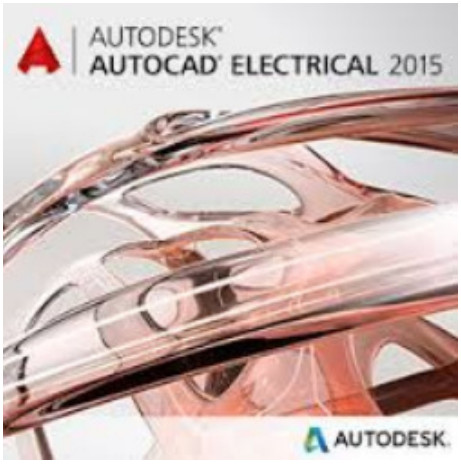
5. Creating electrical controls designs with generic software (i.e. doing everything manually) can be quite tedious, time-consuming, and can be far more prone to errors, and those errors are often not caught before the designs hit the shop floor and could cause even more delays.

6. The manual creation of reports can also cause a lot of time-consuming extra work and delays, especially with an inaccurate bill of materials (BOM).

7. Reusing designs in future projects or even just working on a project with other people in your organization can be very cumbersome, error-prone, and the use of design standards can become quite inconsistent.

Top 10 Reasons to Move from AutoCAD to AutoCAD Electrical[2]

1. Comprehensive symbol libraries
2. Automatic wire numbering and component tagging
3. Automatic project reports
4. Real-time error checking
5. Real-time coil and contact cross-referencing (Parent-Child relationships)
6. Smart panel layout drawings
7. Electrical-specific drafting features
8. Ability to automatically create PLC I/O drawings from spreadsheets
9. Ability to share drawings with customers and suppliers and track their changes
10. Reuse existing drawings to easily find and reuse designs



[1]**The AutoCAD Electrical Productivity Study compares the time required to complete 10 tasks in both basic AutoCAD and AutoCAD Electrical. The conclusion: switching to AutoCAD Electrical can help increase your productivity by as much as 80 percent. To learn more, visit www.autodesk.com/autocadelectrical-whitepapers.[1]**

[2]**Execrpt from the ACADE_JIC_Overview_broch_us.pdf[2]**

Learning Objective 2: 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.

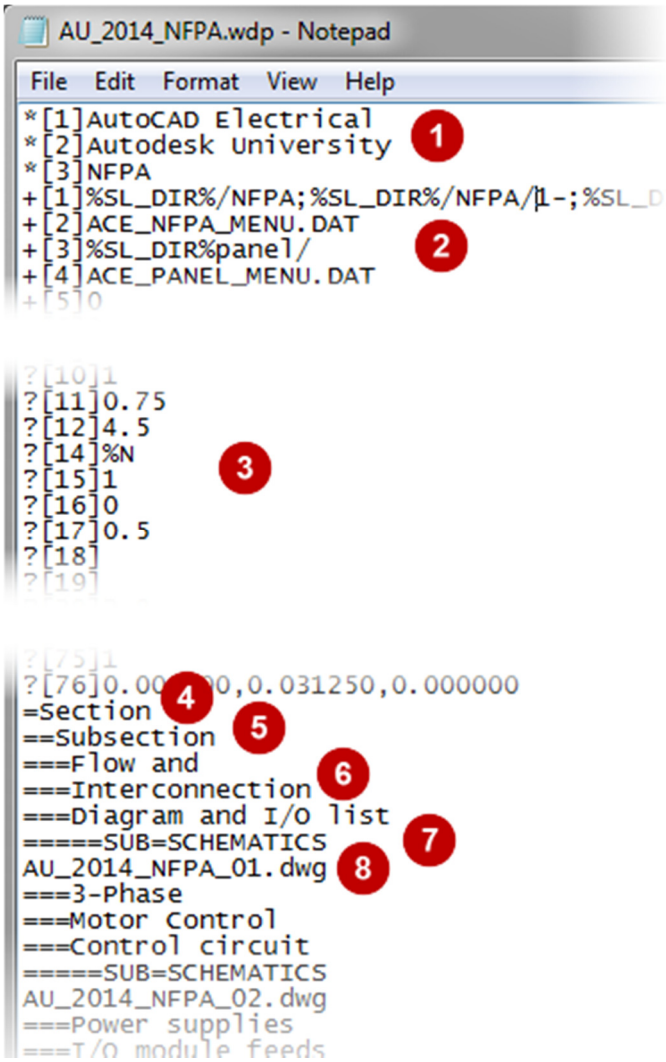
Note: Absolute or fixed paths to drawing files can also be used. To use an absolute path to a drawing file, you must manually edit the project file using any text editor. You cannot enter a fixed path using the project manager.

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:



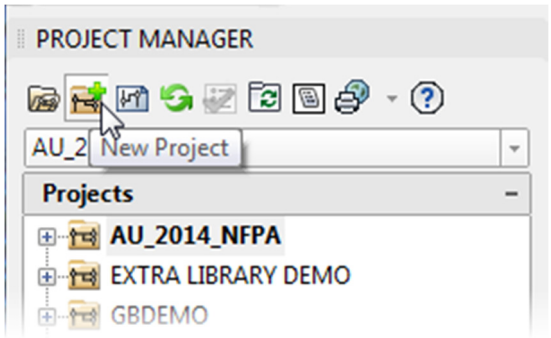
1. Lines starting with "*****[n]" are project description lines.
2. Lines starting with "**+**[n]" are project-wide settings.
3. Lines starting with "**?**[n]" are drawing default settings.
4. Lines starting with "**=**" are drawing section labels.
5. Lines starting with "**= =**" are drawing subsection 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:

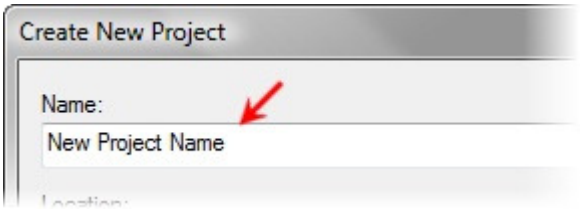
- A project file is not needed if the project consists of a single drawing.
- For more details on what is contained in a project file, go to *AutoCAD Electrical Help > Projects and Drawings*

Creating a New Project Workflow:

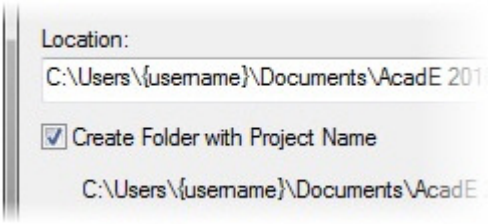
1. In the **Project Manager**, click **New Project**.



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

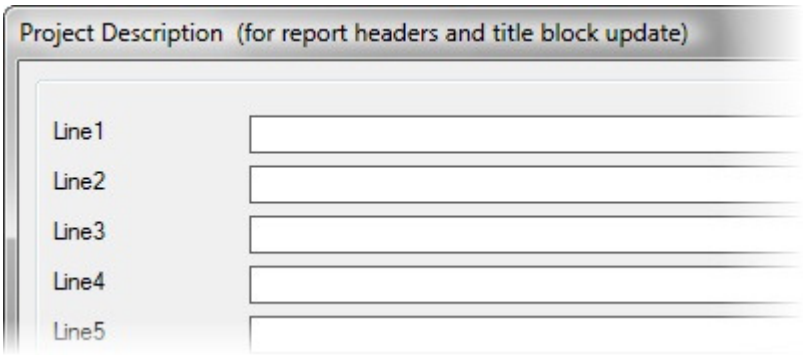


3. Select the **Create Folder with Project Name** check box to create a new folder for the project with the same name that you entered for the project. The folder is created in the path that is specified in **Location**. The location is also the path where the project file is saved. If left empty, **AutoCAD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

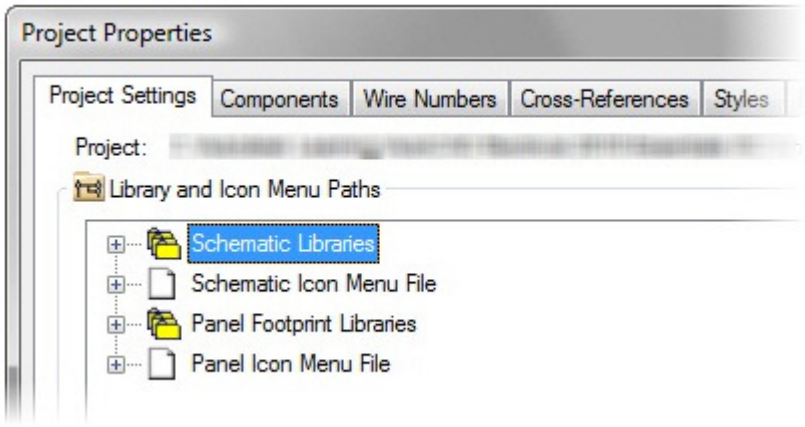


4. If you want to copy project settings from an existing project, click **Browse** to select the existing project file.

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.



7. Click **OK** to create the new project without making changes to the settings.

Learning Objective 3: 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.

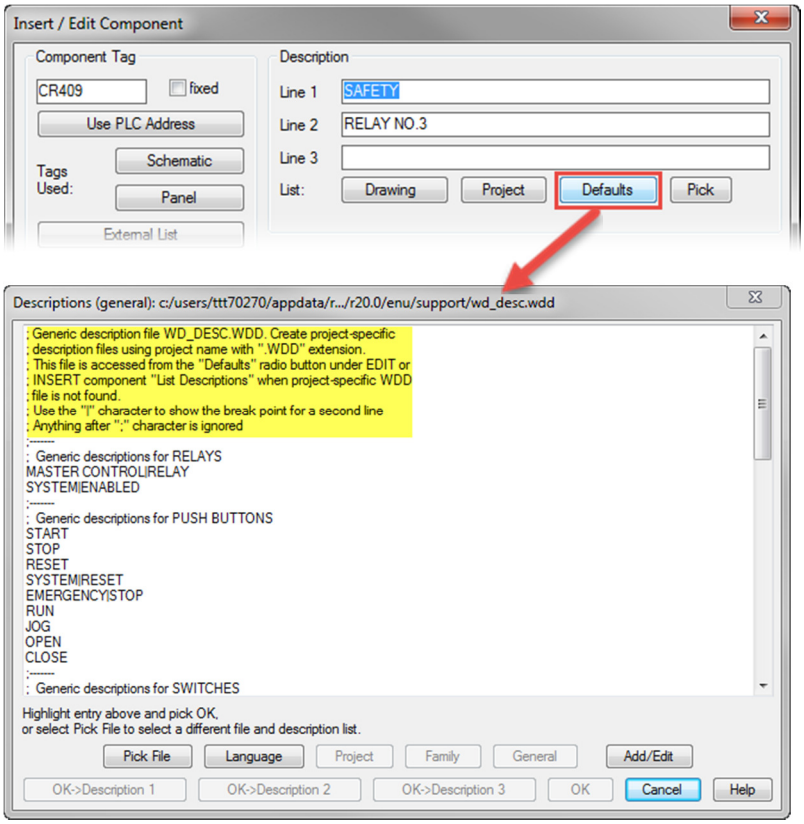
Knowledge of these files, how they are used, and how they can be made project-specific can help make tasks, such as changing drawing descriptions or mapping title block attributes, easier to understand and complete and when they are customized to your company's needs, they set the foundation for everyone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

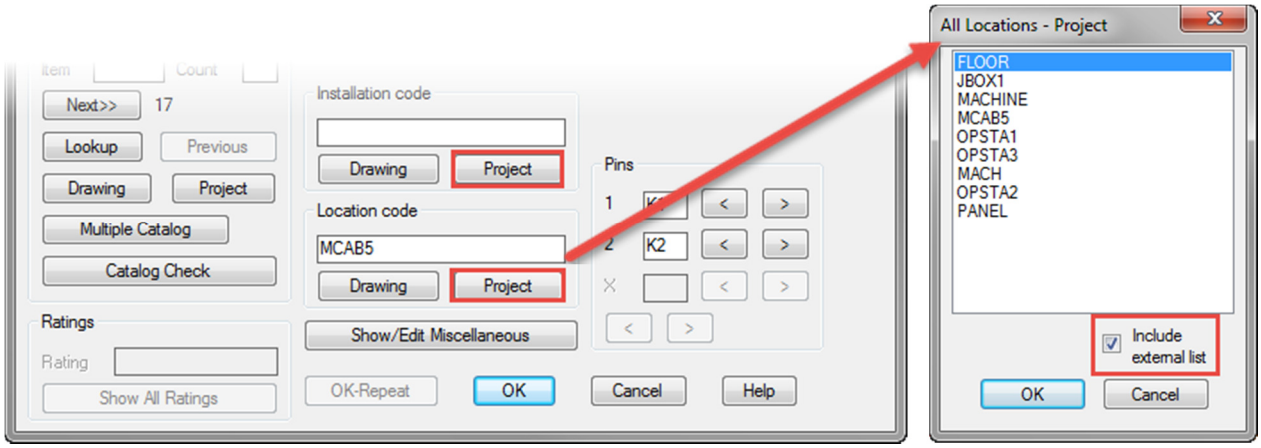
You can use wizards in the software, or any external text editor, such as Notepad, to edit these files.

