AutoCAD® Electrical: Unleash the Spreadsheet!

Nicole Morris – KETIV Technologies, Inc. (www.ketivtech.com)

EC315-1   Learn how to increase the power of AutoCAD Electrical using spreadsheets. Automate, populate, and stimulate your drawings with software you already know how to use. Learn the ins and outs of creating data outside and bringing it into Electrical. See how many designers use spreadsheets to plan a project and reuse this data effectively. Understand the fundamentals of these tools, allowing you to cut production time in half. Become comfortable with how spreadsheets are handled in AutoCAD Electrical, so that you can create your own versions. Samples and datasets will be included. Attendees should have basic knowledge of AutoCAD Electrical.

About the Speaker:

Nicole Morris has been an instructor for Autodesk manufacturing software for 10 years. She has spent the last four years helping companies implement AutoCAD® Electrical as their main productivity tool. Formerly a designer for a multi-disciplined design house, she is familiar with the day-to-day pain points experienced in documenting electrical controls and has helped users overcome these challenges. Currently she instructs basic and advanced classes at KETIV Technologies, Inc. for AutoCAD Electrical 2009 and Inventor 2009.

Email: Nicole.morris@ketivtech.com

Autodesk

AutoCAD Electrical has many ways to extract data from documented schematic and panel drawings. The more information you include while building your schematics, the more complete your data will be for installation and maintenance. More complete document has been proven to improve product development and customer satisfaction.

Much of this information can be created in spreadsheets and imported into your drawings. Information like retrofit and PLC I/O Addresses and Descriptions that is usually compiled in a spreadsheet before hand can be imported as well.

There is a ZIP file which contains several samples available for download on the AU website.

Interconnect Diagrams

You can define the type of connector, pin numbers, and catalog information in a spreadsheet.

The excel file must contain a table with the default 27 columns. The names of the columns in the table are not important, but the position is.
Sample included ImportConnector.xls
The sample contains

- 4 different size connectors with their pins descriptions and catalog data,
- 1 splice
- Wire Numbers, Color and Gauge
- Cable Information

1. To access this command, go to the pulldown menu, Components>Insert Connector>**Insert Connector from List**.
2. Select the file to insert.
3. Select one or more of the listed connectors and splices.
4. Define the type (plug/receptacle, etc.)
5. Select Insert and place on the drawing.
6. Select Wire It (be sure running Osnaps is turned off)

Importing Connector Pin Data

Since there is currently not a way with AutoCAD Electrical to fill out the pin descriptions in excel, I have found a way to do this with the AutoCAD ATTOUT and ATTIN command.
Of course this tool exports to a .txt format. However, with some quick tricks in Excel you can change the format to a structure more easily edited.
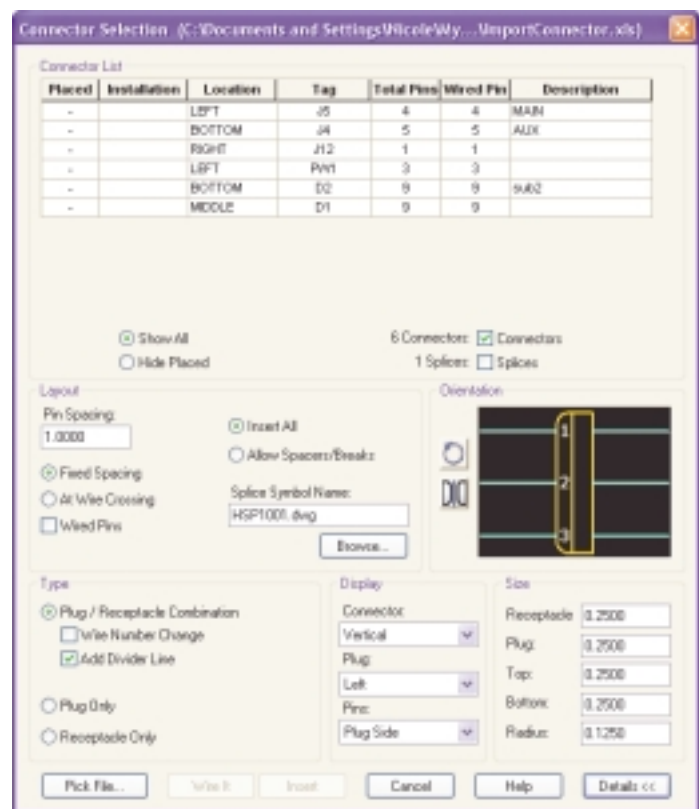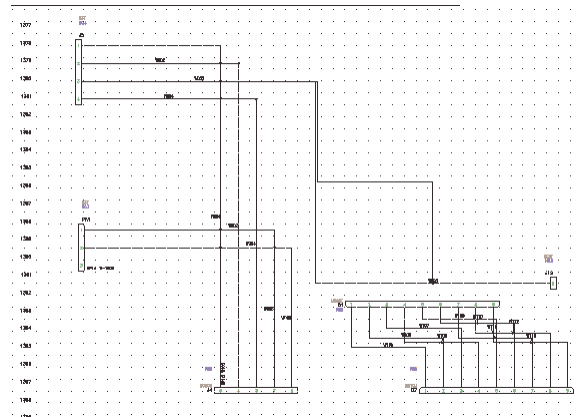
Export the existing Connectors

1. Open your drawing,
2. Type **ATTOUT** at the command line.
3. Select your block
4. Save the text file and watch where it saves the file.

```
Output file: "C:\Documents and Settings\Nicole\My Documents\YYYYYYYYYYY.txt"
created.
```

5. Open excel and go to File Open,
6. Change Files of type to All Files *.*
7. Select the text file created by the ATTOUT command.
8. Bring in as Tab Delimited.

You can fill out the spreadsheet as is, but it's a little difficult to keep track of in this format. So you can transpose the data from columns to rows.
(Transpose Data)

9. Highlight the rows of data including the header row and right click and select Copy.
10. Move to another area of your spreadsheet and right click and select Paste Special.
11. Check the Transpose option.
12. Delete the original data
There are only 2 mandatory fields, HANDLE and BLOCKNAME.
13. Highlight all but the top two rows
14. Go to the pulldown menu Data, select Sort, No Header Row, select Ok.
Since I'm only filling out the descriptions, I typically leave only the following.

