GEN21987

I Feel the Need, the Need for Speed – AutoCAD Electrical Automation

Brian Krystiniak , Dematic Corp.

Tiffany Bachmeier, Autodesk, Inc.

Learning Objectives

- Discover and develop the Spreadsheet to PLC I/O Utility
- Discover and learn how to customize the PLC database
- Discover and learn how to develop circuits
- Discover the WDI file and WDIO.LSP

Description

If you are using AutoCAD Electrical software and love it, but you've been dreaming of getting your entire electrical project done in seconds—dream no more. This is the class for you. We are going to take a deep dive into the Spreadsheet to PLC I/O Utility and show you how to manipulate it to automate the creation of all of your schematic drawings, not just the PLC drawings! Join this class and take your AutoCAD Electrical skills to the next level! This session features AutoCAD Electrical.

Your AU Experts

Brian Krystiniak *has over 18 years of CAD experience involving 2D and 3D modeling, tool and die design, architecture, mechanical, HVAC (heating, ventilating, and air conditioning), and conveyor systems. He is efficient in both plan and elevation views, and he also has some self-taught programming experience using Visual Basic and Visual LISP. He has been an AutoCAD Electrical software trainer for the last 6-plus years, and he helped develop the training manual and labs used by Dematic. His Microsoft Office experience includes advanced proficiency in all disciplines of Office Suite, with some self-taught programming using Visual Basic.*

Tiffany Bachmeier *has been an Autodesk Consultant for the last decade. Her primary focus is as a technical consultant/instructor for AutoCAD Electrical, but she also focuses on AutoCAD, Inventor, and a variety of other products in the Autodesk family. She is an Autodesk Certified Instructor and she (and colleagues) 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 (MSU) and she worked in many different industries gaining valuable CAD experience, including electrical engineering, interior design/architecture, mechanical engineering, software engineering, and she was part of MSU's CAD Development Team. She started on AutoCAD R10 and has carried a strong passion for Autodesk products ever since.*

Learning Objective 1: Discover and Develop the Spreadsheet to PLC I/O Utility

About the Spreadsheet to PLC I/O Utility

Overview: Before using the **Spreadsheet to PLC I/O utility** to automatically create your schematic drawings, you need to learn what the tool is, how it works, and which information you need to modify in the spreadsheet.

Spreadsheet to PLC I/O Utility Defined
You can use your favorite spreadsheet program to lay out your **PLC I/O** requirements. The spreadsheet is read and drawings are automatically created from the information in the spreadsheet file.

The spreadsheet file includes the following:

■ **I/O** catalog numbers

■ **I/O** address assignments

■ **I/O** descriptions

■ Connected component tags

■ Component symbol names

Note: The **Spreadsheet to PLC I/O utility** can read an **Excel** spreadsheet file or an **Access database** file directly. If your file is not in **XLS** or **MDB** format, you must save your data in a comma-delimited **ASCII** file format (usually with a .csv file extension).

Spreadsheet Layout
Each line in the spreadsheet corresponds to a rung or an **I/O point** on the **PLC module**.

You can insert up to **nine** in-line connected devices per **I/O point**. Components for input modules are inserted left to right, whereas components for output modules are inserted right to left. The spacing between devices (established in the **Spreadsheet to PLC I/O Utility Setup** dialog box) is maintained even if no component is defined for a particular column.

Special Codes
You can enter code words in the spreadsheet to force specific actions. The following are special codes you can use:

■ **BREAK:** Entered in the **Address** column. The **PLC module** breaks and continues in the next ladder column. There should be no other **PLC module** data in the row. You can place wired device data in the row.

■ **SPACER:** Entered in the **Address** column, adding a space in the **PLC module**. This causes the module to skip a rung before continuing to insert **I/O point**. There should be no other **PLC module** data in the row. You can place wired device data in the row.

■ **SKIP:** Entered in the **Module Code** column. Makes the **PLC module** skip a ladder and continue in the next ladder. There should be no other data in the row.

Special Formatting
You can enter special codes in the spreadsheet cells to create custom formatting. The following is a list of some of the codes available:

■ Use the pipe **|** character to map a single **Description field** to multiple **Description attributes**.

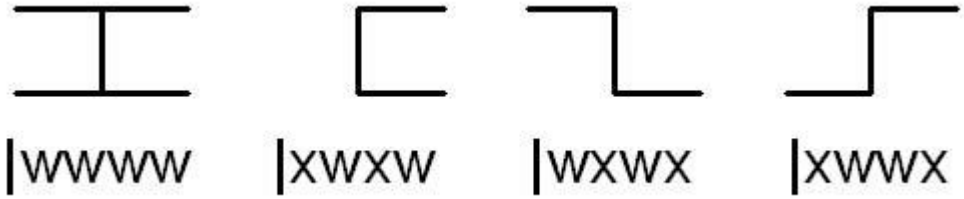
Separate sections of the field using the pipe **|** character. For example, when you use MACHINE|

START in the **Description** field, you place MACHINE in the **Description1** attribute and START in the **Description2** attribute.

■ Define both a tag and a terminal number for a terminal by using the format **TAGSTRIP:TERM**, where the colon (**:**) separates the terminal's **TAG-ID value** from the terminal number that is applied to the **TERM attribute**.

■ Insert a **circuit** (an exploded DWG™ file) by placing an asterisk (*****) in front of a symbol device block name.

Wire Jumpers
Specify that **AutoCAD Electrical** uses a **jumper**, or branch (for example, tying two parallel rungs together) by encoding the jumper as one of the available in-line devices. Use the **|** character as the jumper block name. A wire is added from the current rung to the next one in the sequence. To control removal of wire connections, follow the **|** character with **four characters** to cover, in order, the **upper-left**, **upper-right**, **lower left**, and **lower-right** connections. Use a **W** to keep the wire connection and an **X** to remove it. Some examples are shown in this image.



Settings

You control many aspects of how the drawings are automatically generated through the use of settings and mappings files. The default spreadsheet, ***demoplc.xls***, is set up to match the default column assignments for this tool. If you have a spreadsheet that uses a different column format, you assign the columns to a specific data category. All columns are optional except for the module part number.

You also can use this tool with an existing spreadsheet that differs from the default ***demoplc.xls*** file format. You can change the settings each time you run the program, or you can change them once and save the settings for future use.

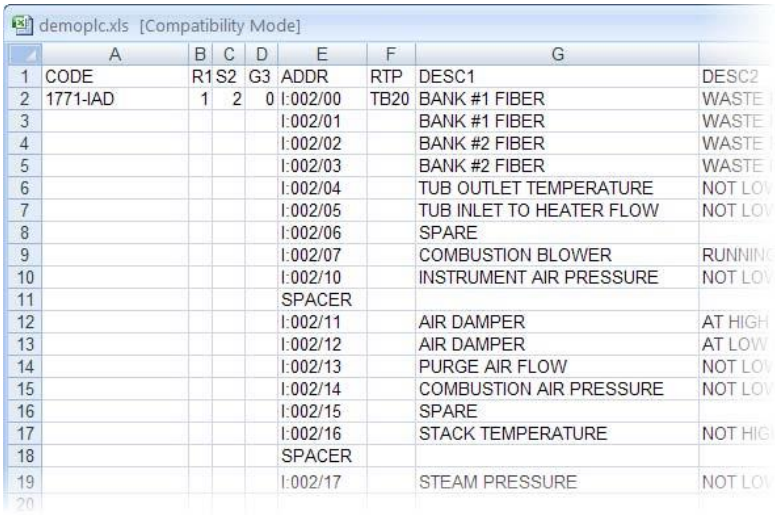
Example of PLC Spreadsheet

In the spreadsheet example, you can see the following:

■ Line 2 lists the **PLC module** and the first input rung information.

■ Lines 3 through 10, 12 through 17, and 19 list standard input rung information, one line for each **PLC** input.

■ Lines 11 and 18 show a spacer on the **PLC module** that has components inserted on a parallel rung.



demoplc.xls [Compatibility Mode]

	A	B	C	D	E	F	G	
1	CODE	R1	S2	G3	ADDR	RTP	DESC1	DESC2
2	1771-IAD	1	2	0	I:002/00	TB20	BANK #1 FIBER	WASTE
3					I:002/01		BANK #1 FIBER	WASTE
4					I:002/02		BANK #2 FIBER	WASTE
5					I:002/03		BANK #2 FIBER	WASTE
6					I:002/04		TUB OUTLET TEMPERATURE	NOT LO
7					I:002/05		TUB INLET TO HEATER FLOW	NOT LO
8					I:002/06		SPARE	
9					I:002/07		COMBUSTION BLOWER	RUNNIN
10					I:002/10		INSTRUMENT AIR PRESSURE	NOT LO
11					SPACER			
12					I:002/11		AIR DAMPER	AT HIGH
13					I:002/12		AIR DAMPER	AT LOW
14					I:002/13		PURGE AIR FLOW	NOT LO
15					I:002/14		COMBUSTION AIR PRESSURE	NOT LO
16					I:002/15		SPARE	
17					I:002/16		STACK TEMPERATURE	NOT HIG
18					SPACER			
19					I:002/17		STEAM PRESSURE	NOT LO
20								

Learning Objective 2: Discover and Learn How to Customize the PLC database

About the PLC Database File Editor

Overview: If your design requirements specify the use of a **PLC module** not listed in the **PLC database**, you need to add a new module to the database. To best complete this task, you need to know what the **PLC Database File Editor** is and what it is used for.

PLC Database File Editor Defined

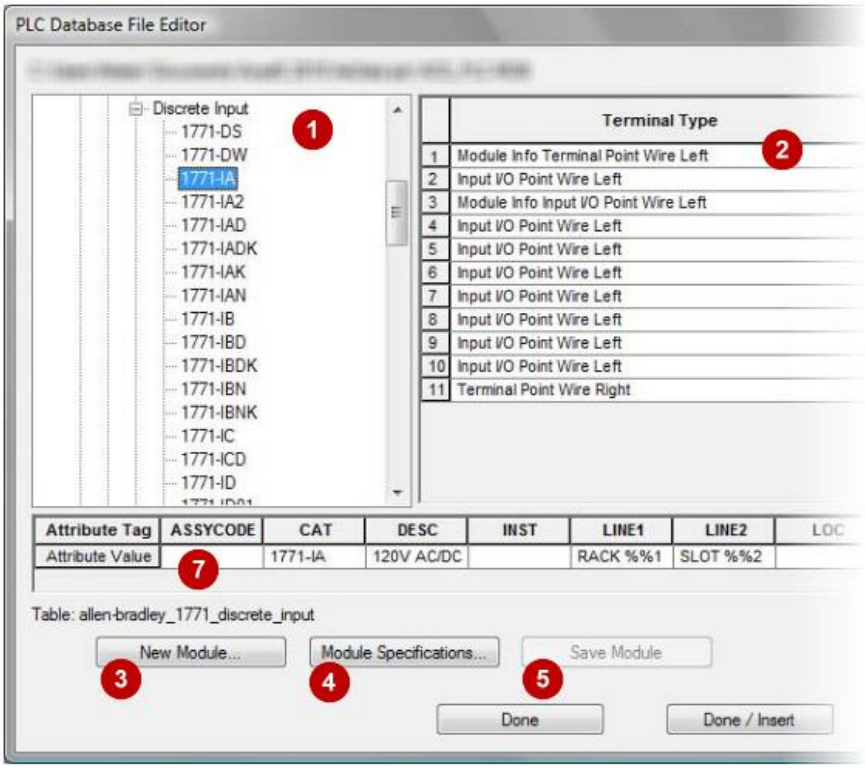
You use the **PLC Database File Editor** to assemble the pieces of the parametric **PLC**. The information you enter, including the creation of prompts and the creation of the terminal list are used by the **Insert PLC (Parametric)** command to build the **PLC** during insertion.

The custom modules you create work just like the modules that ship with the product. They are available automatically through the **Insert PLC (Parametric)** command and can also be used with the **Spreadsheet to PLC IO utility**.

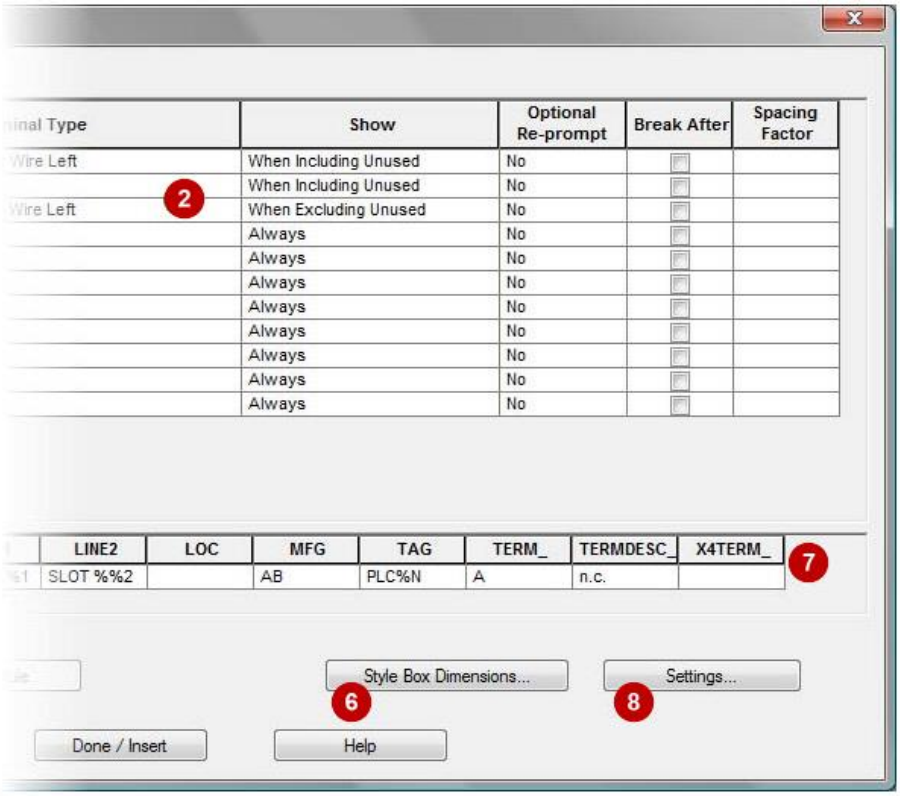
All **PLC** data is stored (by default) within the **PLC database** file, ***ace_plc.mdb***.

Control Areas

The following illustrations indicate the major control areas of the **PLC Database Editor** dialog box. See the descriptions in the table:



Left side of the **PLC Database Editor** dialog box



Right side of **PLC Database Editor** dialog box

1. **Module Tree View:** Lists all of the modules currently entered in the PLC database.
2. **Terminal Type Grid:** Displays and builds the terminal list for the PLC module.
3. **New Module:** Creates a new PLC module entry in the database.
4. **Module Specifications:** Edits an existing module entry.
5. **Save Module:** Saves the changes made while in the dialog box.
6. **Style Box Dimension**s: Changes the size of the PLC representation.
7. **Attribute Tag**: Enters attribute information, including prompt variables for each terminal.

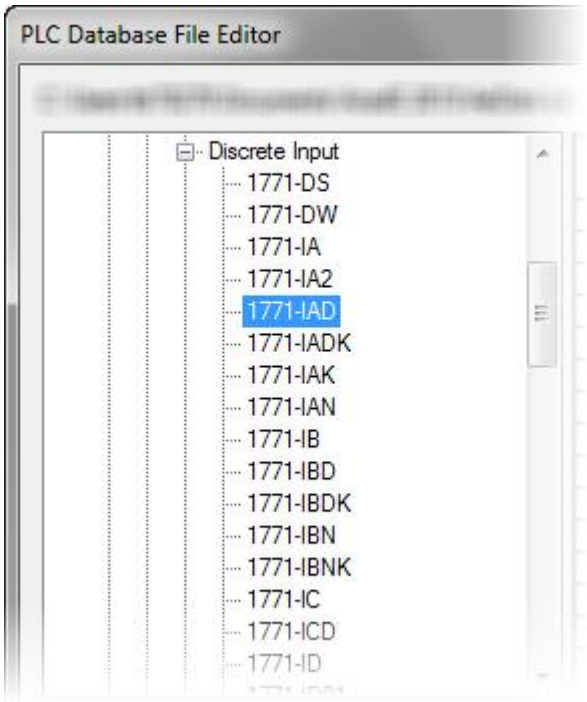
8 **Settings:** Specifies terminal blocks and settings.

PLC Module Specifications

Overview: One of the two main parts of a **PLC module** definition is the specifications of the overall module itself. To effectively create the module, you need to learn the module options and tools that are available for defining a **PLC module**.

