

Synergis University Technology Event - June 3, 2015

Moving AutoCAD® Designs to AutoCAD® Electrical

Todd Schmoock – Synergis Technologies

AutoCAD® Electrical has proven to be easy for creating electrical controls system designs. It has libraries of standard components, and intelligent connections between drawings and tools for updating entire projects at once. But what to you do with previous work? How to you get a project designed in regular AutoCAD® to have this functionality?

AutoCAD® Electrical is the answer again. In this session, we will take an AutoCAD®-based control system design and add AutoCAD® Electrical intelligence. We will replace existing blocks with Electrical components, while keeping the block's attribute data. We will make custom Electrical components and circuits from regular AutoCAD® files for use in future designs. We will even convert existing lines into wires and add Electrical project intelligence to title blocks. Most importantly, we will do all this with tools available in a typical installation of AutoCAD® Electrical!

About the Speaker:

Todd is a Solutions Engineer with Synergis Technologies. He has over 25 years of Drafting, Design, and CAD experience in manufacturing environment using AutoCAD®, AutoCAD® Electrical, and Inventor which he is a certified Inventor Expert. Ten years of this time was spent as a documentation specialist/designer at Honeywell, Inc. where he worked on several government contracts which required strict drafting and design documentation in accordance with government standards. Additional experience comes from working in the technical ceramic, elevator, and specialty gas industry designing equipment for each of these fields. Todd provides training for Synergis customers and has presented classes at Autodesk University, implementation support, and pre-sales support of Autodesk manufacturing products for commercial and educational customers.

todd.schmoock@synergis.com



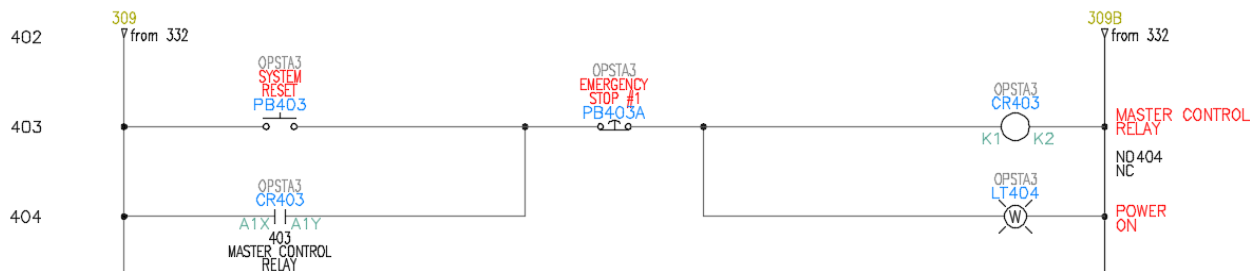
800.836.5440
www.synergis.com/cad

You've purchased and installed AutoCAD® Electrical. You saw the demos, you've attended training. You know how to build new designs in Electrical and you are raring to go!!!

However, the first project you need to do is a variation of a design you already created in AutoCAD®. It's a large enough change that you would like to use the power of Electrical, and this design will likely be the basis for many future projects. What do you do now?

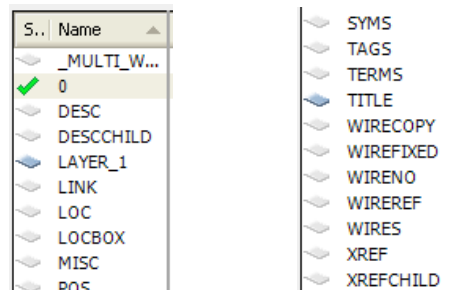
If you are thinking about redoing all your work from scratch in Electrical wait until after this session to make your decision. If you want to take your existing work and make it into an intelligent Electrical design try this tutorial.

Note: In this document the name AutoCAD® Electrical will often be abbreviated to ACE.

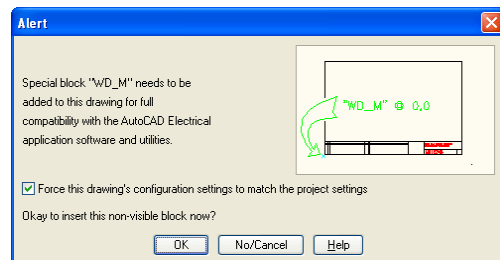


AutoCAD® Electrical uses layers and smart blocks in the model tab to pass the information from drawing to drawing. In addition it has specific tools which use macros and lisp routines to use this information from the model tab to run tools like the report generator and project wide updates.

Here are some of the special layers:



Note that in the standard acad.dwt you would only have layer 0. When you start an AutoCAD® Electrical command on a drawing without these layers you are prompted to insert a hidden block at 0,0 which contains these layers:



Here is an example of a smart block, in this case a schematic component, and the attributes it uses to communicate with the rest of AutoCAD® Electrical. Several of these attributes are invisible in the drawing:

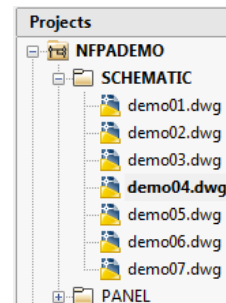


Additionally, AutoCAD® Electrical communicates through wires. This means lines in ACE are not just lines. ACE looks at them like actual wires, connected to the smart blocks. For example: if you have a layer called RED_18AWG you would identify this as a wire layer. Wire numbers could be assigned to lines on this layer and reports could be generated.

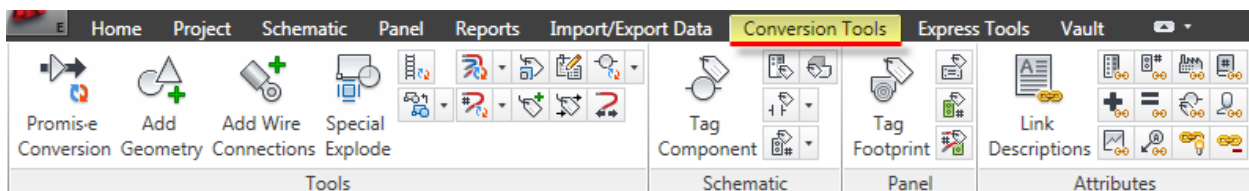
These wire numbers are not just plain AutoCAD® text. They are smart text that can link wires from drawing to drawing. You can run reports to print all wire numbers associated to a complete drawing project:

403

You can put several smart ACE drawings together to make a project. All of the drawings are linked so you can run reports and identify components across the entire project.



To take a regular AutoCAD® drawing and add Electrical intelligence to it, you use AutoCAD® Electrical's conversion Ribbon:



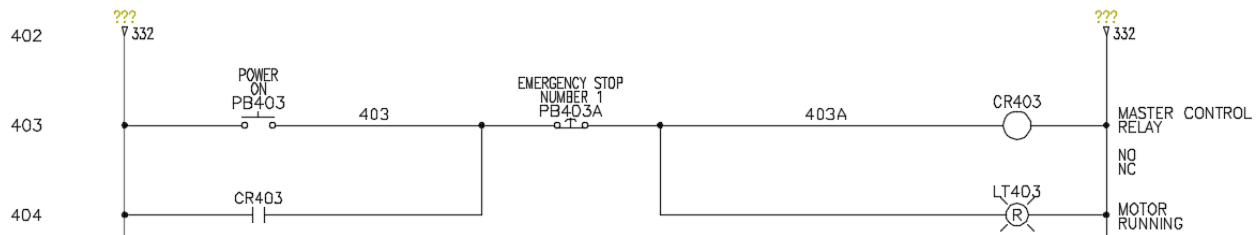
With these tools you can convert AutoCAD® blocks to AutoCAD® Electrical smart blocks, lines to wires, standard text to smart wire numbers, and add all drawings to a project.

Getting Started

For this exercise you will start with an example using a variation of the standard AutoCAD® Electrical exercise file **demo04.dwg**.

- Open **AutoCAD® to Electrical.dwg**:
 “C:\Documents and Settings*<login>*\My Documents\Acade 2011\AeData\Proj”

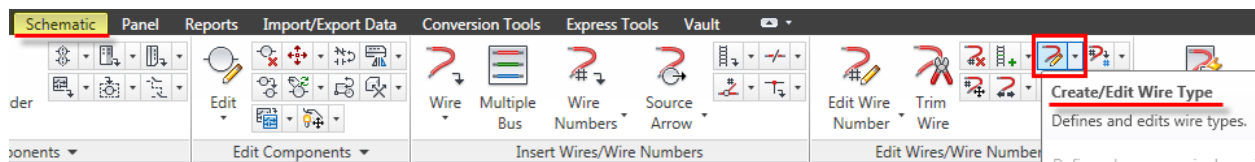
This circuit doesn't have any smart Electrical objects in it. The circuit needs to be converted from lines, blocks, and text to Electrical objects.



