GEN20454

A Quick Spin on AutoCAD Electrical 2017

Randy Brunette
Autodesk, Inc.

Learning Objectives

- Learn about the Project Manager and its functions
- Discover basic schematic creation
- Discover basic panel layout functions
- Learn about error checking and reporting capabilities

Description

This is a class for people who are unfamiliar with AutoCAD Electrical software. We will give an overview of the functionality and the efficiency AutoCAD Electrical software provides for electrical controls design. We will lightly cover the Project Manager, basic schematic creation, panel layouts, and reporting tools. If you are new to AutoCAD Electrical software, or just getting started, this class will be a good place to start! This session features AutoCAD Electrical.

Your AU Expert

Randy Brunette joined Autodesk, Inc., as an 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 for 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 the sole proprietor of a consulting business specializing in AutoCAD Electrical software, 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 software training manuals, videos, and other materials.

Working with Standard Symbols and Library Parts

Overview

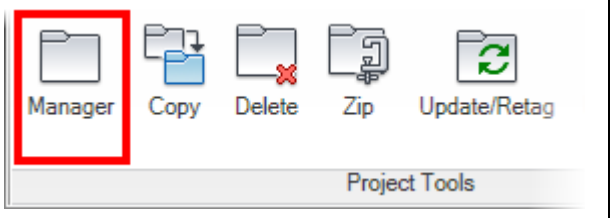
A schematic drawing can contain hundreds of symbols and parts. Since 90% of these consist of standard off-the-shelf components, AutoCAD Electrical software ships with more than 2,000 standards-based schematic symbols, with over a million parts within the manufacturers catalog database. This means you can quickly create schematics for standards-based controls designs.

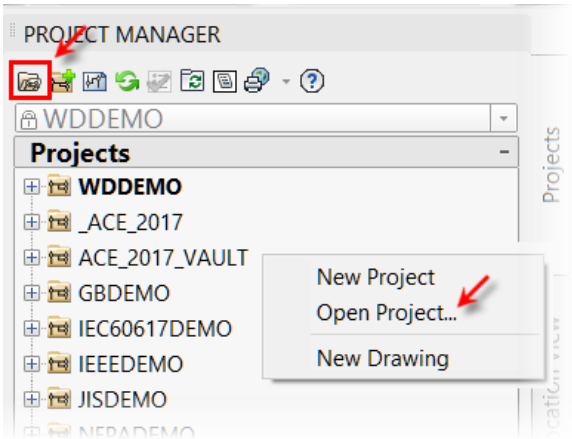
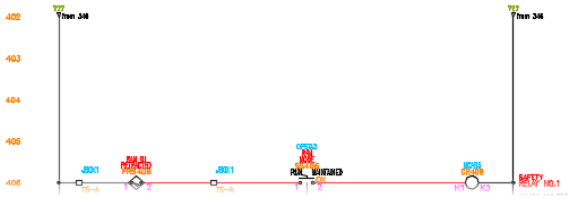
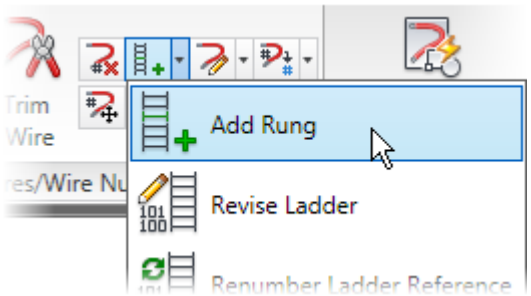
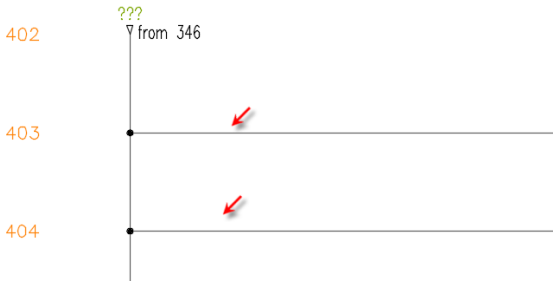
A simple, icon-menu-driven system for inserting electrical, pneumatic, hydraulic, and P&ID devices provides an interface for you to quickly build schematic drawings using simple pick-and-place methods. These "intelligent" symbols automatically break the insertion wire and create full connectivity throughout the project.

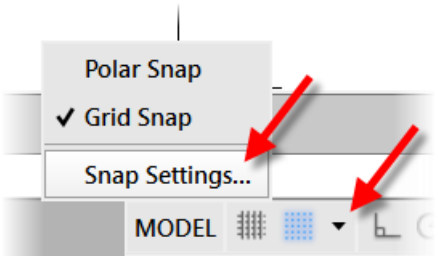
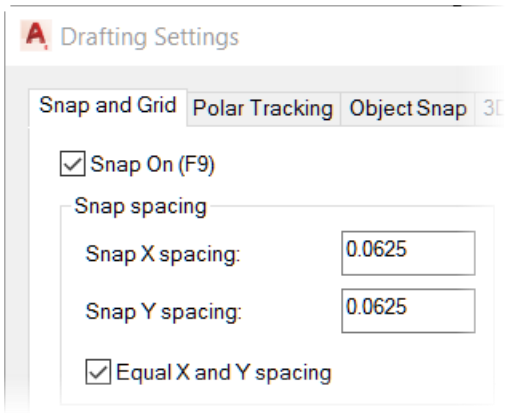
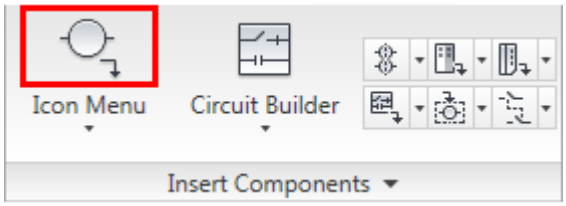
AutoCAD Electrical includes all the functionality of AutoCAD plus a comprehensive set of drafting features developed specifically for designing electrical control systems. Specialized features such as Trim Wire, Copy Component, Copy Circuit, Scoot, and Align, make it much easier to create drawings quickly.

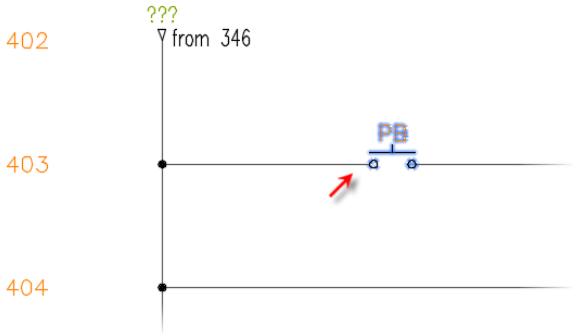
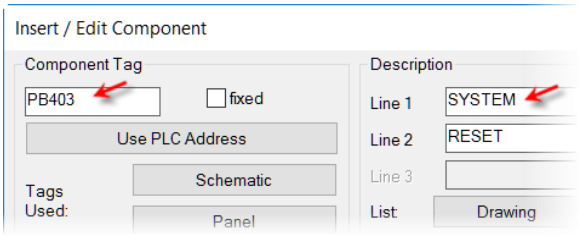
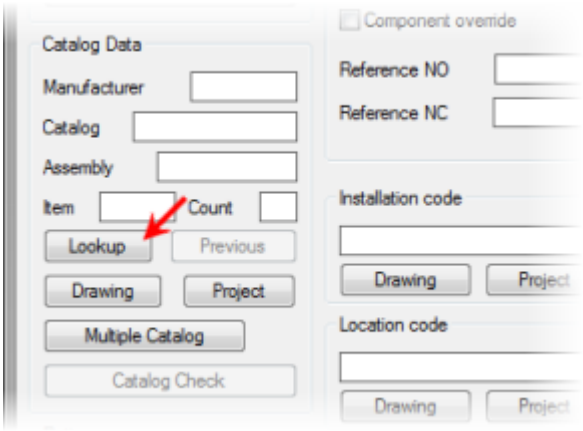
An AutoCAD Electrical project is used to organize and manage all of the drawings that make up an electrical project, including schematic, panel layout, and wiring diagrams. The project file stores drawing file names and locations, project configuration settings, drawing order, and other information important to the electrical design.