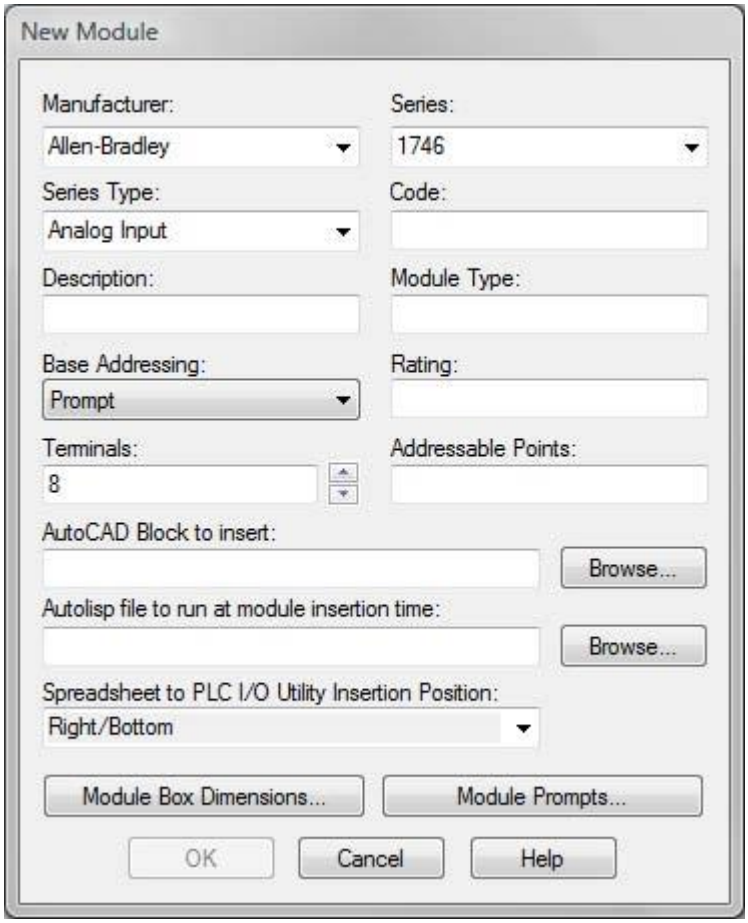
PLC Module Selection List

In the **PLC Database File Editor** dialog box, the module selection list shows an expandable tree view of **PLC** data files that are available in the **PLC database**. The list contains the **Manufacturer**, **Series**, **Type**, and **Part Number** categories. You can right-click any category to display shortcut menus for copying, renaming, deleting, and creating **PLC** data.



New Module

Click **New Module** in the **PLC Database File Editor** to display the **New Module** dialog box. You use this dialog box to enter information about the module you are creating. If information has been previously entered for a different module, you can select it by using the drop-down lists for each data field.



Use the **AutoCAD Block to Insert** option to specify an **AutoCAD®** block file to be inserted directly below the last **I/O point** that is inserted parametrically. These are typically used for block files that represent DIP switch settings and notes on how to configure a **PLC module**.

You can specify an **AutoCAD LISP routine** to run after the program parametrically builds a **PLC module**. These are typically used for inserting groups of symbols and wires or modifying attributes on symbols to accommodate a more customized **PLC module** build.

Module Specifications

You use **Module Specifications** to edit the settings for an existing **PLC module**. This utility displays a dialog box that is very similar to the **New Module** dialog box.

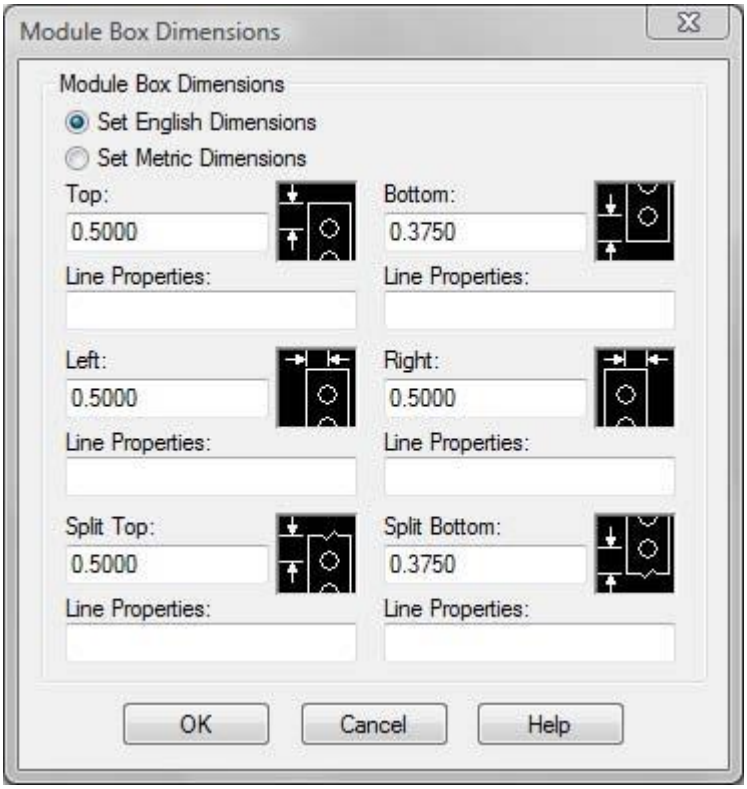
Note: The **Manufacturer**, **Series**, **Series Type**, **Code**, and **Terminals fields** are not available in the **Module Specifications** dialog box, because they are under the control of the tree structure in the **PLC** selection window and the total number of terminals listed in the terminal type grid.

Module Box Dimensions

In the **New Module** and **Module Specifications** dialog boxes you use the **Module Box Dimensions** command to display the **Module Box Dimensions** dialog box. You use this tool to optionally enter custom values for defining the outer box dimensions of the module. If you leave the values blank, the default **Style Box dimension values** are used.

All dimensions are calculated from the insertion point of the PLC module.

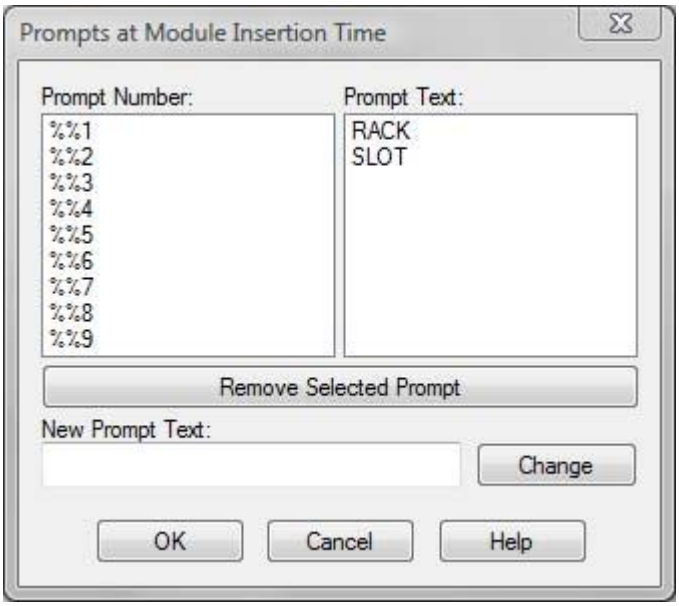
You can set the color and linetype for the box lines by using the **Line Properties** fields. To set the color, enter **COLOR <colorname>** in the box. For **linetype**, enter **LTYPE <linetypename>** in the box.



Module Prompts

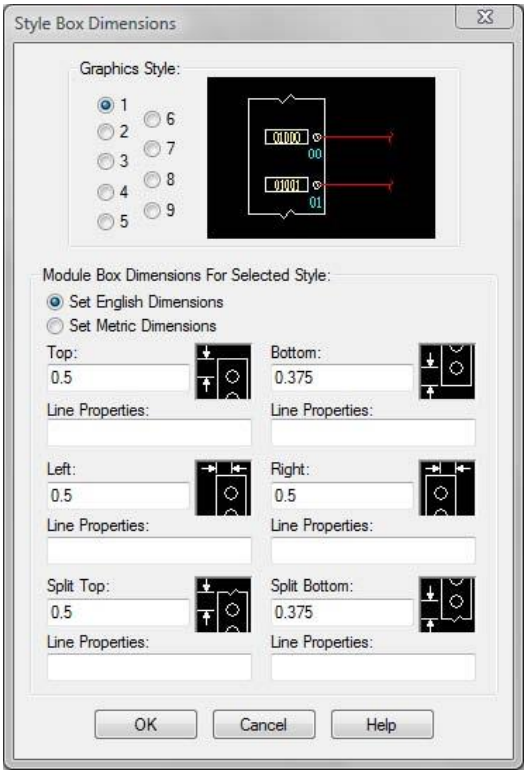
In the **New Module** and **Module Specifications** dialog boxes, you use the **Module Prompts** command to display the **Prompts at Module Insertion Time** dialog box. You can use the **Prompts at Module Insertion Time** tool to create custom prompts when inserting a **PLC**.

Once the **Module Prompt** is assigned, you use the variable to fill attribute fields in the **PLC module** utility to create custom prompts while inserting a **PLC**.



Style Box Dimensions

In the **PLC Database File Editor** dialog box, click **Style Box Dimensions** to display the **Style Box Dimensions** dialog box. The style box dimensions are the default settings supplied for each graphical style of **PLC module**. These are the dimensions used if the **Module Box Dimension** values are blank.



Note: The module box dimensions override the style box dimensions.

Tips

- Select a similar module to the one you are creating from the tree view list before you select **New Module**. The **New Module** dialog box is prefilled with data for the selected **Manufacturer**, **Series**, and **Type**.
- For examples of the use of the variables, see the **Rack** and **Slot** number variables assigned to many of the existing **PLC** definitions.

Terminal List and Settings

Overview: In addition to the module specifications, the second main area of a **PLC module** specification is the **terminal list**. The terminal list drives the appearance and functionality of the final **PLC** component. To create this terminal type list, you need to know the options and tools available within the **PLC Database File Editor**.

Terminal Type Grid Area

When you highlight a module part number from the selection list in the **PLC Database File Editor**, **AutoCAD Electrical** fills the terminal grid with the defined information for the selected module. Each row in the grid contains the block and properties for the corresponding terminal in the **PLC**.

The terminal grid serves as the assembly list for a parametric **PLC module**. During insertion, the

information in the grid controls the appearance of the **PLC**.

To change the properties of a terminal, right-click the **Terminal Type field** for the selected terminal. From the shortcut menu, you can edit the terminal or insert and delete terminals. You can select multiple terminals for editing with standard selection methods.

	Terminal Type	Show	Optional Re-prompt	Break After	Spacing Factor
1	Module Info Input I/O Point Wire Left	Always	No		
2	Terminal Point Wire Left	Always	No		
3	Terminal Point Wire Left and Right	Always	No		
4	Input I/O Point Wire Left	Always	No		
5	Terminal Point Wire Left	Always	No		
6	Terminal Point Wire Left and Right	Always	No		
7	Input I/O Point Wire Left	Always	No		
8	Terminal Point Wire Left	Always	No		
9	Terminal Point Wire Left and Right	Always	No		
10	Input I/O Point Wire Left	Always	No		
11	Terminal Point Wire Left	Always	No		
12	Terminal Point Wire Left and Right	Always	No		

The terminal grid has the following fields:

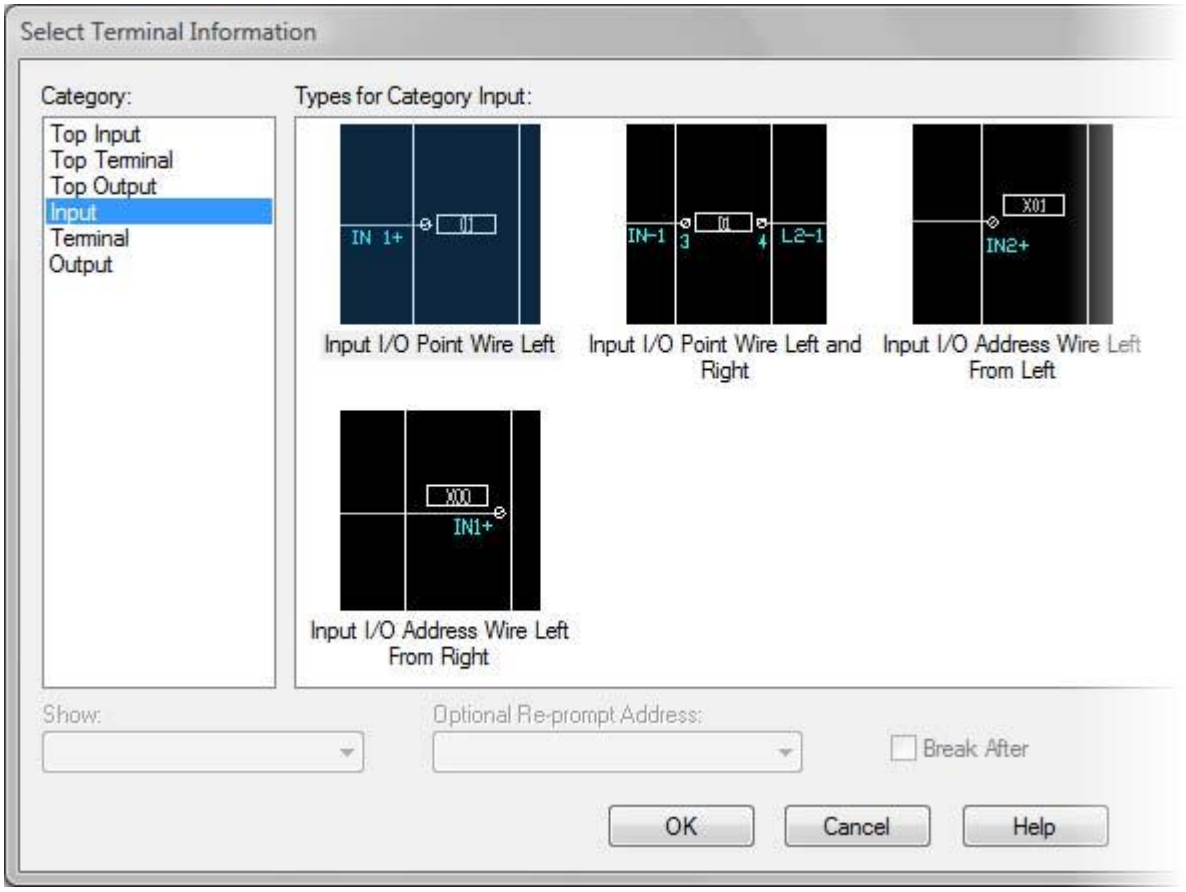
■ **Terminal Type:** Determines the type of terminal to be used. Right-click the cell and click **Edit Terminal** to select the terminal.

■ **Show:** Controls the display state of the unused terminals as determined by the **Include Unused/Extra Connection** option during **PLC module** insertion. Unused terminals are those that do not have wiring connections but may be displayed on an inserted **PLC module**.
■ **Optional Re-prompt:** Re-prompts for a new beginning address number. When the parametric build for a **PLC module** changes from inputs to outputs or from outputs to inputs, you are prompted for a new beginning address number.

■ **Break After:** Specifies for the module to break automatically after the selected terminal.

■ **Spacing Factor:** Overrides the current rung spacing for **I/O** and wire connection point spacing. For example, a value of 2 inserts the point down two times the rung spacing instead of a single rung spacing.

When you select **Edit Terminal**, you use the **Select Terminal Information** dialog box to select the desired category for the terminal. Available terminal types for the selected category are displayed. During this operation, you are selecting the block that is used during the parametric creation of the **PLC module**.



Note: One of the first terminals inserted in the grid is one of the top styles of terminals, either **Top Input**, **Top Terminal**, or **Top Output**. This style of terminal contains the parent attributes such as catalog number, manufacturer, tag, and so on.

Terminal Attributes

In the **PLC Database File Editor** dialog box, the **Terminal Attributes** area displays the attribute values associated with the selected terminal. Some of these values are prefilled, such as the **CAT** field, which is shown filled with the catalog number.

You can also assign prompts for you to enter data during **PLC** insertion, such as the **LINE1** and **LINE2** fields shown in the following image. The **LINE1** and **LINE2** fields are shown with the **%% 1** and **% %2** variables that you define during **PLC module** creation, along with the static **RACK** and **SLOT** text, respectively. You can enter the prompt assignments in the **New Module** and **Module Specifications** dialog boxes.

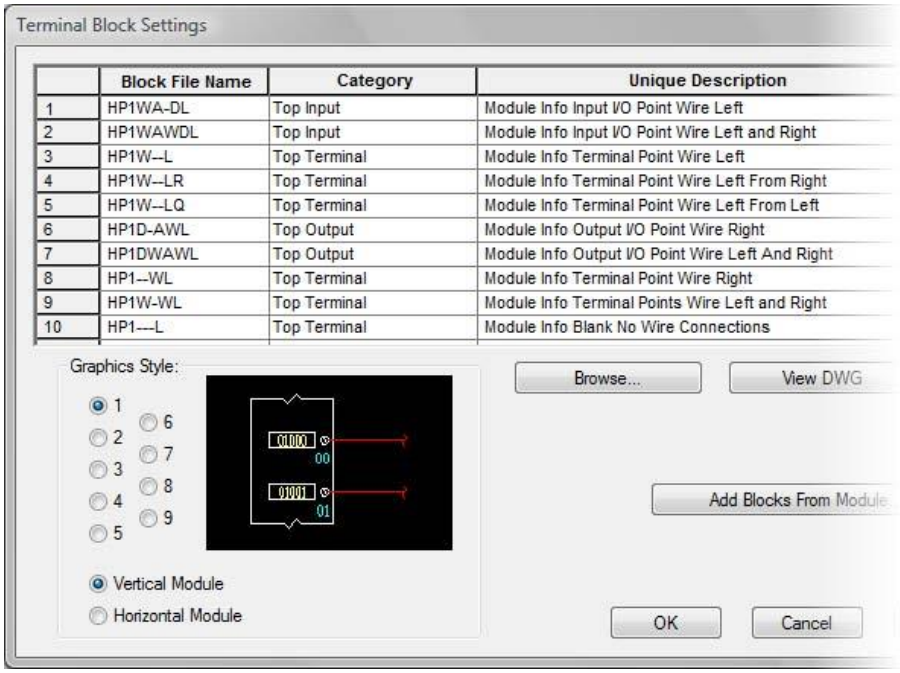
Attribute Tag	ASSYCODE	CAT	DESC	DESCA_	DESCB_
Attribute Value		1746-NI16V	16pt ANALO		

DESCB_	DESCC_	DESCD_	DESCE_	INST	LINE1	LINE2
					RACK %%1	SLOT %%2

LINE2	LOC	MFG	TAG	TAGA_	TE
SLOT %%2		AB	PLC%N		

Terminal Block Settings

In the **PLC Database File Editor**, click **Settings** to display the **Terminal Block Settings** dialog box. You use this dialog box to organize your terminal blocks, establishing which blocks are available for each category and graphical style when building a new **PLC module**.