- Handle (required)
- Blockname (required)
- Tag1 (helpful)
- Desc1 (helpful)
- Term Numbers (help you to know how many pins in a connector
- TermDesc* These will alternate Plug and Jack - I have created a template that allows me to group plug descriptions and jack descriptions. You can delete the descriptions and copy and paste the number of descriptions needed, the number needed is driven by the connector with the most pins.
- The rest of the fields can be removed if you desire.
15. Fill out the data
16. Transpose back to Columns and remove the rows. Repeat Steps 1 thru 4.

4
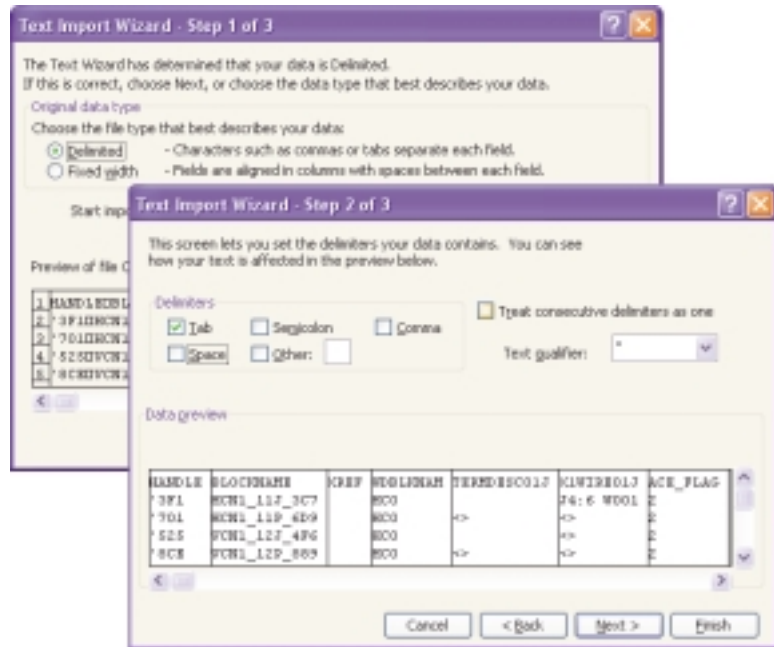
17. Save the file, excel will complain that it can't really save in that format, but it will save anyway. After you save, you can select No.
18. Open your AutoCAD Drawing and ATTIN, select the text file to fill out your connector descriptions.

Since this is an AutoCAD operation, you will need to rebuild the project database. In AutoCAD Electrical go to the Project pulldown menu, Go to Projects, Rebuild Project Database.

Samples Included ConnectorDescriptions.xls and SampleDescriptions.xls

Communicating with Inventor Routed Systems

You may have a need to create a virtual prototype of your wiring or wireharness. If you or someone you work with has access to Inventor Professional or Inventor Routed Systems, you can create a 3D representation of how your machine will be wired. The benefits include the ability to extract wire and cable length as well as show graphic nailboard layouts with this data.

Data that has been created in AutoCAD Electrical can be pushed to that model to determine connectivity, wiretypes, component reference tags and catalog information. This information can be modified in Inventor Professional or Routed Systems and sent back to AutoCAD Electrical to keep the documentation in agreement.

When creating your AutoCAD Electrical drawing, be sure that the wire types match the wiretypes available in the library in Inventor's Routed Systems or Professional. You can add the necessary Wiretypes in AutoCAD Electrical using the Create/Edit Wiretypes tool.

To export a report from AutoCAD Electrical in a format that can be read by Inventor Professional or Routed Systems, go to the pulldown menu, Projects>Reports>**Autodesk Inventor Professional Export**.

The information being exported includes
- Component Tag (ResDes)
- Pin Numbers
- Layer Name (Color and Gage of Wire)
- Wire Numbers
- Cable Information (Including Conductor Info)
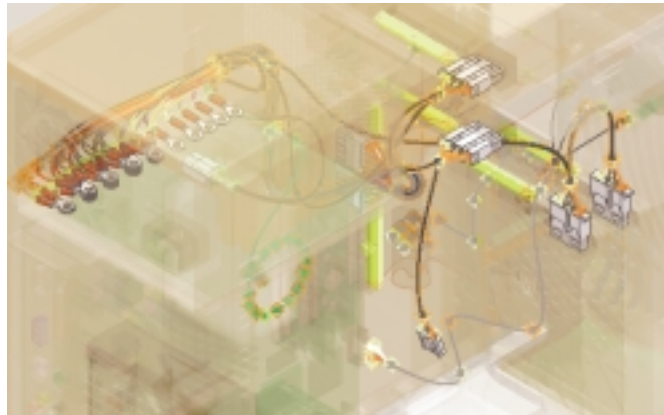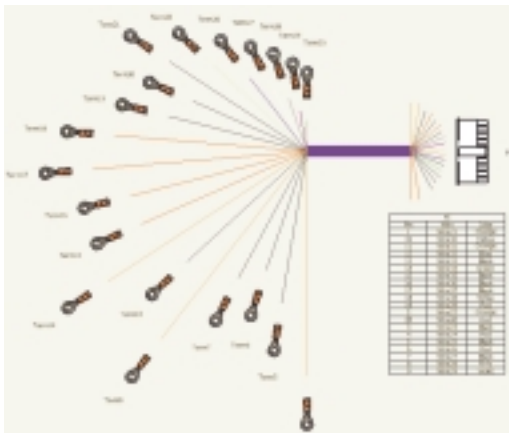
Assign Tags and wire numbers, be sure to have pin numbers that coincide with the model in Inventor. Inventor Professional does not allow you to have more than one wire per wire number, so while a wire going to through a terminal may have multiple wires with the same number, you may have to assign unique fixed wire numbers for these.

Update from Spreadsheet

As you create and modify your drawings in AutoCAD Electrical you can export data to excel, modify the spreadsheet and then Import that spreadsheet to populate attribute fields. Some great uses for this tool are:

- Ensure complete reports
- As-built and Retrofit Documentation
- Reuse of PLC data already laid out in Excel

The image to the right indicates the different type of information that can be isolated and exported. Here we will focus on three of the most commonly used.