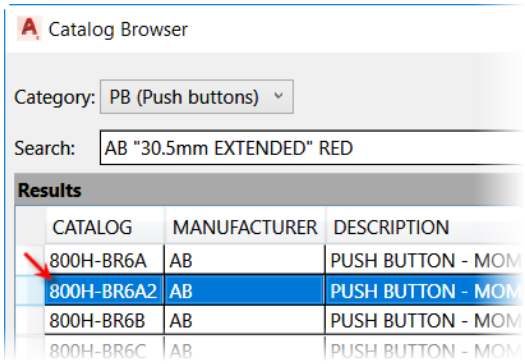
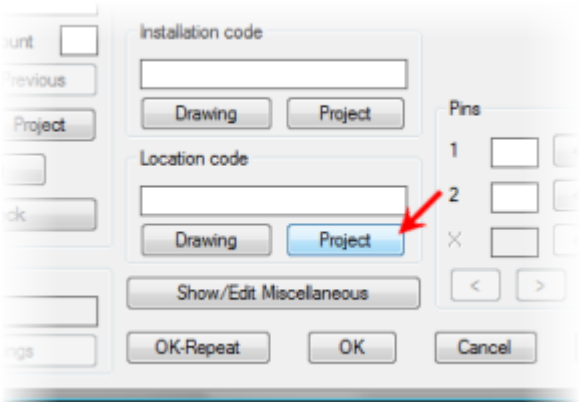
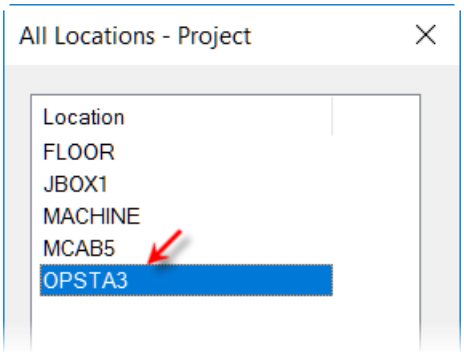
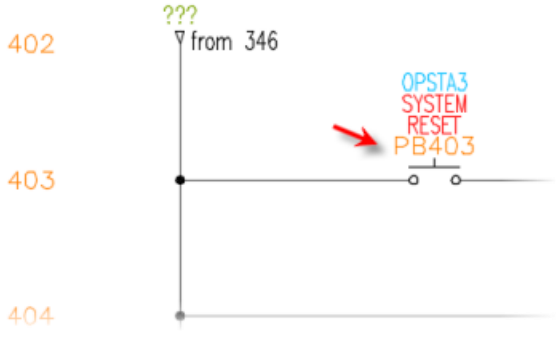
*Exercise-Insert Pushbuttons using Icon Menu and Catalog Browser

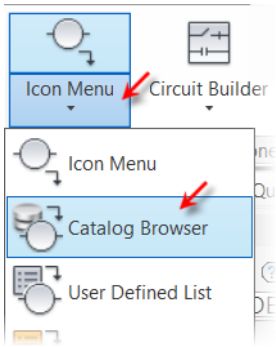
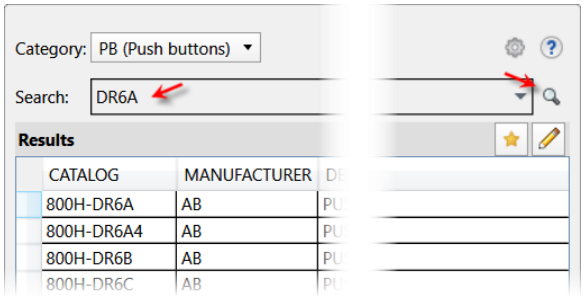
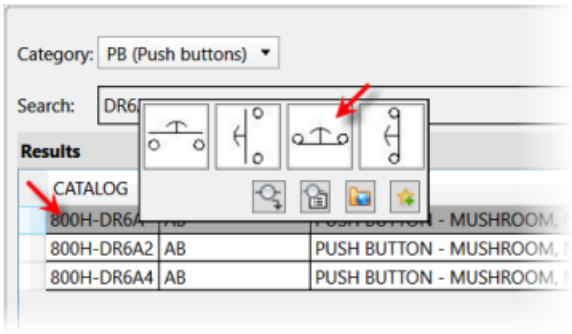
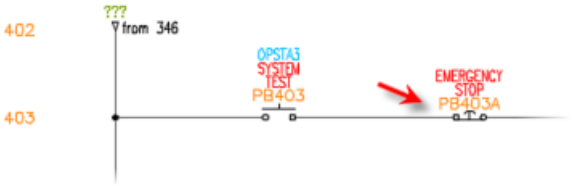
In this lesson, you start editing an existing drawing. You insert rungs and two push buttons, using the Icon Menu to sa parent symbol, and a child symbol, and link them together.	
1. If the Project Manager is not displayed, on the Project tab, Project Tools panel, click Manager.	

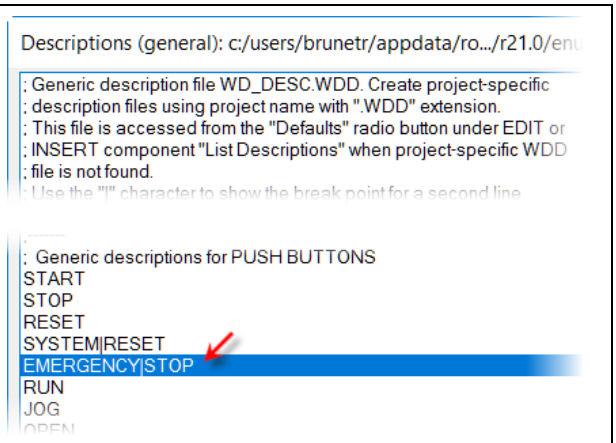
2. In the Project Manager, right-click in an empty area. Click Open Project. Or click the Open Project button. **3.** Browse to the A Quick Spin on AutoCAD Electrical folder. Select *Quick_Spin_On_AcadE.wdp*. Click Open. **4.** In the Project Manager, expand Quick_Spin_On_AcadE to view the sub-folder list. **5.** Expand Schematic to view the drawing list.	
6. Right-click Quick_Spin_2017_04.dwg. Click Open. **7.** Zoom into rungs 402-405.	
8. On the Schematic tab, Edit Wires/Wire Numbers panel, click Add Rung. *Ribbons are an easy to use interface that displays the command icon and the command name all the time. Ribbons commands are designed and located to assist in your design workflow providing additional productivity.*	
9. In the drawing, select insertion points on rung references 403 and 404. ***Note:*** *Select somewhere between the vertical bus and the PLC. Do not select directly on the vertical bus.* *Notice that wire lengths and termination nodes are created automatically.*	

10. ***Note:*** *Verify that Osnap is turned off before insertion. It is helpful during placement of components to have Snap turned on. Recommended setting, 0.0625.* On the Controls menu, (on the bottom right) click the Snapmode flyout, Snap Settings.	
11. In the Drafting Settings dialog box, on the Snap and Grid tab, under Snap Spacing, Select Snap On (F9) **12.** Enter 0.0625 for both Snap X Spacing and Snap Y Spacing. Click OK.	
13. On the Schematic tab, Insert Components panel, click Icon Menu.	
14. In the Insert Component dialog box, under NFPA: Schematic Symbols, click Push Buttons.	
15. Under NFPA: Push Buttons, click Push Button NO.	

16. In the drawing, select an insertion point on rung 403, near the left side. *The component symbol is inserted and the underlying wire is automatically trimmed to the wire connection points on the component. Intelligent wire connections are created in the schematic.*	
17. In the Insert/Edit Component dialog box, • *Notice that the component TAG is automatically created based upon the reference number of the rung you placed it on.* • Under Description, for Line 1, enter **System**. • For Line 2, enter **Test**. *Notice how all description text is automatically capitalized. This is an optional setting.*	
18. *Next, you will assign a catalog number to the component from more than 1 million entries in the default part catalog database that ships with AutoCAD Electrical. The specific custom components your company uses can also be easily added to the database.* Under Catalog Data, click Lookup.	

19. In the Parts Catalog dialog box, select 800H-BR6A2. Click OK. *Descriptions, manufacturer data, physical locations, and many additional specifications can be added for each individual symbol. All of this information is available for export into specialized reports.*	
20. In the Insert/Edit Component dialog box, under Location Code, click Project.	
21. In the All Locations-Project dialog box, select OPSTA3. Click OK.	
22. In the Insert/Edit Component dialog box, click OK.	

You can also insert components directly by part number from the Catalog Browser. **23.** On the Schematic tab, Insert Components flyout, click Catalog Browser.	
24. In the Catalog Browser dialog box, for Category, from the flyout, select PB (Push Buttons) **25.** For Search, enter **DR6A**. *This is only a partial part number.* **26.** Click Search.	
27. From the list, select 800H-DR6A. **28.** From the image menu select the horizontal normally closed symbol.	
29. In the drawing, select an insertion point on rung 403 to the right of PB403.	

30. In the Insert/Edit Component dialog box, for description, click Defaults. **31.** In the Descriptions dialog box, select EMERGENCY\STOP. Click OK. **32.** In the Insert/Edit Component dialog box, click OK. **33.** Close the Catalog Browser dialog box. *This completes the exercise.*	

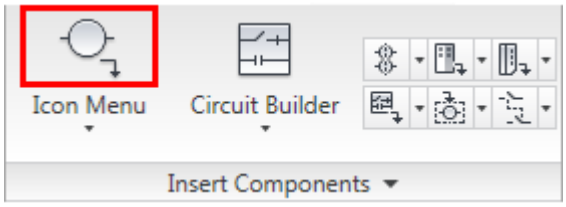
Cross-referencing Coil and Contacts in Real Time

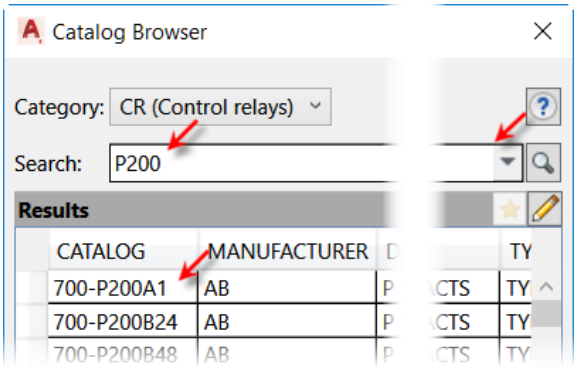
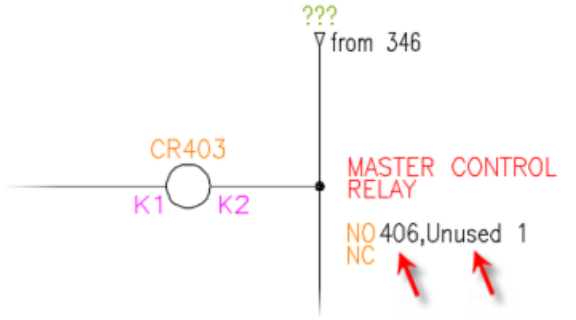
Overview

In basic AutoCAD, you must manually find the tag-ID of the parent components, such as relay coils, decide on the next available set of contact pin number assignments for the child components, such as relay contacts, then update the cross-reference annotation at both ends. This can be time-consuming when you have to search for the correct drawing where the components might reside, especially within a large project set containing many drawings.