Learning Objective 3: Discover and Learn How to Develop Circuits

About Circuits

Overview: To better create your own custom circuits, it is helpful to understand what circuits are and how they are processed.

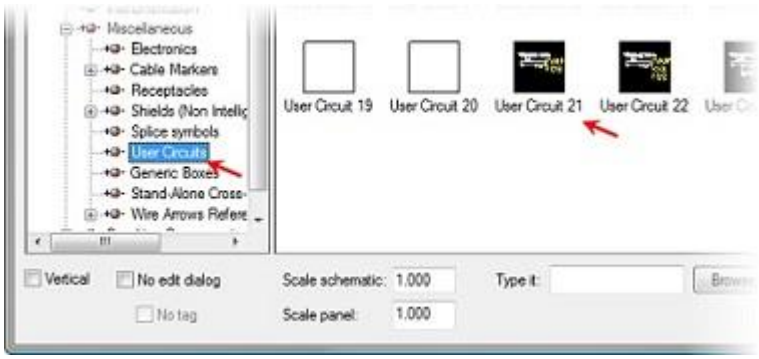
Definition of Circuits

After a circuit is moved, copied, or inserted, the circuit is processed and the component tags and other information is updated where possible. To prevent duplication errors some data may be erased.

A **circuit** is any collection of electrical objects that you manage as one group. A circuit may contain ladders, wires, components, or entire sections of your schematic drawing.

Saving Circuits to an Icon Menu

Overview: Many times during a design process you require quick and easy methods of storing circuits for temporary use. The use of these circuits may only be for your own short-term design needs. Saving Circuits to an icon menu is one method of providing this functionality and speeding up your design process.



Command Access

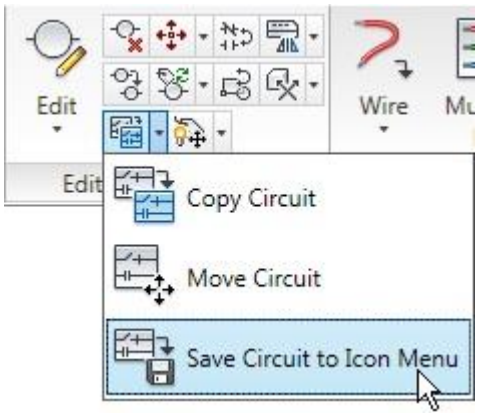
Save Circuit to Icon Menu

Ribbon: Schematics tab > Edit Components panel > Copy Circuit flyout > Save Circuit to Icon Menu

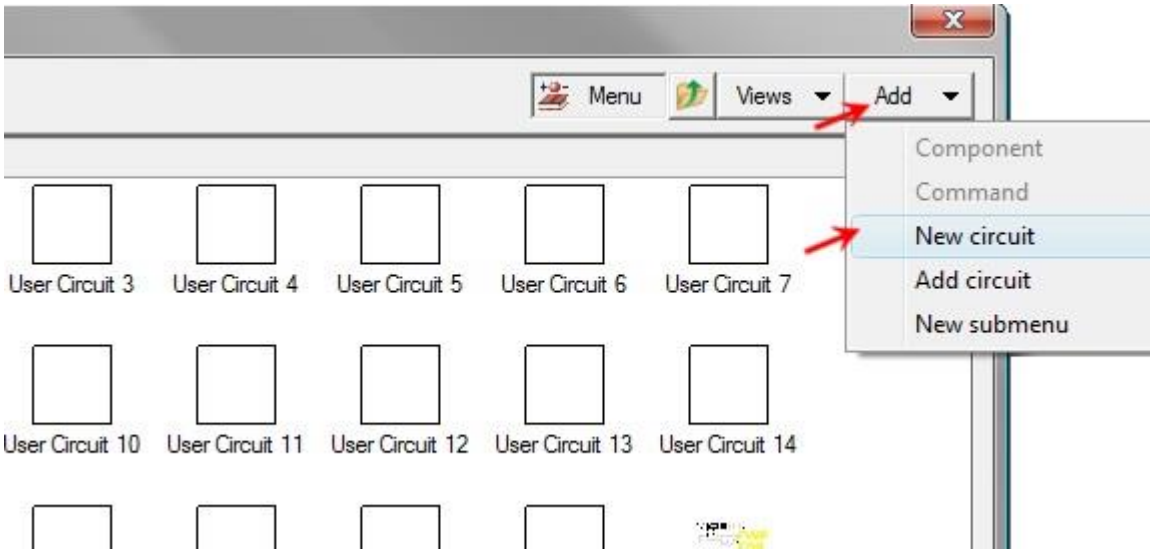
Procedure: Saving Circuits to an Icon Menu

1. The following steps describe how to save a circuit to the **User Circuits icon menu**:

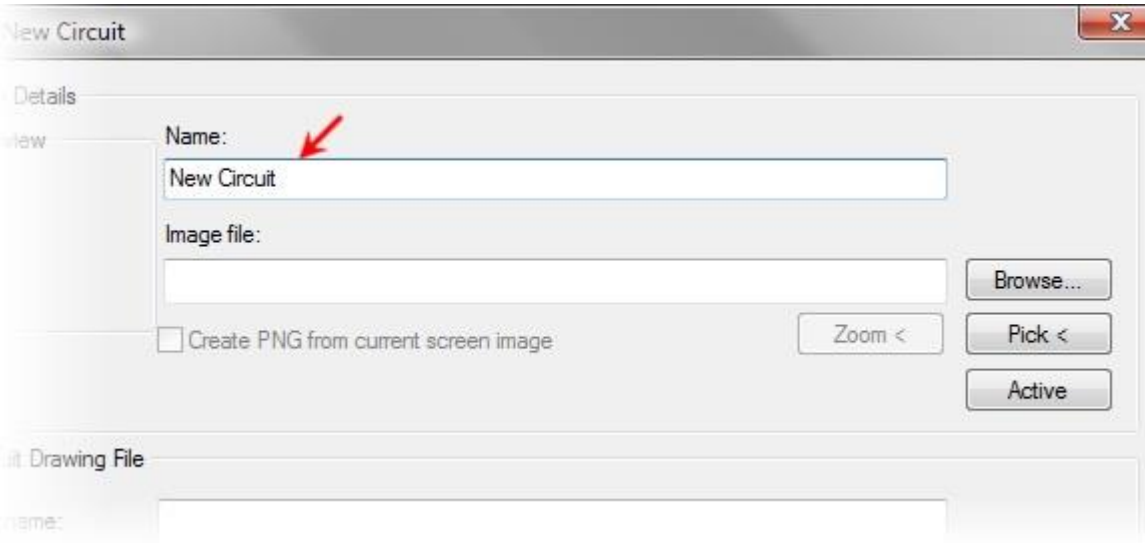
   Start the **Save Circuit to Icon Menu** command.



2. In the **Save Circuit to Icon Menu** dialog box, from the **Add** menu, select **New Circuit**.

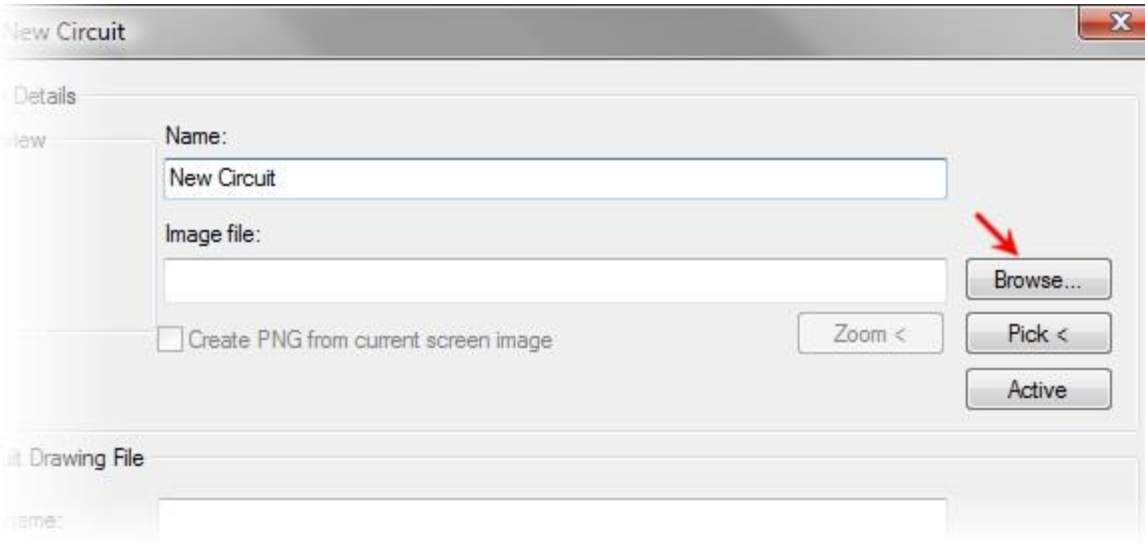


3. In the **Create New Circuit** dialog box, type the name.

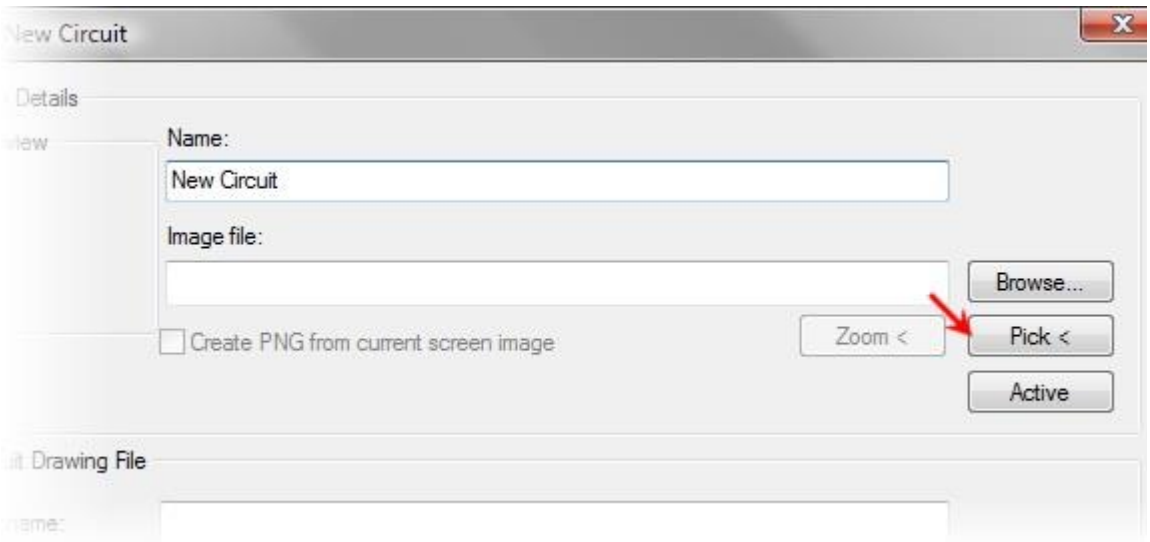


4. To select an existing image file or create a new one, use the following options:

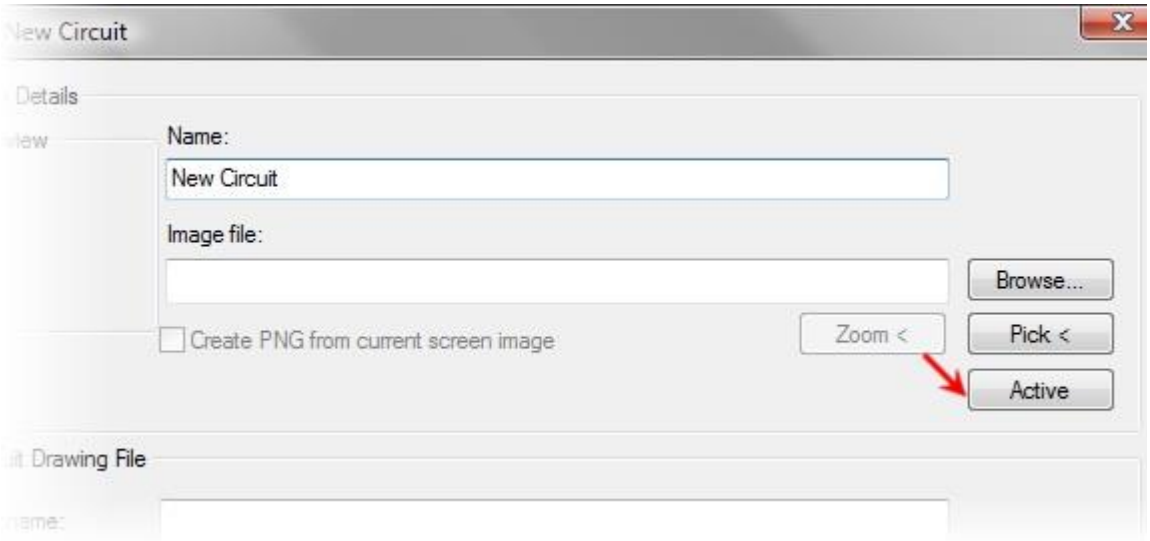
   Option 1: Click **Browse** to select an existing image file.



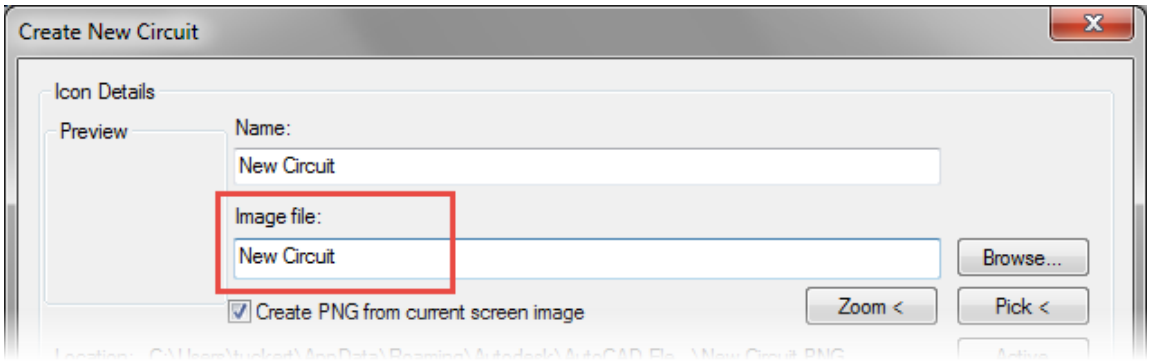
Option 2: Click **Pick** to select an existing block. The name of the block will be entered into the **Name** box.



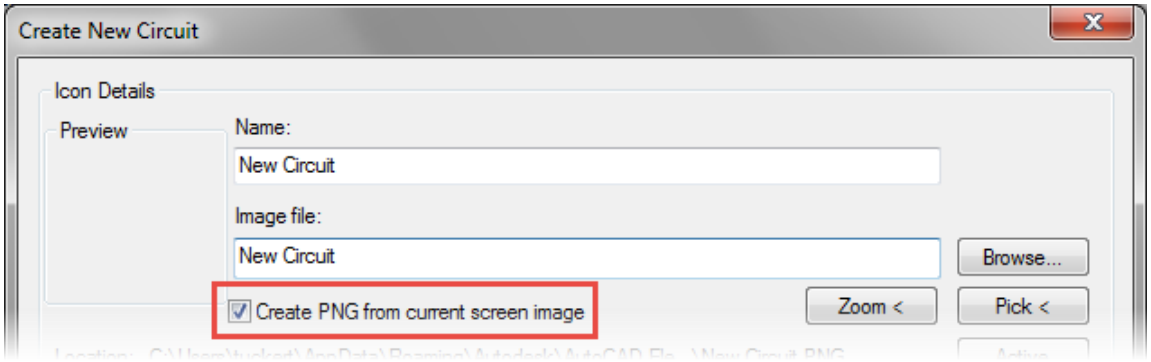
Option 3: Click **Active** to enter the name of the current drawing into the **Image file** box.



