



Cables and Harnessing in AutoCAD Electrical

Randy Brunette – Autodesk, Inc.

MD5595 Modern electrical design is becoming more modular every day. Instead of individual wires attaching to individual screw terminals, pre-made harnesses are used to provide connections between components. Take this class to learn about the cable and harness tools in AutoCAD Electrical. We cover how to manipulate the tools to provide intelligent reports for rack systems, console interfaces, and other non “standard” requirements.

Learning Objectives

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

- Add cable markers, cable part numbers, and color labels
- Create Cable reports listing From/To connections
- Add wire fan in/out markers to define cable wires
- Create rack and console inter-connection designs

About the Speaker

Randy Brunette is an Autodesk Electrical Subject Matter Expert. Randy's duties include helping channel partners and customers through mentoring and understanding their business issues and finding solutions that solve their challenges. Randy has been in the design field using Autodesk products over 30 years, with experience across multiple segments of the Manufacturing industry. He has been in an Application Engineer role for 21 years.

Prior to joining Autodesk, Randy was a sole proprietor of a consulting business specializing in AutoCAD Electrical, traveling in North America and Europe providing consulting services. Randy is a top rated speaker at Autodesk Universities, Technical Academies, and seminars. He has authored AutoCAD Electrical training manuals, videos, and other materials.

Randy.Brunette@autodesk.com

Introduction

Point-to-Point wiring diagrams, terminal strips and cable representations are easier than ever to create in AutoCAD Electrical thanks to the addition of several new tools and components such as the Connector components, the Multiple Wire Bus command, and the Terminal Strip Editor.

These new tools can be combined with each other to simplify otherwise tedious tasks. This presentation will explore both the newer commands and give some insights into some of the older commands.

Wires

Connections

One of the most important tools in AutoCAD Electrical is the reporting functions. In our case, we will be looking for connectivity reports between components such as Wire From/To and Cable From/To reports. The key thing to remember when using AutoCAD Electrical to create these reports is that wires **MUST** be used to provide the connectivity between any components.

This means you always connect components by using wires. A wire must be used even if you only need to list the cables used to connect components. A wire must also be used even if the wire is not shown visibly on the drawing such as when attaching a plug to a connector. Simply, without a wire connecting components, AutoCAD Electrical will not report any connectivity.

Hints

#1-Coziness

An interesting point is that the length of the wire is not important, only that a wire exists. Even a wire as short as 0.001" will provide the connection functionality we need. We'll use this fact to make our drawings appear correct. By placing components within 0.001 of each other, they will appear to touch when at mHost common zoom factors.

#2-Name Calling

The second work around concerns the wire type used for this "hidden" wire. Since all wires show up in reports, we will create a "special" wire layer with a unique name to enable us to recognize our connection wires. For example, I use a wire type layer name "PLUG CONNECTIONS".

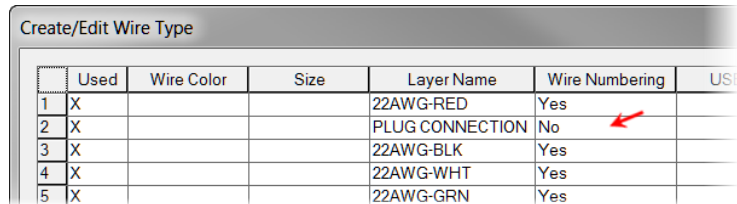
Note: *The use of a "jumper" layer does not work for our purposes. Wires located on a wire type layer that contains the letters "jumper" in it do not show up in reports.*

#3-No Labels

Turn off the automatic wire numbering is an additional tool to help simplify the drawing and reduce any confusion caused by the "hidden" connection wires. (See Wire Layer Numbering topic for details.)

Wire Layer Numbering

For both of these cases wire numbers may not be needed or wanted. Fortunately we have a setting in the Create/Edit Wire Type dialog box to disable wire numbering for a wire. With this option set to No, wire numbers will not be automatically applied to the wire. This will be especially useful for our “hidden” connection wires.



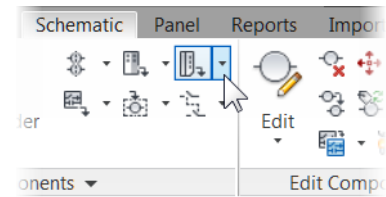
	Used	Wire Color	Size	Layer Name	Wire Numbering	USE
1	X			22AWG-RED	Yes	
2	X			PLUG CONNECTION	No	
3	X			22AWG-BLK	Yes	
4	X			22AWG-WHT	Yes	
5	X			22AWG-GRN	Yes	

Connectors

Let's start with the end of the cable, the connector. There are two basic ways to build connectors, parametric, and stand-alone.