- Components (parents only)
- Wire numbers and layers
- PLC I/O address/descriptions

General Procedure

1. To access this tool, go to the pulldown menu, Projects>Export to Spreadsheet>**Export to Spreadsheet**.
2. Select the type of data you wish to export.
3. Select to export from the Project or Active drawing, select OK.
4. Note the name of the file and the location to where it will be saved.
5. Access the **Update from Spreadsheet** from the same area.
6. Select the file to be opened, right click and Open with excel.
7. Modify the Data
8. Close Excel
9. You will still be in the Update from Spreadsheet dialog. Select type of file that you exported, select Ok.
10. The data will be incorporated into your drawing.
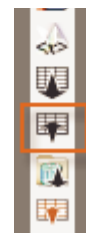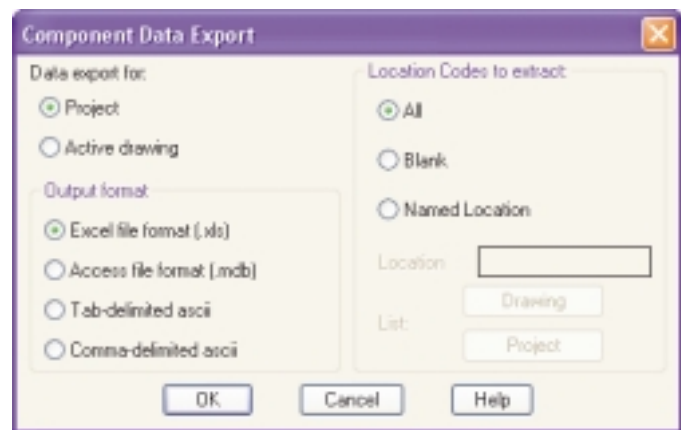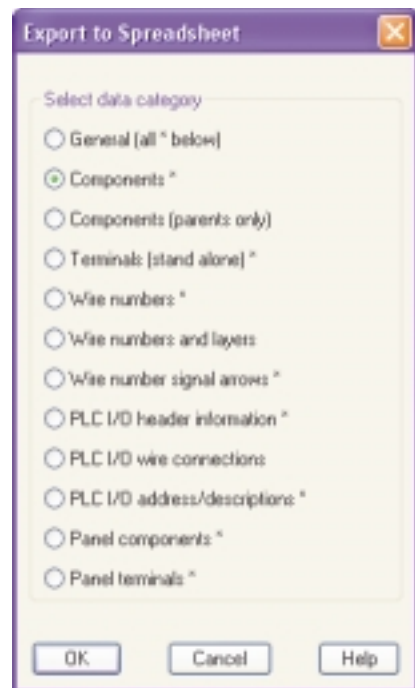
Rules and Guidelines
- Be sure to select the same export type, Project or Active drawing, on Export and Update.
- Consider these files temporary. Do not expect to use the same exported file 6 months from now, a lot can change.
- Do not modify the fields in parentheses like (HDL) and (FILENAME). This information is mapping for the data import.
- You can add miscellaneous or custom attribute names as headers to replace unneeded columns. An example would be COLOR for the lights.
- There is a limit of 255 column headers but less is ok.
- Its Ok to delete columns that don't have parentheses like the Ratings 12 ratings columns, often we use only 4. If expected columns are not there when we import, they are simply ignored.

- Ok to Sort all data, include the top row as header.
- Your spreadsheet edits can update existing or blank attribute values on existing components, terminals, PLC I/O modules, and wire numbers but it cannot insert new attributes into the drawings.

Components (parents only)

- You can modify block names to swap out blocks
- You can enter Catalog information to complete your bill of materials
- Modify Tags, new values will be fixed

Wire numbers and layers

- Modify Wire numbers, new values will be fixed
- Modify Layers, for pre-defined as wire layers.
- New layers can be added, but they then must be added as wire layers in the Create/Edit Wire Type tool.

PLC I/O address/descriptions

- Copy and Paste existing lists of I/O addresses and descriptions.

Copy and paste default wire layer info into dialog

I have had many ask me if there was a way to automatically import a list of layers while maintaining the information in the Color Size and User fields.

It is true that it is best to maintain this list in your template. You can also Copy and Paste form the **Create/Edit Wiretypes Tool** of one drawing, the same dialog box in another. It is also true that you can create a list in Excel to main different types of wire layers, including 3-phase, main controls, etc. Then copy and paste only the wirelayers that you need into the **Create/Edit Wiretypes** Tool.
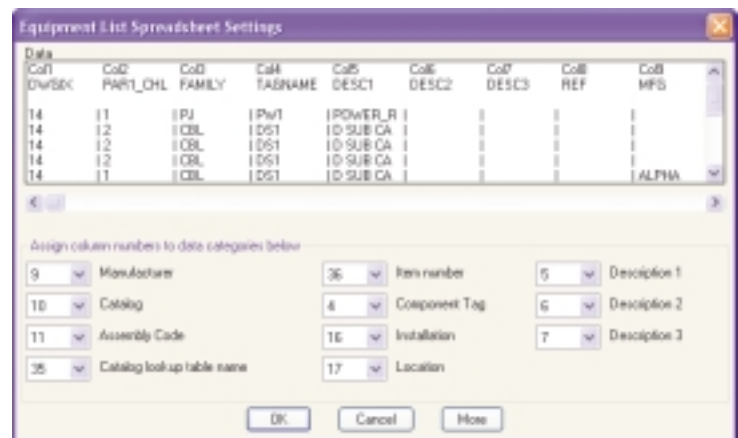
Insert Component (Equipment List)

When you start a project, you may be provided with an equipment list that includes which components and materials you will need to complete this job. This is a great tool to use if you are documenting an existing machine.

Build an excel spreadsheet that includes the important information about your components including catalog data, tag information and descriptions.

Column names are not very important as you will be able to map them to the appropriate place, however, if you use certain names very little mapping will be necessary.