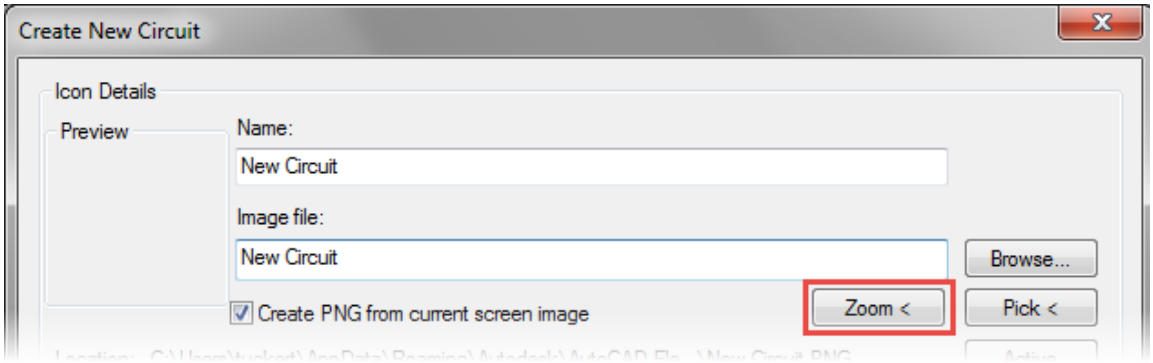
Option 4: Manually enter an image name.



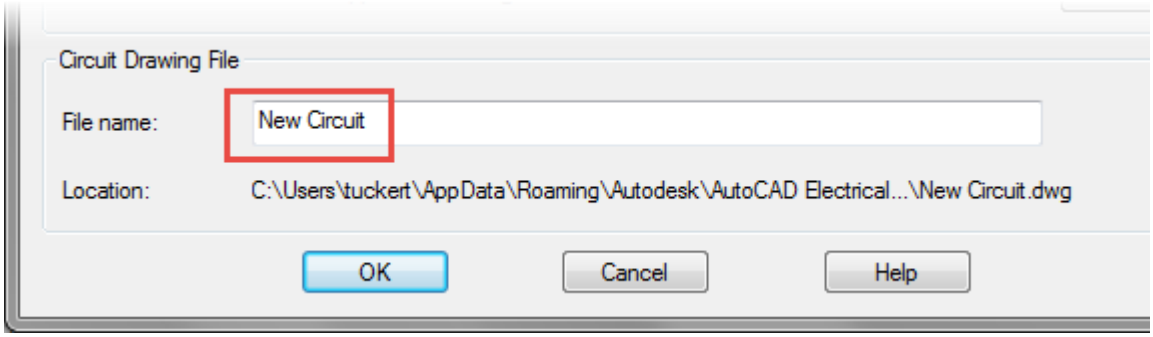
Option 5a: Create an image from the current screen image. Click the box next to **Create PNG from Current Screen Image**.



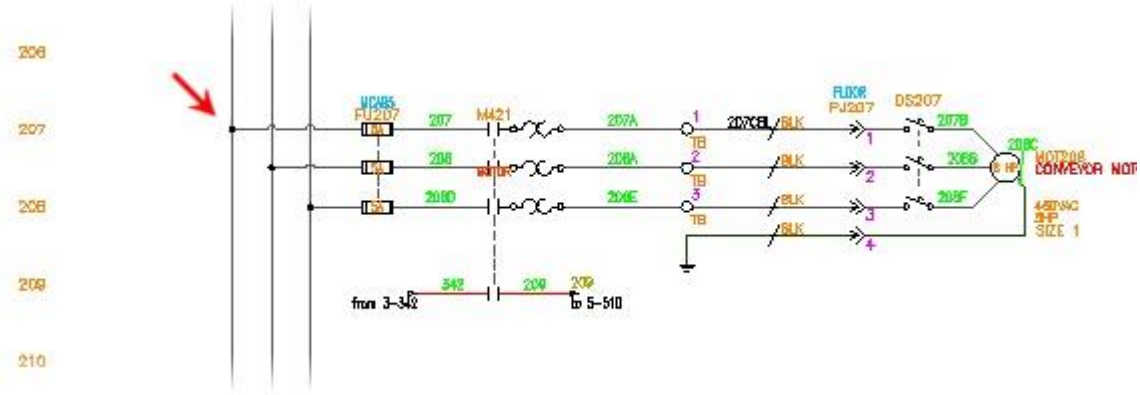
Option 5b: Click **Zoom** to return to the drawing, and **zoom** and **pan** the image to make it as large as possible.



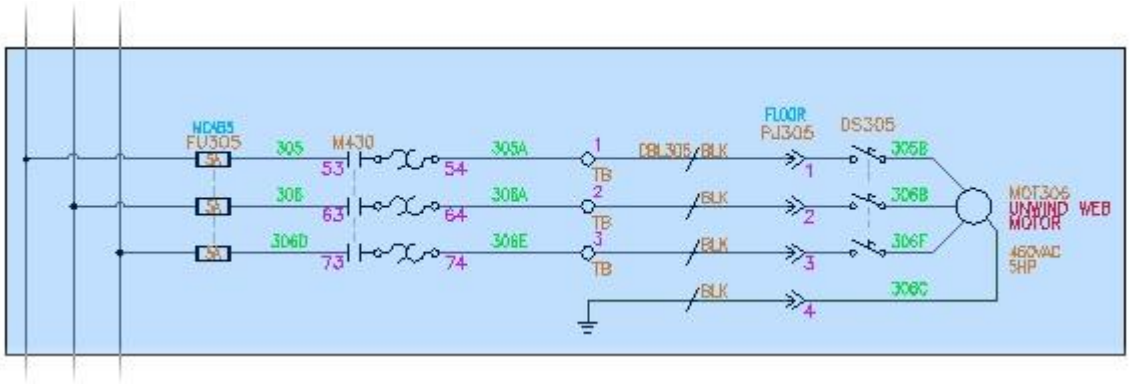
5. Enter a file name. By default, the file is saved in the current user directory.



6. Select a base point for your circuit. Be sure to use a **Snap** mode.



7. Select all of the objects to be included in the circuit. The new circuit is added to the **User Circuit icon menu**.



Guidelines

1. The **Save Circuit to Icon Menu** and **Insert Saved Circuit** commands provide shortcuts to the **User Circuits submenu** in the icon menu system.

   You can also use the **Insert Component** command to achieve the same results. Select **Insert**

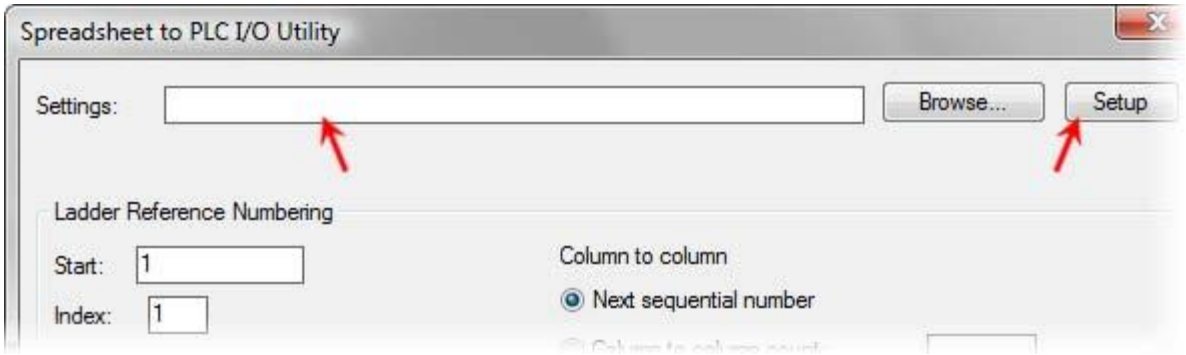
   Component > ...Schematic Symbols > Miscellaneous > User Circuits for circuit selection.
2. The **Name** field is displayed as the text in the icon with text and list views. Be very detailed when naming your circuit. This is an important step that enables you to identify your circuit when you want to insert it. The circuits are often too large and complicated to identify them clearly by using the icon alone.
3. When creating the icon from the current screen image, you must **zoom** in to the circuit you want to save. Try to make the view as large and as clean as possible, showing only graphics necessary to identify the circuit.
4. The advantage of the **Save Circuit to Icon Menu** command is that it provides a shortcut to the **User Circuits** menu. The disadvantage is that, because the circuits are automatically saved in the local **User** directory, they are not available to other users. To share a circuit saved in the **User** directory, you must copy or move the circuit to a shared folder.

Learning Objective 4: Discover the WDI file and WDIO.LSP

Spreadsheet to PLC I/O Utility Setup

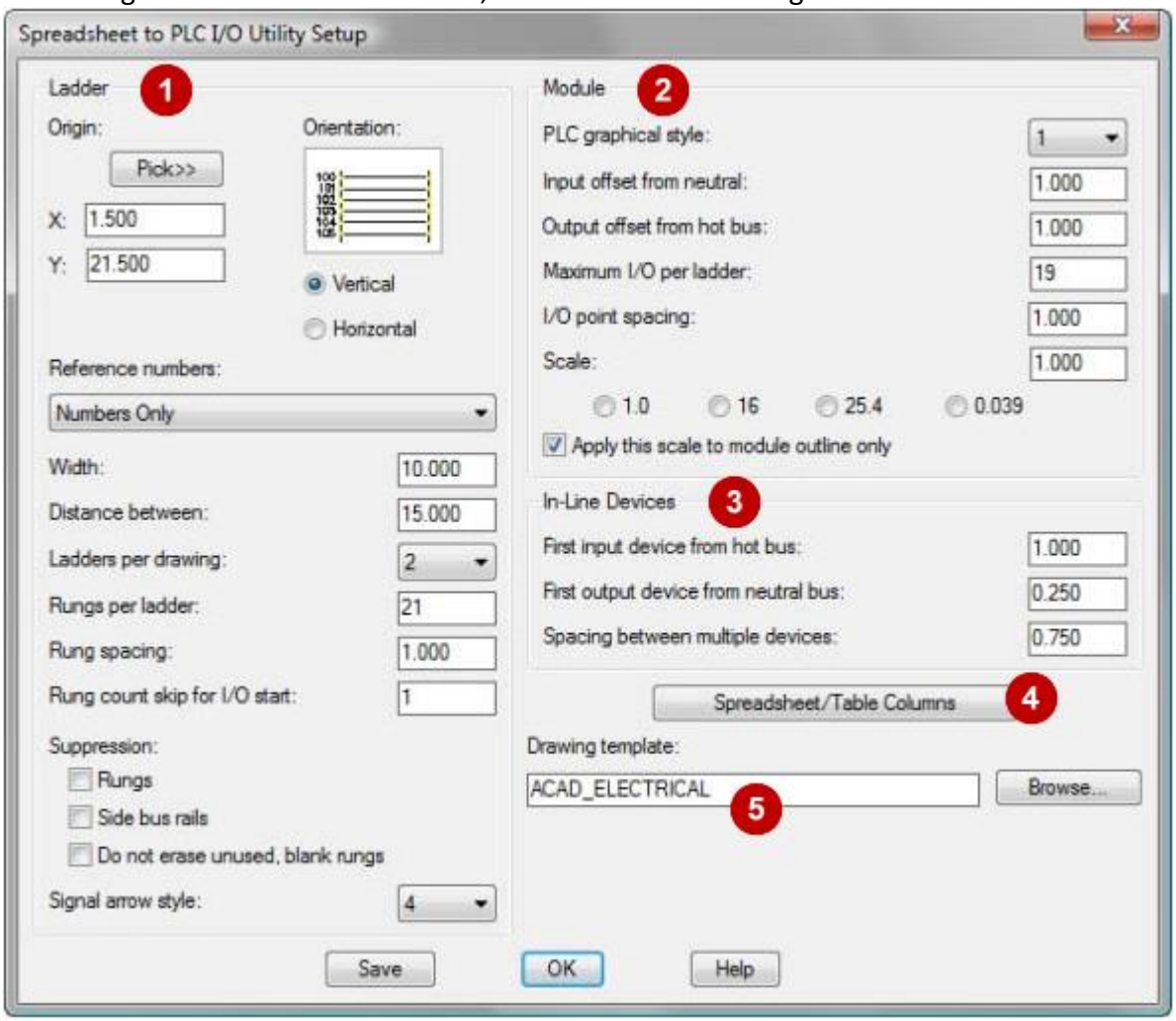
Overview: To customize the **Spreadsheet PLC I/O utility** to match your company **PLC** spreadsheet layouts, you must learn where and what settings can be preconfigured so that information is transferred correctly into the generated drawings.

Setup Options
To edit the setup options for the **Spreadsheet to PLC I/O utility**, start the command. In the dialog box, click **Setup**.



You use the **Spreadsheet to PLC I/O Utility Setup** dialog box to edit the settings used during the drawing generation process. You can save the changes to an external file to reuse at any time.

The dialog box has **five** main sections, as indicated in this image:



1 **Ladder:** Use this area to define the creation of ladders.

2 **Module:** Use this area to define the location and **I/O point** spacing of the **PLC module**.

3 **In-Line Devices:** Use this area to locate the devices on the ladder rungs.

4 **Spreadsheet/ Table Columns:** Use this option to display the **Spreadsheet to PLC I/O Drawing Generator** dialog box in which you map the spreadsheet columns to component attributes.

5 **Drawing Template:** Use this area to select the template that will be used when new drawings are created.

Ladder Configuration

You use the **Ladder** area of the **Spreadsheet to PLC I/O Utility Setup** dialog box to control how the ladders are inserted in the drawings. You can customize most things related to the ladder columns. In addition to the standard ladder settings used in the **Insert Ladder** command, you can also set the following:

■ Origin or insertion point of the first ladder in a drawing

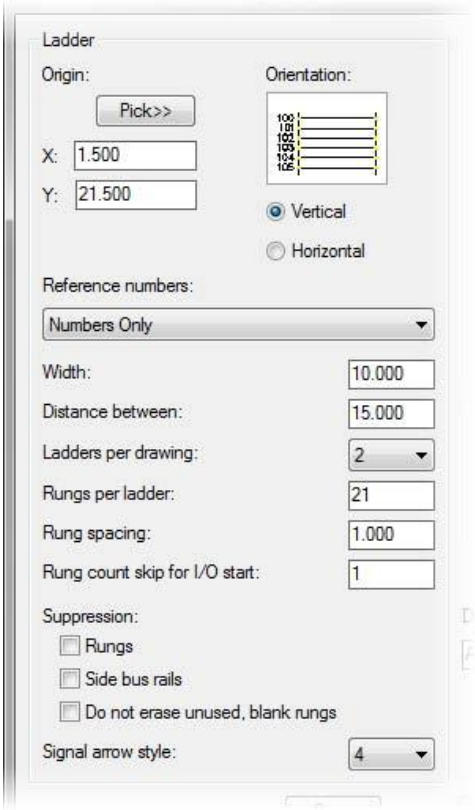
■ Ladder orientation

■ Reference number or style of ladder

■ Distance between multiple ladders

■ Style of the signal arrow used

The default setting in the utility is to erase rungs that connect to **I/O points** that do not have devices inserted on them. This is to help prevent unwanted shorts between wire networks. You can choose to keep these rungs for the manual addition of components by selecting the **Do Not Erase Unused, Blank Rungs** option.



Note: The **Distance Between** setting is the distance between the insertion points of the ladders, not the gap between them. For example, if the ladders are 10 units wide and the Distance Between is set to 15, there will be a space of 5 between the two ladders for component descriptions, and so on.