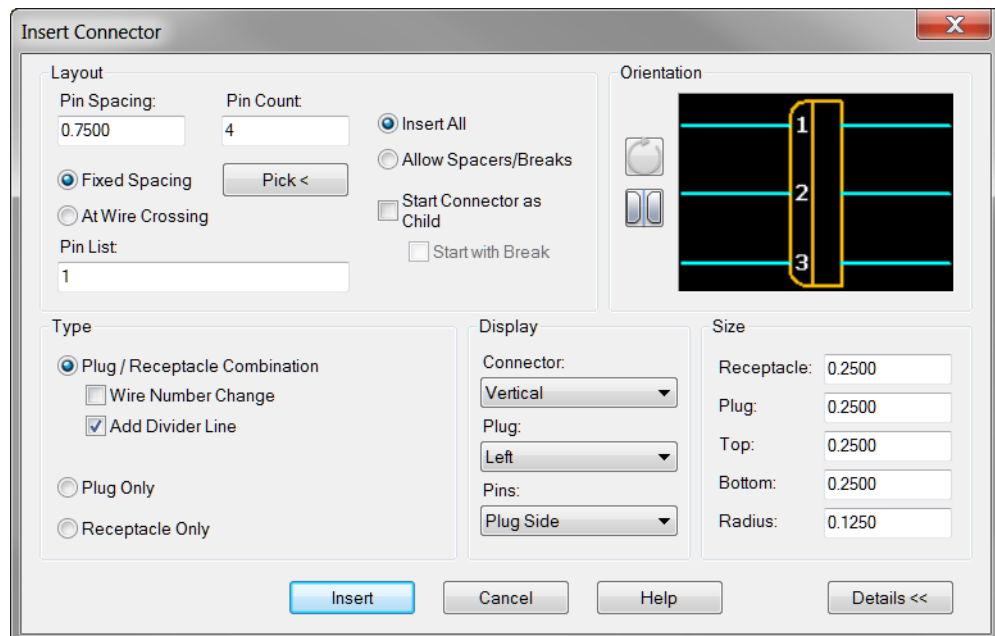
Parametric

The Insert Connector command can be used to create and edit the parametric plug and receptacle symbols used in schematic drawings.



The connector utility builds the plug parametrically based on options selected. You can fix the spacing between pins, or allow the pin spacing to be controlled by the wires the connector crosses, or a combination of both. You can break the connector into as many pieces as needed, using spacers and breaks similar to the functionality of the parametric PLC insert. The component parts broken in this manner automatically establish the parent/child relationship.

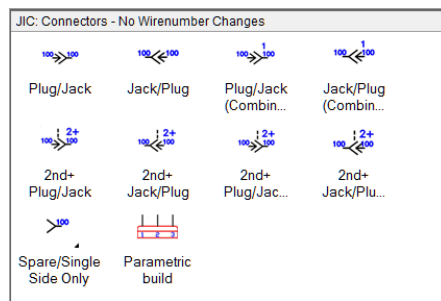
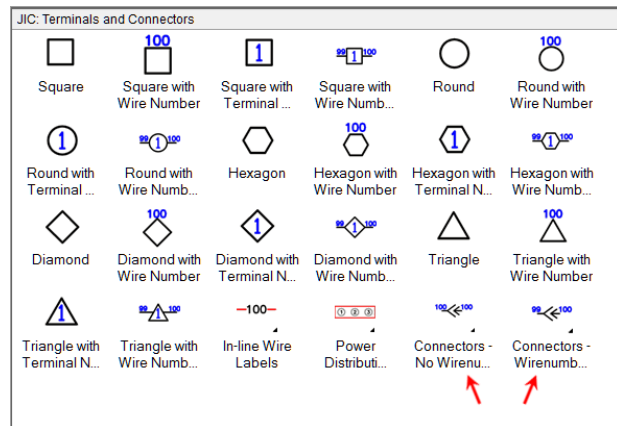
Once the pin is placed on the drawing, you can easily edit the component, changing and moving pin locations, changing pin designations, etc.



Stand-alone

Connectors can also be built up using individual pieces using plug and receptacle symbols to represent the pins assemble the connector. In this case, you insert the first pin of the connector as the parent, assigning Bill of Material and other information to it. The remaining pins are inserted as children and get associated to the parent.

Access this connector type using the Icon Menu, Terminals/Connectors sub menu.



Hint

In some cases, such as a rack connection diagram, you may want only basic connection information to appear in the reports. For example, CableEnd#1 connects to Component#1-Port#1, and CableEnd#2 connects to Component#2-Port#3.

