AutoCad Electrical 2019 Tutorials (here)

Table of Contents

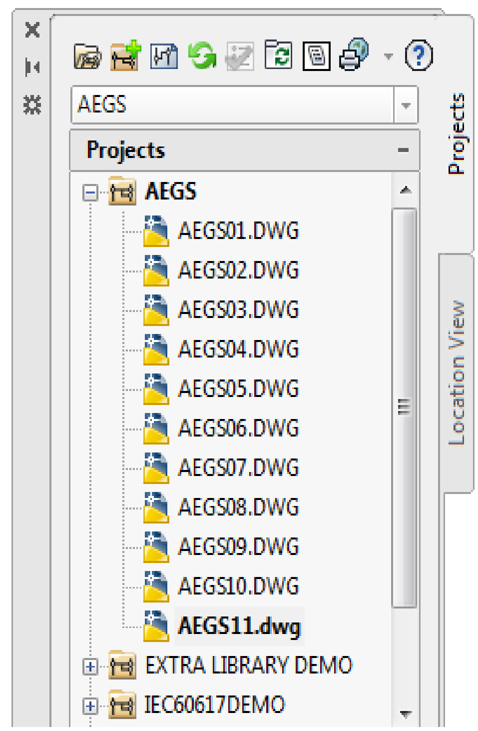
Projects

Create a project and add drawings with Project Manager.

Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Projects to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Create a project
- Set project properties
- Create a drawing
- Add drawings to a project
- View drawings in a project



Workflow Topic

1. To Work with Projects
   Create a project and modify project properties.
2. To Work with Drawings
   Create a drawing, add drawings to the project, and view drawings in Project Manager.

To Work with Projects

Create a project and modify project properties.
AutoCAD Electrical toolset is a project-based system. An ASCII text with a **.wdp** extension defines each project.

This **Project File** contains:

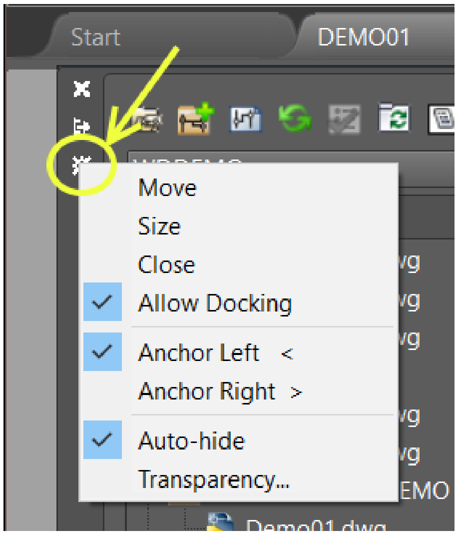
- a list of project information,
- default project settings,
- drawing properties, and
- drawing file names.

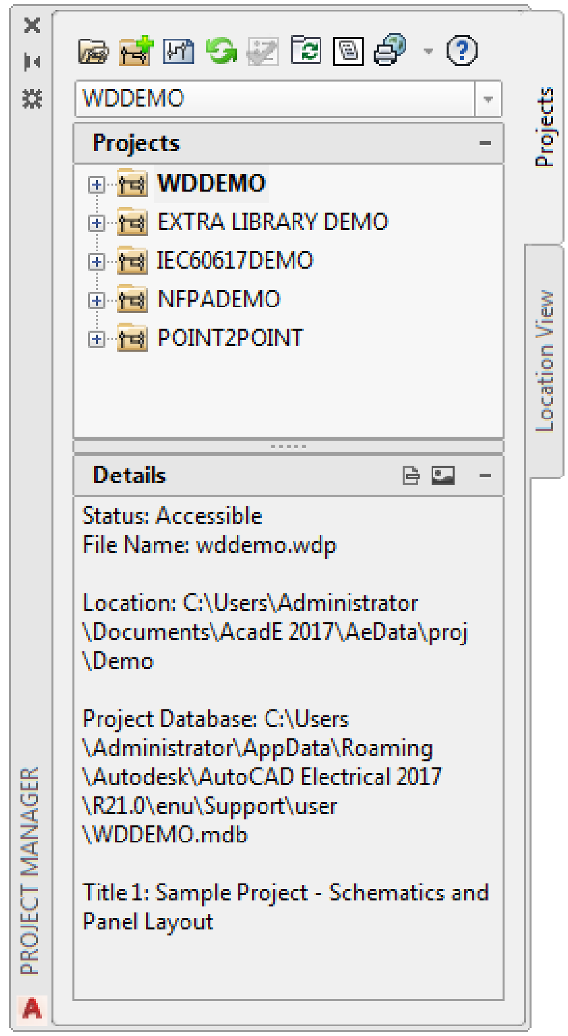
You can have an unlimited number of projects; however, only one project can be active at a time.

Use the **Project Manager** to:

- add new drawings,
- reorder drawing files,
- organize drawings in subfolders, and
- change project settings.

Right-click the **properties** icon to display options to move, size, close, dock, hide, or set the transparency for the Project Manager.

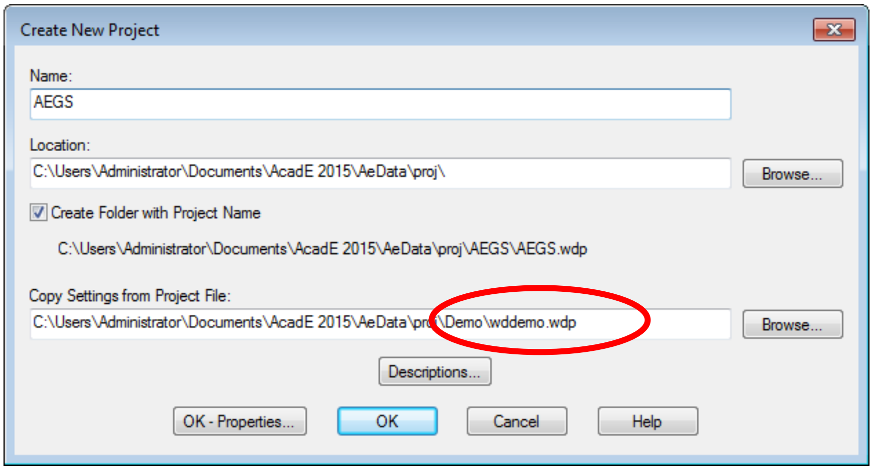




Create an AutoCAD Electrical toolset project

1. Click Project tab ➤ Project Tools panel ➤ Manager.
2. In the Project Manager, click the New Project tool. 
3. In the Create New Project dialog box, specify:
   Name: AEGS

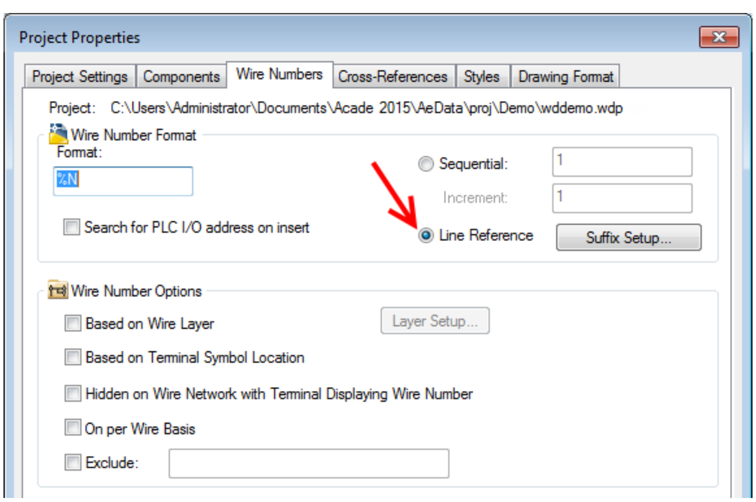
   A name must be entered to define any of the project properties. The .wdp extension is not required in the edit box.
4. Make sure ***wddemo.wdp*** is specified in the Copy Settings From Project File edit box.
5. Click OK-Properties.



Your new project is added to the current projects list and automatically becomes the active project. The Project Properties dialog box displays, where you can modify your project default settings. All information defined on these tabs are saved to the **project definition file** as project defaults and settings.

Set Project Properties

1. In the Project Properties dialog box, click the Components tab.
2. In the Component Tag Format section, verify that Line Reference is selected. This selection creates unique reference-based tags when multiple components of the same family are located at the same reference location. When **reference-based tagging** is used, a suffix variable is required to keep components of the same family type unique. For example, three push buttons on line reference 101 could be labeled PB101, PB101A, and PB101B. Click Suffix Setup to change the suffix variable.
3. Click the **Wire Numbers** tab.
4. In the Wire Number Format section, verify that Line Reference is selected. This selection creates **unique reference-based wire number tags** for multiple wire networks beginning at the same reference location. When reference-based numbering is used, a suffix variable is required to keep wires on the same reference line or in the same reference zone unique. Click **Suffix Setup** to change the suffix variable.
5. Review the various options on the different tabs of the Project Properties dialog box.
   Note: In the Project Properties dialog box, icons indicate whether the settings apply to project settings or drawing defaults.
6. Click OK.



To Work with Drawings

Create a drawing, add drawings to the project, and view drawings in Project Manager.

- A single project file can have drawings located in many different directories.
- There is no limit to the number of drawings in a project.
- You can add drawings to your project at any time.
- When you create a drawing, using New Drawing tool, it is automatically added to the active project.
- Many of the **drawing settings** used by AutoCAD Electrical toolset are stored in a **smart block** on the drawing named **WD_M.dwg**.
- Each AutoCAD Electrical toolset drawing should contain only one copy of the WD_M block. If multiple WD_M blocks are present, the settings cannot be stored and read consistently.

Create a drawing

1. In the Project Manager, click the New Drawing tool. 
2. In the **Create New Drawing** dialog box, specify:

Name: AEGS11	**Description 1**: Bill of Materials Report

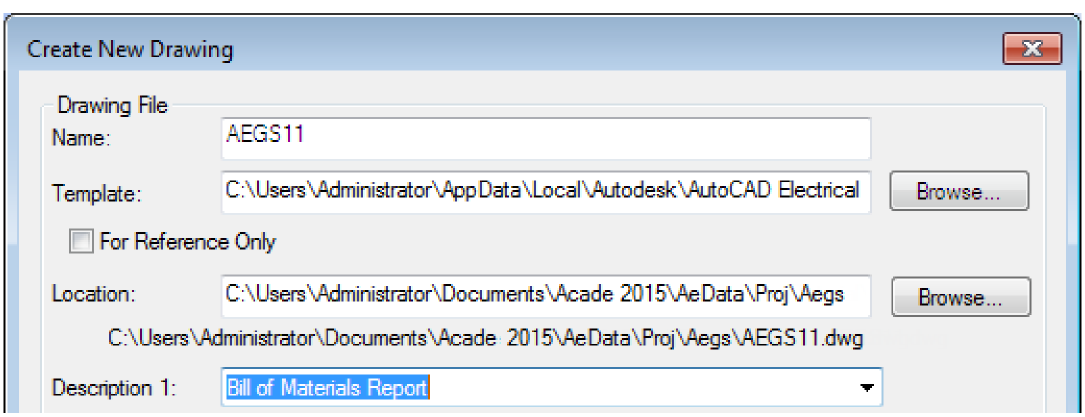
3. Click **Browse** next to the Template edit box.

   A set of templates (*.dwt files) installed with AutoCAD Electrical toolset contain settings for various kinds of drawings, such as acad.dwt and ACAD_ELECTRICAL.dwt.

   You can create your own templates, or use any drawing as a template. You can save a drawing at any stage of completion as a template file. When you use a drawing as a template, the settings in that drawing are used in the new drawing. The changes you make to a drawing that is based on a template do not affect the template file.

   AutoCAD Electrical toolset fully supports the use of AutoCAD template files. **To make an AutoCAD drawing compatible with AutoCAD Electrical toolset, select an AutoCAD Electrical toolset command to modify the drawing**.

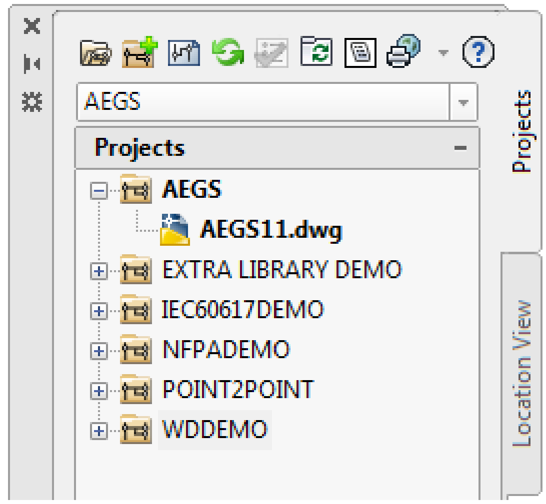
4. In the Select template dialog box, select ACAD_ELECTRICAL.dwt, and click Open.



5. In the Create New Drawing dialog box, click OK.

> **Note**: You could click OK-Properties to display the **Drawing Properties** dialog box. This dialog box has options like the options found in the Project Properties dialog box. It defines drawing-specific settings that are maintained inside the **WD_M block** of the drawing.

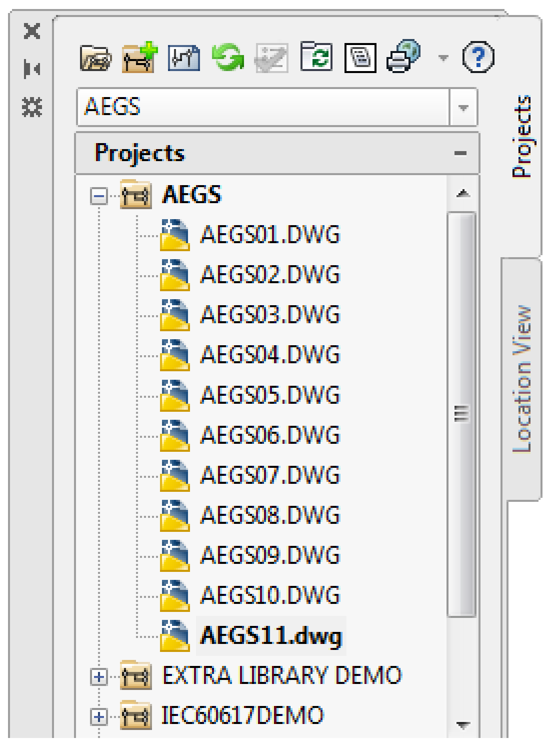
6. In the Project Manager, double-click the project name (AEGS) to display the drawing files. AEGS11 is the only file in the list.



Add drawings to the project

1. In the Project Manager, right-click AEGS, and select Add Drawings.
2. In the Select Files to Add dialog box, select drawings AEGS01.dwg to AEGS10.dwg and click Add.
3. When asked whether to apply the project default values to the drawing settings, click Yes.

   The Project Manager lists the files under the AEGS folder. New drawings that you add from this point on are added at the end of the drawing order. You now have access to the files required for the exercises in this book.



   Note: Two projects can reference the same drawing file. However, if both projects try to modify the same drawing with a project-wide tagging or cross-referencing function, it can lead to **conflicts**.

   The drawing order in the Project Manager determines how AutoCAD Electrical toolset processes the drawings during project-wide operations such as resequencing and wire numbering.

4. Drag the drawings to put them in order, with AEGS11 at the end of the list.

   Note: The active drawing displays in bold text in the project drawing list.

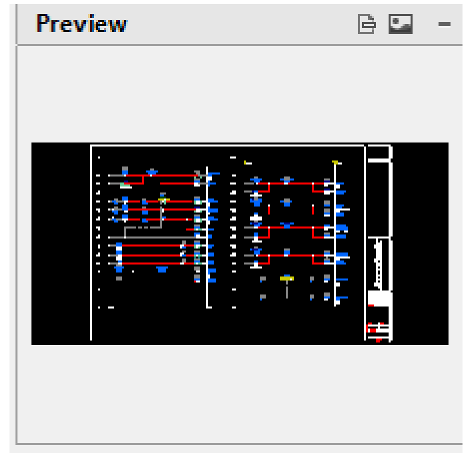
Add the description of a drawing you add

1. In the Project Manager, right-click AEGS10.dwg, and select Properties ➤ Drawing Properties.
2. In the Drawing Properties ➤ Drawing Settings dialog box, Drawing File section, specify:

   Description 1: Connector Drawing

3. Click OK.
4. In the Project Manager, select AEGS10.dwg.
5. In the Project Manager, Details section, review the drawing descriptions.

View drawings in a project



1. In the Project Manager, select AEGS04.dwg.
2. In the Project Manager, Details section, click **Preview**.
3. Continue to click the drawing name you want to preview or use the up and down arrow keys to scroll through the drawing files.
4. When you finish viewing the drawings, click **Details** to return to the drawing details view.

Title Block Setup

Create a title block and use a WDT file or the WD_TB attribute to map AutoCAD Electrical toolset project and drawing values to attributes on your title block.

Title Block Utility

AutoCAD Electrical toolset can link project description lines and some of the drawing properties to attributes on the drawing title block.

The title block utility:

- Automates project-wide title block updates.
- Supports multiple title blocks per drawing.
- Maps AutoCAD Electrical toolset project description lines to specific attributes.
- Maps AutoCAD Electrical toolset per-drawing values to specific attributes.
- Maps AutoLISP values, system variables, or environment variables to specific attributes.

AutoCAD Electrical toolset uses two methods to map the AutoCAD Electrical toolset values to attributes on the title block:

- **WD_TB attribute method** - **mapping information embedded on the title block**. This option is self-contained in the drawing and requires no external file. It is limited to the number of characters that can be placed on a single attribute.
- **WDT file method** - **external attribute mapping file**. This option can update the attributes on existing title blocks, even if the title block does not contain the WD_TB attribute.

Create a Title Block

Create a title block drawing and add the attribute definitions.

The title block is a border drawing inserted as an AutoCAD block on another drawing. The title block border drawing can be inserted as a block on an AutoCAD drawing template file. If your drawing title block consists of an AutoCAD block with attributes, AutoCAD Electrical toolset can link to it.

1. Start a blank new drawing and draw your border using standard AutoCAD commands and objects.

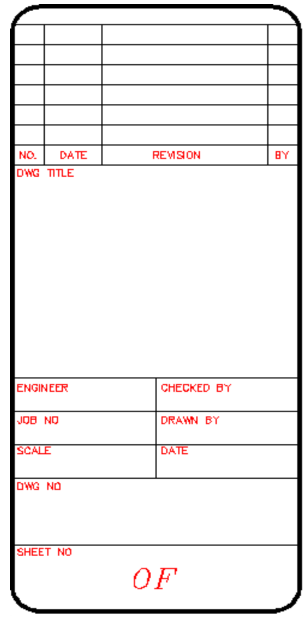
Or

1. Open ACADE_TITLE_BORDER.DWG in Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs.

   This drawing contains a sample border without any of the attribute definition objects.

2. Zoom in for attribute definition placement.

3. Enter ATTDEF at the command prompt to insert attribute definition objects.
   Note: When the border drawing is inserted as a block on another drawing, attribute definition objects become attributes.

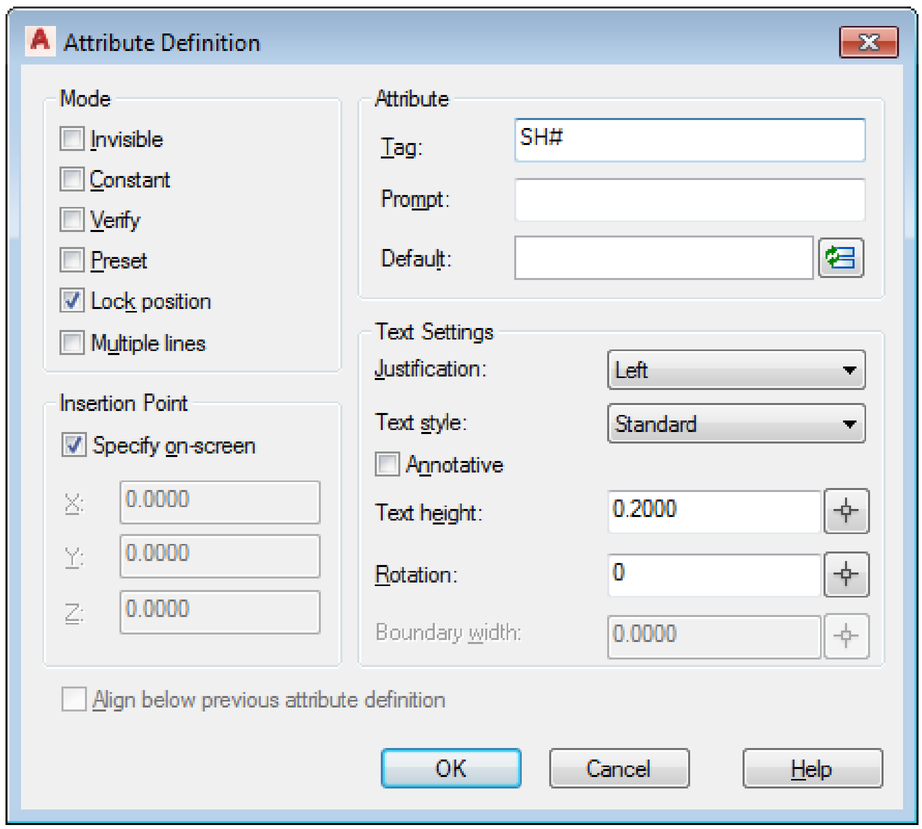


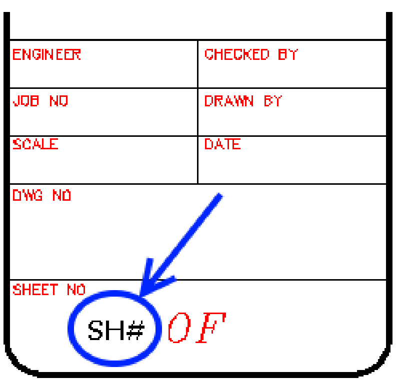
4. Enter the Tag name SH#.

5. Set any other attribute definition properties and values, such as text style, height, and justification.

6. Select OK.

7. Specify the insertion point.





8. Repeat for each attribute definition for the title block as shown.

9. Enter SAVEAS at the command prompt.

10. Enter **File name:** acade_title.

11. Select **Files of type:** AutoCAD Drawing (*.dwg)

12. Click Save.

Title Block Setup

Create the **WDT file** or define the **WD_TB** attribute which maps the project and drawing values to the title block attributes.

There are two different methods that can be used to map the project and drawing values to the title block attributes.

- **WDT file method** - an external text file maps the project and drawing values to attributes on the title block. Use this method if you have many attributes to map or will change it frequently.
- **WD_TB** attribute method - a WD_TB attribute must be present on the title block. The WD_TB attribute value maps the project and drawing values to other attributes on the title block. Use this method if you want the mapping embedded right in the drawings.

WDT File Method

A **text file** defines which AutoCAD Electrical toolset values are mapped to the drawing title block attributes. Use the Title Block Setup utility to create or modify the WDT mapping file.

1. Click Project tab ➤ Project Tools panel ➤ Manager.
2. If AEGS is not the active project, activate the AEGS project.

    If AEGS is in the list of open projects:

    - Select AEGS and right-click.
    - Click Activate.

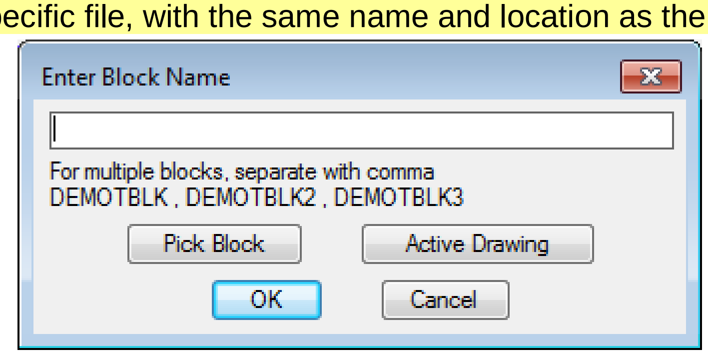
    If AEGS is not in the list of open projects:

    - Select the project list drop-down.
    - Click Open Project.
    - On the Select Project File dialog box, navigate to and select the AEGS.WDP file.
    - Click Open.

3. Open the title block base drawing created previously, ACADE_TITLE.DWG, that contains the attribute definition objects.
    Note: Title Block Setup, WDT file method, can also be used on a drawing with the title block inserted as a block.
4. Click Project tab ➤ Other Tools panel ➤ Title Block Setup.
5. Select the title block link method: **Method 1: <Project> .WDT file**.

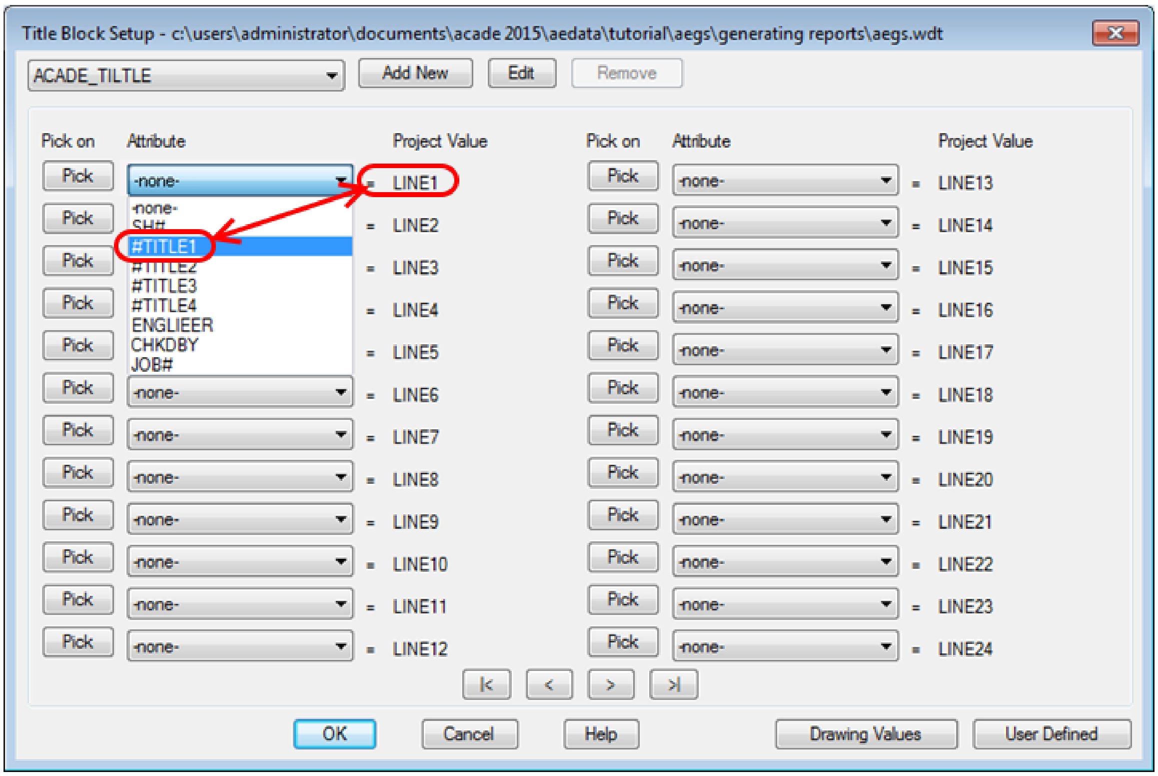
    A project-specific file, with the same name and location as the active project and a WDT extension, defines the attribute mapping.

6. Click OK.



7. Click Active Drawing.
   Note: If running Title Block Setup on a drawing with ACADE_TITLE inserted as a block, select Pick Block and select on the block.
8. Click OK.

Title Block Setup reads the attribute definitions. The Title Block Setup dialog box displays. Each drop-down list contains all the attribute definition objects found on the drawing.



Note: If no attribute definition objects are found on the drawing, an alert displays.

9. In the Title Block Setup dialog box, select the attribute from each list to map to its corresponding project description line.

TITLE#1 ➤ LINE1 TITLE#2 ➤ LINE2 JOB# ➤ LINE4 DRAWNBY ➤ LINE6

10. Click **Drawing Values** to assign drawing specific values.
11. Select the attribute from each list to map to its corresponding drawing value.

DWG# ➤ Drawing (%D value) SH# ➤ Sheet (%S value) SHTS ➤ Sheet Maximum
TITLE#3 ➤ Drawing Desc. 1 TITLE#4 ➤ Drawing Desc. 2

12. Click OK.
13. Title Block Setup creates AEGS.WDT with the selected mappings.

WD_TB Attribute Method

An invisible attribute on a title block of the drawing, named "WD_TB," is encoded with the mapping information. This method eliminates the need for an external mapping text file.

1. Click Project tab ➤ Project Tools panel ➤ Manager.
2. If AEGS is not the active project, activate the AEGS project.

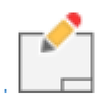
   If AEGS is in the list of open projects:

   - Select AEGS and right-click.
   - Click Activate.

   If AEGS is not in the list of open projects:

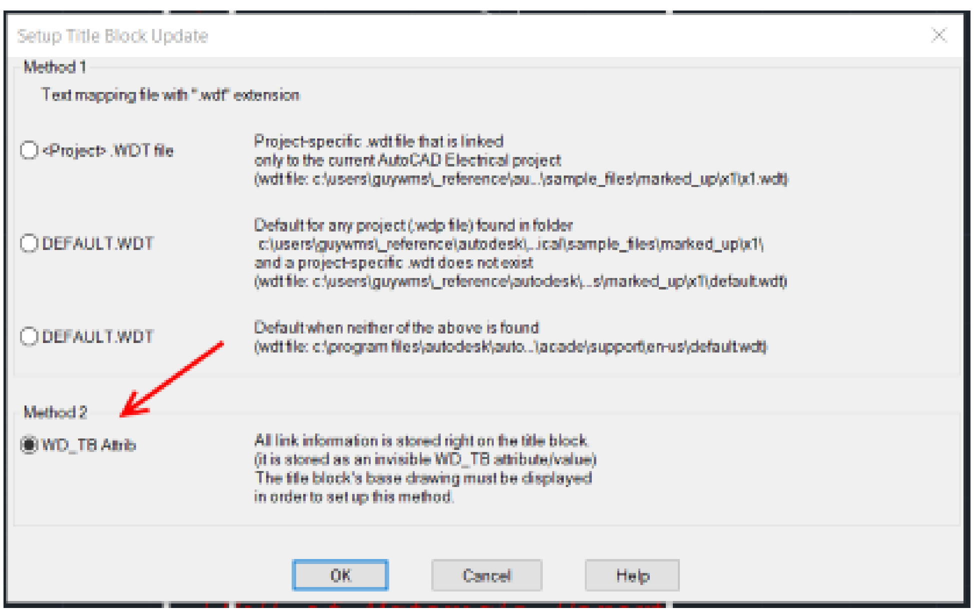
   - Select the project list drop-down.
   - Click Open Project.
   - On the Select Project File dialog box, navigate to and select the AEGS.WDP file.
   - Click Open.

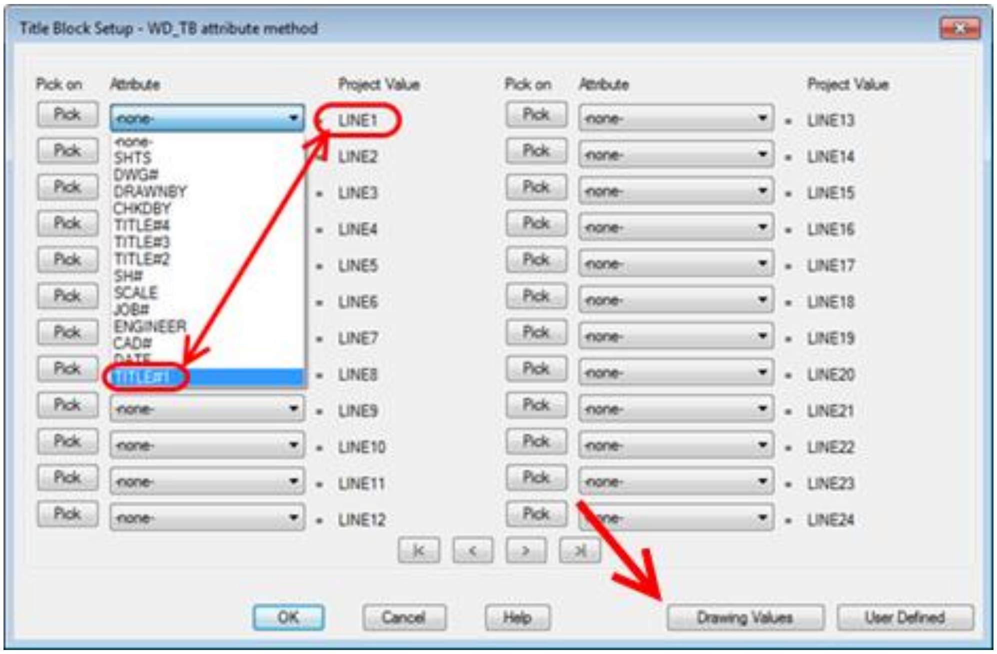
3. Open the title block base drawing created previously, ACADE_TITLE.DWG, that contains the attribute definition objects.
4. Click Project tab ➤ Other Tools panel ➤ Title Block Setup.



5. Select the title block link method: **Method 2: WD_TB attrib**.
6. Click OK.

Title Block Setup reads the attribute definitions. The Title Block Setup dialog box displays. Each drop-down list contains all the attribute definition objects found on the drawing. (next page)





Note: If no attribute definition objects are found on the drawing, an alert displays.

7. In the Title Block Setup dialog box, select the attribute from each list to map to its corresponding project description line.

TITLE#1 ➤ LINE1 TITLE#2 ➤ LINE2 JOB# ➤ LINE4 DRAWNBY ➤ LINE6

8. Click **Drawing Values** to assign drawing specific values.
9. Select the attribute from each list to map to its corresponding drawing value.

DWG# ➤ Drawing (%D value) SH# ➤ Sheet (%S value) SHTS ➤ Sheet Maximum
TITLE#3 ➤ Drawing Desc. 1 TITLE#4 ➤ Drawing Desc. 2

10. Click OK.

    Title Block Setup updates the WD_TB attribute definition with the selected mappings. If a WD_TB attribute definition does not exist, Title Block Setup inserts it at 0,0.

11. Save the drawing.

Create a Drawing Template

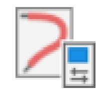
Add the title block, set drawing properties, and **define wire layers** while creating a drawing template.

When a drawing template file is used to start a new drawing it can:

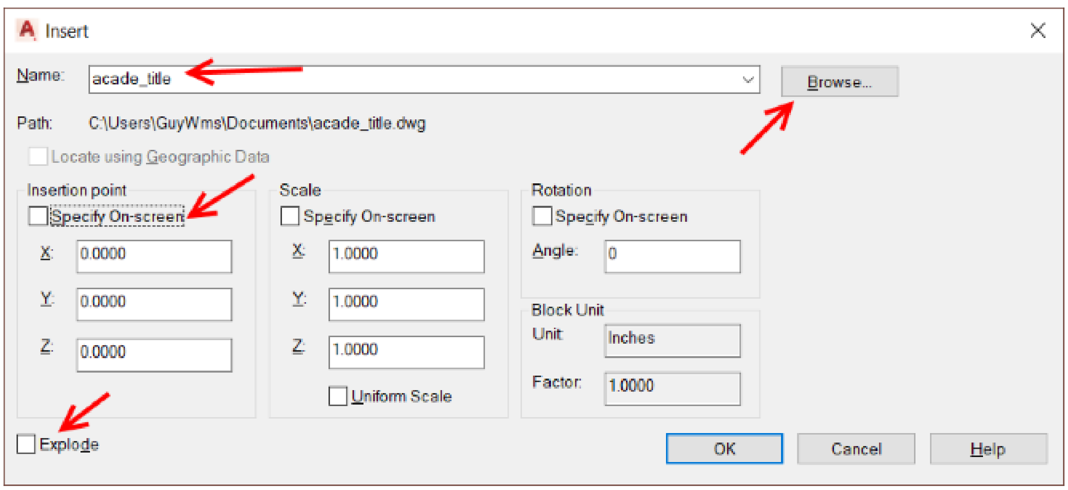
- Predefine AutoCAD Electrical toolset drawing properties such as component tagging, wire numbering format, and so on.
- Predefine layers and layer properties.
- Predefine wire layers.
- Provide your drawing border and title block.

By default, **drawing template files** are stored in the template folder.

1. Enter **QNEW** at the command prompt to start a new drawing.
2. Select the **acad.dwt** template.
3. Click Open.
4. Enter **INSERT** at the command prompt.
5. Click Browse.
6. Navigate to and select the title block ACADE_TITLE.DWG created for the border.
7. Click Open.
8. On the Insert dialog box, make sure the **Explode** option is not checked.
9. Specify the **insertion point** at 0,0,0.
10. Click OK.
11. If prompted for attribute values, leave them blank.

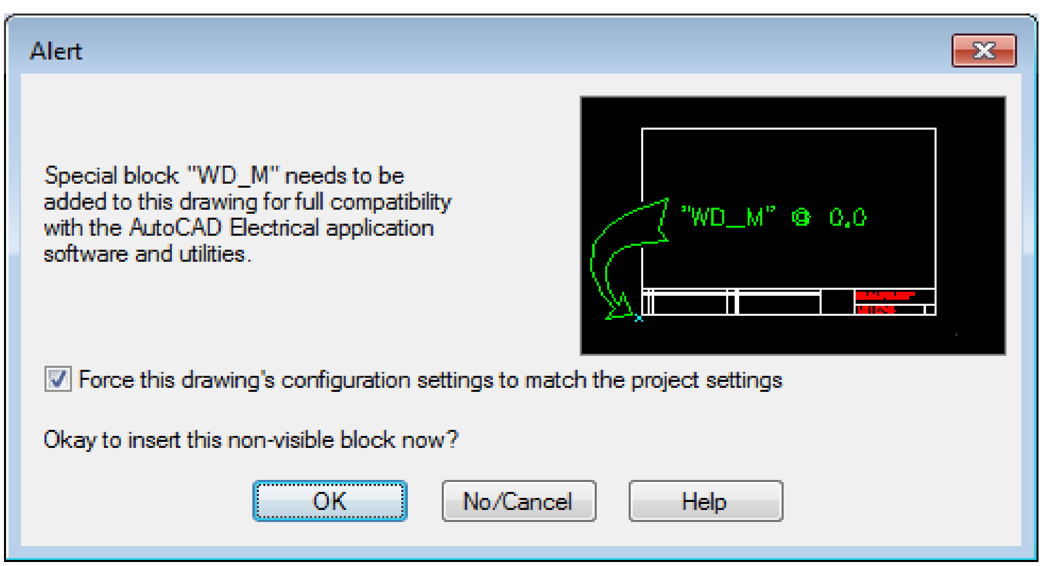


Note: Attributes are invisible if no default values are assigned.



12. Click Schematic tab ➤ Other Tools panel ➤ Drawing Properties.

The alert displays.



13. Click OK to insert the WD_M block.

14. Set the default drawing properties such as component tagging, wire numbering, cross-referencing, and so on.
    Note: No specific changes are needed for this tutorial.
15. Click OK.

16. Click Schematic tab ➤ **Edit Wires/Wire Numbers** panel ➤ **Create/Edit Wire Type**.
17. Add wire layers as needed. Set the properties, color, linetype, and lineweight for **each layer**. For example:

    - In the Create/Edit Wire Type dialog box, click inside the **Wire Color** column for a blank row and enter **RED** for a new wire layer.
    - Click inside the **Size** column and enter **12** for the size.

      The **Layer Name** RED_12 is automatically created.

    - Click Color.
    - Select Red and click OK.
    - Click OK.

    The layer is created and defined as a wire layer.

18. Enter SAVEAS at the command prompt.
19. Set the file type as **AutoCAD Drawing Template** (*.dwt).
20. Enter the file name, AEGS_ELECTRICAL.
21. Click Save.

The Template Options dialog box displays.

22. Select OK.
23. Close the drawing, AEGS_ELECTRICAL.DWT.

Use the Template

Create a drawing using the template containing the title block.

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, click the **New Drawing** tool. 
3. In the Create New Drawing dialog box, specify:

Name: AEGS11 **Description 1:** Title Block **Description 2:** Exercise

4. Click Browse next to the Template edit box.
5. In the **Select Template** dialog box, select AEGS_ELECTRICAL.dwt, and click **Open**.
6. In the **Create New Drawing** dialog box, click OK.
7. On the **Apply Project Defaults to Drawing Settings** dialog box, click **No**.

   Project Manager creates the drawing using the template containing the title block.

Project Description Lines

Add project description values which can be mapped to title block attributes.

1. Click Project tab ➤ Project Tools panel ➤ Manager.
2. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
3. In the Project Manager, right-click the project name, and select **Descriptions**. (a drawing must be open)
4. In the Project Description dialog box, enter values:

   - **Line 1:** Tutorial Project
   - **Line 2:** AutoCAD Electrical toolset
   - **Line 4:** Job #01000
   - **Line 6:** {your name}

5. Click OK. [these are placed in the "details" window at bottom of project manager.]

Drawing Values

Add drawing values which can be mapped to title block attributes.

1. In the Project Manager, double-click to expand the AEGS project.
2. Right-click on drawing AEGS11 and select Properties ➤ **Drawing Properties**.
3. Enter values:
   - **Sheet:** 11
   - **Drawing:** 0211

   Note: Drawing Description 1 and 2 were defined when the drawing was created.

4. Click OK.
5. Save the drawing.

Title Block Update

Update the title block attributes with the mapped AutoCAD Electrical toolset values.

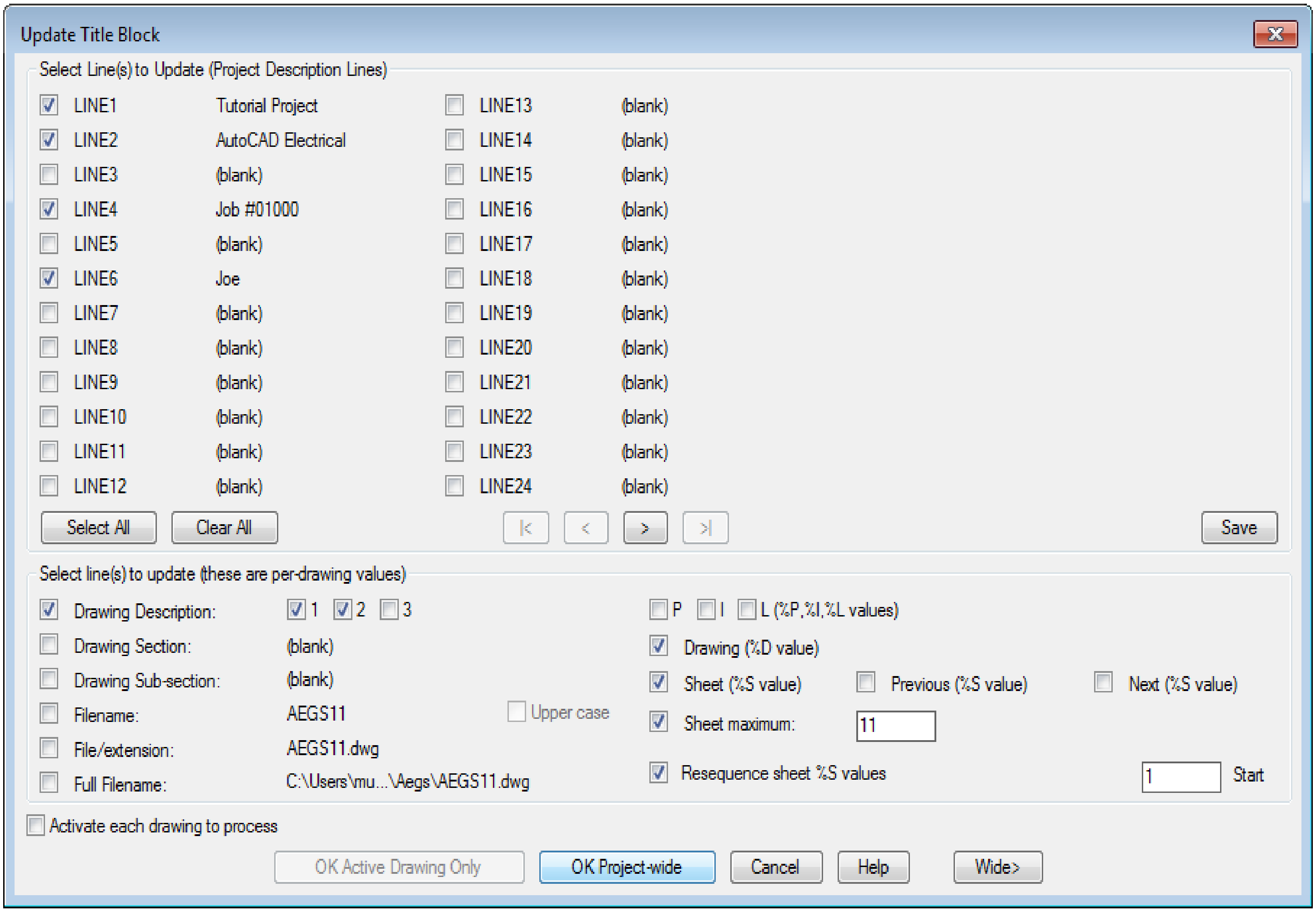
1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click to expand the AEGS project.
3. Double-click drawing AEGS11 to open it.
4. Click Project tab ➤ Other Tools panel ➤ **Title Block Update**.

   - The Update Title Block dialog box displays.

5. Select the project and drawing values to update on the title block.

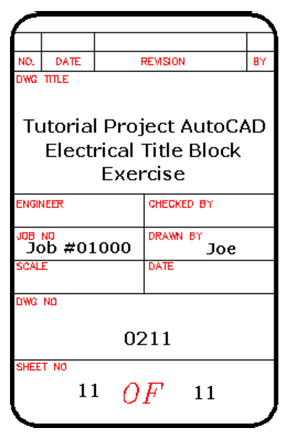
   - LINE1
   - LINE2
   - LINE4
   - LINE6

- Drawing Description: 1 and 2
- Drawing (%D value)
- Sheet (%S value)
- Sheet maximum
- Resequence sheet %S values: 1



6. Click **OK Project-Wide**.
7. Select drawings AEGS01 through AEGS05, and AEGS11 to process. Click Process v.

   Note: Drawings AEGS01 through AEGS05 are supplied with the WD_TB attribute on the title block for this exercise.
8. Click OK.



Customize Project Description Labels

Define the labels for project description lines displayed on dialog boxes.

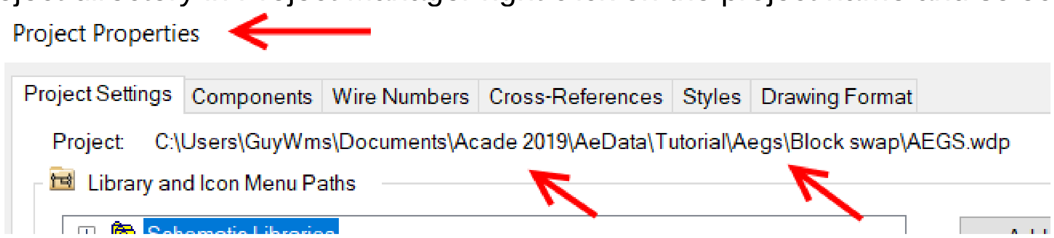
The title block and project description dialog boxes in AutoCAD Electrical toolset display generic labels like “LINE1”, “LINE2”, and so on. You can change these labels so they match up with the link to the title block. For example, you have linked the AutoCAD Electrical toolset data “LINE4” value to the “JOB#” attribute on the title block. What you want to see when AutoCAD Electrical toolset displays a title block-related dialog box is not “LINE4” but “Job Number.” A text file with a WDL extension defines the custom labels.