- **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

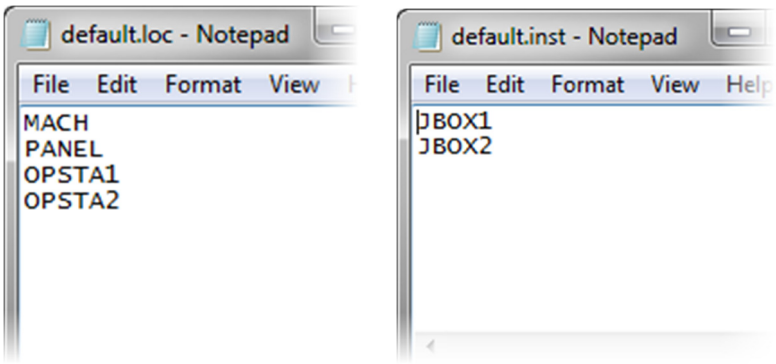


Out-of-the-box component reference file *wd_desc.wdd*

- **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

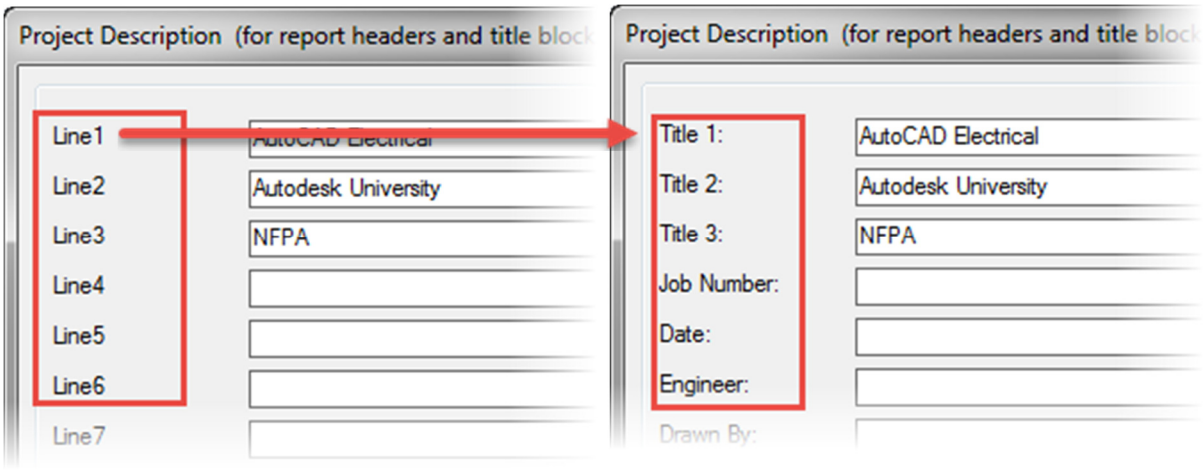
You work for a company that completes schematic designs and builds panels for many different companies. Your clients use different nomenclature, and in some cases, different languages for the component descriptions and labels.

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.

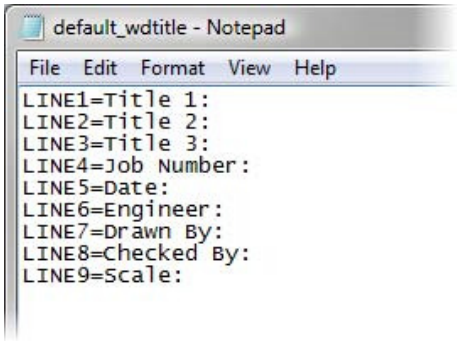
Setting Up Automated Title Block Updating

- **Define Project Line Labels** (*default_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.

Generic line labels: **Custom line labels:**

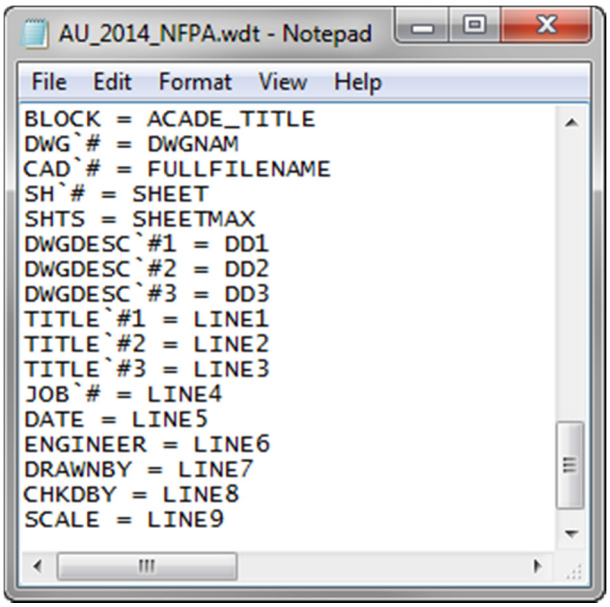


  - Either a project-based mapping file or a default mapping file can be used for this purpose. You name these files *<projectname>_wdtitle.wdl* or *default_wdtitle.wdl*, respectively. The software searches first for a file that matches the current project name. If a file is not found, the default file 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:



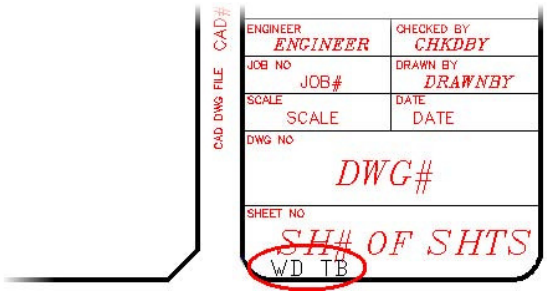
- **Title Block Mapping**
  - Before updating title blocks in your project, you must define how the project and drawing data is mapped to the matching title block attributes. To accomplish this task, you need to understand the formatting involved when using the internal attribute or the external ASCII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



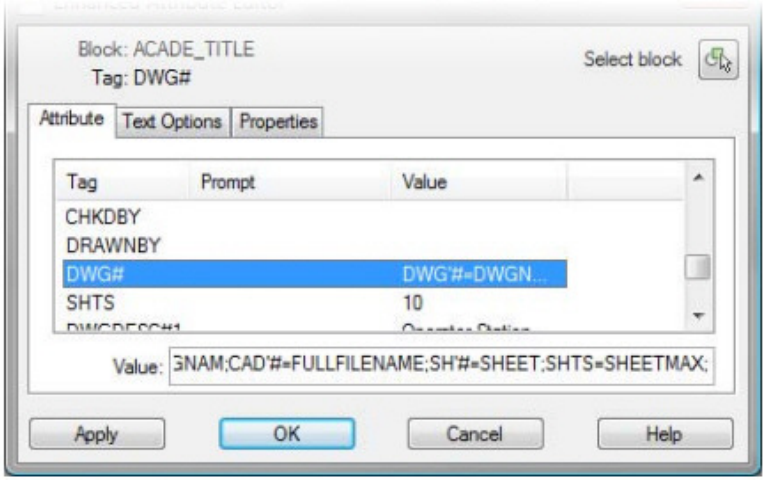
Example of external project specific WDT file

- **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.

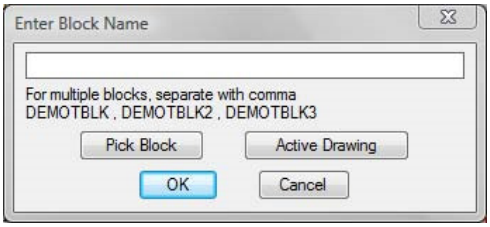


- **Title Block Setup Tool**
  - The Title Block Setup tool automates the formatting of the mapping data and makes the mapping process easier by listing the available project and drawing data as well as the available block attributes.
  - **Command Access:**

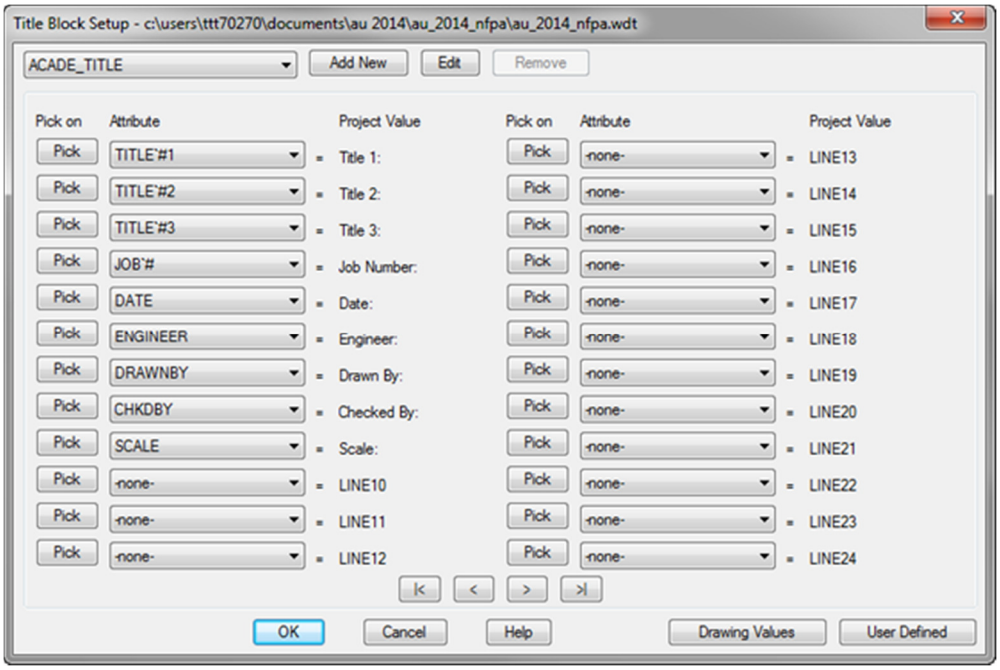
Title Block Setup

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

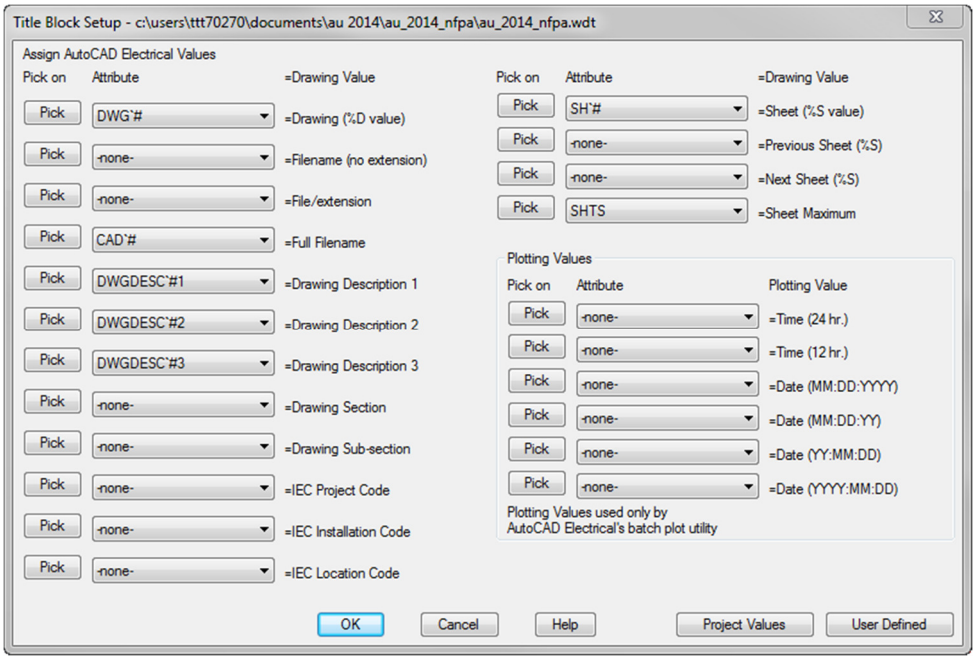
- **Enter Block Name**
  - After you select the mapping method in the **Setup Title Block Update** dialog box, the **Enter Block Name** dialog box is displayed. You use this dialog box to enter the title block names to search for attributes to map information to. You can select only a single drawing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



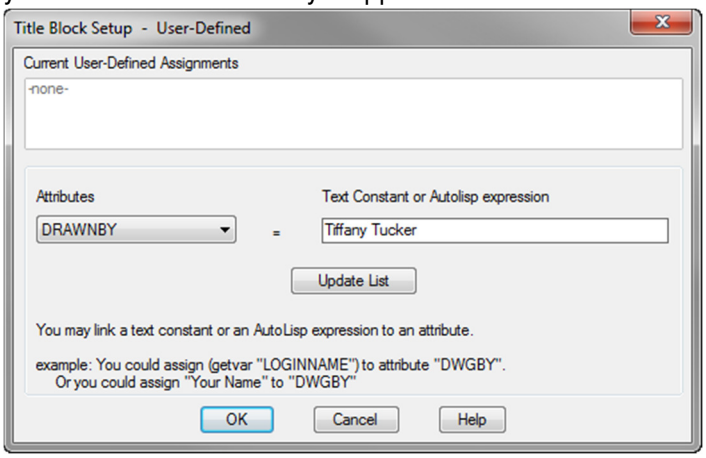
  - You use the Project Values, Drawing Values, and User Defined buttons to move between dialog boxes in the Title Block Setup tool. Each dialog box is specific to the type of data being mapped to the block attributes.
- **Project Values**
  - The **Title Block Setup - Project Values** dialog box is used 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**.