Achieving this is as simple as changing the labels for the connection points on the components. Instead of have all the connections labeled, you would enter blanks for the unneeded points and enter the appropriate description for the single connection point required. Before and after examples are shown below.

Multiple Wire Busses and Point to Point Wiring

Once the connector is inserted, the wires need to be attached. Of course you can always use the Insert Wire tool to draw individual wires one at a time, but here's where the Multiple Wire Bus tool can be very handy.

Several options are available in the Multiple Wire Bus tool for the insertion of the wires. The base functionality works great for the insertion of three phase busses, either starting at existing busses or creating new ones.

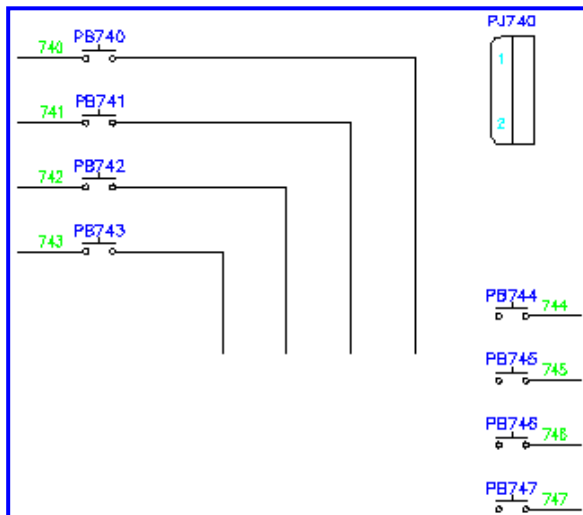
When attaching to existing components such as connectors, PLCs, switches, push buttons, etc. use the Component (Multiple Wires) option.

When using the Component option you do not need to set the number of wires. This number is determined by the connections you select for wiring during the command. Spacing for the starting points of the wires is determined by the connections selected, but the spacing settings are used as the wires traverse the drawing, make turns, etc.

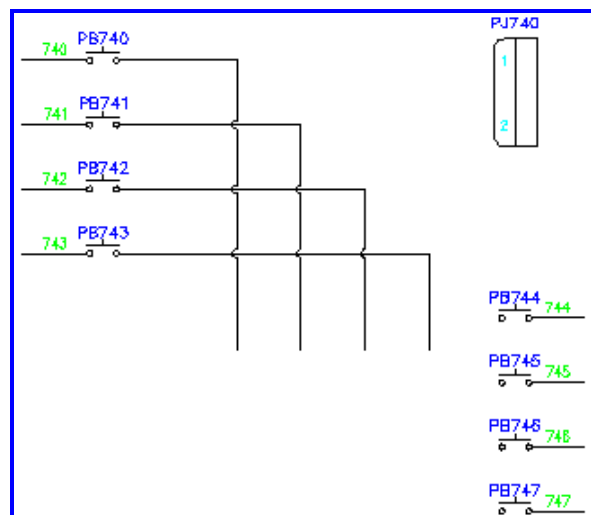
After selecting Component (Multiple Wires) and clicking OK, simply window around all of the wire connection points you want to connect to. Only one symbol orientation can be selected at a time, for example, only horizontal connections or vertical connections.

Little green X's indicating the available wire connection points are displayed during the selection process. If you zoom, these connection indicators may disappear, but don't worry the connection points are still where you expect them to be.

After selection, simply drag the wires to the desired connection points, again highlighted with the little green X's. During the dragging process you can select intermediate stopping or turning points, using the Continue option, and flip the wire layout method using the Flip option.



Before Flipping

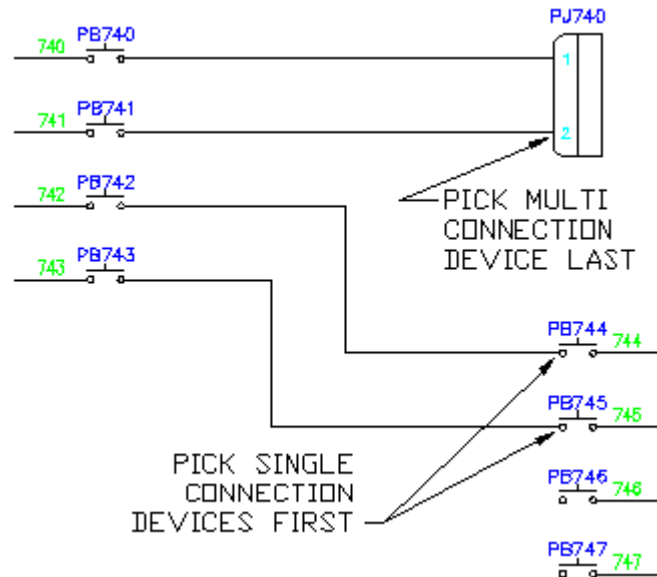


After Flipping

If you connect to a device that has multiple connection points, the utility will find the other connection points on the device and automatically connect to them. When connecting to devices with a single connection point, you select each connection point individually.