Using AutoCAD Electrical you can reduce the risk of assigning too many contacts to any component, and minimize time spent manually tracking assignments. AutoCAD Electrical sets up parent/child relationships between components, keeping track of how many contacts are assigned to any coil or multi-contact device, and alerting you when the limit has been exceeded.

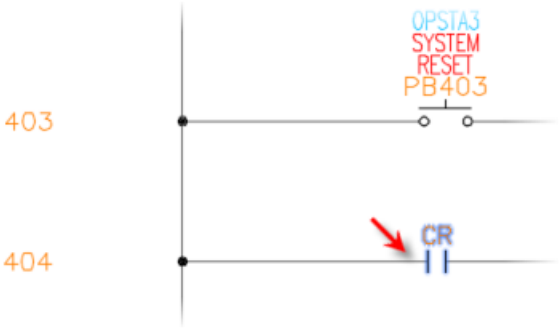
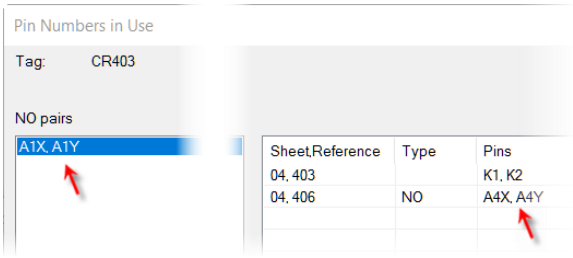
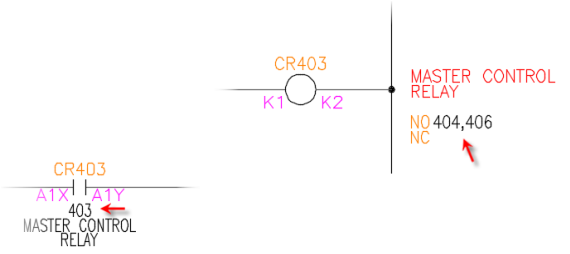
*Exercise-Insert Parent Coil

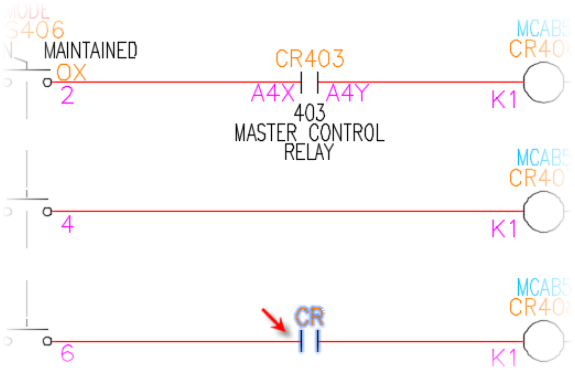
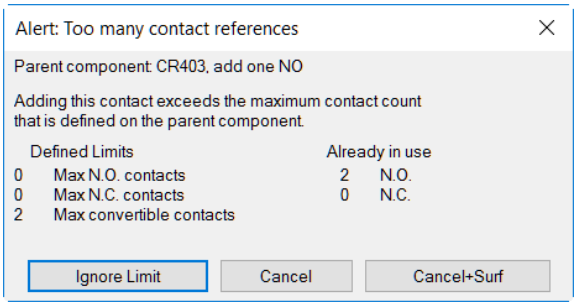
In this lesson, you insert a parent relay coil, and select a catalog part number assignment that includes pin list data.	
1. On the Schematic tab, Insert Components panel, click Icon Menu.	
2. In the Insert Component dialog box, under NFPA: Schematic Symbols, click Relays/Contacts.	
3. Under NFPA: Relays and Contacts, click Relay Coil.	

4. In the drawing, select and insertion point on rung 403, near the right side, directly above the other coils. **5.** In the Insert/Edit Component dialog box, for Description, click Defaults. **6.** In the Descriptions dialog box, select Master Control|Relay. Click OK.	
7. In the Insert/Edit Component dialog box, Under Catalog Data, click Lookup. **8.** In the Catalog Browser, for Search, enter **P200A1**. Click Search. **9.** Select 700-P200A1. **10.** Scroll to the right to the Contacts column. *This part contains only two normally open contacts.* **11.** Click OK.	
In the Insert/Edit Component dialog box, under Pins, notice this part number contains pin list data for the coil and contacts. **12.** Click OK.	
13. If prompted, in the Assign Symbol dialog box, click Map Symbol. *Notice how an existing contact was located in the project and the cross reference information automatically updated the coil. Also notice the optional listing of the unused contact.* *This completes the exercise.*	

Exercise-Insert Relay Child Contacts

In this exercise, you insert a relay child contacts and link them to the parent relay coil. Data, including pin list information is transferred to the child component. When inserting more contacts than available on the parent coil an alert is displayed.

1. On the Schematic tab, Insert Components panel, click Icon Menu. **2.** In the Insert Component dialog box, under NFPA: Schematic Symbols, click Relays/Contacts. **3.** Under NFPA: Relays and Contacts, click Relay NO Contact.	
4. In the drawing, select an insertion point on rung 404 near the left side. **5.** In the Insert/Edit Child Component dialog box, under Component Tag, click Parent/Sibling. **6.** In the drawing, select anywhere on CR403.	
Notice all information from parent coil is copied to the child contact. **7.** Under Pins, click List. **8.** *In the Pin Numbers In Use dialog box, notice all contacts for the relay are listed, both used and the one remaining NO pair.* Click OK.	
9. In the Insert/Edit Child Component dialog box, click OK.	

10. Repeat steps 1-3 above. **11.** In the drawing, select an insertion point on rung 408 directly under the child contact CR403. **12.** In the Insert/Edit Child Component dialog box, under Component Tag, click Parent/Sibling.	
13. In the drawing, select on any CR403 coil or contact reference. **14.** *Since only two contacts were available for this part number and both were already used, the Too Many Contact References alert dialog box is displayed.* Click Cancel. *This ends the exercise.*	

Automating Wire Numbering and Component Tagging

Overview

Many hours can be consumed manually assigning wire numbers, and then renumbering them as design changes are made throughout the course of the schematic design.

Using AutoCAD Electrical you can automatically assign unique wire numbers and component tags in your drawings, and reduce errors and the time you spend tracking design changes. AutoCAD Electrical automatically places sequential or reference-based numbers on all wires and components based on the chosen configuration. AutoCAD Electrical can determine if an inserted wire number will "bump into" anything and, if needed, will automatically add leaders from the wire number to the wire.

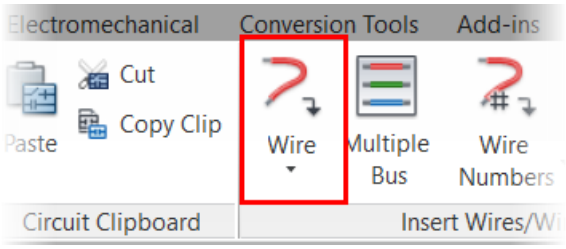
Avoid costly errors before the build phase begins by catching and removing errors during design. AutoCAD Electrical monitors and alerts users to potential design errors as they occur by constantly comparing the requested changes with the current project.

*Exercise-Automatic Wire Numbering and Component Tagging

In this lesson, you will add wire numbers to wires in the active drawing. Then you change the format for the component tags and retag the components in the active drawing.

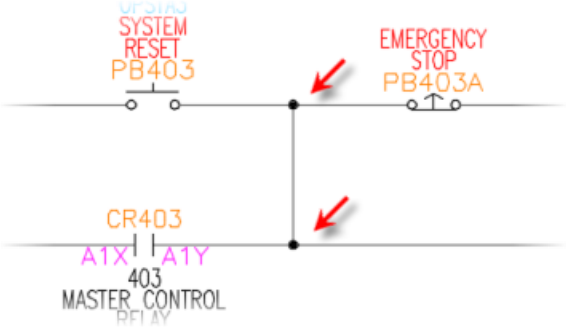
First we need to add and trim some wires to complete the circuit.