Sample Included Equipmentlist.xls

1. To access this tool, go to the pulldown menu Components>Insert Component (Lists)>**Insert Component (Equipment List).**
2. Select the file where your equipment is defined.
3. Select Read Default Settings or Read Settings you have previously saved.
4. Select Spreadsheet/Table Columns to map which column applies to which field, select OK.
5. Save Settings, select OK.
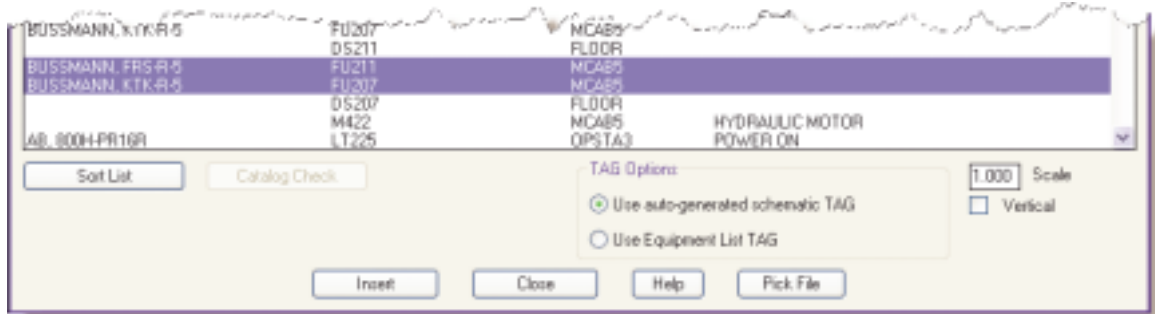
6. Select components you want to import and select Insert.
7. You will be asked which schematic symbol to use.

> Tip: You can determine in advanced which icon will be used for which catalog number by adding entries to the **Schematic Database File Editor**. Provide the manufacturer, catalog number and which block to use.

Spreadsheet to PLC I/O Utility

While documenting PLC I/O modules in the typical workflow of AutoCAD Electrical greatly streamlines the task, you can use a spreadsheet to build the entire diagram which further reduces the time it takes to document PLC I/O. Of course it is necessary to take the time to learn how to use this tool and understand exactly how the process is controlled.

To build a spreadsheet that can document your PLC Modules it is helpful to first understand the layout and where the settings are coming from.

*   **Spreadsheet to PLC I/O Utility Settings** determines the style and position of the modules and their components.
*   **PLC Database Editor** determines the position of the module on the ladder and also dictates how each point on the module will behave.
*   **The Spreadsheet** can be an Excel Spreadsheet or an Access file. This file determines, which module is used, attribute values and components on the Input or Output side of the module.

Understanding the Tool

A sample spreadsheet is provided with AutoCAD Electrical which describes 6 modules and their information. It is helpful to use this sample to understand the basics.

1. To access this tool, go to the pulldown menu Components>Insert PLC I/O Modules>**Spreadsheet to PLC I/O Utility.**
2. Select the *demoplc.xls* file with a single click. Right click Open to open the spreadsheet in Excel.

Line 2   Module Info and first rung information
Line 3 thru 19   I/O data including address, description and
                              components on the input or output

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
1	CODE	R1	S2	G3	ADDR	RTP	DESC1	DESC2	DESC3	DESC4	DESC5	VOLTAGE	D1TAG	D1DESC	D1BLK
2	1771-OAD	1	2	0	I:002/00	TB20	BANK #1 FIBER	WASTE REMOVAL BLOWER	START HS100BR	REMOTE		120VAC INPU	TB1		HT0001
3					I:002/01		BANK #1 FIBER	WASTE REMOVAL BLOWER	STOP HS100BS				TB1		HT0001
4					I:002/02		BANK #2 FIBER	WASTE REMOVAL BLOWER	START HS1001BR	REMOTE			TB1		HT0001
5					I:002/03		BANK #2 FIBER	WASTE REMOVAL BLOWER	STOP HS1001BS				TB1		HT0001
6					I:002/04		TUB OUTLET TEMPERA	NOT LOW(ALARM)					TB1		HT0001
7					I:002/05		TUB INLET TO HEATER	NOT LOW (TRIP)					TB1		HT0001
8					I:002/06		SPARE								
9					I:002/07		COMBUSTION BLOWER	RUNNING					TB1		HT0001
10					I:002/10		INSTRUMENT AIR PRES	NOT LOW (IPS)					TB1		HT0001
11					SPACER										
12					I:002/11		AIR DAMPER	AT HIGH FIRE (>80%)(PFS)					TB1		HT0001
13					I:002/12		AIR DAMPER	AT LOW FIRE (<20%)(LFS)					TB1		
14					I:002/13		PURGE AIR FLOW	NOT LOW (PAS)					TB1		HT0001
15					I:002/14		COMBUSTION AIR PRE	NOT LOW (AS)					TB1		HT0001
16					I:002/15		SPARE								
17					I:002/16		STACK TEMPERATURE	NOT HIGH					TB1		HT0001
18					SPACER										
19					I:002/17		STEAM PRESSURE	NOT LOW (>20PSIG)					TB1		HT0001
20															
21															

Close Excel file and select Open in the PLC I/O Dialog.

3. Select the Setup Option on the top right.

- Select your company's template here, but be sure to change the Ladder origin point so that each new sheet has the correct start point.
- Remap the spreadsheets if your spreadsheet columns are not in the same order.

4. Select Save and give the settings file a new name. In the future, this can be retrieved without having to open the settings dialog.

Spreadsheet to PLC I/O Utility Setup

Ladder

Origin:
Pick>>
X: 1.500
Y: 21.500

Orientation:
⦿ Vertical
◯ Horizontal

Reference numbers:
Numbers Only
Width: 9
Distance between: 15.000
Ladders per drawing: 2
Rungs per ladder: 21
Rung spacing: 1.000
Rung count skip for I/O start: 1

Suppression:
☐ Rungs
☐ Side bus rails
☐ Do not erase unused, blank rungs

Signal arrow style: 4

Module
PLC graphical style: 1
Input offset from neutral: 1.000
Output offset from hot bus: 1.000
Maximum I/O per ladder: 19
I/O point spacing: 1.000
Scale: 1.000
◯ 1.0   ◯ 16   ◯ 25.4   ◯ 0.039
☑ Apply this scale to module outline only

In-Line Devices
First input device from hot bus: 1.000
First output device from neutral bus: 0.250
Spacing between multiple devices: 0.750

Spreadsheet/Table Columns

Drawing template:
ACAD_ELECTRICAL   Browse...