Note from your Uncle Randy:

When selecting mixed components, some with multiple connections and some with single connections, try to select all of the single connection devices first. The Multiple Wire Bus command continues to prompt for additional connection points after each selection, until it finds a multiple connection device. After finding a multiple connection device the utility connects to the connection points it finds, and ends the remaining wires at the default locations, according to the selected device and spacing settings.



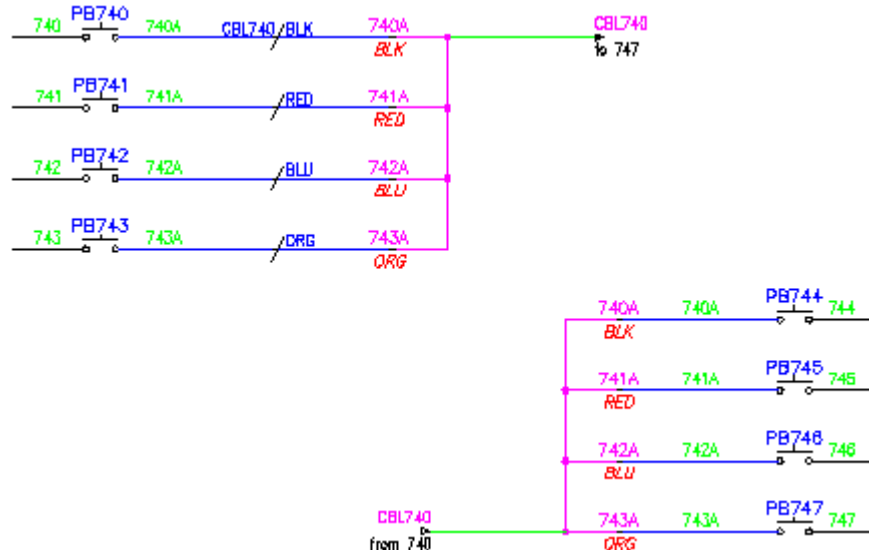
Cables and Wire Fan In/Out

The Wire Fan In/Out utility is used when you want to show a grouping of wires combining together into a single wire to cable, and when the cable separates again into the individual wires.

The basic functionality is the same as when using Source/Destination arrows. Both provide a link between wire segments located in different locations.

Source/Destination arrows are designed to be placed on the end of a wire. Fan In/Out markers are placed in-line with a wire.

Fan In/Out markers also place the outgoing or destination portion of the

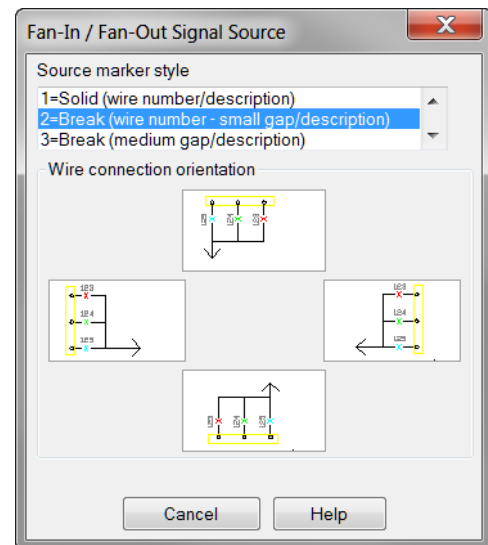
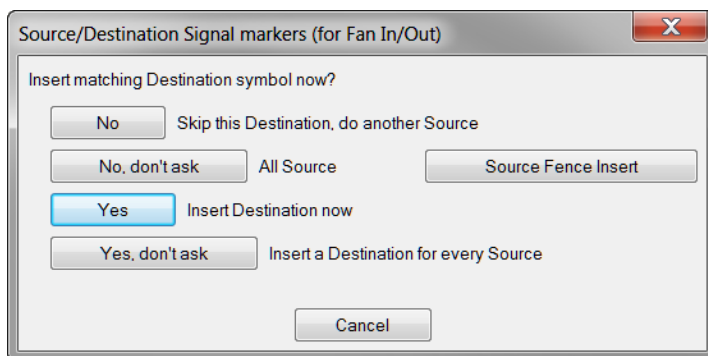


wire segment on a special non-wire layer. Because the segments are on a non-wire layer AutoCAD Electrical doesn't process the joined lines as wires connected or shorted together.