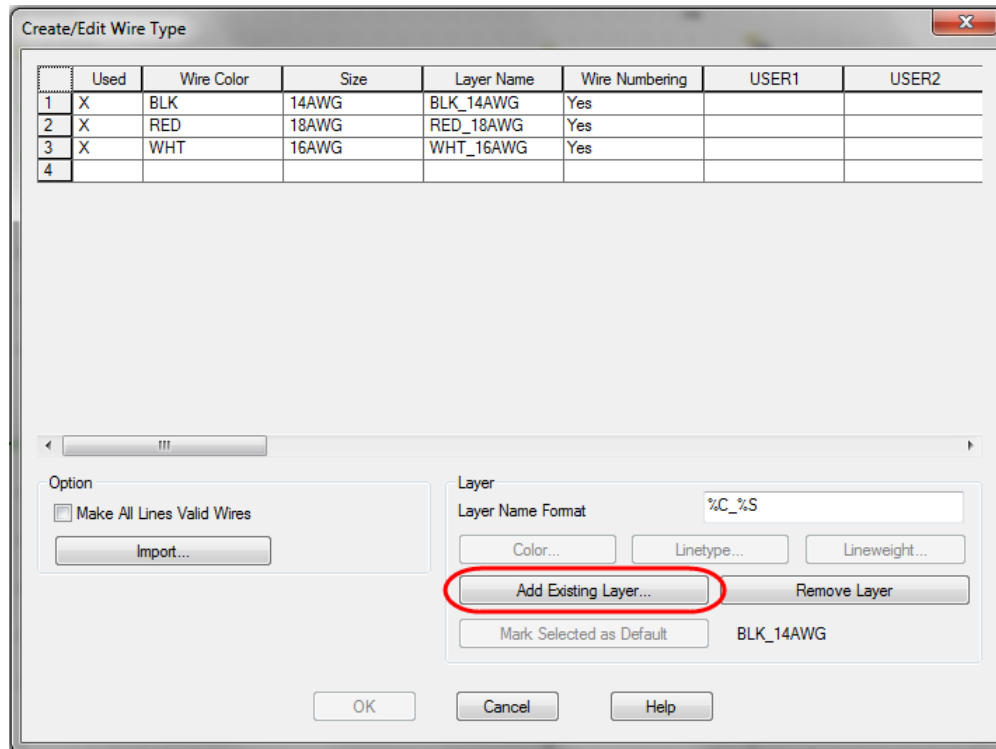
Converting Lines to Wires

The first step when converting a line to a wire is to identify which layers are going to be wire layers. The tool for this is **Create/Edit Wire Type**:

Note: You have to go to the Schematic Ribbon for this tool. Autodesk did not add this tool to the Conversion Tools Panel.

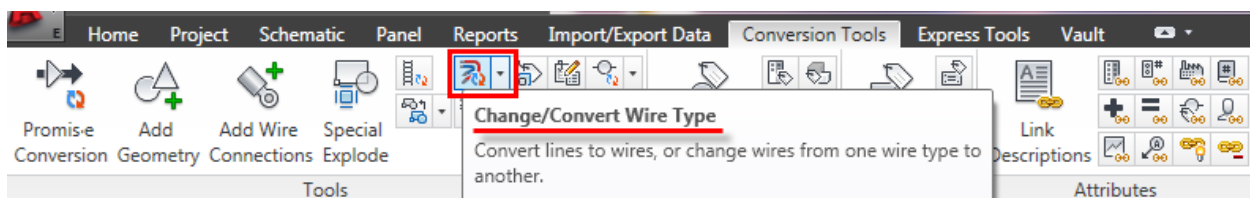


With this command's dialog box you can change existing layers that you have identified for wires to ACE smart wires:

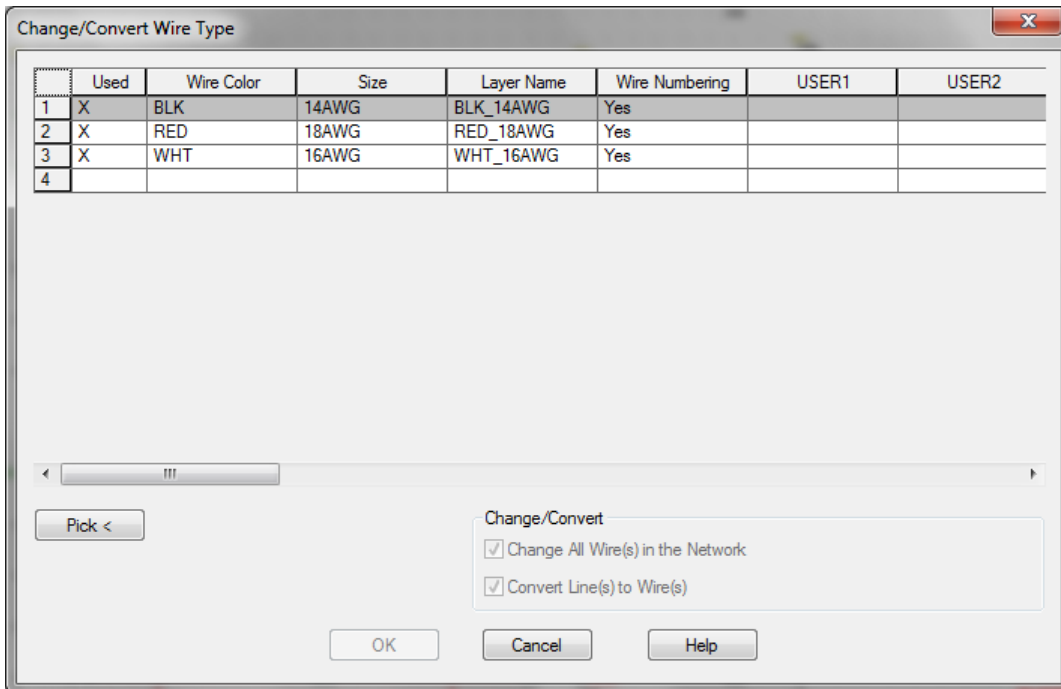


You use the **Add Existing Layer...** option to add an additional layer to this list, or create a new wire by adding information to the next available row.

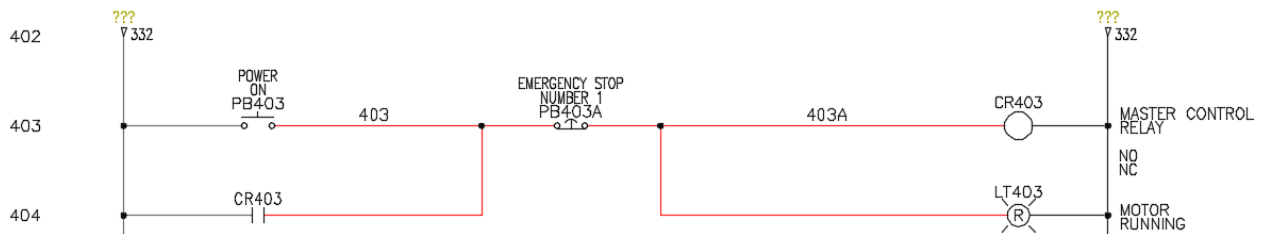
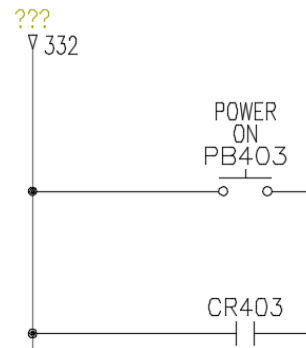
To move lines to a wire layer, which will make them wires from ACE's point of view, use the **Change/Convert Wire Type** tool.



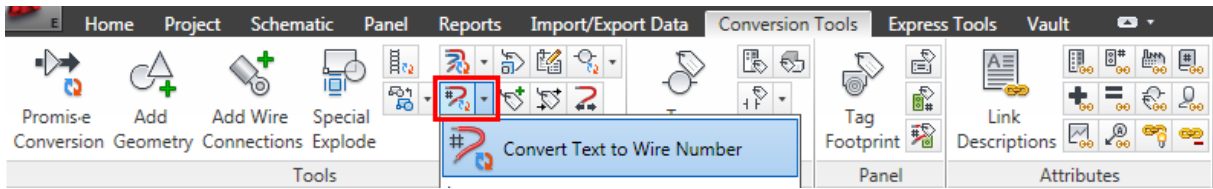
In this dialog you select the desired wire layer you want to move the AutoCAD® line to, and then pick the lines. *Note: This tool is also used to change ACE wires from one type to another.*



- Start the **Change/Convert Wire Type** command.
- Click on the **BLK_14AWG** type, which is the first row.
- Click **OK**.
- Select the two lines to the left of the **PB403** push button and **CR403** contact:
- Press **Enter** to complete the selection.
- Repeat the process to convert all the middle lines to **RED_18AWG** and the two lines to the right of the **CR403** relay coil and **LT403** to **WHT_16AWG**.

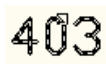


Now, look at the text that identifies the wire numbers. At this point it is just standard AutoCAD® text. Since the lines are now valid wires you can easily change this text to ACE wire numbers by using this tool:



- Start the Convert Text to Wire Number tool.
- Select the valid wire layer near the AutoCAD® text which identifies the wire number.



- Then pick the text:  Now it's a smart ACE wire number.

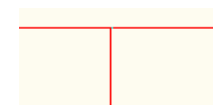
This tool put it on the WIREFIXED layer to avoid any automatic edits. This is verified by the color change. This can also be verified by selecting the text and checking its layer.