Save   OK   Help

5. Select Ok, to close the Setup dialog and return to the utilities main dialog.

6. Enter the starting rung number
7. Enter column to column count
8. Enter Drawing to drawing count
9. Starting file name or use the activate drawing. If more sheets are need the utility will automatically use the template and start point defined in Setup Dialog.
10. Select Free Run
11. Check Add new drawings to the active project.

PLC Module Database Editor

Module Position

Position of the PLC Module in the ladder is determined in the **PLC Database Editor**.

To access this tool go to the pulldown menu>Components>Insert PLC Modules>**PLC Database Editor.**

Open the PLC Database Editor and choose Module Specifications. You can select the following options:

- **Center** - Centered between the two bus lines
- **Left/Top** -Inserted near the left bus (top for IEC).
- **Right/Bottom** - Inserted near the right bus (bottom for IEC).

Note: The distance from the bus is determined in the settings of the Spreadsheet to PLC I/O utility.

Each line in the module determines whether the rung is an input, output or terminals. Only input and output, spacers and breaks can have components on them. Terminal types will draw the rungs but will not include any components.

Building the Spreadsheet

After getting acquainted with the basics, there are some tools that can help you to get the desired results.
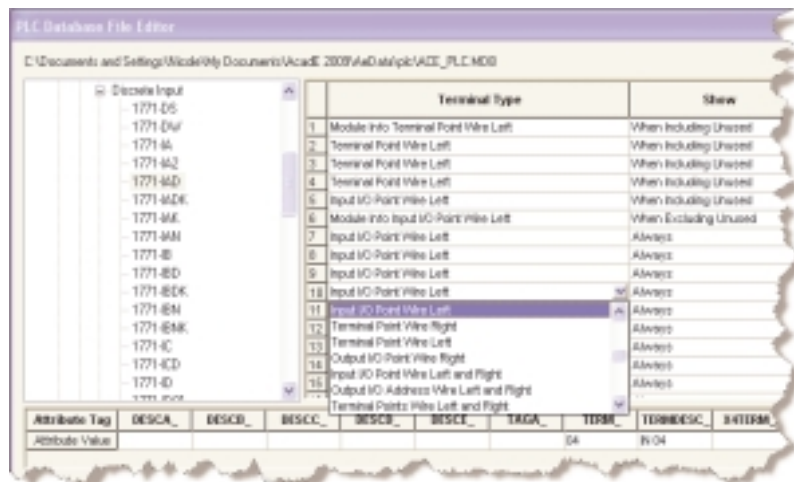
Special Codes

- **BREAK** - Enter BREAK in the Address Column and it will force the PLC Module to break at this point. Setting on the import dialog will help control whether that break continues after skipping rungs or moves to the next ladder. You can add components on this row.
- **SPACER** - Enter SPACER in the Address Column and it will stretch the PLC Module at this point while still adding a rung. You can add components on this row.
- **SKIP** - Enter SKIP in the Module Column and the next module will skip a whole ladder before adding the module to the next ladder. Unfortunately you cannot add components without a PLC Module so you cannot add components into the blank ladder.

More Attributes

You can fill out more attributes than the ones available on the spreadsheet.
For the rack column, enter rack number then subsequent attributes and their values separted by semicolon.   **3;LOC=MACHINE;DESC1=PALLET**

Entering Components

There are four fields to describe components on the rungs. In each group of four, the column headers will change.   Where *n* is the position on the rung
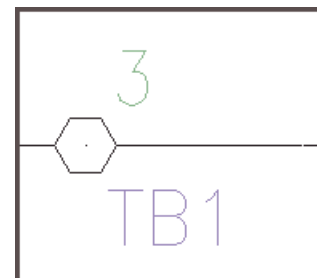

D*n*TAG   D*n*DESC   D*n*BLK   D*n*LOC

- **D*n*TAG** - Enter the tag name, the program does not pick up the TAG number automatically, only the family name.

A terminal number can be added to the tagstrip name while separated by a colon in the D*n*TAG field.

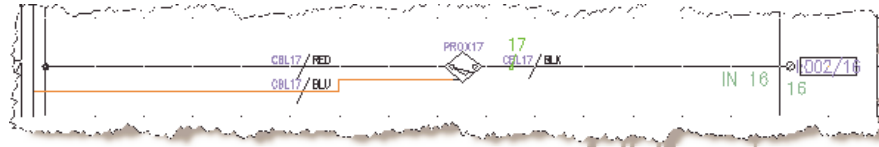**TB1:3**   results

- **D*n*DESC -** Description is Optional.   However, the DnDESC field you can fill in multiple description attributes by using the pipe "|" symbol.

EMERGENGY|STOP   EMERGENCY
                                                                   STOP

- **D*n*BLK -** Enter the block name in this field. It will insert the blocks as long as they can be found in the project path.

Insert Circuit -
Enter an asterisk in
front of a symbol device
block name.

- **D*n*LOC -** Enter the location of the component. This attribute is optional.

Component Spacing

Up to 9 components can be placed on a single rung. The distance from the bus and the spacing between each component is defined in the Spreadsheet to PLC I/O Setup.

Components on Input wires insert from the left to the right.
Components on Output wires insert from the right to the left.
If the ladder is too narrow, your components will run into the PLC module.