Module Settings

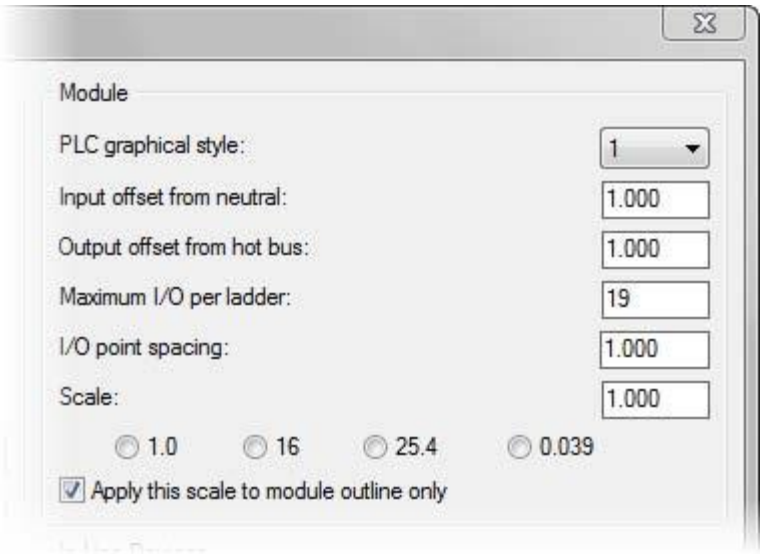
You use the **Module** area of the **Spreadsheet to PLC I/O Utility Setup** dialog box to control how the PLC module is inserted on the ladder. You have options available to set the following:

■ **PLC graphical style**

■ Location on the ladder

■ Number of **I/O points** and their spacing

■ Scale of the module

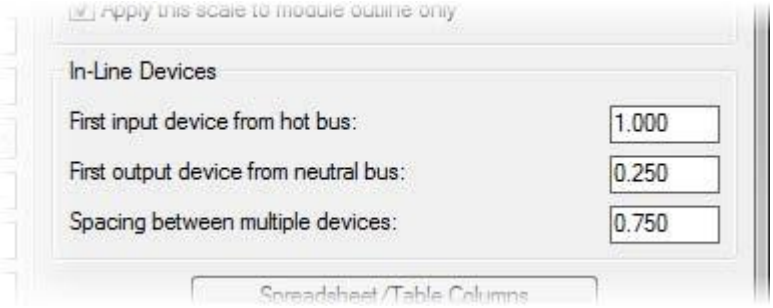


In-Line Device Settings

You use the **In-Line Devices** area of the **Spreadsheet to PLC I/O Utility Setup** dialog box to control how the inline devices are inserted. You have options available to set the following:

■ Spacing from the hot or neutral buses

■ Spacing between devices on a rung

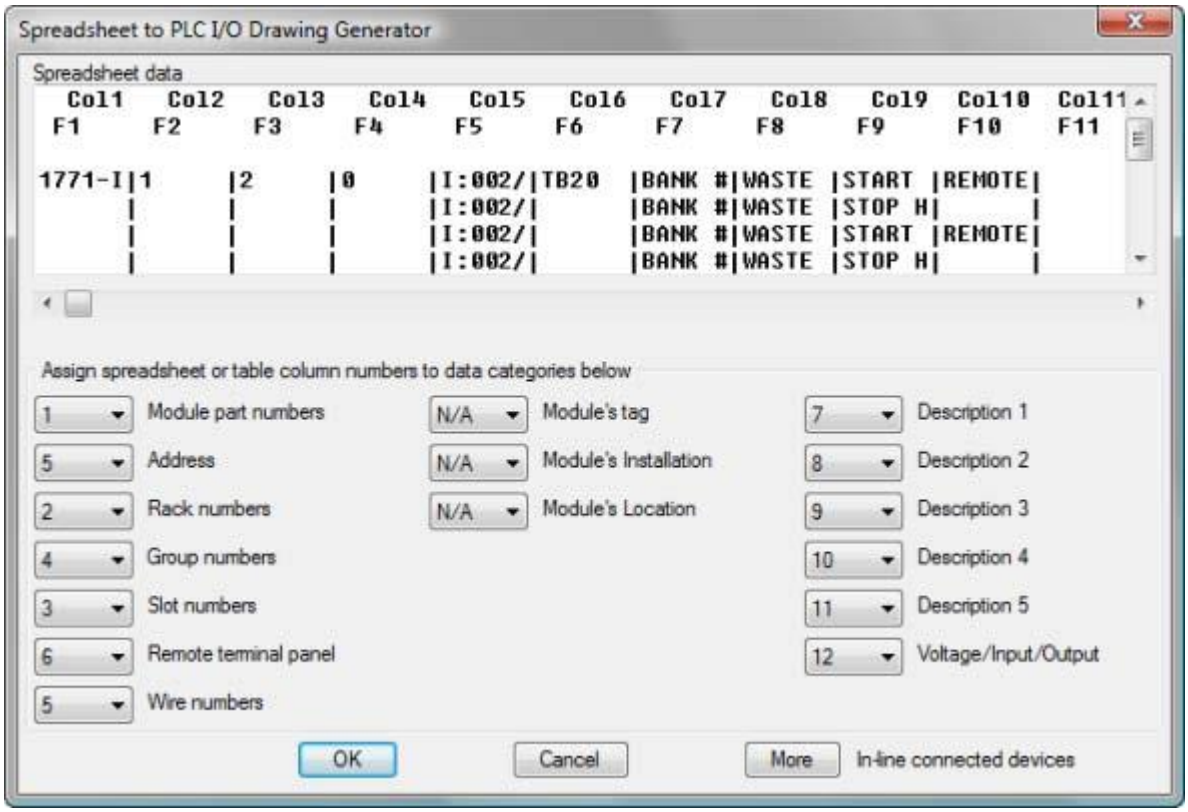


Spreadsheet/Table Columns

To map spreadsheet information to the appropriate fields, in the **Spreadsheet to PLC I/O Utility Setup** dialog box, select **Spreadsheet/Table Columns**.

The **Spreadsheet to PLC I/O Drawing Generator** dialog box displays an **ASCII** representation of the spreadsheet under **Spreadsheet Data**. In the lower area of the dialog box, you map the

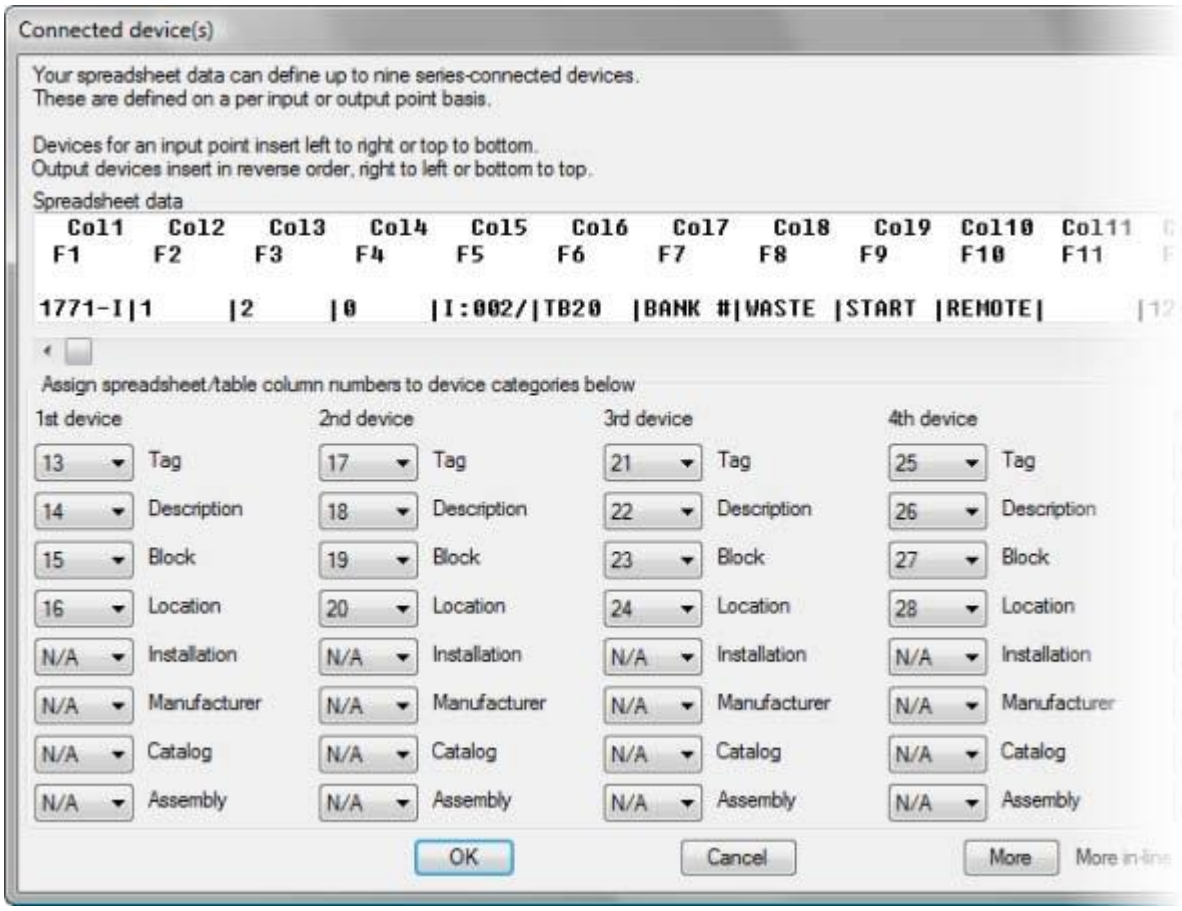
columns of the spreadsheet to the component attributes. For each listed attribute, you use the associated column list to select a spreadsheet column.



Connected Components

In the **Spreadsheet to PLC I/O Drawing Generator** dialog box, click **More** to create the mappings on the drawings for the connected devices.

You use the **Connected Devices** dialog box to map the spreadsheet data to connected devices.



Drawing Template Selection

You use the **Drawing Template** area of the **Spreadsheet to PLC I/O Utility Setup** dialog box to select a template file used when new drawings are created.

Note: Do not use a template containing ladders. The **Spreadsheet to PLC I/O utility** automatically creates the ladders as required.

Additional Class Material

The files uploaded to the Additional Class Materials are needed to create the demonstrated automatic routine of the demoplc cabinet. In order to successfully run the PLC Utility download the following files from the Additional Class Materials and save them to the following locations:

---

wdio.lsp (Electrical 2015) – C:\Users\Public\Documents\Autodesk\Acade **2015**\en-US\Support

wdio.lsp (Electrical 2016) – C:\Users\Public\Documents\Autodesk\Acade **2016**\Support\en-US

wdio.lsp (Electrical 2017) – C:\Program Files\Autodesk\AutoCAD **2017**\Acade\Support\en-US\Shared

wdio.dcl (Electrical 2015) – C:\Users\Public\Documents\Autodesk\Acade **2015**\en-US\Support

wdio.dcl (Electrical 2016) – C:\Users\Public\Documents\Autodesk\Acade **2016**\Support\en-US

wdio.dcl (Electrical 2017) – C:\Program Files\Autodesk\AutoCAD **2017**\Acade\Support\en-US\Shared

NOTE: When adding the wdio files to your system, I would recommend to renaming the original to another file name (i.e. Original_wdio.lsp). This way if you have made any modifications to your Lisp or DCL files they will not be lost.

---

ace_plc.mdb – C:\Users***{username}***\Documents\Acade ***{version}***\AeData\en-US\Plc

---

Templates (.dwt) – C:\Users***{username}***\AppData\Local\Autodesk\AutoCAD Electrical ***{version}***\R2**X**.0\enu\Template

---

Circuits and symbols (.dwg) – C:\Users\Public\Documents\Autodesk\Acade ***{version}***\Libs\NFPA

Setting files & Spreadsheet (.wdi & .xls) – C:\Users***{username}***\AppData\Roaming\Autodesk\AutoCAD Electrical ***{version}***\R2**X**.0\enu\Support\User

---

NfpaDemo2 Project Folder – C:\Users***{username}***\Documents\Acade ***{version}***\AeData\Proj

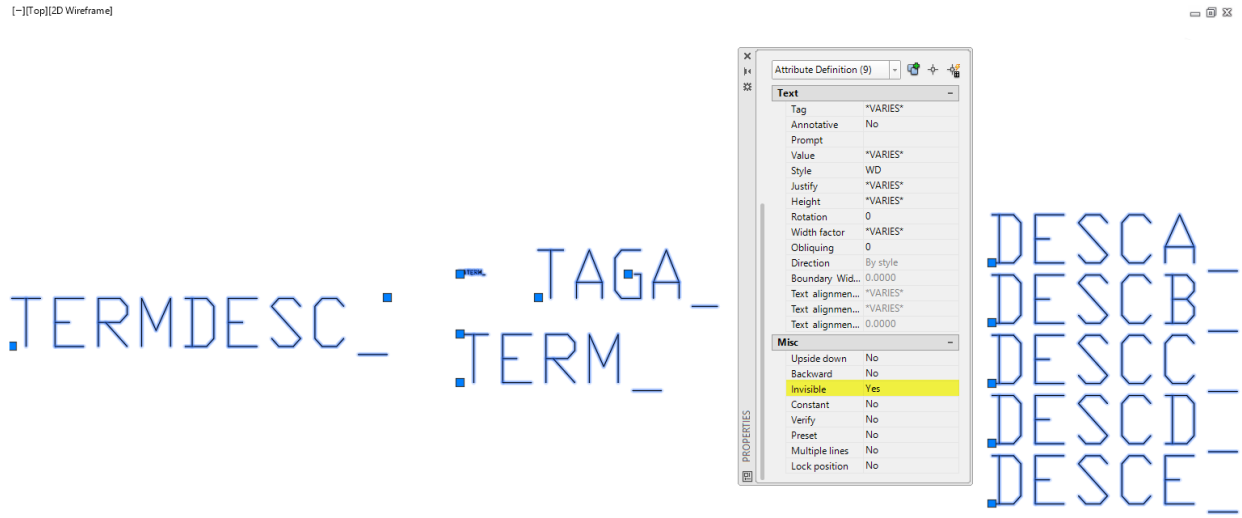
Creating automatic Schematic sheets with the PLC Utility

The following instructions show how to create the customized files needed to create an automatic schematic routine.

Start with Hp1wa-d.dwg and save as Hp1wa-blank.dwg (remove all geometry and set all attributes as Invisible).

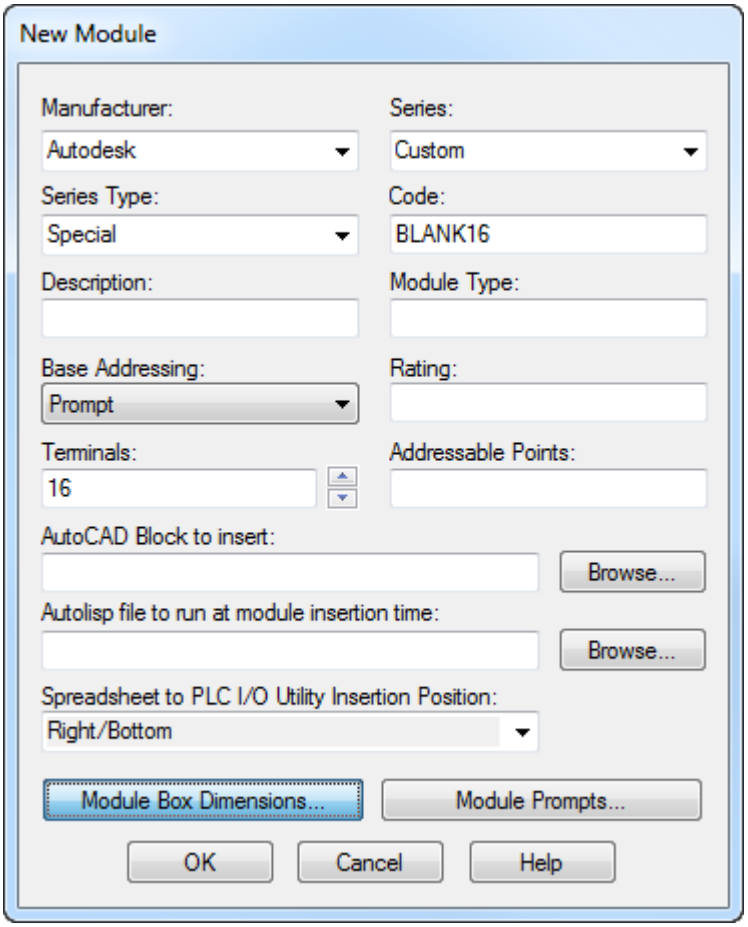
Hp1wa-d.dwg

Hp1wa-blank.dwg



This block allows for a blank PLC module to be placed in the schematics to add the symbols defined in the PLC Spreadsheet without display I/O addresses or rung comments. This is basically a place holder allowing the symbols to be created on the drawings.

Create a new Module as follows:

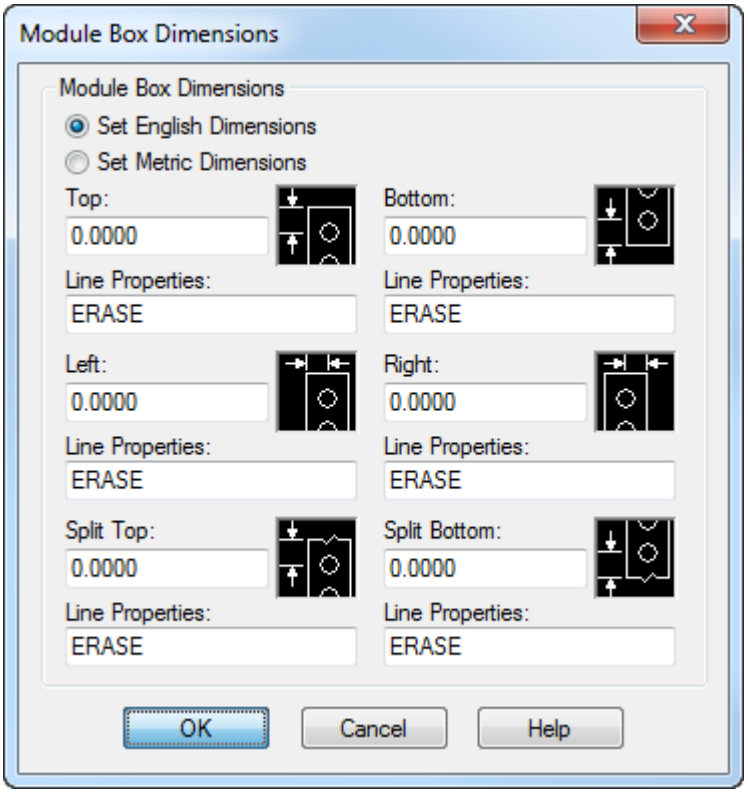


Do the above steps for BLANK16 and BLANK 44.