Fan In/Out Markers

The Fan In/Out utility provides the ability to place multiple source or destination markers in a single operation. If both the In and Out markers are on the same drawing and in relatively close proximity you can also place the matching marker in the same operation.

You choose the marker style and the wire connection orientation. The orientation determines which part of the underlying wire is going to remain on the wire layer and which part is placed on the multi-wire layer. Don't be concerned if the arrow representation is in the wrong location, the arrow simply indicates the side of wire that will be changed, for example, right, left, top, or bottom.



When the Fan In/Out symbol is placed, the underlying wire is broken and the destination side is placed on the multi-wire layer (described later).

Use the description field of the fan in/out markers to list the wire colors. These labels help to identify the individual cable wires on the schematic but do not show up in reports.

Note from your Uncle Randy:

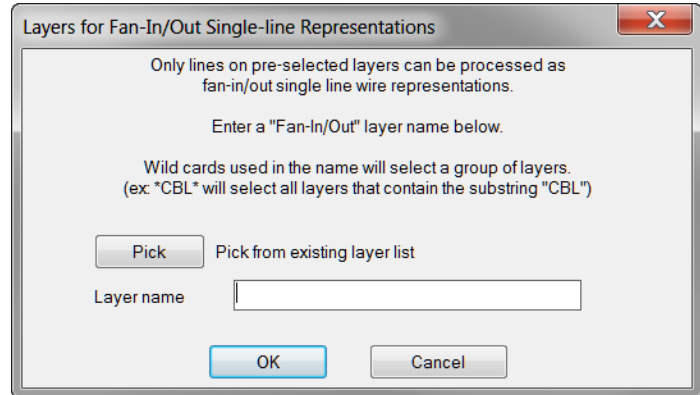
The Help file states "If you enter the color of the wire in the Description field, AutoCAD Electrical reports use this information in the Wire Color field." This is not working at this time.

Multi-Wire Layer

The multi-wire layer is necessary to display the connecting wires of the Fan In/Out. This layer is not a common wire layer and is ignored by the standard wire processing utilities. Without moving the lines to a non-wire layer, the fan wires would appear to be connected together and give undesirable results.

Layers to be used as multi-wire layers are set in the Drawing Properties dialog box under Styles. The default layer is `_Multi_Wire`.

This layer has some special properties when relating to cable and fan in/out markers. When a cable marker is attached to multi-wire network, all of the wires attached to the Fan In/Out markers are associated to the cable. The Cable From/To report will list the cable name in addition to the wire from/to information.



Cables and Cable Markers

In AutoCAD Electrical there is no actual “cable” object. A cable marker itself becomes the cable object that is listed in reports. All of the Bill of Material and other information is stored on attributes in the cable marker.

Cable markers are usually placed on wires that show the cable routing. This allows the cables to use the wire routing utilities to create connection lists, etc.

Cable conductor symbols have a specific naming convention:

The first character is an “H” or a “V” for a horizontal or vertical orientation.

The second and third characters are “W0”, representing a wire or cable.

The fourth character is a “1” or a “2”. A “1” for a parent marker, and a “2” for a child.

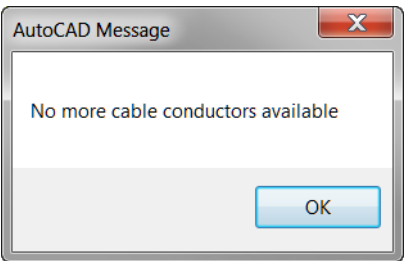
All remaining characters are unspecified and can be used to uniquely name the symbol.

This is a similar naming convention to terminals that do not change the wire number as it passes through the symbol.

The conductors in a cable can be listed in the catalog database table “_W0_CBLWIRES”. Enter each conductor for a cable in the order you normally want to insert them. AutoCAD Electrical uses both the W0 table, which lists the cable part number and the _W0_CBLWIRES table together to provide error checking similar to the use of pin lists.