It is helpful to plan how you want your components spaced. I have included 3 grids that can help you plan. PLC Position-center.dwg   PLC Position-left.dwg   PLC Position-right.dwg
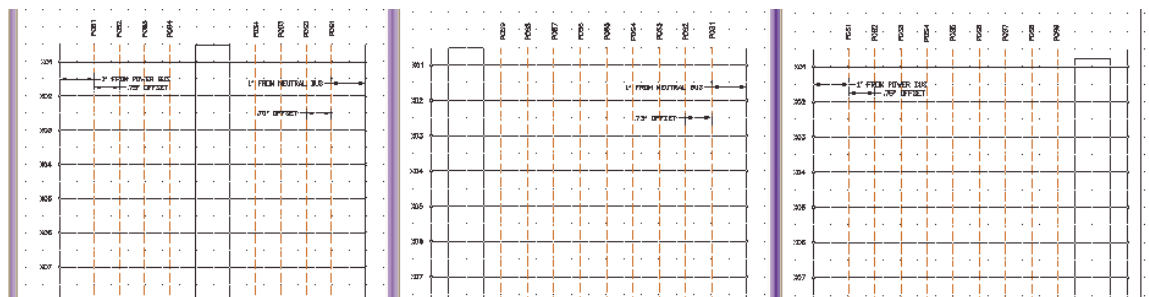
Wire Jumpers

Wire jumpers can be created by placing a special code in one of the 9 component positions. The code is placed at the top of the jumper each time. It starts with a | symbol followed by 4 letters **W** for connection and **X** for no connection.

Leave the component field below empty unless you want to jumper into another component or terminal.

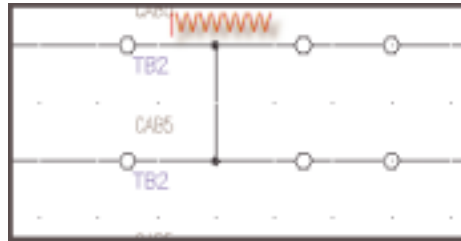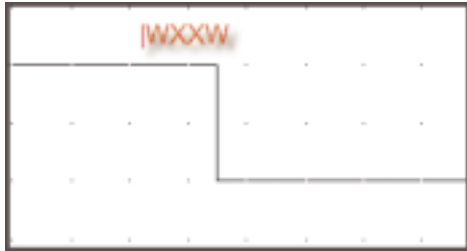| AK	AL	AM	AN	A
D7TAG	D7DESC	D7BLK	D7LOC	D8TA
		MWWWW		
		HT0003		

12
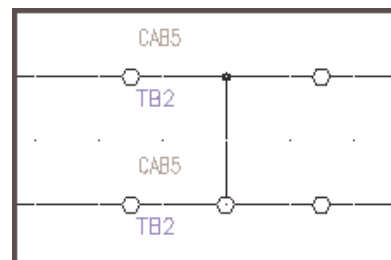
Each of the 4 letters signifies a position
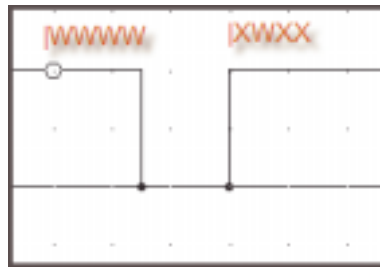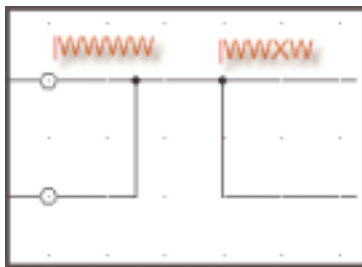- 1st char Top Left
- 2nd char Top Right
- 3rd char Next rung bottom left
- 4th char Next rung bottom right

Here are some images that show the effect of different combinations.
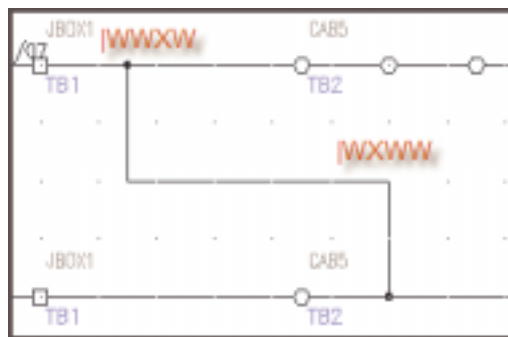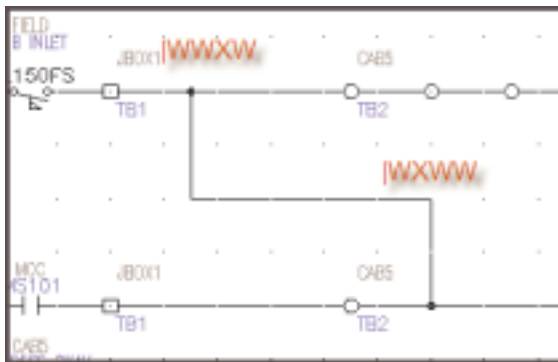
- One entry in the spreadsheet

- Two entries in one row

- One entry in one row, one entry in the next row

I have included an example of some of these advanced options to help you in building the diagrams the way you want SStoPLC.xls
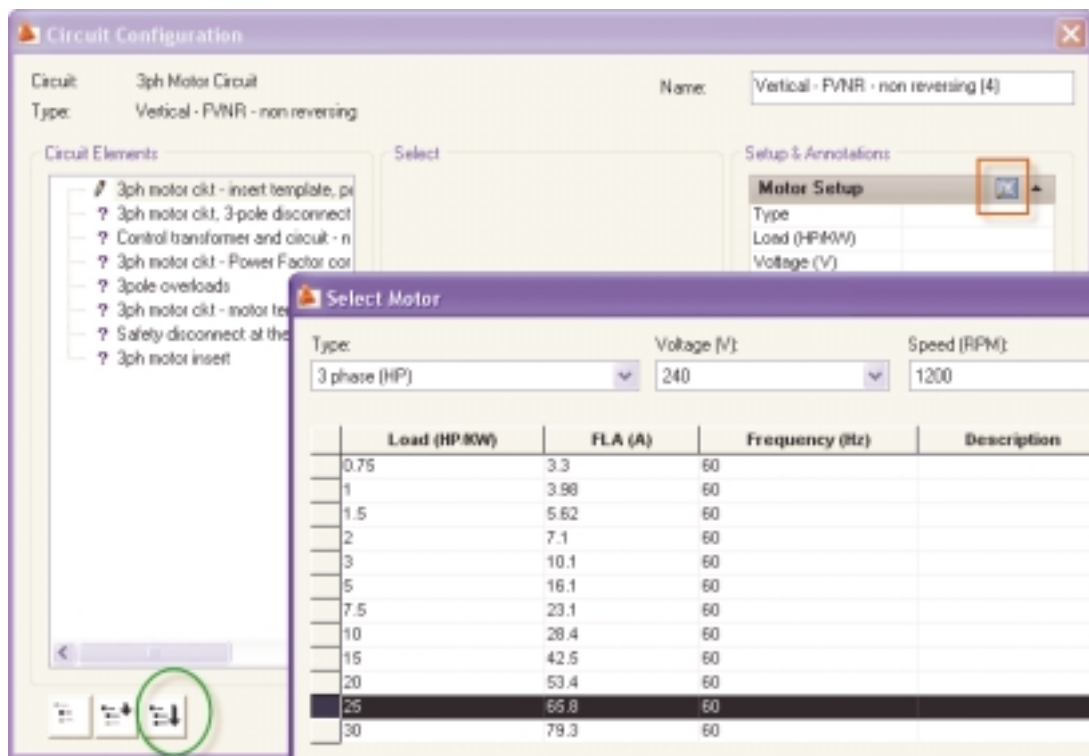
Circuit Builder