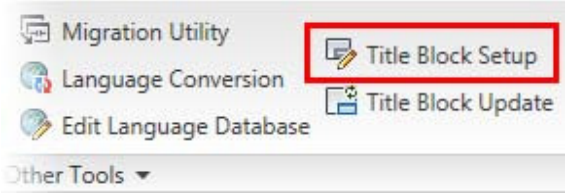


- **User-Defined Values**
  - You use the **Title Block Setup - User-Defined** dialog box to map custom information to title block attributes. You can enter a fixed value, such as your name, or you can enter LISP expressions to generate calculated values.
  - In this example, you enter your name in the **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.

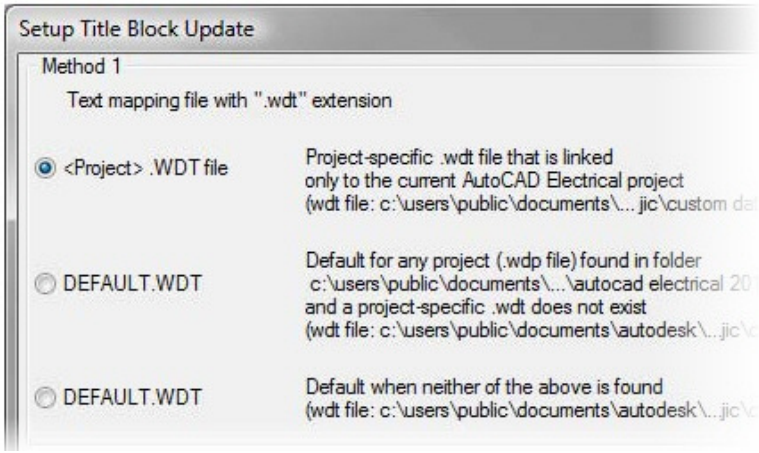


Workflow: Creating a Title Block Mapping File

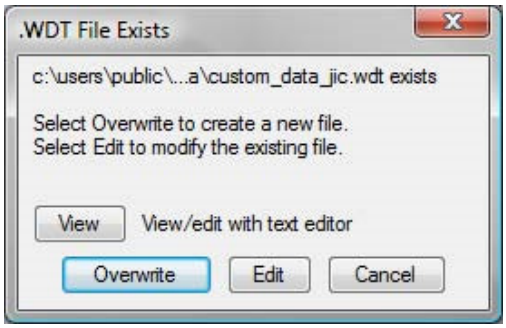
1. On the **Project** tab of the Ribbon, **Other Tools** panel, click **Title Block Setup**.



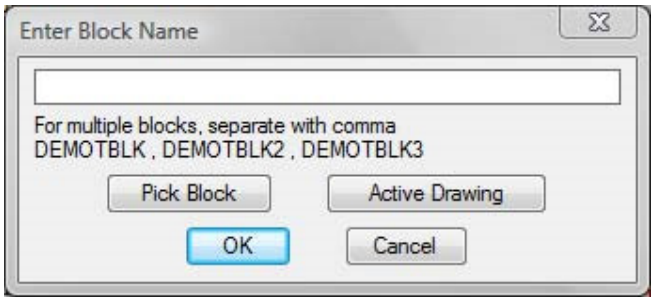
2. In the **Setup Title Block Update** dialog box, click a mapping option.



3. If an external mapping file exists, the *.WDT* File Exists dialog box is displayed. Choose the desired option.

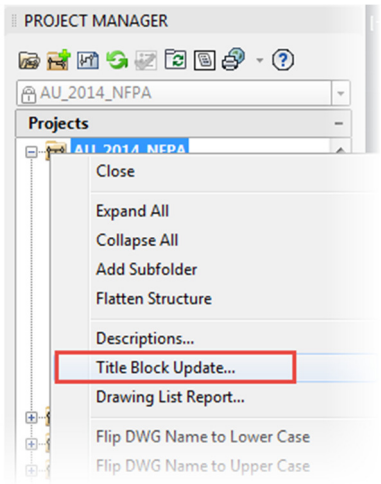


4. In the **Enter Block Name** dialog box, enter or select the title block files to be referenced.



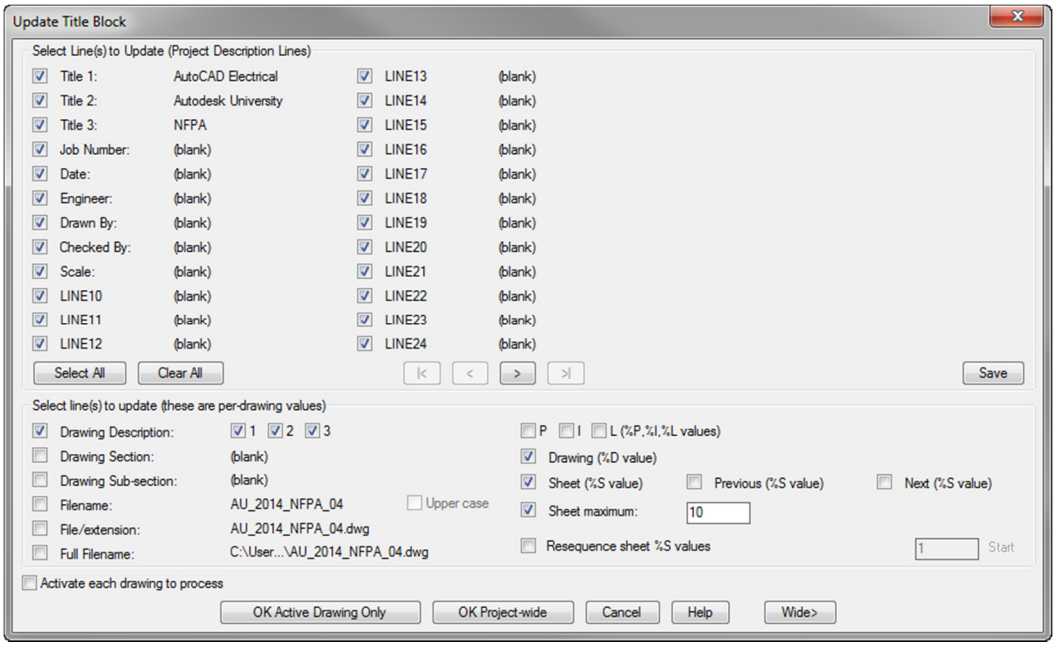
Workflow: Updating a Title Block

1. Start the **Title Block Update** command.



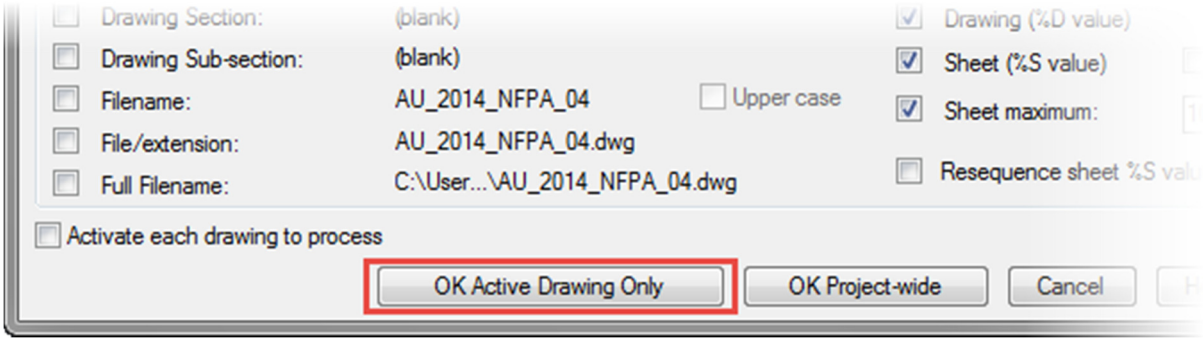
2. In the **Update Title Block** dialog box, select the information values you want to use to update the title block attributes. Only the selected mappings are updated.

If you select values that are blank, or if the appropriate attribute is not found on the title block, the mapping for that item is ignored.

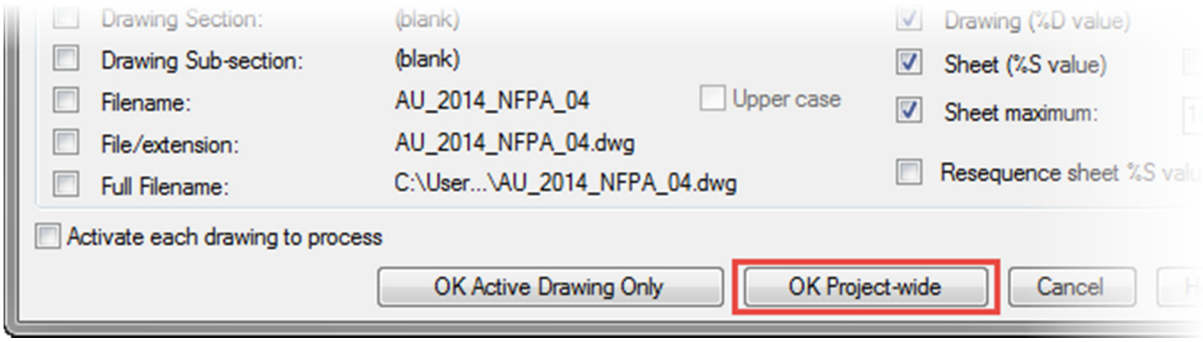


TIPS: In the **Update Title Block** dialog box, click **Save** to save your update selections as the default for the next time the dialog box is opened.

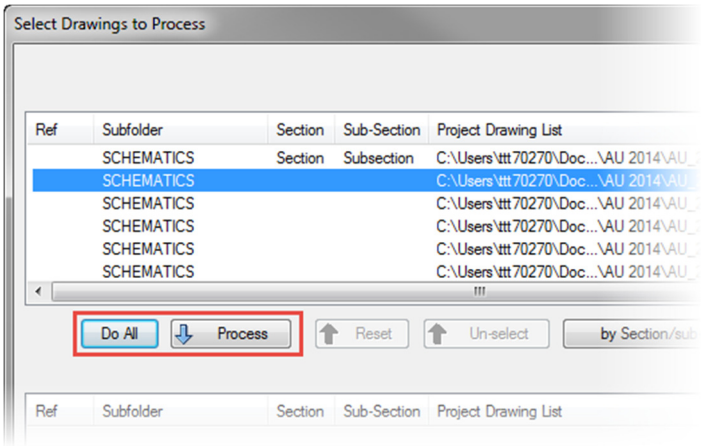
3. Click **OK Active Drawing Only** to update title block attributes on the active drawing title block. (The command ends.)



4. Click **OK Project-Wide** to update title block attributes on multiple drawings in the active project.



5. In the **Select Drawings to Process** dialog box, select the drawings you want to update and add them to the **Drawings to Process** list or click **Do All**. Click **OK**. The title block is updated in each selected drawing in the background and the drawing is saved. When all drawings are processed, the command prompt is blank.



Learning Objective 4: Creating AutoCAD Electrical Templates from vanilla AutoCAD Templates

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.