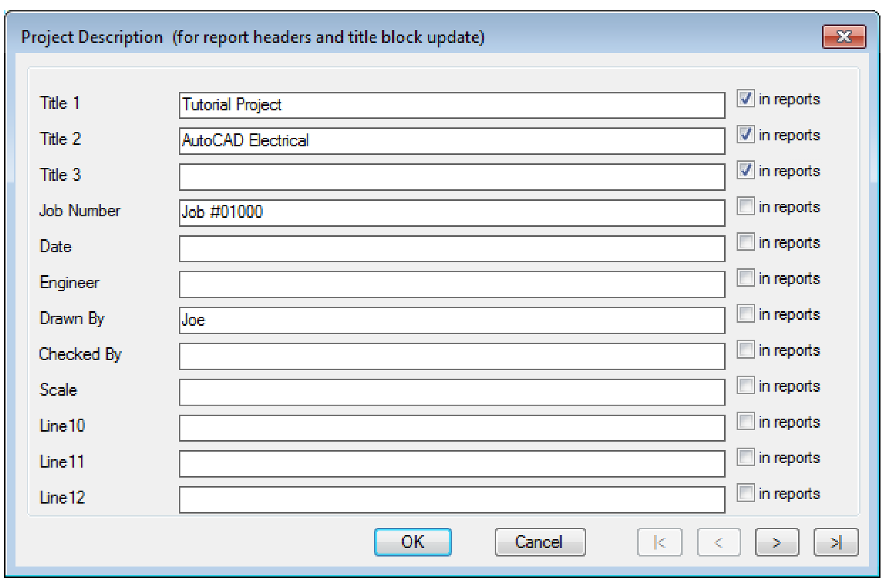
1. Use any generic text editor like Notepad or Wordpad and start a new text file.
2. Enter the lines as shown:

LINE1 = Title 1	LINE2 = Title 2	LINE3 = Title 3	LINE4 = Job Number	LINE5 = Date
LINE6 = Drawn By	LINE7 = Eng.	LINE8 = Checked By	LINE9 = Scale	

3. Save the file as AEGS_WDTITLE.WDL in the project folder: Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs.
4. Switch over to AutoCAD Electrical toolset.
5. Click Project tab ➤ Project Tools panel ➤ Manager.
6. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
7. In the Project Manager, right-click the project name, and select **Descriptions**.

To find the project directory in Project Manager right click on the project name and select Properties.





The labels match the values in the WDL file.

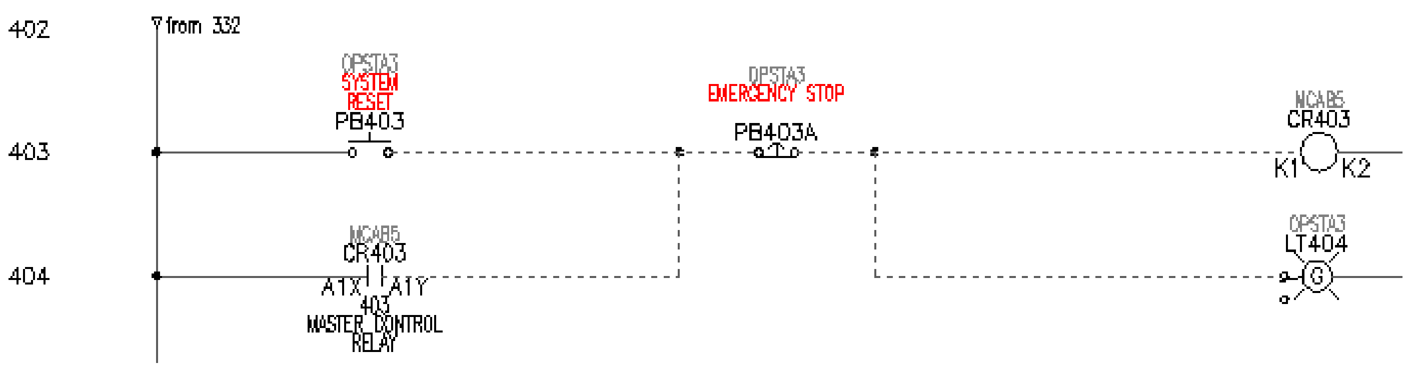
Wiring (here)

Insert and modify wires and ladders.

Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Wiring to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Insert wires
- Add ladder rungs
- Trim wires
- Insert a ladder
- Resequence ladder line reference numbers



About Wires

AutoCAD Electrical toolset treats AutoCAD® line entities as wires when the lines are placed on an AutoCAD Electrical toolset defined **wire layer**. The number of wire layers available in AutoCAD Electrical toolset is unlimited. These lines get tagged with wire numbers and show up in various wire connection reports.

Two wire segments connect if the end of one wire segment touches or falls within a small trap distance of any part of the other wire segment. This connection can be at the end of the other wire or anywhere along the length of the other wire. If the wire end falls within a trap distance from the wire connection-point attribute of a component, AutoCAD Electrical toolset considers a wire connected to a component.

The following rules determine the **wire layer** for a new wire segment:

- Wires that do not connect to an existing wire at either end are put on the **default wire layer** defined in the Create/Edit Wire Type dialog box.
- Wires that begin at an existing wire take on the same layer as the beginning wire.
- Wires that begin in space or at a component and end at an existing wire take on the layer of the ending wire.

Insert Wiring

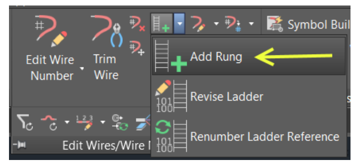
Add ladder rungs and draw wires.

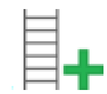
You can start or end a wire segment in empty space, from an existing wire segment, or from an existing component. If you start from a **component**, the wire segment snaps to the wire connection terminal closest to your pick point on that symbol. If the wire segment ends at another wire segment, a DOT (block name wddot.dwg) is applied if appropriate. If it ends at another component, the segment connects to the wire connection terminal closest to your pick point on that symbol.

Note: When inserting wires, if a wire already occupies a wire connection point, the new wire is drawn as an angled wire connection.

Insert wiring

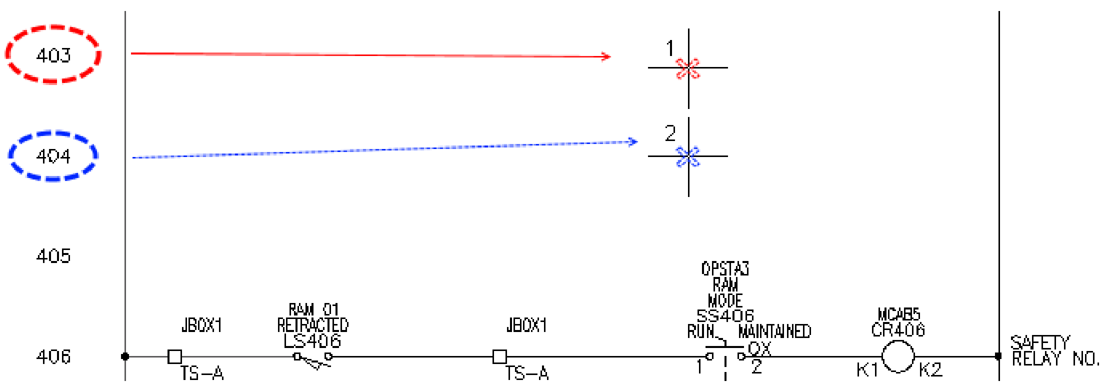
1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS04.dwg.
4. Zoom in on the upper left corner of the drawing. Make sure the hot and neutral vertical wires are displayed.
5. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Modify Ladder drop-down ➤ Add Rung.
6. Respond to the prompts as follows:



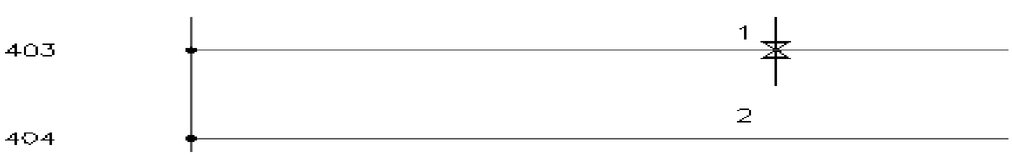


Add rung passing through this location or [wiretype (T)]: *Select a location between the two vertical bus wires beside line reference 403 (1)*

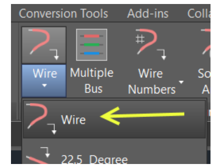
Add rung passing through this location or [wiretype (T)]: *Select a location between the two vertical bus wires beside line reference 404, underneath the newly created rung (2), press* ENTER



Two horizontal wires are created automatically between the vertical bus wires at the closest line reference location.



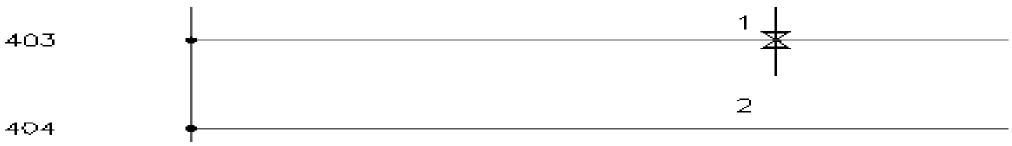
Create two vertical wires between two horizontal wires



1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wires drop-down ➤ Wire.
2. Respond to the prompts as follows:

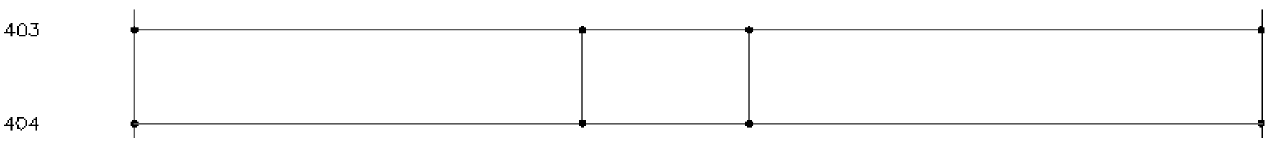
Specify wire start or [wireType/X=show connections]: *Select the top wire at line reference 403(1)*

Specify wire end or [V=start Vertical/H=start Horizontal/Continue]: *Select the lower wire at line reference 404 (2)*

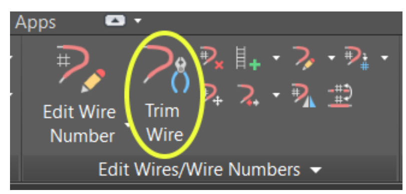


The color of temporary graphics changes for a new wire when AutoCAD Electrical toolset can connect the wire to an existing wire. Each component wire connection point displays as a green x at the wire connection when you enter X + ENTER during wire insertion. If you pan or zoom, repeat the command to view the wire connection points.

3. Insert another wire to the right of the new wire.
4. Press ENTER to exit the command.

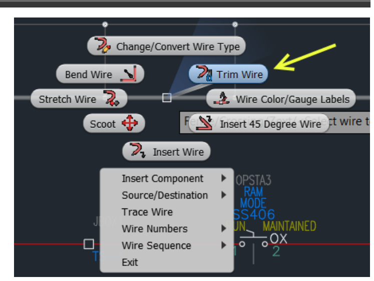


Trim a Wire



After you insert wires, you can trim them. The Trim Wire tool removes wire segments. You can trim single or multiple wires.

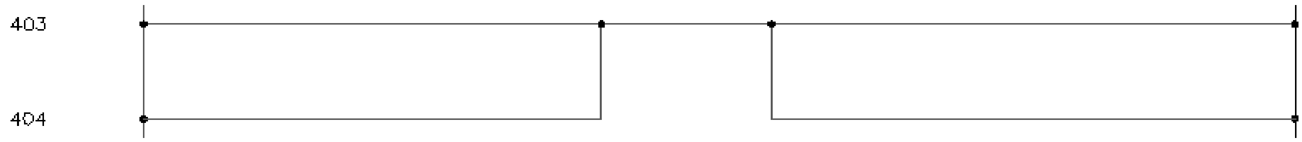
Trim a wire



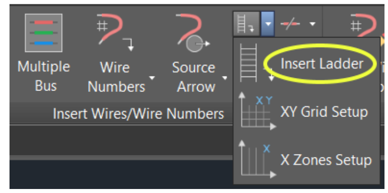
1. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Trim Wire.
2. Respond to the prompts as follows:

Fence/Crossing/Zext/<Select wire to TRIM>:

Select the wire segment at line reference 404 between the two vertical wires (1), then right-click to exit the command

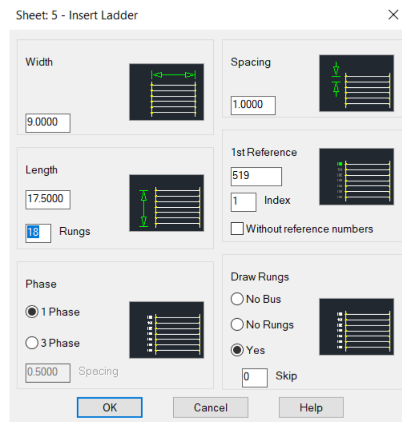


Insert a Single-Phase Ladder



Add a ladder with a defined width, number of rungs, and first rung reference number.

You can insert a ladder into a drawing at any time. A drawing can have multiple ladders, as well as single-phase and three-phase ladders. The ladders can have different parameters, such as: rung spacing, number of rungs, ladder width.

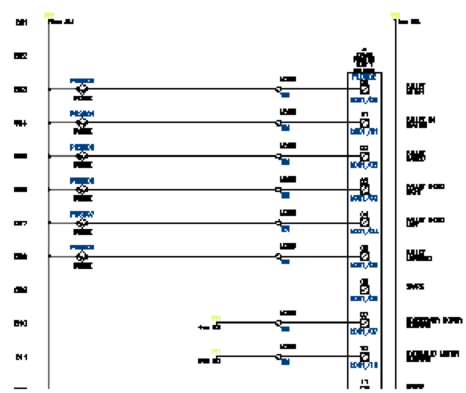


Insert a **Single-Phase** ladder

1. Open AEGS05.dwg.
2. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Ladder drop-down ➤ Insert Ladder.
3. In the Insert Ladder dialog box, specify:

Width: 9.000	Spacing: 1.0000	1st Reference: 519
Index: 1	Rungs: 18	Phase: 1 Phase
Draw Rungs: Yes	Skip: 0	

You do not specify the Length since it is automatically calculated once the first Reference, Index, and Rungs are specified. **Note**: Reference 519 represents Page 5, Reference 19.



4. Click OK.
5. Respond to the prompts as follows:

Specify start position of first rung or [wireType]: *Enter* 16, 21 *press* ENTER

Note: You can also specify the start position of the first rung by left-clicking a location on the drawing with your mouse. A single phase ladder is inserted in the drawing.

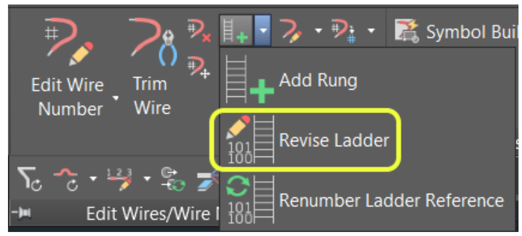
Resequencing Ladders

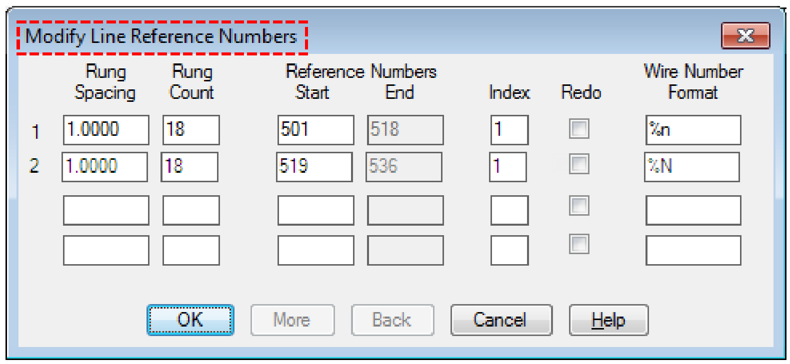
AutoCAD Electrical toolset drawings can be easily **renumbered** and **retagged** with a minimum of manual clean-up. You can resequence: **line reference numbers**, **component tags**, and **wire numbers**. It is useful when a drawing has been copied from a previous project and the line reference numbers and tagging format of the drawing do not conform to the project requirements.

1. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Modify Ladder drop-down ➤ **Revise Ladder**

The Modify Line Reference Numbers dialog box displays a list of ladders in the drawing.

2. Change the beginning line reference numbers for each ladder. Change the first ladder to 101 (column 1, line 01) and the second ladder to 201 (column 2, line 01).
3. Click OK.





Schematic Components (here)

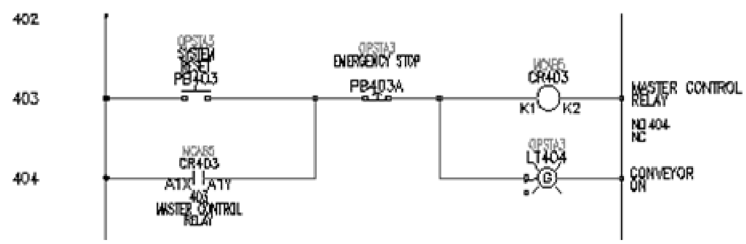
Schematic Components Tutorial

Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Schematic components to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Insert and modify schematic components.

Follow the workflow topics listed below to accomplish these tasks:

- Insert a parent component
- Scoot a component
- Insert a child component
- Align components
- Edit a component
- Link components
- Edit catalog information
- Add a catalog entry



About Schematic Components

Understand what makes a block an AutoCAD Electrical toolset **schematic component**.

An AutoCAD Electrical toolset schematic component is an AutoCAD block with certain expected attributes. When inserting components, use AutoCAD Electrical toolset to:

- Break wires
- Assign unique component tags
- Cross-reference related components
- Enter values for catalog information, component descriptions, location codes, and so on

AutoCAD Electrical toolset supplies a schematic symbol dialog box for finding and inserting schematic components. It also triggers some additional features.

- Automatic wire breaks
- Component tagging
- Real-time cross-referencing
- Component annotation

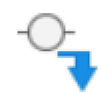
Inserting Components

Insert a parent component, and assign **description**, **location**, and **catalog** values.

AutoCAD Electrical toolset employs a parent/child relationship for schematic components. The **parent coil** symbol and the **child contact** symbols represent a relay coil with a certain number of contacts. When the parent coil symbol is inserted, it is assigned a unique **component tag**. When the child contact symbols are inserted, the child is related to the parent and the parent tag is assigned to the child symbol. In this exercise, you insert components on the wires previously defined in AEGS04.dwg.

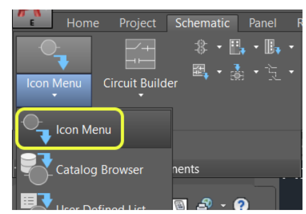
Insert a Parent component

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS04.dwg.
4. Zoom in on the upper left corner of the drawing.



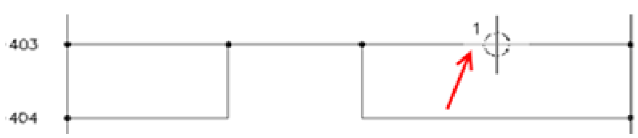
5. Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Icon Menu.

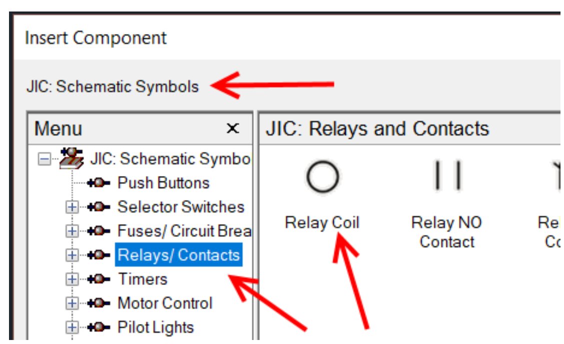
6. In the **Insert Component**: JIC Schematic Symbols dialog box, click **Relays/ Contacts**.



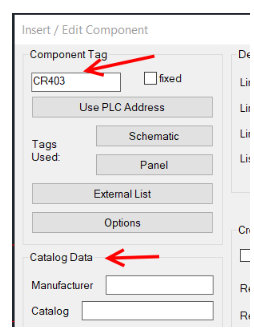
7. In the JIC: Relays and Contacts dialog box, click **Relay Coil**.
8. Respond to the prompts as follows:

   Specify insertion point: *Position the component on the wire at line reference 403 near the neutral wire and click (1)*





   If you select directly on the wire or near to it, the coil symbol breaks the underlying ladder wire and reconnects. If the underlying wire did not break, you did not select close enough to the wire. To try again, click Cancel on the Insert/Edit Component dialog box.

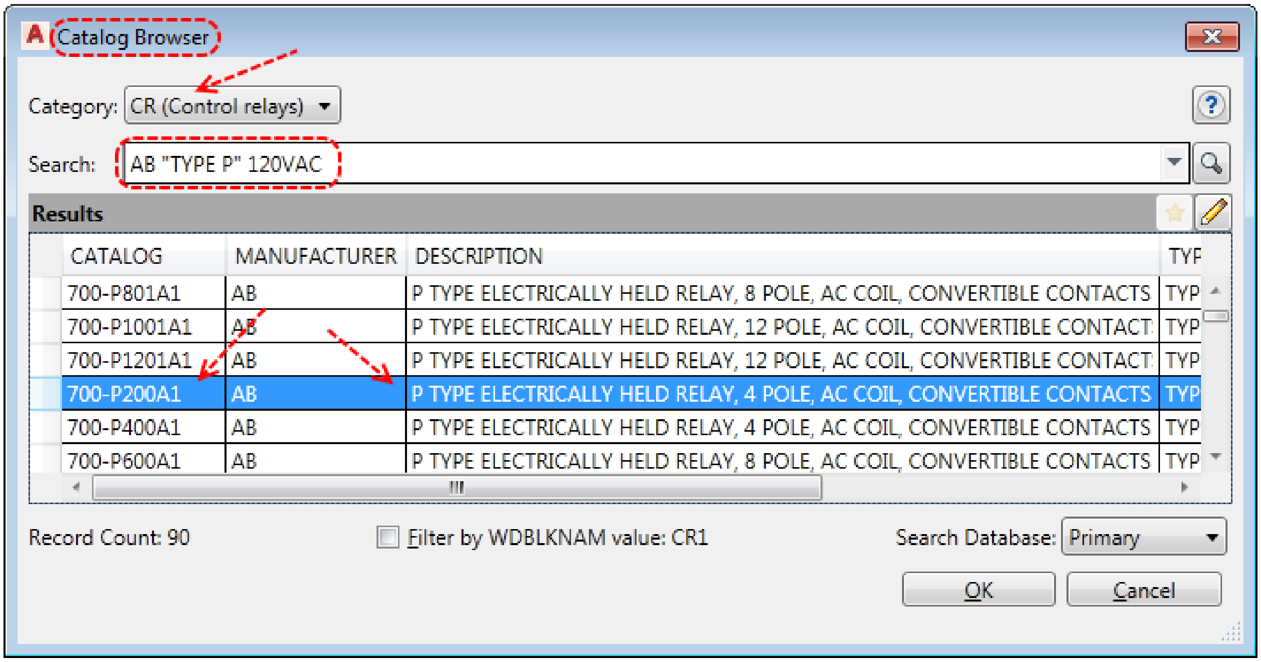


   Right-click or press ENTER to repeat the command. Turning on **Snap** helps (0.125 is a good setting to use). This tool inserts components into alignment with underlying wires, it does not align components side-to-side. If you want to insert components in neat columns, you have three options: use AutoCAD **Snap** when inserting components; use the **Scoot** command to move components and connected wires in place; or use the **Align Component** tool.

9. In the Insert/Edit Component dialog box, verify that the Component Tag is set to CR403.

   AutoCAD Electrical toolset automatically determines the unique tag name for the new relay based on the line reference location that you inserted the symbol on. "CR" indicates that it is a control relay and "403" indicates that the symbol is on line reference 403. If you inserted this symbol on line reference 404 then the tag name would be "CR404."

10. In the Catalog Data section, click Lookup.
11. On the Catalog Browser dialog box, enter **AB "TYPE P" 120VAC** as the search string.
12. Click .
13. Select 700-P200A1.



14. In the Catalog Browser dialog box, click OK.

    The selected manufacturer code and catalog number display in the Insert/Edit Component dialog box. When you click OK on the dialog box, the values transfers to the symbol.

Note: Sample catalog information is provided with AutoCAD Electrical toolset in **Access Database format (.mdb)**. If your company uses its own internal coding system instead of manufacturer catalog numbers, substitute those numbers into the catalog database.

15. In the Insert/Edit Component dialog box, Description section, specify:

    Line 1: MASTER CONTROL   Line 2: RELAY

    Up to three lines of description text can be entered as a description for components. If the third description line is unavailable, the symbol does not carry an attribute for a third line of description.

    Note: You can specify a description by entering text or by clicking Defaults to select from a list of standard component descriptions.

16. In the Insert/Edit Component dialog box, **Location code** section, click **Drawing**.

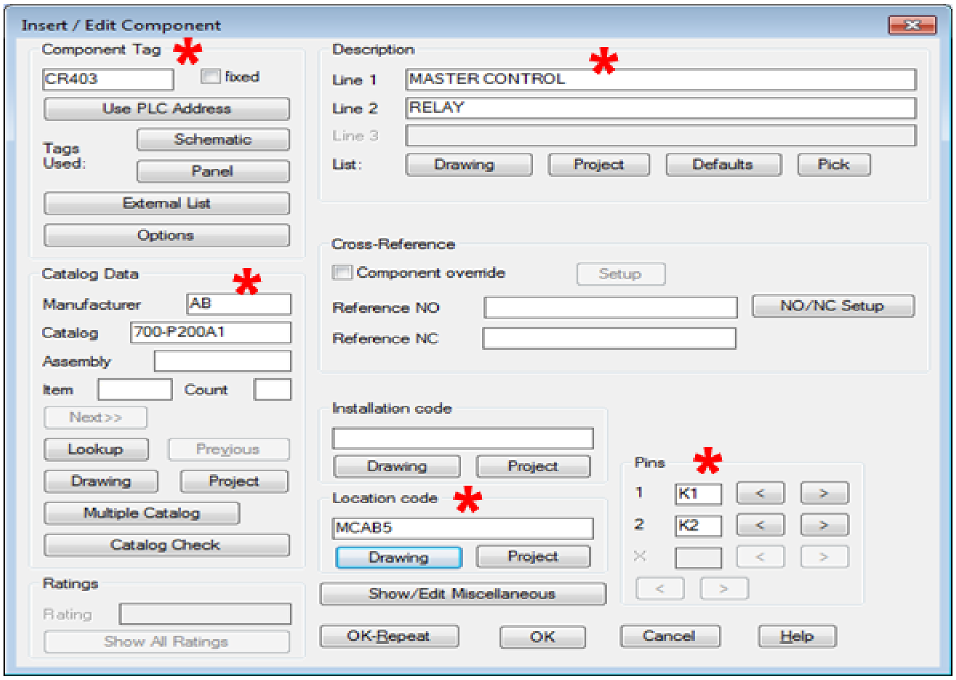
    AutoCAD Electrical toolset does a quick read of the drawing file and returns a list of all location codes used so far.

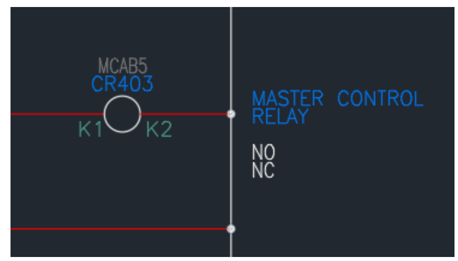
17. In the All Locations - Drawing dialog box, select MCAB5 and click OK.

Note: You can also include **an external "LOC" location list** in the project "LOC" list to help with consistency. To use this feature, create a file called ***default.loc*** and put it in an AutoCAD Electrical toolset search directory. The format for this text file is each location on its own line in the file with no leading spaces. You can also create a **project-specific file** by naming it the same as your project but with a **.loc extension**.

18. In the Insert/Edit Component dialog box, the pin values are inserted based on the selected catalog number:

Pins: 1: K1 Pins: 2: K2





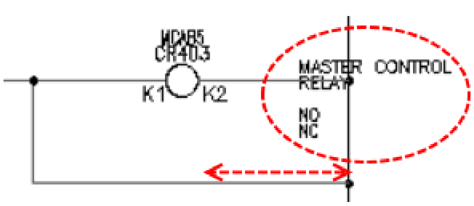
19. In the Insert/Edit Component dialog box, click OK. Any values entered here are saved as attribute values on the symbol itself.

Relocating Components

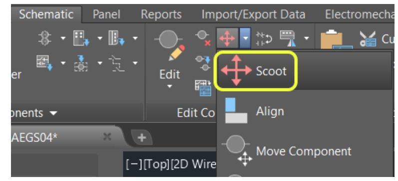
Scoot a component along a wire, and insert child components.

If the component was not inserted in the correct location, you can scoot the component. Use the **Scoot** tool to select a component or wire number and slide it back and forth along the wire while keeping everything connected. You can select a wire or a whole rung of circuitry. If there are any parent components among the scooted items, you are asked if you want to retag the scooted components.

The Scoot tool works on: wire numbers, components, terminals, PLC I/O modules, jogs in dashed link lines, signal arrows, wires, and wires with wire-crossing loops.

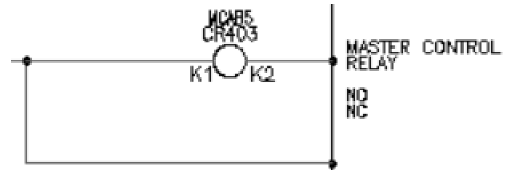


Scoot a component



1. Click Schematic tab ➤ Edit Components panel ➤ Modify Components drop-down ➤ Scoot.
2. Respond to the prompts as follows:

Select component, wire, or wire number for SCOOT: *Select the component that was inserted at line reference 403*



The cursor changes to a box.

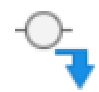
Select component, wire, or wire number for SCOOT: *Move the cursor to the right and click, right-click to exit the command*

The component moves to its new location.

You can use the Scoot tool to grab a component or a wire number and slide it back and forth along a wire. You can grab a wire or a whole rung of circuitry and scoot it to a new position, while keeping everything connected.

The steps to insert a parent component and a child component are the same, except when you annotate the symbol.

Insert a child component

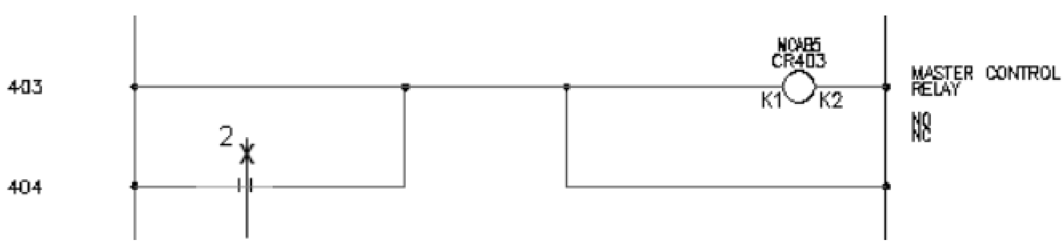


1. Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Icon Menu.
2. In the **Insert Component**: JIC Schematic Symbols dialog box, click Relays/ Contacts.

3. In the JIC: Relays and Contacts dialog box, click Relay NO Contact.

4. Respond to the prompts as follows:

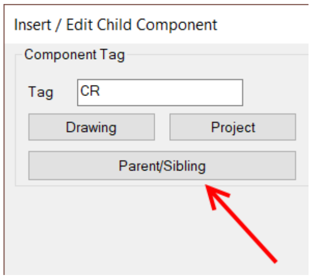
Specify insertion point: *Position the cursor on the wire at line reference 404 near the hot wire and click (1)*

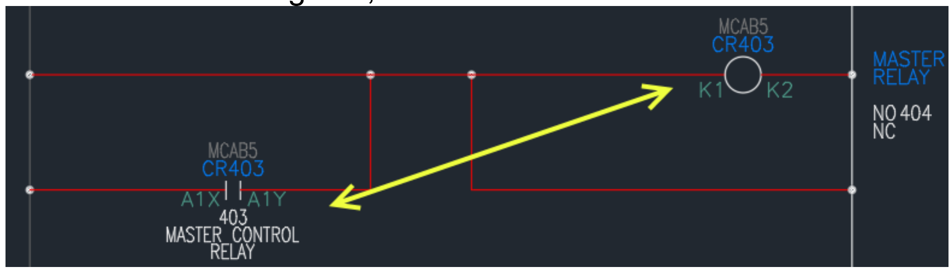


The Insert/Edit Child Component dialog box displays. Notice that AutoCAD Electrical toolset did not automatically assign a tag name for the relay contact; there is just a generic "CR" in the edit box. Determine the relay contact tag name. **A relay contact is a child component that must link to a parent relay coil on a drawing in the active project**. The child gets the same tag name that is found on the parent relay coil.

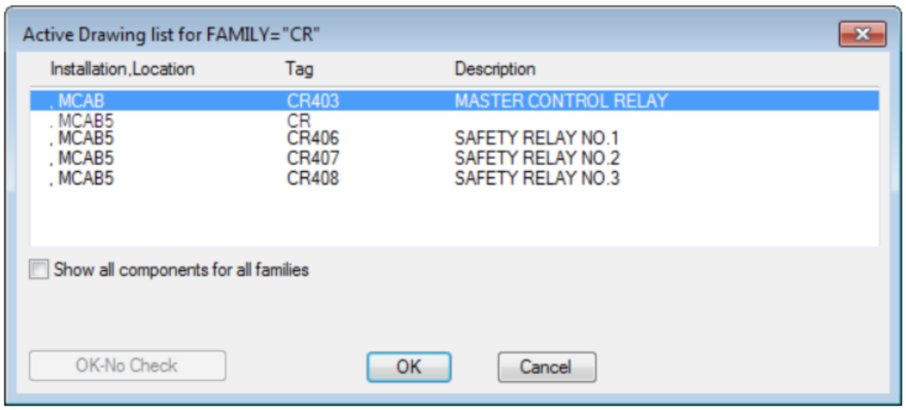
Assign the tag name by clicking Parent/Sibling and picking the parent in the drawing. Or, click Drawing or Project to select from a list of components with the same family name.

5. In the Insert/Edit Child Component dialog box, Component Tag section, click **Drawing**.
6. In the Active Drawing list for FAMILY="CR" dialog box, select:





MCAB5 CR403 MASTER CONTROL RELAY

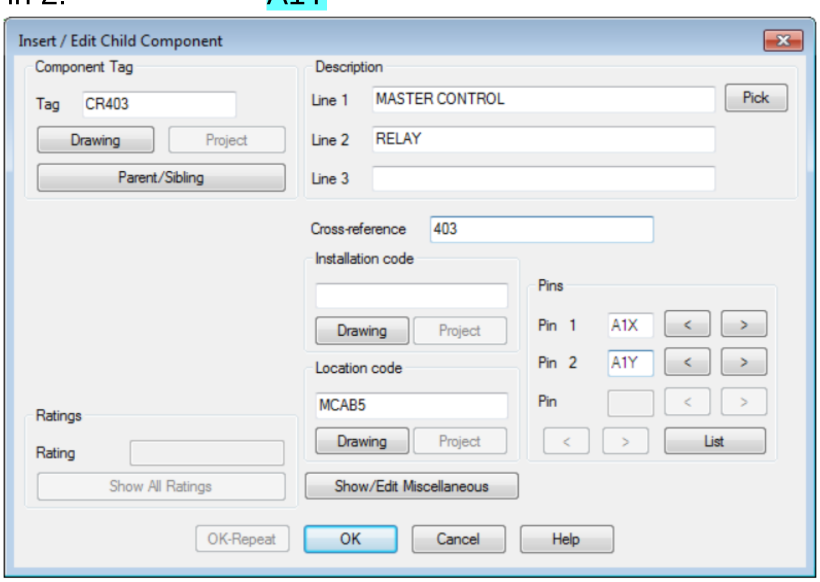


7. Click OK.

The values of the parent are immediately transferred to the contact.

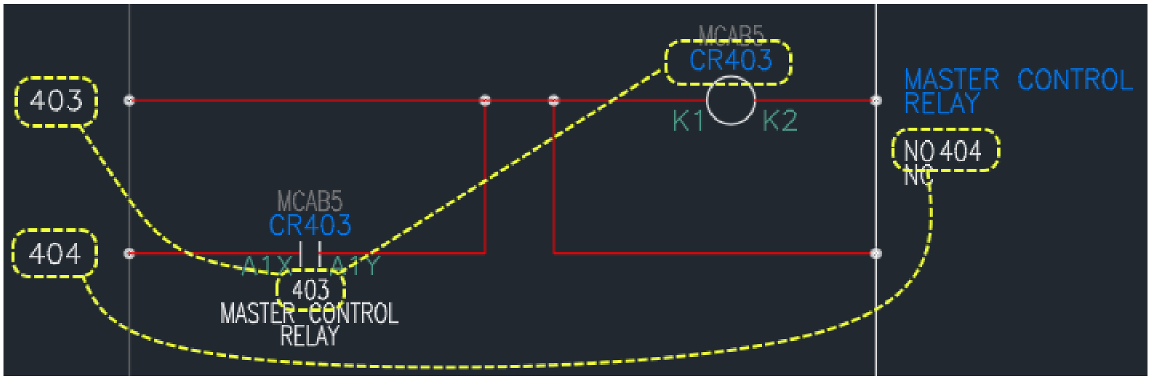
8. In the Insert/Edit Child Component dialog box, verify that the following options are specified:

Component Tag:	CR403
Description: Line 1:	MASTER CONTROL
Description: Line 2:	RELAY
Cross-reference:	403
Location code:	MCAB5
Pins: Pin 1:	A1X
Pins: Pin 2:	A1Y



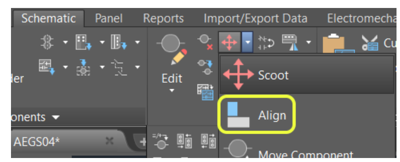
9. In the Insert/Edit Child Component dialog box, click OK.

The child component is inserted. It is cross-referenced in real time. The coil is annotated with the line reference number of the new child contact. The child contact gets annotated with the line reference location of the parent coil.



Aligning Components

Align multiple components to a selected component.



Align a component

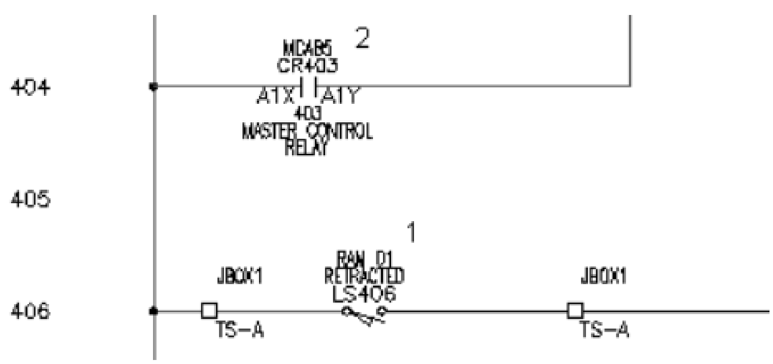
1. Click Schematic tab ➤ Edit Components panel ➤ Modify Components drop-down ➤ Align.
2. Respond to the prompts as follows:

Pick component to align with (Horizontal/<Vertical>):

Select the normally open limit switch component near the hot wire at line reference 406 (1)

A dashed line displays.

Select objects: *Select the previously inserted child contact component near the hot wire at line reference 404 (2), right-click*

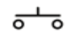


The aligned component is placed.

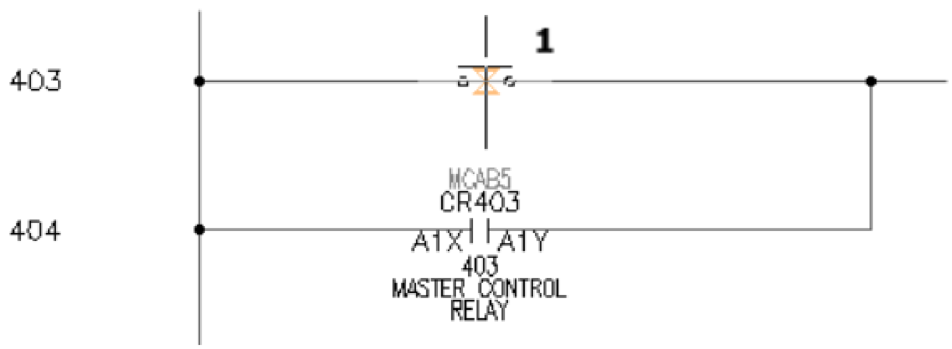
Inserting Components (continued)

Now you insert a system reset push button, pilot light, and an emergency stop push button to make up the circuit.

Insert a system reset button 

1. Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Icon Menu.
2. In the Insert Component: JIC Schematic Symbols dialog box, click Push Buttons. 
3. In the JIC: Push Buttons dialog box, click Push Button NO. 
4. Respond to the prompts as follows:

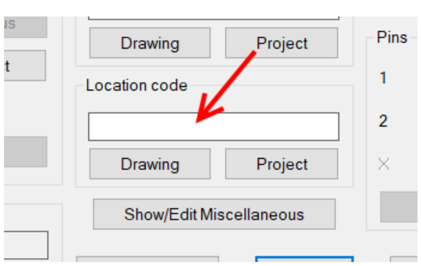
Specify insertion point: *Position the push button on the wire at line reference 403 near the hot wire and click (1)*

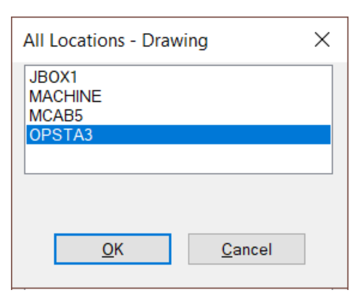


5. In the Insert/Edit Component dialog box, verify the following: **Component Tag: PB403**

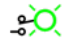
   AutoCAD Electrical toolset automatically assigned the tag name based on the line reference.

6. In the Descriptions section, specify: **Line 1: SYSTEM** **Line 2: RESET**
7. In the Location code section, click **Drawing**.
8. In the All Locations - Drawing dialog box, select **OPSTA3** and click OK.
9. In the Insert/Edit Component dialog box, click OK.

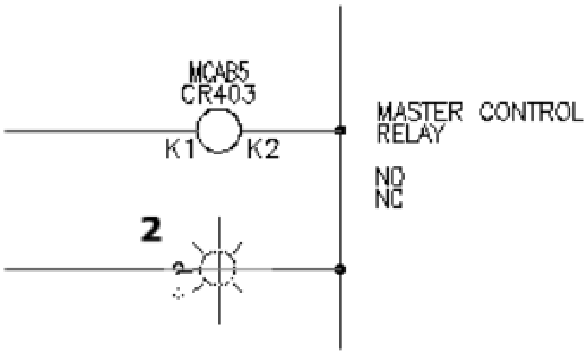




Insert a pilot light

1. Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Icon Menu.
2. In the Insert Component: JIC Schematic Symbols dialog box, click Pilot Lights. 
3. In the JIC: Pilot Lights dialog box, click Green Press to Test. 
4. Respond to the prompts as follows:

   Specify insertion point: *Position the pilot light on the wire at line reference 404 near the neutral wire and click (2)*



   Tip: having **Snap** turned on makes positioning the pilot light easier.

5. In the Insert/Edit Component dialog box, verify:

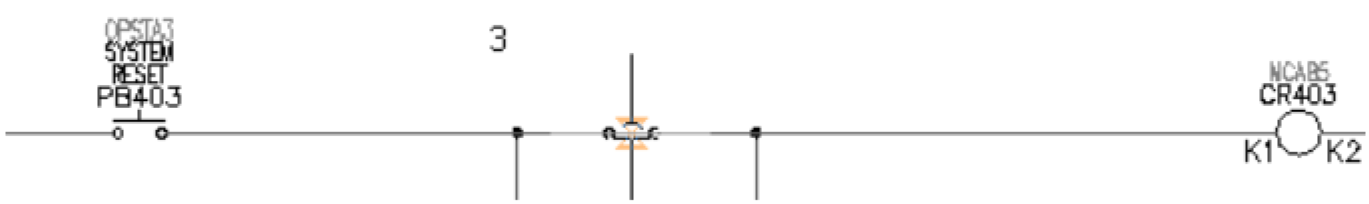
Component Tag: LT404 Line 1: CONVEYOR Line 2: ON

6. In the Location code section, click **Drawing**.
7. In the All Locations - Drawing dialog box, select **OPSTA3** and click OK.
8. In the Insert/Edit Component dialog box, click OK.

Insert a push button for emergency stop

1. Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Icon Menu.
2. In the Insert Component: JIC Schematic Symbols dialog box, click Push Buttons. 
3. In the JIC: Push Buttons dialog box, click Mushroom Head NC. 
4. Respond to the prompts as follows:

   Specify insertion point: *Position the push button on the middle of the wire at line reference 403 and click (3)*

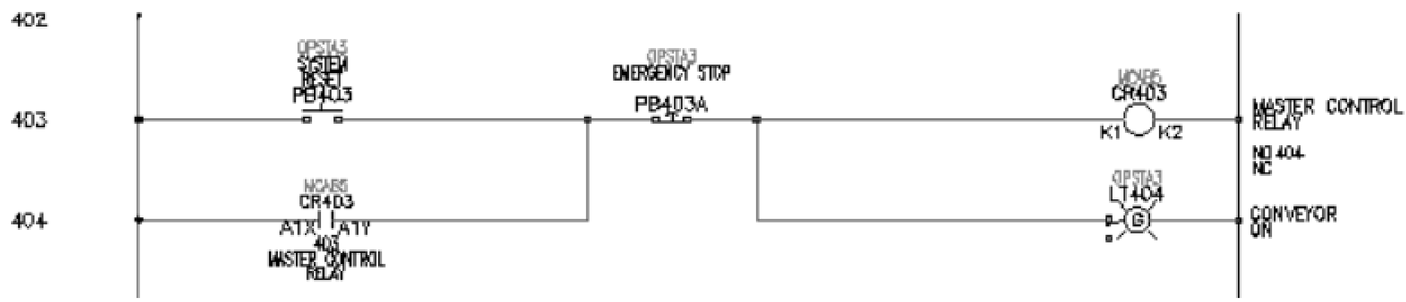


5. In the Insert/Edit Component dialog box, verify: Component Tag: **PB403A**

AutoCAD Electrical toolset automatically assigned the tag name based on the line reference. It added the “A” suffix since it is your second push button on this line reference.

6. In the Descriptions section, specify: **Line 1: EMERGENCY STOP**
7. In the Location code section, click **Drawing**.
8. In the All Locations - Drawing dialog box, select OPSTA3 and click OK.
9. In the Insert/Edit Component dialog box, click OK.

Your finished schematic resembles the following:

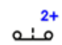


Editing Components

You can go back to a component at any time and change it.

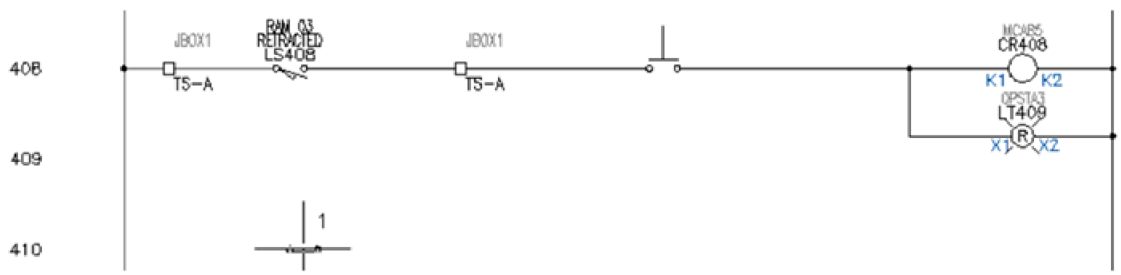
The Edit Component tool can change: description, tag, catalog number, location code, terminal numbers, and rating values.

Insert a child contact

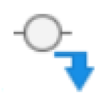
1. Zoom in on the blank ladder rung at line reference 410.
2. Press F9 to turn on **SNAP**.
3. Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Icon Menu.
4. In the Insert Component: JIC Schematic Symbols dialog box, click Selector Switches. 
5. In the JIC: Selector Switches dialog box, click 2nd+ NC Contact. 

6. Respond to the prompts as follows:

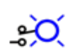
Specify insertion point: *Position the selector switch at line reference 410 near the left side of the ladder and click (1)*



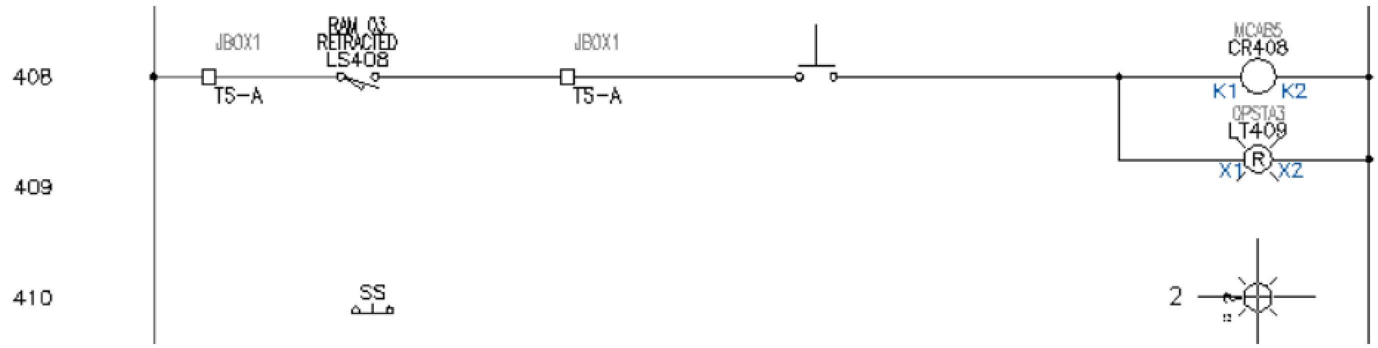
7. In the Insert/Edit Child Component dialog box, click OK.