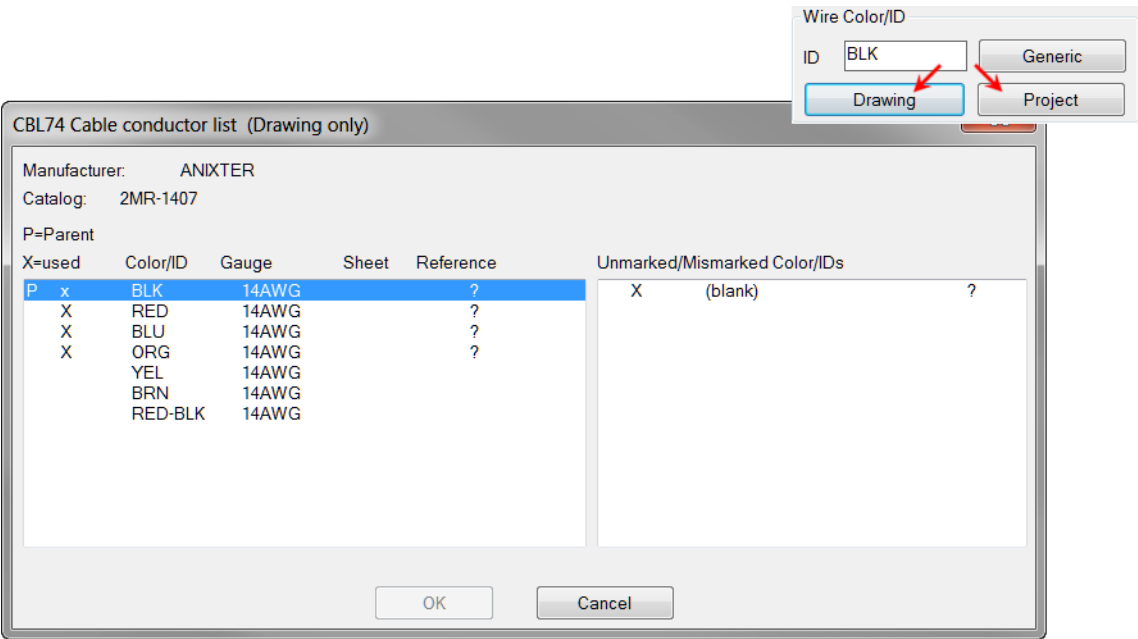
CATALOG	MANUFACTU	CONDUCTOI	GAUGE	RE
1172C	ALPHA	BLK	22AWG	
1172C	ALPHA	WHT	22AWG	
1173C	ALPHA	BLK	22AWG	
1173C	ALPHA	WHT	22AWG	
1173C	ALPHA	RED	22AWG	
1174C	ALPHA	BLK	22AWG	
1174C	ALPHA	WHT	22AWG	
1174C	ALPHA	RED	22AWG	
1174C	ALPHA	GRN	22AWG	
1175C	ALPHA	BLACK	22AWG	
1175C	ALPHA	WHITE	22AWG	

With this information and the use of cable markers, you can track which conductors of a cable have been used. Cable colors are automatically entered in the order listed in the table. When you try to insert more conductors than are available, a message box appears.



Dismissing the dialog enables you to bypass the error checking and insert a generic marker.

You can check for which conductors have been used or miss-marked by using the Wire Color/ID Used: Drawing or Project tools.



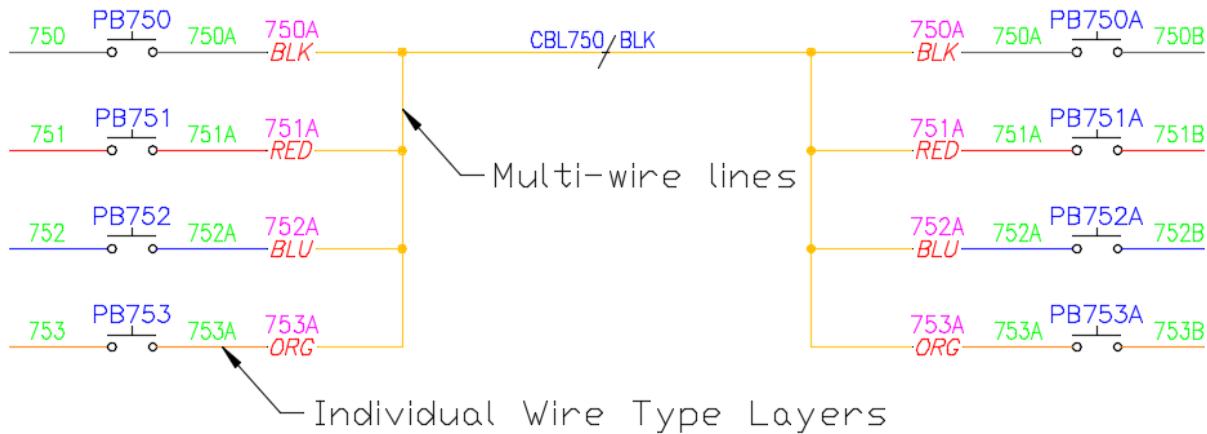
Note from your Uncle Randy:

Currently, once the “No Conductors Available” flag is tripped, the flag is not reset when you correct the problem. The dialog box continues to re-appear. Just dismiss the warning dialog box and do a Used: Drawing or Project lookup. The available wires are listed correctly in the list.