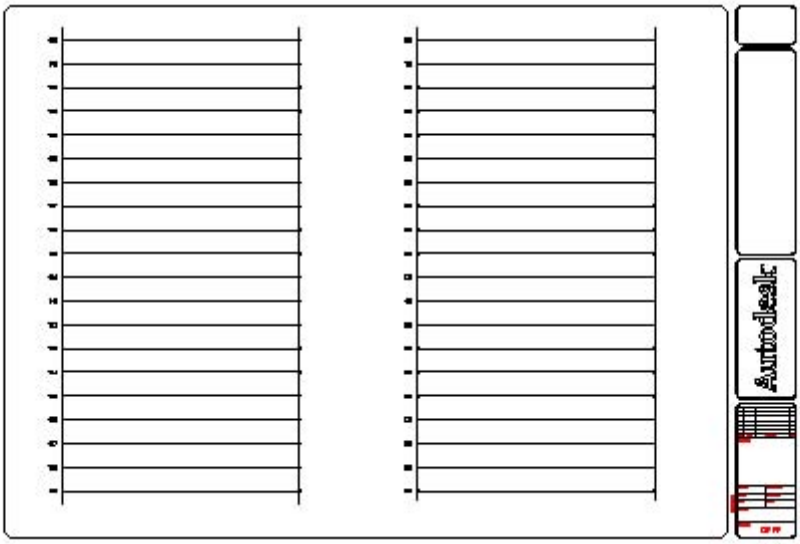
Preset Drawing Graphics

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.

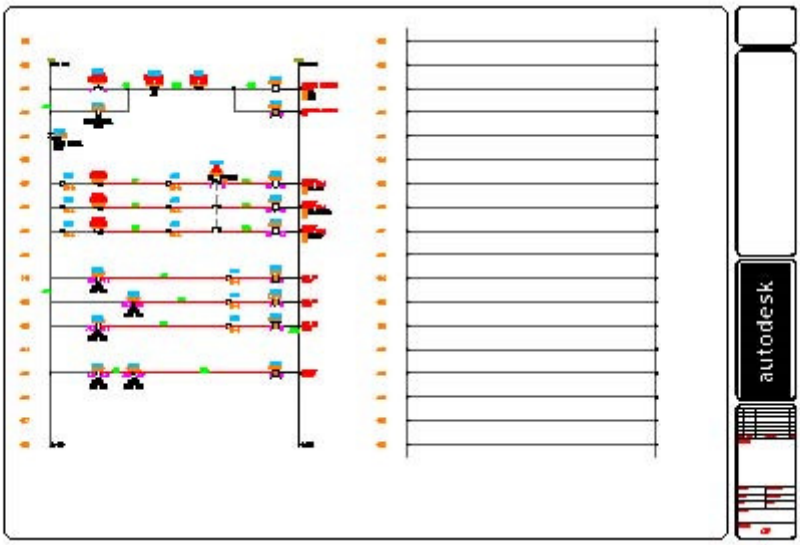
Note: After creating a drawing that is based on a *DWT* file, if you modify the new drawing, the changes do not affect the *DWT* file.

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:

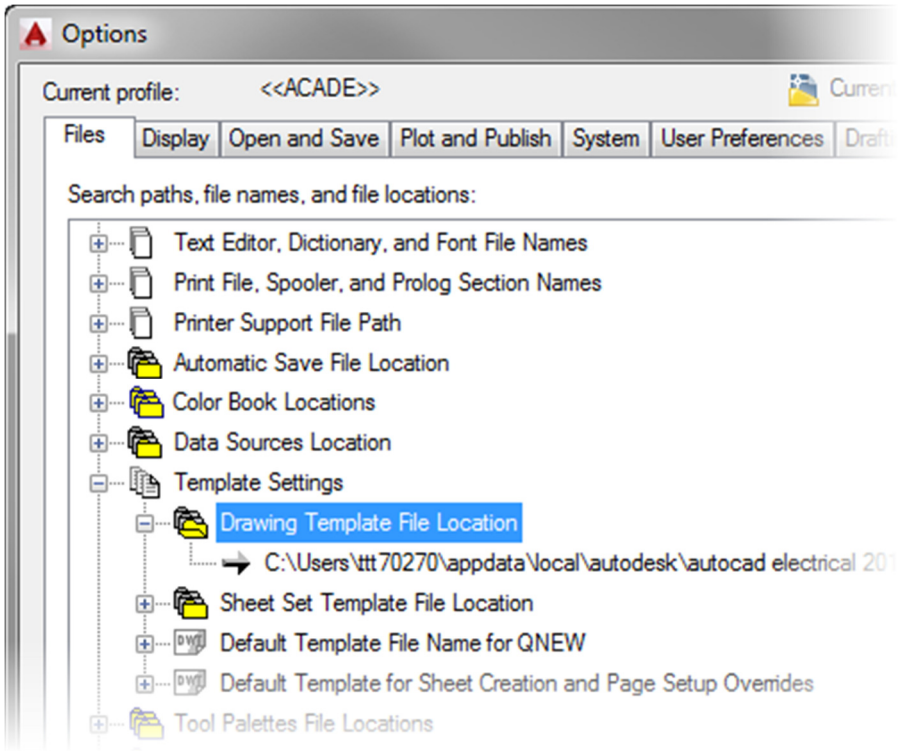
- Blocks, such as symbols or other objects that you commonly use in your drawings
- Ladders, circuits, and other graphical entities

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.

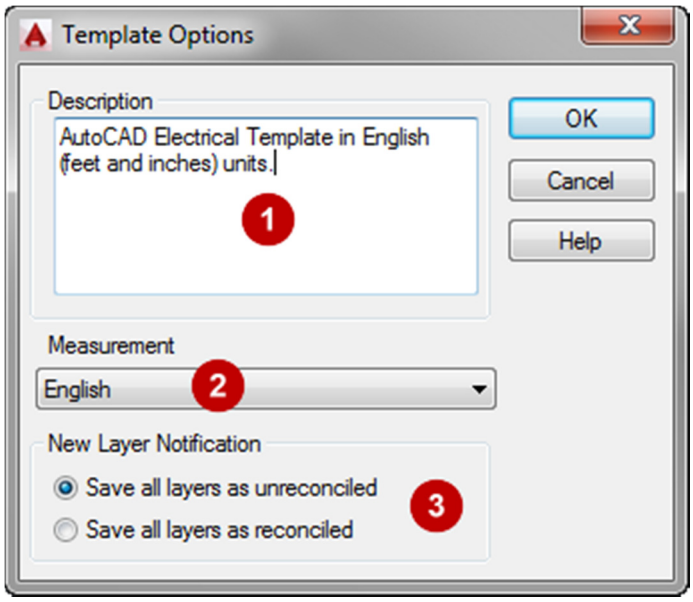
Command Access

Save As

Application menu: Save As > AutoCAD Drawing Template

Command line: **SAVEAS**

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



1 **Description**: Specifies a description for the DWT file.

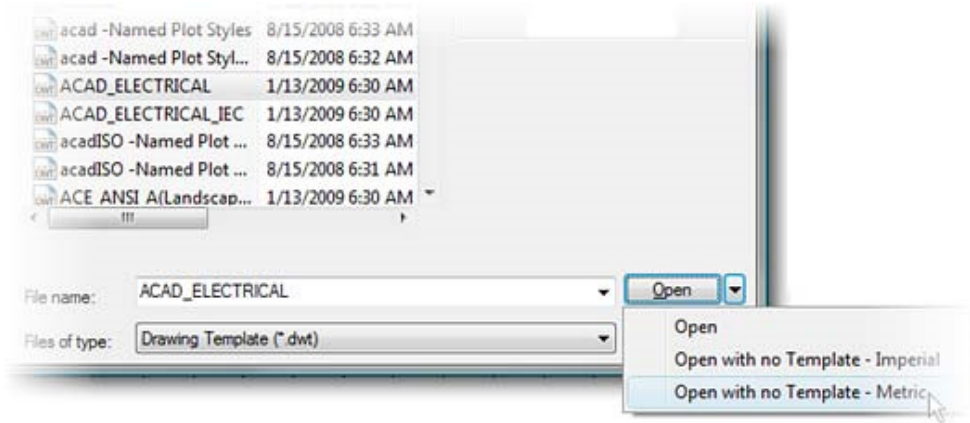
2 **Measurement**: Determines whether drawings based on this template use English or Metric units.

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.

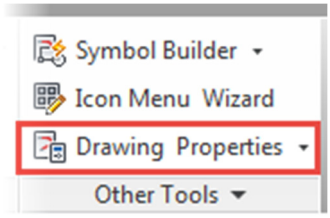
Note: All the layers in a *DWT* file are saved as unreconciled by default.

Creating Drawing Templates Workflow

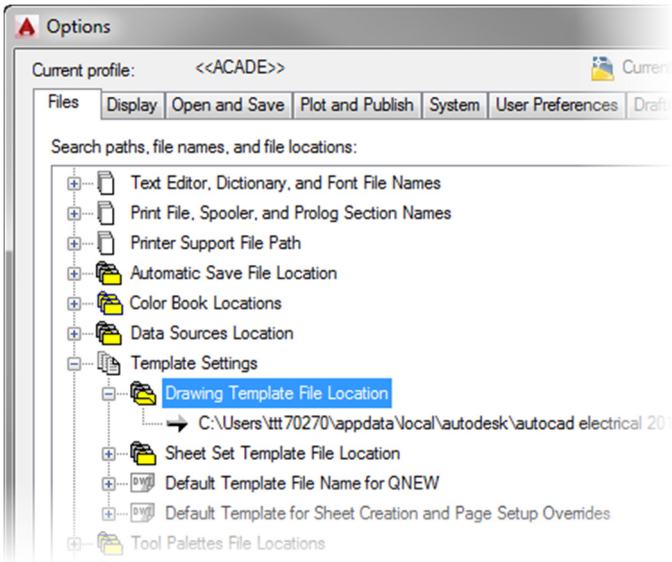
1. Create a new drawing by using an existing template or by using the no template options.



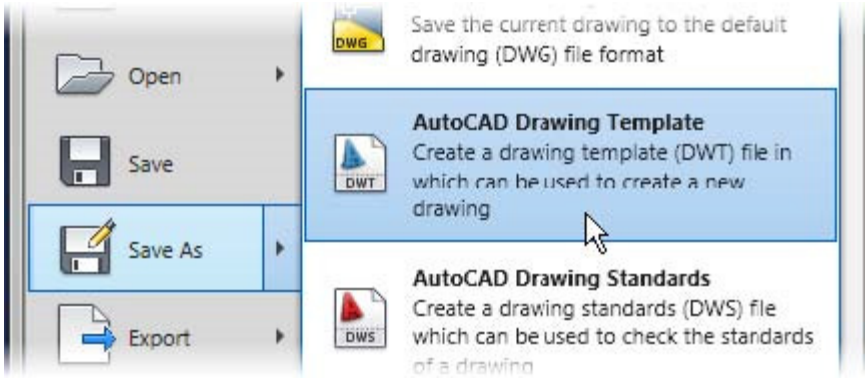
2. Modify the drawing to include the required layers (see Create Wire Types workflow below), styles, layout settings, title blocks, and set drawing properties. Starting any **AutoCAD® Electrical** command, such as **Drawing Properties**, automatically inserts the wd_m.dwg block.



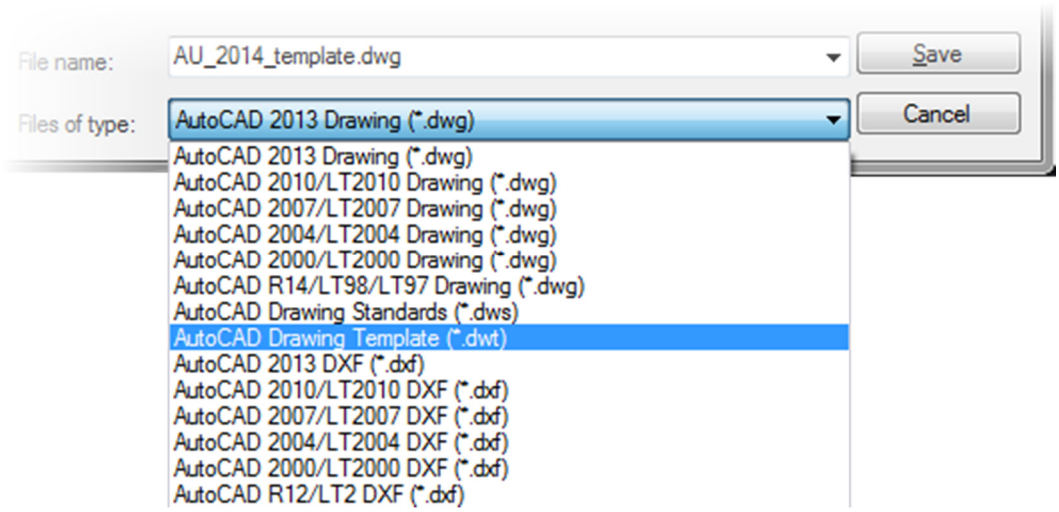
3. Adjust the **Drawing Template File Location** path in the **Options** dialog box, if required.