Insert a pilot light

1. Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Icon Menu.
2. In the Insert Component: JIC Schematic Symbols dialog box, click Pilot Lights. 
3. In the JIC: Pilot Lights dialog box, click Blue Press to Test. 
4. Respond to the prompts as follows:

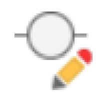
Specify insertion point: *Position the pilot light at line reference 410 near the neutral wire but exactly in line with the selector switch and click (2)*



5. In the Insert/Edit Component dialog box, verify: **Component Tag: LT410**
6. In the Descriptions section, specify: **Line 1: MAINT** **Line 2: MODE**
7. In the Insert/Edit Component dialog box, click OK.

Edit a child contact

1. Press F9 to turn off SNAP .
2. Click Schematic tab ➢ Edit Components panel ➢ Edit Components drop-down ➢ Edit.

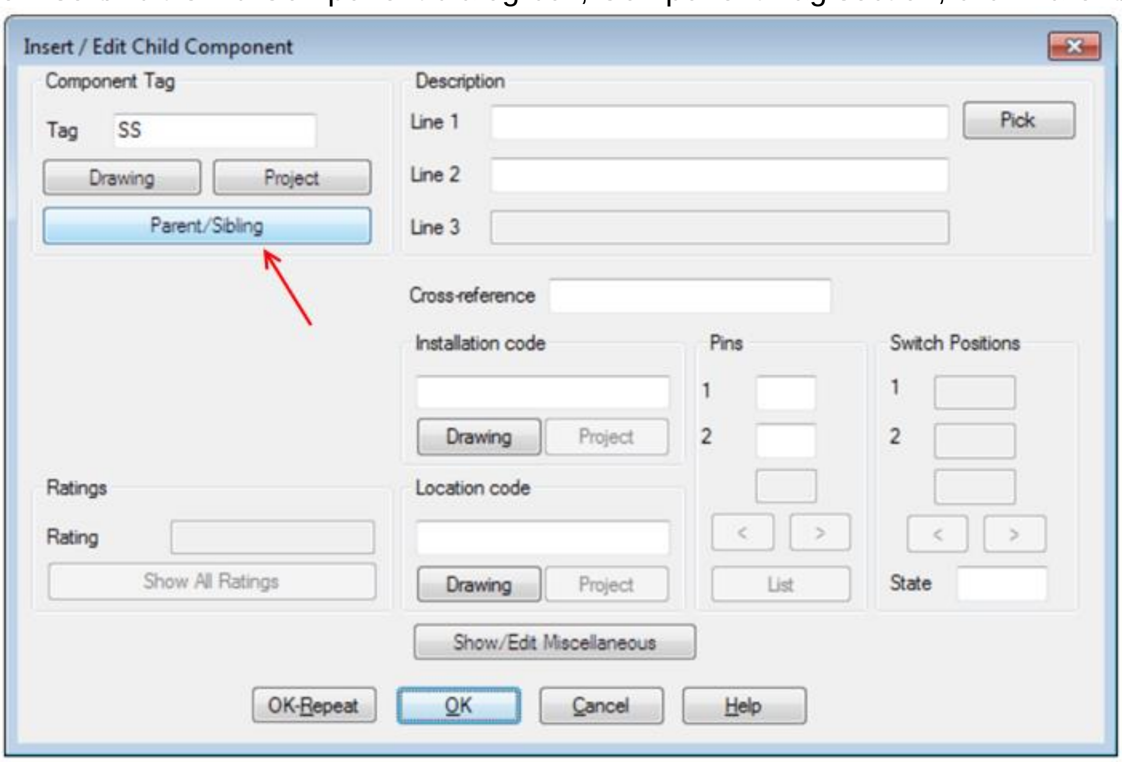


Note: You can also right-click on a component and select Edit Component from the menu.

3. Respond to the prompts as follows:

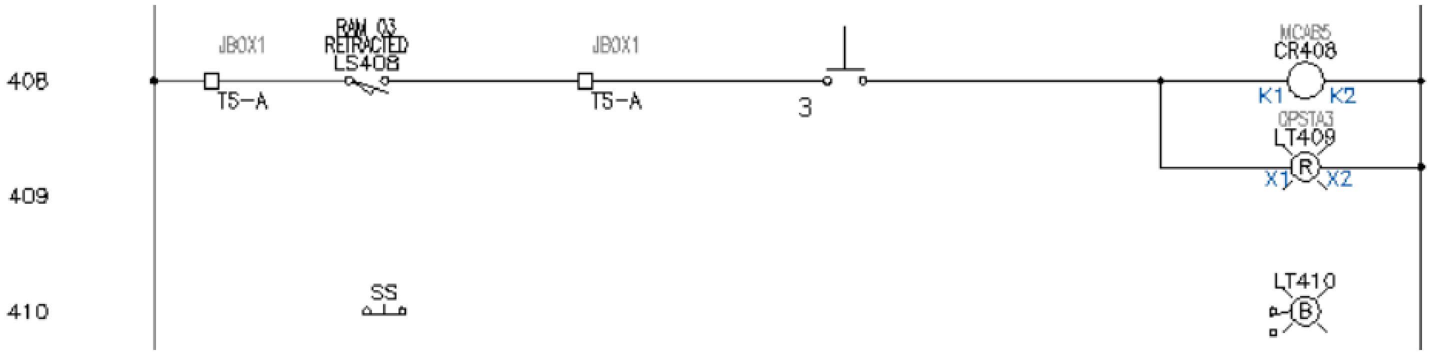
Select component/cable/location box to EDIT: *Select the selector switch on line reference 410*

4. In the Insert/Edit Child Component dialog box, Component Tag section, click Parent/Sibling.

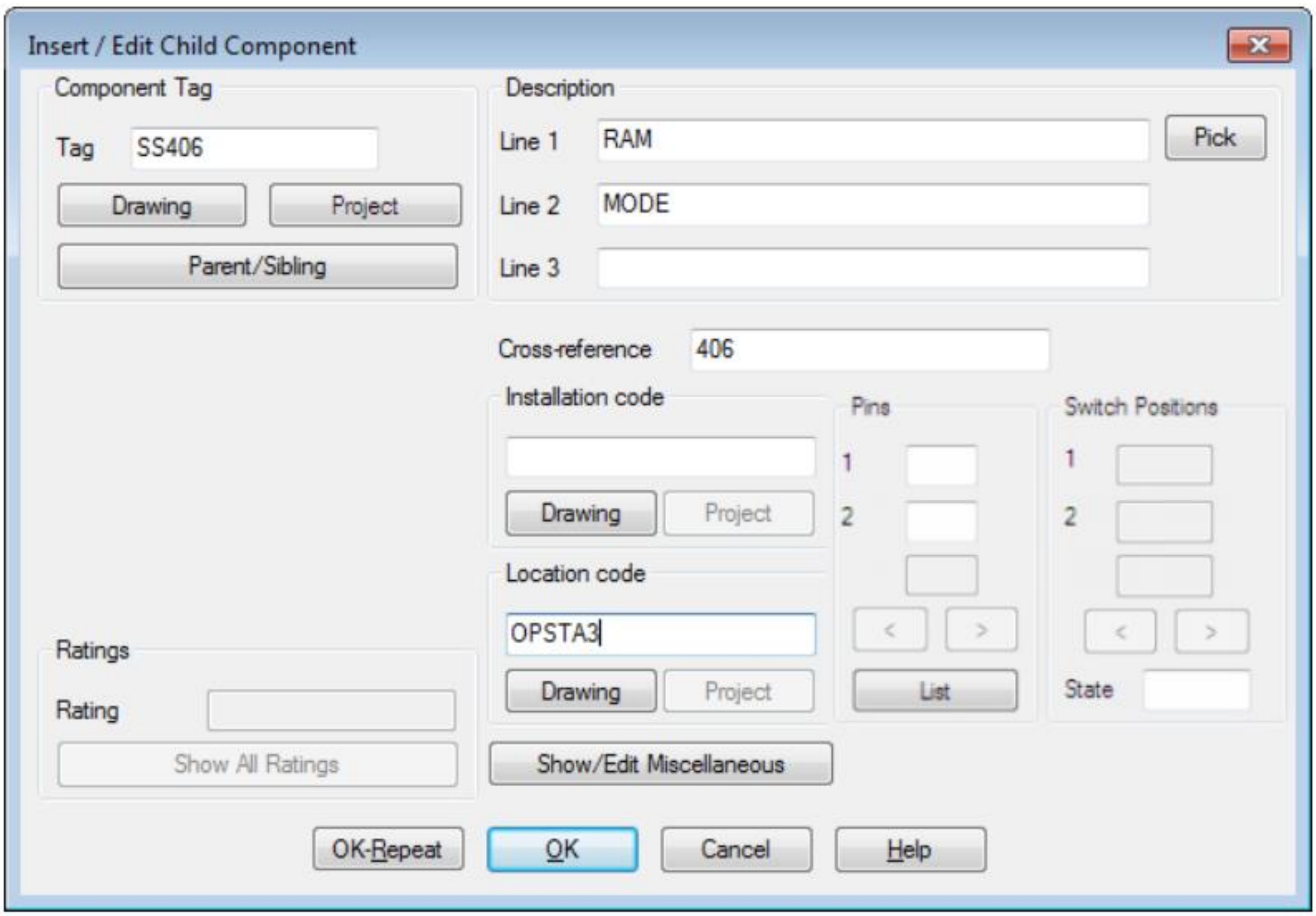


5. Respond to the prompts as follows:

Select component: *Select the bottom sibling contact (3) of the existing switch on line reference 408*

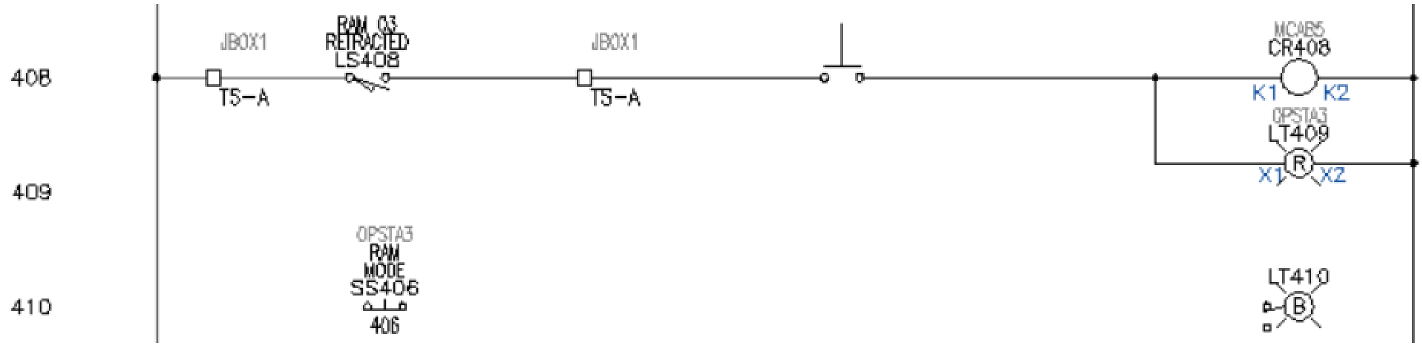


AutoCAD Electrical toolset reads the sibling contact and transfers the appropriate annotation to your new switch contact.



6. In the Insert/Edit Child Component dialog box, click OK.

   The sibling contact information displays on the drawing.



408
409
410
JBOX1
TS-A
RAM 03
RETRACTED
LS408
JBOX1
TS-A
MCAB5
CR408
K1
K2
OPSTA3
LT409
R
X1
X2
OPSTA3
RAM
MODE
SS406
406
LT410
B

Linking Components

Add dashed **link lines** to related components.

In this exercise, you link the selector switch you inserted to the existing RAM MODE selector switch residing on line reference 406 through 408 using dashed link lines.

Connect components using wires

1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wires drop-down ➤ Wire.
2. Respond to the prompts as follows:

   Specify wire start or [wireType/X=show connections]: *Click the wire connection point on the right-hand side of the switch contact (4)*

   Specify wire end or [Continue]: *Drag the wire to the right and click the wire connection point on the left-hand side of the blue pilot light (5)*

   Specify wire start or [Scoot/wireType/X=show connections]: *Click the left-hand side of the switch contact*

   Specify wire end or [Continue]: *Drag the wire to the left and click the left-hand vertical bus wire*

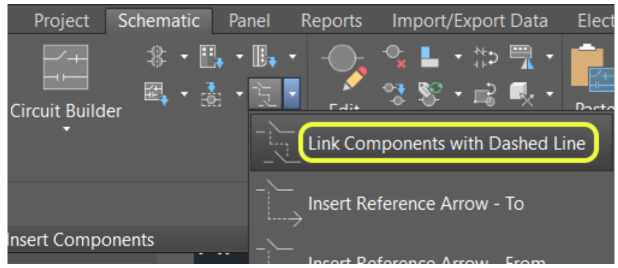
   The wire automatically ends on the bus and inserts a wire connection dot.

3. Repeat the process to connect the right-hand side of the blue pilot light to the vertical bus wire.
4. Right-click and select Enter to finish creating the wire connections.

If you lay a wire over the top of a series of components, AutoCAD Electrical toolset automatically breaks and reconnects to the underlying wire connection points.

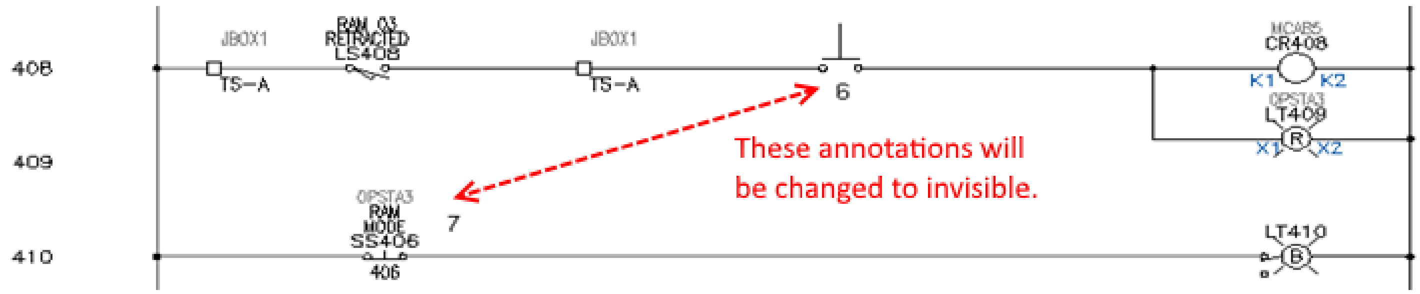
Link components

1. Click Schematic tab ➤ Insert Components panel ➤ Dashed Link Line drop-down ➤ Link Components with Dashed Line.
2. Respond to the prompts as follows:



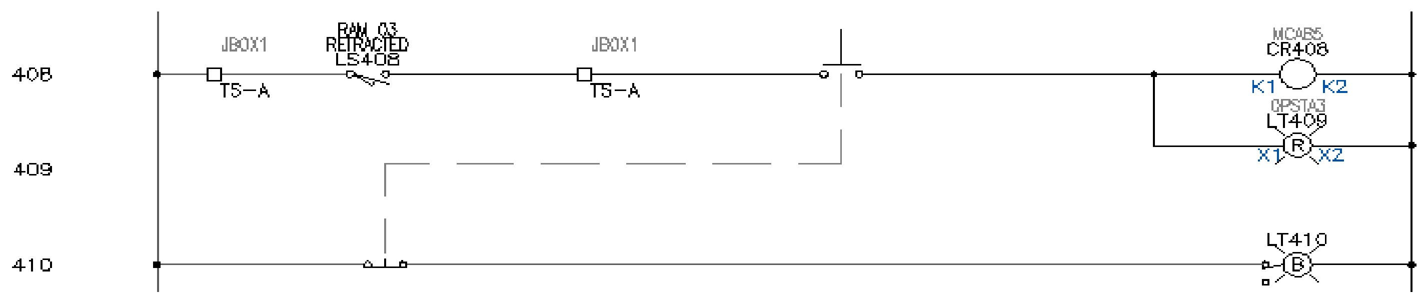
Component to link from: *Click the contact of the switch on line reference 408 (6)*

Component to link to: *Click anywhere on your new switch contact (7), right-click* or press Enter.



The annotation of the contact is changed to **invisible**. A dashed link line is drawn from the bottom of the upper contact to the top of your new contact.

Your finished schematic resembles the following:



Note: The Scoot command is fully compatible with dashed line links. Scooting one contact left or right causes both links to update automatically. You can even scoot the horizontal “jog” in the dashed link line up or down.

Editing Catalog Information

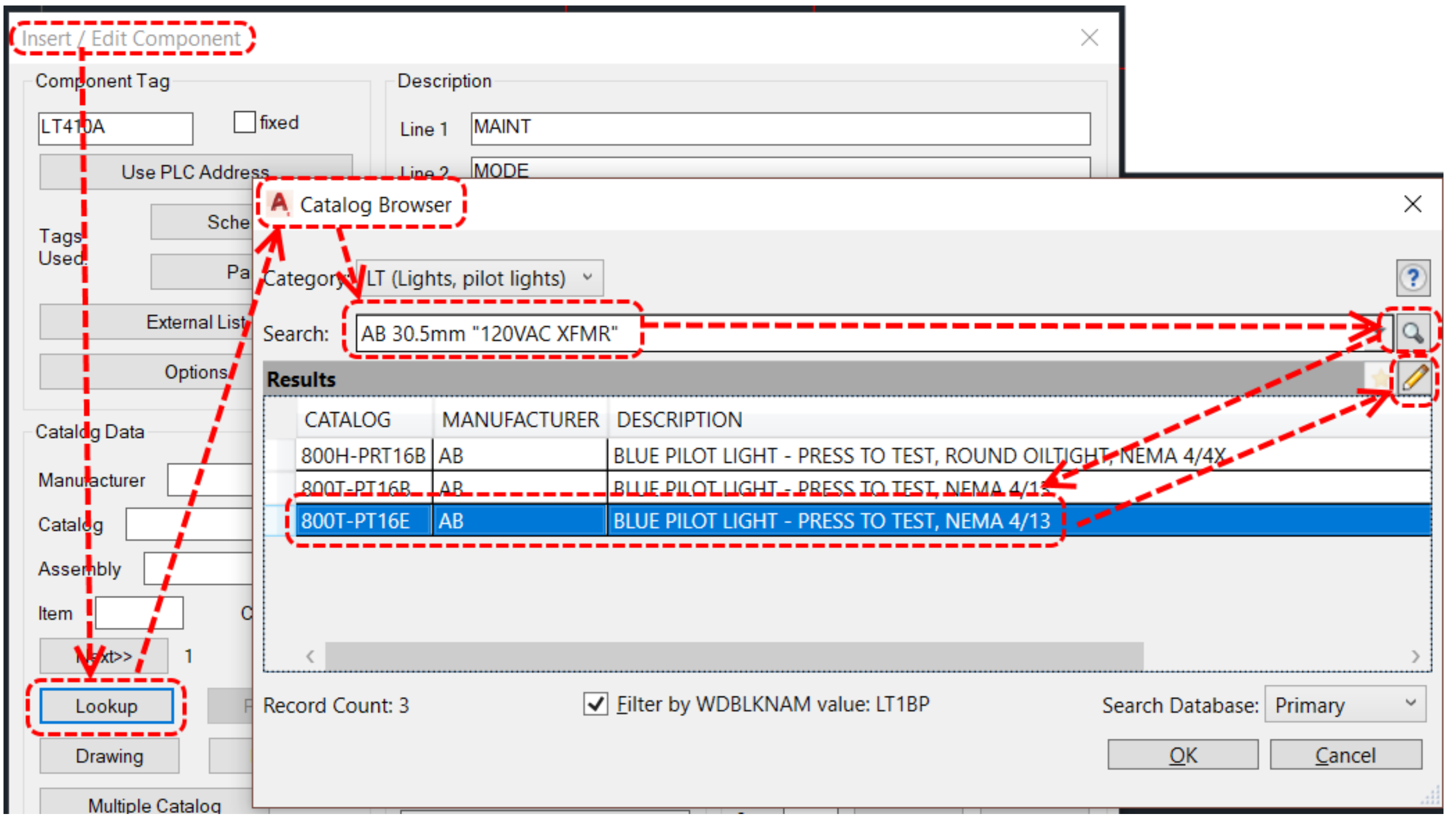
Change the catalog assignment for a component and create a catalog entry in the catalog database.

Sample catalog information is supplied with AutoCAD Electrical toolset. The information is held in tables in an Access Database file (.mdb) that is populated with sample vendor data.

Use **search** to display catalog numbers selectively for a component type.

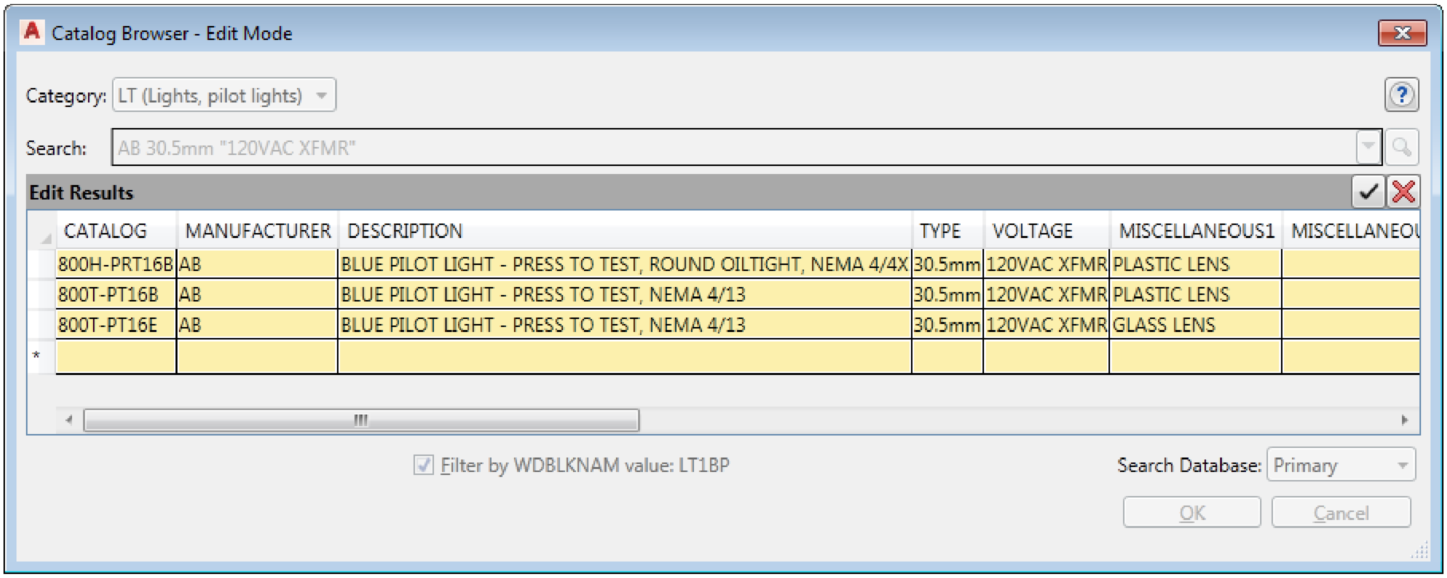
Search catalog data

1. Right-click LT410 and select Edit Component.
2. In the Insert/Edit Component dialog box, Catalog Data section, click Lookup.
3. Enter the search AB 30.5mm "120VAC XFMR" and click  .
4. Change the catalog assignment to 800T-PT16E.

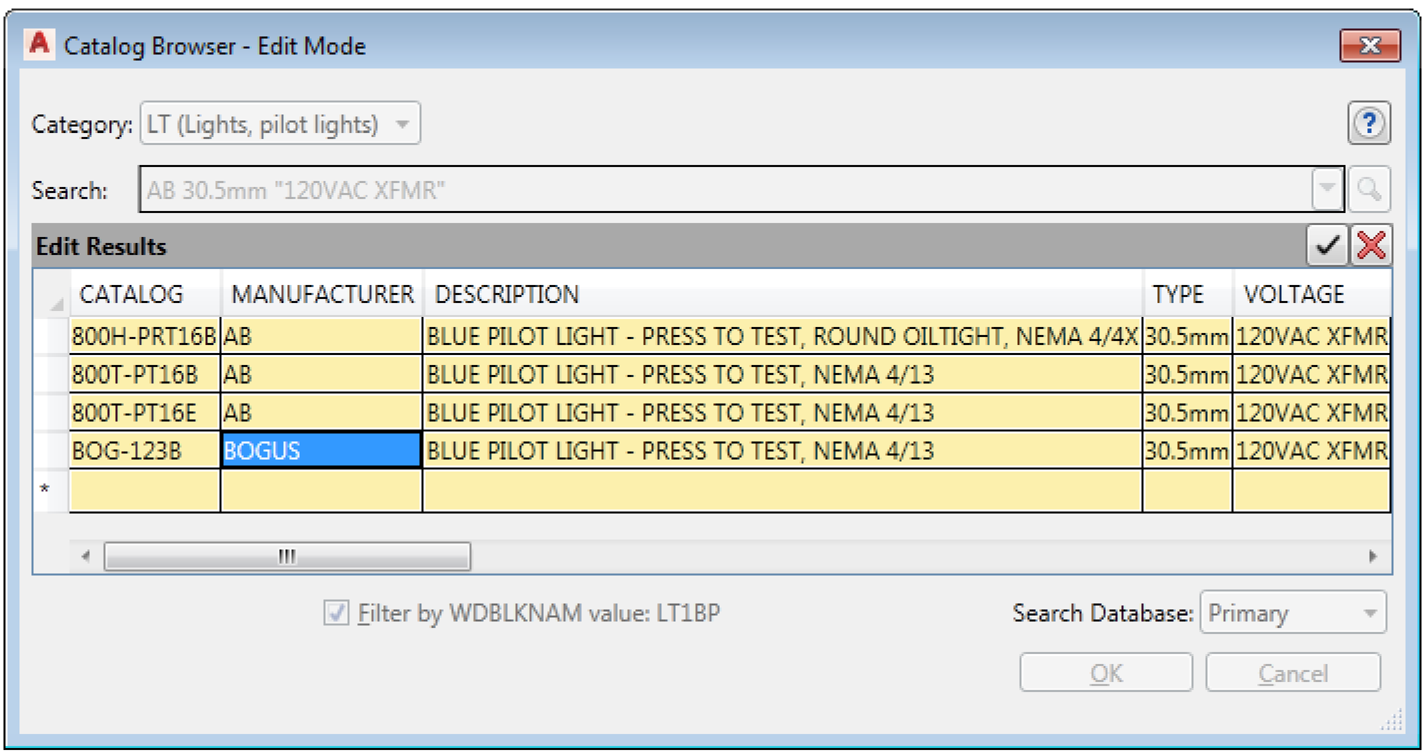


Add a catalog entry

1. In the Catalog Browser dialog box, click  to start the edit mode.

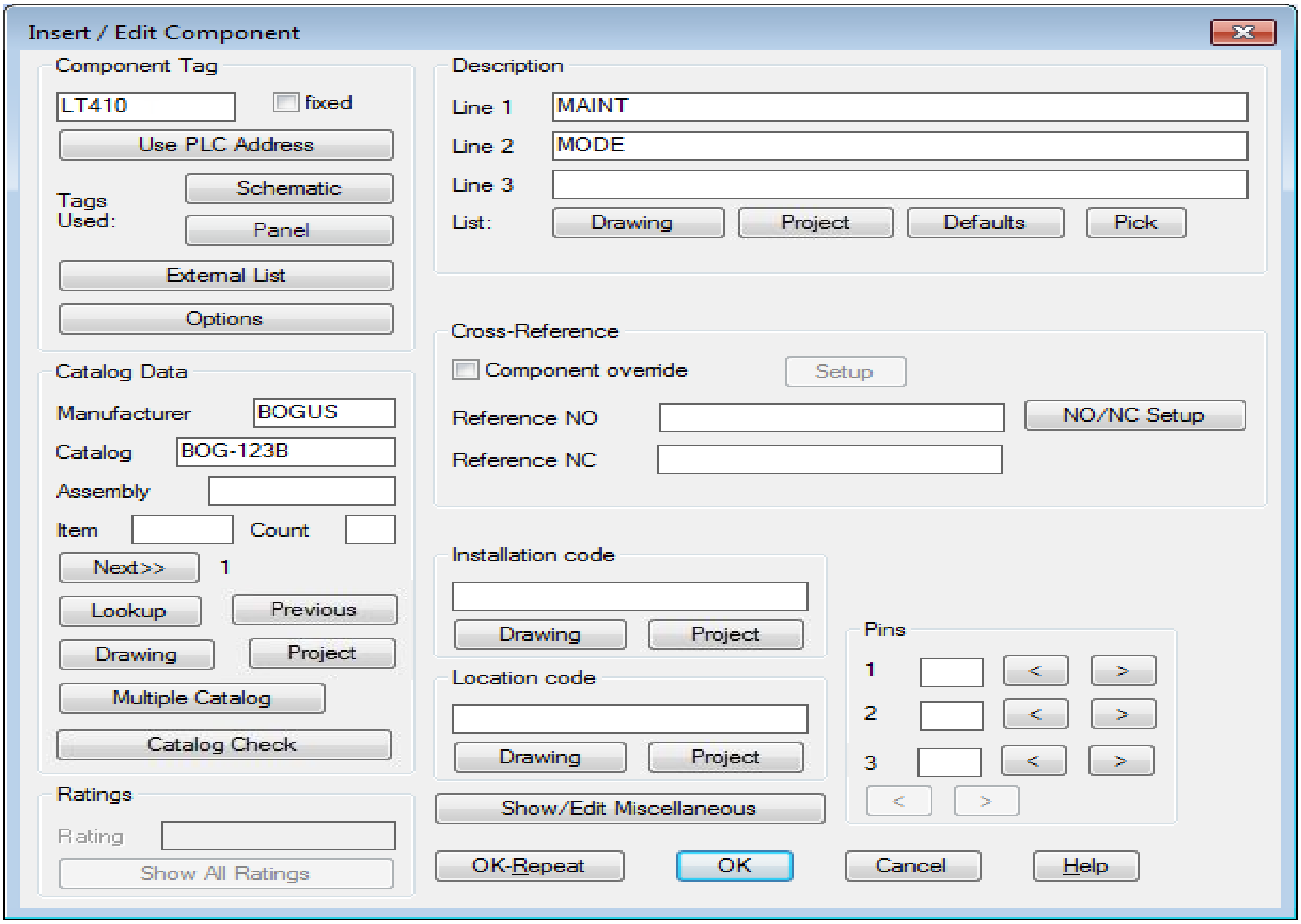


2. Right-click on the row for catalog 800T-PT16E and select Copy row.
3. Right-click in the blank row at the bottom and select Paste.
4. Click the Catalog cell and enter BOG-123B.
5. Click the Manufacturer cell and enter BOGUS.



6. Click  to save the changes.
7. Change the search to BOGUS 30.5mm "120VAC XFMR" and click .

8. Select the BOG-123B catalog entry and click OK.
9. In the Insert/Edit Component dialog box, click OK.

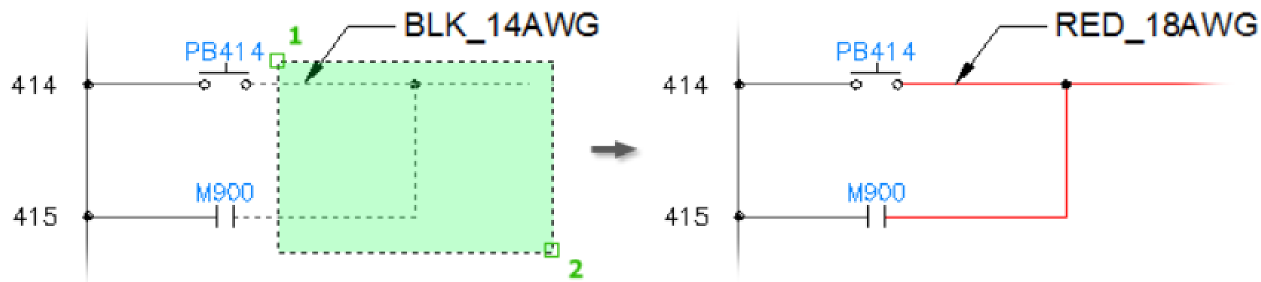


Wire Layers Tutorial

Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Wire layers to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

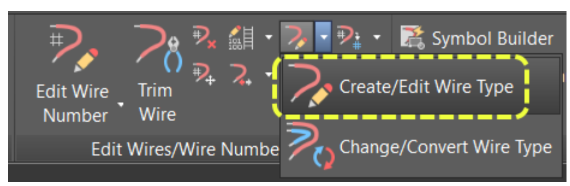
Follow the workflow topics listed below to accomplish these tasks:

- Create wire layers
- Change wire layer assignments



Creating a Wire Layer

Define a new wire layer and assign **size**, **color**, and **wire numbering** properties.

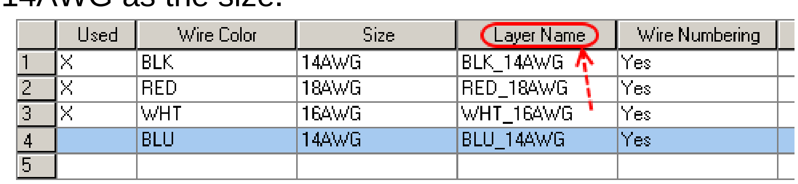


Create wire layer

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS04.dwg.
4. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Modify Wire Type drop-down ➤ Create/Edit Wire Type.

   The Create/Edit Wire Type dialog box lists all the valid wire layers that are defined for the active drawing. The **wire layer name** and the wire properties like color, size, and user-defined properties are listed in the grid.

5. Click inside the Wire Color column for a blank row and enter BLU as the wire color.
6. Click inside the Size column and enter 14AWG as the size.



	Used	Wire Color	Size	Layer Name	Wire Numbering
1	X	BLK	14AWG	BLK_14AWG	Yes
2	X	RED	18AWG	RED_18AWG	Yes
3	X	WHT	16AWG	WHT_16AWG	Yes
4		BLU	14AWG	BLU_14AWG	Yes
5					

The **Layer Name** is automatically created.

7. Click Color in the Layer section. Select blue and click OK.

   Note: If you want the new wire layer to be the default, click Mark Selected as Default.

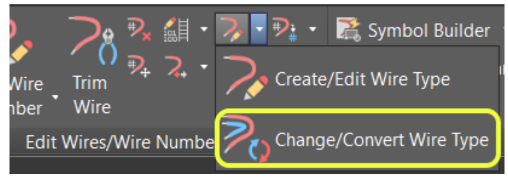
8. Click OK.

Changing a Wire Layer Assignment

Change wires to a different wire layer.

When a wire is inserted, the wire ends up on the first valid wire layer as defined in the Drawing Properties dialog box. You can place wires on different wire layers. You can use the **AutoCAD PROPERTIES command** to move a wire to the correct layer or you can use the **Wire Layer utility**.

Change wire layer assignments



1. Zoom in on the upper left corner of the drawing.
2. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Modify Wire Type drop-down ➤ Change/Convert Wire Type.

The **Change/Convert Wire Type** dialog box lists all the valid wire layers that are defined for the active drawing. The wire layer name and the wire properties like color, size, and user-defined properties are listed in the grid. An "X" in the Used column indicates the layer name is currently being used.

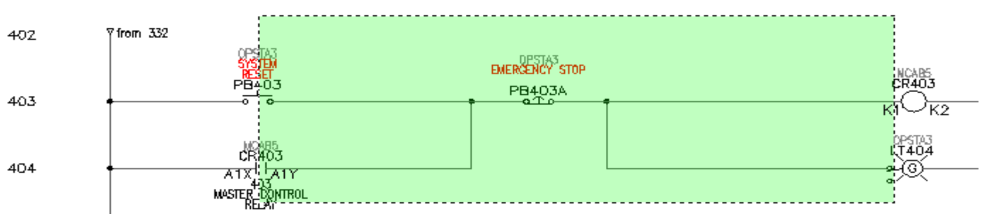
3. Select RED_18AWG.

The wire type highlights in blue in the dialog box indicating that it is the wire type to change.

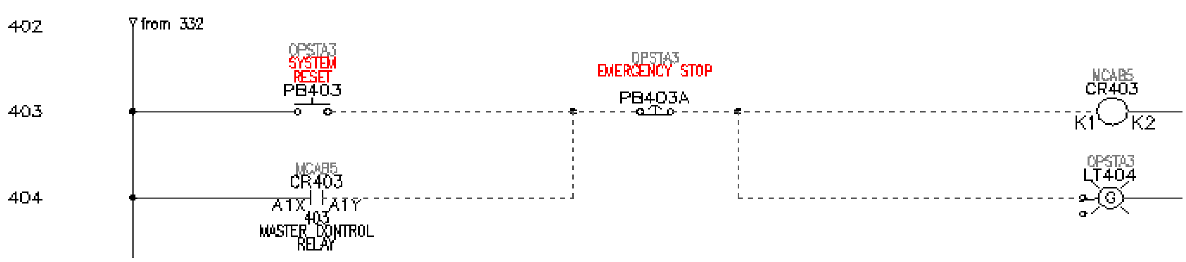
	Used	Wire Color	Size	Layer Name	Wire Numbering	USER1	USER2
1	X	BLK	14AWG	BLK_14AWG	Yes		
2	X	RED	18AWG	RED_18AWG	Yes		
3	X	WHT	16AWG	WHT_16AWG	Yes		
4							

4. Click OK and respond to the prompts as follows:

Select Objects: *Window from left to right around the wires as shown and press* ENTER



Before you press ENTER, the wires display as dashed lines to indicate that they have been selected. Once you press ENTER the lines display in red indicating that they have been moved to the RED_18AWG wire layer.



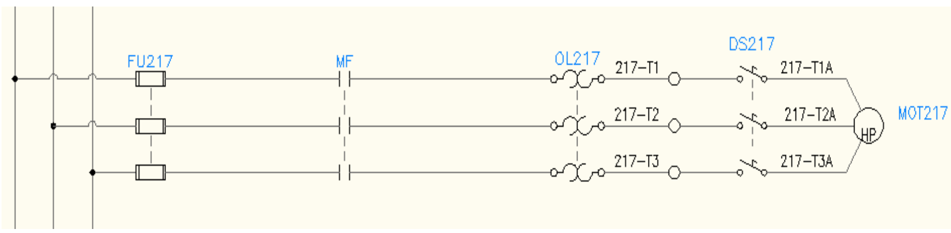
Circuits Tutorial

Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Circuits to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Create circuits with Circuit Builder. Save and insert a saved circuit.

Follow the workflow topics listed below to accomplish these tasks:

- Move a circuit
- Insert a circuit using Circuit Builder
- Save and insert a saved circuit
- Insert a saved circuit using WBLOCK



Move an Existing Circuit

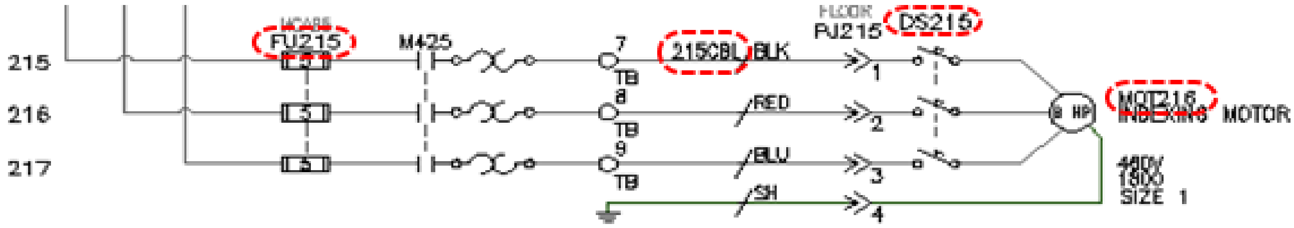
Move a circuit to a new location and update all related components.

When you move a circuit, most of the parent components contained in the circuit automatically retag since the drawing is set up for reference-based component tagging. In the process of moving the circuit, you change the reference locations of the moved components. Related **child** components update to match the new **parent** tags, including references on other drawings in the project.

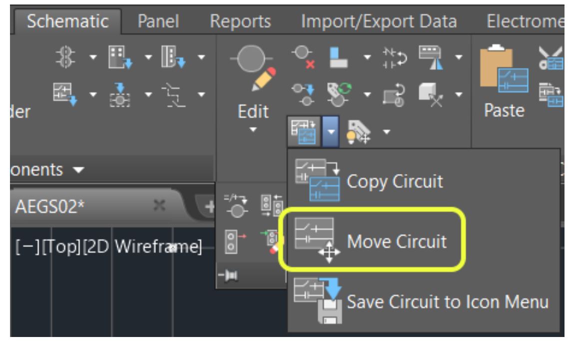
Note: Tagging updates vary depending on your default tagging configurations.

Move the location of a circuit

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS02.dwg.
4. Zoom in on the lower left corner of the drawing. Make sure the 3-phase motor circuit at line reference 215 is visible.



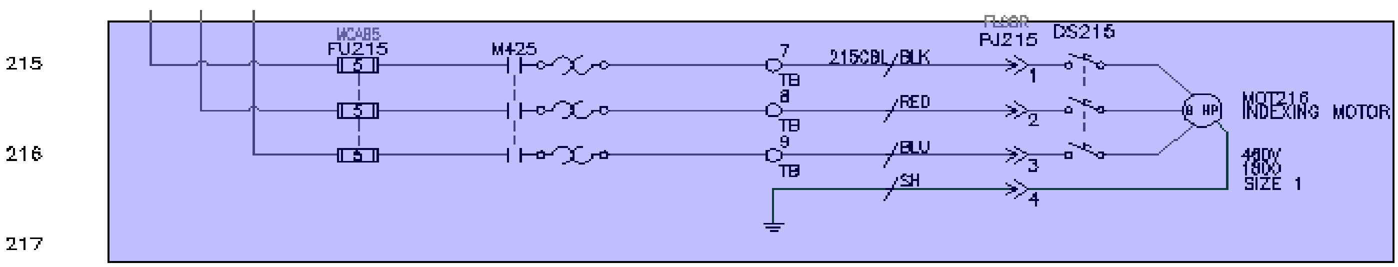
This circuit has component tags



- "FU215" on the 3-pole fuse
- "215CBL" on the multi-conductor cable
- "DS215" on the disconnect switch
- "MOT216" on the motor

5. Click Schematic tab ➤ Edit Components panel ➤ Circuit drop-down ➤ **Move Circuit**.
6. Respond to the prompts as follows:

Select Objects: *Window select the circuit on line reference 215 to capture the connection wire and dots that tie in to the vertical bus, right-click*

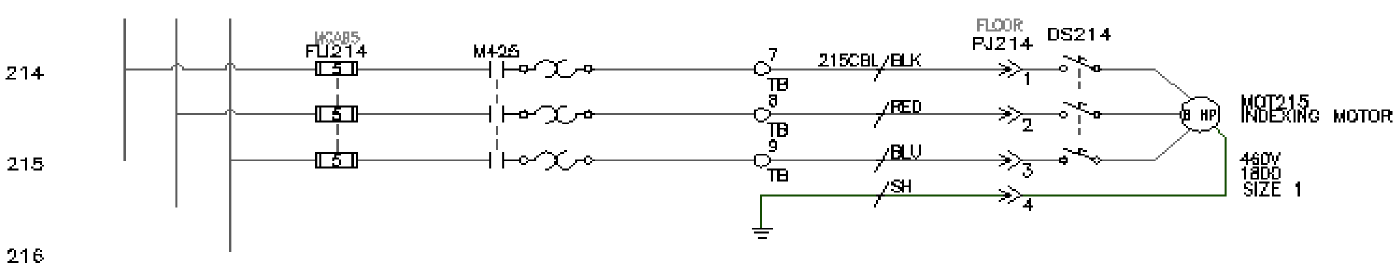


Press F9 to turn on **SNAP**.

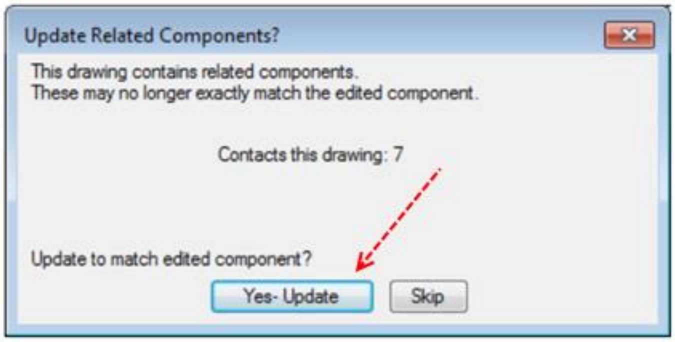
Specify base point or displacement:

Select a base point and then select a point on line reference 214

The circuitry is **moved**, the affected components are **retagged**, and cross-references are **updated** based on the new line reference. Each of the listed parent component tags decrement by one. For example, fuse FU215 became FU214.



7. In the Update Related Components dialog box, click Yes-Update.

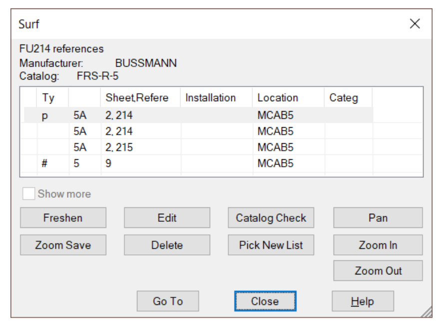


Related child references on the active drawing update to match the newly retagged parent components.

8. In the Update other drawings dialog box, click OK.

   Related child components and panel layout references on other drawings update to match the parent components on the moved circuit.

9. If asked to save the drawing, click OK.
10. Click Project tab ➤ Other Tools panel ➤ Surfer drop-down ➤ Surfer.
11. Select **FU214** on the drawing.



   The Surf dialog box displays three references on sheet 2 and one reference on sheet 9.

12. Double-click the reference on Sheet 9.

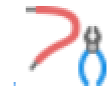
   Surfer goes to the panel layout drawing and zooms in on the physical representation of this 3-pole fuse. Notice that the physical representation of the fuse block tag updated because the circuit was moved.

13. Double-click the first entry in the dialog box to return to the original AEGS02.dwg drawing.
14. Click Close.

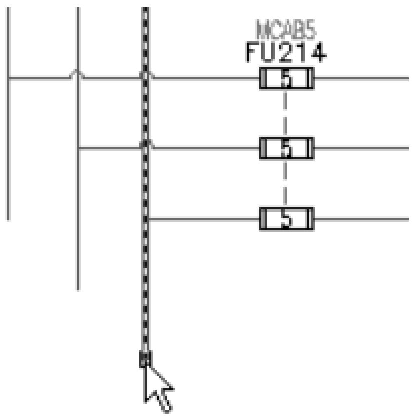
   Moving the motor circuit up one line reference spacing opened up a bit more room to add a new circuit below it. The next step is to extend the 3-phase bus down to line reference 218 and over to the right to begin building a new motor circuit.

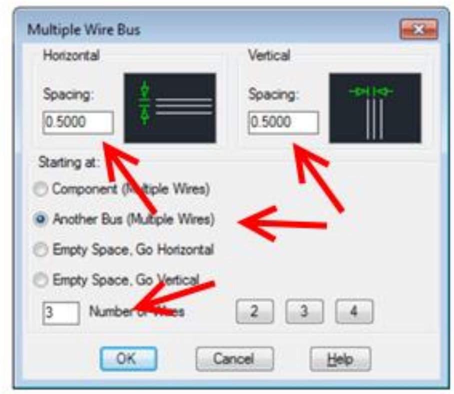
Extending the 3-phase bus

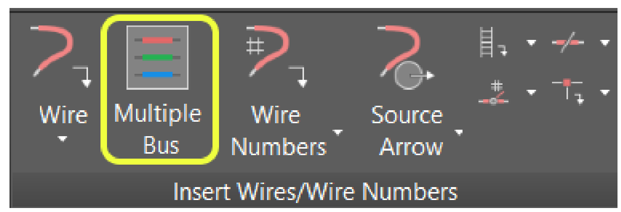
1. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Trim Wire.
2. Respond to the prompts as follows:



Fence/Crossing/Zext/<Select wire to TRIM>: *Click the bottom ends of the three dangling wires, right-click*







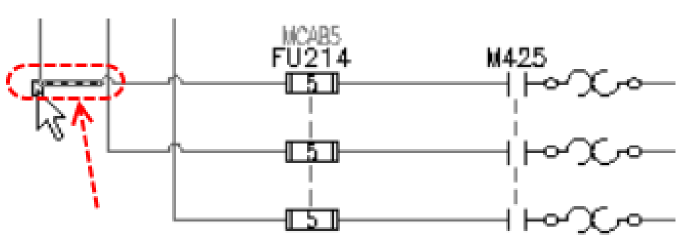
You can insert vertical or horizontal 3-phase wiring.
Three-phase wiring automatically breaks and reconnects to any underlying components that it finds in its path. If it crosses any existing wiring, wire-crossing gaps are inserted.

3. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ **Multiple Bus**.
4. In the Multiple Wire Bus dialog box, select:

Horizontal Spacing: 0.5	Vertical Spacing: 0.5	Starting at: Another Bus (Multiple Wires)	Number of Wires: 3

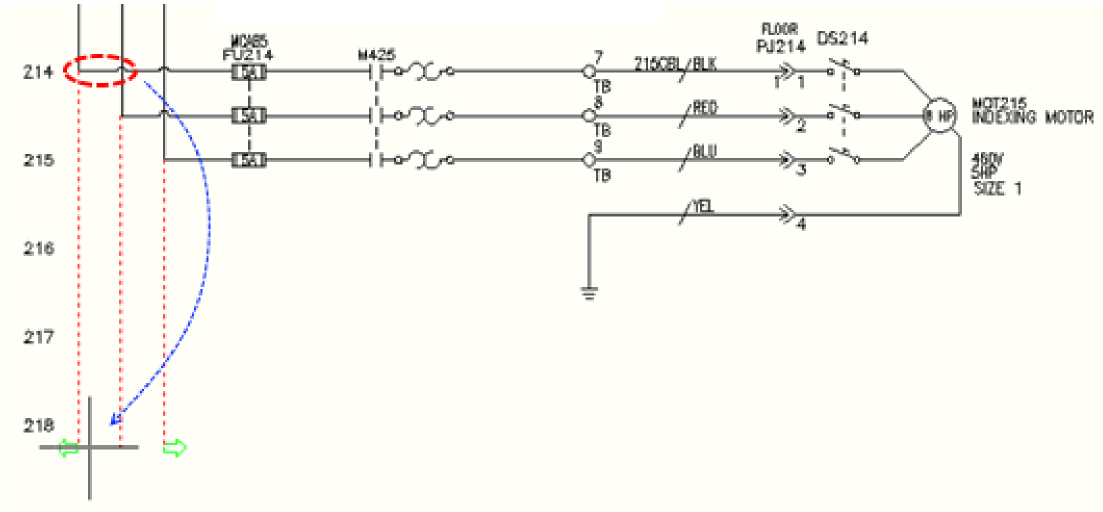
5. Click OK.
6. Respond to the prompts as follows:

Select existing wire to begin multi-phase bus connection: *Select the bottom corner of the left-most vertical bus on line reference 214 as shown*



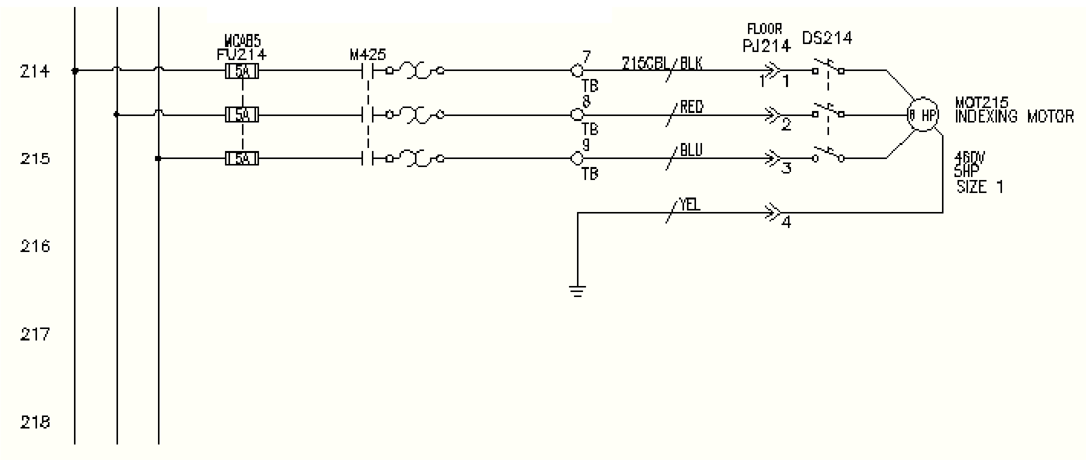
Select existing wire to begin multi-phase bus connection: to *Pull the cursor down to line reference 218*

Temporary graphics show the proposed routing of the extended bus.



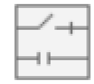
7. Click to create the wires.
8. Right-click to exit the command.

The 3-phase bus and wire connection dot symbols are inserted on the drawing. (Bus wires are extended, wire dots are added).



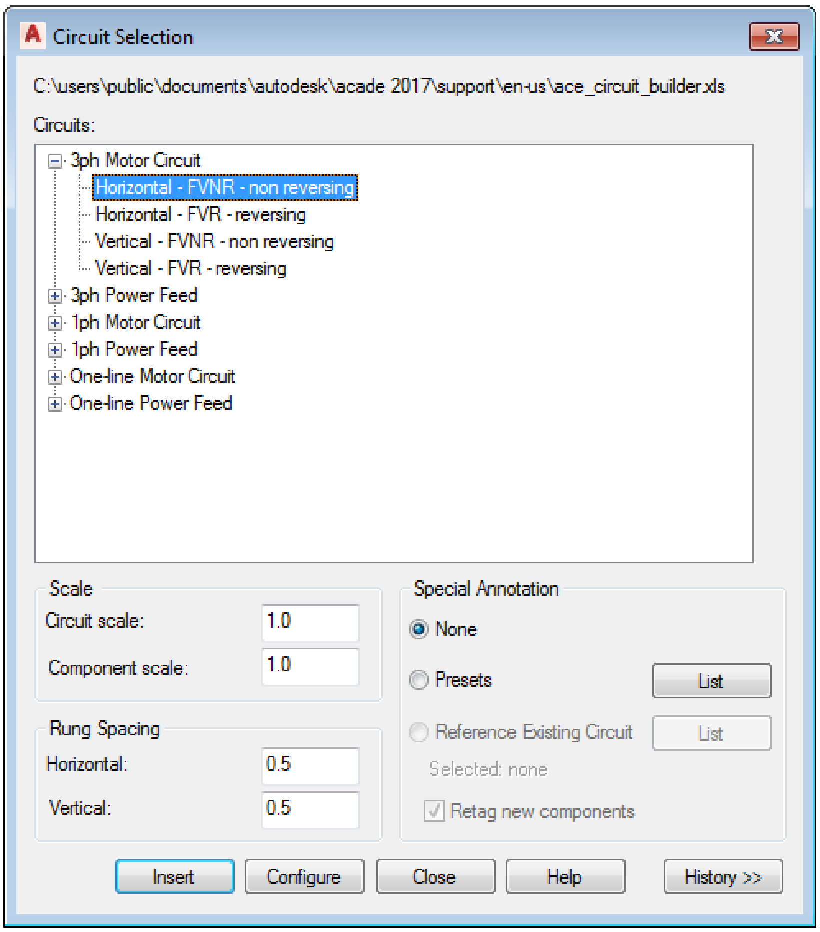
Insert and Configure a Circuit

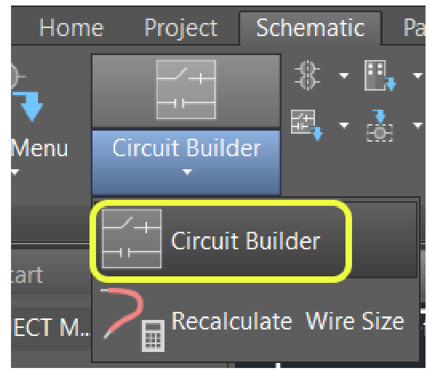
Use **Circuit Builder** to insert and configure a 3-phase motor circuit.



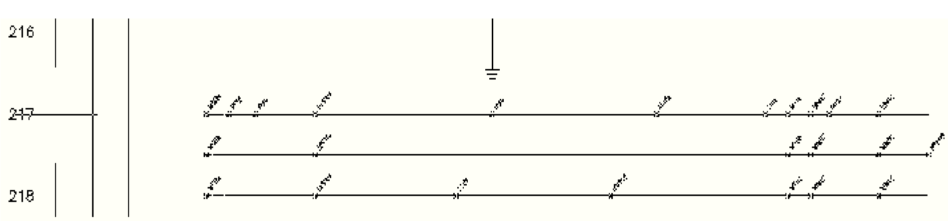
Insert and configure the circuit

1. Click Schematic tab ➤ Insert Components panel ➤ Circuit Builder drop-down ➤ Circuit Builder.
2. The **Circuit Selection** dialog box displays.





3. Expand 3ph Motor Circuit.
4. Select **Horizontal - FVNR - non reversing**.
5. Change the Rung Spacing: Horizontal to **0.5**.
6. Select **Configure**.
7. Specify insertion point at rung 217.



Circuit Configuration

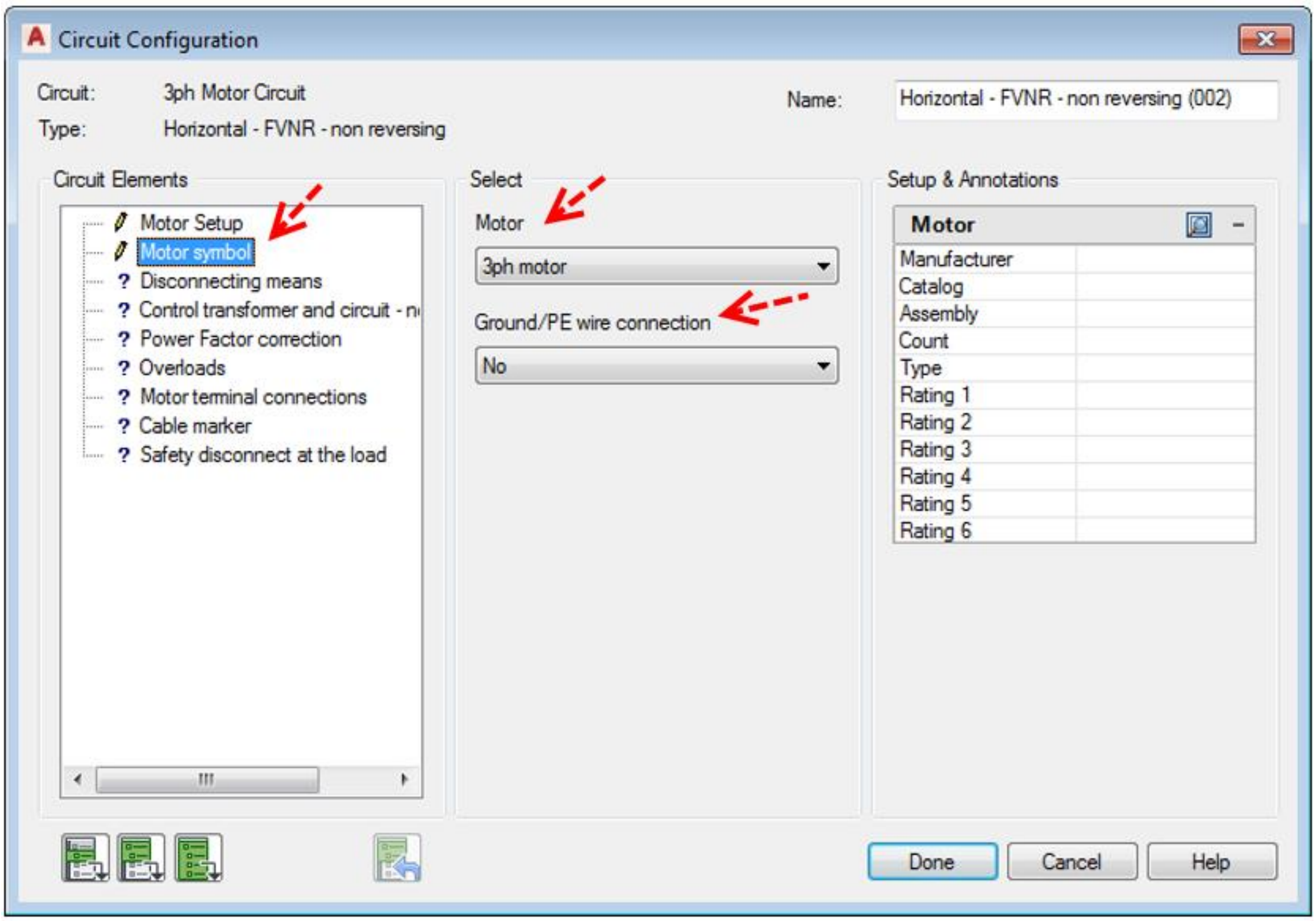
Circuit Builder inserts a template drawing. This template contains the base wiring for the circuit and strategically positioned “marker blocks”.

The “**marker blocks**” control what circuit elements are presented in the Circuit Configuration dialog box. For example, a “marker block” indicates the need for a Disconnecting Means in the circuit. Various options for the Disconnecting Means are presented in the dialog box. The option selected for this circuit element is inserted at the location of the “marker block”. Circuit Builder dynamically builds the complete circuit based on the selections you make on this dialog box.

1. In the Circuit Elements section, select **Motor symbol**.

   In the Select section, select Motor: **3ph motor**,

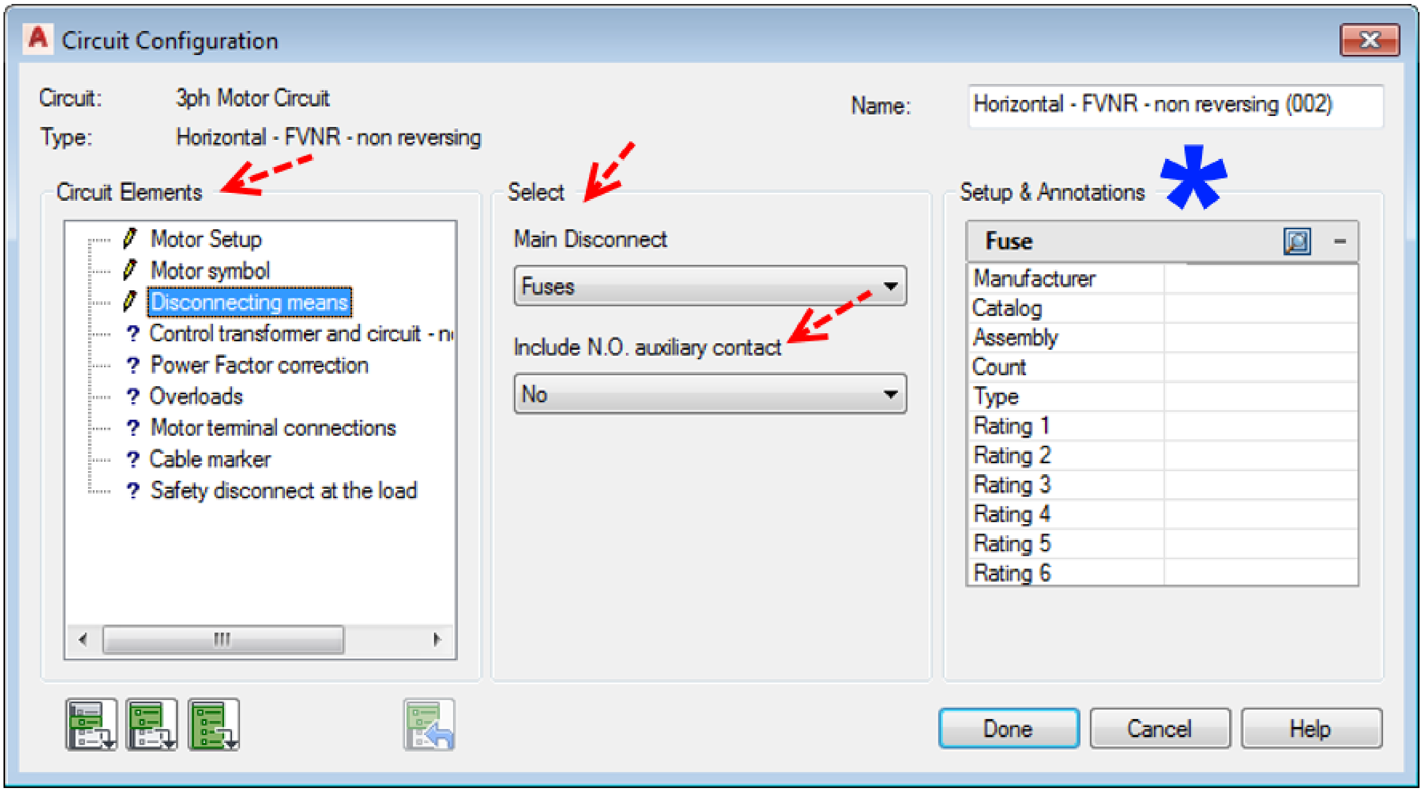
   Ground/PE wire connection: **No**.



2. In the Circuit Elements section, select **Disconnecting Means**.

   In the Select section, select Main Disconnect: **Fuses**,

   Include N.O. auxiliary contact: **No**.

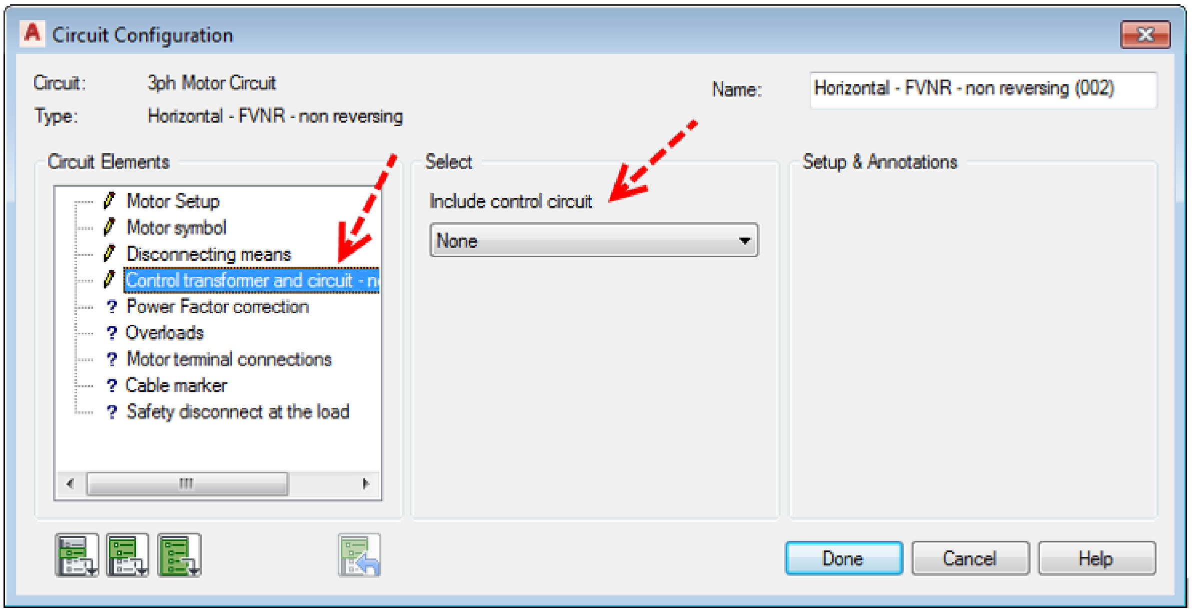


Setup & Annotation section: The options within this section change according to your selections in the Circuit Elements and Select sections. Type in values or select the Browse button to access a lookup table.

Select an entry from the lookup table to obtain values for the individual settings. If the circuit option is a component, the catalog lookup opens.

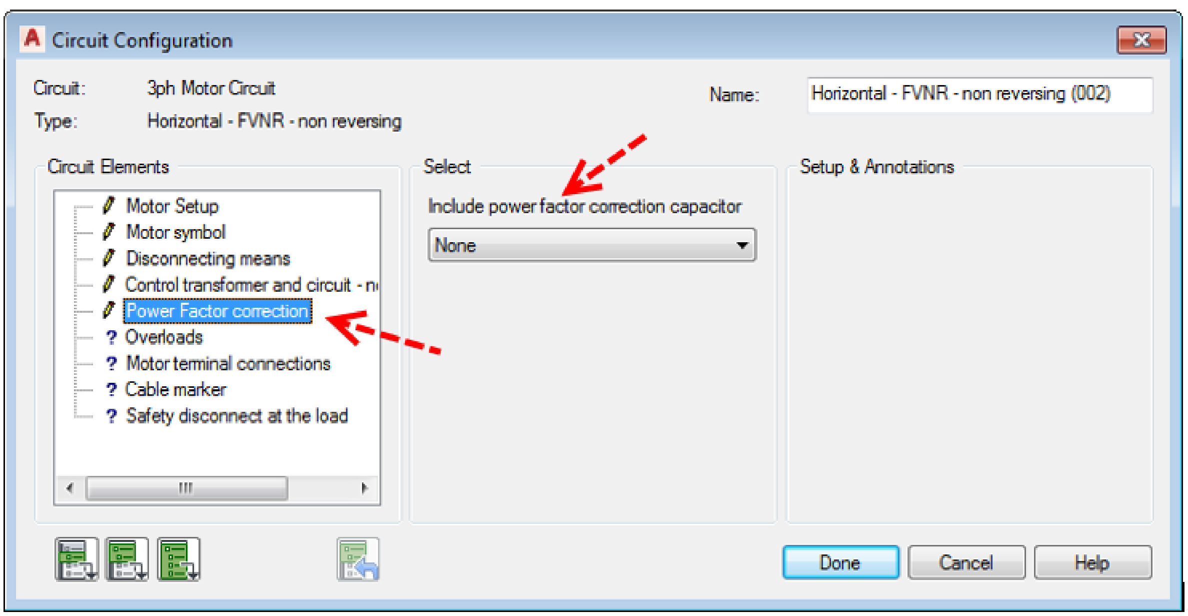
3. In the Circuit Elements section, select **Control transformer and circuit - non-reversing**.

   In the Select section, select Include control circuit: **None**.

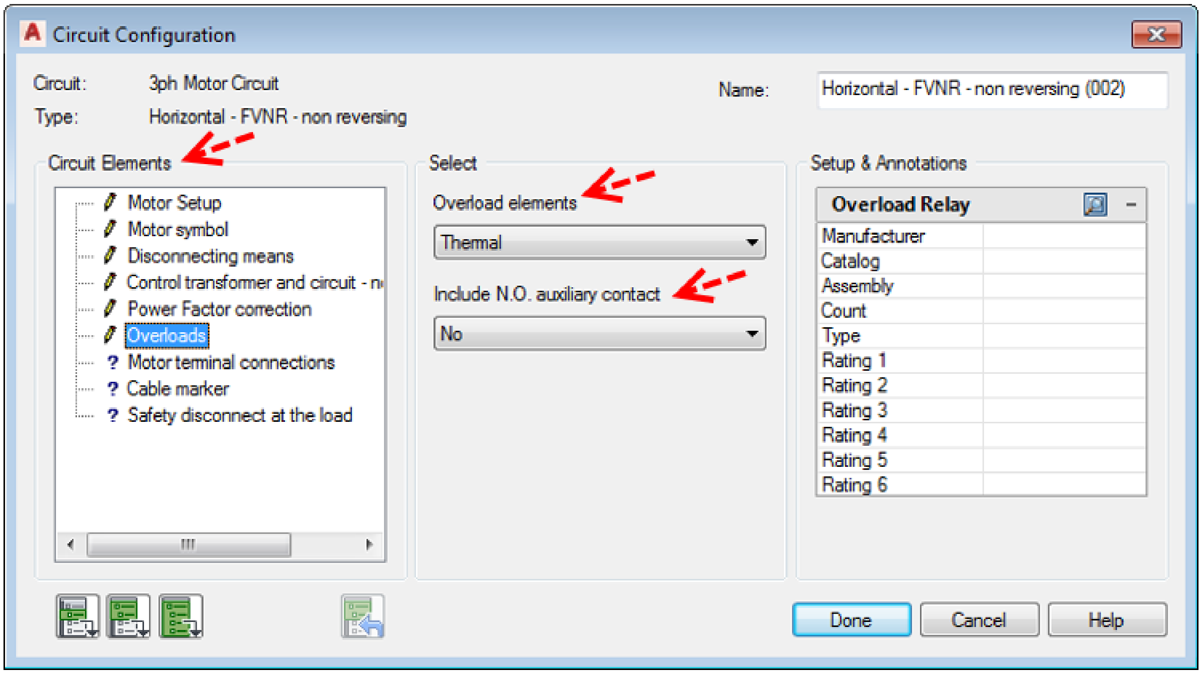


4. In the Circuit Elements section, select **Power Factor correction**.

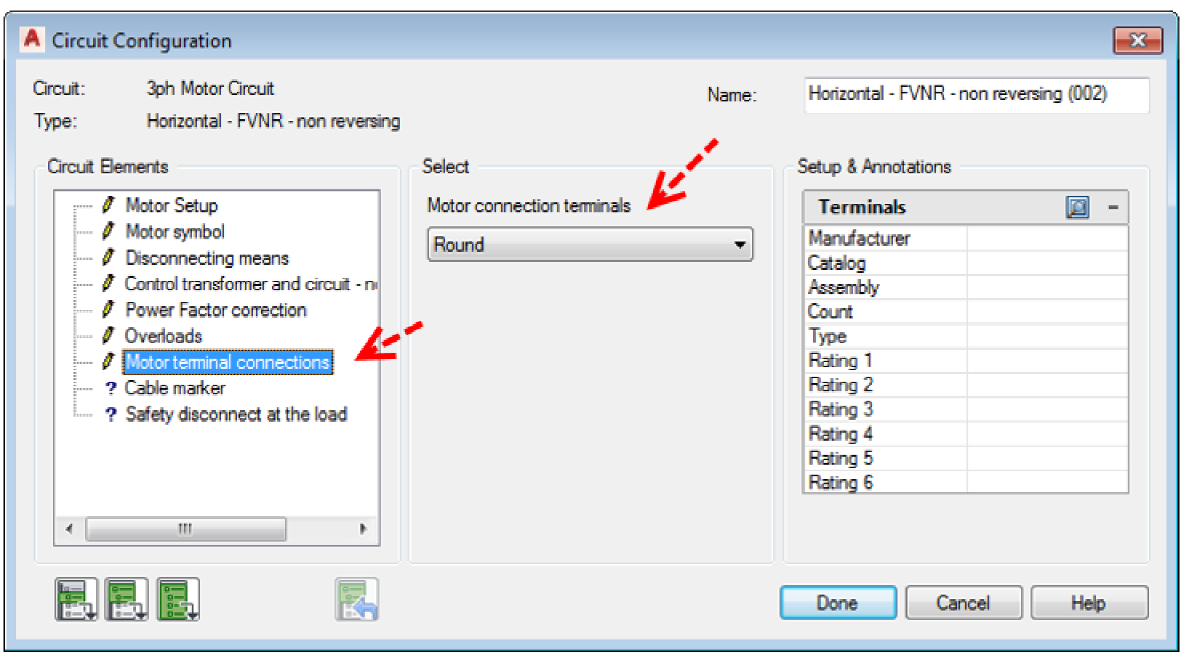
   In the Select section, select Include power factor correction capacitor: **None**.



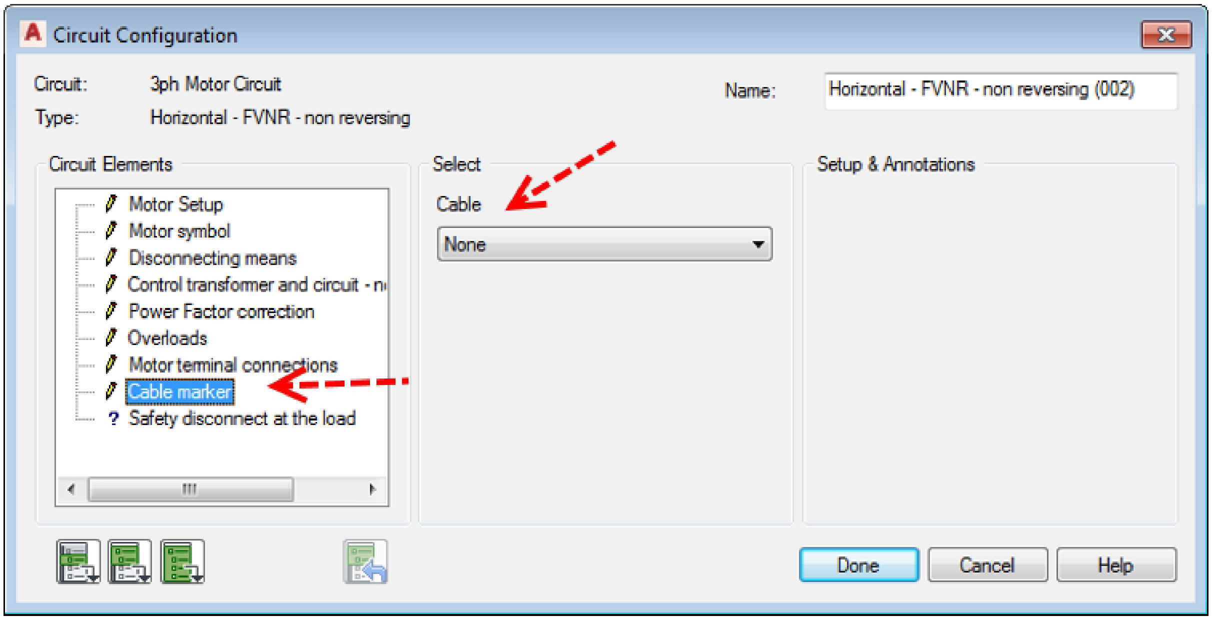
5. In the Circuit Elements section, select **Overloads**.
   In the Select section, select Overload elements: **Thermal**,
   Include N.O. auxiliary contact: **No**.



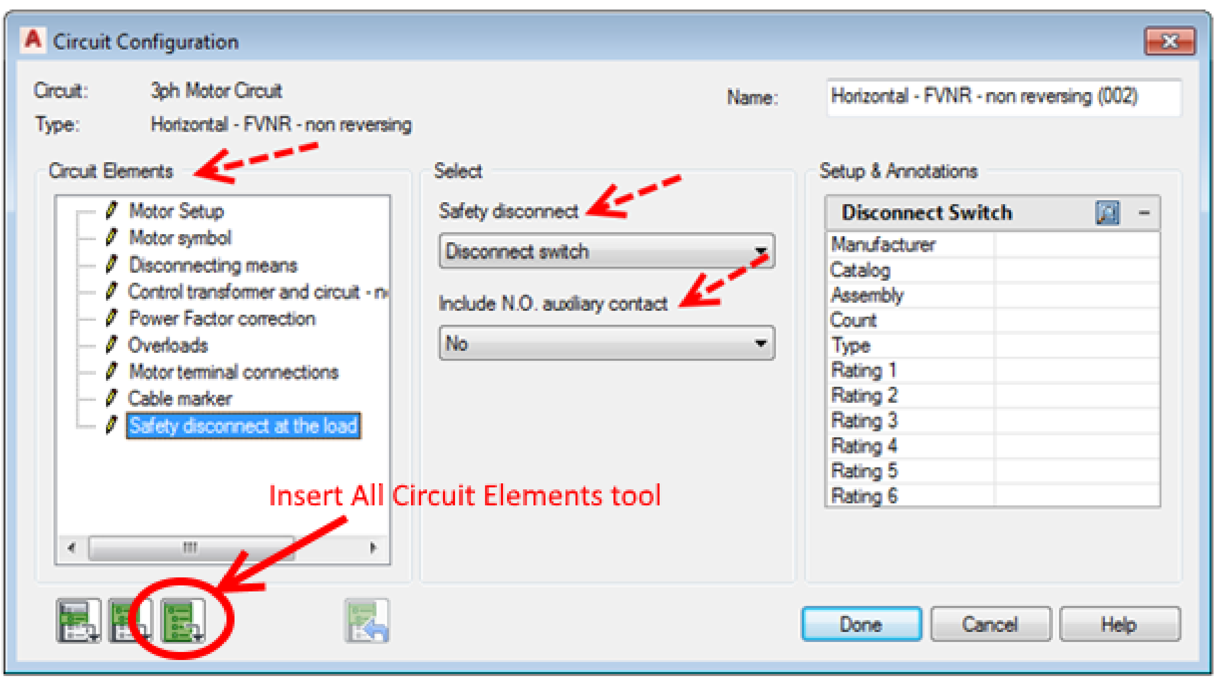
6. In the Circuit Elements section, select **Motor terminal connections**.
   In the Select section, select Motor connection terminals: **Round**.



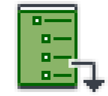
7. In the Circuit Elements section, select **Cable marker**.
   In the Select section, select Cable: **None**.

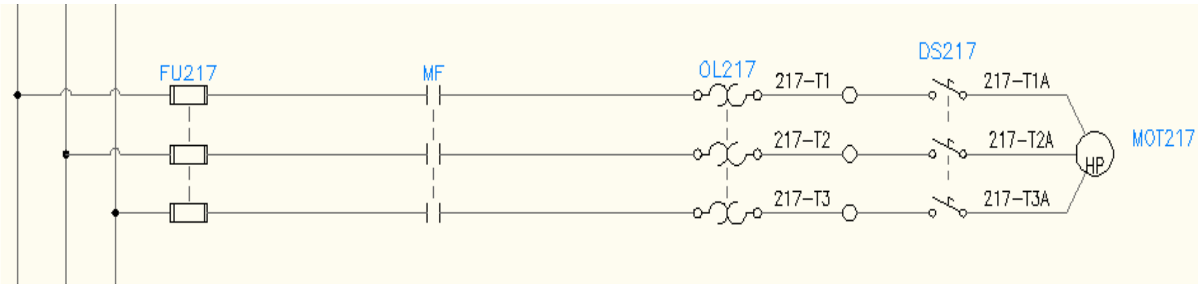


8. In the Circuit Elements section, select **Safety disconnect at the load**.
   In the Select section, select Safety disconnect: **Disconnect switch**
   Include N.O. auxiliary contact: **No**.



9. Select the **Insert All Circuit Elements** tool. Circuit Builder inserts each of the selected circuit elements.

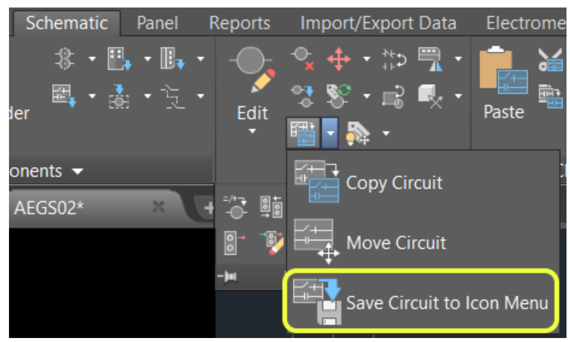




10. Select Done.

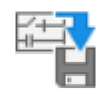
Note: See the Circuit Builder topics later in this section for more examples.

Save and Insert a Circuit



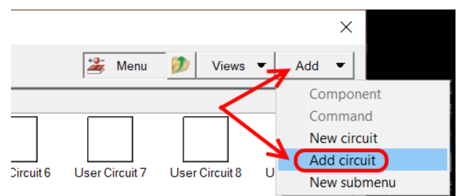
Save existing circuits to the icon menu **for use in the future**.

AutoCAD Electrical makes saving and inserting pre-drawn circuits easy and convenient. You can save and insert from a user circuits page on the Insert Component icon menu. You can also use the normal AutoCAD **WBlock** command to save selected circuitry to disk.



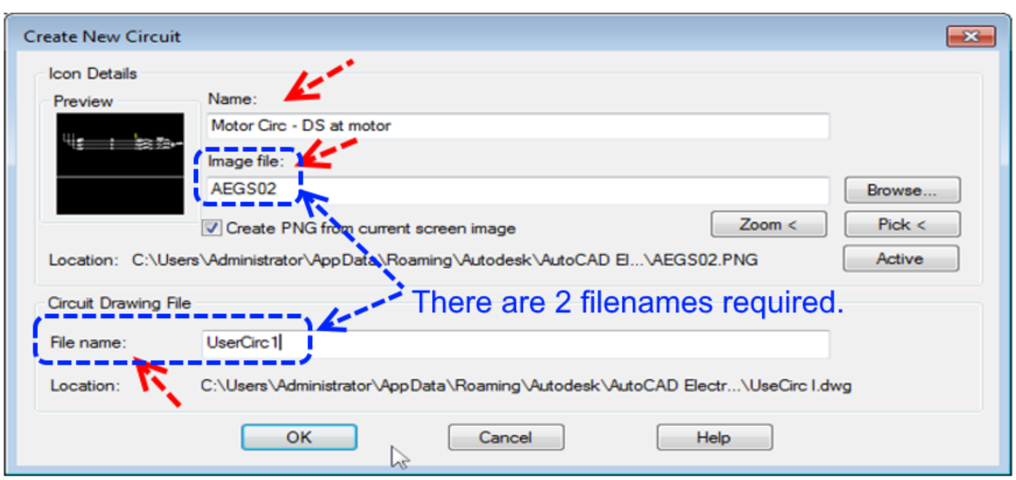
Use the **Insert Circuit** command to insert WBlocked circuits into the active drawing.

Save your circuit for use in the future



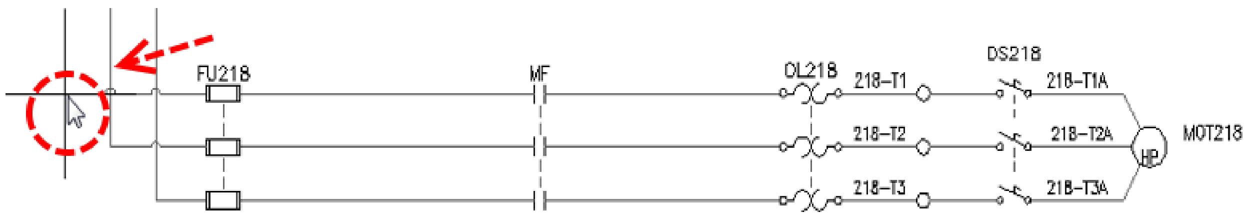
1. Zoom around the circuit so that it fills your screen .
2. Click Schematic ➤ Edit Components panel ➤ Circuit drop-down ➤ **Save Circuit To Icon** Menu.
3. In the dialog box, click Add ➤ **New** circuit. [**NEW** CIRCUIT, NOT ADD]
4. On the **Create New Circuit** dialog box, specify:

Name: Motor Circ - DS at motor	**Image file**: Click Active and check Create PNG from current screen image	**Filename**: UserCirc1

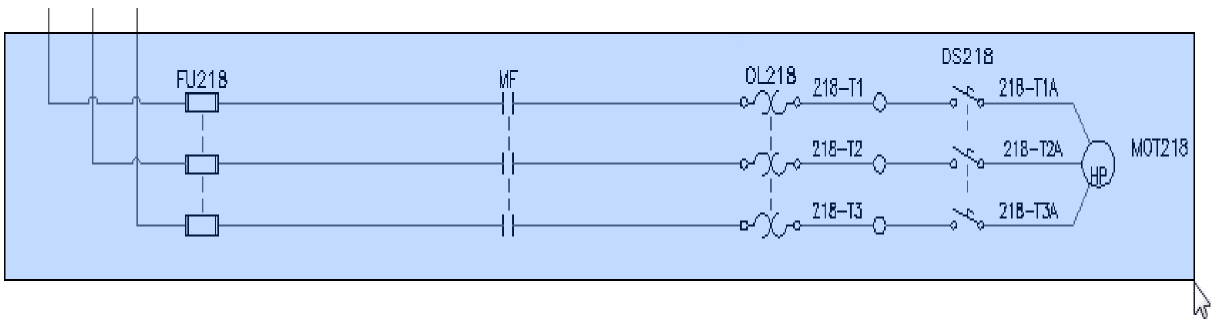


5. Click OK.
6. Respond to the prompts as follows:

   Base point: *Select the left-most wire connection point where the circuit ties into the left-hand vertical bus wire*



   Select objects: *Window around the circuit from left to right to capture all the components and wiring,* ***but exclude the vertical bus****, press* ENTER

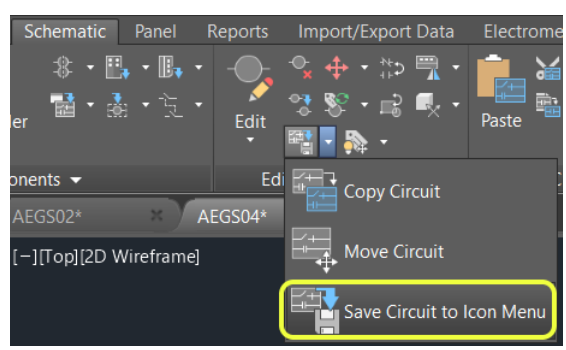


7. On the **Save Circuit to Icon** Menu dialog box, click OK.

   The circuit is saved to your AutoCAD Electrical toolset user folder. It can be quickly accessed from the Insert Component icon menu or from the Insert Saved Circuit tool.

The new motor has a 3-pole motor contactor child reference but there is not a parent motor starter relay coil to operate it. The **motor start coil circuit must be added** on a control schematic in the project drawing set and linked back to the new motor circuit.

Insert motor start coil circuit to control schematic

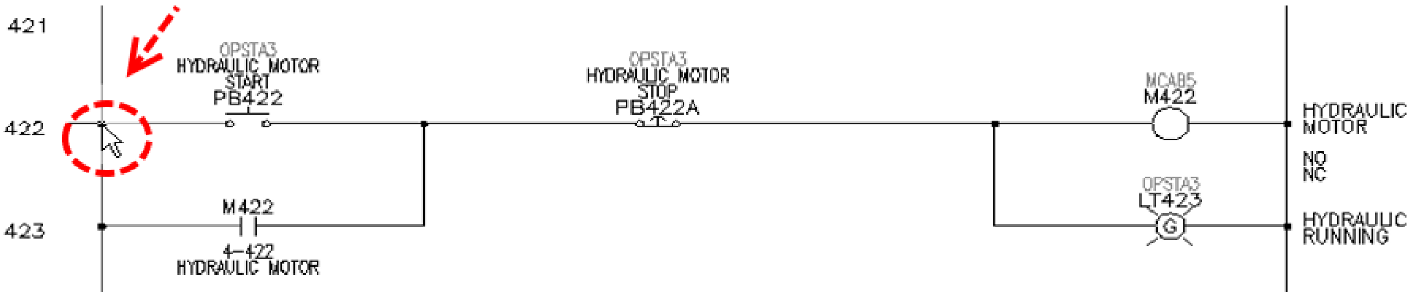


1. Open AEGS04.dwg.
2. Zoom on the upper-right hand ladder column so the full circuit on line reference 422-423 displays.
3. Click Schematic tab ➤ Edit Components ➤ Circuit drop-down ➤ **Save Circuit To Icon Menu**.
4. On the Save Circuit to Icon Menu dialog box, click Add ➤ New circuit.
5. On the Create New Circuit dialog box, specify:

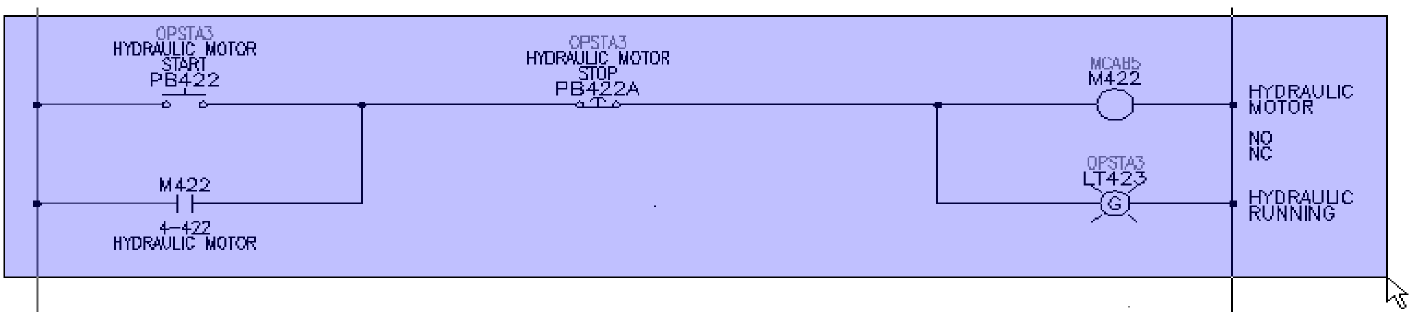
Name: Motor starter circ	Image file: Active and Create PNG from current screen image	File name: UserCirc2

   Click OK.

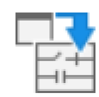
6. Respond to the prompts as follows: Base point: *Select the left-most wire connection point at line reference 422*



Select objects: *Window around the circuit from left to right to capture all the components and wiring, but exclude the vertical bus, press* ENTER



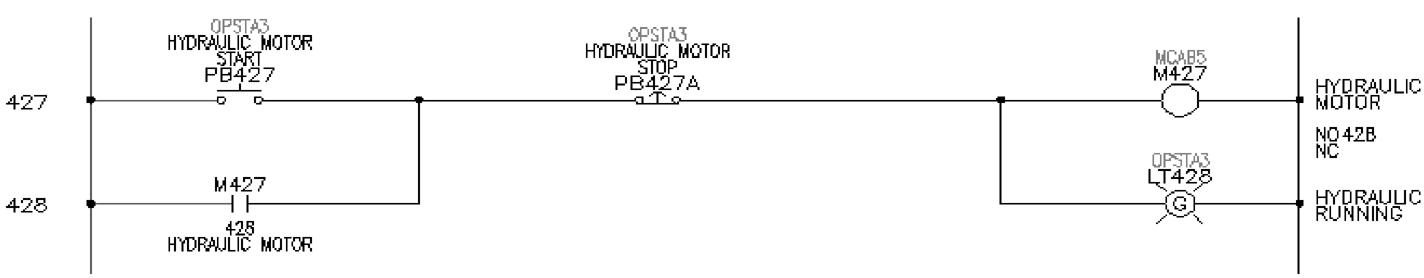
7. On the **Save Circuit to Icon Menu** dialog box, click OK.



Insert a circuit you saved for reuse

1. Pan to display the blank area between line references 426 - 432.
2. Click Schematic tab ➤ Insert Components panel ➤ Circuit drop-down ➤ Insert Saved Circuit.
3. In the JIC: Saved User Circuits dialog box, select the Motor starter circ button.
4. In the Circuit Scale dialog box, click OK.
5. Respond to the prompts as follows:

Specify insertion point: *Place the circuit insertion point on the vertical bus wire at line reference 427, left-click to insert the circuit.*



The circuit inserts and updates. Tags automatically update to reflect the new line reference number, and parent/child relationships defined inside of the circuit update accordingly.

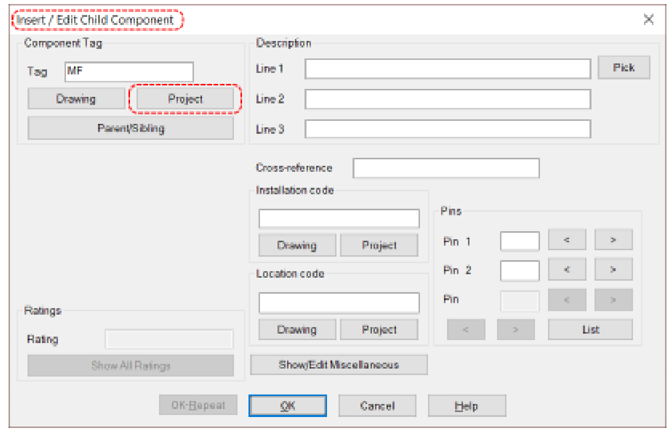
6. Right-click the M427 coil symbol and select **Edit Component**.
7. In the Insert/Edit Component dialog box, specify: **Description Line 2: MOTOR NO. 2**
Click OK.