There are several methods available for having wiring information appear in reports.

Method 1, Cable parent only, Wire Layers, Local destination

This example has the Cable marker parent placed on the Multi-line layer. Wires going into the Fan In/Out markers are use a standard wire layer for each cable wire. For example, you might use layers named BLK_18GA_Cbl3Cond, RED_18ga_Cbl3Cond, and WHT_18ga_Cbl3Cond. The destination markers are shown locally (on the same drawing). **(Connected.)**

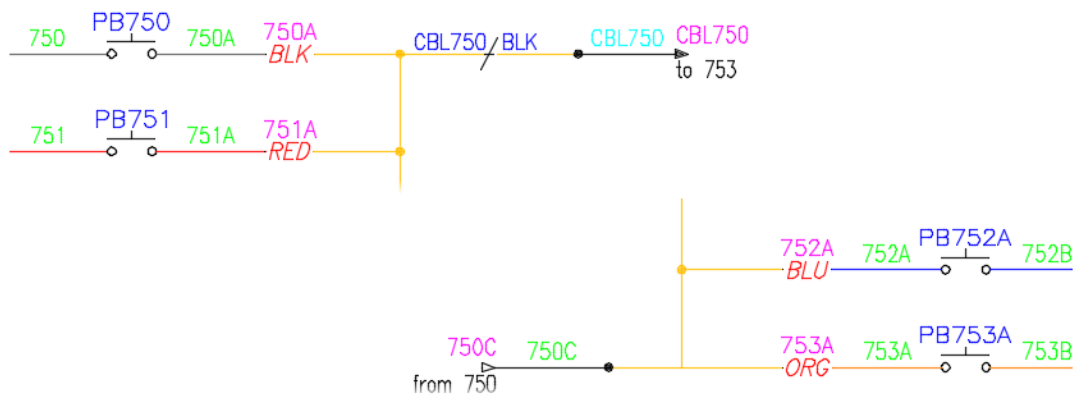


CBL	CBLMFG	CBLCAT	WLAY1	WIRENO	CMP1	CMP2
CBL750	ANIXTER	2MR-1404	BLK-18AWG	750A	PB750	PB750A
CBL750	ANIXTER	2MR-1404	RED-18AWG	751A	PB751	PB751A
CBL750	ANIXTER	2MR-1404	BLU-18AWG	752A	PB752	PB752A
CBL750	ANIXTER	2MR-1404	ORG-18AWG	753A	PB753	PB753A

Remote destination

This example will show the same report results as above but illustrates the solution to a different problem.

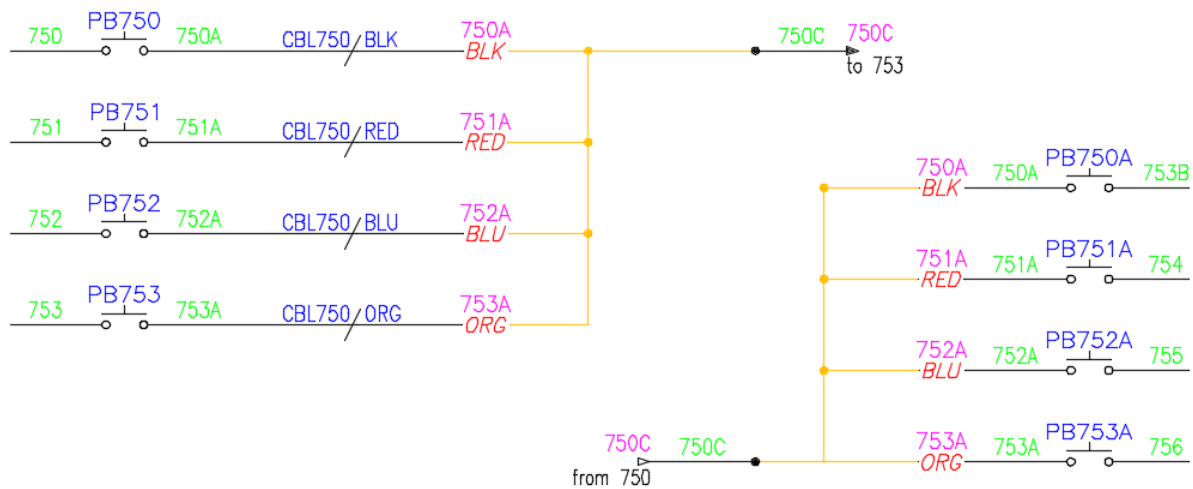
Source/Destination arrows do not recognize a multi-wire layer as a wire. A solution is to place a wire segment attached to the multi-wire lines. You can label this wire with a fixed wire number to identify the cable as shown. You can also hide the wire number if desired.