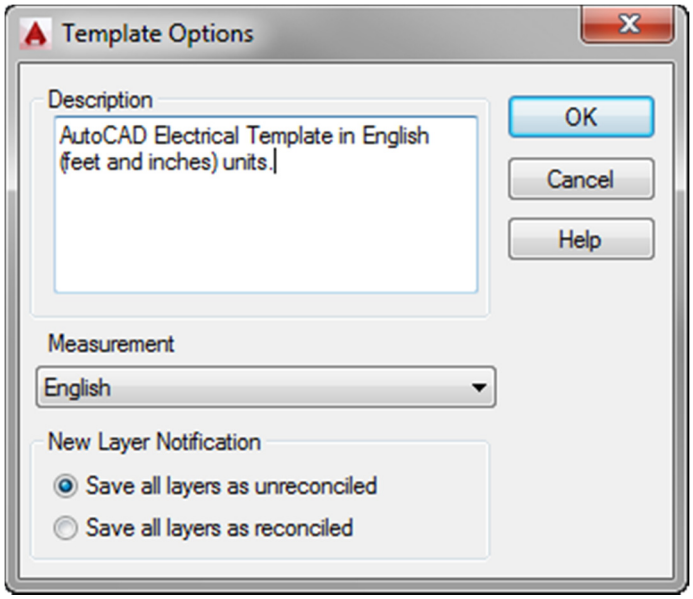
4. On the **Application menu**, click **Save As > AutoCAD Drawing Template**.



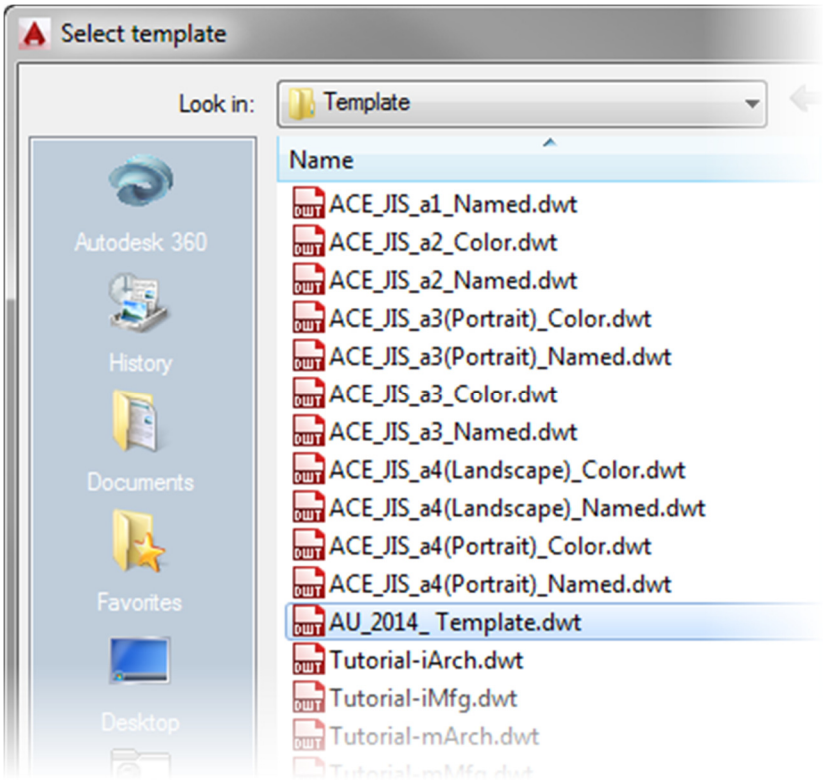
5. In the **Save Drawing As** dialog box, verify that **AutoCAD Drawing Template** (*.*dwt*) is selected from the **Files of Type** list.



6. In the **Template Description** dialog box, enter a description, select the measurement unit, and specify the new layer notification.

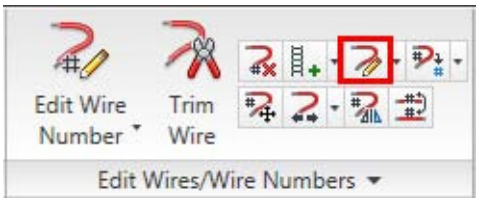


7. Open the newly created template and verify that the drawing contains the settings that you created.

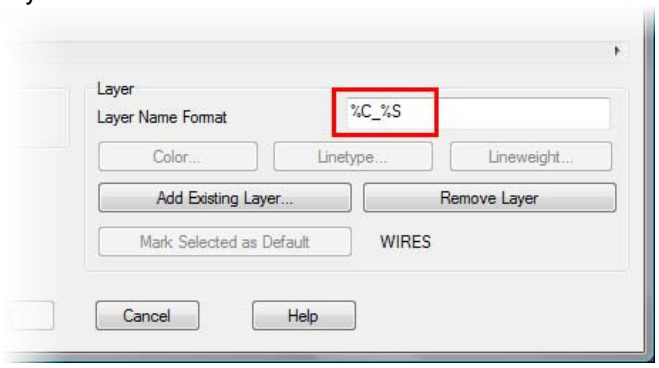


Creating Wire Types Workflow

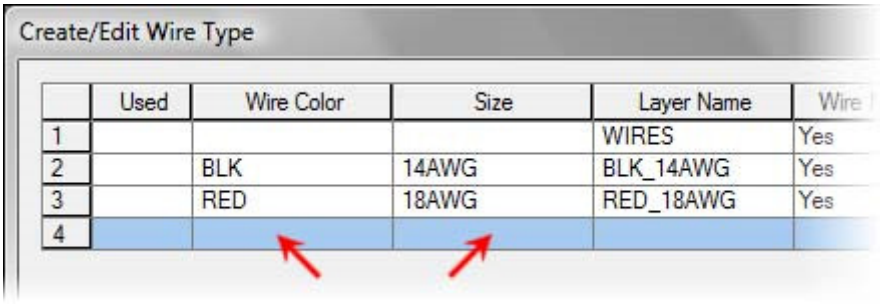
1. Start the **Create/Edit Wire Type** command.



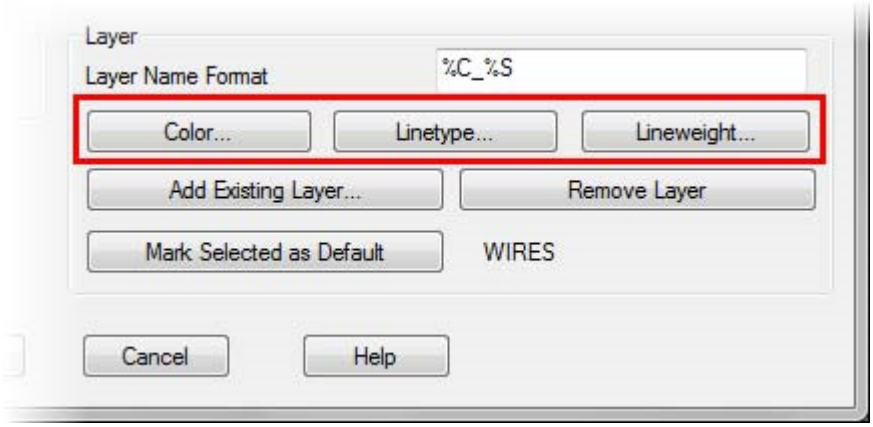
2. Enter the desired layer name format.
Note: You must change the format *before* creating a new layer if you want it to affect the layer name.



3. Enter data for the new wire type in the grid.



4. Change the layer properties for the selected layer(s) as desired.



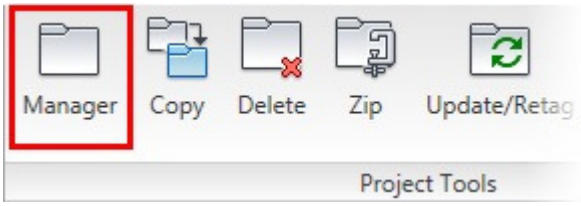
EXERCISE: Exploring the Basics of AutoCAD Electrical

Objective: This exercise will take you through all of the basics of creating schematic drawings. You will complete the following:

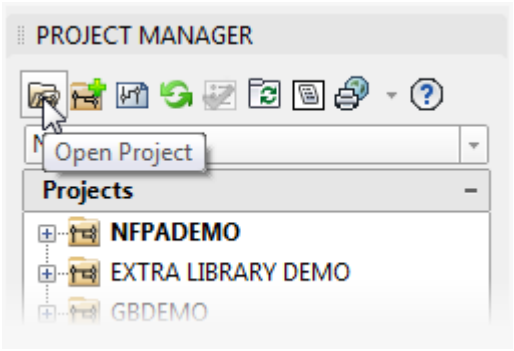
- Insert ladder rungs and a relay coil component.
- Insert push-button components and add part-catalog information.
- Add wire branches and relay coil and child contacts.
- Add wire numbers and generate a Bill of Material report.

Instructions

1: If the **Project Manager** is not displayed, on the **Project** tab, **Project Tools** panel, click **Manager**.



2: In the **Project Manager**, click **Open Project**.



3: Browse to where you saved the project files from the AU website. Select *AU_2014_NFPA.wdp*. Click **Open**.

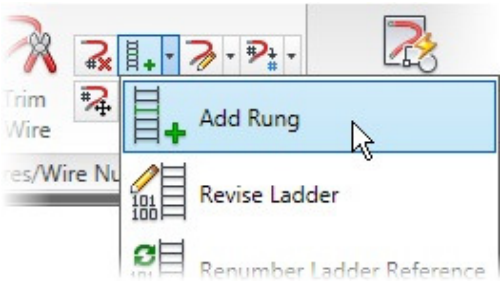
4: In the **Project Manager**, click the expansion node to the left of *AU_2014_NFPA* to expand the drawing list.

5: Double-click *AU_2014_NFPA _04.dwg* to open the drawing.

6: To begin you will insert Ladder Rungs and a Relay Coil:

Zoom in to the upper-left corner of the drawing to rungs 403-404. Make sure both the hot and neutral vertical wires are displayed.

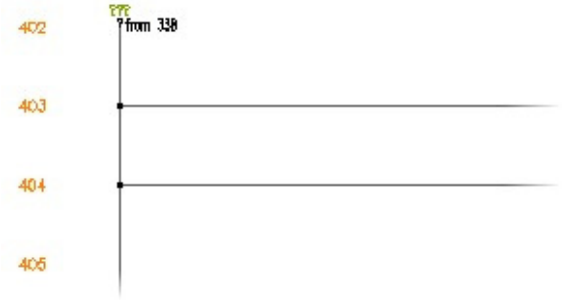
7: On the **Schematic** tab, **Edit Wires/Wire Numbers** panel, click **Add Rung**.



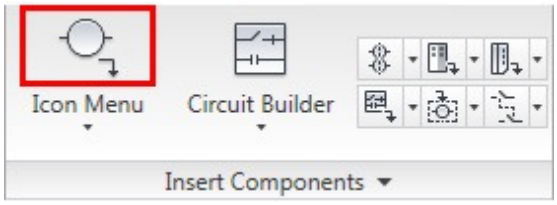
8: Select insertion points for two rungs at rung references 403 and 404.

Note: Be sure to click anywhere between the vertical buses, not on the bus.

9: Notice that the rung automatically snaps to the nearest rung reference, and connection symbols are added as necessary.



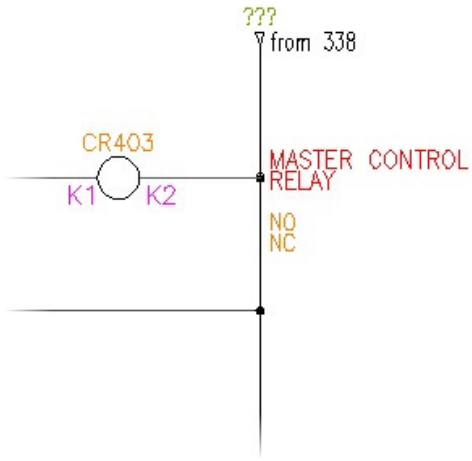
10: On the **Schematic** tab, **Insert Components** panel, click **Icon Menu** to insert the first component, a relay coil.



11: In the **Insert Component** dialog box, click **Relays/Contacts**.

12: In the **NFPA: Relays and Contacts** window, click **Relay Coil**.