8. In the Update Related Components dialog box, click Yes-Update.

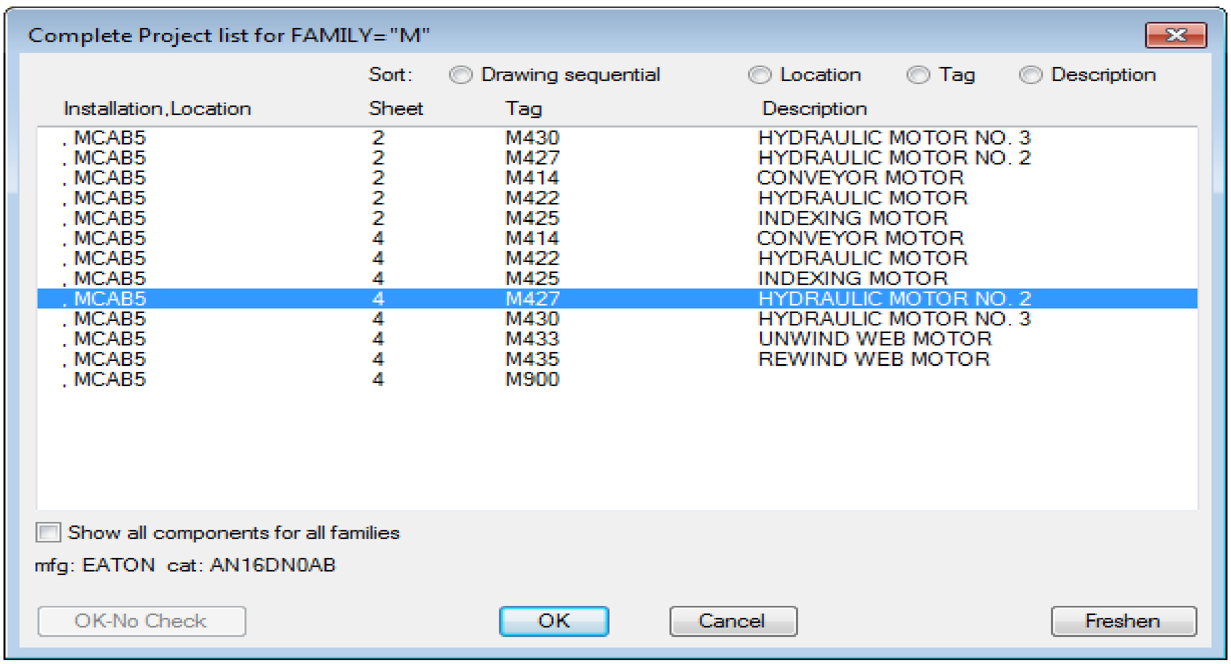
Linking the parent coil to the child contactor

1. Open AEGS02.dwg and zoom on the untagged 3-pole motor contact/overloads on line reference 217.
2. Right-click the “M” contact and select Edit Component.

   The Insert/Edit Child Component displays.



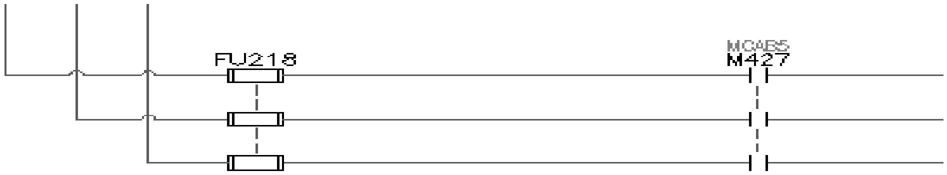
3. In the Insert/Edit Child Component dialog box, Component Tag section, click Project.
4. In the Complete Project list for Family=”M” dialog box, select M427 HYDRAULIC MOTOR NO. 2 and click OK.



The tag M427 is now displayed in the Component Tag edit box. Notice that the description, cross-reference, and location code boxes have also updated.

5. In the Insert/Edit Child Component dialog box, click OK.
6. In the Update linked components dialog box, click OK.

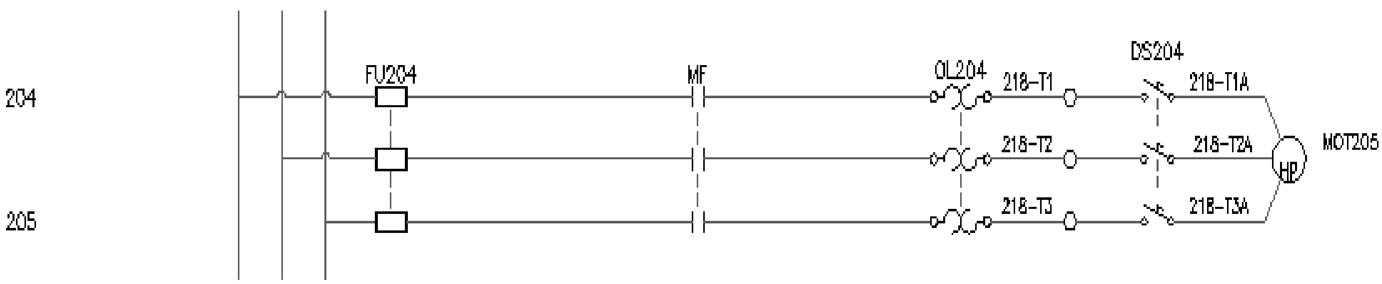
   The components are now linked. If you go back to drawing AEGS04.dwg and look at the motor starter coil, it shows references to these three child contacts (plus one seal contact around PB427).



Using the icon menu to add a motor

1. Reopen drawing AEGS04.dwg and zoom to the blank area at line references 430-431.
2. Repeat the steps for inserting the saved Motor starter circ circuit.
3. In the Circuit Scale dialog box, click OK.
4. Insert the circuit at line reference 430.
5. Right-click the M430 coil symbol, and select Edit Component.
6. In the Insert/Edit Component dialog box, specify: **Description Line 2: MOTOR NO. 3** Click OK.
7. In the Update related components dialog box, click Yes-Update.
8. Open drawing AEGS02.dwg and zoom to the blank area at line references 204-206.
9. Repeat the steps for inserting a saved circuit, but this time insert the Motor Circ - DS at motor circuit.
10. In the Circuit Scale dialog box, click OK.
11. Respond to the prompts as follows:

    Specify insertion point: *Position the motor circuit so that the insertion point lands on the left-hand vertical bus at line reference 204, left-click to insert the circuit.*

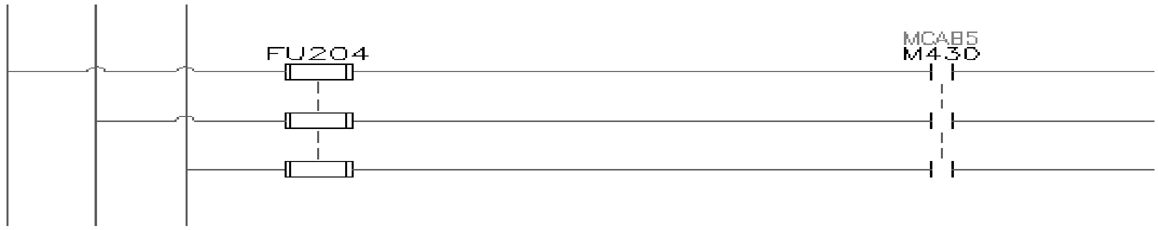
    

    Notice that the fuse, disconnect, and motor **automatically retag** based on their reference locations.

12. Right-click the M child motor contact symbol, and select Edit Component.
13. In the Insert/Edit Child Component dialog box, Component Tag section, click Project.
14. In the Complete Project list for Family="M" dialog box, select M430 HYDRAULIC MOTOR NO. 3 and click OK.

    The tag M430 is now displayed in the Component Tag edit box. Notice that the description, cross-reference, and location code boxes have also updated.

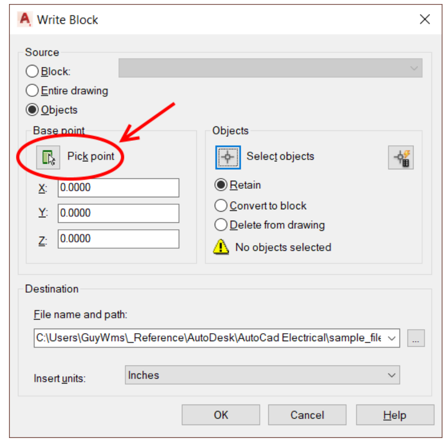
15. In the Insert/Edit Child Component dialog box, click OK.
16. In the Update linked components dialog box, click OK.


FU204
MCAB5
M43D

Insert a Saved Circuit Using WBlock

Reuse a circuit that has been saved as a DWG file.

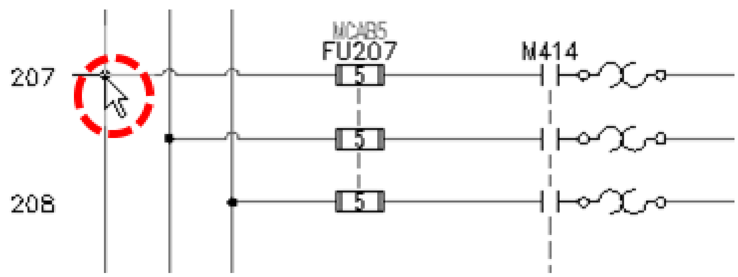
Another method for saving and inserting circuits is to use the AutoCAD **WBlock** command to save the circuit to disk. A separate **Insert Circuit** command is used to browse to a selected saved circuit and insert it into the active drawing. This method allows unlimited circuits to be constructed and saved to disk. They can be arranged into a set of shared subfolders for easy browsing and retrieval using the Insert Circuit command.



Saving a circuit using WBlock

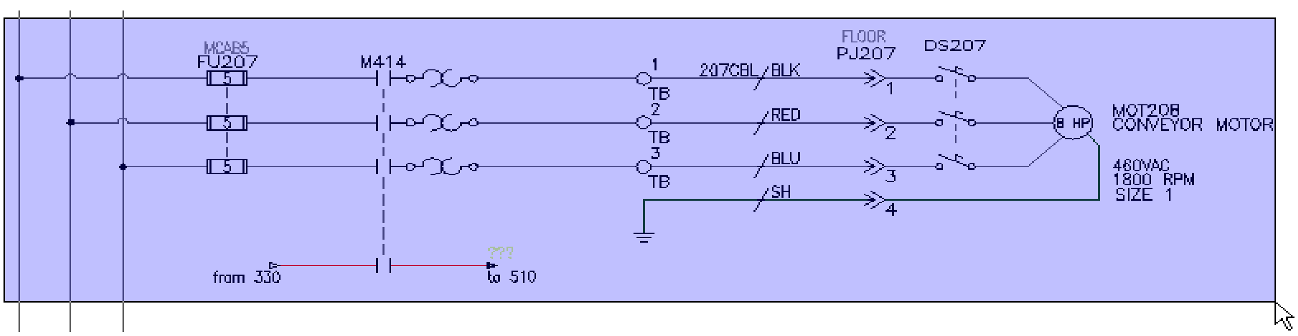
1. Pan to display the 3-phase motor circuit at line references 207 - 209.
2. Enter **wblock** at the command line and press ENTER.
3. In the **Write Block** dialog box, click Pick point.
4. Respond to the prompts as follows:

   Specify insertion base point: *Select the intersection of the left vertical bus with the upper horizontal wire at line reference 207*



5. In the Write Block dialog box, click **Select objects**.
6. Respond to the prompts as follows:

Select objects: *Window from left to right around the full circuit, right-click*

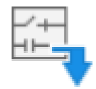


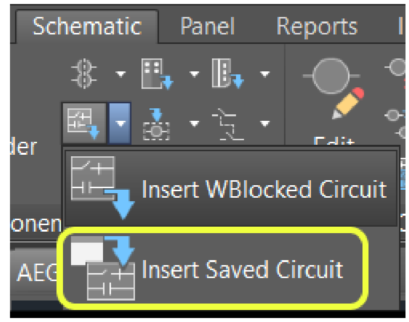
7. In the Write Block dialog box, enter a name for the saved circuit. Take note of the location where the drawing file is being saved.
8. Click OK.

Inserting a WBlocked circuit

1. Click Schematic ➤ Insert Components panel ➤ Circuit drop-down ➤ **Insert WBlocked Circuit**.
2. In the Insert Wblocked Circuit dialog box, browse to the folder containing the circuit you saved.
3. Select the WBlocked motor circuit, and click Open.
4. In the Circuit Scale dialog box, select:

    Move all lines to wire layers
    Keep all source arrows
    Update circuit's text layers as required
    Click OK.

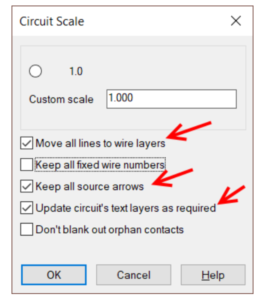




5. Respond to the prompts as follows:

    Specify insertion point: *Select any blank spot on your drawing*

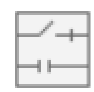
    The parent component tags that are not set to Fixed automatically retag based on the insertion point. It is like the behavior when inserting a circuit using the icon menu method.



6. Delete the circuit.

Insert a One-Line Motor Control Circuit

Use **Circuit Builder** to insert a one-line motor circuit and size the wires based on industry standards.

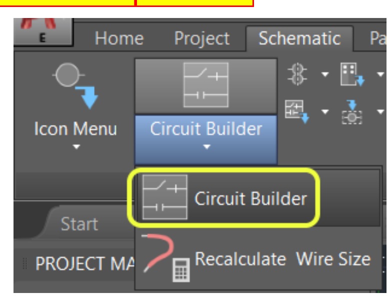


1. In the Project Manager, Project Drawing List, double-click One-Line.dwg.

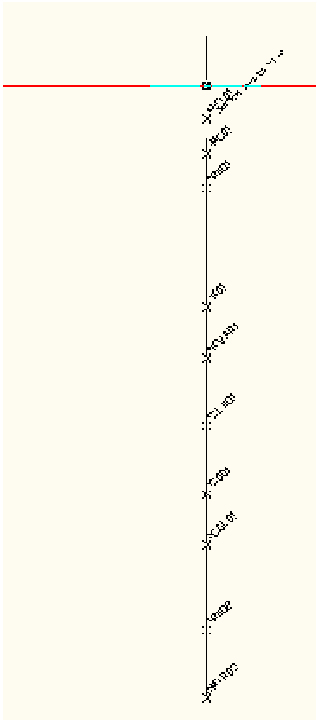
    One-Line.dwg contains a one-line bus. This wire is drawn on a wire layer defined as No Wire Numbering. Such a wire layer behaves normally for inserting, breaking, and scooting components. These wires also show up in the from/to report.

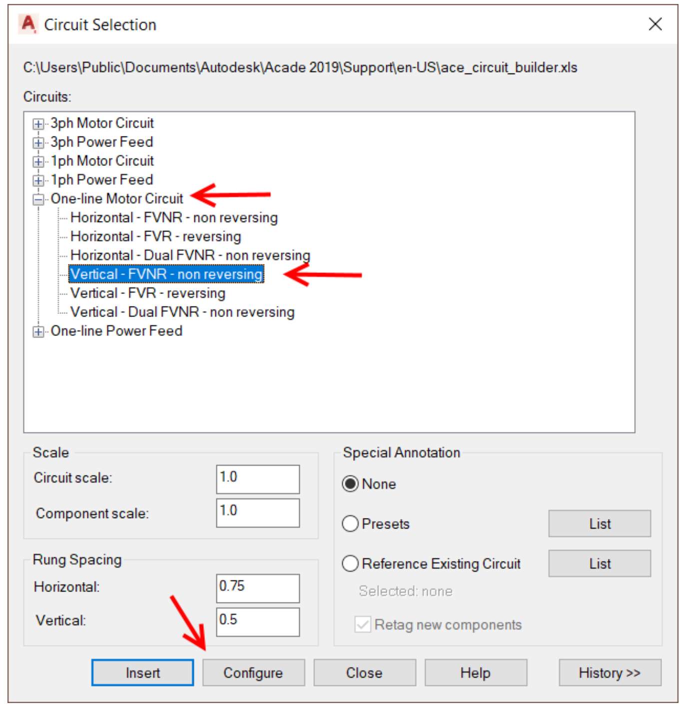
    Wire numbers are not placed on these wires during the Insert Wire Numbers process.

Insert the one-line circuit



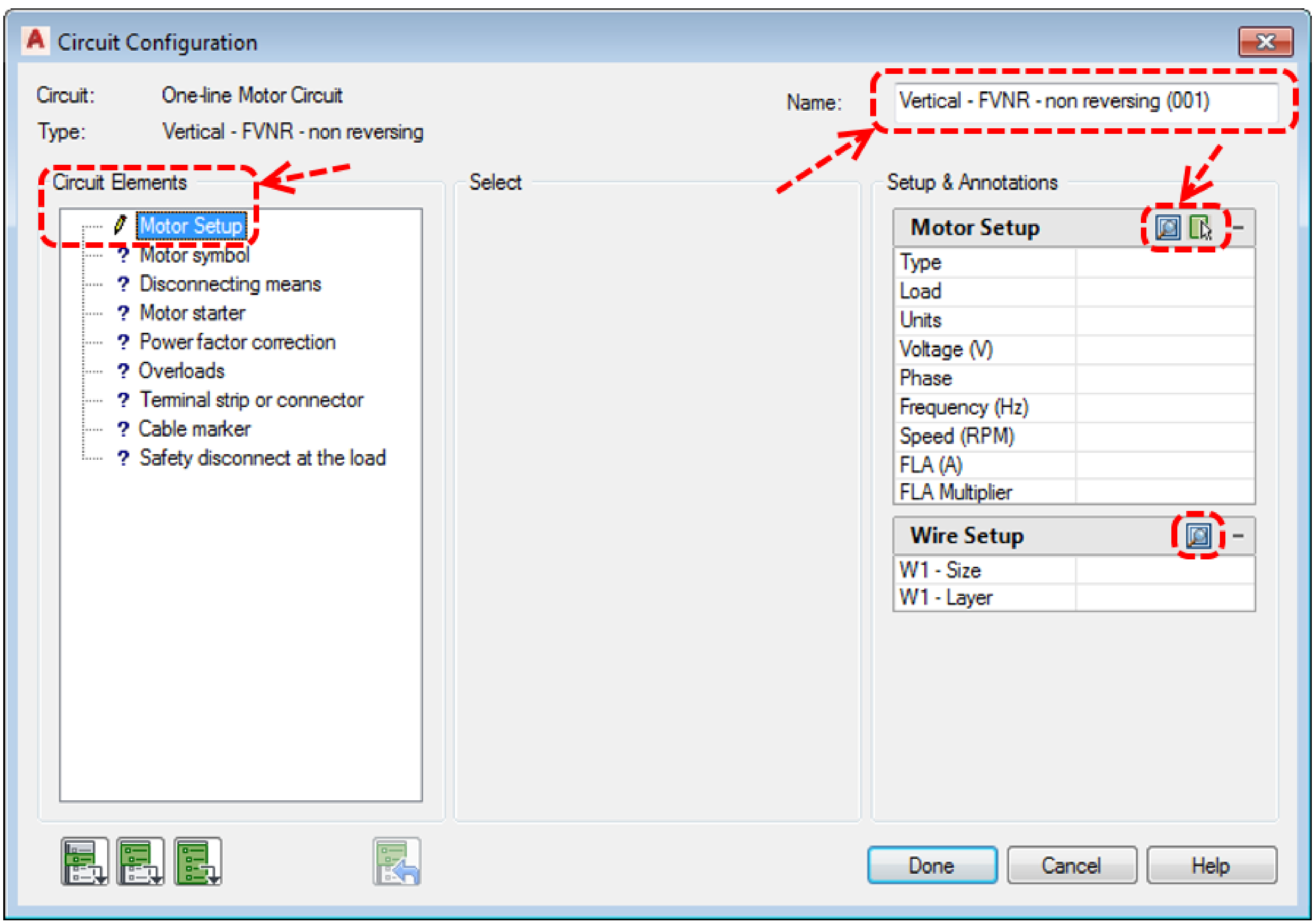
1. Click Schematic tab ➤ Insert Components panel ➤ Circuit Builder drop-down ➤ Circuit Builder. The Circuit Selection dialog box displays.
2. Expand One-line Motor Circuit.
3. Select **Vertical - FVNR - non reversing**.
4. Click Configure. Dialog will display after insertion point selected. (Shown below).
5. Specify an insertion point on the one-line bus.

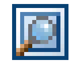




The Circuit Configuration dialog box displays.

6. In the Circuit Elements section, select **Motor Setup**.



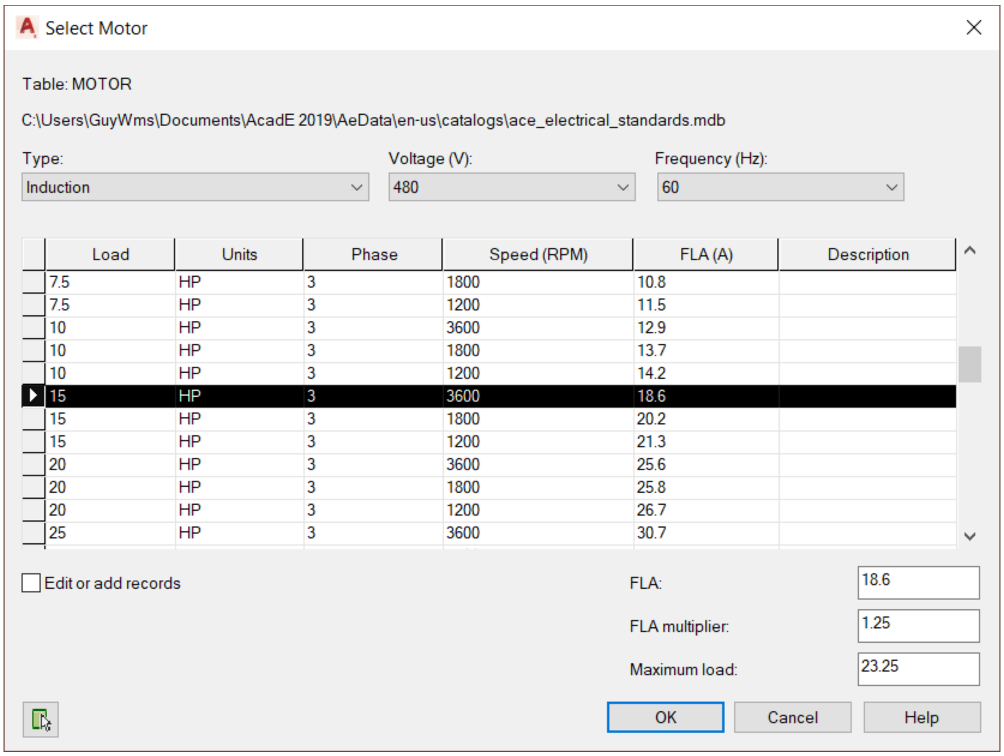
7. In the Setup & Annotations: Motor Setup section, select the Browse button. 

   The Motor Table Not Found dialog box displays. The sample project is set up to use the NEC standard. However, a MOTOR_NEC table is not supplied, only a default MOTOR table.

8. Select Use default table.

   The Select Motor dialog box displays.

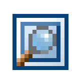
9. Select Type: **Induction**, Voltage (V): **480**, and Frequency (HZ): **60**.
10. Select the row that shows Load: **15**, Units: **HP**, Phase: **3**, Speed (RPM): **3600**, FLA (A) **18.6**.



Note: The values used to populate this dialog box are defined in the MOTOR* tables in the electrical standards database file, *ace_electrical_standards.mdb*.

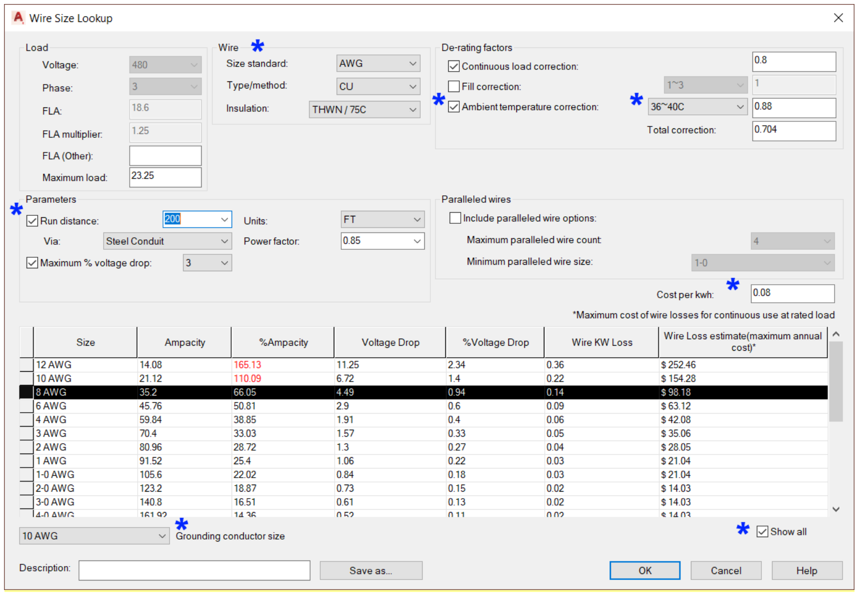
11. Click OK.

    The values are entered in the Motor Setup section. A default wire size, based on the load for the motor, is selected and shown in the Wire Setup section.

12. In the Setup & Annotations: Wire Setup section, select the Browse button. 

    The **Wire Size Lookup** dialog box displays. The minimum wire size is preselected. The size is based on the load for the selected motor. (shown below)

Note: When Show all is on, wires where the %Ampacity value is greater than 100% and less than 300%, are shown in red.



The values in the Load section are populated with the values from the Motor Setup. The options available within this dialog box are defined in the electrical standards database file, ace_electrical_standards.mdb.

13. In the Wire section, select Size standard: **AWG**, Type/method: **CU**, Insulation: **THWN / 75C**.
14. In the De-rating factors section, select the **Ambient temperature correction** option.

    This option directs Circuit Builder to use a de-rating factor for an **elevated ambient temperature**. These values are defined in the electrical standards database file.

15. Select **36~40C** from the drop-down list.

    The de-rating factor is extracted from the electrical standards database file and entered in the dialog box. The wire size grid is adjusted based on the new total de-rating factor. Based on this de-rating factor the minimum wire size can change.

16. Select the **Run distance** option.

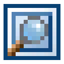
    This option directs Circuit Builder to consider the **length of the wire run in the voltage drop calculation**. Additional columns display in the wire selection grid showing Voltage drop, wire KW loss, and wire loss cost estimate.

17. Select **200** from the drop-down list.

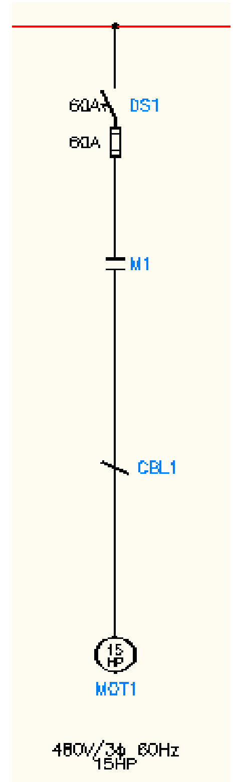
    Circuit Builder displays parallel energy loss calculations to allow you to make better green design decisions. For example, you can oversize the conductors for a motor to reduce conductor heating losses. It results in a higher initial cost, material, and installation labor, which is recovered many times over in reduced energy losses in the wiring during the life of the motor.

18. Select a wire size in the grid based on the values shown.
19. Select a Grounding conductor size. The minimum size is preselected based on the load of the motor.
20. Click OK.

21. Select Circuit Elements: **Motor Symbol**.

22.  In the Setup & Annotations: Motor section, select the Browse button.

The Catalog Browser dialog box displays.

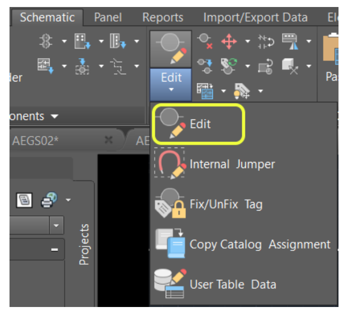


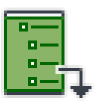
23. Search for and select a catalog value and click OK.

Note: Circuit Builder does not preselect the catalog based on the parameters entered previously.

24. Continue selecting Circuit Elements:

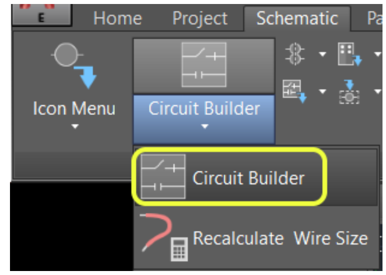
Disconnecting means:
Disconnect switch and fuses
Motor starter: **Yes**
Power factor correction: **No**
Overloads: **None**
Terminal strip or connector: **None**
Cable marker: **Yes**
Safety disconnect at the load: **None**



25. Click  to insert all circuit elements.
26. Click Done.
27. Click Schematic tab ➤ Edit Components panel ➤ Edit Components drop-down ➤ Edit. 
28. Select the motor symbol.
29. On the **Insert/Edit Component** dialog box, enter **FIELD** for the Location code and **MY MOTOR** for Description Line 1.

30. Save the drawing.

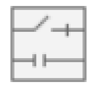
Insert a One-Line Dual Power Feed Circuit



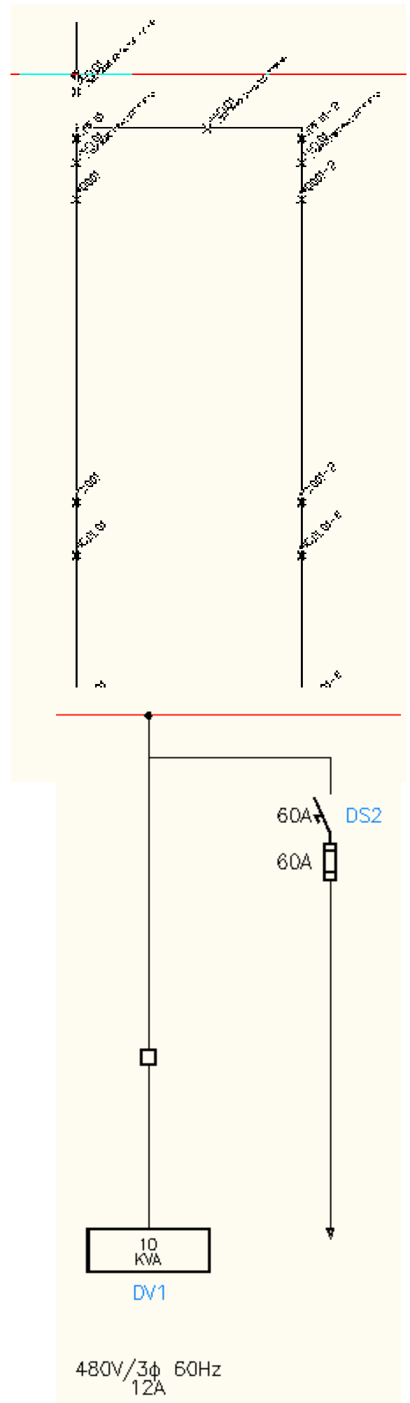
Use Circuit Builder to configure and insert a dual power feed circuit.

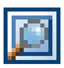
A dual circuit has two distinct circuits running off the same bus-tap. Each circuit can be independently configured.

1. Click Schematic ➤ Insert Components panel ➤ Circuit Builder drop-down ➤ Circuit Builder.
2. The Circuit Selection dialog box displays.
3. Select **One-line Power Feed: Vertical - Dual feed**.
4. Click Configure.
5. Specify an insertion point on the one-line bus.



The Circuit Configuration dialog box displays. Some circuit elements have a "**(2)**" prefix. These elements make up the second circuit in the dual circuit.

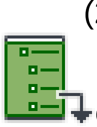


6. In the Circuit Elements section, select **Load Setup**.
7.  In the Setup & Annotations: Load Setup section, select the Browse button. The Select Load dialog box displays.
8. Select Type: **Transformer**, Voltage (V): **480**, and Phase: **3**.
9. Select an entry from the grid and click OK.
10. Continue selecting Circuit Elements for the first circuit:

    Load: **Generic box**
    Disconnecting means: **None**
    Terminal strip or connector: **Square**
    Cable marker: **None**

11. In the Circuit Elements section, select **(2) Load Setup**.
12.  In the Setup & Annotations: Load Setup section, select the Browse button. The Select Load dialog box displays.
13. Select Type: **Transformer**, Voltage (V): **480**, and Phase: **3**.
14. Select an entry from the grid and click OK.
15. Continue selecting Circuit Elements for the second circuit:

    (2) Load: **Source arrow**
    (2) Disconnecting means: **Disconnect switch and fuses**
    (2) Terminal strip or connector: **None**
    (2) Cable marker: **None**

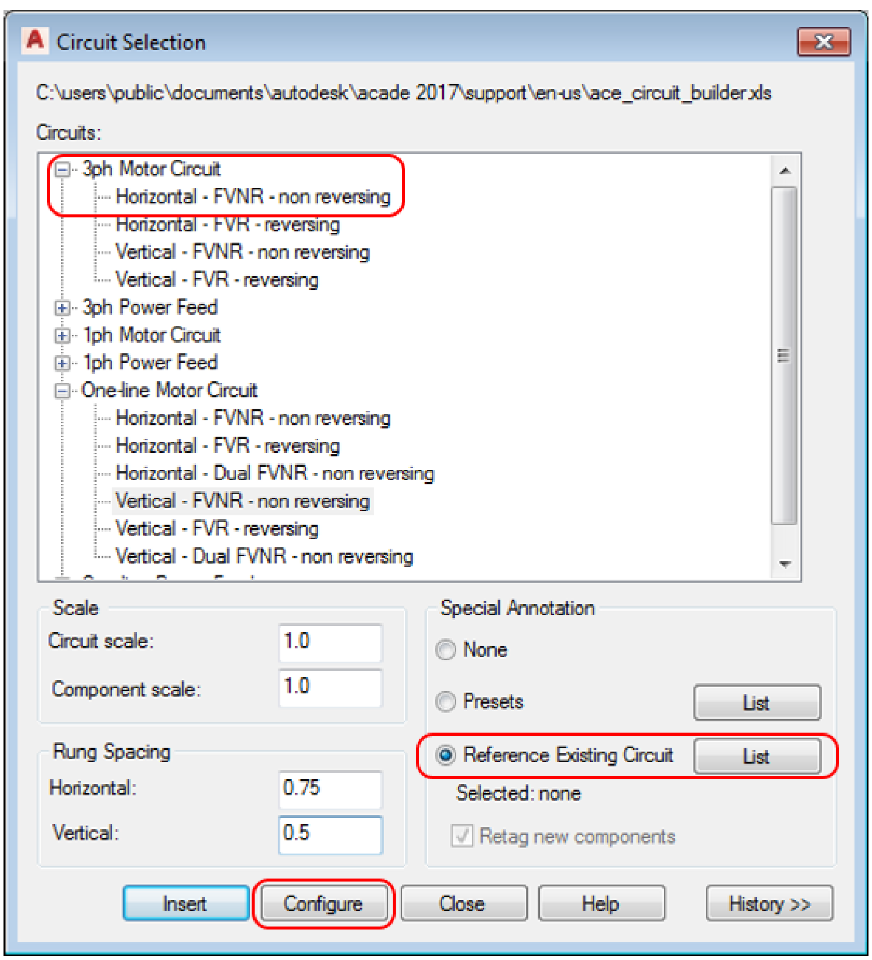
16.  Click to insert all circuit elements.
17. Click Done.
18. Save the drawing.

Reference an Existing Circuit

Use Circuit Builder to insert and annotate a circuit based on an existing circuit.

When a new circuit is inserted, you can reference an existing circuit picked from a list of circuits pulled from the active project. The components, values, descriptions, and tag assignments from the selected circuit, become defaults for the new circuit. Tags are recalculated if the option "Retag new components" is selected.

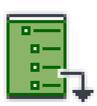
1. Start a new blank drawing and save it as Three-Line.dwg.
2. In Project Manager, right-click on the project name and select **Add Active Drawing**.
3. Click Yes to apply the project default values to the drawing settings.
4. Click Schematic ➤ Insert Wires/Wire Numbers panel ➤ Insert Ladder drop-down ➤ Insert Ladder.
5. Insert a 3-phase ladder.
6. Click Schematic tab ➤ Insert Components panel ➤ Circuit Builder drop-down ➤ Circuit Builder.
7. The Circuit Selection dialog box displays.
8. Select **3ph Motor Circuit: Horizontal - FVNR - non reversing**.
9. Select Reference Existing Circuit.
10. Select the List button. The Existing Circuits dialog box displays.
11. Select the one-line motor control circuit inserted on One-Line.dwg, **MOT1**.
12. Click OK.
13. Turn off the Retag new components check box.

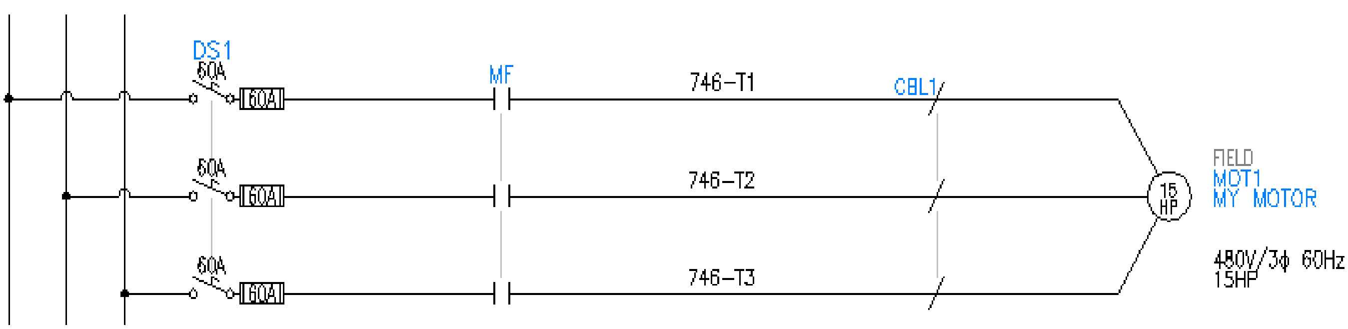


This directs Circuit Builder to use the tags from the one-line circuit for the components with matching marker block code values.

14. Select Configure.
15. Select an insertion point on the bus for the new circuit.
16. Verify that the same circuit elements as the referenced one-line motor circuit are selected. The default options are based on the referenced circuit.

Circuit Elements	Select
Motor symbol	Motor: **3ph motor** Ground/PE wire connection: **No**
Disconnecting means	Main Disconnect: **Disconnect switch and Fuses** Include N.O. Auxiliary contact: **No**
Control transformer and circuit - non-reversing	Include control circuit: **None**
Power Factor correction	Include power factor correction capacitor: **None**
Overloads	Overload elements: **None** Include N.O. auxiliary contact: **No**
Motor terminal connections	Motor connection terminals: **None**
Cable marker	Cable: **Yes**
Safety disconnect at the load	Safety disconnect: **None** Include N.O. auxiliary contact: **No**

17.  Click to insert all circuit elements.
18. Click Done.



The circuit is inserted and the component values from the one-line circuit are applied. The motor symbol receives the same catalog value and horsepower rating. The main disconnect switch receives the same rating values for the switch and the fuses. The motor symbol receives the values modified on the one-line circuit after it was inserted.

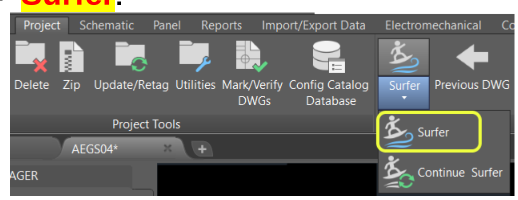
Surf Tutorial

Time required	10 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Surf to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

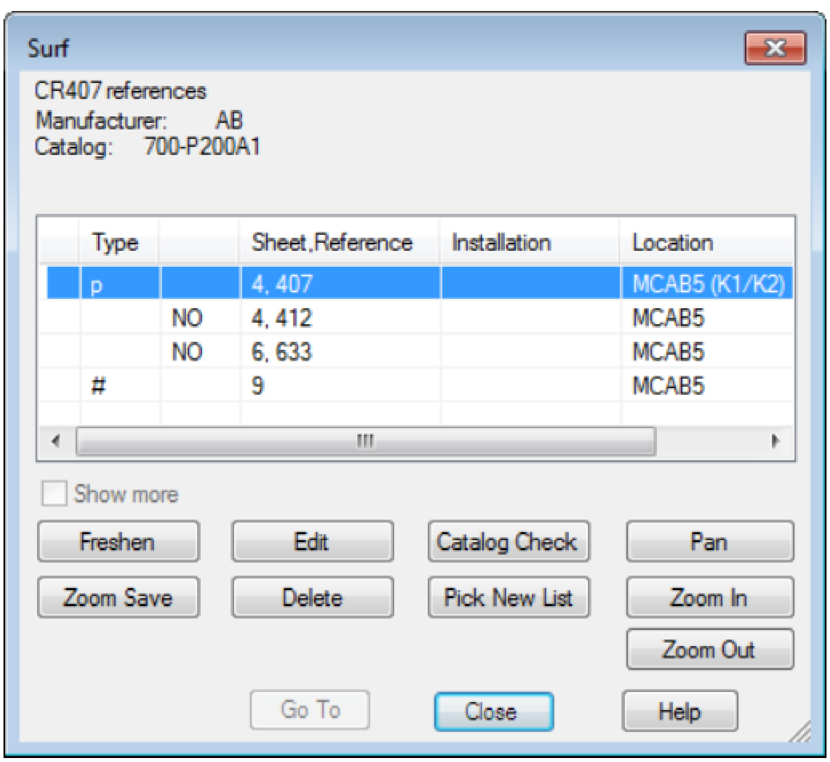
Moving Between Symbols

Use the AutoCAD Electrical toolset Surf utility to quickly move from component reference to reference across the project drawing set.

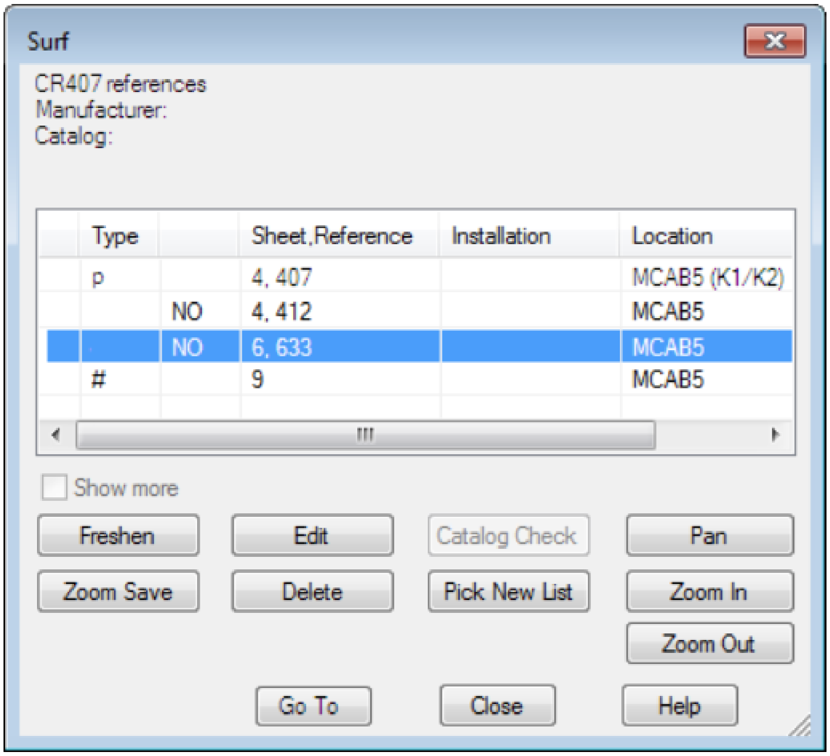
1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS04.dwg.
4. Zoom on the upper left-hand portion of the first ladder column.
5. Click Projects tab ➤ Other Tools panel ➤ Surfer drop-down ➤ **Surfer**.
6. Click anywhere on relay coil CR407.



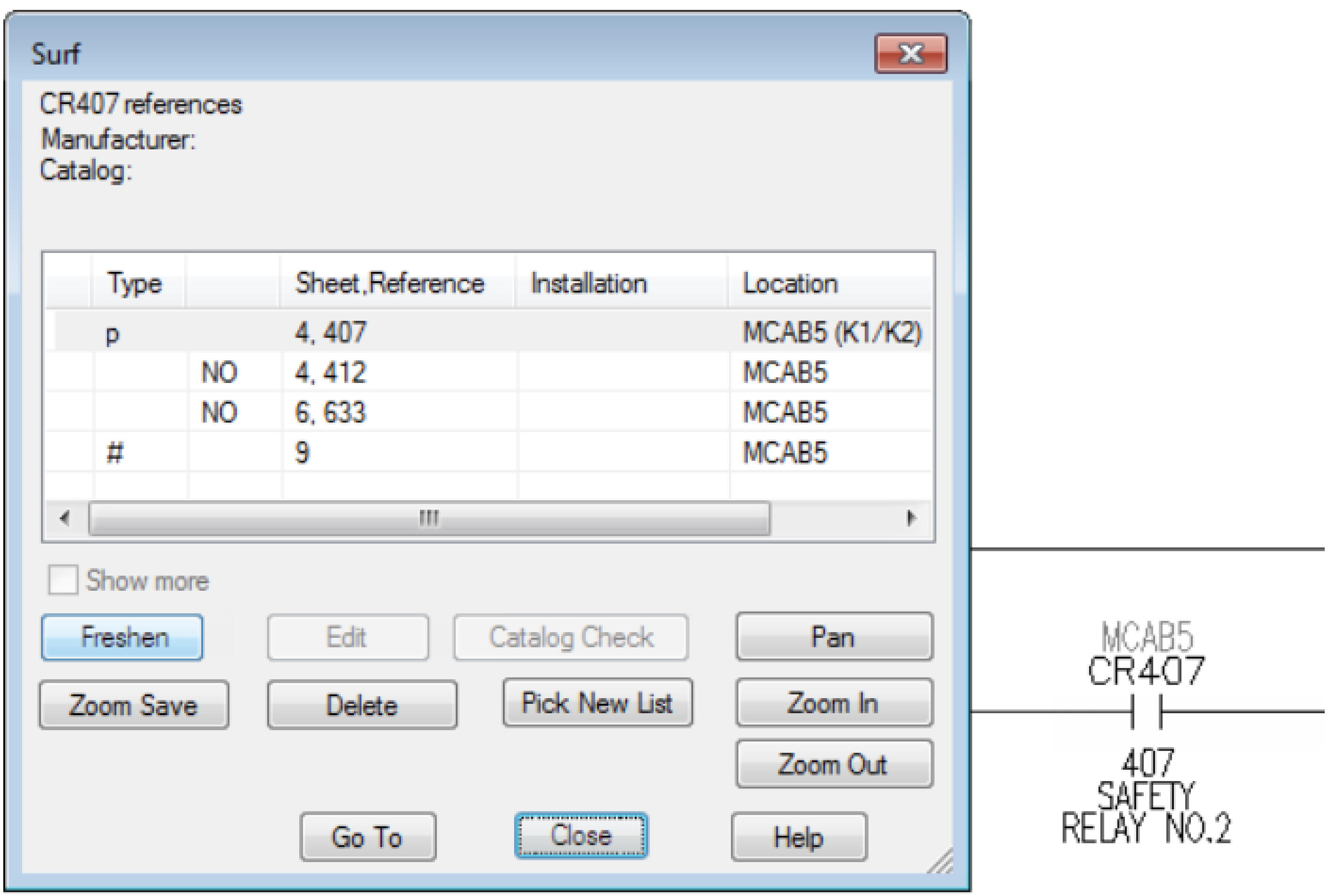
All instances of CR407 appear in the Surf dialog box.



7. Select the reference on sheet 6.
8. Click Go To.



The instance of CR407 on sheet 6 is surfed to and displayed in the drawing next to the Surf dialog box.



- Select the reference on sheet 9.
- Click Go To.

You can edit or delete the component using options in the Surf dialog box.

- Double-click the first entry in the Surf dialog box to return to the original AEGS04.dwg drawing.
- Click Close.

Note: If AutoCAD Electrical toolset senses that a change has been made to the drawing while surfing, drawing files are saved.

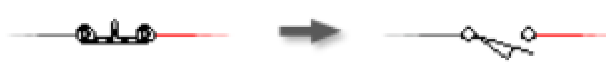
Block Swap Tutorial

Swap components while maintaining wire connections with **Swap/Update Block**.

Time required	10 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Block swap to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

You learn to:

- Swap components while maintaining wire connections and attribute values.

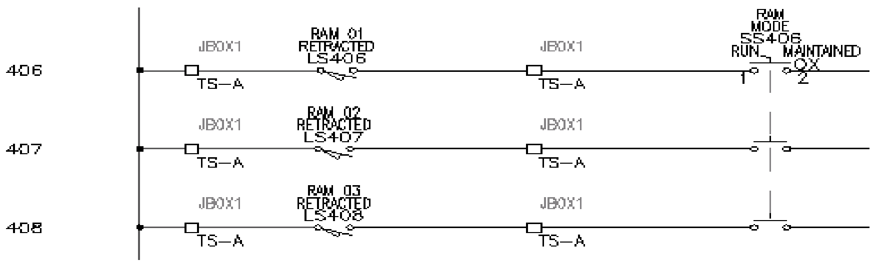


Swapping Components

Use the Swap Block tool to swap one component for another.

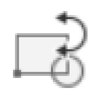
Swap switches while keeping wire connections

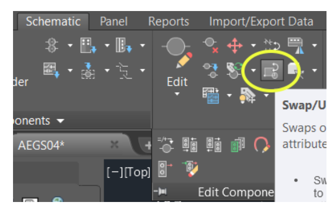
1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS04.dwg.
4. Zoom in on the limit switch on line reference 406.