1. On the Schematic tab, Insert Wires/Wire Numbers panel, click Wire.

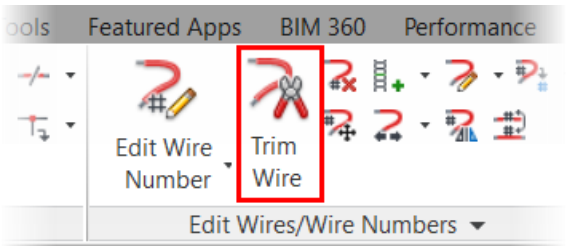


2. In the drawing, select a starting point for the wire on rung 403 in the middle between PB403 and 403A.

3. Select the endpoint for the wire directly below the start point on rung 404.

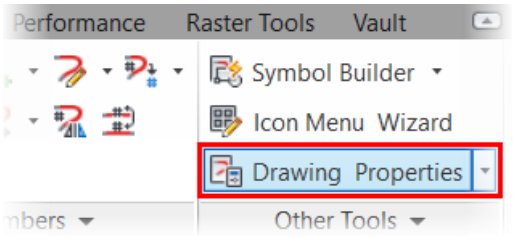
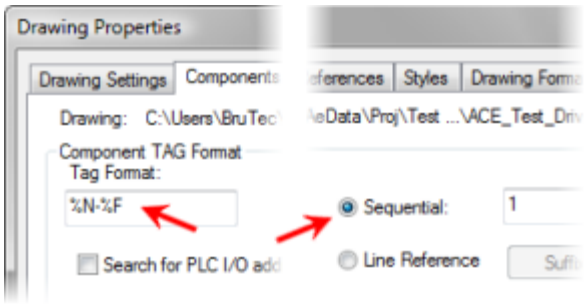


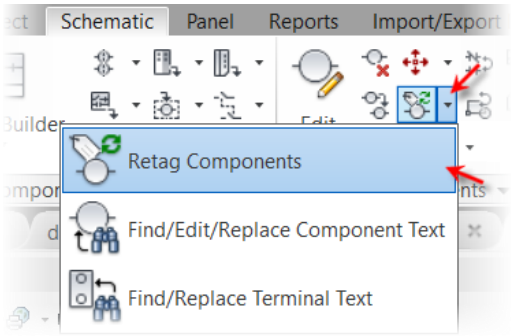
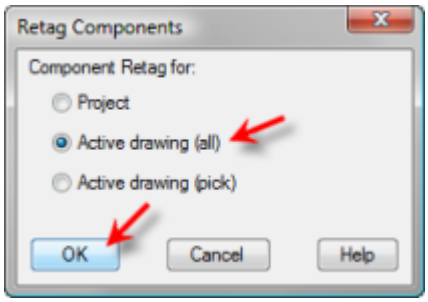
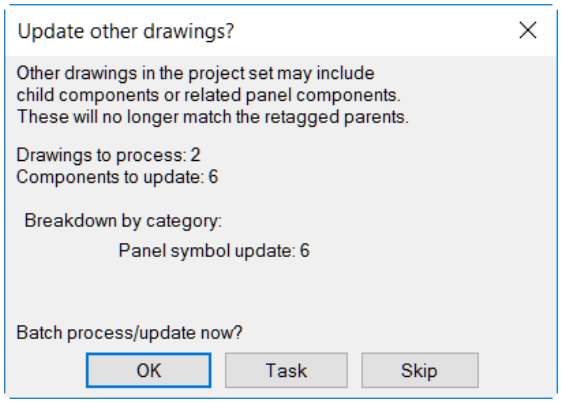
4. On the Edit Wires/Wire Numbers panel click Trim Wire.



5. In the drawing, select anywhere on rung 404 to the right of the new wire you just added.

The selected wire segment is trimmed and unneeded connecting dots are automatically removed.

6. On the Schematic tab, Insert Wires/Wire Numbers panel, click Wire Numbers. **7.** *In the Sheet 4 - Wire Tagging dialog box you have options to number your wires by reference number or sequentially. You can also override the default format and create your own custom wire numbering styles.* Click Drawing-Wide. *Unique wire numbers are added to every wire segment on the drawing. Wire number copies can also be added as desired.*	
8. *At any point you can change the style or format of the wire numbers, or component tags.* On the Schematic tab, Other Tools panel, click Drawing Properties.	
9. In the Drawing Properties dialog box, Components tab, • For Tag Format, enter **%N-%F**. (Be sure to enter exactly as shown, including the dash.) • Select Sequential. • Click OK. *The codes represent parts of the component tag. In this example, %N is the component Number, and %F is the component family, for example PB-Push Button. These two codes are separated by a dash.* *Notice that the component number and family name are reversed with this custom annotation style.* *This example shows how easy it is to modify the annotation style using the Component TAG format in the Drawing Properties dialog box.*	

10. On the Schematic tab, Edit Components panel, Retag Components flyout, Retag Components flyout, click Retag Components.	
11. In the Retag Components dialog box, click Active Drawing (All). Click OK. *Notice that this command allows you to make changes to individual components, an individual drawing, or throughout the entire project automatically.*	
12. *The Update Other Drawings dialog box provides several options.* *OK - Automatically search for and update related components immediately.* *Task - Save the task for later execution.* *Skip - Skip the automatic process and perform the updates manually*. Click OK. **13.** If prompted, in the Qsave dialog box, click Always Qsave. *This completes the exercise.*	

Panel Layouts

Overview

Creating a panel layout can be a pain staking task. Trying to find and match the correct footprints for the parts used, making sure every part in the schematic is accounted for in the panel and vice versa, and so on.

Using AutoCAD Electrical you can extract a list of components from the schematic or from the panel and simply select the components you want to insert. Based on the catalog number, AutoCAD Electrical locates the correct matching component and provides tools to help you locate it in the drawing. This includes placement of entire rows or columns of footprint, or sorting based on Installation and Location codes.

*Exercise-Panel Layout From Schematic List

In this lesson, you will extract a list of schematic components and insert footprints on the panel layout for the two push buttons you previously inserted.

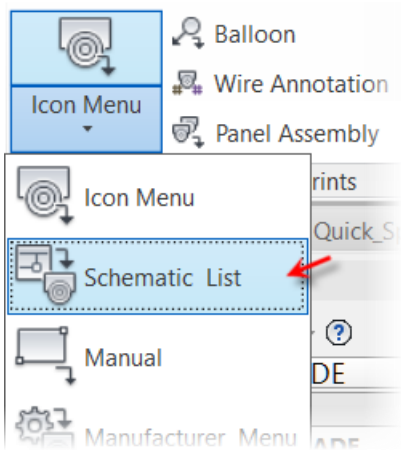
1. In the Project Manager, under Quick_Spin_On_AcadE, expand the Panel sub-folder.

2. Right-click on Quick_Spin_2017_2017_08.dwg. Click Open.

3. On the Panel tab, Insert Component Footprints panel, Icon Menu flyout, click Schematic List.

4. In the Schematic Components List dialog box, under Extract Component List For, select Project. Click OK.

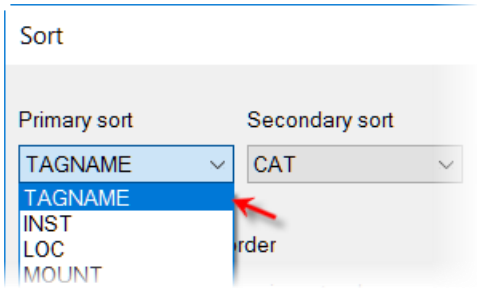
5. In the Select Drawings to Process dialog box, select Do All. Click OK.



6. In the Schematic Components dialog box, click Mark Existing.

7. Click Sort List.

8. In the Sort dialog box, under Primary Sort, select TAGNAME from list. Click OK.

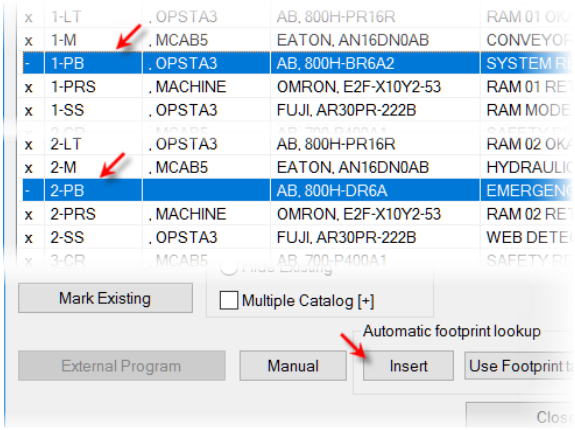


9. In the Schematic Components dialog box, scroll through list to locate and select 1-PB and 2-PB, the two pushbuttons you inserted earlier.

(Hint: Use the CTRL key to select both.)

10. For Rotate, enter **0.**

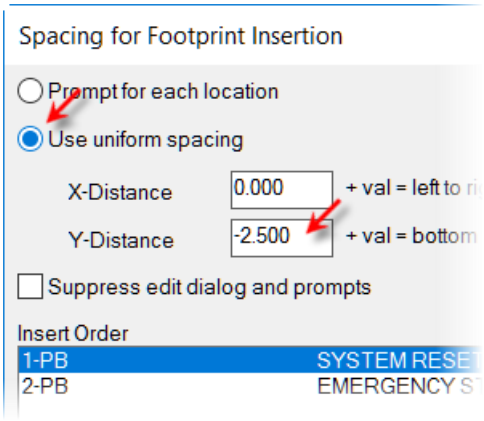
11. Under Automatic Footprint Lookup, click Insert.