BLANK16 is used for a typical NFPA style schematic drawings and is defined with 1” rung spacing, 1.25” spacing between symbols, and 11.25” rail spacing.

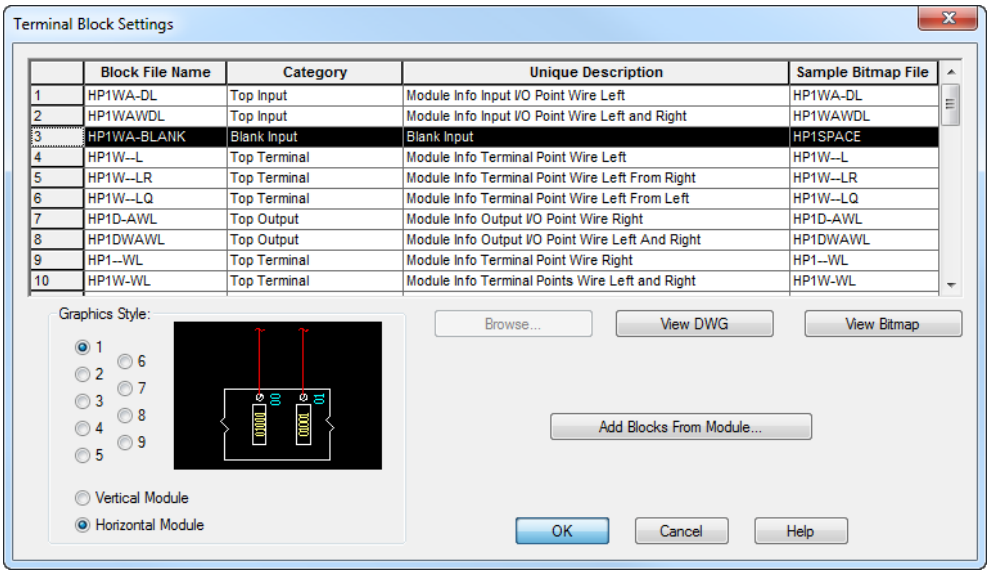
BLANK44 is used with the out-of-the-box title block and covers the entire drawing in a 0.5” by 0.5” grid. Allowing the user to put a symbol anywhere on the drawing as long as the insertion point falls on that grid.

Set the Module Box Dimensions as follows:

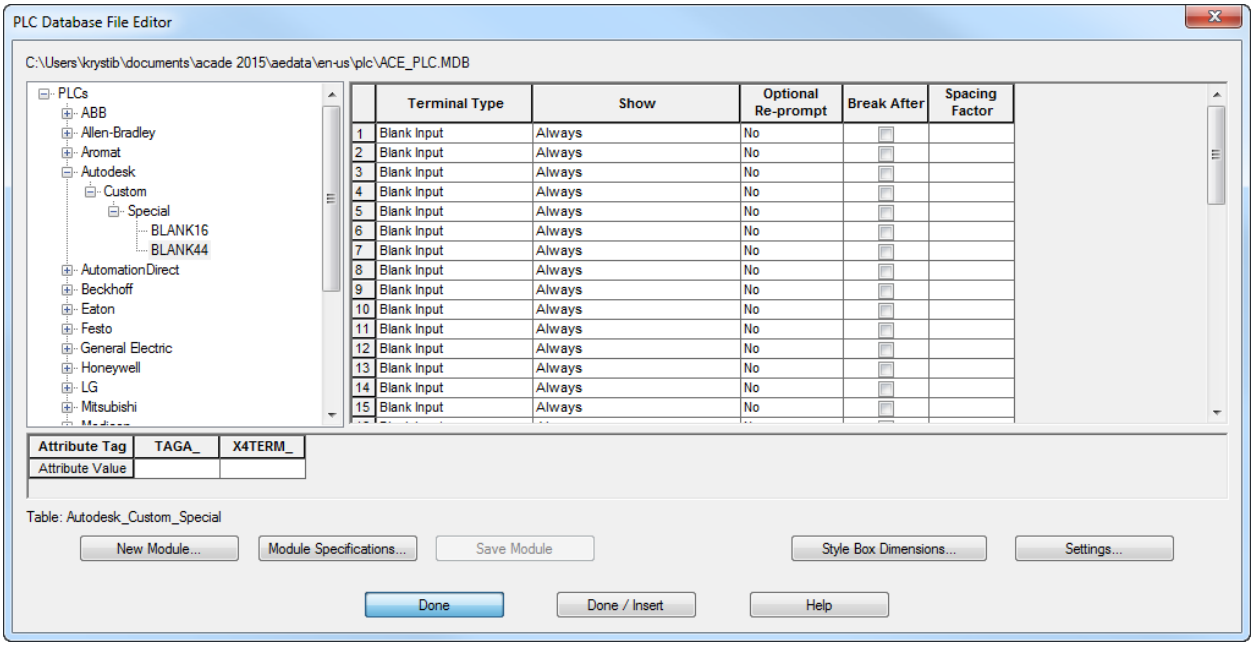


This prevents AutoCAD Electrical from creating a box around the PLC and keeps the PLC information completely invisible on the drawing.

Add the Hp1wa-blank to the Terminal Block Settings, type “Blank Input” in the *Category* column, and “Blank Input” in the Unique Description.

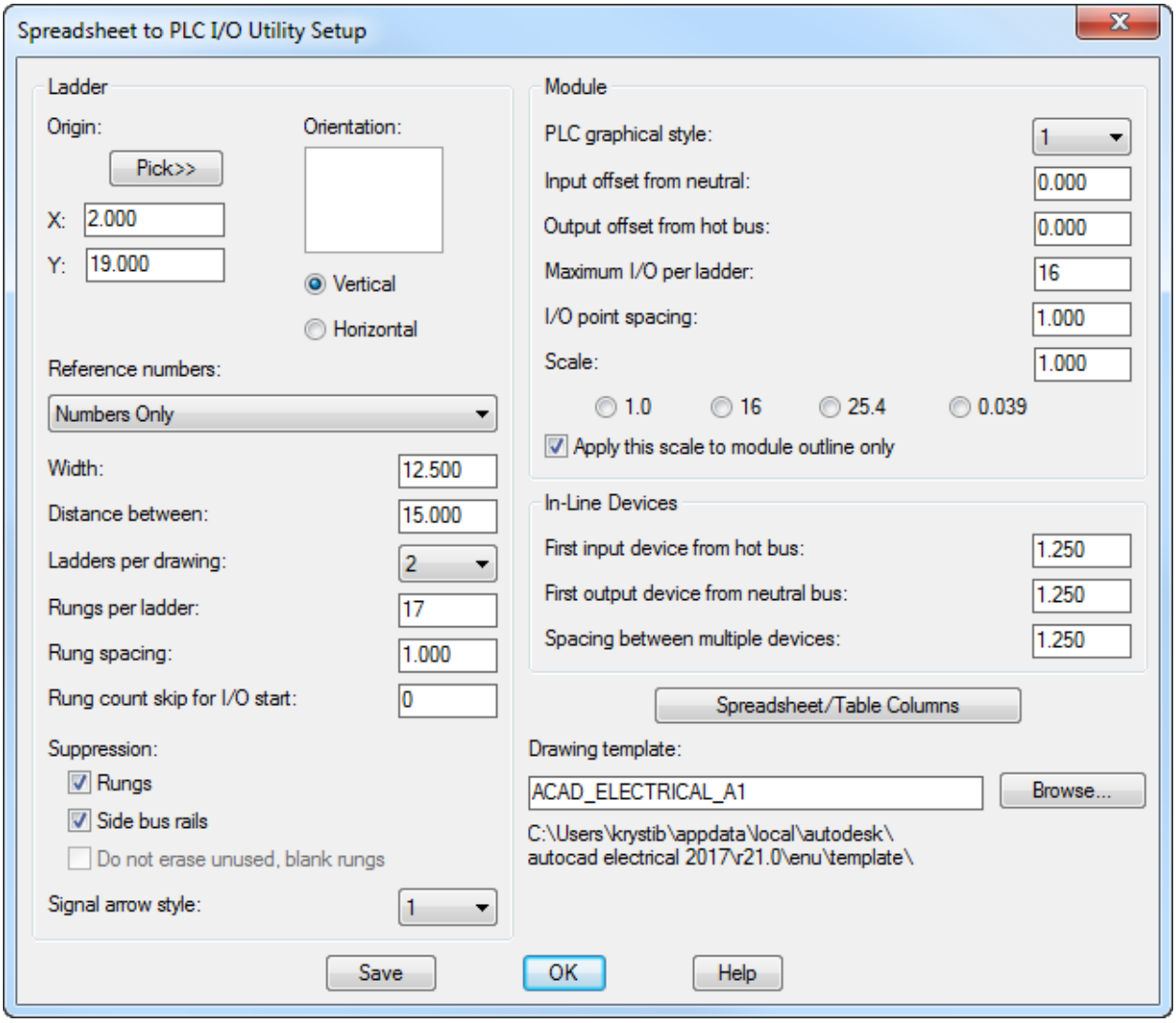


Assign 16 I/0 points to BLANK16 and 44 I/O points to BLANK44. Assign them all as the “Blank Input” Terminal Type created in the previous step.

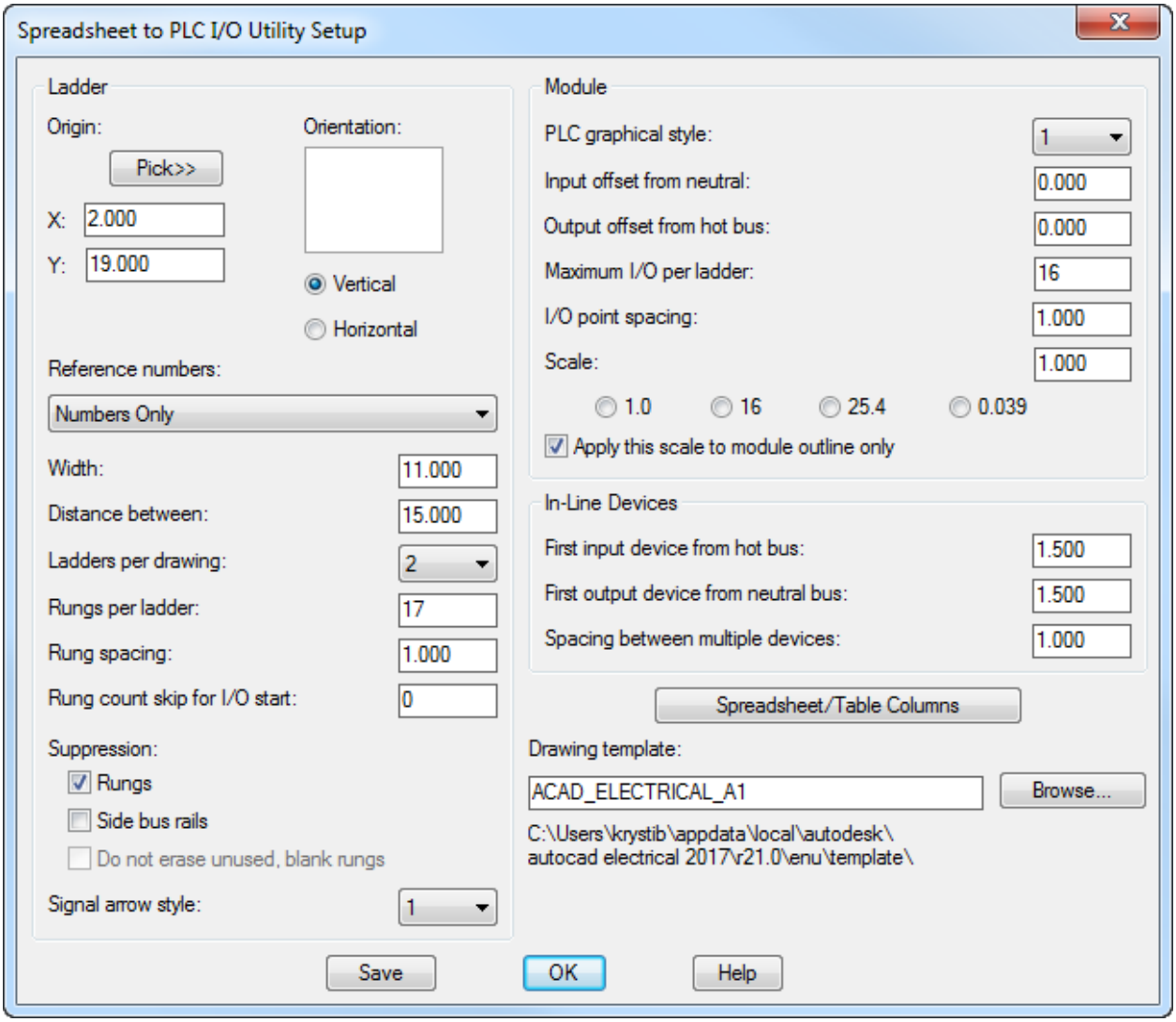


The Spreadsheet to PLC I/O Utility Setup (WDI files) for the demo are setup as follows:

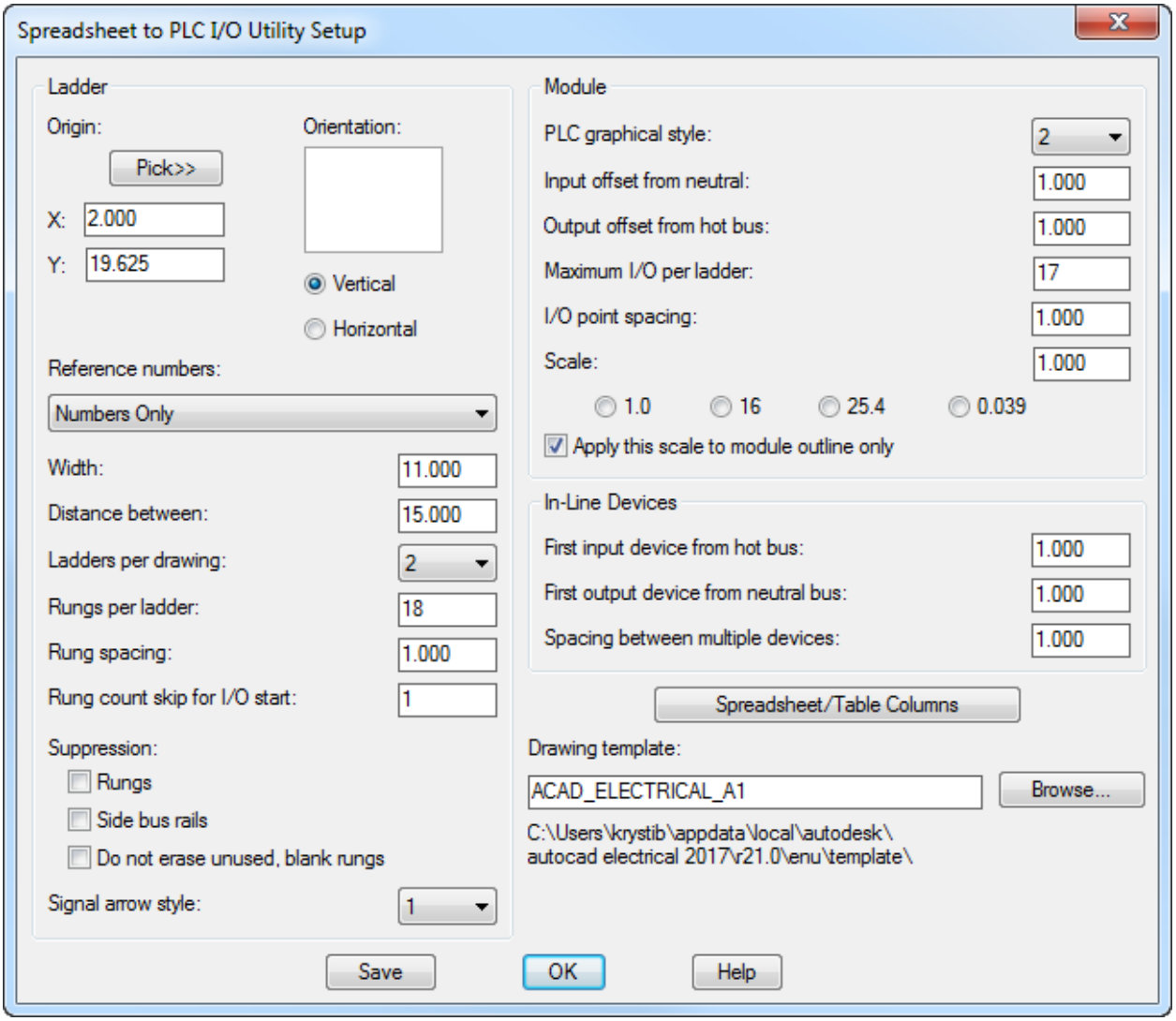
AU2016 – MTR.wdi – Used for the motor wiring and power distribution drawings:



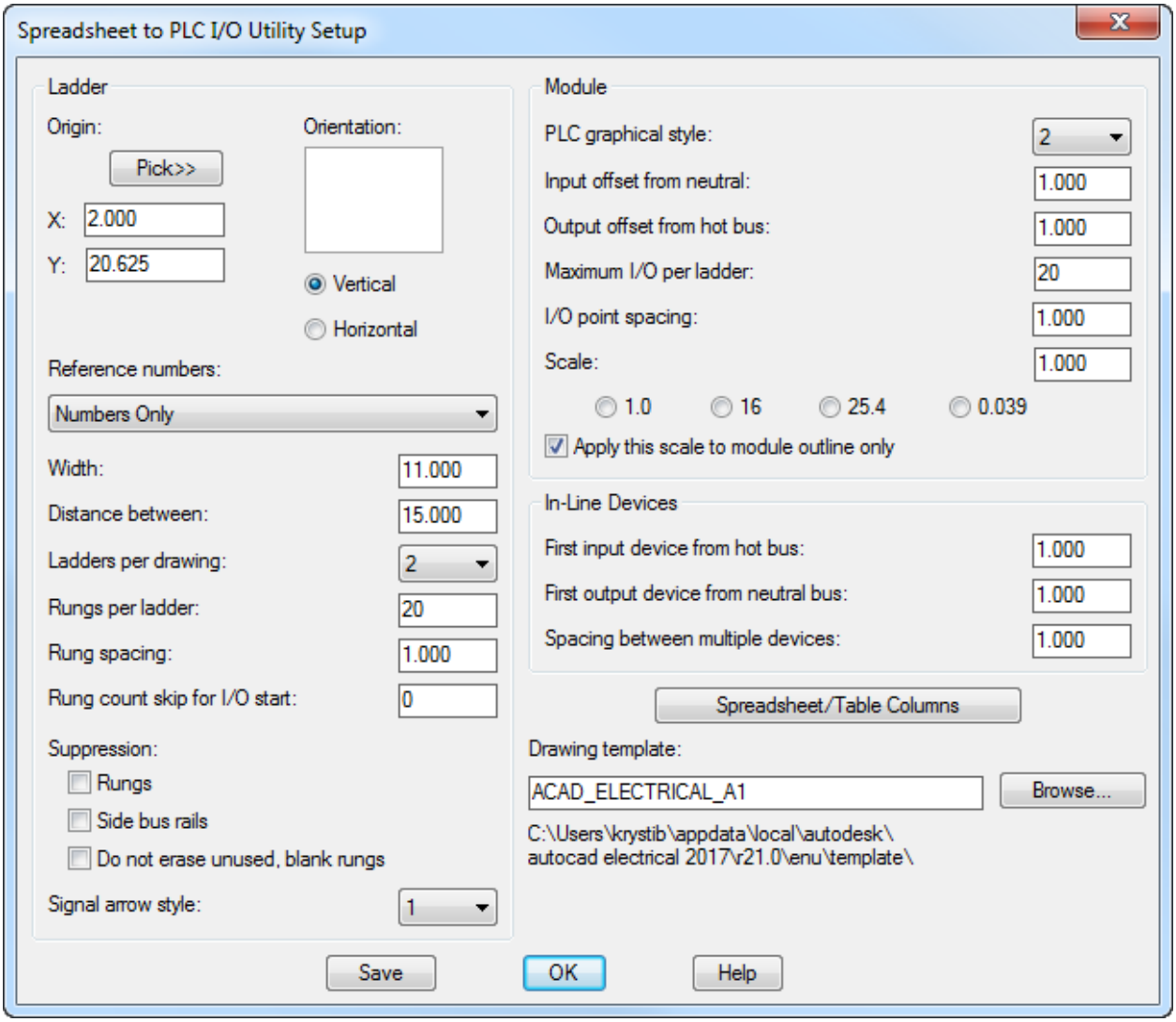
AU2016 – SCH.wdi – Used for the schematic drawings (the only difference is the Side bus rails have been added):



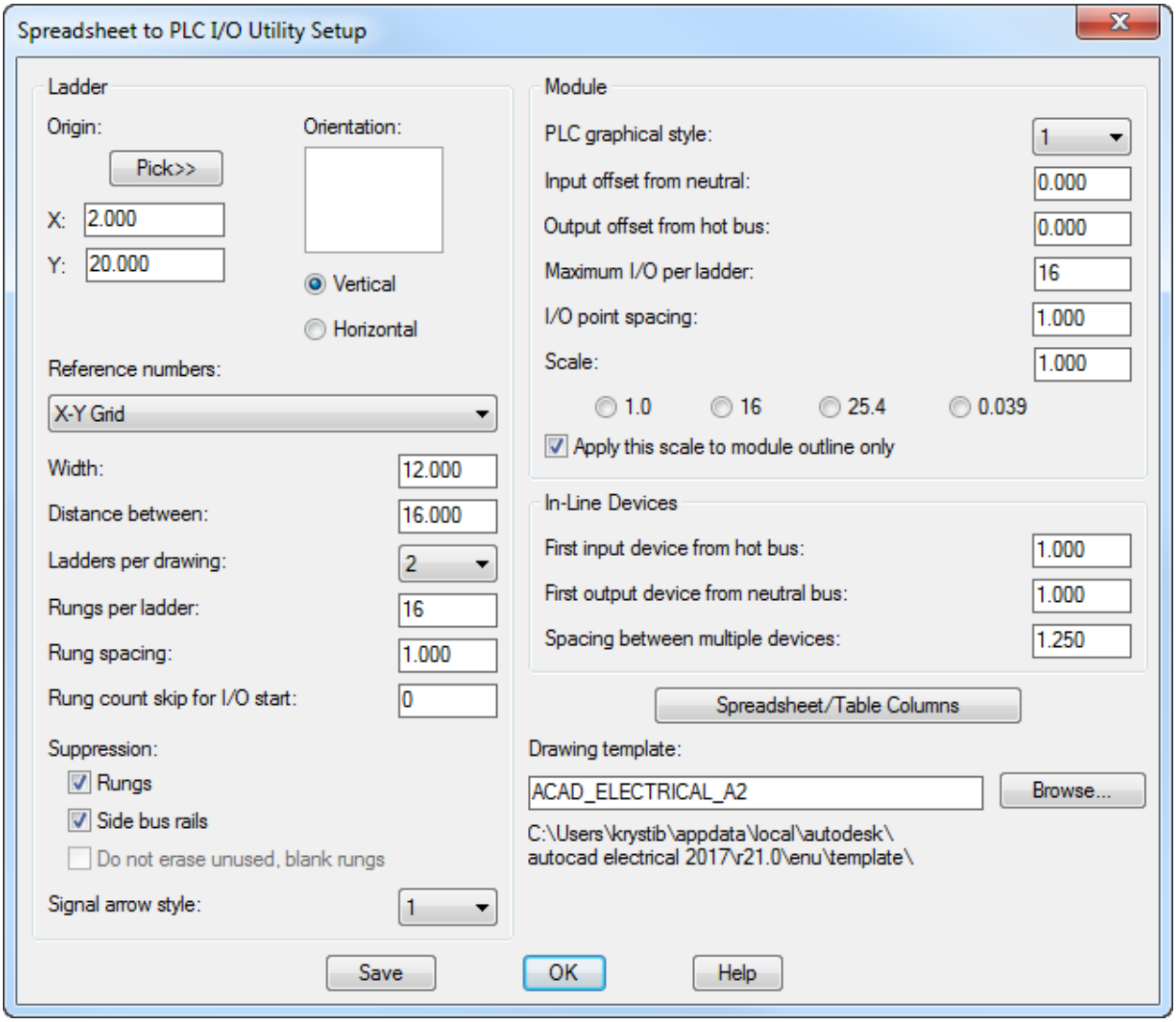
AU2016 – 1769-IO.wdi – Used for the first I/O card:



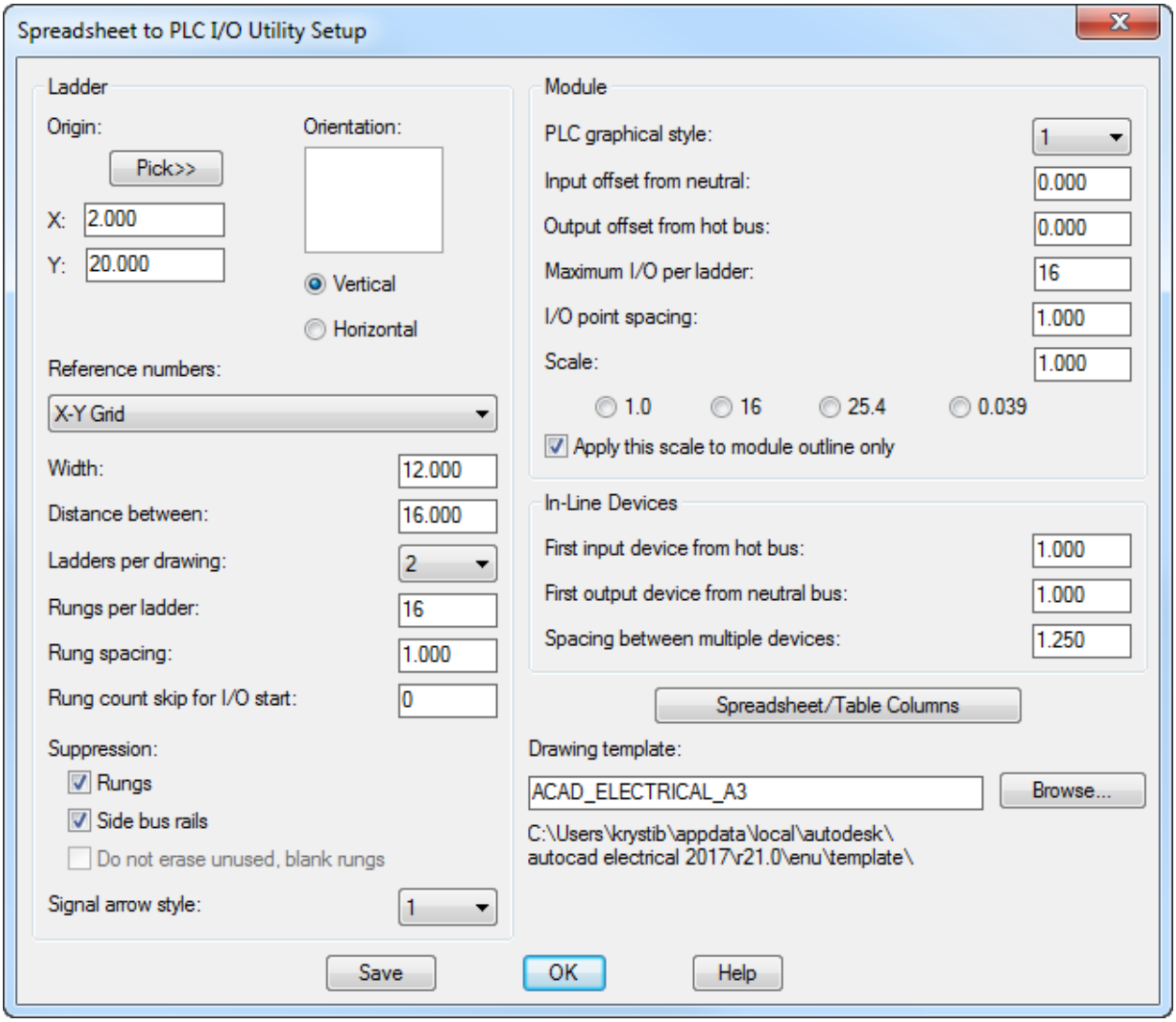
AU2016 – 1756-IO.wdi – Used for the rest of the I/O cards, showing how cards can be changed:



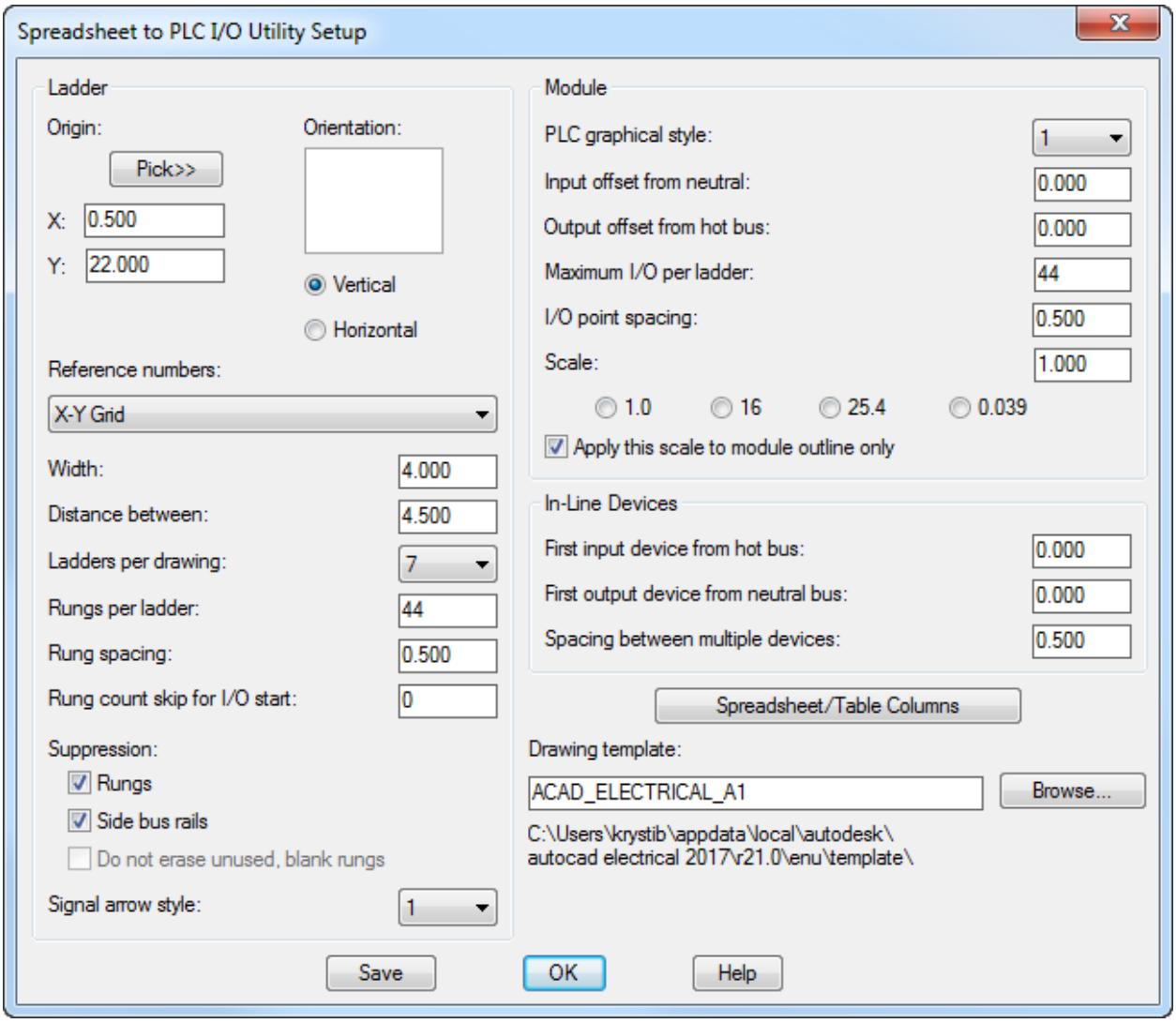
AU2016 – OPSTA.wdi – Used to add the Operator Station to the drawing set:



AU2016 – PANEL.wdi – Used to add the sub-panel to the drawing set:



AU2016 – BLANK.wdi – Used to add a symbol anywhere on the Title Block in a 0.5” x 0.5” grid. To get a blank sheet with no ladder references; use XY grid option. Make a full drawing of symbols with no ladder references:



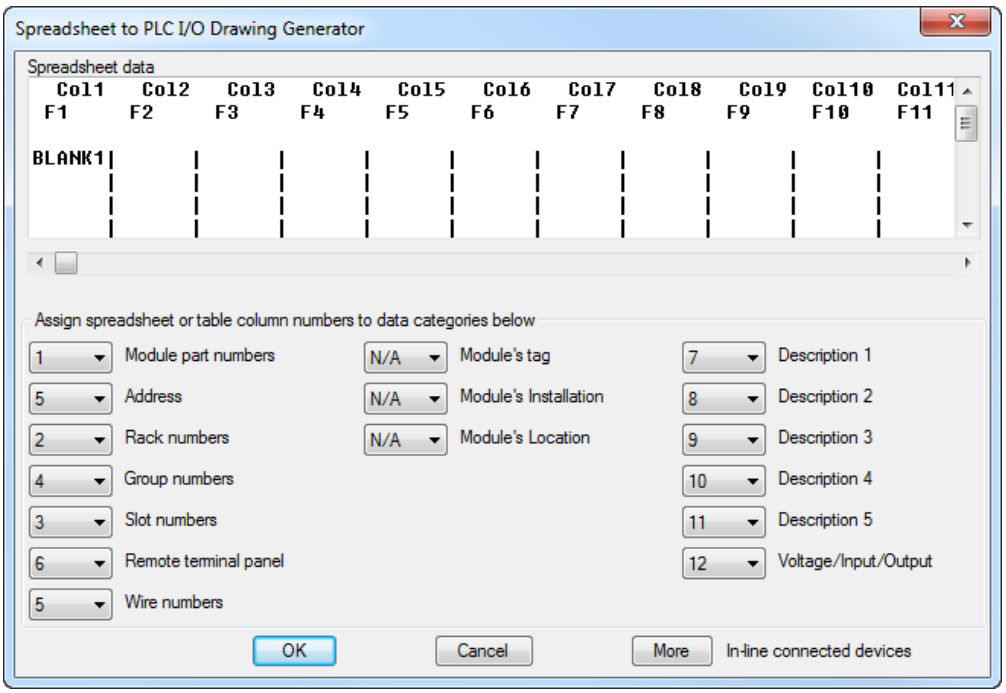
NOTE: Some modifications need to be made to the WDI file using Notepad or another text editor:

```
(setq rung_skip_btwn_module 2)
(setq refnums 4)
(setq scale 1.0000)
(setq GBL_wd_scl_plcbld 1.0000)
(setvar "TEXTSIZE" 0.1250)
(setq tsize 0.1250)
(setq useprototype "C:\\Users\\krystib\\appdata\\local\\autodesk\\autoc
\ACAD_ELECTRICAL_A1.dwt")
(setq h_or_v_rungs "H")
(setq GBL_wd_scl_plc_borderonly 1)
(setq suppress_rails 1)
(setq use_colskip 0)
(setq use_skip 1)
(setq skipcnt 100)
(setq colskipcnt 30)
```