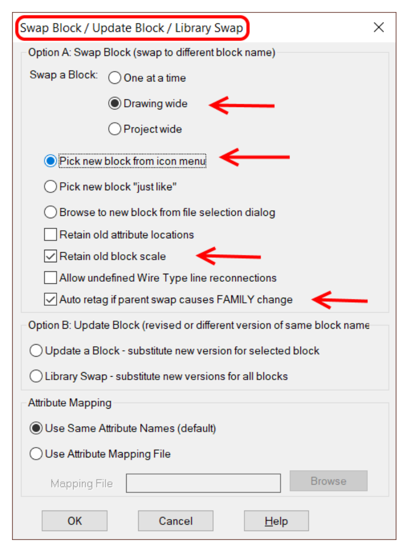


5. Click Schematic tab ➤ Edit Components panel ➤ Swap/Update Block.
6. In the Swap Block/ Update Block/ Library Swap dialog box, specify:

   Option A: Swap a Block - drawing wide
   Pick new block from icon menu
   Retain old block scale
   Auto re tag if parent swap causes FAMILY change
   Attribute Mapping: Use Same Attribute Names (default)
   Click OK.



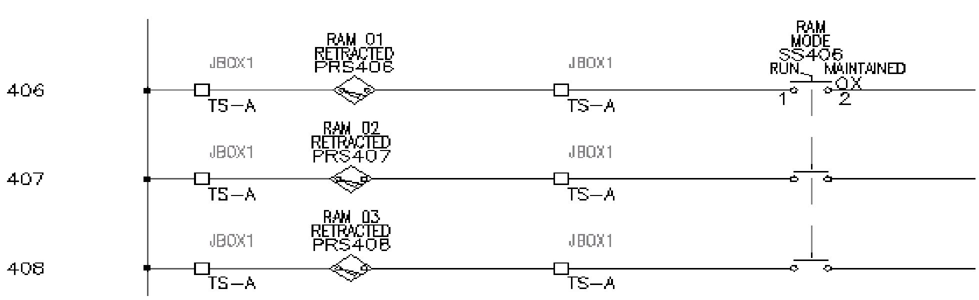




7. In the Insert Component: JIC Schematic Symbols dialog box, click Miscellaneous Switches. 
8. In the JIC: Other Switch Types dialog box, click Proximity Switch  NO.
9. Respond to the prompts as follows:

   Select component type to swap out: *Select the limit switch, LS406*

   The limit switch symbol disappears and the proximity switch symbol inserts. All existing text annotation transfers to the new symbols and the wires reconnect.

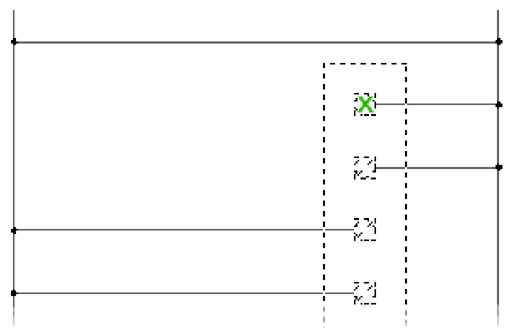


PLC Tutorial

Time required	30 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\PLC to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Insert a PLC module
- Remove ladder rungs
- Use multiple insert component
- Annotate PLC I/O descriptions



Inserting PLC Modules

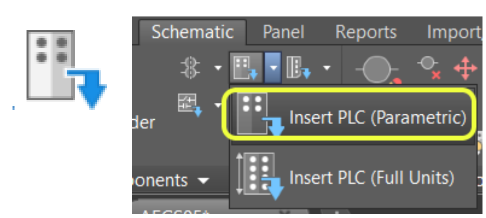
Select, insert, and annotate a PLC module.

AutoCAD Electrical toolset generates any of hundreds of different PLC I/O modules on demand. The modules generate in various different graphical styles, all without a single, complete I/O module library symbol resident on the system. Modules adapt to the underlying ladder rung spacing, whatever that value is. They can be stretched or broken into two or more pieces at insertion time.

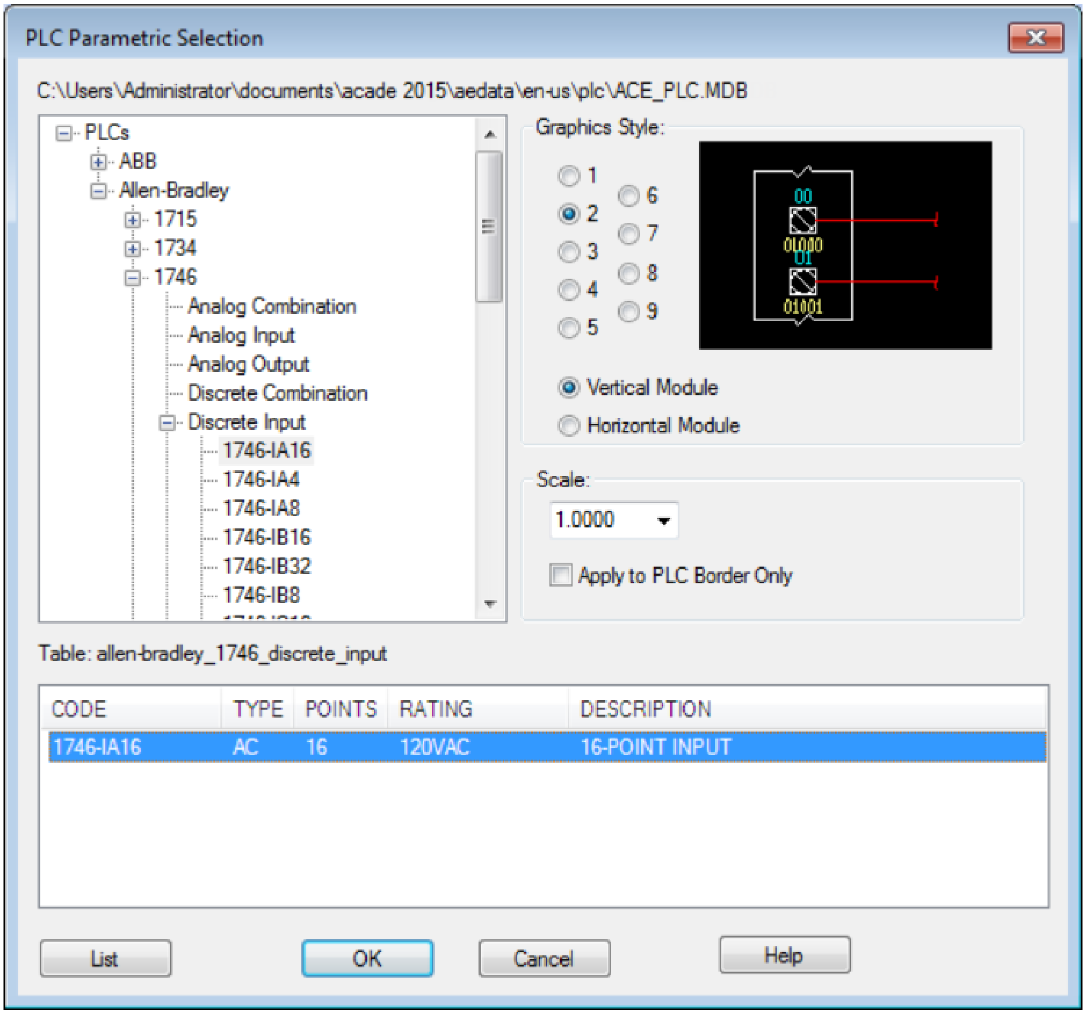
To insert a PLC module, you select the module and pick a location. AutoCAD Electrical toolset builds and inserts the module, using a small set of library symbols.

Insert a PLC module

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS05.dwg.
4. Click Schematic ➤ Insert Components panel ➤ Insert PLC drop-down ➤ Insert PLC (Parametric).
5. In the PLC Parametric Selection dialog box, select:

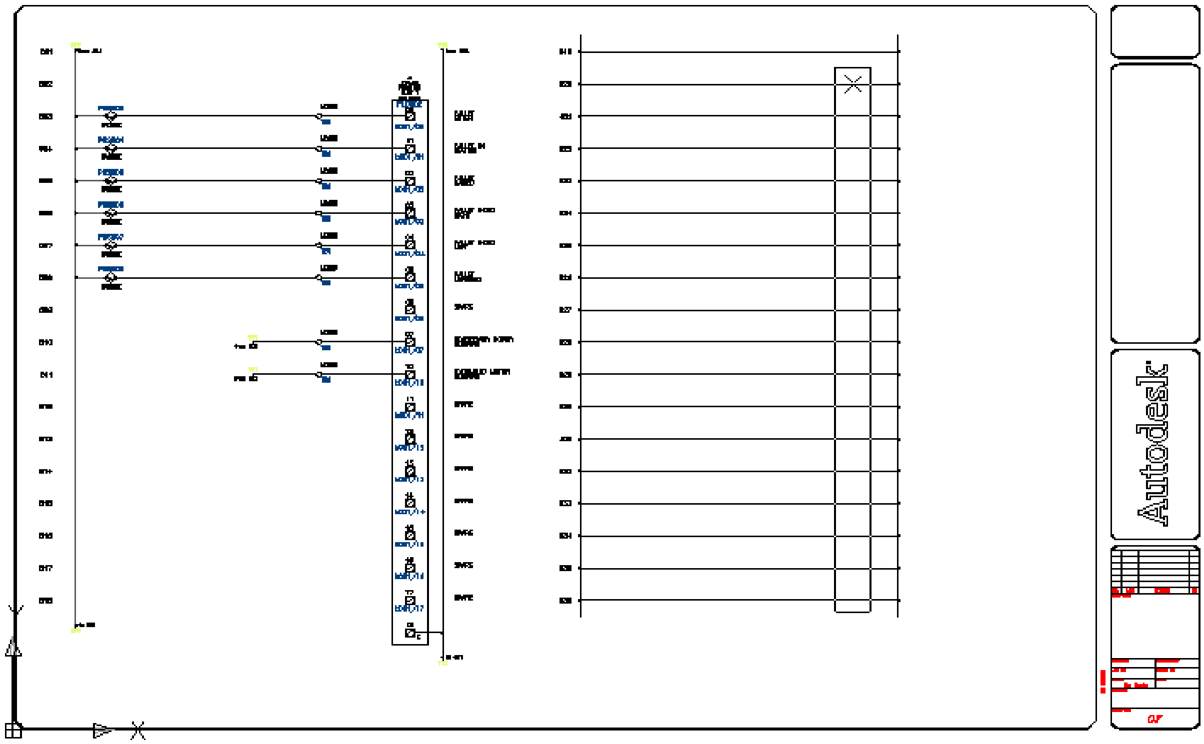


Manufacturer:	Allen-Bradley
Series:	1746
Type:	Discrete Input
Part Number:	1746-IA16
Graphics Style:	2, Vertical Module



6. Click OK.
7. Respond to the prompts as follows:

   Specify PLC module insertion point or [Z=zoom, P=pan]: *Pick a point on wire line reference 520 closer to the right side, ensure the X is near the horizontal wire, click*



8. In the Module Layout dialog box, verify the default settings:

   Spacing: 1.0000
   I/O Points: Insert all
   Click OK.

   AutoCAD Electrical toolset reads the vertical rung spacing of your ladder and calculates how long the module is going to be. It multiplies the rung spacing by the number of wire connections specified by the module you selected.

   Temporary graphics display a representation of the module (with the spacing defined) to help position the module on the ladder.

9. In the I/O Point dialog box, specify:

   Rack Number: 1
   Slot Number: 1

   Note: Specify the values by either entering text into the edit boxes or by clicking the arrows.

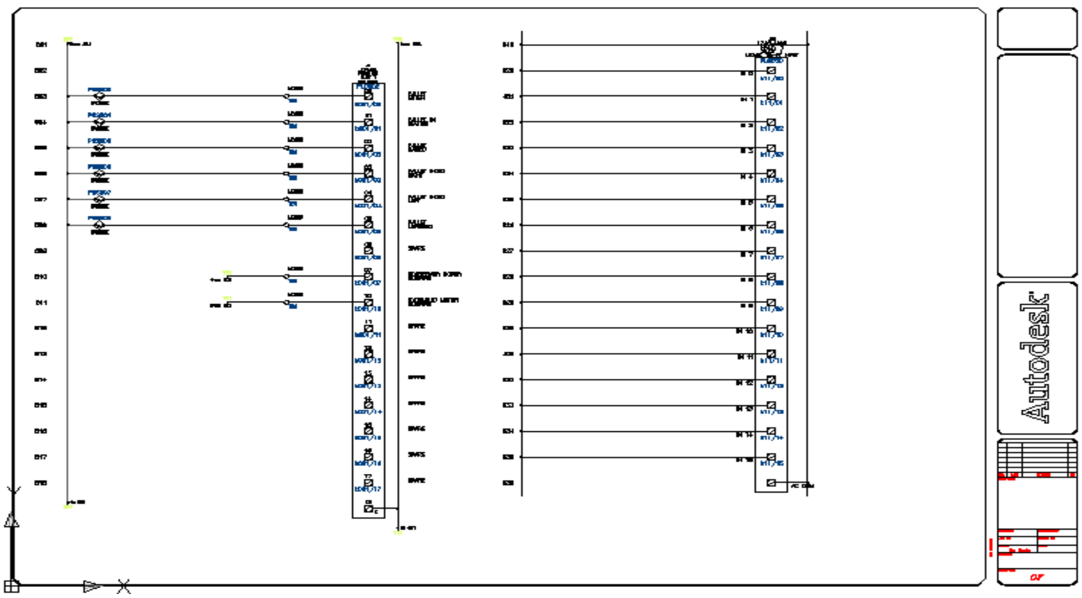
10. Click OK.
11. In the I/O Address dialog box, specify:

    Beginning address: I:11/00

    Note: You can also select the beginning address from the Quick picks list.

12. Click OK.
13. In the I/O Addressing dialog box, click Decimal.

The PLC module is inserted into your drawing with incremental address numbers already annotated as the module goes in, it breaks and reconnects to underlying wires.



You can break an I/O module into as many pieces as you want at insertion time. It is great for high-density modules that do not fit into a single ladder column. **Use the Allow spacers/breakers option** in the Module Layout dialog box at insertion time to do it.

You can also add extra space between adjacent I/O points using the Stretch Block tool. This feature leaves extra room when you know ahead of time that a certain I/O point will have additional components wired tied to a single I/O point after a PLC module is inserted.

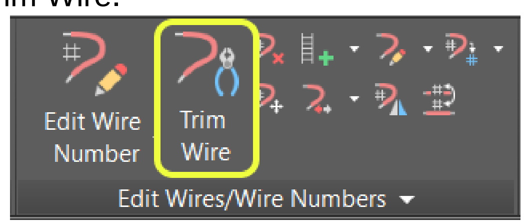
Note: It can be used on any block, not just a PLC module.

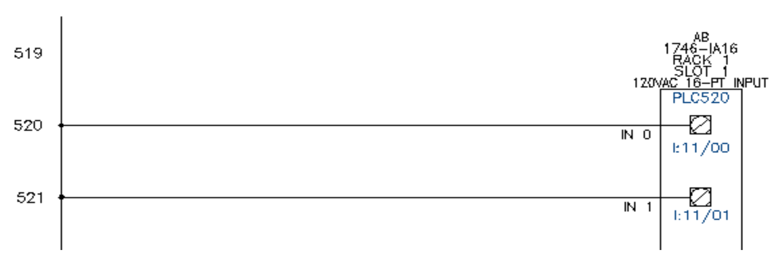
Remove ladder rungs

1. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Trim Wire.
2. Respond to the prompts as follows:

   Fence/Crossing/Zext/<Select wire to TRIM>: *Select the ladder rung at line reference 519, right-click*

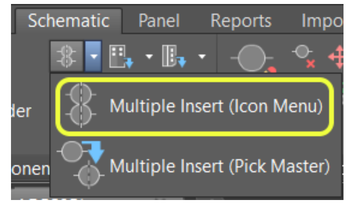
   The ladder rung is removed from your drawing.





Using Multiple Insert Component

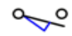
Use the Multiple Insert Component tool to insert a string of normally open limit switches into wires that are tied to the PLC module.



Insert a limit switch

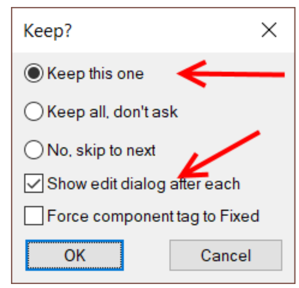
1. Click Schematic ➤ Insert Components panel ➤ Multiple Insert drop-down ➤ Multiple Insert (Icon Menu).

2. In the Insert Component: JIC Schematic Symbols dialog box, click Limit Switches. 

3. In the JIC: Limit Switches dialog box, select Limit Switch, NO. 
4. Respond to the prompts as follows:

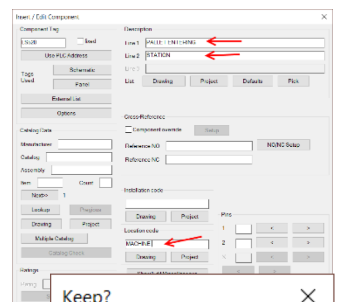
   Component Fence, From Point: *Select above the wire at line reference 520 (1)*

   Component Fence, From Point: to: *Drag below the wire at line reference 522, click the point (2), right-click*



5. In the Keep dialog box, select:
   Keep this one
   Show edit dialog box after each

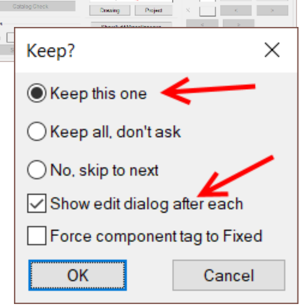
   Click OK



6. In the Insert/Edit Component dialog box, specify:
   Component Tag: LS520
   Description: Line 1: PALLET ENTERING
   Description: Line 2: STATION
   Location code: MACHINE

   Click OK.

   Note: In the Insert/Edit Component dialog box, Component Tag section, you can use the Use PLC Address button to add the I/O Address as the component tag.



7. In the Keep dialog box, select:
   Keep this one
   Show edit dialog box after each

   Click OK

8. In the Insert/Edit Component dialog box, specify:
    Component Tag: LS521
    Description: Line 1: PALLET INSIDE
    Description: Line 2: STATION
    Location code: MACHINE

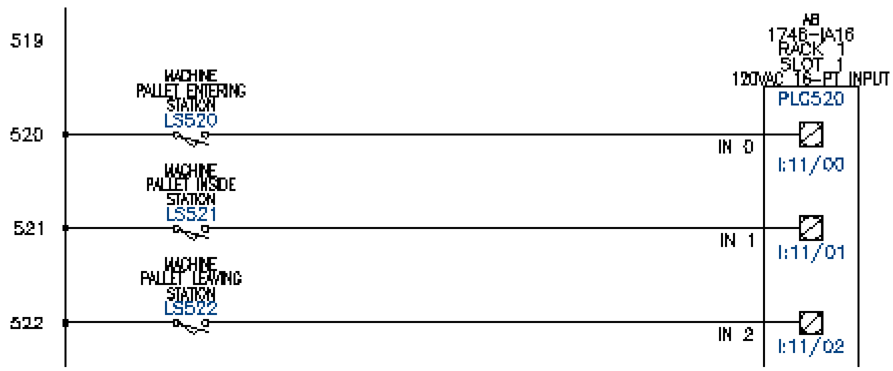
   Click OK.

9. In the Keep dialog box, select:
    Keep this one
    Show edit dialog box after each
    Click OK

10. In the Insert/Edit Component dialog box, specify:
    Component Tag: LS522
    Description: Line 1: PALLET LEAVING
    Description: Line 2: STATION
    Location code: MACHINE

    Click OK.

The normally open limit switches are inserted into the drawing.

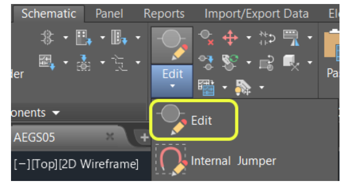


Annotating PLC I/O Descriptions

Edit a PLC module to add I/O descriptions.

You can add description text to a PLC module using the **Edit Component** tool. You can change the descriptions at any time. However, edit each split PLC piece separately.

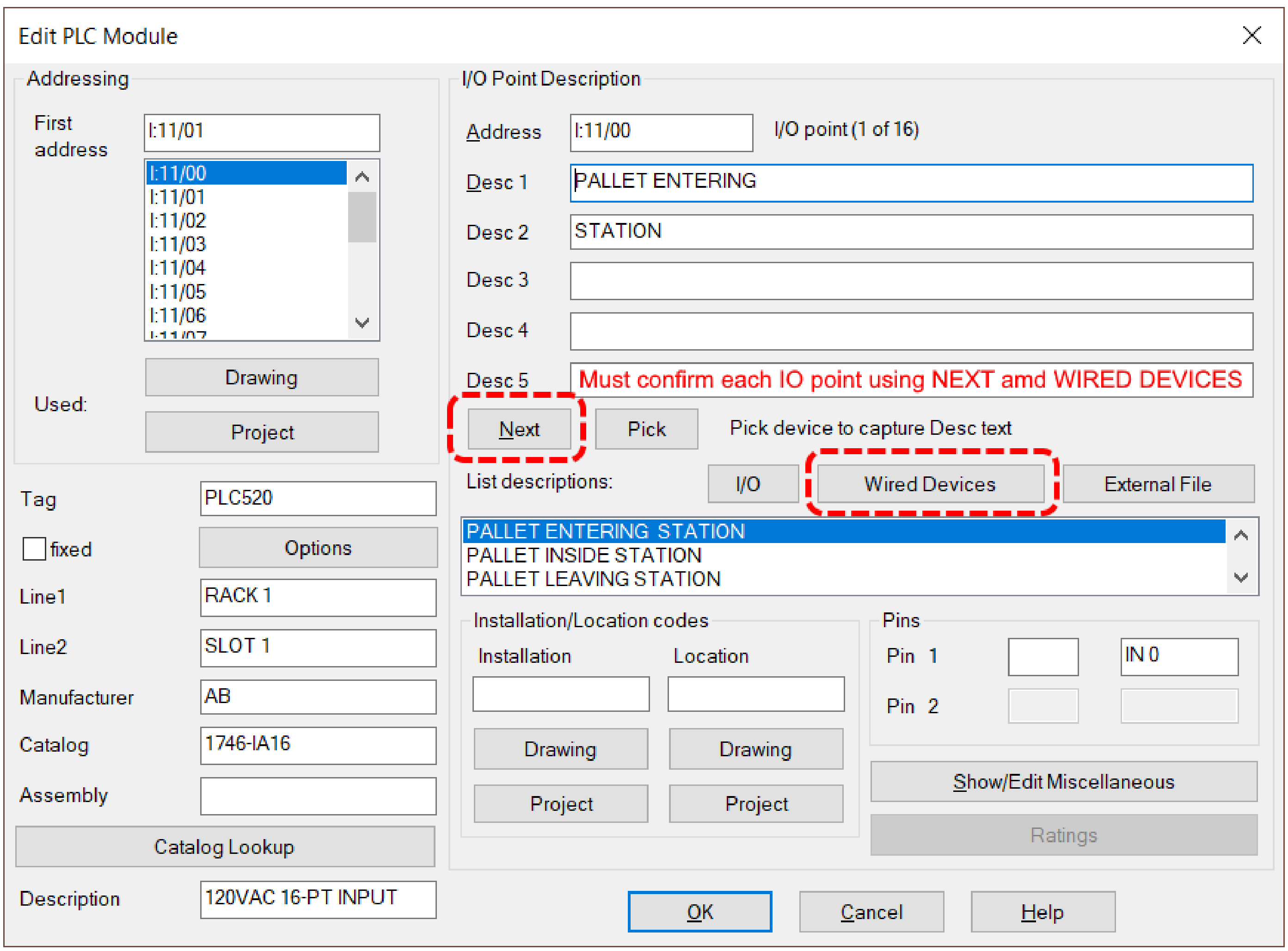
Add description text



1. Click Schematic tab ➤ Edit Components panel ➤ Edit Components drop-down ➤ Edit.
2. Respond to the prompts as follows:

Select component/cable/location box to EDIT: *Select anywhere on the top portion of the PLC module*

The Edit PLC Module dialog box displays.



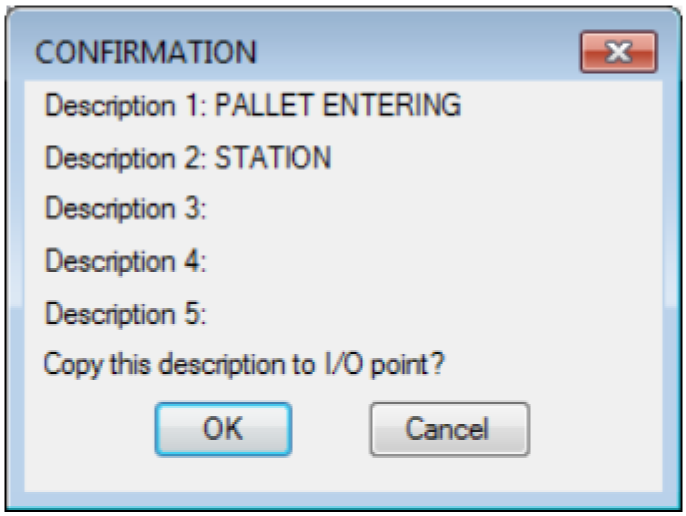
This dialog box provides spaces for you to enter description text for each I/O point. Assume that the descriptions already assigned to the connected limit switches are like what you want to use for the PLC I/O point descriptions.

3. In the Edit PLC Module dialog box, click Wired Devices. (be sure the wires from external devices actually touch the connection lug on the PLC.)

   AutoCAD Electrical toolset follows the connected wire for each I/O point backwards. If it finds a connected component, the component description text is retrieved. Each description displays in a dialog box list.

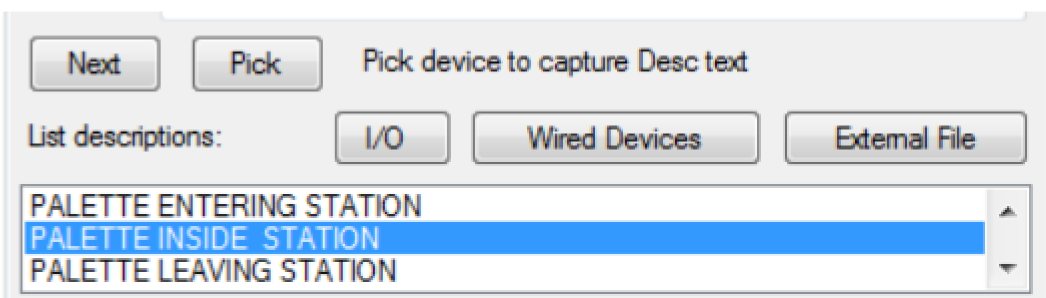
4. For the first I/O address (I:11/00), select the first description (PALLET ENTERING STATION) in the extracted device list.

   The Confirmation dialog box displays.



5. Make sure that the correct description is specified and click OK.
6. Click Next to highlight I/O address 1:11/01 in the Addressing list.

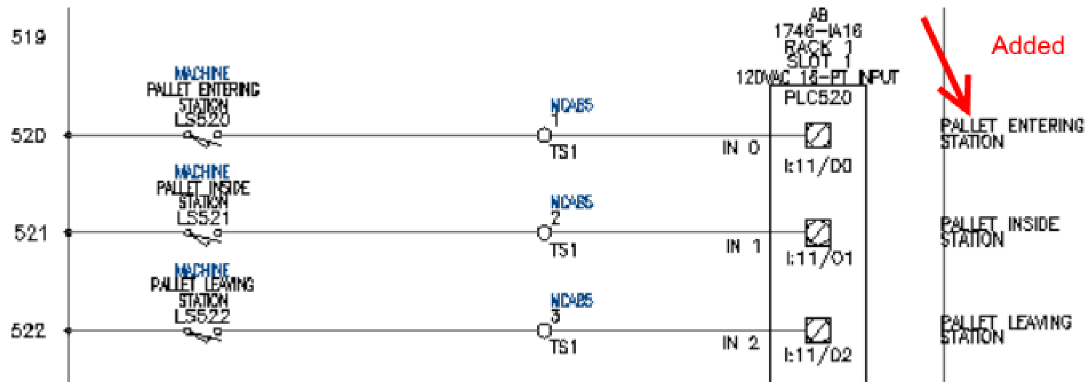
   The corresponding device description highlights automatically.



7. Select the highlighted description, PALLET INSIDE STATION, and click OK.
8. Repeat this process for the remaining I/O point.
   Note: Alternately you can use Pick to capture existing description text from a connected device. To do so, in the Edit PLC Module dialog box, click Pick, and then select the component whose text you want to copy. AutoCAD Electrical toolset reads the existing DESC text values on the component and transfers a copy to the DESC boxes in the Edit PLC Module dialog box.

9. In the Edit PLC Module dialog box, click OK.

   Your descriptions appear on the module.



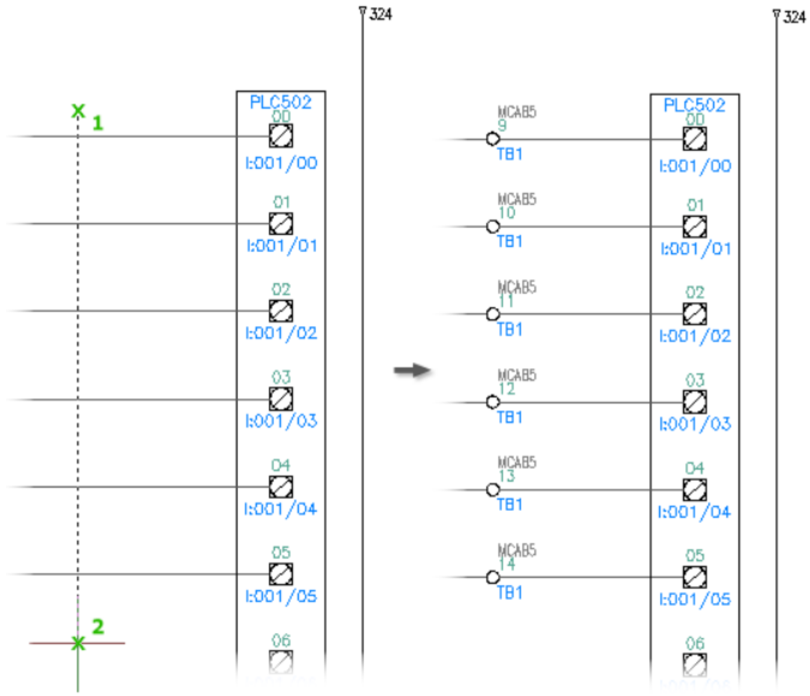
Note: If your PLC description is not where you want it, use the Scoot tool to scoot the description to a new location.

Schematic Terminals Tutorial

Time required	45 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Schematic terminals to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Insert terminals
- Assign terminal block properties
- Define a multi-level terminal
- Associate terminals



About Schematic Terminals

Understand schematic and panel terminals and relationships.

AutoCAD Electrical toolset supports two types of relationships for terminals: schematic-to-schematic and schematic-to-panel.

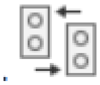
Note: Since one-line terminal symbols likely represent multiple, independent terminals, they cannot be associated to other schematic or panel terminals. A one-line terminal must be updated manually. A one-line terminal symbol is defined by a WDTYPE attribute value of “1-”.

Schematic-to-Schematic - The schematic-to-schematic relationship defines separate schematic terminal symbols as one multi-level (also referred to as multi-tier or multi-stack) terminal block. On the schematic drawing, each schematic terminal symbol represents one level of the multi-level terminal block. **Note**: Multiple terminal symbols for one level are not currently supported.

The **number of levels** for the block is defined as a block property. Each level carries certain characteristics, such as a label, wires per connection, left pin, and right pin. Each schematic terminal symbol carries all the block properties for each level so that removing one terminal symbol does not remove the block properties. If a block property is modified, all the terminal symbols update.

An ID value held on the LINKTERM attribute or Xdata, associates the terminal symbols. When a terminal symbol is inserted, by default it is seen as a standalone terminal (it has no associations) and receives a new LINKTERM value. When the terminal is associated to another, the LINKTERM value updates so that each terminal carries the same LINKTERM value. Changing or removing the LINKTERM value breaks any associations that terminal has.

To associate schematic terminals, first add block properties. The number of terminals you can associate is limited to the number of levels defined in the block properties. Once block properties are established you can associate schematic terminals to build a multi-level terminal block by:



- Click Schematic tab ➤ Edit Components panel ➤ ➤ Associate Terminals. You select a **master terminal** and then select each terminal symbol to associate to the master.

- Clicking Pick on the Insert/Edit Terminal Symbol dialog box adds the edited symbol into an association with the picked terminal.
- Clicking Add/Modify on the Insert/Edit Terminal Symbol dialog box. It adds the edited symbol into an association with any schematic terminal in the project.

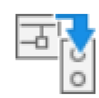
Prebuilt circuits can contain associated terminals. These relationships are maintained when the circuit is inserted. Copying a circuit also maintains these relationships within the copied circuit.

When the Bill of Materials report is run, these separate terminal symbols that make up one multi-level terminal, are counted as one in the quantity.

Schematic-to-Panel

The schematic-to-panel relationship is used mainly for updating. If the schematic or panel is modified, the other updates to reflect the changes. This relationship is like component relationships, which are based on the TAG value. The TAGSTRIP, Installation, and Location values must match for the terminals to associate together. The association number on the LINKTERM is also taken into account when creating a relationship between the schematic terminal and its panel representation. Block properties are not required to associate a schematic to panel terminal. Once they are associated, modifications on one results in modifications on the other.

You can associate a schematic and panel terminal automatically by:



- Click Panel tab ➤ Terminal Footprints panel ➤ ➤ Insert Terminals drop-down ➤ Insert Terminal (Schematic List).
- Click Schematic tab ➤ Insert Components panel ➤ Insert Components drop-down ➤ Terminal (Panel List).

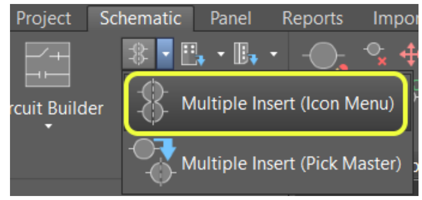
For multi-level terminals, the Insert Terminal (Schematic List) tool shows only one terminal for insertion regardless of how many schematic terminal symbols/levels there are for that multi-level block. The Insert Terminal (Panel List) tool shows one terminal for each level for insertion.

Note: Panel terminals inserted by the Terminal Strip Editor are automatically associated to the schematic representation.

You can click the **Associate terminals tool** to select terminals to associate or click Add/Modify on the Panel Layout - Terminal Insert/Edit dialog box to add the panel terminal to an association with a schematic terminal on any drawing in the project.

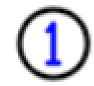
Insert Terminals (as in Terminal Blocks)

Use the multiple insert tool to insert and annotate a string of terminals.



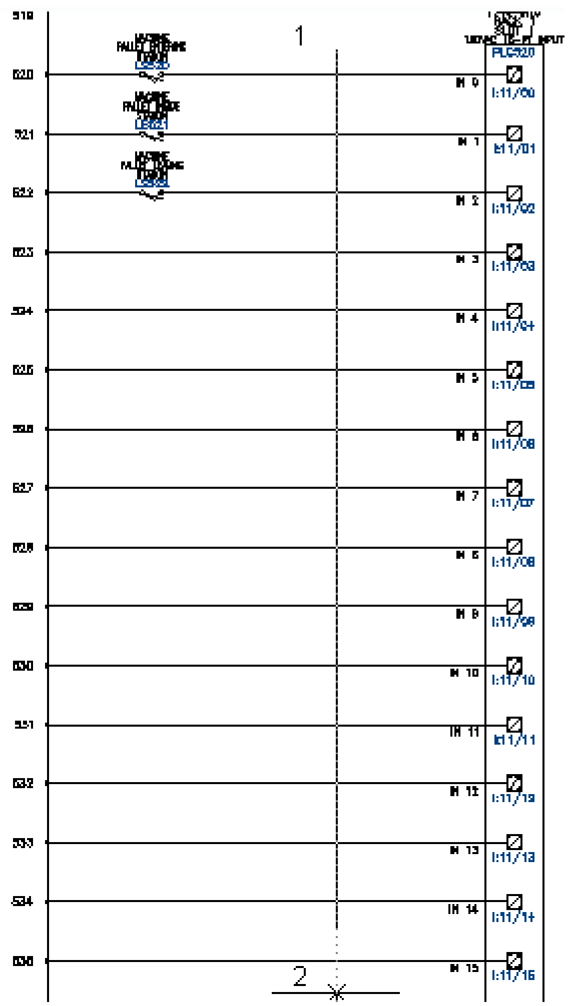
Insert terminals

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS05.dwg.
4. Click Schematic ➤ Insert Components panel ➤ Multiple Insert drop-down ➤ Multiple Insert (Icon Menu).
5. In the Insert Component: JIC Schematic Symbols dialog box, click Terminals/Connectors.
6. In the JIC: Terminals and Connectors dialog box, click Round with Terminal Number.
7. Respond to the prompts as follows:



Component Fence, From Point: *Select above wire at line reference 520 (1)*

Component Fence, From Point: to: *Select below wire at line reference 535 (2), left click to end command, right-click to add terminal*

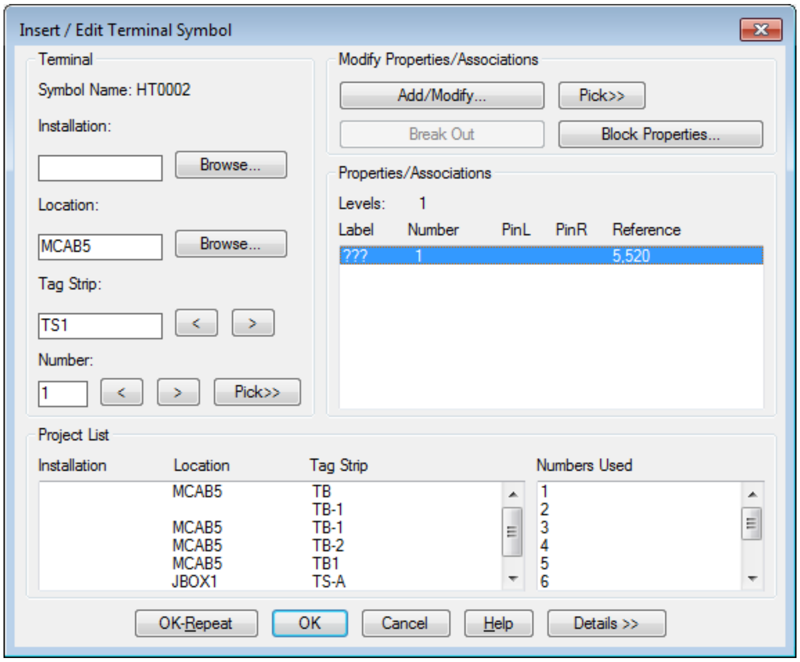


8. In the Keep dialog box, select Keep this one.

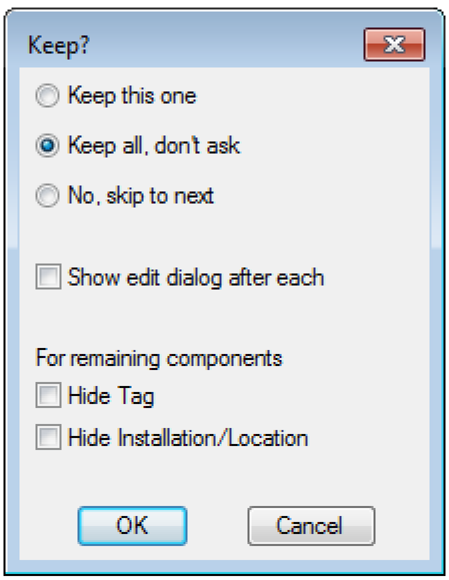
   Click OK.

9. In the Insert/Edit Terminal Symbol dialog box, Terminal section, specify:

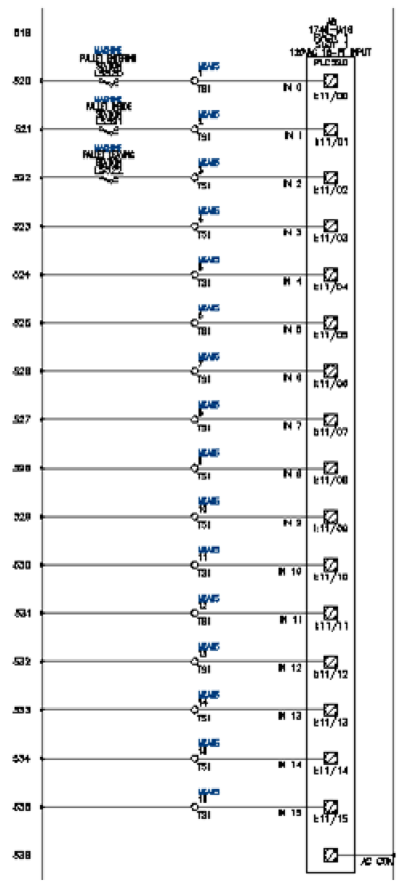
   Location: MCAB5
   Tag Strip: TS1
   Number: 1



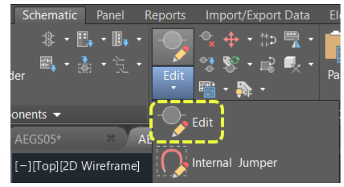
10. Click OK.
11. In the Keep dialog box, select as indicated:



12. Click OK.



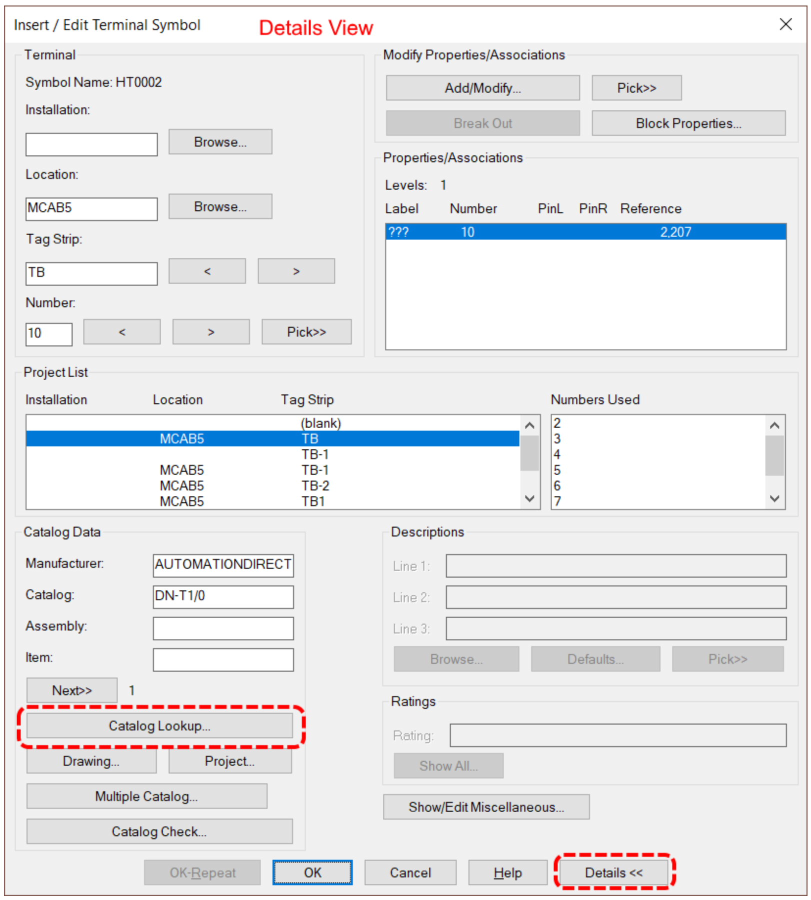
Multi-Level Terminals



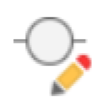
Define a **multi-level terminal** by selecting an appropriate catalog value.

Multi-level terminals

1. In the Project Manager, Project Drawing List, double-click AEGS02.dwg.
2. Click Schematic tab ➤ Edit Components panel ➤ Edit Components drop-down ➤ Edit.
3. Select the round terminal on rung 217. The Insert/Edit Terminal Symbol dialog box displays, where you can annotate the terminal properties and associations.
4. In the Insert/Edit Terminal Symbol dialog box, Project List section, select Tag Strip TB.
5. Enter **Location:** MCAB5 and **Number:** 10.



6. Click Details >>.
7. In the Catalog Data section, click Catalog Lookup.
8. On the Catalog Browser dialog box, enter the search string SIEMENS MULTI-LEVEL 20AMPS.
9. Click the search icon. 
10. Select part 8WA1 011-3JF16 and click OK. The Manufacturer and Catalog information for the selected part displays in the Catalog Data section of the Insert/Edit Terminal Symbol dialog box.
11. On the Insert/Edit Terminal Symbol dialog box, click OK.
12. Click Schematic tab ➤ Edit Components panel ➤ Edit Components drop-down ➤ Edit.
13. Select the middle terminal between rungs 217 and 218. The Insert/Edit Terminal Symbol dialog box displays.
14. In the Insert/Edit Terminal Symbol dialog box, Project List section, select Tag Strip TB.
15. Enter **Location:** MCAB5 and **Number:** 11.

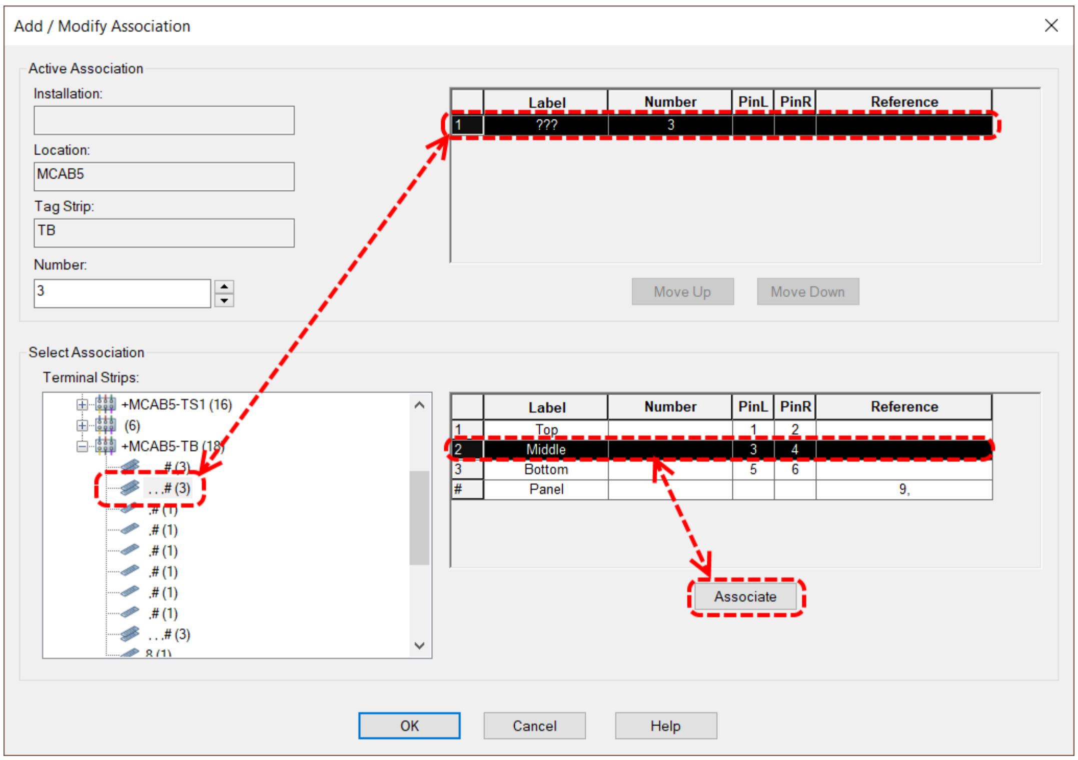


Modify Multi-Level Associations

Use the Add/Modify Association dialog box to assign terminals to specific levels of a multi-level terminal.

Modify multi-level terminal associations

1. On the Insert/Edit Terminal Symbol dialog box, Modify Properties/Associations section, click **Add/Modify**.
2. On the Add/Modify Association dialog box, Select Association section, expand the active project node. The active node is bold in the list.