The Circuit Builder is the first rules-based system that has been added to AutoCAD Electrical. You can define your 3-phase motor control layout as desired, streamlining the process to achieve a faster documentation.
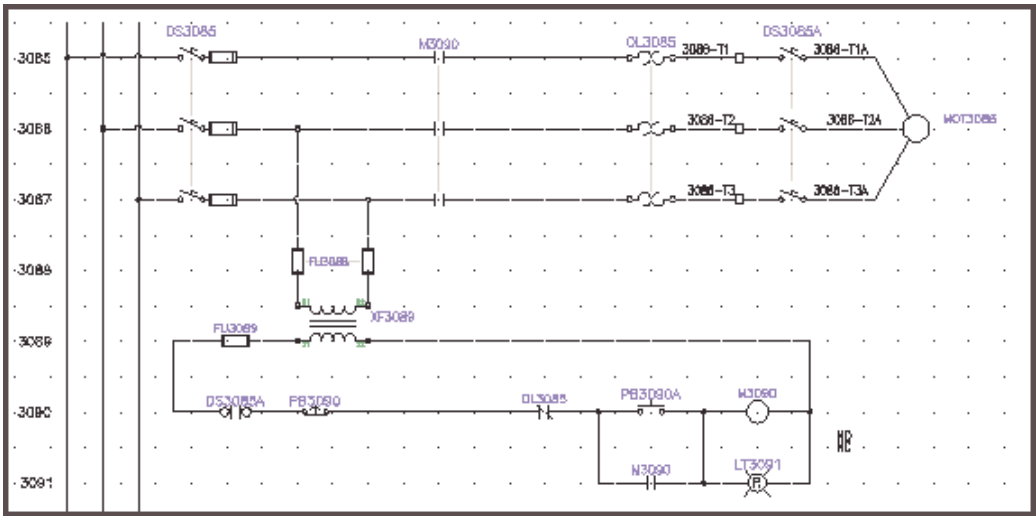
Using the Circuit Builder

We will first start by understanding how to use the tool as it is installed. Later we will learn how to customize and adjust to our own company's standards.

To access this tool, go to the pulldown menu, Components>**Circuit Builder**.

1. Select the type of MCC that you wish to build. You can use the history to find previously used MCC circuits once you have used and save the configuration.
2. Select Configure.
3. Select the first bus, (top bus for vertical MCC, left bus for horizontal MCC)
4. In the Circuit Elements area, select the first line
5. In the Setup & Annotations area, you can drop down and select each line individually or you can select the Configure Motor Options button, highlighted in red on the image below.
6. Select each line and select the configuration for each.
7. When you are satisfied with your input, select the larger arrow below to insert the entire circuit.
8. When the circuit is complete, the dialog box will return. You can enter a name in the upper right corner and select Done. This will save the configuration to the history.

Customize the Circuit Builder

After using the Circuit Builder, you may want to add additional information that your company commonly uses. It helps to first understanding the elements involved. Making small changes and displaying the effects is a helpful way to understand each of these tools.

- **Circuit Builder Spreadsheet** contains the selection options available in the Circuit Builder Dialog box, including choices and default entries
- **Drawing Templates** contains place holders for the placement of each device. There are several samples provided in the AutoCAD Electrical install.
- **Annotation Database** contains data for specific component information including annotation like ratings.

Circuit Builder Spreadsheet

This spreadsheet, called ***ace_circuit_builder.xls***, can be found in following folder:
- Windows XP: C:\Documents and Settings\All Users\Documents\Autodesk\AcadE {version}\Support\
- Windows Vista: C:\Users\Public\Documents\Autodesk\AcadE {version}\Support\

	A	B	C	D
1	CATEGORY	TYPE	DWG_TEMPLATE	SHEET_NAME
2	3ph Motor Circuit	Horizontal - FVNR - non reversing	ace_cb1_FVNR_H.dwg	3ph_H
3		Horizontal - FVR - reversing	ace_cb1_FVR_H.dwg	3ph_H
4		Vertical - FVNR - non reversing	ace_cb1_FVNR_V.dwg	3ph_V
5		Vertical - FVR - reversing	ace_cb1_FVR_V.dwg	3ph_V
6				

\ACE_CIRCS / 3PH_V / 3PH_H /

Ready

The Circuit Builder Spreadsheet contains three tabs.

- ACE_Circs - Maps the type to a specific drawing template and which of the two default tabs is being used. Rather than relying on the history to build your circuit. You can create your own custom type. Described the TYPE, indicate which drawing template to use and which of the tabs are related.

	A	B	C	D	E	F	G		
1	CODE	COMMENTS	UI	UI_TITLE	UI_PROMPT_LIST	UI_VAL	UI_!	COMMAND_LIST	ANNOTATE_
2	MC01	3ph motor ckt - insert template, prompt f		!MCC_CTRL	3				
18	Q001	3ph motor ckt, 3-pole disconnecting mea		Main Disc	Circuit Breaker	2			
19			X		Fused Disconnect	4			
20					Disconnect switch (non-fused)	6			
21					Circuit Breaker-Thermal	8			
22					Circuit Breaker-Current Limiter	10			
23					Fuses	12			
24					None	0			
25									

ACE_CIRCS \ **3PH_V** / 3PH_H /

- 3PH_V - Displays the options available in the Circuit Builder Dialog box when selecting a vertical layout for your MCC.
  - An "X" in the UI__DEF column indicates which is the default.
  - The CODE column is mapped to the code for the attribute values in the drawing template for each marker block. Each code represents an action that takes place at the insertion of that marker block.
- 3PH_H - Displays the options described above for the horizontal layout for MCC.