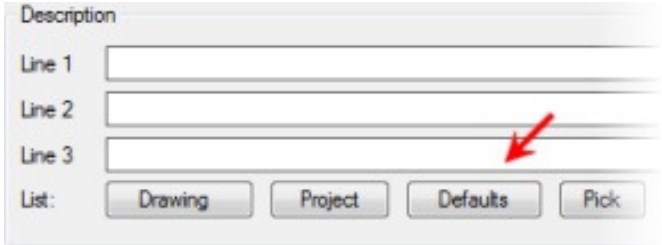
13: Select the insertion point for the relay coil on rung 403, near the right side, directly above **CR407**.



14: Now you annotate the component, adding description and catalog information. You can manually type the desired information, but many tools are provided for typing the information from various reference files automatically. Do the following:

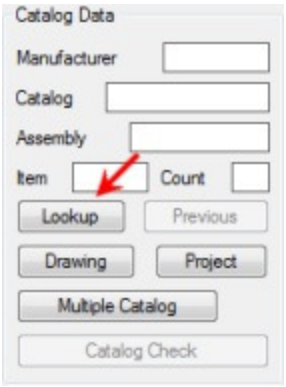
- Notice the tag name is automatically assigned **CR403**.

- In the **Insert/Edit Component** dialog box, under the **Description** area, click **Defaults**.



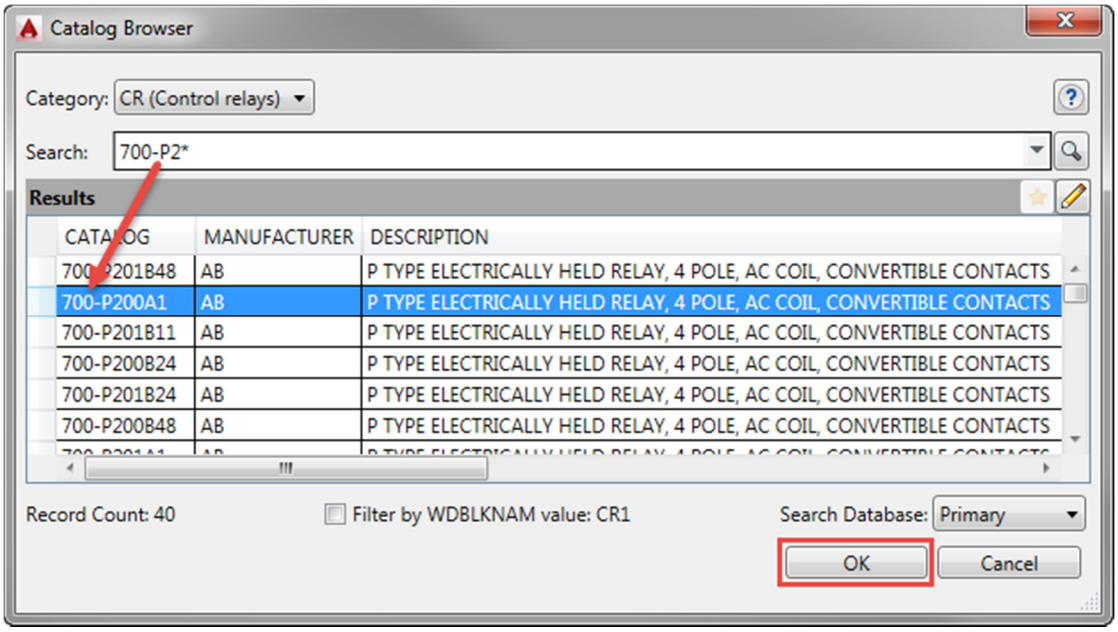
15: In the **Descriptions** dialog box, select **Master Control|Relay**. Click **OK**.

16: In the **Insert/Edit Component** dialog box, under **Catalog Data**, click **Lookup**.

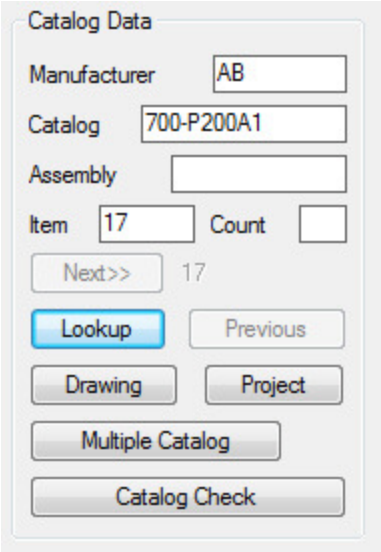


17: In the **Catalog Browser** dialog box, browse the parts catalog database to find the desired part number. You filter the available options using search bar at the top of the dialog box.

- Type **700-P2*** in the **Search** field.
- Select part number **700-P200A1** then click **OK**.



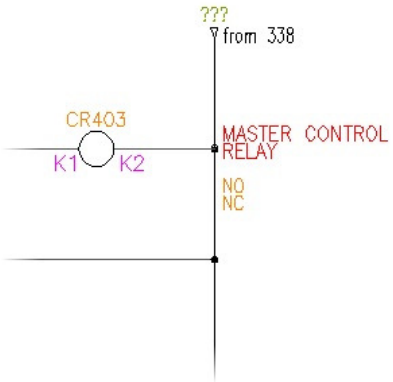
18: Notice that the information is transferred to the **Insert/Edit Component** dialog box.



19: Notice that the selected part number also contains pin-number information, which is automatically entered in the **Pins** area of the **Insert/Edit Component** dialog box.

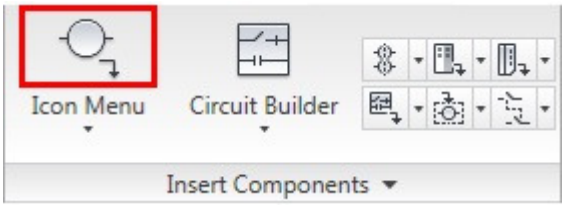
Click **OK**.

20: Notice that the information is transferred to attributes on the inserted component.



This completes the insertion of the relay coil. This basic process is repeated for most component insertions.

21: This next section inserts push buttons and adds part catalog information:
On the **Schematic** tab, **Insert Components** panel, click **Icon Menu** to add a push button to reset the circuit.



22: In the **Insert Component** dialog box, click **Push Buttons**.

23: In the Symbol preview window, click **Push Button NO**.

24: Select the insertion point on rung **403**, near the left side.

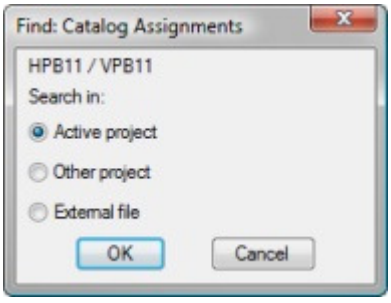
25: Notice that the tag name is automatically entered as **PB403**.

26: In the **Insert/Edit Component** dialog box, under **Descriptions**, click **Defaults**.

27: In the **Descriptions** dialog box, click **System| Reset**. Click **OK**.

28: Under **Catalog Data**, click **Project**.

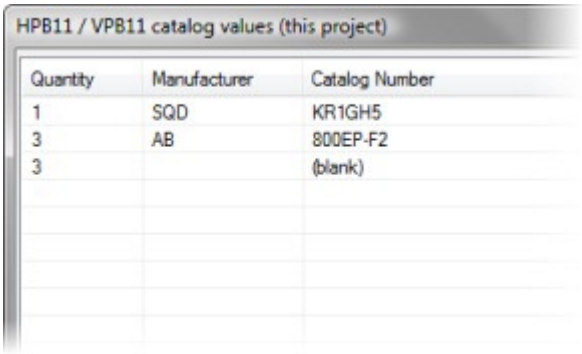
29: In the **Find: Catalog Assignments** dialog box, click **Active Project**. Click **OK**.



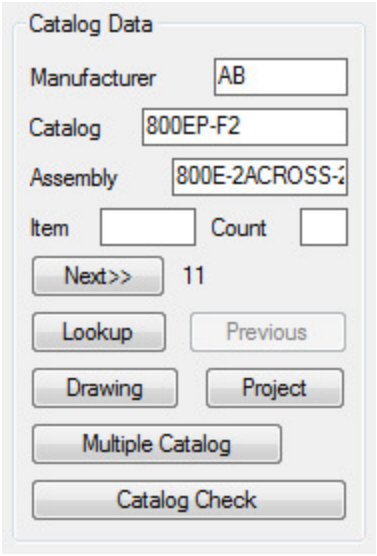
30: In the **Qsave** dialog box, click **Always QSave**.

Note: **AutoCAD Electrical** always stores and works with the data that is saved in the drawings themselves. To ensure that the data is up to date, you are requested to save the current drawing.

31: All drawings in the current project are searched, and any push buttons found are listed in the **HPB11/VPB11 Catalog Values (this project)** dialog box. Select **AB**, **800EP-F2**.

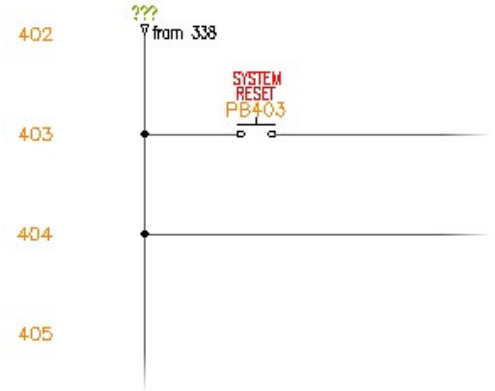


32: Click **OK**. The catalog data, including subassembly information, is transferred to the **Insert/Edit Component** dialog box.

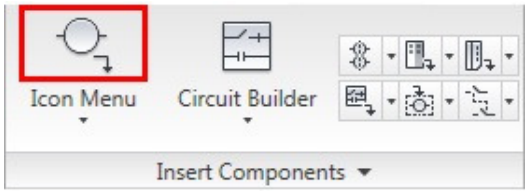


Note: You seldom need to type information more than once. In this project's dataset, push buttons are already used elsewhere in this project. You want to search for and use the same part numbers that have been selected.

33: Click **OK**. This completes the push-button insertion.



34: On the **Schematic** tab, **Insert Components** panel, click **Icon Menu** to add an emergency stop push button.



35: In the **Insert Component** dialog box, click **Push Buttons**.

36: In the Symbol preview window, click **Mushroom Head NC**.

37: Select the insertion point approximately in the middle of rung **403**.

38: Notice the tag name is automatically entered as **PB403A**. Because this is the second push button on the **403** rung, the tag receives an **A** as a suffix.

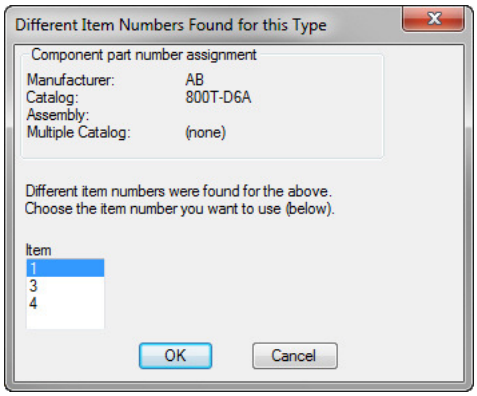
39: In the **Insert/Edit Component** dialog box, under **Descriptions**, click **Defaults**.

40: In the **Descriptions** dialog box, select **Emergency|Stop**. Click **OK**.

41: In the **Insert/Edit Component** dialog box, in the **Catalog Data** area, click **Lookup**.

42: In the **Catalog Browser** dialog box:

- Type **800T*** in the **Search** field.
- Select part number **800T-D6A** then click **OK**.
- If the **Different Item Numbers Found for this Type** dialog box pops up, select an Item Number from the list then click **OK**.

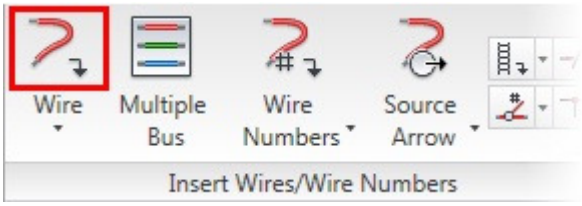


43: In the **Insert/Edit Component** dialog box, click **OK**.
- If asked to **Update other drawings?** Click **OK**.
- If a **Mismatched Item Number** warning pops up, click **OK**.

This completes the push-button insertion.