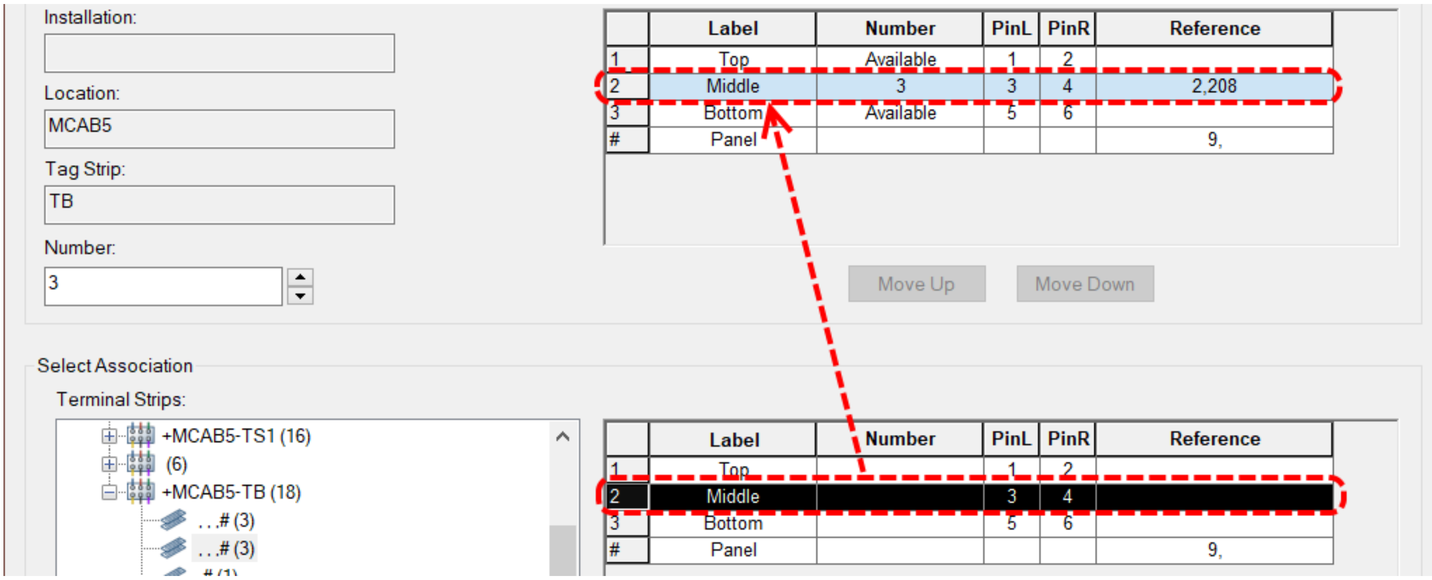


3. Select the terminal block node you inserted on line reference 217 (10, , (3)).

   The terminal numbers defined on the block are listed, separated by commas. The number of levels defined in the block properties displays at the end of the node string in parenthesis. For example, 1,21,GND (3). An empty space represents a level not represented on the schematic: 1, , GND (3). A '???' represents a terminal assigned to the level, but the terminal does not have a number assignment: 1,???,GND (3).

   Note: The grid to the right populates with the definition for the selected terminal: Level 1 has Label = TOP, Number = 11, Reference = 2,218.

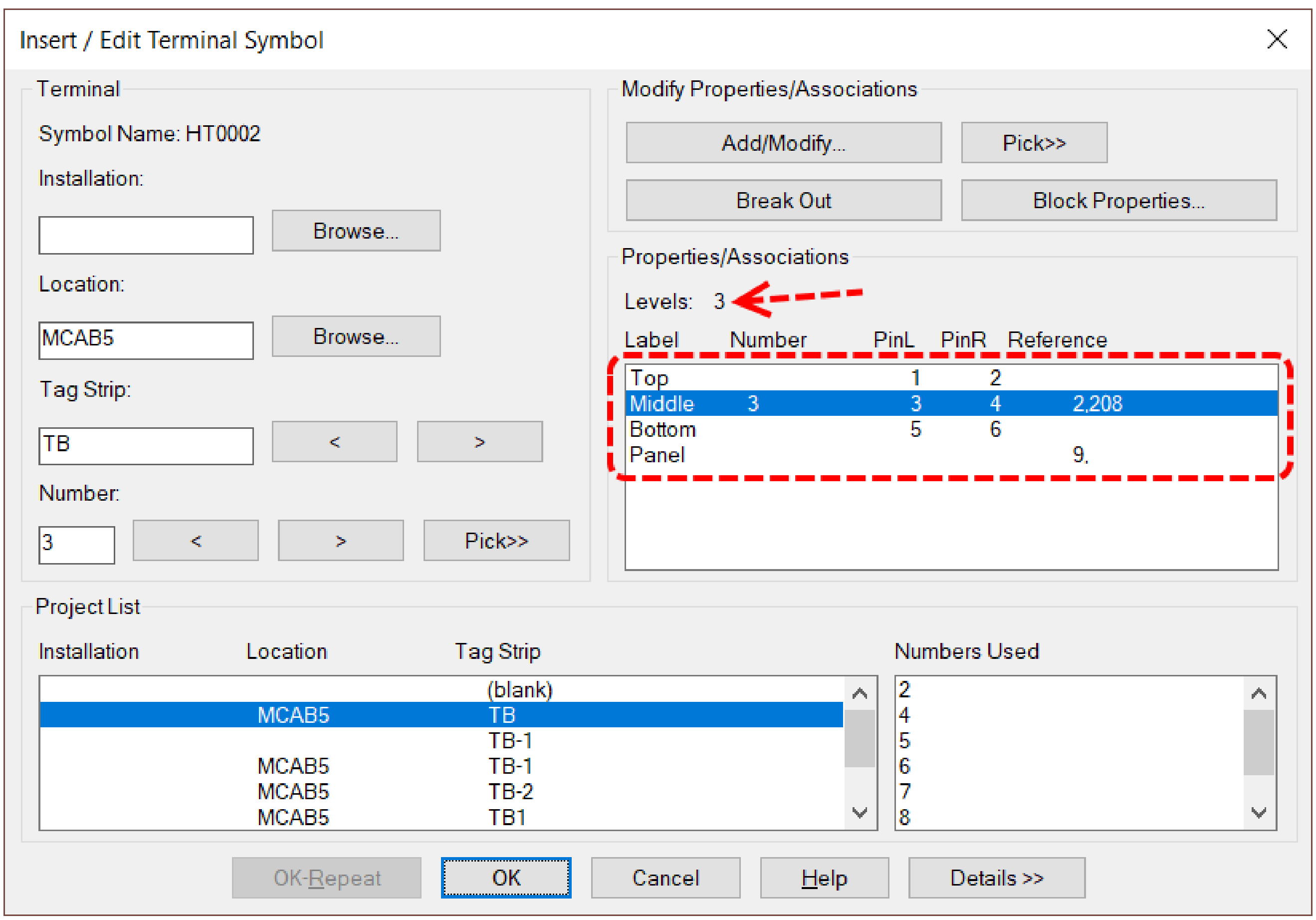
4. Select Level 2 in the grid and click Associate.



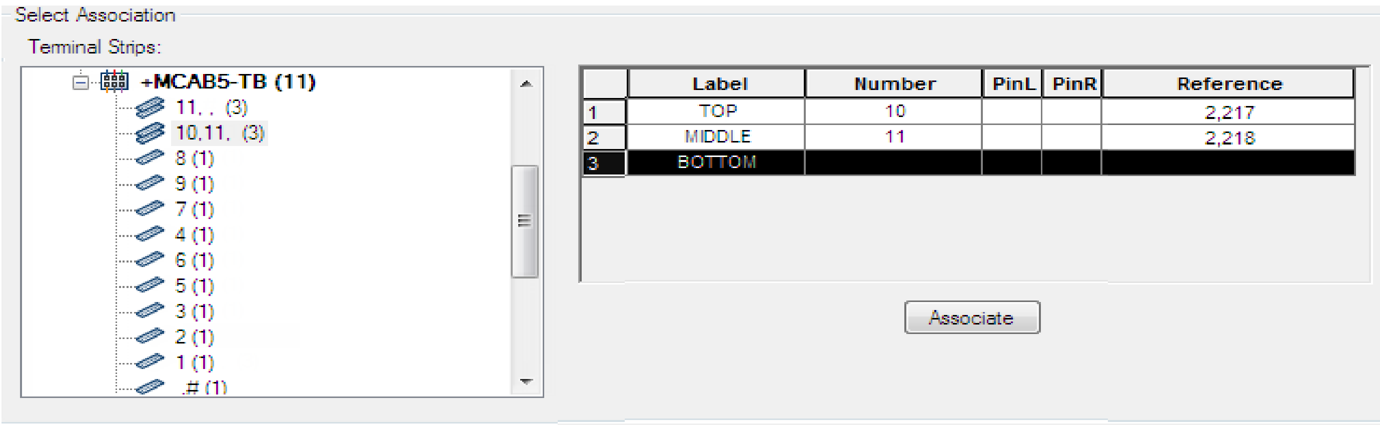
Once you click Associate, the middle level updates with the terminal number in the grid in the Active Association section of the dialog box.

5. Click OK.

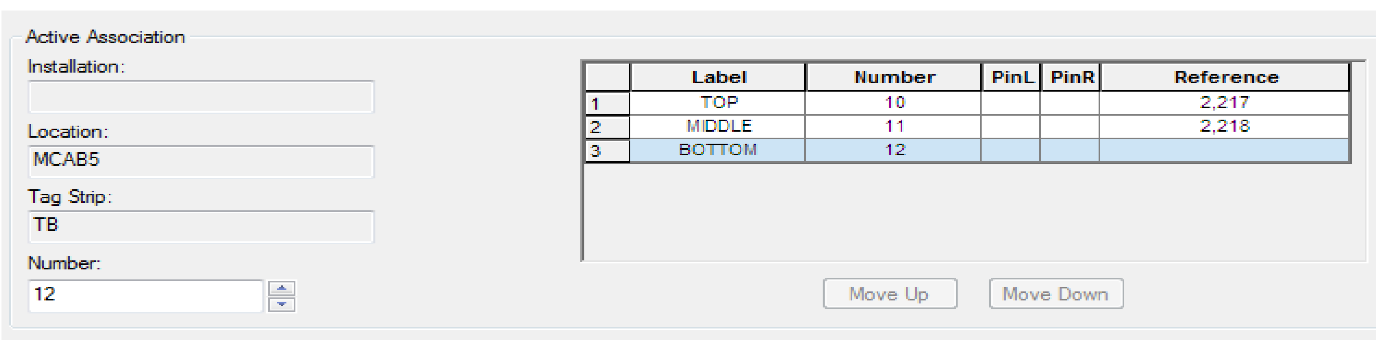
The level assignments display in the Properties/Associations section of the Insert/Edit Terminal Symbol dialog box. **Notice that the terminal is three levels and levels 1 and 2 are now assigned**.



6. On the Insert/Edit Terminal Symbol dialog box, click OK.
7. Click Schematic tab ➤ Edit Components panel ➤ Edit Components drop-down ➤ Edit.
8. Select the bottom terminal on rung 218. The Insert/Edit Terminal Symbol dialog box displays.
9. In the Insert/Edit Terminal Symbol dialog box, Project List section, select Tag Strip TB.
10. Enter **Location:** MCAB5 and **Number:** 12.
11. On the Insert/Edit Terminal Symbol dialog box, Modify Properties/Associations section, click Add/Modify.
12. On the Add/Modify Association dialog box, Select Association section, expand the active project node.
13. Select the terminal block node you inserted on line reference 217 (10,11, (3)). Notice that the node properties updated to reflect that levels 1 and 2 are assigned and that level 3 is still blank/available.
14. Select Level 3 in the grid and click Associate.

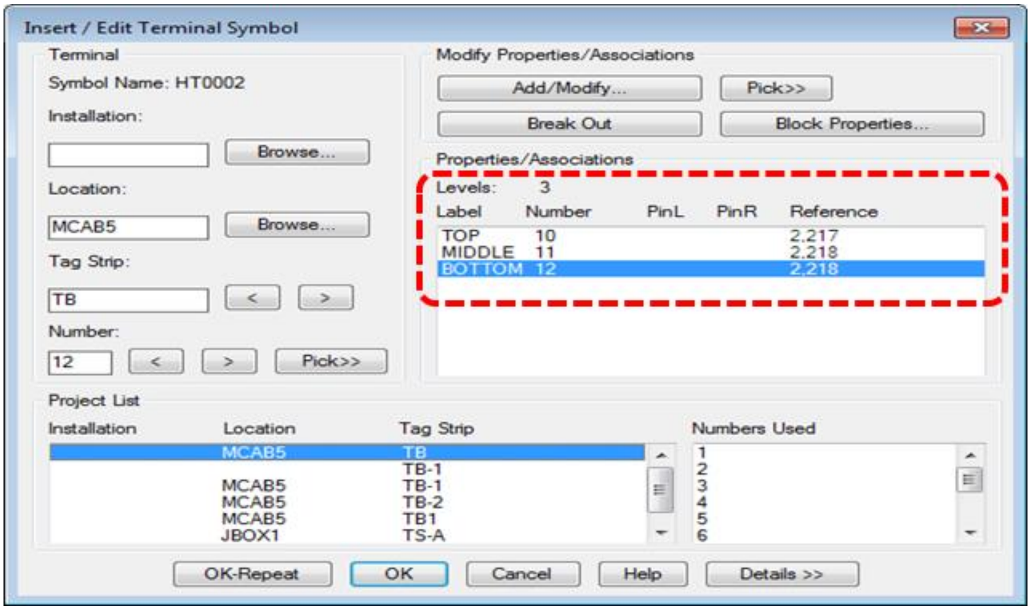


Once you click Associate, the bottom level updates with the terminal number in the grid in the Active Association section of the dialog box. You can rearrange the levels by selecting a level and clicking Move Up or Move Down.

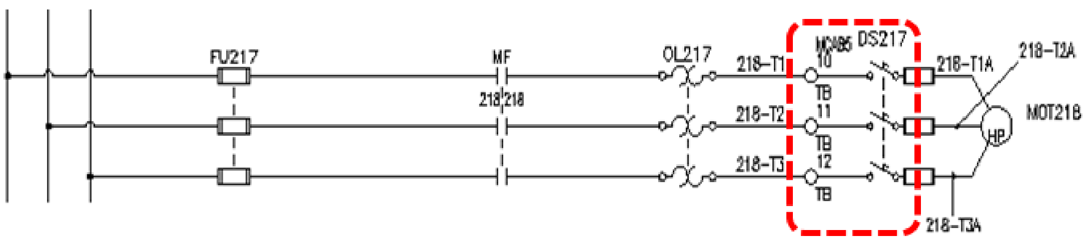


15. Click OK.

The level assignments display in the Properties/Associations section of the Insert/Edit Terminal Symbol dialog box. Notice that levels 1, 2, and 3 are now assigned.



16. On the Insert/Edit Terminal Symbol dialog box, click OK.

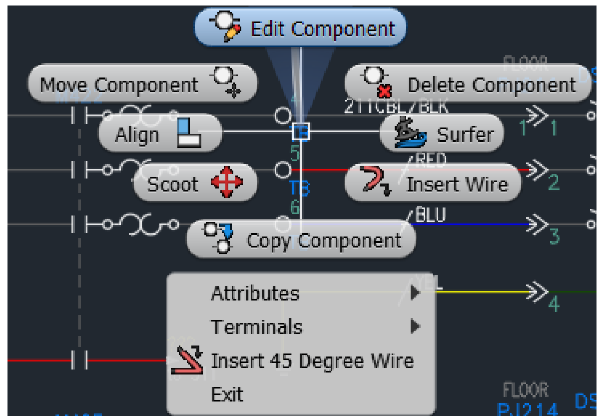


Terminal Properties

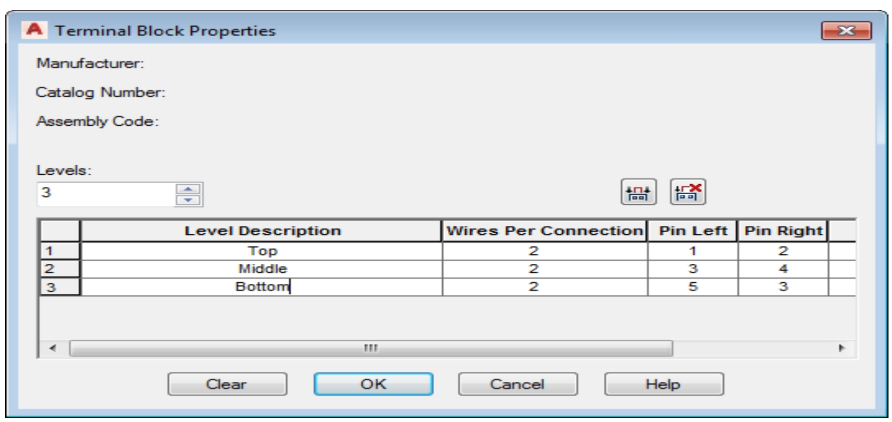
Define a multi-level terminal by assigning appropriate properties.

Modify terminal properties

1. Right-click terminal 4 on line reference 211 and select Edit Component.
2. On the Insert/Edit Terminal Symbol dialog box, Catalog Data section, delete the Manufacturer and Catalog information.
3. In the Modify Properties/Associations section, click Block Properties.
4. On the Terminal Block Properties dialog box, specify:

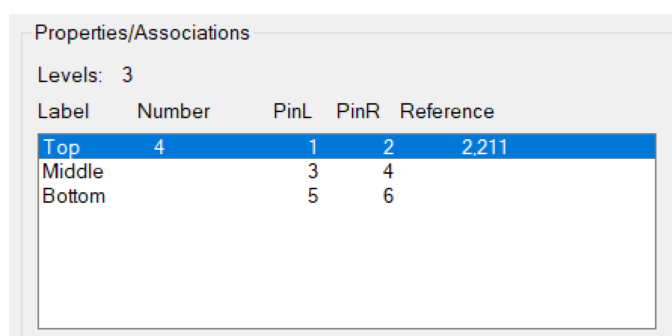


   Levels: 3
   Level 1
   Level Description: Top
   Wires Per Connection: 2
   PinL: 1
   PinR: 2
   Level 2
   Level Description: Middle
   Wires Per Connection: 2
   PinL: 3
   PinR: 4
   Level 3
   Level Description: Bottom
   Wires Per Connection: 2
   PinL: 5
   PinR: 6 (image to right has 3)



   Click OK.

   Notice on the Insert/Edit Terminal Symbol dialog box, Properties/Associations section that the block now has three levels. Terminal 4 is assigned to the top level of the block.

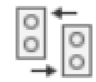


5. On the Insert/Edit Terminal Symbol dialog box, click OK.
6. On the Update other drawings dialog box, click OK.
7. If asked to save the drawing, click OK.

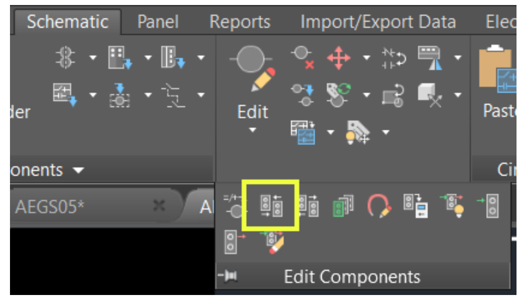
Associate Terminals

Use the Associate Terminals command to assign terminals to specific levels of a multi-level terminal.

Associate terminals



1. Click Schematic tab > Edit Components panel > > Associate Terminals.
2. Respond to the prompts as follows:

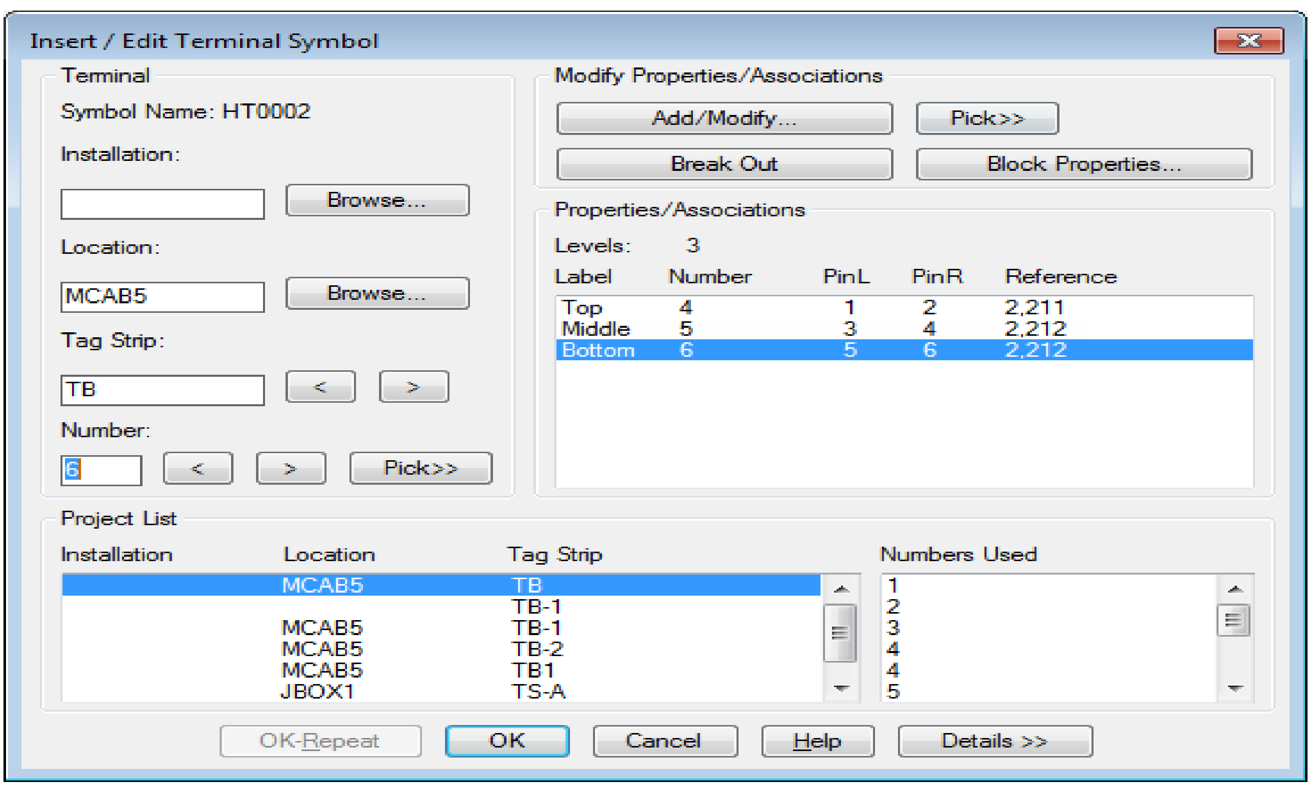


Select "Master" terminal: Select terminal 4 on line reference 211
Pick terminal: Select terminal 5
Pick terminal: Select terminal 6, right-click

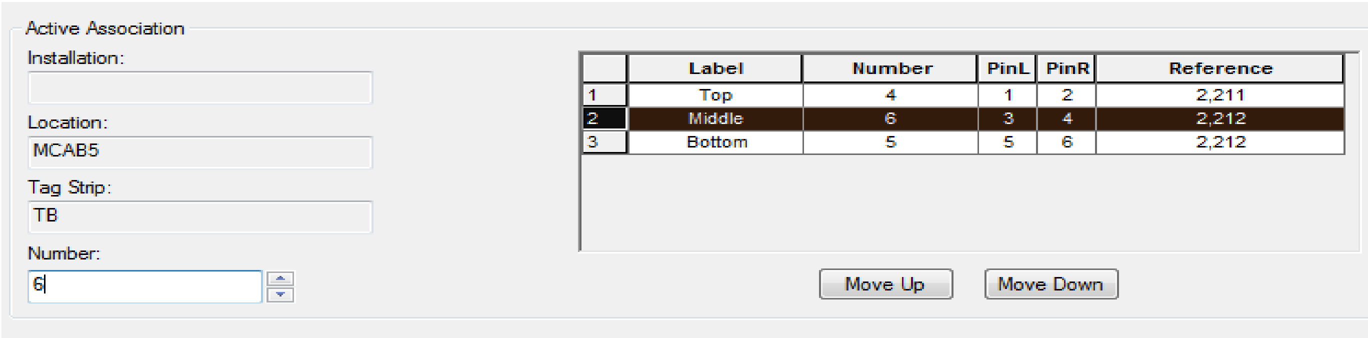
Note: The command prompt area indicates that the terminal was added as level 02 or level 03 once you pick the terminal.

3. Right-click terminal 6 and select Edit Component.

On the Insert/Edit Terminal Symbol dialog box, Properties/Associations section, all three levels have been assigned. You can now move a terminal to another level using the Add/Modify Association dialog box.



4. On the Insert/Edit Terminal Symbol dialog box, Modify Properties/Associations section, click Add/Modify.
5. On the Add/Modify Association dialog box, Active Association section, highlight level 3 in the grid and click Move Up.



The grid updates to reflect the move. Notice that terminal 6 is now assigned to level 2.

6. Click OK.
7. On the Insert/Edit Terminal Symbol dialog box, click OK.
8. If asked to update related components, click Yes-Update.

Note: If the terminals are not all on the same drawing you can associate them using the Add/Modify Association dialog box.

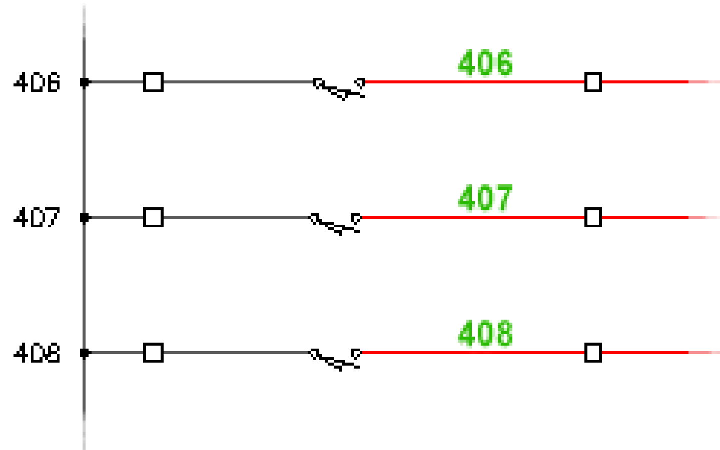
Wire Numbers Tutorial

Insert wire numbers and signal arrows.

Time required	45 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Wire numbers to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Insert wire numbers
- Insert I/O based wire numbers
- Delete wire numbers
- Understand signal arrows
- Insert a source arrow
- Insert a destination arrow



About Wire Numbers

Wire numbers can be assigned to any existing wires on an individual selection, an entire drawing, selected drawings in a project, or an entire project.

AutoCAD Electrical toolset assigns a unique wire number to each wire network.

A wire network consists of one or more wires that are electrically connected.

Inserting Wire Numbers

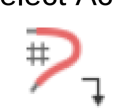
Use the Wire Numbers command to add wire numbers on individual wires, drawing-wide, and project-wide.

You can process and tag wires with sequential wire numbers or with wire numbers based upon the line reference location start of the wire network. When wire numbers are automatically inserted into a drawing, the numbers are not duplicated.

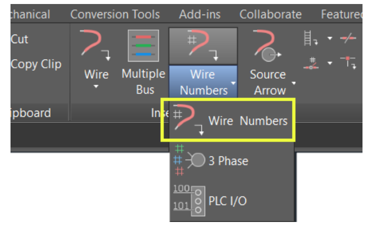
AutoCAD Electrical toolset works from left to right, top to bottom as it processes wire networks by default. You can change the direction of wire numbering using the Project Properties ➤ Wire Numbers dialog box (in the Project Manager. Right-click the project name, and select Properties. In the Project Properties dialog box, click the Wire Numbers tab).

Insert wire numbers automatically

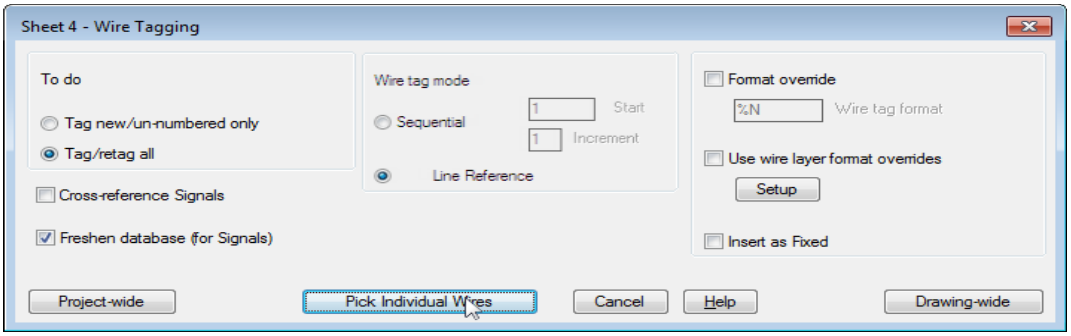
1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. In the Project Manager, Project Drawing List, double-click AEGS04.dwg.
4. Zoom in on the top portion of the wire network on the left side of the drawing.



5. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wire Numbers drop-down ➤ Wire Numbers.

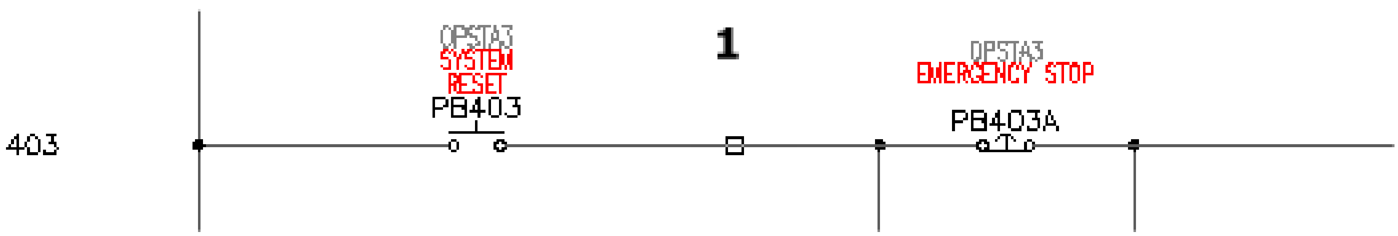


6. In the Sheet 4 - Wire Tagging dialog box, click Pick Individual Wires.

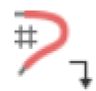


7. Respond to the prompts as follows:

   Select objects: *Select the wire segment between the two push buttons on line reference 403 (1), right-click*



   The wire number is placed.

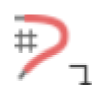


Add wire numbers to the entire drawing

1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wire Numbers drop-down ➤ Wire Numbers.
2. In the Sheet 4 - Wire Tagging dialog box, click Drawing-wide.

   Wire numbers are assigned to each segment in your drawing.

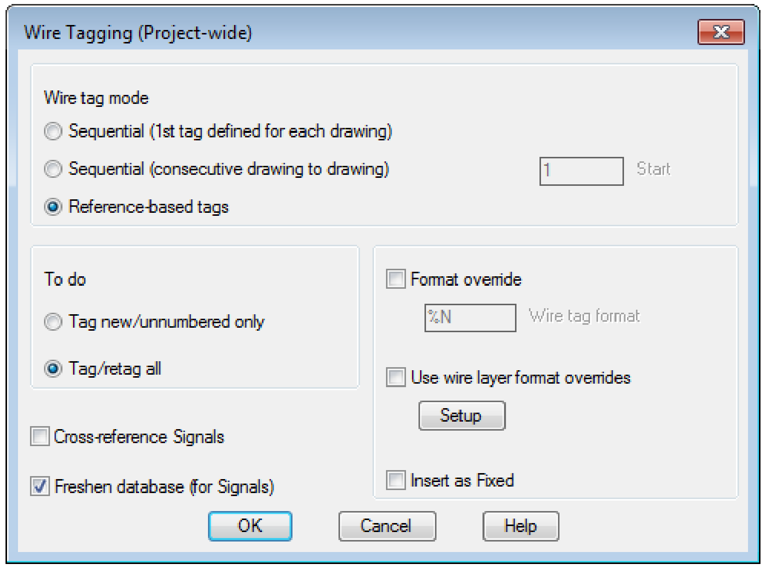
Add wire numbers project-wide

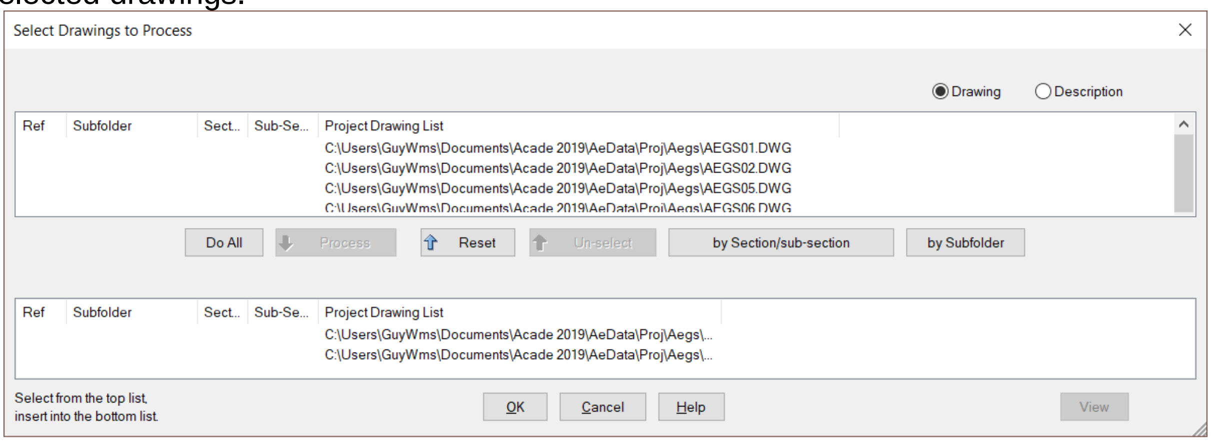


1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wire Numbers drop-down ➤ Wire Numbers.
2. In the Sheet 4 - Wire Tagging dialog box, click Project-wide.
3. In the Wire Tagging (Project-wide) dialog box, verify:

Wire tag mode: Reference-based tags
To do: Tag/retag all
Freshen database (for Signals)

4. Click OK.
5. In the Select Drawings to Process dialog box, Project Drawing List section, press SHIFT as you select AEGS03.dwg and AEGSO04.dwg. Click Process.
6. Verify AEGS03.dwg and AEGS04.dwg are listed as the drawings to process and click OK.
7. If asked to save the drawing, click OK. Wire numbers are processed for the selected drawings.





Inserting I/O Based Wire Numbers

Insert wire numbers based on the I/O address that each PLC connected wire touches.

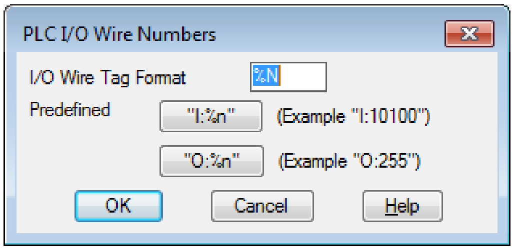
The wire numbers insert with your specified format as fixed wire numbers. If a wire number retag is run later on, **fixed wire numbers do not change**.

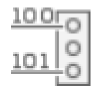
Note: If you want PLC I/O based wire numbering to be the automatic default for a drawing, set it up in the Drawing Properties dialog box. **Select the Search for PLC I/O address on insert toggle.**

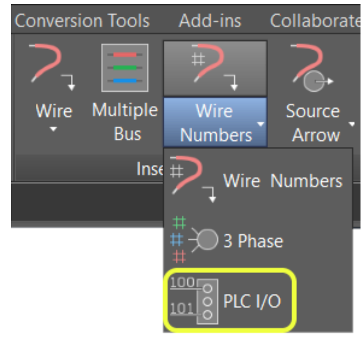
Insert PLC I/O wire numbers

1. Open *AEGS05.dwg*.

2. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wire Numbers drop-down ➤ PLC I/O. The PLC I/O Wire Numbers dialog box displays.







The default format is **%N, the address number**. The wire number is the same as its connected I/O address number.

3. Click I:%n to change the wire number format.

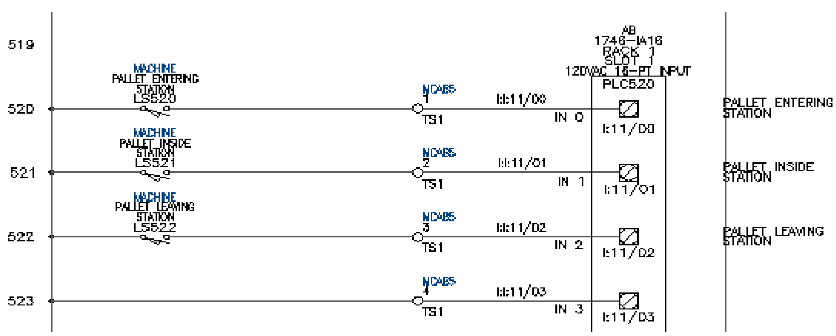
   It adds an 'I' prefix to each wire number that ties to the input module.

4. Click OK.
5. Respond to the prompts as follows:

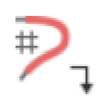
   Select I/O module to process: *Select anywhere on the PLC module*

   Select objects: *Select all the connected wires to process, right-click*

The wire numbers are inserted with the specified format. If some of the I/O points short-circuit to other I/O points, the last point wire number prevails for that common wire network.

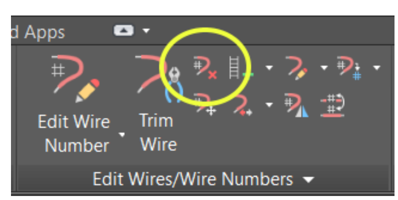


Deleting a Wire Number



Use the Delete Wire Numbers tool to remove unwanted wire numbers.

You can select a wire number or pick on any wire of the network.



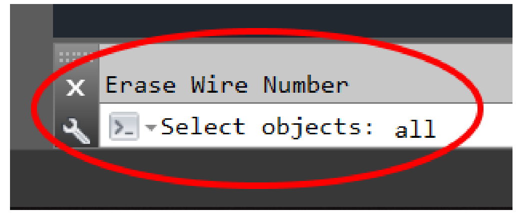
Delete a wire number

1. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Delete Wire Numbers.

2. Respond the prompts as follows:

   Select objects: *Enter* all, *press* ENTER

   The wires in the network change to dashed lines, representing the wires from which the wire numbers will be erased.

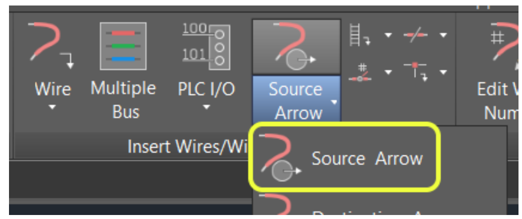


3. Press ENTER again to erase the wire numbers.

Source Signal Arrows

Insert a **source arrow** on a wire that breaks and continues in a new location.

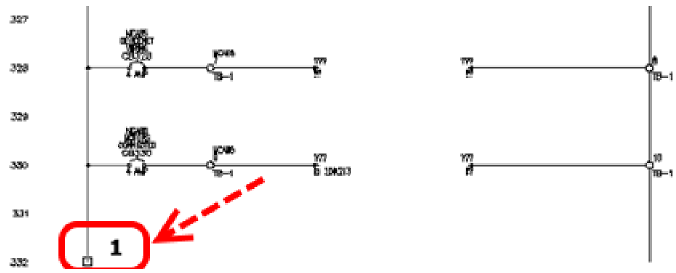
AutoCAD Electrical toolset uses a named source/destination concept. You identify a wire network to be the **source**, insert a source arrow on that network, and assign a source code name to it. On the wire network that is to be a continuation of the same wire number (whether on the same drawing or a different drawing in the project), insert a **destination arrow**. Give it the same code name that you gave to its source. It matches source code names with destination names and copies source wire numbers over to the destination wire networks.



Attach a source signal arrow

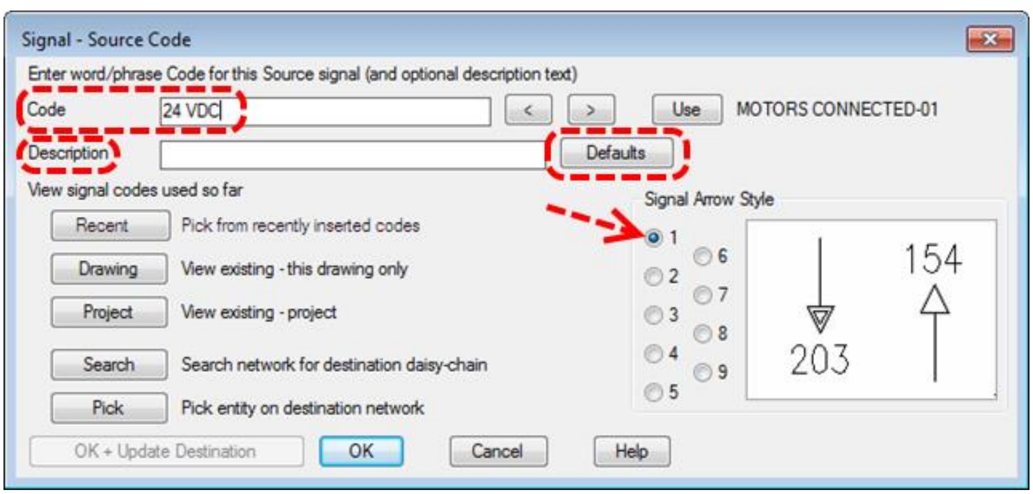
1. Open *AEGS03.dwg*.
2. Click Schematic ➤ Insert Wires/Wire Numbers panel ➤ Signal Arrows drop-down ➤ Source Arrow.
3. Respond to the prompts as follows:

Select wire end for Source: *Select the end of the hot wire on the schematic on the right side of the drawing at line reference 332 (1)*



4. In the Signal - Source Code dialog box, specify:

   Code: 24 VDC
   Signal Arrow Style: 1

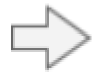


AutoCAD Electrical toolset allows one description line on a source arrow. This description can then be carried over to the associated destination arrow. You can define some default description lines to make them easier to enter without typing them in each time. AutoCAD Electrical toolset looks for a file called **WDSRCDST.WDD**. This file is a simple text file with each line being read as a separate description. If this file exists, the Defaults button is available on the Signal - Source Code and Insert Destination Code dialog boxes.

5. Click OK.
6. In the Source/Destination Signal Arrows dialog box, click No.

Note: Click No to insert the signal arrows on the next drawing. Click OK to insert the signal arrows on the current drawing.

7. To access AEGS04.dwg

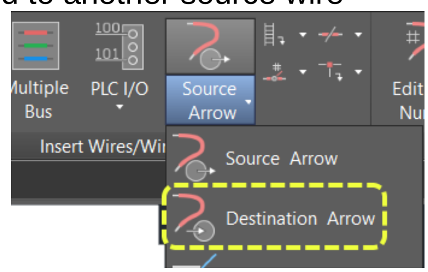
   Click Project tab ➤ Other Tools panel ➤ Next DWG. 

   Now you are ready to insert a destination signal arrow. (below)

Destination Signal Arrows

Attach a destination signal to a wire segment of a wire network and relate it to the source signal arrow.

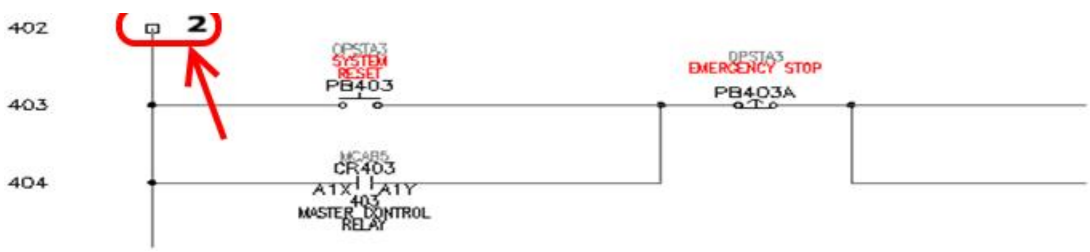
After the source signal arrow is attached to a wire in the drawing, you can attach a destination signal to a wire segment of a wire network. It enables the wire number assigned to another source wire network to carry over to the current network automatically.



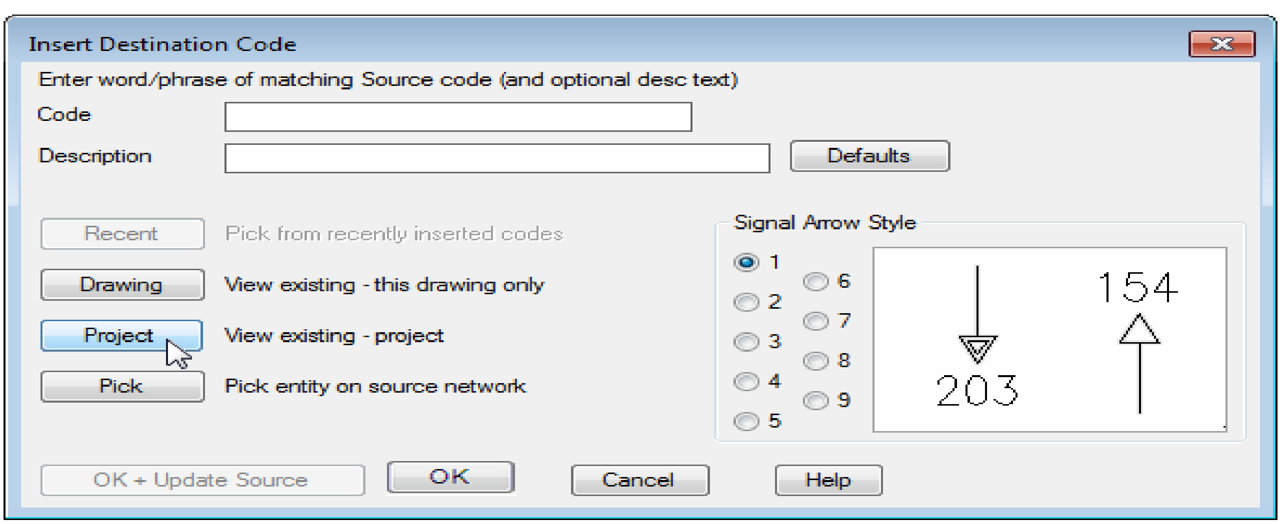
Attach a destination signal

1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Signal Arrows drop-down ➤ Destination Arrow.
2. Respond to the prompts as follows:

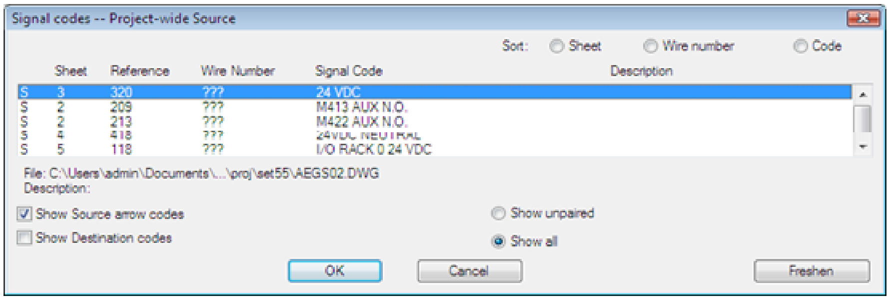
   Select wire end for Destination: *Select the top of the hot wire on the schematic on the left side of the drawing at line reference 402 (2)*



3. In the Insert Destination Code dialog box, click Project.



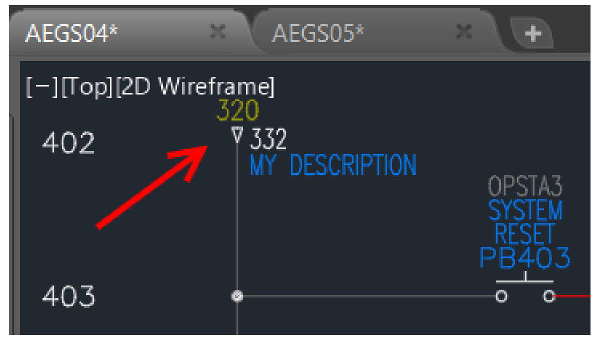
4. In the Signal codes -- Project-wide Source dialog box, select the following:



5. Click OK.
6. In the Insert Destination Code dialog box, verify:

   Code: 24 VDC
   Signal Arrow Style: 1
   Click OK + Update Source.

   The cross-references for your signal insert into the drawing above the hot wire.



Attach source and destination signals to the neutral wires.

1. To return to AEGS03.dwg

   Click Project tab ➤ Other Tools panel ➤ Previous DWG.

2. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Signal Arrows drop-down ➤ Source Arrow.
3. Respond to the prompts as follows:

   Select wire end for Source: *Select the bottom of the neutral wire at line reference 332 (3)*

4. In the Signal - Source Code dialog box, specify:

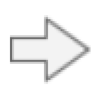
   Code: 24 VDC NEUTRAL
   Click OK.

5. In the Source/Destination Signal Arrows dialog box, click No.

   Note: Click No to insert the signal arrows on the next drawing. Click OK to insert the signal arrows on the current drawing.