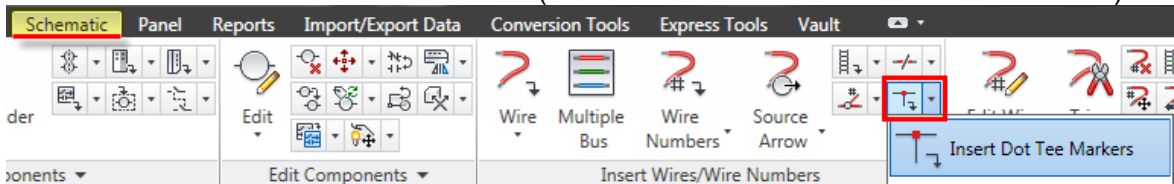
Inserting Wire Connections

In AutoCAD® wire connections are usually individual blocks. If you need to move or delete a wire in AutoCAD® you would also have to handle the connections manually. However, if these are ACE wire connection symbols you will not have to deal with them. They will be removed automatically if you delete a wire, or moved with the wire if the wire is shifted.

- Delete the current AutoCAD® wire connection block to the right of the PB403 push button, leaving the two red wires:

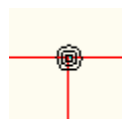


- Start the **Insert Dot Tee Marker** tool (This tool is located on the Schematic ribbon):



- Place the marker at the endpoint of the vertical line.
- After the Dot Tee Marker is inserted hit the **ESCAPE** key to exit the command.

Your drawing should look like this:



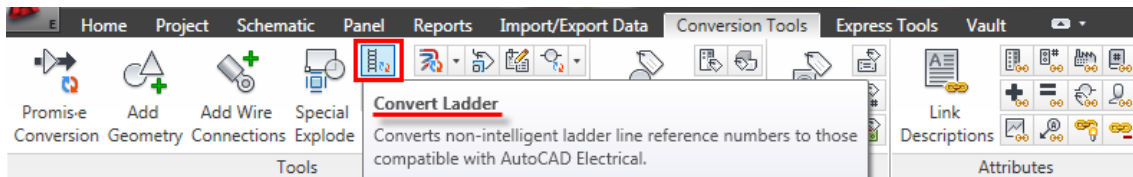
- Repeat this process for the wire connection to the right of PB403A.

Migrating a Ladder to AutoCAD® Electrical

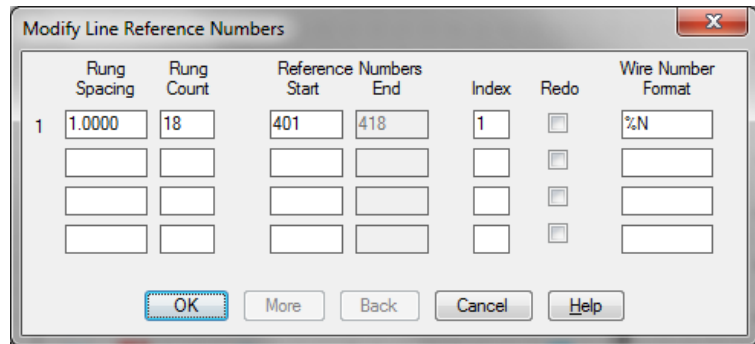
Even after the lines are made into wires the ladder isn't fully an AutoCAD® Electrical smart object. In an ACE ladder like the one in this exercise, the upper left wire number actually keeps track of the ladder's specifications.

The **Convert Ladder** tool can be used to add the missing intelligence to the existing AutoCAD® text.

- Zoom in on the upper left part of the drawing.
- Start the **Convert Ladder** tool.

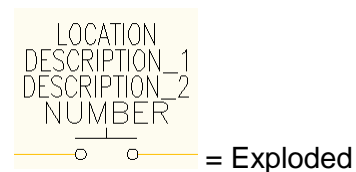
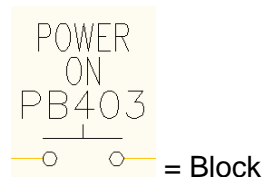


- Select the top-left text
- Press **Enter** or right-click.
- Check that the following information is shown:
- Rung Spacing: **1.000**
- Rung Count: **18**
- Reference Start: **401**
- Index: **1**
- Wire Number Format: **%N**
- Press **OK** to finish the command.



Replacing AutoCAD® blocks with ACE Components

As mentioned earlier this push button's block is just a regular AutoCAD® block:



There are several methods that can be used to replace these AutoCAD® blocks with smart AutoCAD® Electrical blocks (components).

1. Swap/Update Block
2. Convert to Schematic Component
3. Block replacement – Attribute Mapping
4. Special Explode

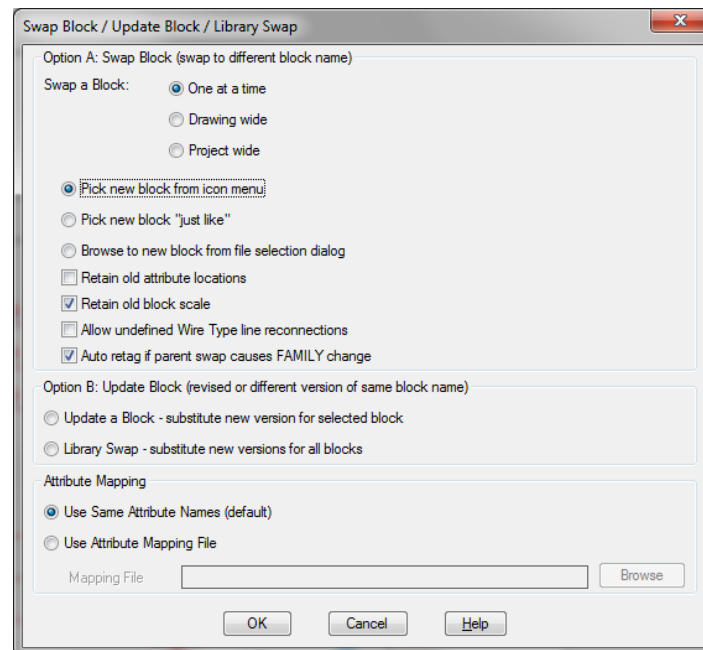
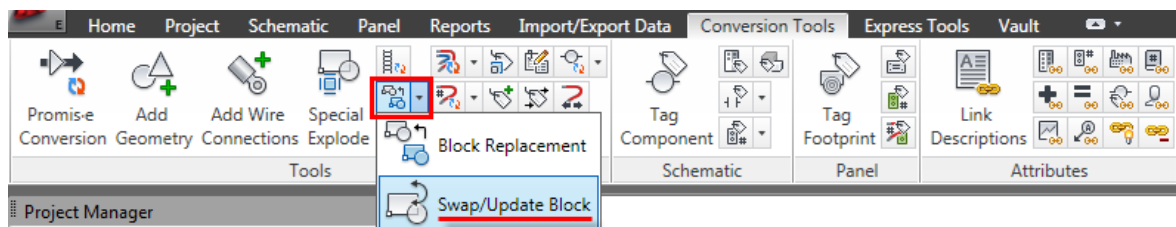
The **Attribute Mapping** option takes more time to set up than is available in this lab. The workflow is included as an appendix at the end of this document. Once the mapping is set up it can easily be used to convert multiple drawings and projects.

The **Special Explode** tool is useful for converting a single drawing that doesn't have very many existing blocks. It is also included in the appendix.

*Note: AutoCAD® Electrical has an excellent tool for creating smart blocks from regular AutoCAD® geometry and adding the desired attributes, called the **Symbol Builder**. The Symbol Builder is typically covered during basic Electrical courses, so it won't be covered here.*

Smart Block Creation – Swap/Update Block

The Swap/Update Block tool is useful for converting a single block definition within a drawing or an entire project. It must be repeated for each block definition. It can be found on the conversion tool bar and the main electrical tool bar:



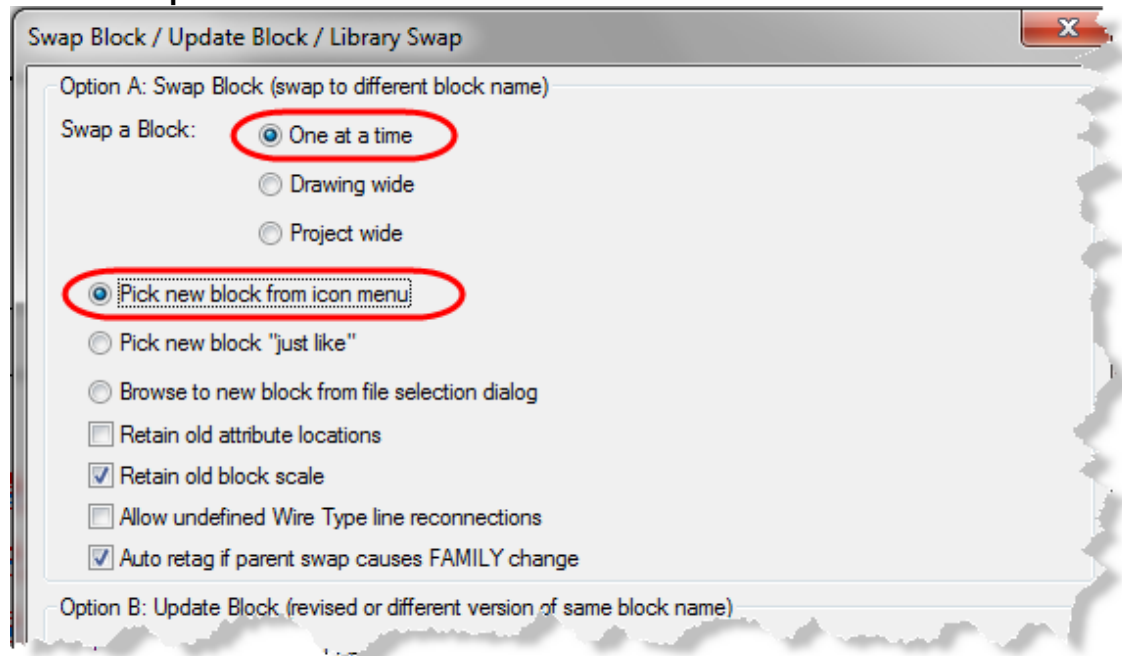
The dialog box has the three options:

- Option A allows you to swap to a different block name.
- Option B gives you the option to swap a block with another block which has the same block name.