12. *Since you selected more than one footprint to insert you are presented options for locating multiple components.*

In the Spacing For Footprint Insertion dialog box, select Uniform Spacing.

13. For Y- Distance, enter **-2.5**. Click OK.



14. In the drawing select an insertion point directly below the Power On light and to the left of the Conveyor Running light.

The footprint for 1-PB is placed and all information is transferred from the schematic component to the footprint.

15. In the Panel Layout – Component Insert/Edit dialog box, click OK.

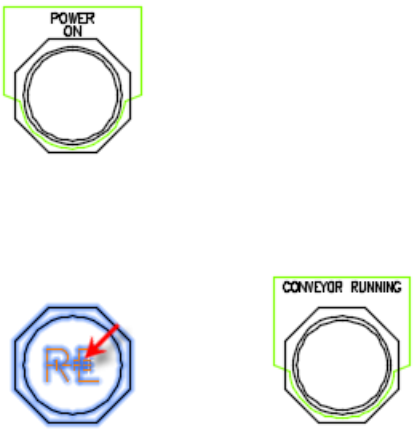
The footprint for 2-PB is automatically placed and all information is transferred.

16. In the Panel Layout – Component Insert/Edit dialog box, click OK.

17. *In the Panel Layout-Component Insert/Edit dialog box both 1-PB and 2-PB are marked as existing.*

Click Close.

This completes the exercise.



Reports

Overview

One of best time saving features of AutoCAD Electrical is its ability to extract data from the drawings and format the data into reports. Pretty much any data added to the drawings can be extracted into one of the available report types.

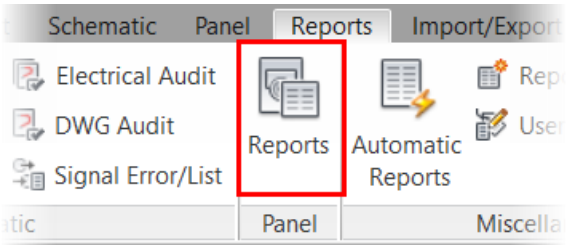
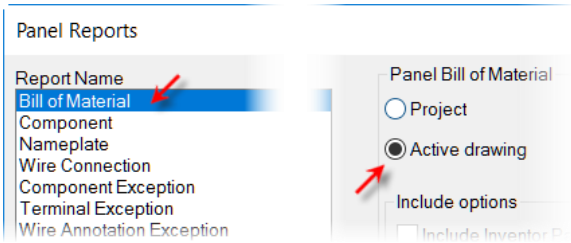
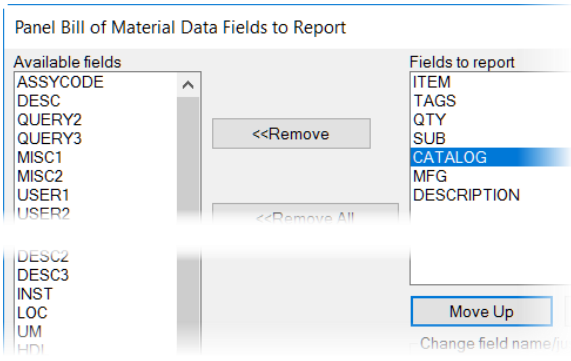
Bill of Material or BOM reports are one of the most commonly used type of reports. Using the Manufacturer and Catalog attribute values, detailed data is extracted from the part catalog database for each component.

A report that is seldom created manually but is easy to create in AutoCAD Electrical is the Wire From/To report. This report supplies of list of every wire in the drawing and includes information about the wire such as the wire type, wire number, the location-component-pin the wire starts at, and the location-component-pin for the end point of the wire.

All reports can be easily formatted to match company standards or desired layout. All reports can either be saved to an external file or placed on the drawing as a table.

*Exercise-Panel BOM Report Saved on Drawing

In this lesson, you will extract a Bill of Material for the active drawing and place it on the drawing as a table entity.

1. On the Reports tab, Panel panel, click Reports.	
2. In the Panel Reports dialog box, under Report Name, select Bill of Material. **3.** Under Panel Bill of Material, select Active Drawing. **4.** Click OK. **5.** If prompted, in the Qsave dialog box, click always Qsave.	
The format of the report is easily changed to meet your requirements. **6.** In the Report Generator dialog box, click Change Report Format. **7.** In the Fields to Report dialog box, under Fields to Report, select Catalog. **8.** Click Move Up. Click OK.	

9. In the Report Generator dialog box, click Put On Drawing.

10. In the Table Generation Setup dialog box, under First New Section Placement, click Pick.

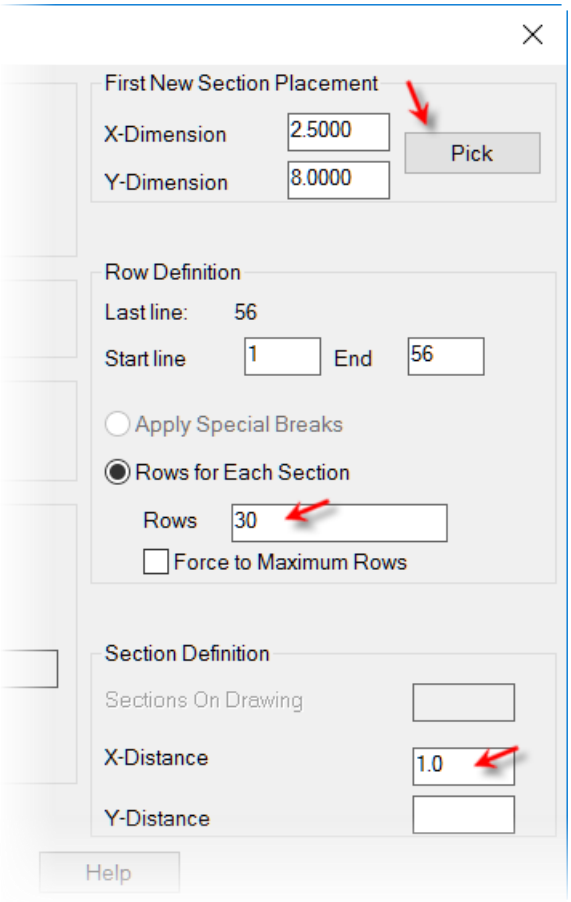
11. In the drawing select an insertion point just below the left edge of the panel at approximately 2.5,8.0.

Under Row Definition, the length of the BOM report is displayed as 56 lines, too many to fit on the drawing properly. You will split the table into two sections.

12. In the Table Generation Setup dialog box, under Row Definition, for Rows, enter **30**.

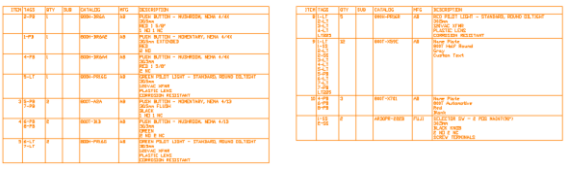
13. Under Section Definition, for X-Distance, enter **1.0**.

14. Click OK.



15. In the Report Generator dialog box, click Close.

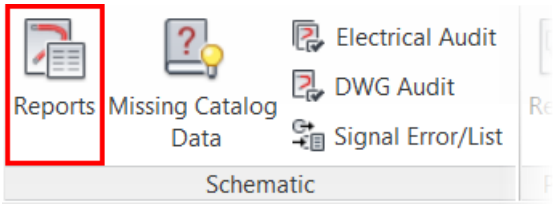
This completes the exercise.



*Exercise-Schematic Wire From/To Report Saved to File

In this lesson, you will extract a Wire From/To report simulating a conduit run between two cabinets and save the report as an Excel file.

1. On the Reports tab, Schematic panel, click Reports.

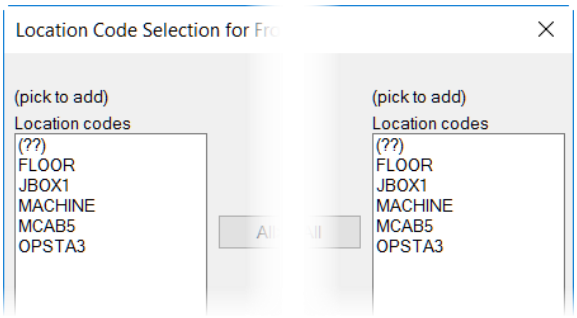


2. In the Schematic Reports dialog box, under Report Name, select From/To.

3. Under From/To select Project. Click OK.

4. In the Select Drawings to Process dialog box, click Do All. Click OK.

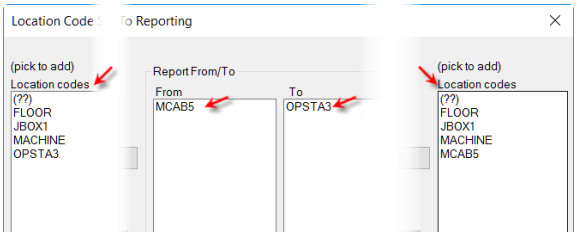
The Location Code Selection dialog box provides the ability to filter by Location codes for wires to include in report.



5. In the Location Code Selection dialog box, under the left side Location Codes, select MCAB5.

6. In the Location Code Selection dialog box, under the left side Location Codes, select OPSTA3.

7. Click OK.



8. In the Report Generator dialog box, click Save to File.

9. In the Save Report to File dialog box, select Excel Spreadsheet Format. Click OK.

A default location and file name for the report are displayed.