44: In the next section you will add Wire Branches and Relay Coil Child Contacts:

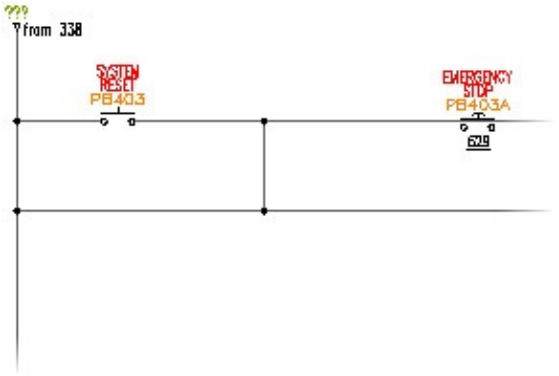
On the **Schematic** tab, **Insert Wires/Wire Numbers** panel, click **Wire** to add two wires that create connecting branches for the circuit.



45: For the wire start point, select a point on rung **403** between **PB403** and **PB403A**.

46: Select the wire endpoint on rung **404** directly below the wire start point.

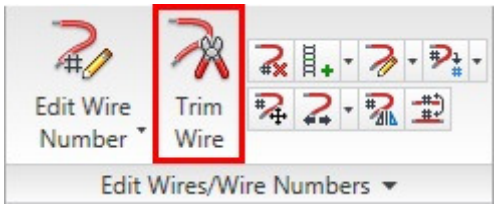
47: Notice that connecting dots are added automatically.



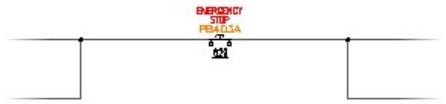
48: For the second wire branch, select the wire start point on rung **403** between **PB403A** and **CR403**.

49: Select the wire endpoint on rung **404** directly below the wire start point. Press **ENTER** to end the command.

50: On the **Schematic** tab, **Edit Wires/Wire Numbers** panel, click **Trim Wire** to trim the wire on rung **404** between the two wire branches that you added.

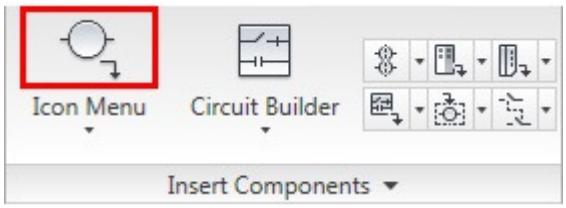


51: Select a point on rung **404** between the two wire branches that you added. Press **ENTER** to end the command.



52: Notice that the selected wire is removed. If the connecting dots are no longer needed, they are also removed.

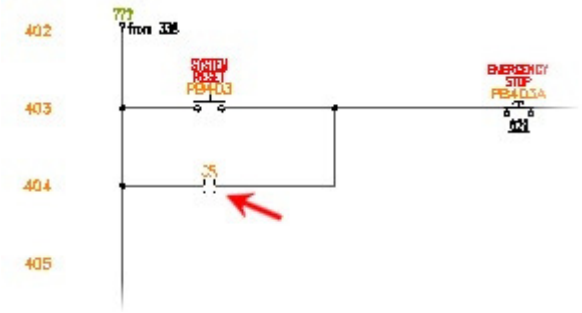
53: On the **Schematic** tab, **Insert Components** panel, click **Icon Menu** to add a contact from **CR403** to latch the circuit after **CR403** is activated.



54: In the **Insert Component** dialog box, click **Relays/Contacts**.

55: In the Symbol preview window, click **Relay NO Contact**.

56: Select the insertion point on rung **404** directly below **PB403**.



This is a child contact of the parent coil.

57: In the **Insert/Edit Child Component** dialog box, under **Component Tag**, click **Parent/Sibling** to select the parent coil and transfer data from the parent to the child.

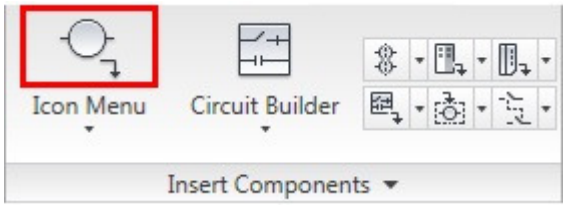
58: Select a point anywhere on **CR403**.

Note: Anywhere on the text works best.

59: In the **Insert/Edit Child Component** dialog box, click **OK** to transfer data, such as tag, description, and pin numbers, to the child component from the parent.

60: The remaining steps will add Wire Numbers and Generate a BOM Report:

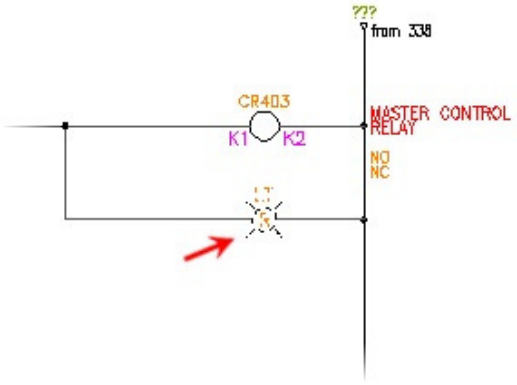
On the **Schematic** tab, **Insert Components** panel, click **Icon Menu** to insert a red light to signal when the circuit is engaged.



61: In the **Insert Component** dialog box, click **Pilot Lights**.

62: In the Symbol preview window, click **Red Standard**.

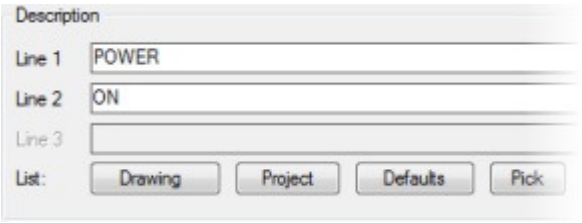
63: Select the insertion point on rung 404 directly below CR403.



64: Instead of using the description defaults, you can manually type the description:

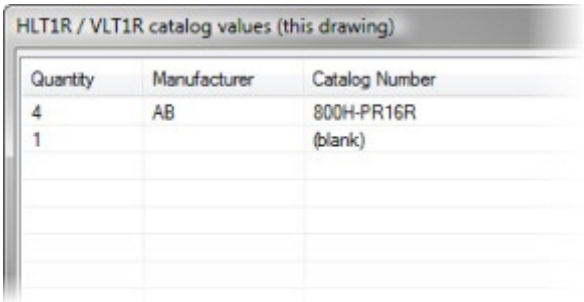
In the **Insert/Edit Component** dialog box, under **Description**:

- For **Line 1**, type **POWER**
- For **Line 2**, type **ON**

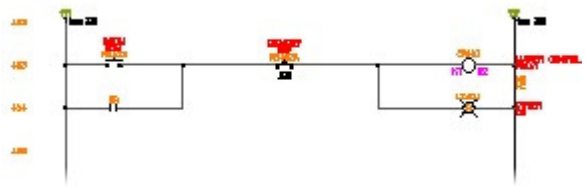


65: Under **Catalog Data**, click **Drawing**.

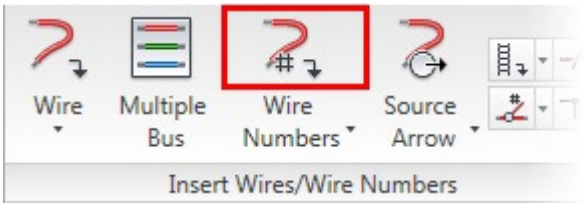
66: In the **HLT1R/VLT1R Catalog Values** dialog box, select **AB**, **800H-PR16R**. Click **OK**.



67: In the **Insert/Edit Component** dialog box, click **OK**.

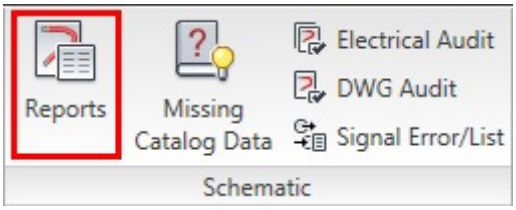


68: On the **Schematic** tab, **Insert Wires/Wire Numbers** panel, click **Wire Numbers** to add wire numbers.



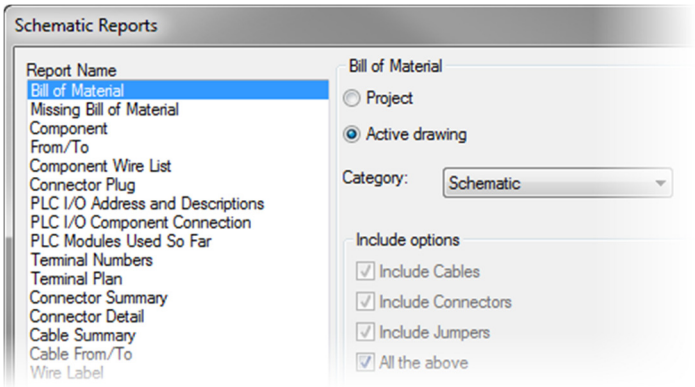
69: In the **Wire Tagging** dialog box, click **Drawing-Wide**.
The drawing is searched for wire networks. A wire number is placed on each network found.

70: The last step is to extract a Bill of Material report from the components in the drawing. On the **Reports** tab, **Schematic** panel, click **Reports**.

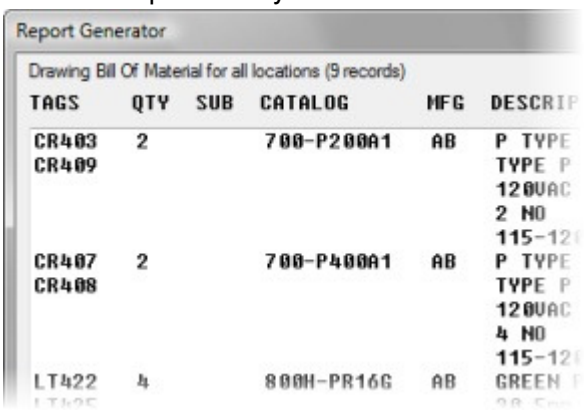


71: In the **Schematic Reports** dialog box, under **Report Name**, select **Bill of Material**.

72: In the **Bill of Material** area, click **Active Drawing**. Click **OK**.



73: Notice the Bill of Material report is generated from component data within the drawing. You can save the report to any of five different file formats, or place it on the drawing in the form of a table.



74: In the **Report Generator** dialog box, click **Close**.

This completes the exercise.

AutoCAD Electrical
Installation Guide

Deployment & Implementation Prerequisites

For this implementation to be successful, you must have:

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

Deployment & Implementation Steps

Deployment (see images below)

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

Implementation

- Create Drawing Template(s)
  - Create/Define Wire Types
  - Define Standard Drawing Properties
  - Make sure your title block is in Paper Space

- Create Project Template(s)
  - Define Standard Project Properties

- Decide what ACADE Support Files to utilize and create
  - Examples
    - Project Line Descriptions: *default_wdtitle.wdl*
    - Title Block Setup: *default.wdt*
    - Default Descriptions: *wd_desc.wdd*
    - Installation/Location Code External Files: *default.inst*, *default.loc*

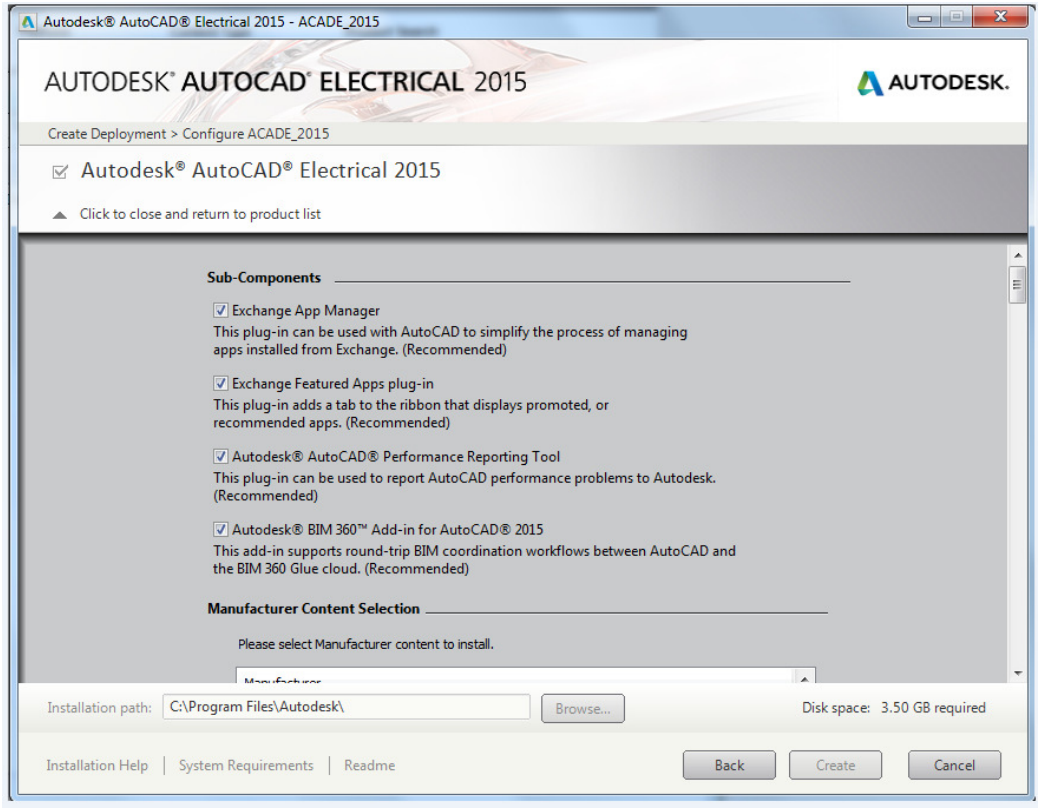
- Create a complete Standard Project(s) to be used as a template for all future projects
  - Creation of custom schematic symbols, as needed
    - Add symbols to the Icon Menu, as needed
  - Creation of custom panel footprints, as needed
    - Update the Footprint Database, as needed
  - Create new part 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