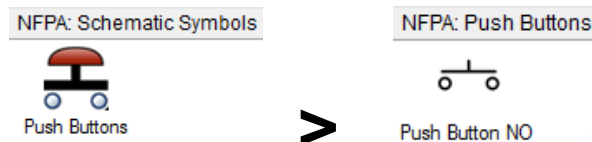
- The last option allows you to map attributes using an external mapping file. This is shown in the Appendix.

For this lab you will use Option A, Swap a Block.

- Pick **Swap a Block > One at a time > Pick new block from icon menu.**



- Select **OK**.
- Pick **Push Buttons** in the icon menu.
- Select **Push Button NO**.

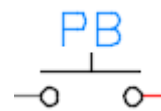


Unfortunately when using Swap Block in this scenario the attribute information does not carry over to the newly placed block:

Before:

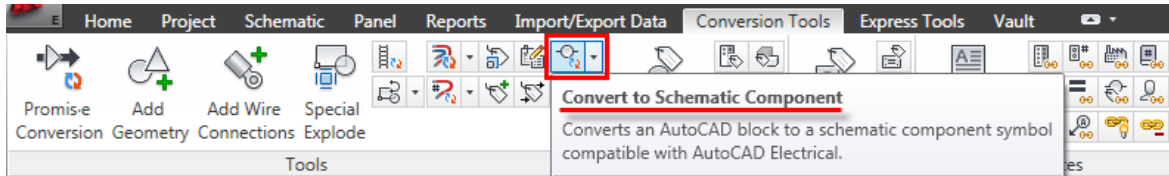


After:



Smart Block Creation - Convert to Schematic Component

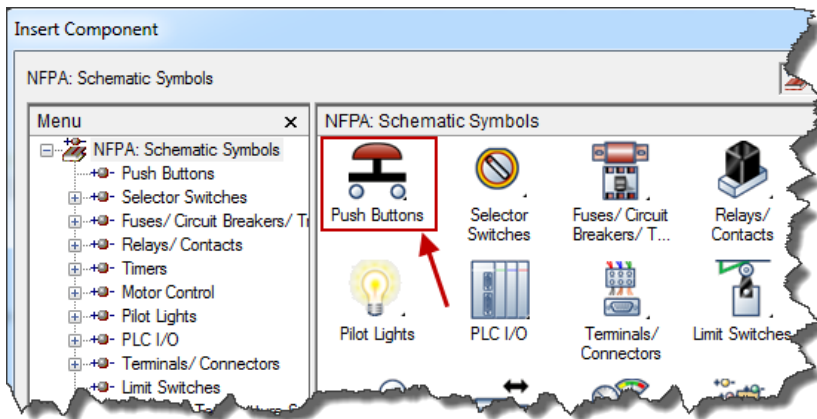
The **Convert to Schematic Component** tool can be used to replace an individual block with an existing schematic component smart block.



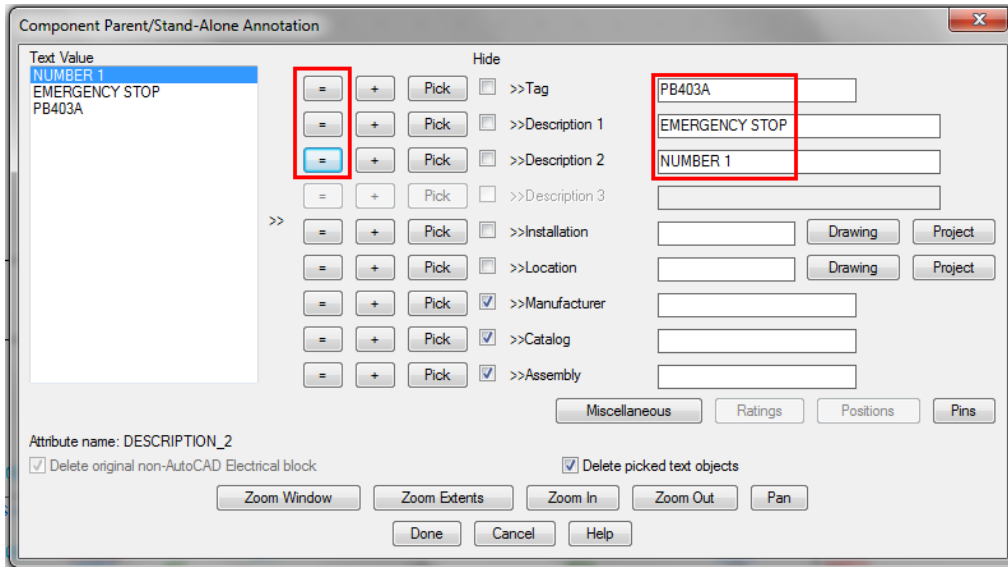
With this tool you pick an existing regular AutoCAD® block, the component that will replace it, then the existing text or attributes that are going to be mapped to the components attributes.

- Start the **Convert to Schematic Component** tool.
- Select the push button PB403A to be replaced.
- Select the desired schematic component from the schematic component menu:

EMERGENCY STOP
NUMBER 1
PB403A
—
o T o



- Place the schematic component over the existing block. *Note: Snap is helpful here.*
- Map the text and attribute values on the left of the dialog box to the appropriate component attributes by highlighting the text and clicking the = sign. The + sign can be used to add additional text to the same attribute.
- Map the following (See image below):
PB403A = Tag
EMERGENCY STOP = Description 1
NUMBER 1 = Description 2



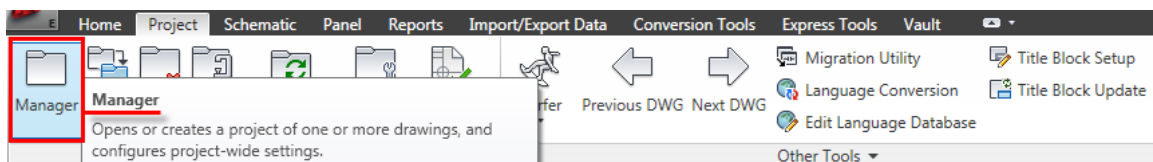
- Leave the other values blank. Click Done.

This tool will delete the original block by default. There is also a **Miscellaneous** button that shows additional component attributes. Once the component is in place the Edit Component tool can be used to add additional information to the component, such as catalog information, location, etc.

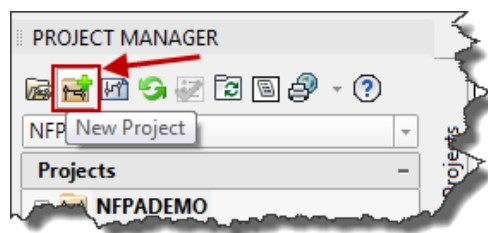
Working with Projects

Now the drawing is converted to a smart ACE drawing, but it is still just a stand-alone drawing. This is where AutoCAD® Electrical Project files come into play. Project files group drawings together. They know the location and order of the all drawing files so you can extract information from the drawings, such as running a report on all vendor part numbers in all the drawings.

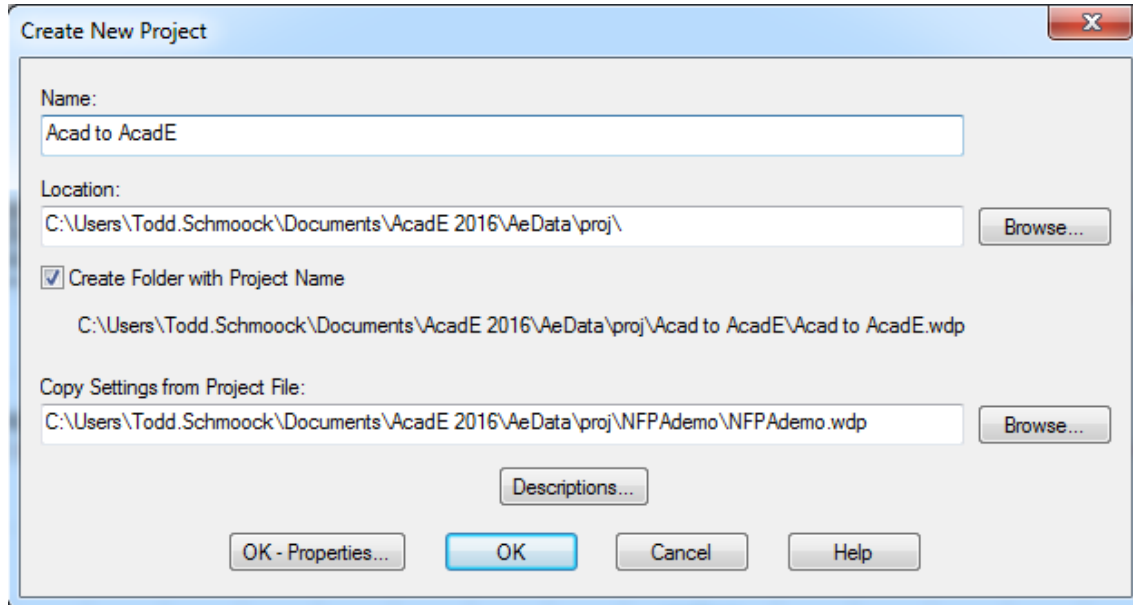
- If the Project Manager is not displayed you can activate it from the Project ribbon:
It is the first tool located on the Project ribbon:



- Select the **New Project** tool:



- Enter the name of your project and give it a location.
- Select **OK**.