Method 2, Cable children, Cable wire layer

This example shows the use of a cable marker and its children to identify the wires of a cable. Insert the cable markers on only one side of the circuit. The description fields of the Fan In/Out markers are helpful here to identify wire colors on the schematic.

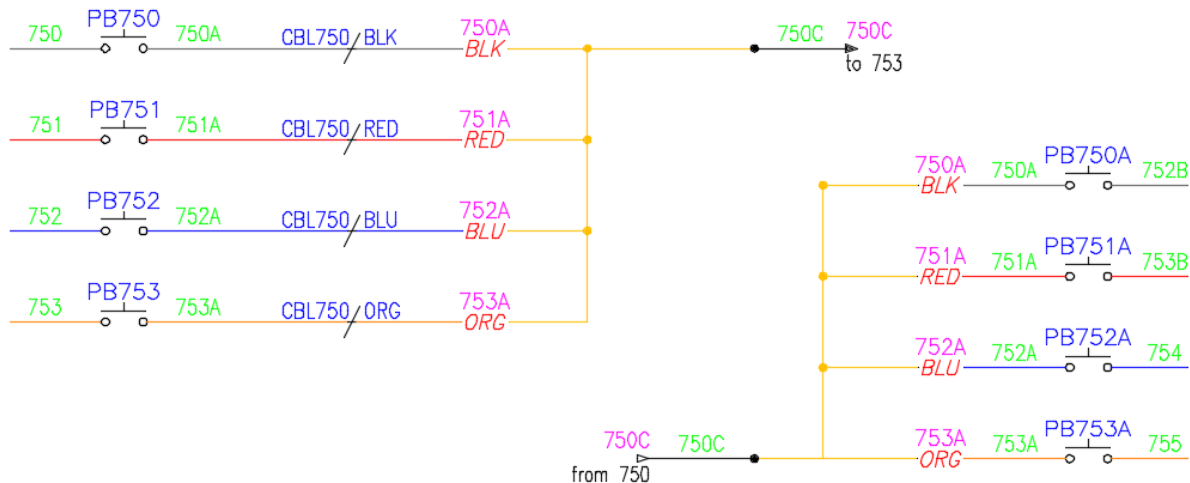
Using this method, individual wire layers are not required. The cable wire colors will appear in reports as the CBLWC field.



CBL	CBLMFG	CBLCAT	CBLWC	WLAY1	WIRENO	CMP1	CMP2
CBL750	ANIXTER	2MR-1404	BLK	WIRES	750A	PB750	PB750A
CBL750			RED	WIRES	751A	PB751	PB751A
CBL750			BLU	WIRES	752A	PB752A	PB752
CBL750			ORG	WIRES	753A	PB753	PB753A

Bonus Example

This example shows the use of a cable marker and its children to identify the wires of a cable and the use of individual wire layers. Some information such as CBLWC and WLAY1 can be similar.



CBL	CBLMFG	CBLCAT	CBLWC	WLAY1	WIRENO	CMP1	CMP2
CBL750	ANIXTER	2MR-1404	BLK	BLK-18AWG	750A	PB750	PB750A
CBL750			RED	RED-18AWG	751A	PB751	PB751A
CBL750			BLU	BLU-18AWG	752A	PB752A	PB752
CBL750			ORG	ORG-18AWG	753A	PB753	PB753A

Summary

AutoCAD Electrical has a tremendous amount of flexibility and functionality built into it. This paper showed you how much power some of the commands have, such as the Multiple Wire tool automatically finding multiple connection points, and how to use the rules of AutoCAD Electrical to your own advantage, such as the No Wire Number option.

The final note from your Uncle Randy is "If it seems like there should be a better way, there probably is." You may just need to explore and experiment to find it.

Presentation Planning

1. Powerpoint with hints.
2. Use Multiple Bus to add wires
 - a. Change Wire types.
3. Add Cable marker with BOM data.
4. Add plugs
 - a. Plugs require wires to attach to Component
5. Wires are not desired in reports, Plug to Component list
 - a. Change wires to Plug Connection type
 - b. Set Plug Connection to No Wire Number
6. Show Fan In/Out
 - a. Cable Marker on _Multiwire layer (special or false wire layer)
 - b. Move Cable Marker to _Multiwire layer