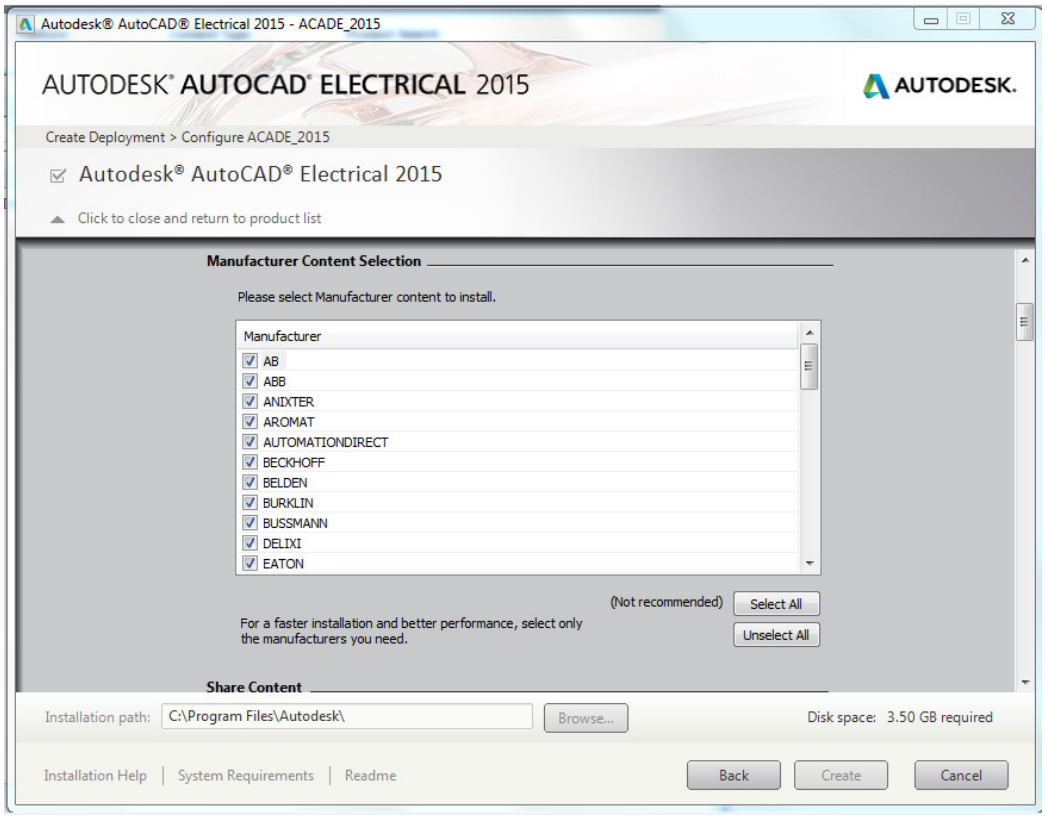
TIPS:

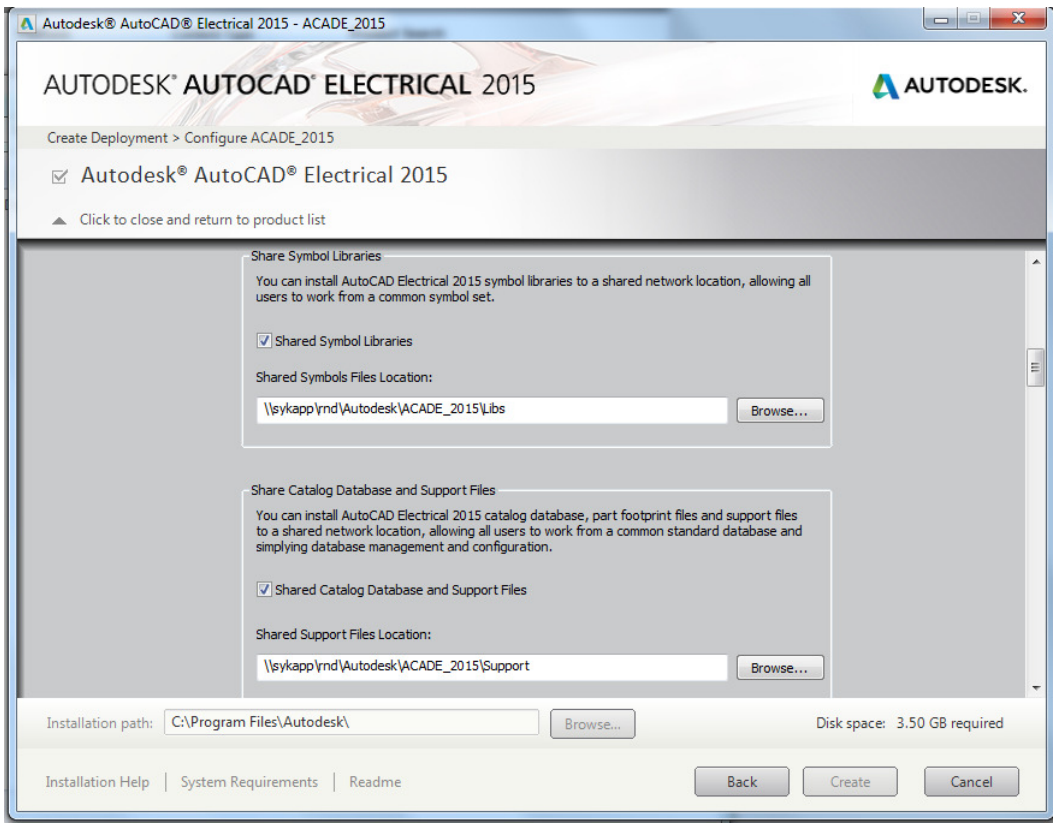
- *Try to work as "out of the box" as you can with ACADE. It will make your implementation much smoother. (Even if it means, changing the way some of your symbols look, etc.)*
- *Although your project files can access drawings from many different folders, it is a best practice to keep the project file,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

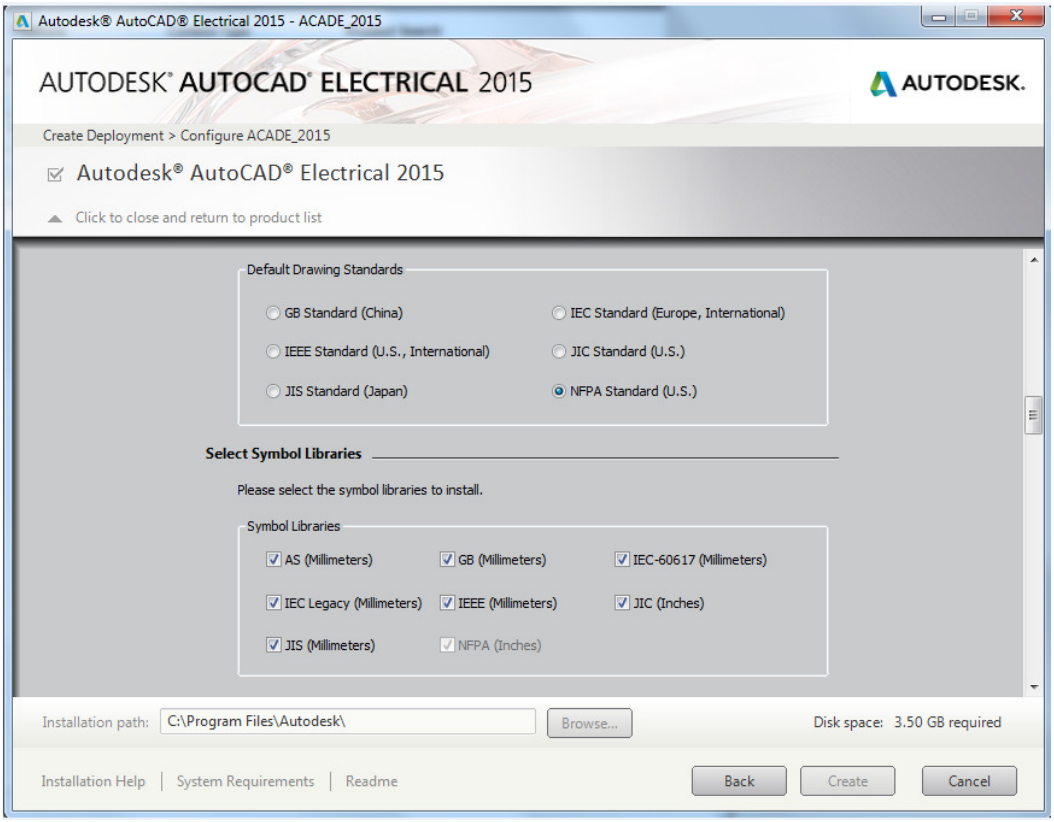
Implementation Steps for a Vaulted Environment

- 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




Autodesk® AutoCAD® Electrical 2015 - ACADE_2015
AUTODESK® AUTOCAD® ELECTRICAL 2015
AUTODESK.
Create Deployment > Configure ACADE_2015
Autodesk® AutoCAD® Electrical 2015
Click to close and return to product list
Manufacturer Content Selection
Please select Manufacturer content to install.
Manufacturer
AB
ABB
ANIXTER
AROMAT
AUTOMATIONDIRECT
BECKHOFF
BELDEN
BURKLIN
BUSSMANN
DELIXI
EATON
(Not recommended)
Select All
Unselect All
For a faster installation and better performance, select only the manufacturers you need.
Share Content
Installation path:
C:\Program Files\Autodesk\
Browse...
Disk space: 3.50 GB required
Installation Help
System Requirements
Readme
Back
Create
Cancel


Autodesk® AutoCAD® Electrical 2015 - ACADE_2015
AUTODESK® AUTOCAD® ELECTRICAL 2015
AUTODESK.
Create Deployment > Configure ACADE_2015
Autodesk® AutoCAD® Electrical 2015
Click to close and return to product list
Share Symbol Libraries
You can install AutoCAD Electrical 2015 symbol libraries to a shared network location, allowing all users to work from a common symbol set.
Shared Symbol Libraries
Shared Symbols Files Location:
\\sykapp\rnd\Autodesk\ACADE_2015\Libs
Browse...
Share Catalog Database and Support Files
You can install AutoCAD Electrical 2015 catalog database, part footprint files and support files to a shared network location, allowing all users to work from a common standard database and simplying database management and configuration.
Shared Catalog Database and Support Files
Shared Support Files Location:
\\sykapp\rnd\Autodesk\ACADE_2015\Support
Browse...
Installation path:
C:\Program Files\Autodesk\
Browse...
Disk space: 3.50 GB required
Installation Help
System Requirements
Readme
Back
Create
Cancel


Autodesk® AutoCAD® Electrical 2015 - ACADE_2015
AUTODESK® AUTOCAD® ELECTRICAL 2015
AUTODESK.
Create Deployment > Configure ACADE_2015
Autodesk® AutoCAD® Electrical 2015
Click to close and return to product list
Default Drawing Standards
GB Standard (China)
IEC Standard (Europe, International)
IEEE Standard (U.S., International)
JIC Standard (U.S.)
JIS Standard (Japan)
NFPA Standard (U.S.)
Select Symbol Libraries
Please select the symbol libraries to install.
Symbol Libraries
AS (Millimeters)
GB (Millimeters)
IEC-60617 (Millimeters)
IEC Legacy (Millimeters)
IEEE (Millimeters)
JIC (Inches)
JIS (Millimeters)
NFPA (Inches)
Installation path:
C:\Program Files\Autodesk\
Browse...
Disk space: 3.50 GB required
Installation Help
System Requirements
Readme
Back
Create
Cancel