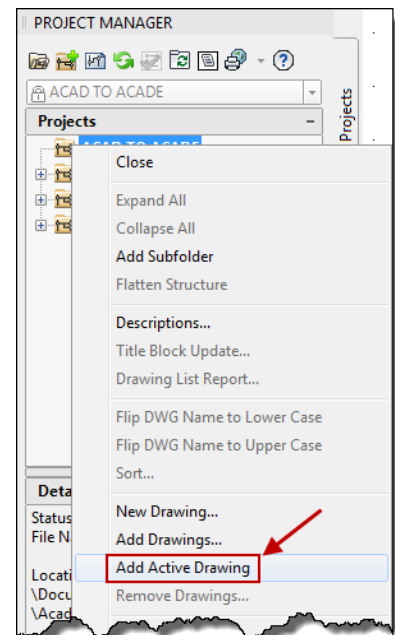
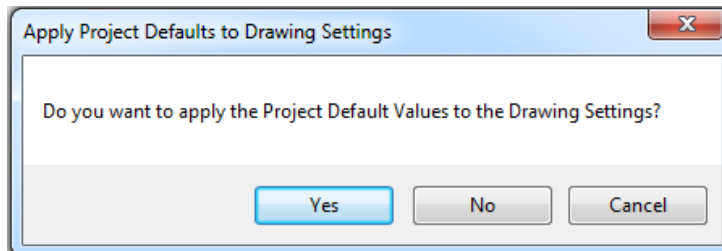


Your new project is listed at the top of the project manager and is bold to indicate it is the active project:

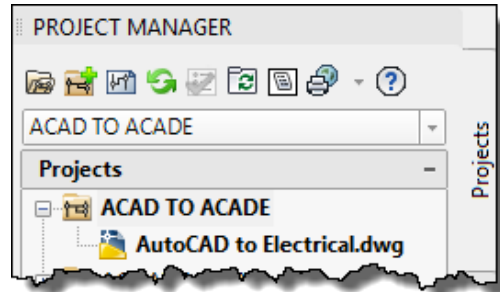
- Right-click on the project file
- Select **Add Active Drawing**

You will be asked if you want to “Apply Project Defaults to Drawing Settings”. Each ACE project and drawing have settings (properties) associated with it. Generally it is recommended to keep the project and the drawing the same and only deviate when necessary.

- Select **Yes**:



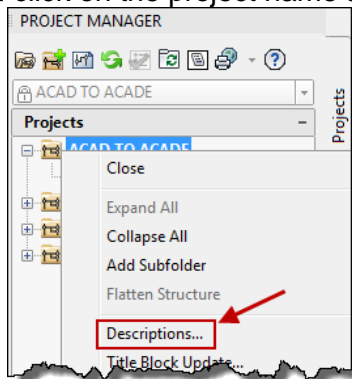
You will see the project file update and when you double-click on the project file you will see your drawing listed below the project:



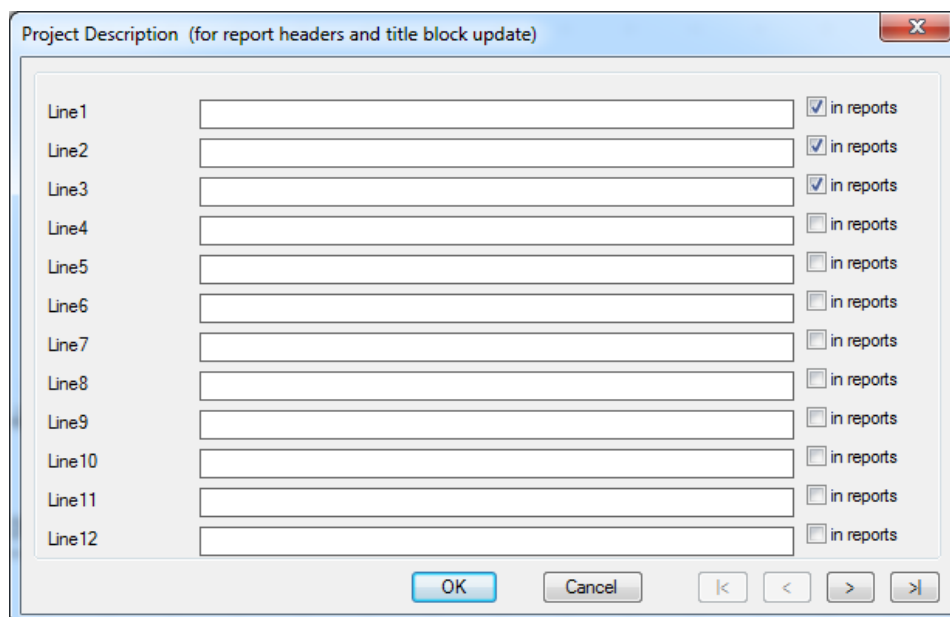
Adding Project Information to Existing Title Blocks:

AutoCAD® Electrical projects and drawings have data that can be used to populate and update title blocks and reports.

- Right-click on the project name and choose **Descriptions...**



The AutoCAD® Electrical project information is currently empty, but that will be changed shortly. The names for each line, such as Line1, Line2, etc, will also be changed to be better description labels. *Note: The check mark for “in reports” means the data on that line is available for use in reports such as Bills of Materials. It doesn’t affect the title block.*



To change the line titles we have to create a text file and list the lines we want to change, and then save the file in our project folder.

- Open Notepad
- Add the following:

LINE1=Title 1:

LINE2=Title 2:

LINE3=Title 3:

LINE4=Job Number:

LINE5=Date:

LINE6=Engineer:

LINE7=Drawn By:

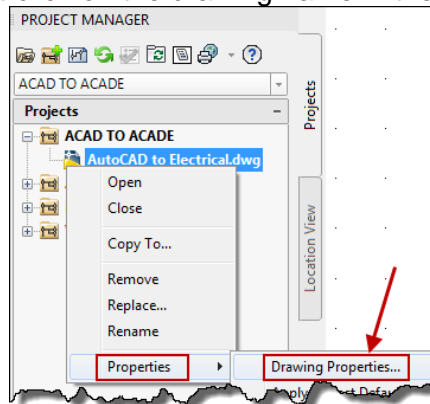
LINE8=Checked By:

LINE9=Scale:

- Save the text file to the project location with filename of *default_wdtitle.wdl*.

Keep Notepad open since we will be using it for our mapping file.

- Right-click on the drawing name in the Project Manager and choose **Properties...**



- Add the following information to the drawing properties:

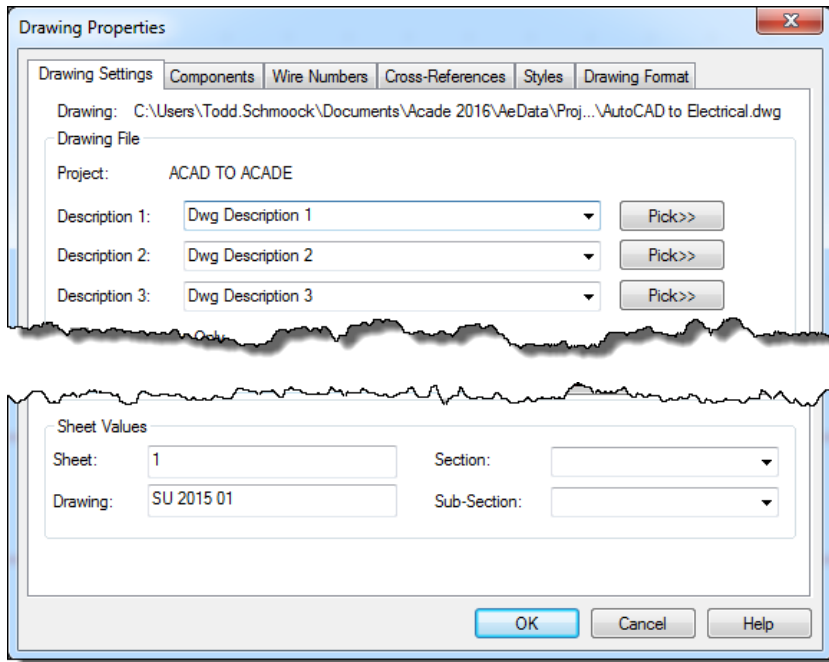
Description 1: = Dwg Description 1

Description 2: = Dwg Description 2

Description 3: = Dwg Description 3

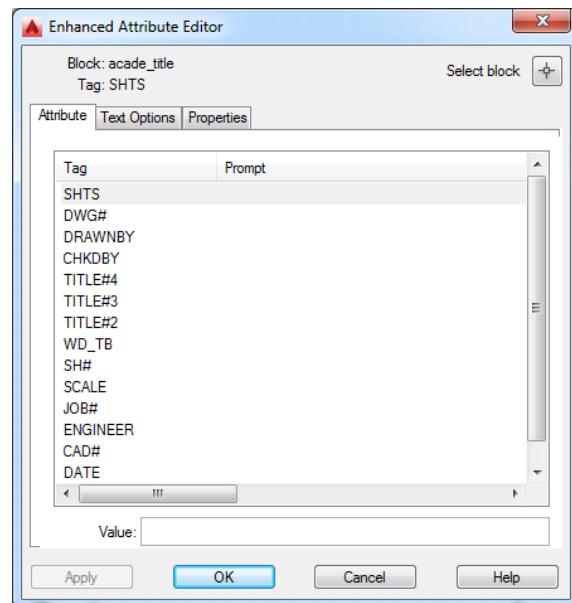
Sheet: = 1

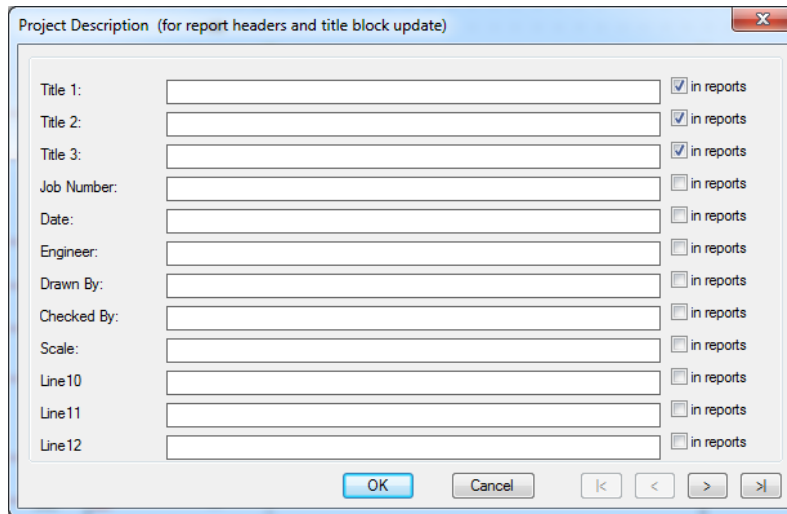
Drawing: = SU 2015_01



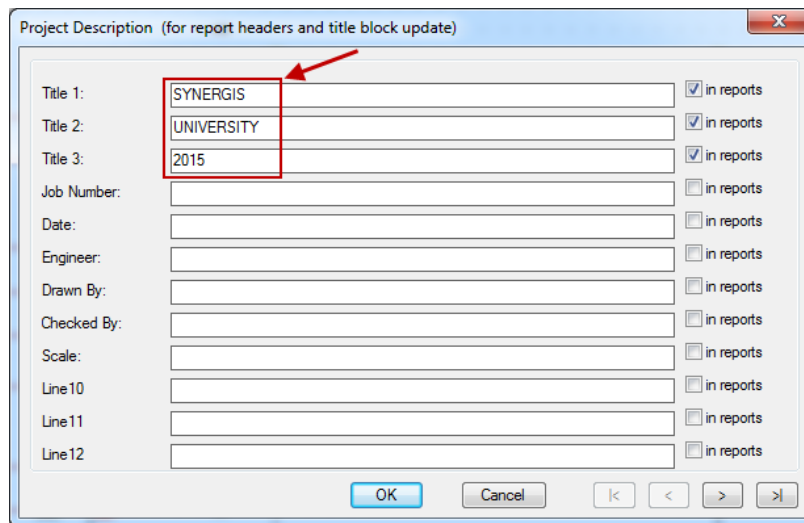
At this point the AutoCAD® title block isn't using the information stored in either the project descriptions or the drawing properties. The current title block attributes and their values are shown here:

- Right click on the project name in the Project Manager and select **Descriptions....** Notice the changes to the labels.





- Add Synergis to Title 1, University to Title 2, and 2010 to Title 3.
- Close the Project Description dialog.



Now you are ready to create the attribute mapping file.

- Go back into Notepad and open a new file.
- Add the following lines of text:

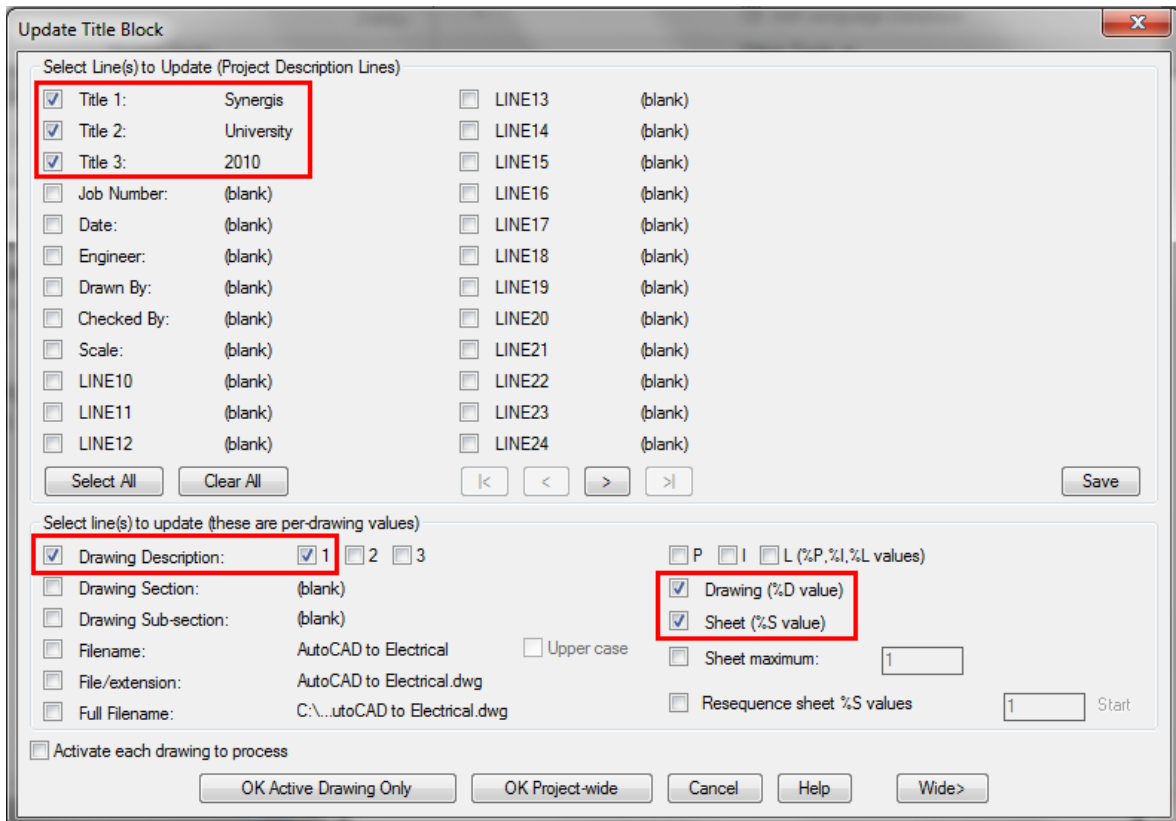
```
BLOCK = acad_e_title  
TITLE#1 = LINE1  
DWG# = DWGNAM  
SH# = SHEET  
SHTS = SHEETMAX  
TITLE#2 = DWGDESC
```

- Save it to the project location with filename of “*default.wdt*” and close Notepad.
- Zoom into the title block area. Right click on the project name in the project manager and select **Title Block Update...**:

AutoCAD® Electrical uses this text file to identify what block is being updated and what attributes get which ACE property value.

- The title block is named “*acade_title*”. This first line identifies the block with attributes and tells ACE that everything under it will identify the drawings attributes. The attribute names are on the left. On the right, after the “=”, is the ACE property name. So the attributes listed on the left will be populated by the ACE properties listed on the right.
- TITLE#1 is a title block attribute which will be populated project file’s description Line1 which was previously changed to “Title 1”.
- DWG# is a title block attribute which will be populated by the ACE Drawing Property DWGNAME (Drawing: in the drawing property dialog) to which you added AU 2009_01.
- SH# is a title block attribute which will be populated by the ACE Drawing Property Sheet Number (Sheet: in the drawing property dialog), to which you added 1.

The upper section of the Update Title Block dialog box shows the project description lines. You can see the Title 1, Title 2, and Title 3 values that were populated earlier. The lower section is the properties from ACE’s drawing properties.



Select the following check boxes:

Title 1; Title 2; & Title 3

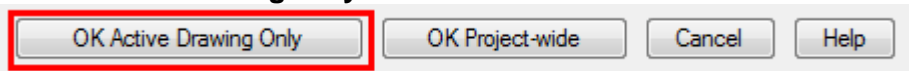
Drawing Description and 1

Drawing (%D value)

Sheet (%S value)

Note: You could select all check boxes in both the upper and lower selection since only values with information that are mapped will be inserted. You can run this option again if you need to make changes at a later date.