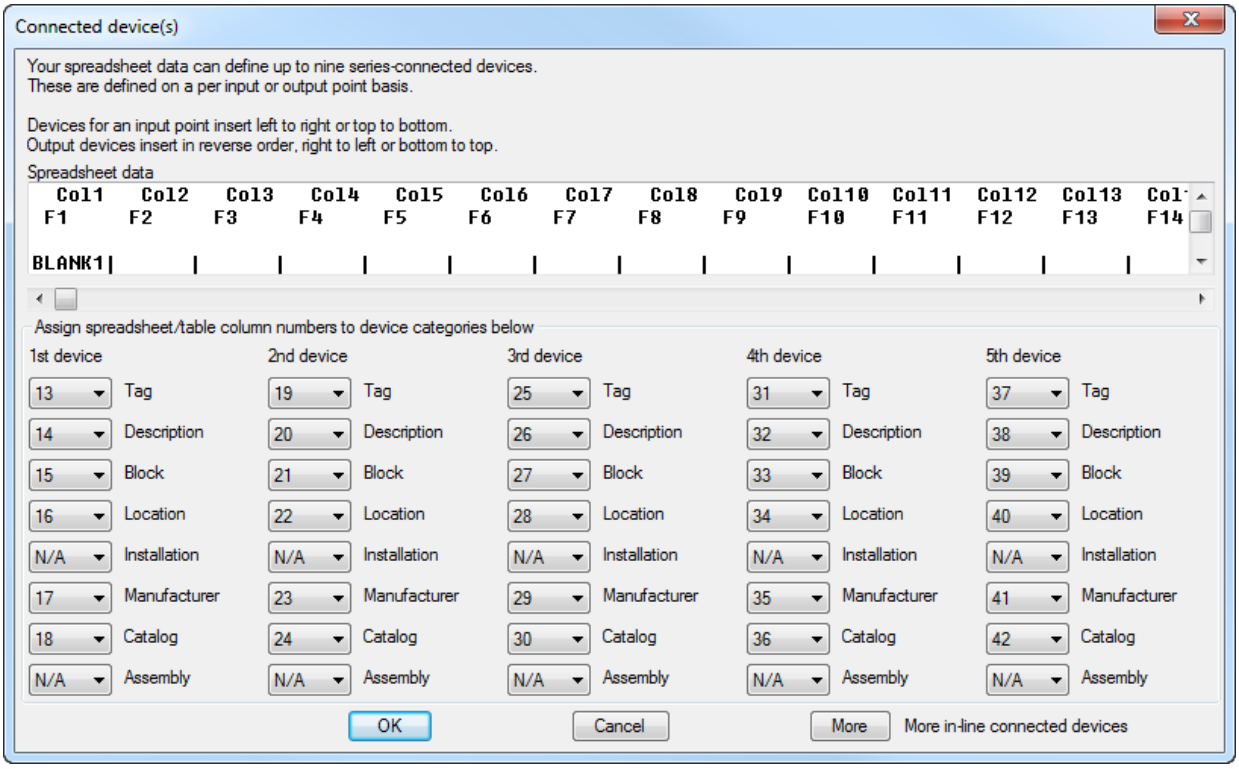
IMPORTANT! – The above step is crucial in order for the line number to work correctly. Please make sure all the WDI files are updated with the above highlighted changes.

In order to use all 9 devices on a PLC card, setup the Spreadsheet/Table Columns as follows:

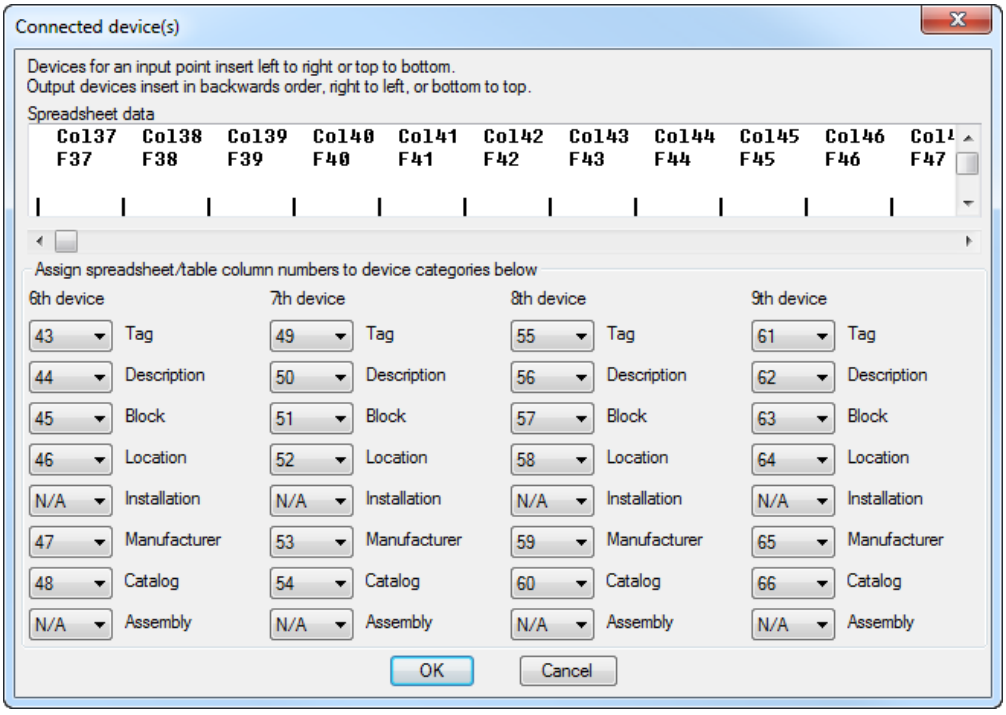
Dialog Box #1



Dialog Box #2



Dialog Box #3



AU2016 – demoplc.xls

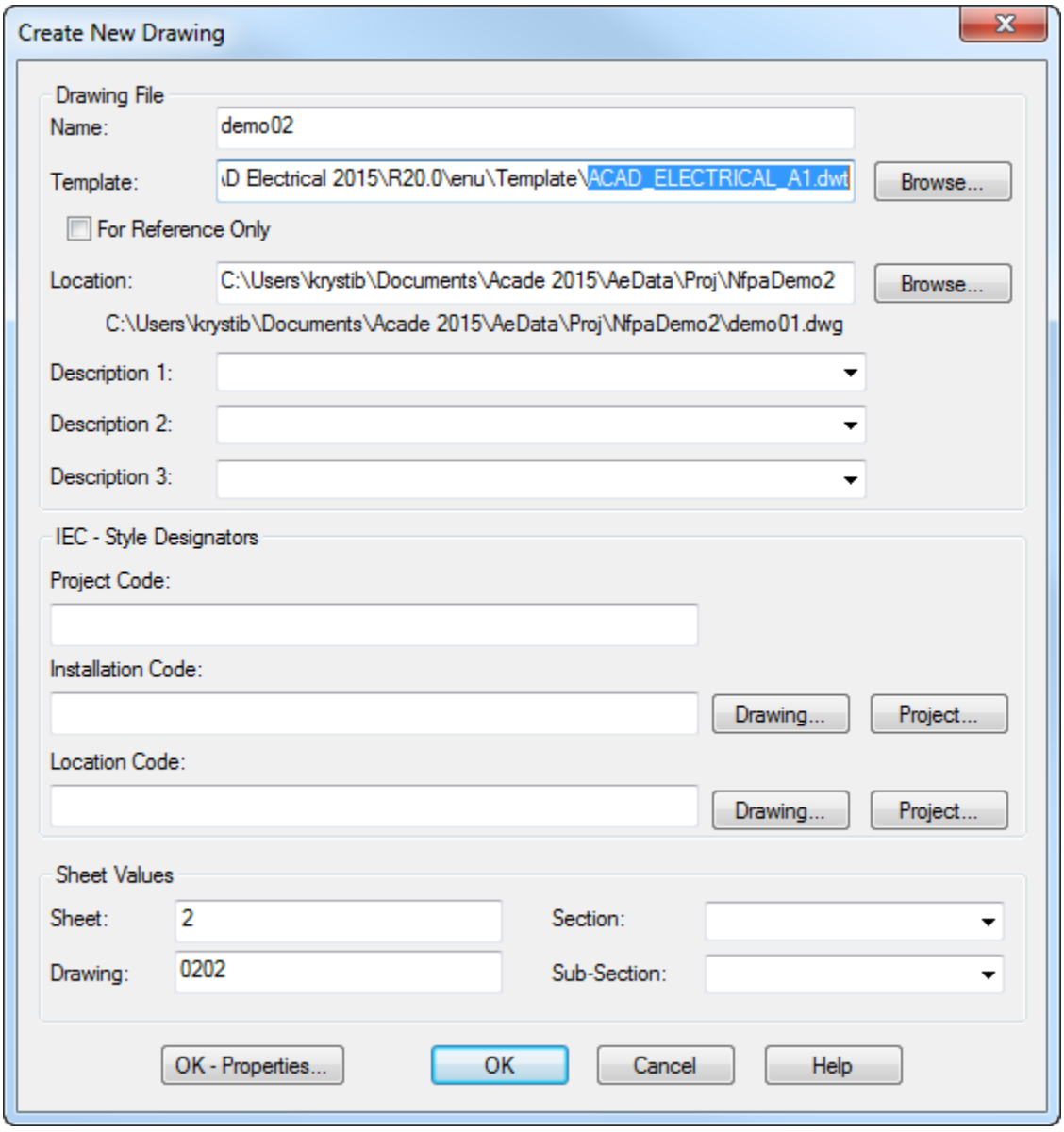
CODE Column:

1. BLANK16 – Used to create 16 rows for motor and schematic wiring
2. BLANK44 – Used to create 44 rows for specialty drawings
3. NEW_DWG= – Use this code to switch your drawing styles. This is new code written into the wdio.lsp routine.
    a. NEW_DWG=AU2016 – MTR.wdi
    b. NEW_DWG=AU2016 – SCH.wdi
    c. NEW_DWG=AU2016 – 1769-IO.wdi
    d. NEW_DWG=AU2016 – 1756-IO.wdi
    e. NEW_DWG=AU2016 – OPSTA.wdi
    f. NEW_DWG=AU2016 – PANEL.wdi
4. 1769-IA16 – Code for I/O Module
5. 1756-IA16 – Code for I/O Module
6. 1756-OA16 – Code for I/O Module
7. SKIP – Use this code at the end to execute the last wdi change.

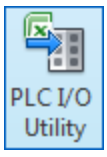
RUN THE PLC I/O UTILITY

Once the files have been downloaded and saved to the correct file paths or the above steps have been manually created, it is time to run the PLC I/O Utility.

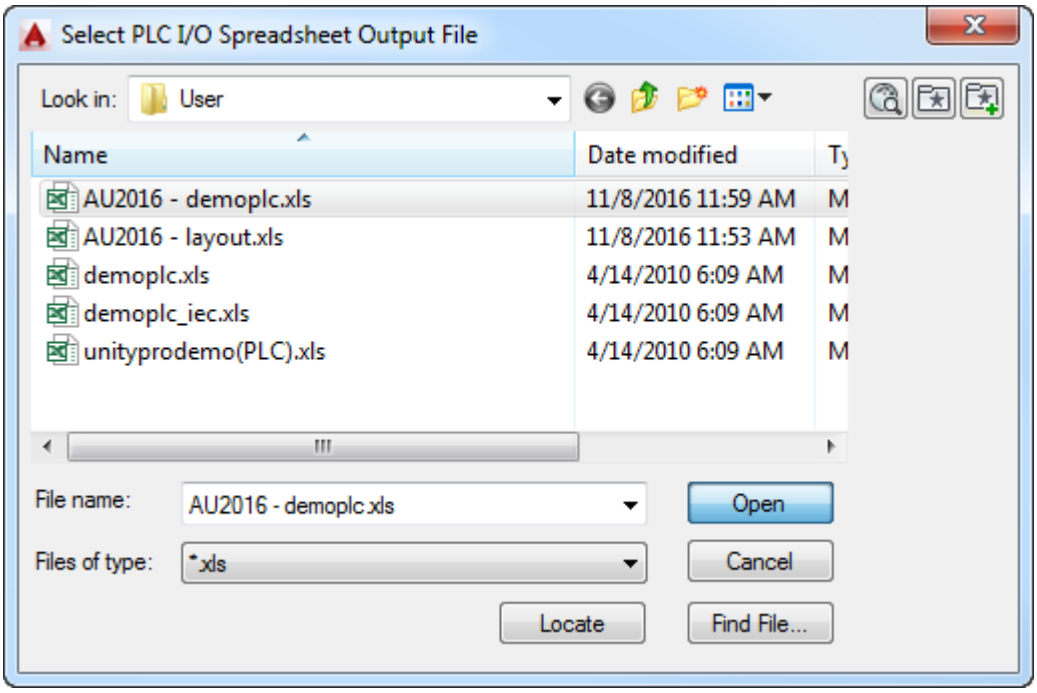
1. Open the NfpaDemo2 project and create a new drawing demo02, start with the ACAD_ELECTRICAL_A1.dwt template:



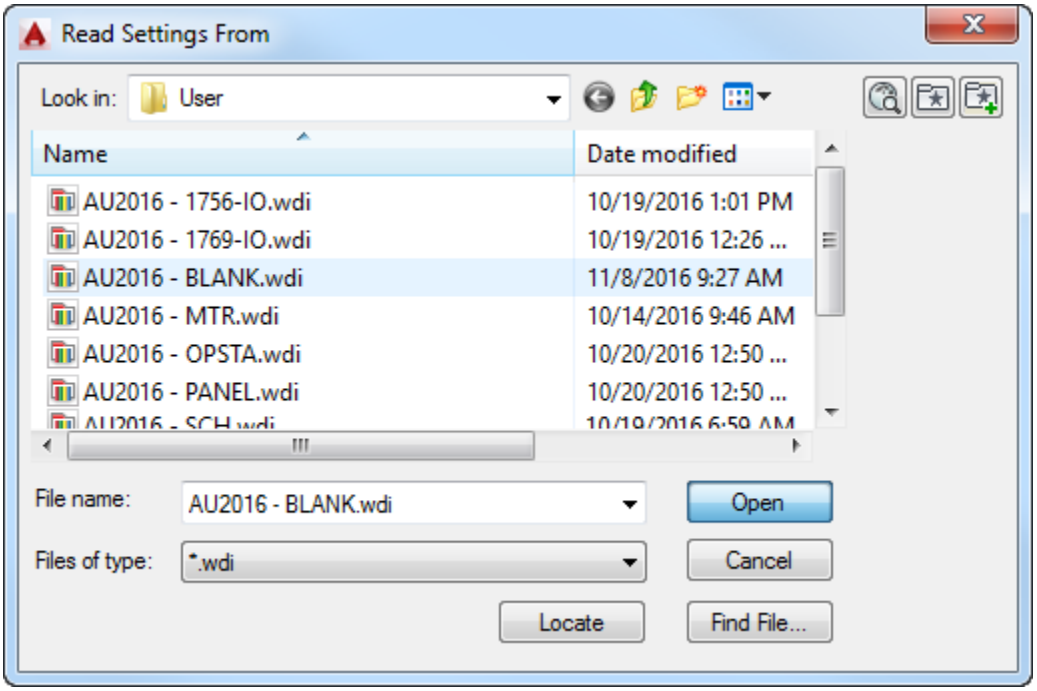
2. Start the **PLC I/O Utility**



3. Select the **AU2016 – demoplc.**xls from the dialog box



4. Click **Browse…** and select the **AU2016 – MTR.wdi**



5. In the *Spreadsheet to PLC I/O Utility* dialog box, change the following:
    a. Start: = **200**
    b. Make sure *Use active drawing* is checked
    c. Click *Free run*
    d. Sheet: = **2**
    e. Check *Add new drawings to active project*
    f. Click **Start** and watch the fun begin!