10. In the Select File For Report dialog box, click Save.

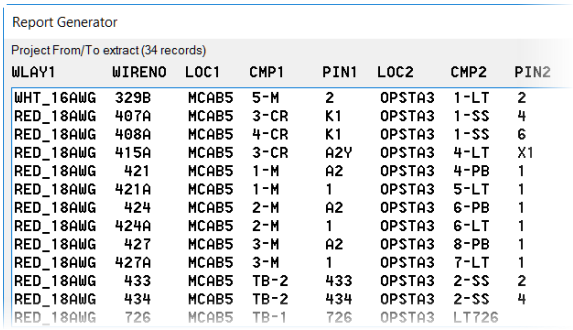
You have the option to create a custom script routine that can upload the file to your MRP system, email the file, or other special tasks.

11. In the Optional Script File dialog box, click Close – No Script.

12. In the Report Generator dialog box, click Close.

13. In the Location Code Selection dialog box, click Cancel.

This completes the exercise.


Report Generator

Project From/To extract (34 records)

WLAY1	WIRENO	LOC1	CMP1	PIN1	LOC2	CMP2	PIN2
WHT_16AWG	329B	MCAB5	5-M	2	OPSTA3	1-LT	2
RED_18AWG	407A	MCAB5	3-CR	K1	OPSTA3	1-SS	4
RED_18AWG	408A	MCAB5	4-CR	K1	OPSTA3	1-SS	6
RED_18AWG	415A	MCAB5	3-CR	A2Y	OPSTA3	4-LT	X1
RED_18AWG	421	MCAB5	1-M	A2	OPSTA3	4-PB	1
RED_18AWG	421A	MCAB5	1-M	1	OPSTA3	5-LT	1
RED_18AWG	424	MCAB5	2-M	A2	OPSTA3	6-PB	1
RED_18AWG	424A	MCAB5	2-M	1	OPSTA3	6-LT	1
RED_18AWG	427	MCAB5	3-M	A2	OPSTA3	8-PB	1
RED_18AWG	427A	MCAB5	3-M	1	OPSTA3	7-LT	1
RED_18AWG	433	MCAB5	TB-2	433	OPSTA3	2-SS	2
RED_18AWG	434	MCAB5	TB-2	434	OPSTA3	2-SS	4
RED_18AWG	726	MCAB5	TB-1	726	OPSTA3	LT726	

What’s New for 2017

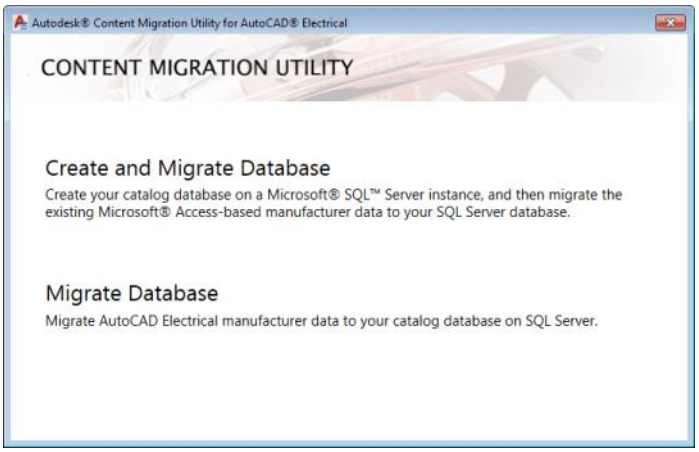
Microsoft® SQL™ Server Support for Catalog Data

Starting with AutoCAD Electrical 2017, AutoCAD Electrical now supports SQL Server for your catalog data. You can create an SQL Server instance and migrate the default Microsoft Access-based catalog and footprint databases to your SQL Server instance.

You can either choose to configure your SQL Server instance when you are installing AutoCAD Electrical 2017 or set up your SQL Server from the product.

Autodesk Content Migration Utility

Content Migration Utility, is a lightweight GUI-based migration tool that allows you to create AutoCAD Electrical catalog database (catalog and footprint databases) on an SQL Server instance. Using the same utility, you can migrate the default catalog data, which is furnished in Microsoft® Access tables to an SQL Server instance.



In the Create and Migrate database workflow, the utility prompts you to connect to an instance of SQL Server. After a successful connection, the utility allows you to name your catalog and footprint target database tables in SQL Server. Once you have created your target databases, use the utility to navigate and select your existing catalog and footprint databases (.mdb) to begin the migration process.

In the Migrate database workflow, the utility prompts you to select the catalog and footprint databases, which exists on the SQL Server. After you have selected, the utility allows you to navigate and select your existing catalog and footprint databases (.mdb) to begin the migration process.

Resequence Item Numbers

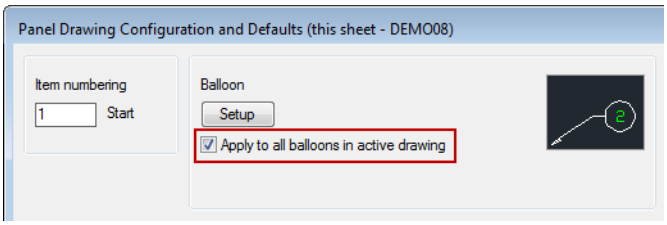
Item number resequencing is updated as follows:

- Drawings are no longer opened for processing, improving the performance.
- The manufacturers are sorted across the project, not per drawing.
- The entire project is always processed, keeping item numbers in sync for components with the same catalog.
- The project property, Reset with each Drawing, has been removed and is no longer valid.

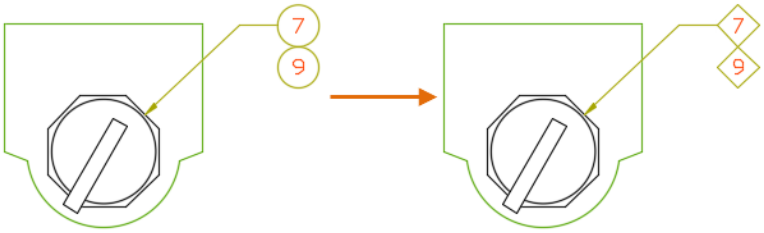
Note: *Use the AEITEMRESEQUENCEMODE system variable to return to the pre-2017 item number resequencing method.*

Item Number Balloon Update

If you change your drawing's balloon setup in the Panel Drawing Configuration, you now can update all the existing balloons.



After you OK, each balloon on the active drawing updates to match the balloon setup.

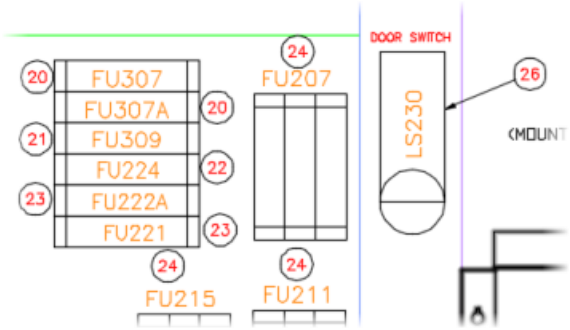


* Exercise- Resequence Item Numbers and Update Balloons

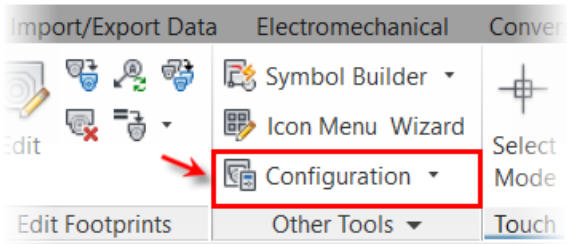
In this lesson, you use the updated Resequence Item Number functionality on your drawing data.

1. In the Project Manager, right-click *Quick_Spin_2017_09.dwg*. Click Open.

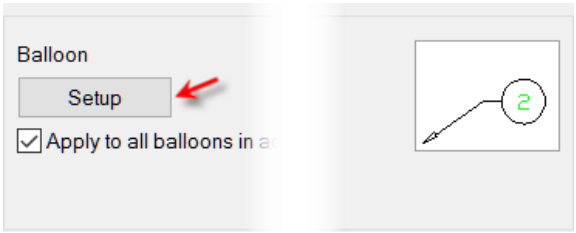
2. Zoom into the fuses in the upper middle of the panel drawing.



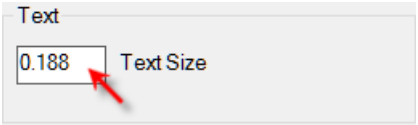
3. On the Panel tab, Other Tools panel, click Configuration.



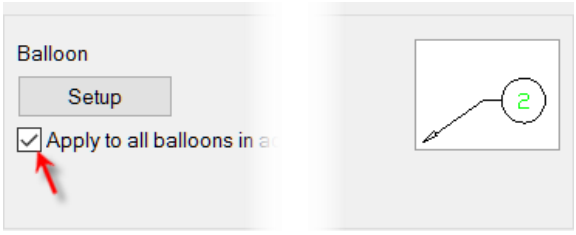
4. In the Panel Drawing Configuration dialog box, under Balloon, click Setup.