- Select **OK Active Drawing Only:**



Your title block is now populated with the values you added earlier and looks like this:

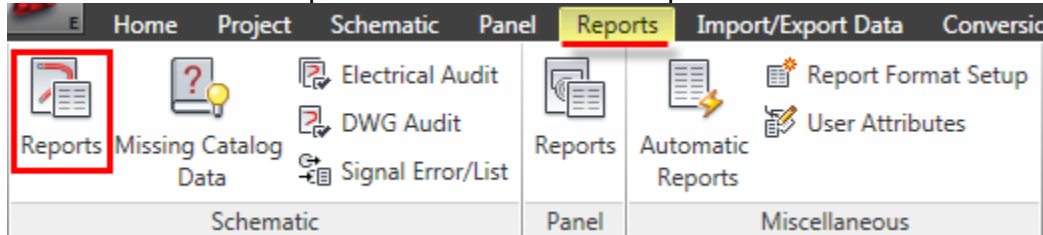
NO.	DATE	REVISION	BY
DWG TITLE			
SYNERGIS UNIVERSITY 2015 Dwg Description 1			
ENGINEER		CHECKED BY	
JOB NO		DRAWN BY	
SCALE		DATE	
DWG NO			
SU 2015 01			
SHEET NO			
1 OF			

Note: The total number of sheets value did not populate since we did not select "Sheet maximum" check box.

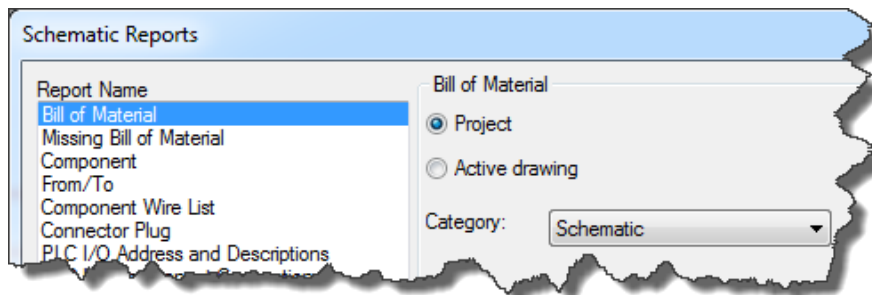
Running Bill of Material, wire connections, and other reports on the upgraded files

Now that you have converted the drawing you can run reports on the entire project.

- Start the Schematic Reports tool located on the Reports ribbon:

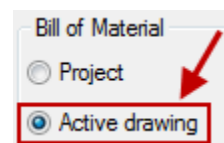


The Schematic Report dialog box has several options for running reports as shown in the upper left of this dialog box:

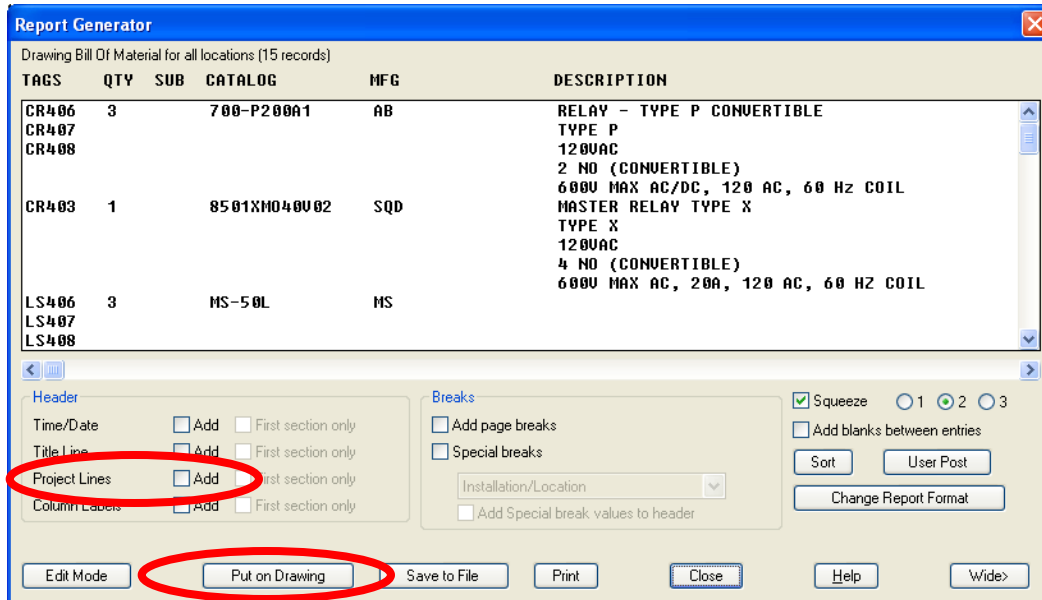


For this lab you will look at a couple of these options. Start with a Bill of Material report on the Active drawing.

- Select the radio button for **Active drawing**.
- Click **OK**.

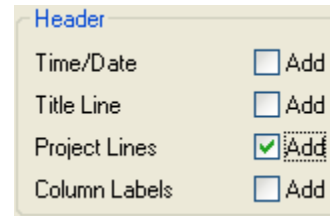


The Report Generator dialog lists the components:

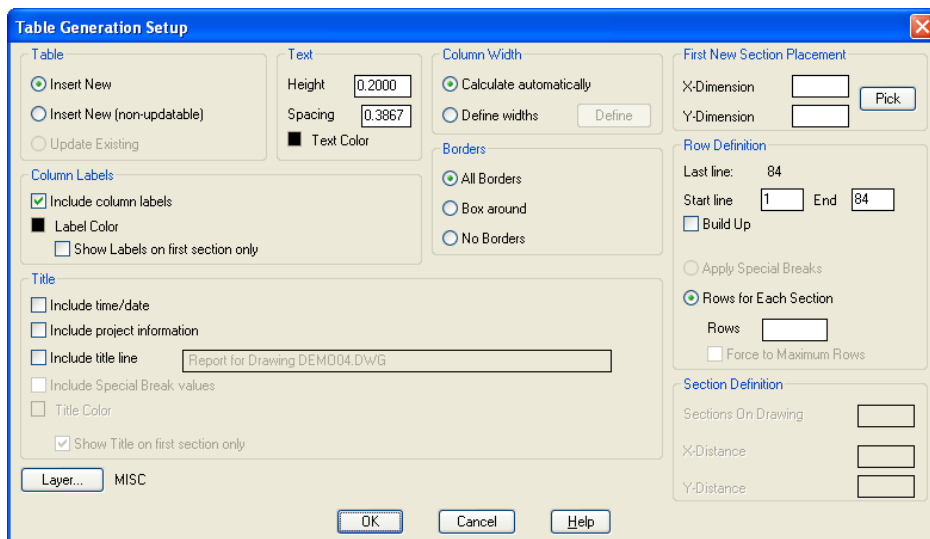


There are several options in this dialog box to format the information.

- Select the **Add** check box to the right of the Project Lines, in the Header section. This will include the project description lines that were added at the beginning of the lab.



- Select the button for **Put on Drawing**. You will get another dialog box which allows you to format how you want it to look on the drawing:

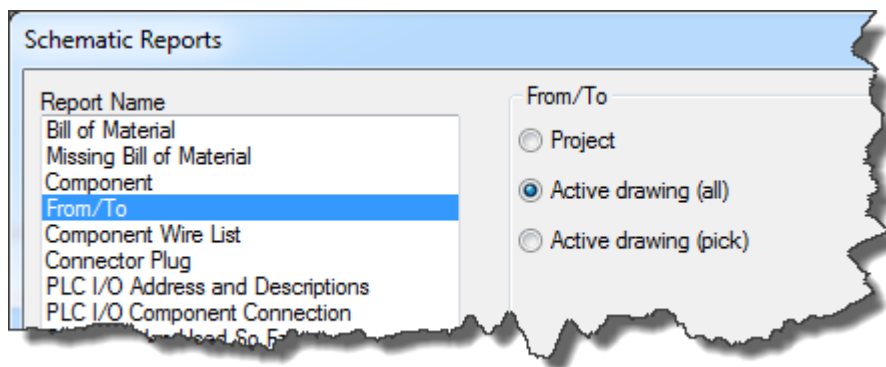


- Select **OK** to take all the defaults.
- Place the information to the right of the border.

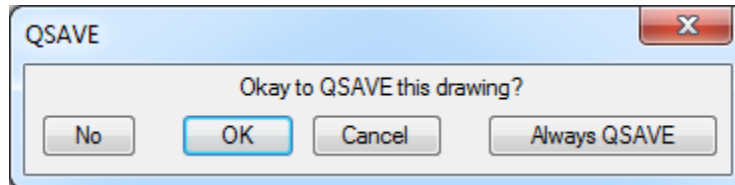
After placing the information on the drawing you will be placed back into the Report Generator dialog box.

Run another report and look at some of the formatting option available to you.

- Select the **Close** button at the bottom.
- Start the report tool again.
- Choose the **From/To** report.
- Select the **OK** button at the bottom.