A few of the many things you can modify:
- Change the default
- Add additional options
- Enter pre-determined annotation for specific components.

Drawing Templates

The drawing template contains marker blocks whose insertion point indicates a point of action by the spreadsheet. Each block contains a value that is a code that corresponds to a line in the ***ace_circuit_builder.xls.***

There are several samples provided with each of AutoCAD Electrical's libraries (i.e. JIC125)
ACE_CB1_*.dwg
ACE_CB2_*.dwg

While the naming convention is not important, it keeps them together. Also their names are indicative of their behavior.
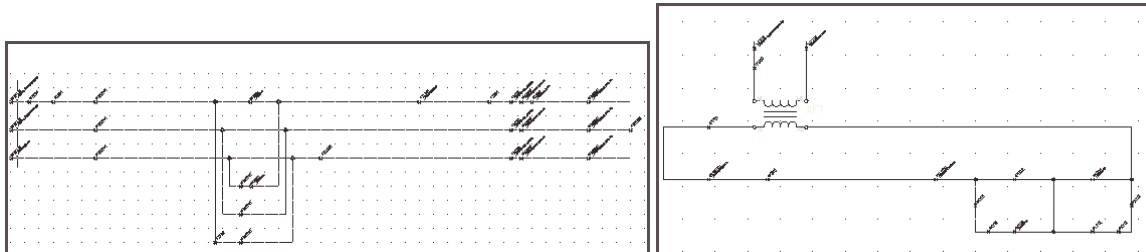
ACE_CB1_*.dwg type are main layout formats.
ACE_CB2_*.dwg type are extensions from the original layout

- Marker Blocks – The marker block must be named **ace_cb_marker_block**   The block contains three attributes whose tags are:
  - o **Code** – matches the CODE column in the spreadsheet
  - o **Order** – determines the sequence of the actions
  - o **Misc1** – contains additional information tags, descriptions, etc.

These blocks can control insertion of single or multi-phase components, assign wire types, include wire numbers, branching, and annotation.

A few of the many things you can modify:
- Change the behavior by code
- Change the order or adjust the order for added components.
- Enter pre-determined descriptions.

Annotation Database

The Circuit Builder can annotate the circuit elements using their the MCC database MCC.mdb or your catalog database.
- Windows XP: C:\Documents and Settings\{username}\My Documents\Acade {version}\AeData\Catalogs\
- Windows Vista: C:\Users\{username}\Documents\Acade {version}\AeData\Catalogs\
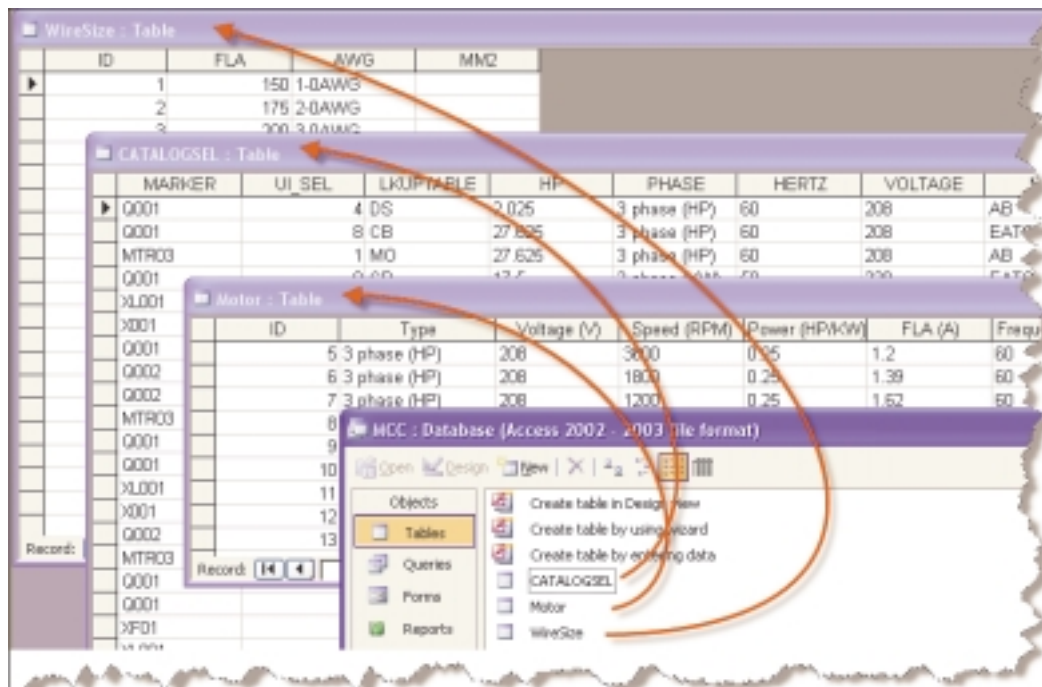
LOOKUP_CMD column value in the *ace_circuit_builder.xls*, indicates which lookup file to use for the circuit element.

Fore More Information on
Circuit Builder go to Electrical Help>Contents Tab>Advanced Productity>Customize the Circuit Builder.

Conclusion

In the fast paced world of industrial controls, it is helpful to have as many tools at your disposal as possible. You can now see that there are a myriad of ways that spreadsheets can aid you in documenting your designs in AutoCAD Electrical. Experiment and come up with your own variations, and by all means, ***Unleash the Spreadsheet!***
.

Resources

Autodesk Official Training Courseware – AutoCAD Electrical 2009

AutoCAD® Electrical - Advanced Productivity Tools

AutoCAD® Electrical - Help Topics

Special thanks to the many designers and engineers I have taught and learned from over the years.