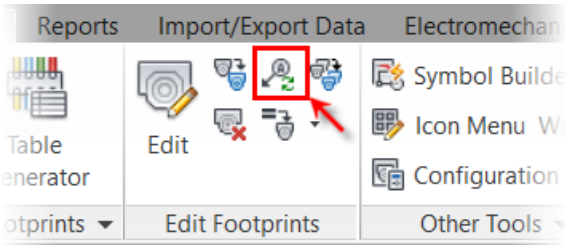
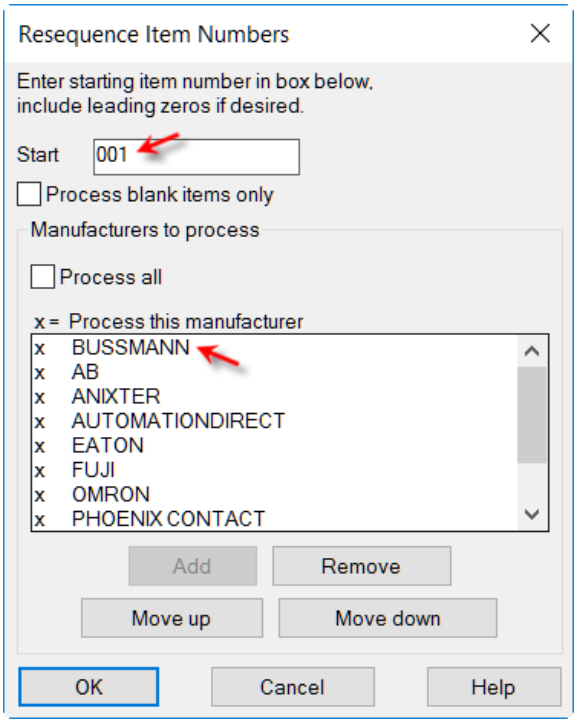
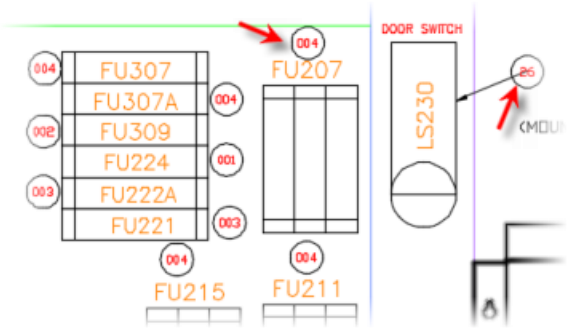


5. In the Panel Balloon Setup dialog box, under Text, enter. **0.188.** Click OK.



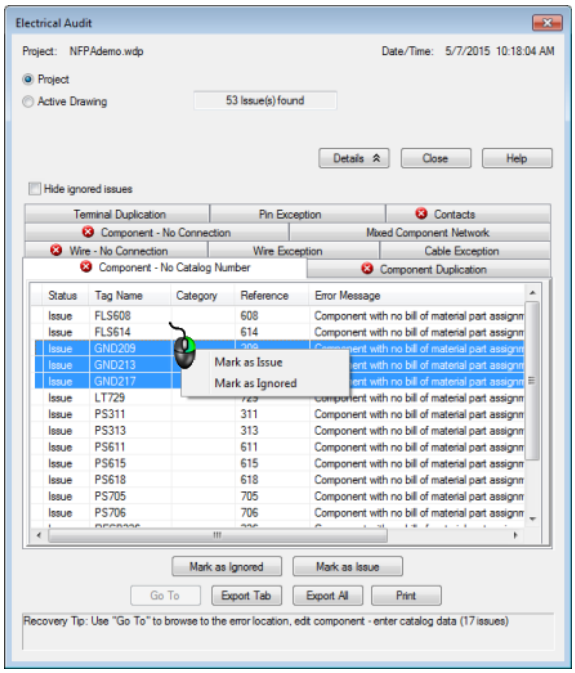
6. In the Panel Drawing Configuration dialog box, under Balloon, select Apply To All Balloons In Active Drawing. Click OK.



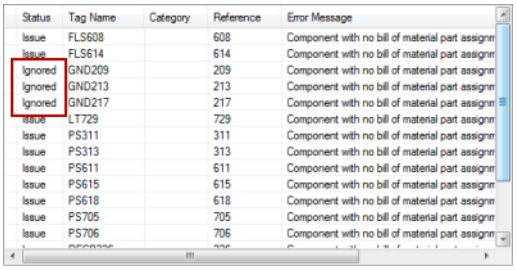
7. On the Panel tab, Edit Footprints panel, click Resequence Item Numbers.	
8. In the Resequence Item Numbers dialog box, for Start, enter **001**. *Manufacturers are listed alphabetically by default, and items numbers will be assigned by manufacturer as listed.* **9.** Under Manuafactures to process, de-select Process All. **10.** From the list, select Bussmann. **11.** Click Move Up as needed to reposition Bussmann at the top of the list. Click OK.	
All item numbers are updated to new format. Notice components without manufacturer and catalog data are not updated. *This completes the exercise.*	

Ignore Errors in Electrical Audit

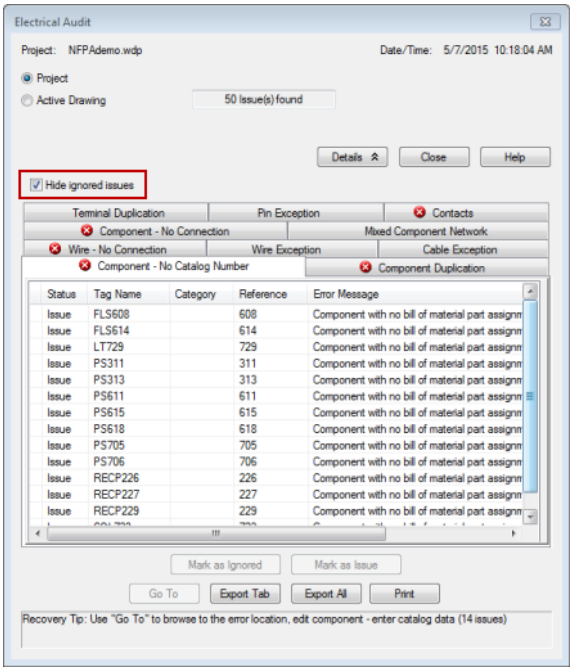
Once you have determined an error is not something that you need to fix, you can set the error's status to Ignored.



Ignored errors are indicated in the Status column.

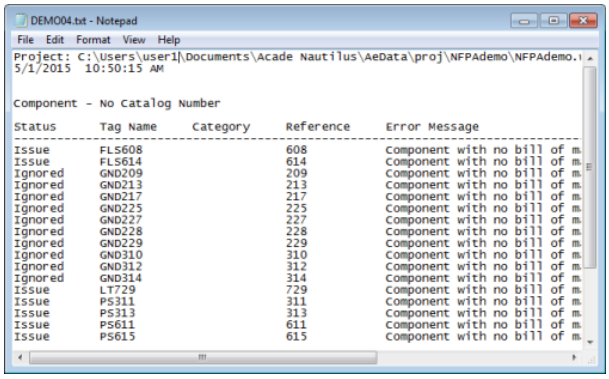


Once you have marked the errors you want to ignore, you can hide those errors.



The error status is remembered each time you run the audit and across sessions of AutoCAD Electrical.

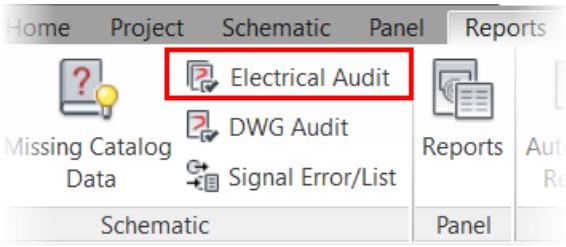
Ignored errors are included in the output text file when you export, even if they are hidden in the dialog. If an error has been marked as ignored, it is indicated in the Status column in the text file.



* Exercise-Ignore Errors in Electrical Audit

In this lesson, you mark erros in the Electrical Audit to ignore and hide the ignored errors in the display list.

1. On the Reports tab, Schematic panel, click Electrical Audit.



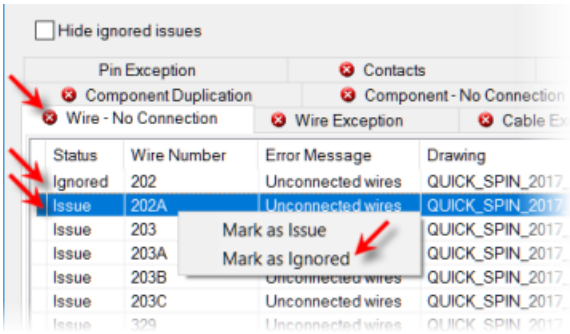
2. In the Electrical Audit dialog box, click Details.

3. On the Wire-No Connection tab, right-click Wire Number 202.

4. Click Mark As Ignored.

Notice you can also revert the status, marking ignored items as issues.