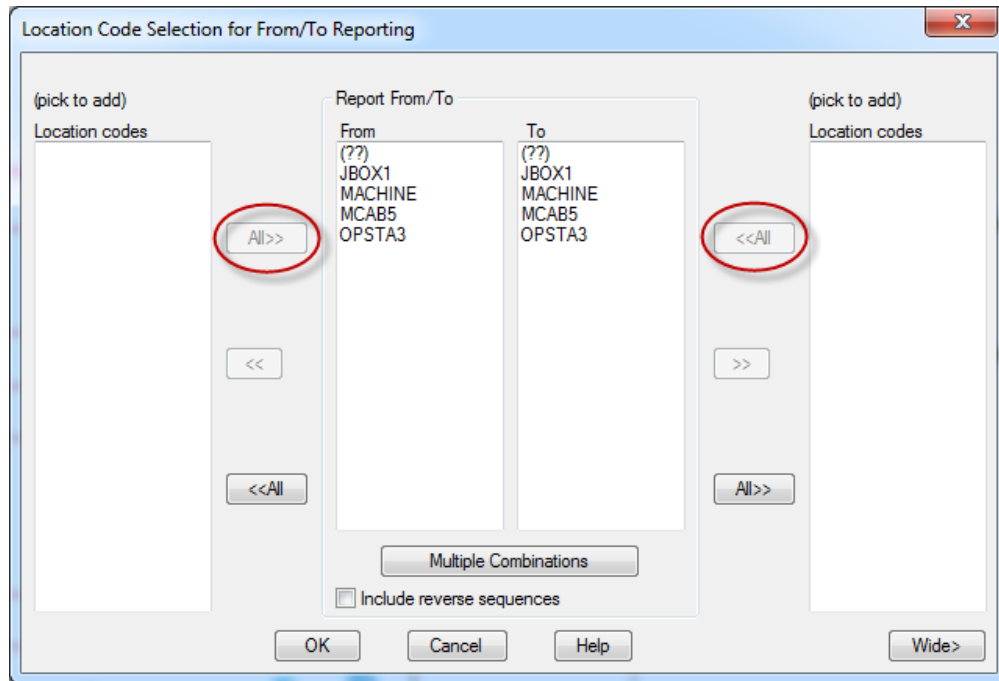


You may get a QSAVE dialog box. If you do select **OK**.



You are now given the Location Code Selection for From/To Reporting dialog box. This dialog gives the option to identify the Location codes added to the report.

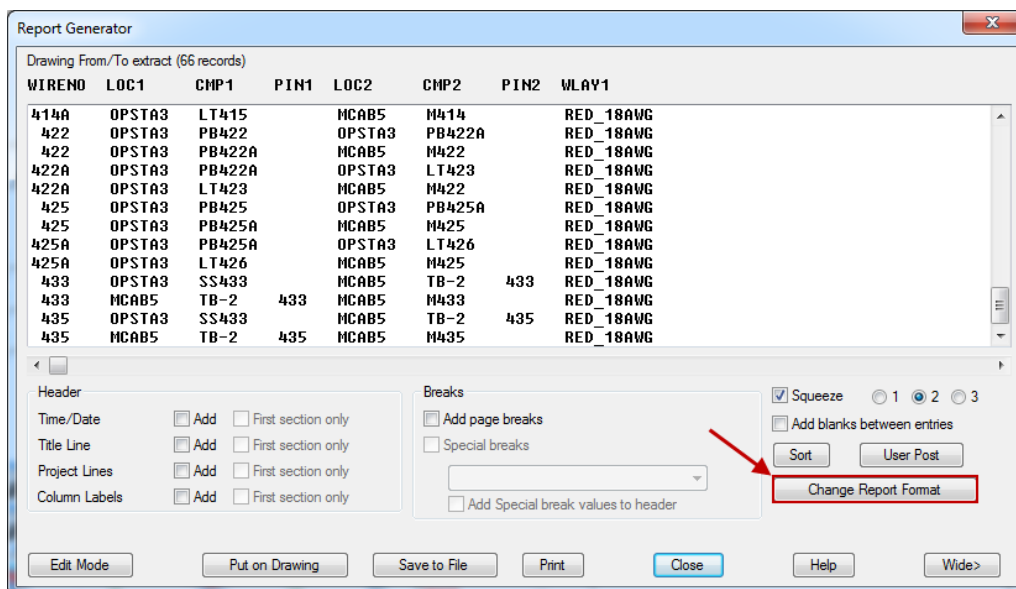
- Select upper **All** on each side of the dialog box.



- Select **OK**.

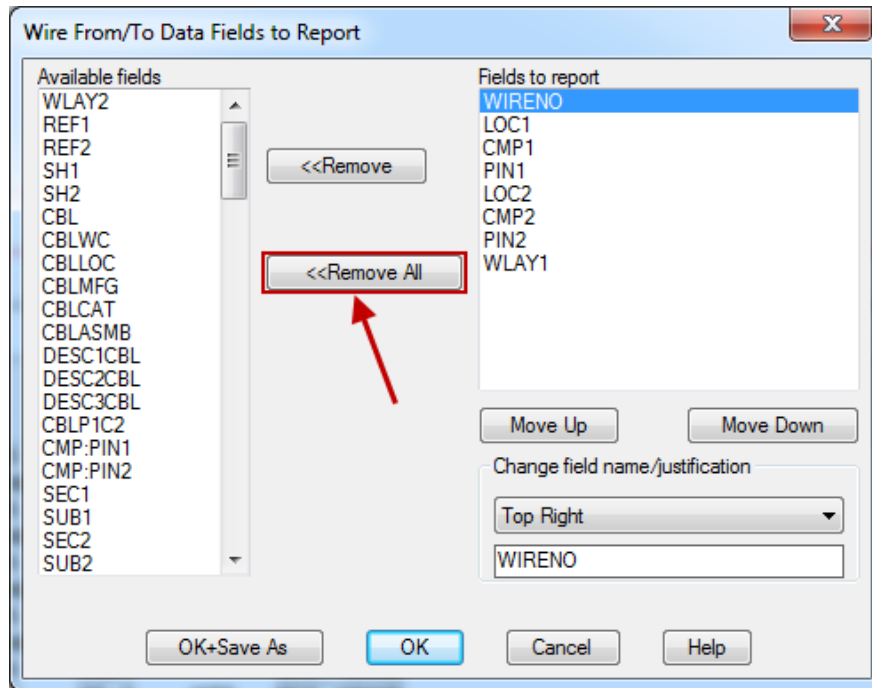
You will get the Report Generator dialog box showing where each wire is connected based on the location codes.

Note: Your dialog box may look different depending on if someone has modified the columns they wanted to see during an earlier session.

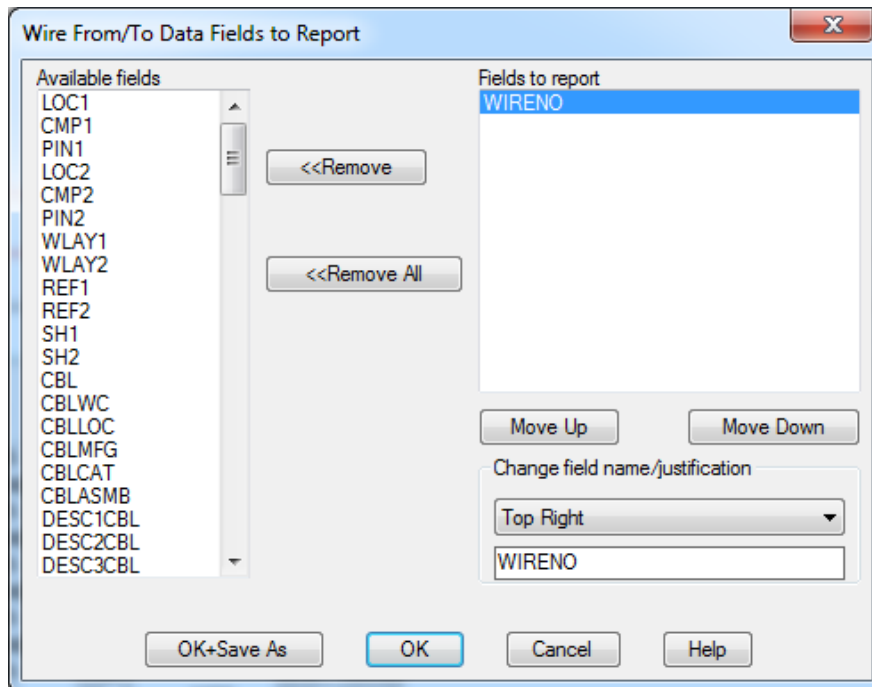


- Select the **Change Report Format**.

You will get the Wire From/To Data Fields to Report dialog box:



- Select the **Remove All** button.
- Add **WIRENO**.



- Select **OK**.

In Conclusion:

During this session you have learned how to take a schematic as it was drawn in regular AutoCAD® and add AutoCAD® Electrical intelligence to it. This included how to convert wires, convert wire numbers, make wire connections, convert blocks, create blocks, and add project information to existing title blocks. Then, bill of materials and other reports were generated based on this intelligent data.

While this was only done on a portion of a drawing the same techniques could be used throughout an entire drawing. Several of the tools included the option to update the full list of drawings in a project.