5. Repeat Steps 2-4 for wire number 202A.

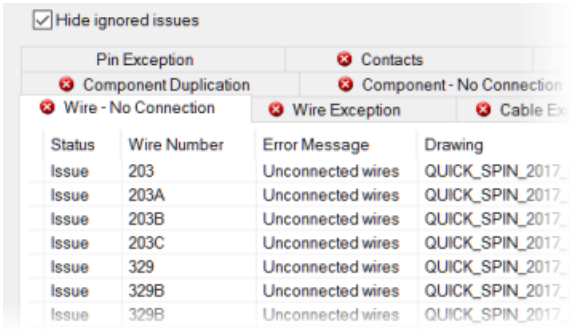


6. Click Hide Ignored Issues.

Ignored items are hidden from list. Deselecting the checkbox will unhide them.

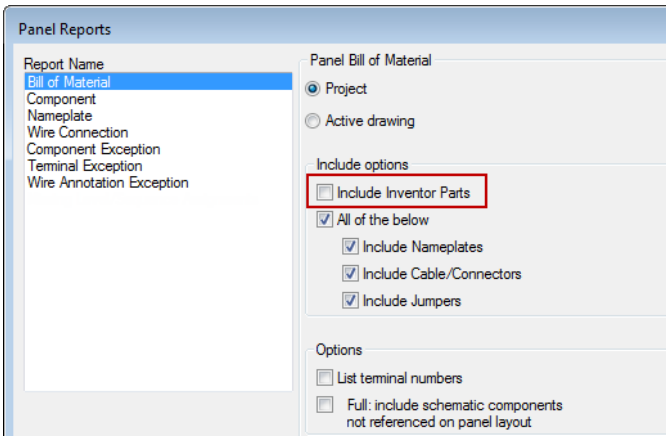
7. Click Close.

This completes the exercise.



AutoCAD Electrical and Inventor Combined BOM

If your project is linked to an Inventor assembly as part of an electromechanical project, you can generate a combined Bill of Material report. An option to include the Inventor parts is now available when you generate either a Schematic or Panel Bill of Material report.



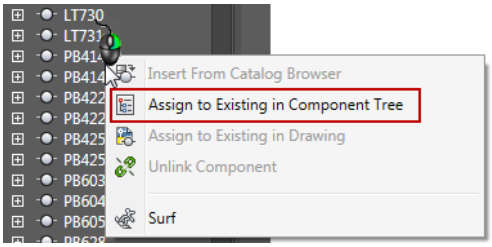
With this option, any electrical parts on your Inventor assembly are included in the report. The Source column in the report indicates whether the component is from AutoCAD Electrical, Inventor, or linked and represented in both.

The option to Include Inventor Parts is also added to the Report Format File Setup for both the Panel and Schematic Bill of Material reports.

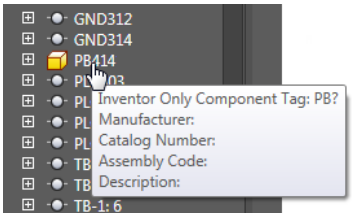
Link Components in Location View tab

When you work in an electromechanical project, you can now link the AutoCAD Electrical component to the Inventor part right on the Location View tab of Project Manager.

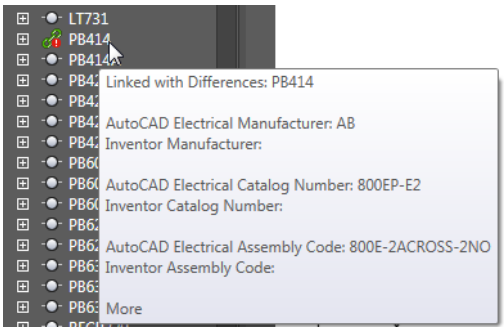
On the Location View tab, right-click either the AutoCAD Electrical component or Inventor part in the tree and click Assign to Existing in Component Tree.



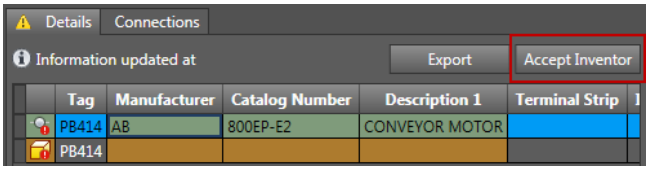
Find and select the other representation in the tree. In this example, since I started with the AutoCAD Electrical component, I select the Inventor pushbutton.



The components are now linked, but there are some differences.



You can resolve these differences on the Details pane in the Location View tab.



* Exercise – Electro-Mechanical Interface Updates

In this lesson, you explore several updates to the electro-mechanical interface between AutoCAD and Electrical and Inventor. The exercise data set includes and is already linked to an EMX file containing data from an Inventor assembly.

You will create a combined Inventor-AutoCAD Electrical Bill of Material and Link components using the Component tree on the Location View tab.

(See the 2015 class "Like Chocolate and Peanut Butter: AutoCAD Electrical and Inventor Are Better Together!" for more details on the interface.)

1. In the Project Manager, expand the Electro-Mechanical sub-folder.

2. Right-click *Quick_Spin_2017_10.dwg*. Click Open.

3. On the Electromechanical tab, Setup panel, click Electromechanical Link Setup.

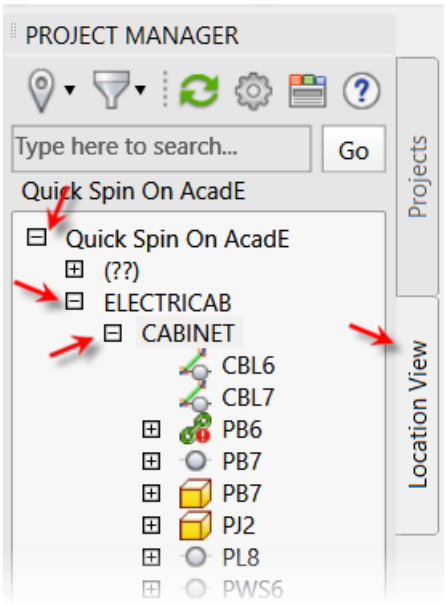
4. In the Electromechanical Link Setup dialog box, click Link.

5. Browse to the dataset directory and select Quick_Spin_On_AcadE.emx. Click Open.

6. In the Electromechanical Link Setup dialog box, click Close.

7. In the Project Manager, click the Location View tab.

8. On the Location View tab, expand Quick Spin On AcadE, ElectriCab, and Cabinet.

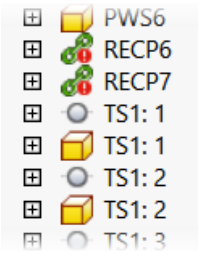


Notice the existence of:

Linked ( *),*

Logical ( *Electrical), and*

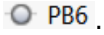
Physical ( *Mechanical) components.*



Next you will link several physical and logical components together using the Location View tab.

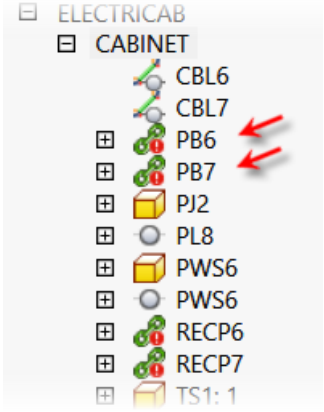
9. On the Location View tab, Component tree, right-click on the physical icon for PB6  .

10. From the menu, click Assign to Existing in Component Tree.

11. In the Component tree, select the logical icon for PB6  .

The two components are linked together.

12. Repeat Steps 9-11 for PB7.



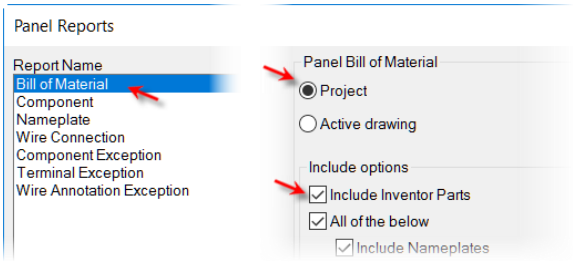
13. On the Reports tab, Panel panel, click Reports.

14. In the Panel Reports dialog box, under Report Name, select Bill of Material.

15. Under Panel Bill of Material, select Project.

16. Under Include Options, select Include Inventor Parts.

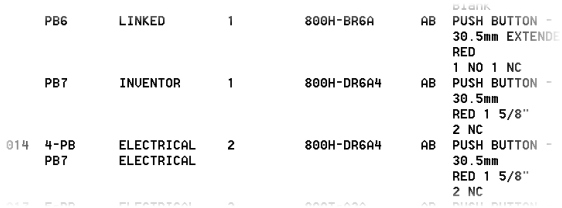
17. Click OK.



Bill of Material report is created listing the source of Linked (both), Inventor only, or Electrical only.

18. In the Report Generator dialog box, click Close.

This completes the exercise.



References

Download Trial Software

You can download a trial of the AutoCAD Electrical software. It is a fully functioning version and will run on your system for 30 days. Remember, this is risk free. The drawings you create are fully AutoCAD compatible, the electrical intelligence is carried in AutoCAD standard blocks and attributes.

Download the software at:
http://usa.autodesk.com/autocad-electrical/

AutoCAD Electrical Forum

http://forums.autodesk.com/t5/AutoCAD-Electrical/bd-p/38

AutoCAD Electrical Knowledge Base

http://usa.autodesk.com/adsk/servlet/ps?siteID=123112&id=3570520&linkID=9240737

AutoCAD Electrical Website and free trial

http://usa.autodesk.com/autocad-electrical/