6. To open AEGS04.dwg

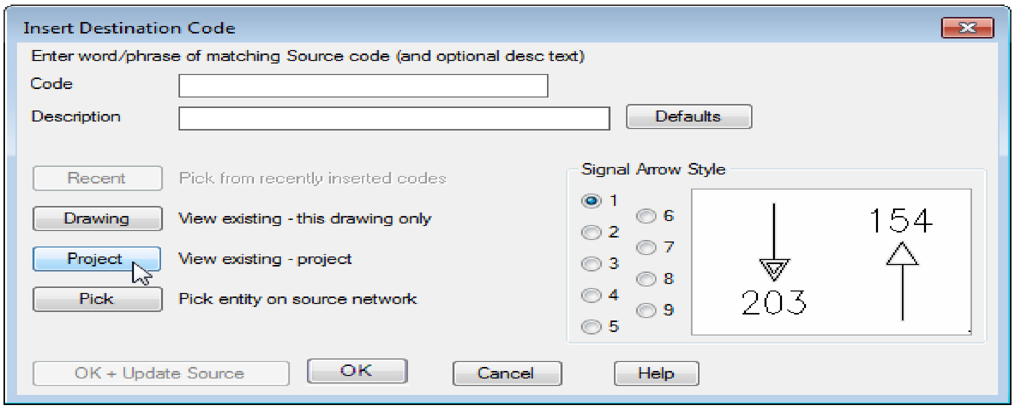
   Click Project tab ➤ Other Tools panel ➤ Next DWG.



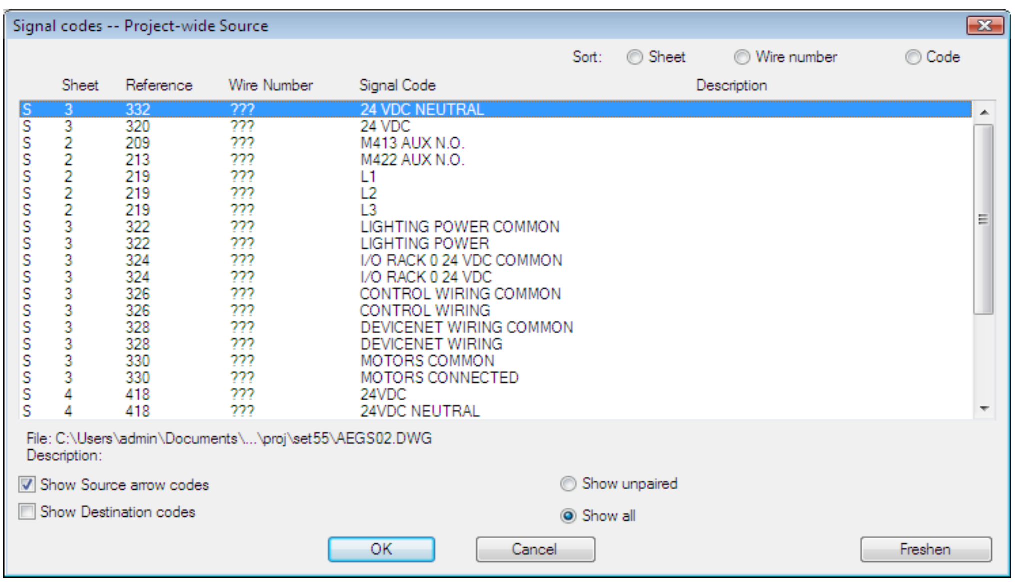
7. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Signal Arrows drop-down ➤ Destination Arrow.
8. Respond to the prompts as follows:

Select wire end for Destination: *Select the top of the neutral wire at line reference 402 (4)*

9. In the Insert Destination Code dialog box, click Project.



10. In the Signal codes -- Project-wide Source dialog box, select the following:

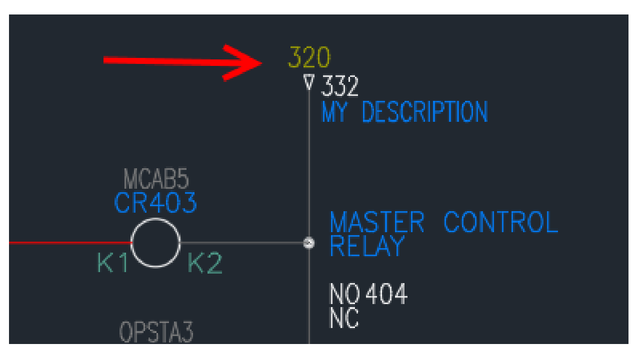


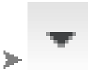
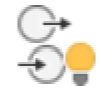
11. Click OK.
12. In the Insert Destination Code dialog box, verify:

Code: 24 VDC NEUTRAL
Signal Arrow Style: 1
Click OK + Update Source.

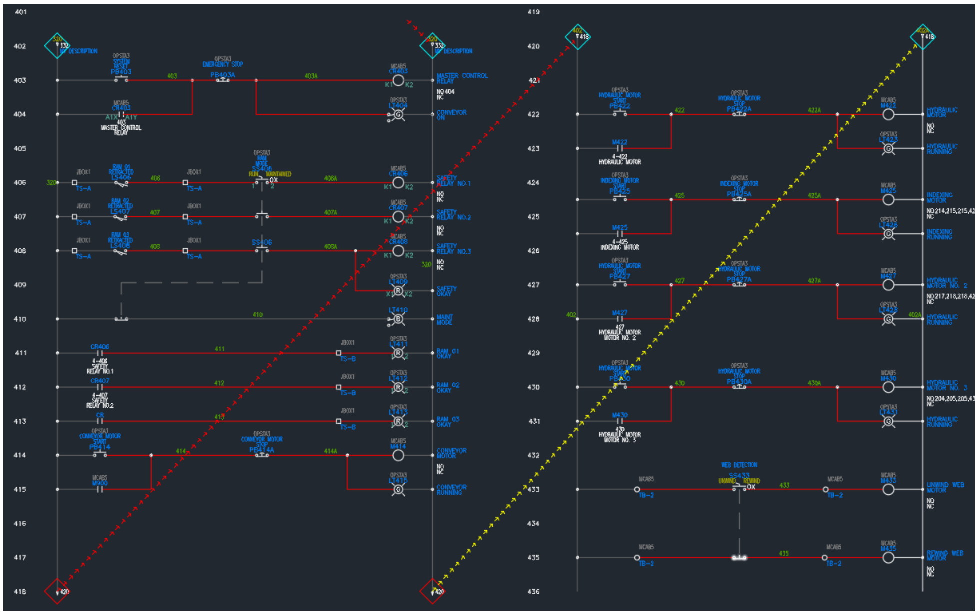
Note: If asked to change the destination wire layer, click Yes.

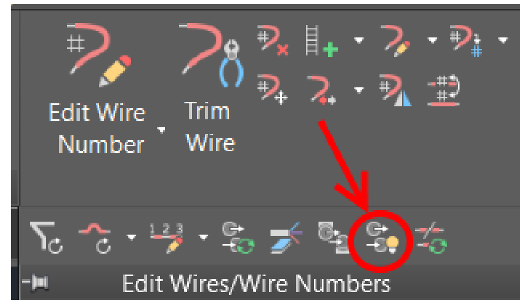
The cross-references for your signal insert into the drawing above the neutral wire.



13. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤  ➤ Show Signal Paths. 

Temporary graphics illustrate the flow of the signals on your drawings.





Note: There is no limit to the number of source and destination links you can set up. One source network can jump to multiple destinations on one or many drawings. A wire can carry both a destination signal and a source signal pointing to the next daisy-chained destination.

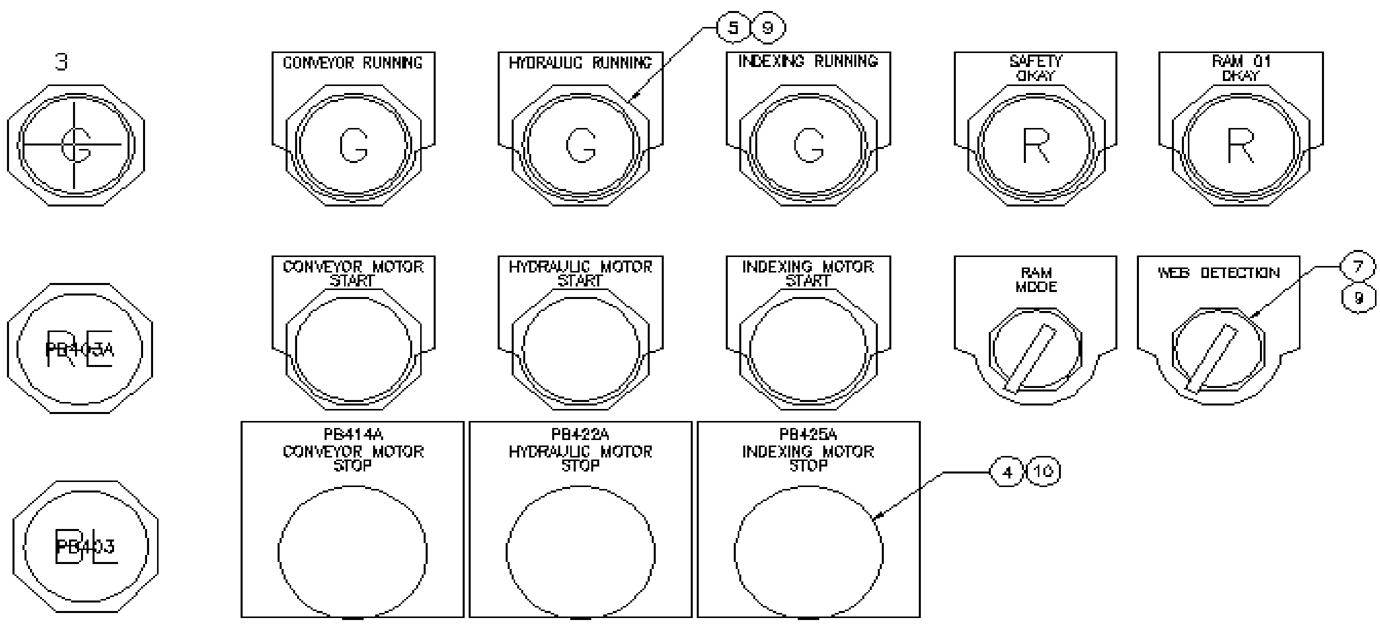
Panel Layout Tutorial

Insert and edit panel footprints. Insert and modify a graphical terminal strip with Terminal Strip Editor.

Time required	45 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Panel layout to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Insert panel footprints based on schematic components
- Insert nameplates
- Use the Terminal Strip Editor

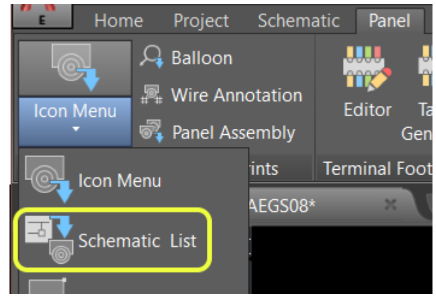


Insert Footprint (Schematic List)

Select from a list of schematic components and place the panel footprints directly into a panel layout.

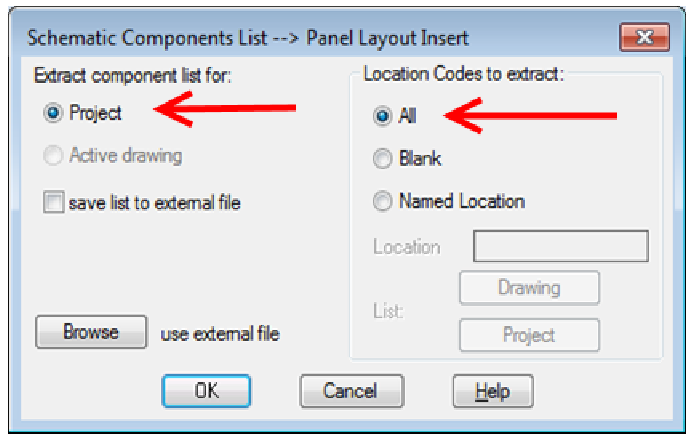
The footprint remains linked to the original schematic component, so you can perform bidirectional updating between schematic components and the associated footprint blocks.

Select schematic component footprints

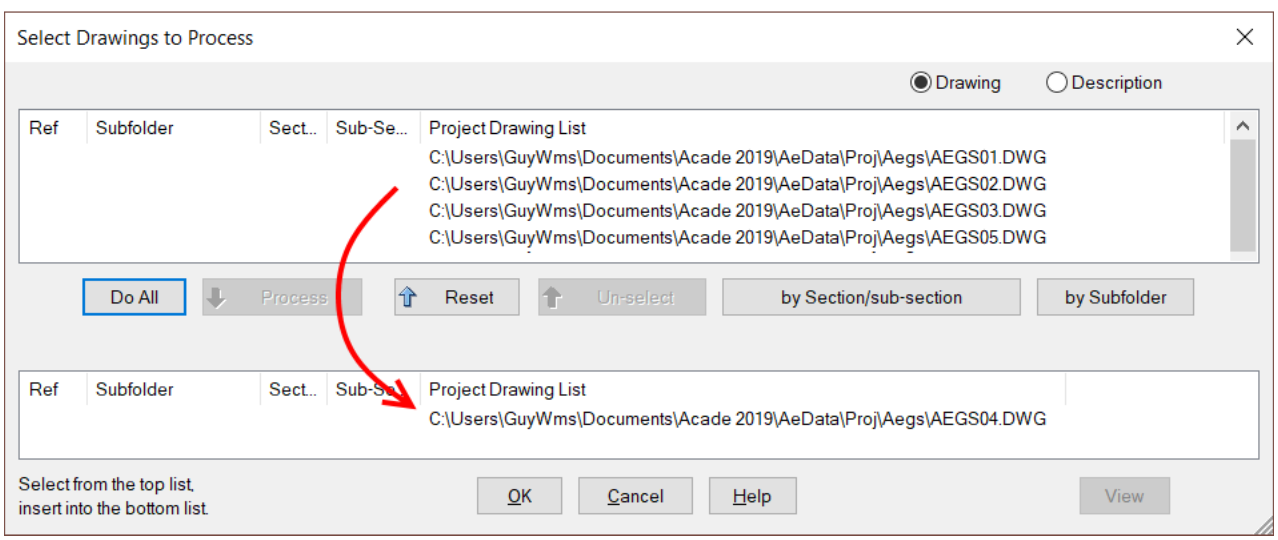


1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. Open AEGS08.dwg.
4. Click Panel tab ➤ Insert Component Footprints panel ➤ Insert Footprints drop-down ➤ Schematic List.
5. In the Schematic Component List -- Panel Layout Insert dialog box, verify:

   Extract component list for: Project
   Location Codes to extract: All

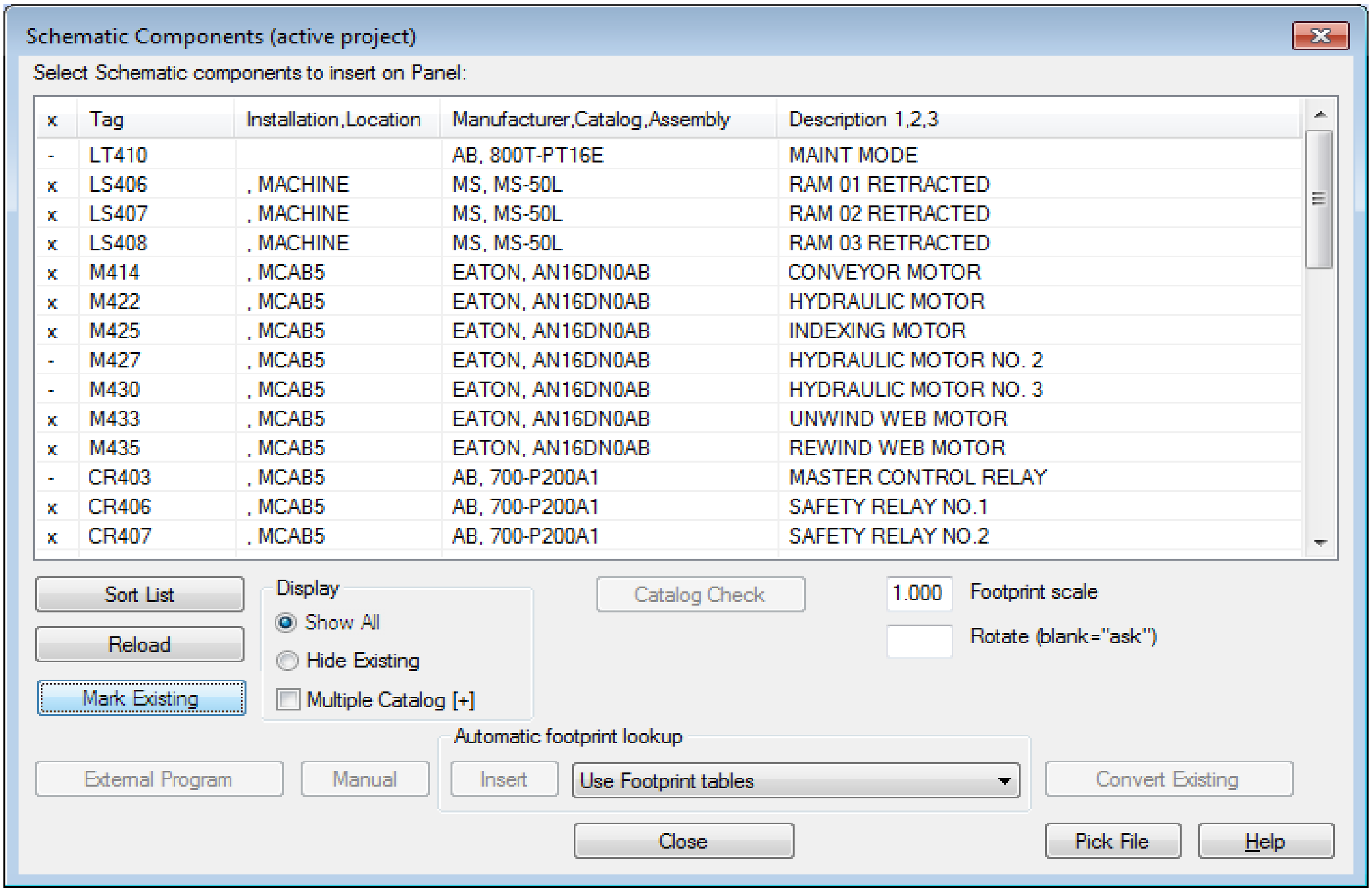


6. Click OK.
7. In the Select Drawings to Process dialog box, select AEGS04.dwg and click Process.
8. Verify that AEGS04.dwg is listed in the Drawing to Process section and click OK.
9. In the Schematic Components (active project) dialog box, click **Mark Existing**. An ***x*** marks the footprints that are already placed in the project.

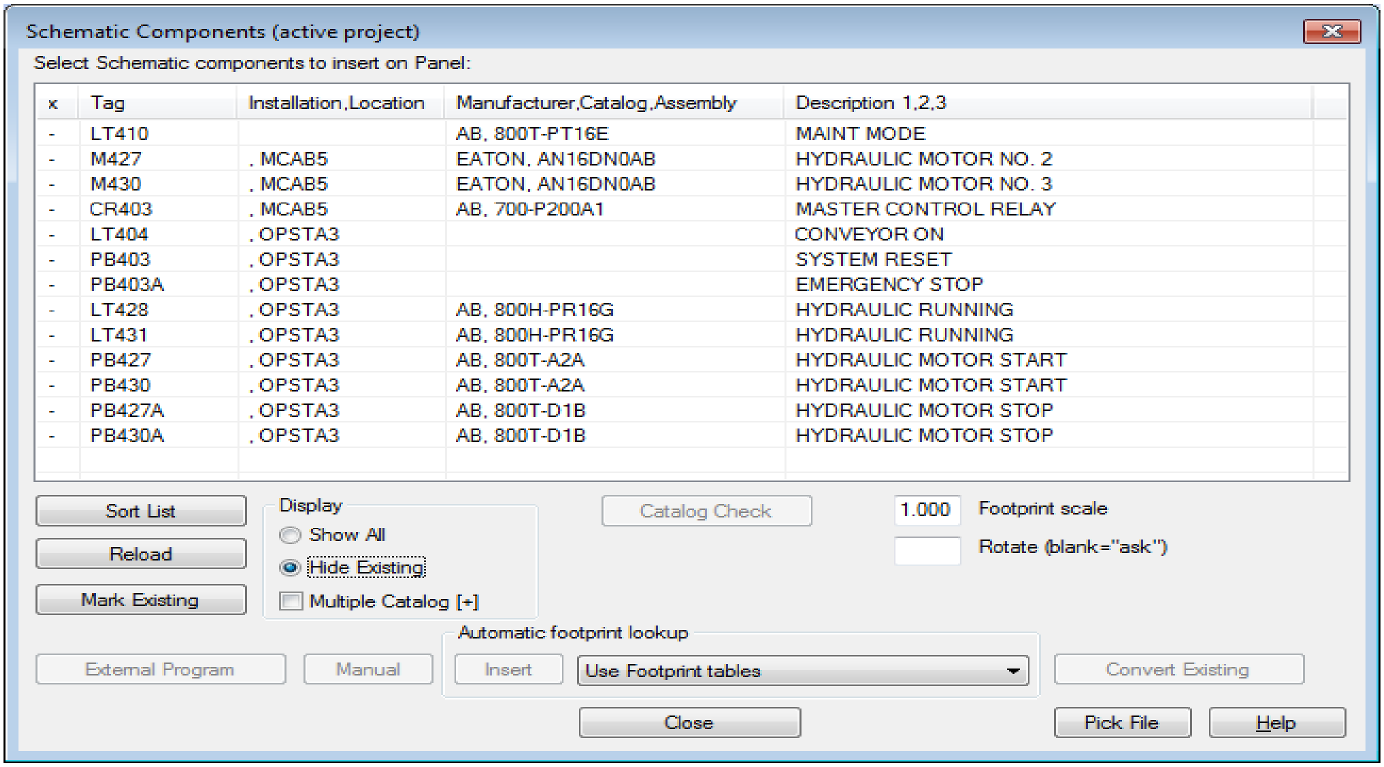


You cannot insert the same component multiple times. If you select an item with an x, the Insert button is disabled.

Note: An *o* next to a component in the list indicates that a panel component with a matching component tag was found, but the catalog information does not match.



10. In the Schematic Components (active project) dialog box, Display section, select **Hide Existing**. The schematic component footprints not yet inserted into the panel layout are displayed.

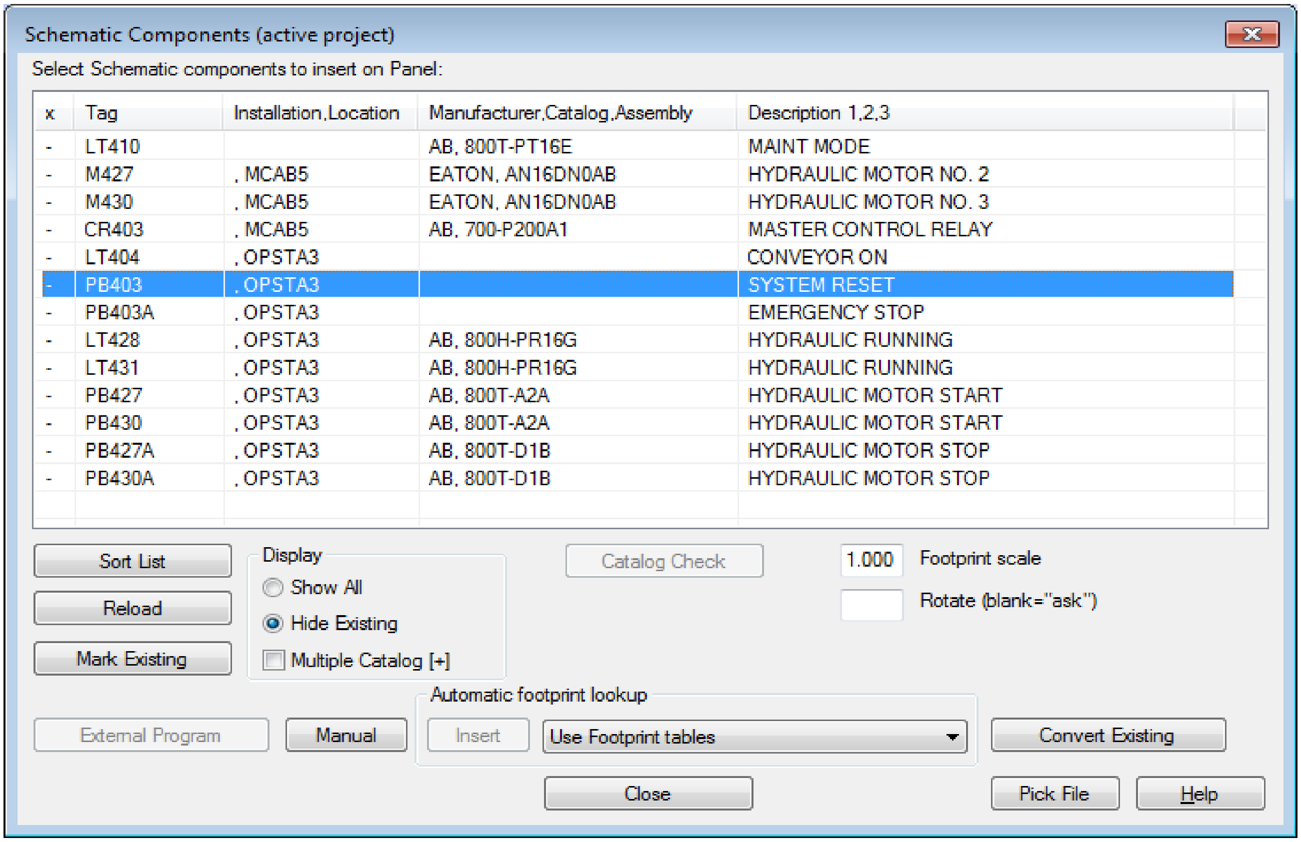


Now you can begin to insert schematic component footprints **manually** on the panel layout.

Insert the system reset footprint manually

1. In the Schematics Components (active project) dialog box, select:

   PB403 OPSTA3 SYSTEM RESET.



2. Click **Manual**.

   Note: The **Manual** button is used when schematic component footprints do not have a manufacturer and catalog number defined.

   The next step is to make a catalog assignment for the automatic footprint.

3. In the Footprint dialog box, Choice A section, click Catalog lookup.

   Note: Use Choice B to enter a graphic without selecting a catalog number.

4. On the Catalog Browser dialog box, enter the search string AB 800T.
5. Click .
6. Change the catalog assignment to:

   800T-A2A 1 NO 1 NC BLACK PUSH BUTTON - MOMENTARY, NEMA 4/13

   Click OK.

7. In the Footprint dialog box, Choice A section, verify:

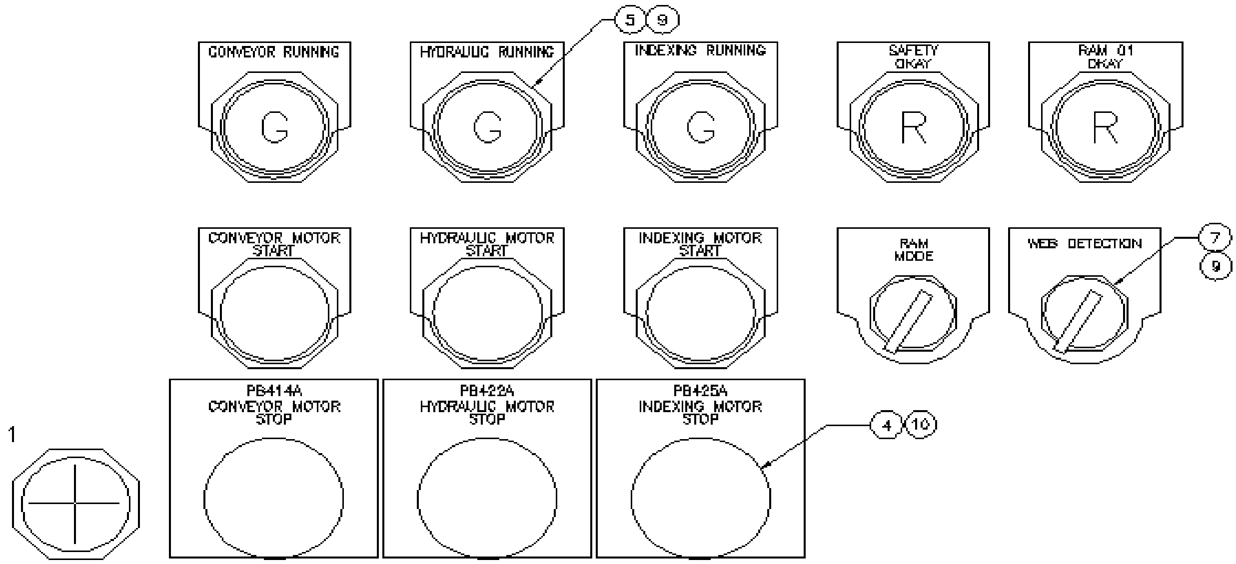
   Manufacturer: AB
   Catalog: 800T-A2A

   Click OK.

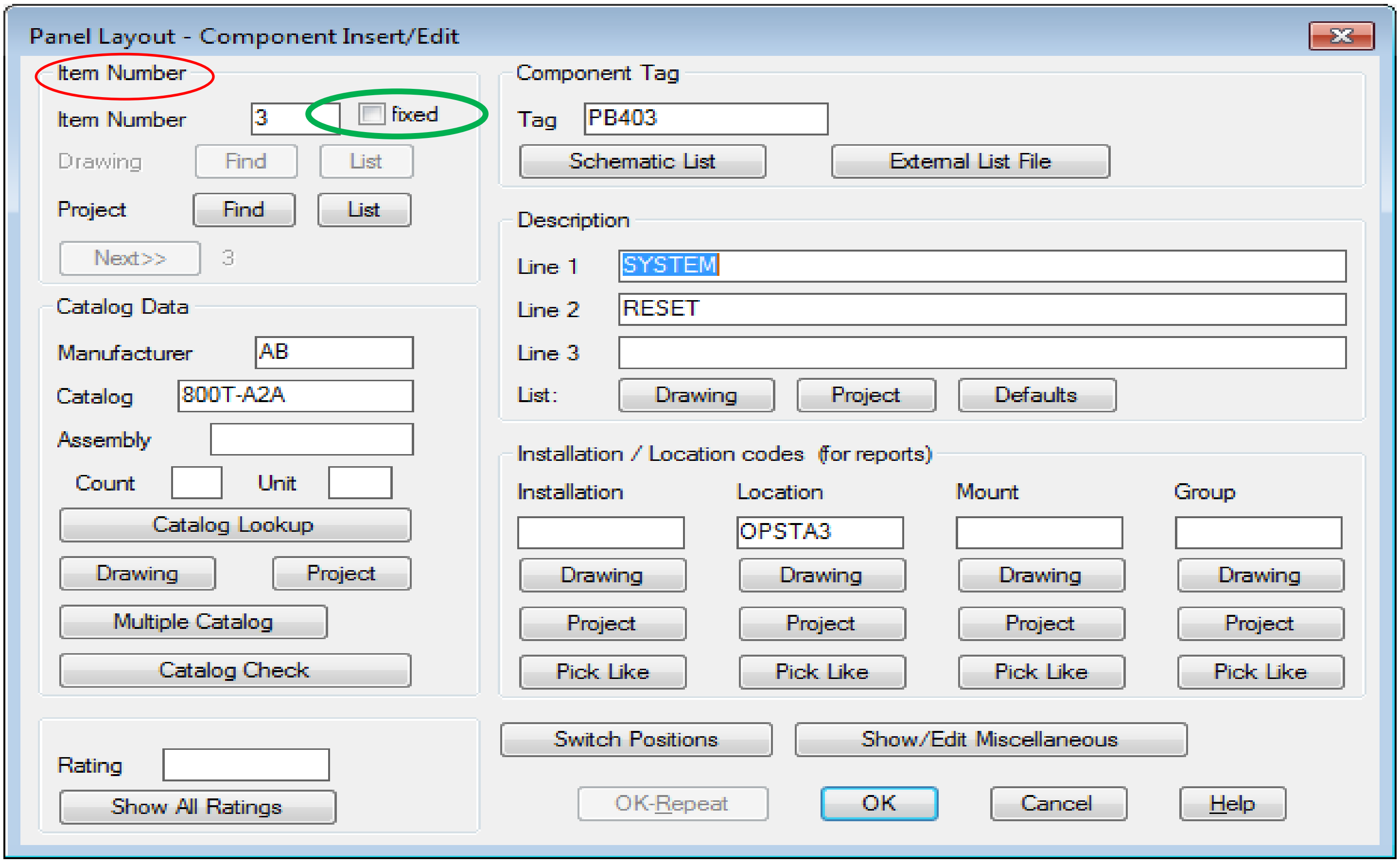
8. Respond to the prompts as follows:

Select Location for PB403: *Select to the left of PB414A (1)*

Select Location for PB403**:** <Ortho on> select ROTATION: *Right-click to place the push button*



The component may already have an **Item Number** assigned. If AutoCAD Electrical toolset finds a component with the same catalog information, it automatically assigns the same item number to this new component. If no item number is assigned, and you think a matching component exists, use one of the Find buttons to look through the drawing or project. If no matching component is found, click Next to assign an item number to this footprint. This button updates each time you insert a footprint and assign an item number. This item or detail number is used for **BOM** and component reporting and can be referenced by optional balloon labels tied to the footprint. If you do not want the item number to change if Resequence Item Numbers is run later on, check **fixed** next to the item number.



Note: The Panel Layout - Component Insert/Edit dialog box displays each time you insert a panel footprint. Information from the schematic representation is automatically carried over to the panel footprint representation.

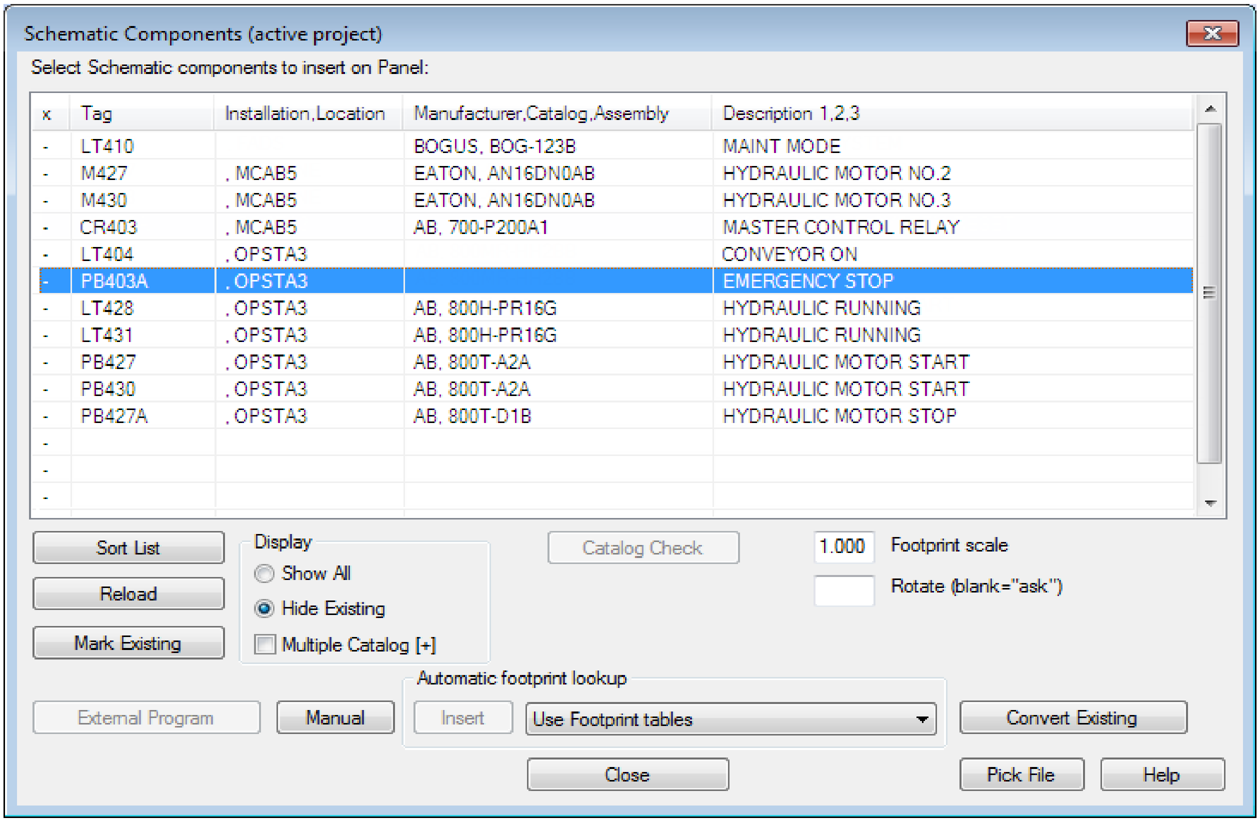
9. In the Panel Layout - Component Insert/Edit dialog box, click OK.

   The Schematics Component (active project) dialog box redisplays. You can continue inserting components from the schematic list of the project.

Insert the emergency stop footprint manually

1. In the Schematic Components (active project) dialog box, select:

   PB403A OPSTA3 EMERGENCY STOP.



2. Click Manual.
3. In the Footprint dialog box, Choice A section, click Catalog lookup.
4. On the Catalog Browser dialog box, enter the search string AB 30.5mm Red.
5. Click .
6. Change the catalog assignment to 800T-D6A 1NO-1NC PUSH BUTTON-MUSHROOM, NEMA 4/13 and click OK.
7. In the Footprint dialog box, Choice A section, verify:

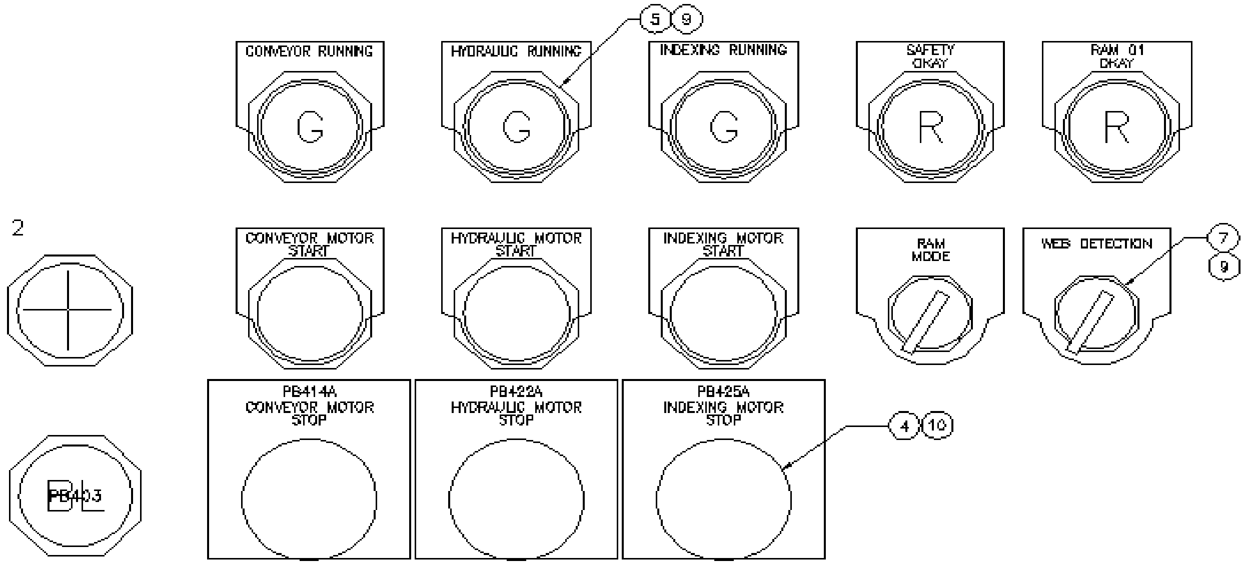
    Manufacturer: AB
    Catalog: 800T-D6A

    Click OK.

8. Respond to the prompts as follows:

    Select Location for PB403A: *Select to the left of Conveyor Motor Start (2)*

    Select Location for PB403A: <Ortho on> select ROTATION: *Right-click to place the push button*

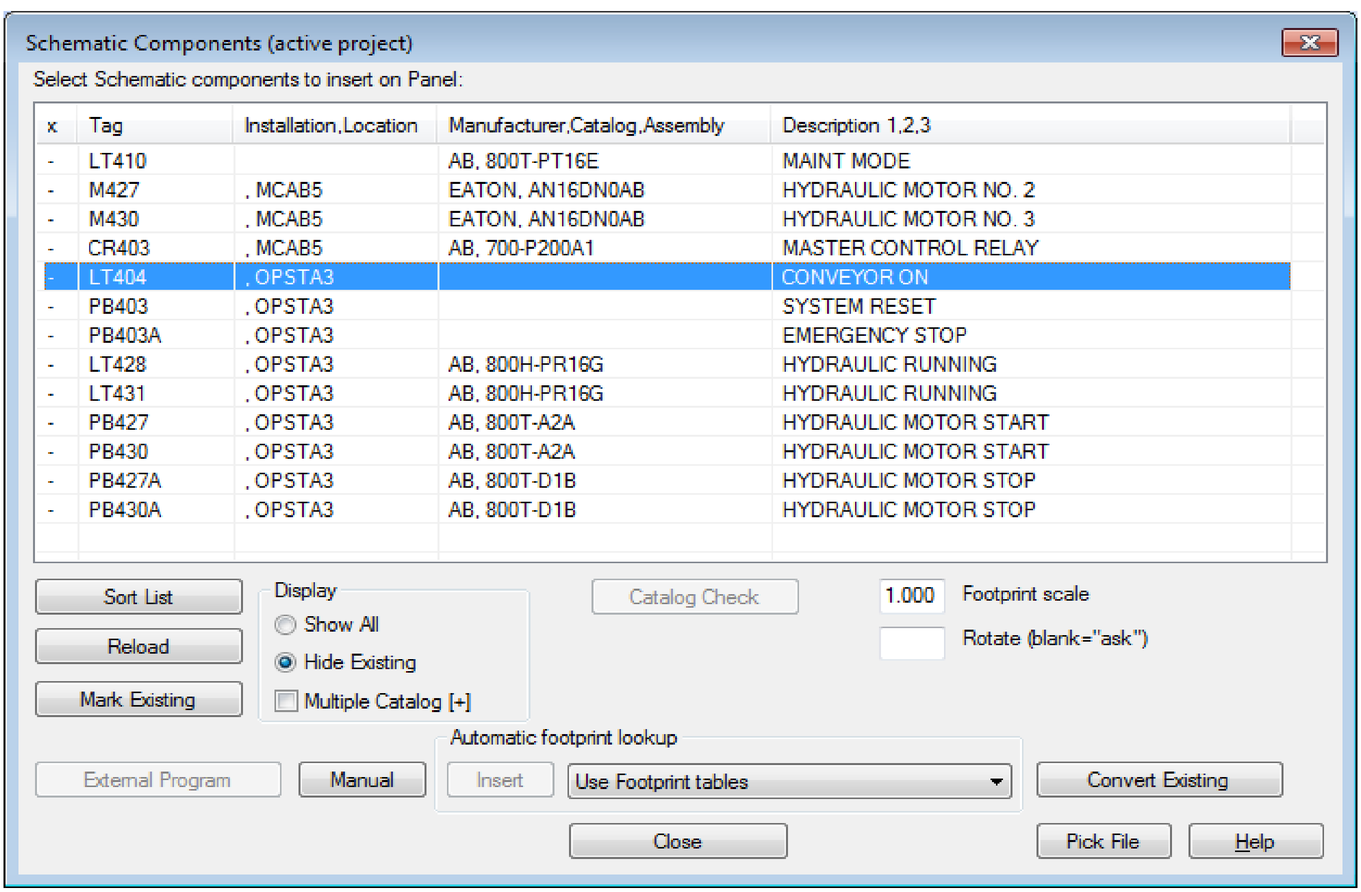


9. In the Panel Layout - Component Insert/Edit dialog box, click OK.

Insert the light footprint manually

1. In the Schematic Components (active project) dialog box, select:

   LT404 OPTSTA3 CONVEYOR ON.



2. Click Manual.
3. In the Footprint dialog box, Choice A section, click Catalog lookup.

4. On the Catalog Browser dialog box, enter the search string AB 30.5mm.
5. Click .
6. Change the catalog assignment to:

    800H-QRT24G PLASTIC LENS 24VAC/VDC FULL VOLT GREEN PILOT and click OK.

    Note: Click a column header to sort the catalog records based on the values in a specific field.

7. In the Footprint dialog box, Choice A section, verify:

    Manufacturer: AB
    Catalog: 800H-QRT24G

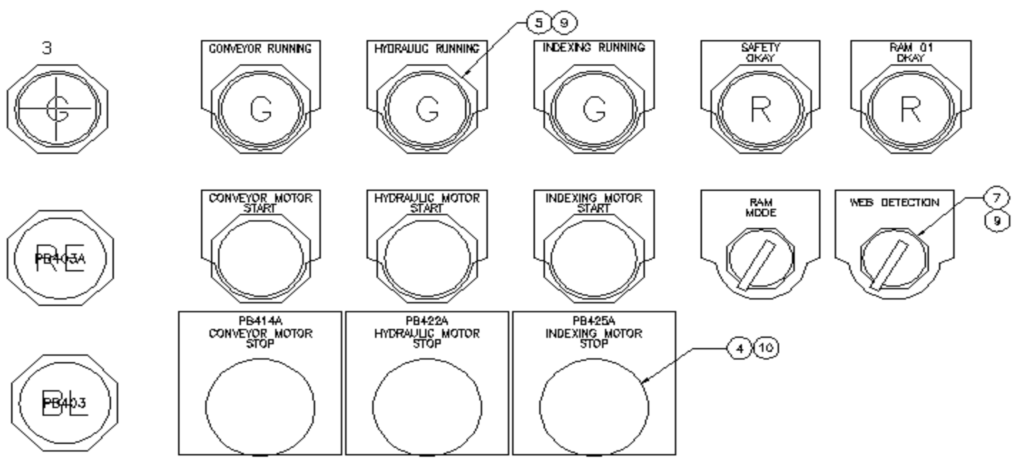
    Click OK.

8. Respond to the prompts:

    Select Location for LT404: *Select to the left of the Conveyor Running light (3)*

    Select Location for LT404: **<Ortho on>** select ROTATION:

    Right-click to place the pilot light



9. In the Panel Layout - Component Insert/Edit dialog box, click OK.

    In the Schematics Components (active project) dialog box, notice the master control relay must still be placed.

10. In the Schematic Components (active project) dialog box, click Close.

    Note: You can modify a footprint at any time using the Edit Footprint tool. Since there is bidirectional update capabilities between the schematic and the panel layout drawings, it is possible to introduce some inconsistencies between the two during edit. AutoCAD Electrical toolset alerts you to check other drawings first, and then update any affected drawings.

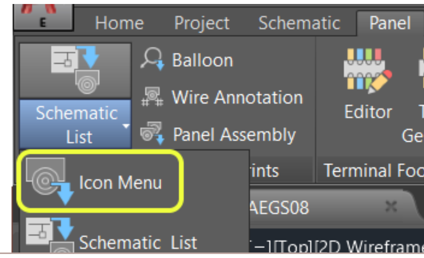
11. In the Update other drawings dialog box, click OK.
12. If asked to save the drawing, click OK.

Adding Nameplate Footprints

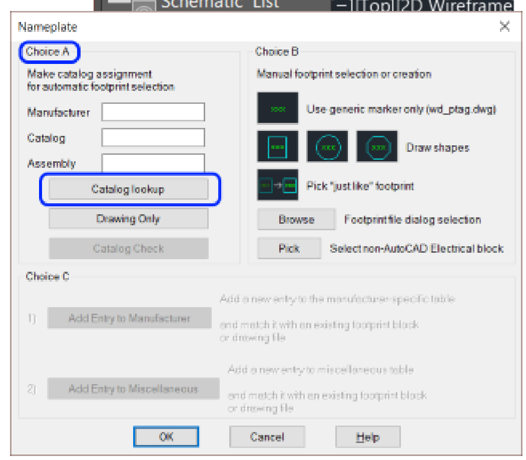
Add nameplates to the panel layout and link them to existing component footprints.

Nameplates can be inserted from the main panel icon menu or from a vendor menu.

Insert an automotive type nameplate



1. Click Panel tab ➢ Insert Component Footprints panel ➢ Insert Footprints drop-down ➢ Icon Menu.
2. In the Insert Footprint: Panel Layout Symbols dialog box, click Nameplates.
3. In the Panel: Nameplates dialog box, click Nameplate, Catalog Lookup.
4. In the Nameplate dialog box, Choice A section, click Catalog Lookup.
5. On the Catalog Browser dialog box, enter the search string AB 800T Automotive.
6. Click the search icon.
7. Change the catalog assignment to 800T-X701 Red Blank Name Plate and click OK.
8. In the Nameplate dialog box, Choice A section, verify:



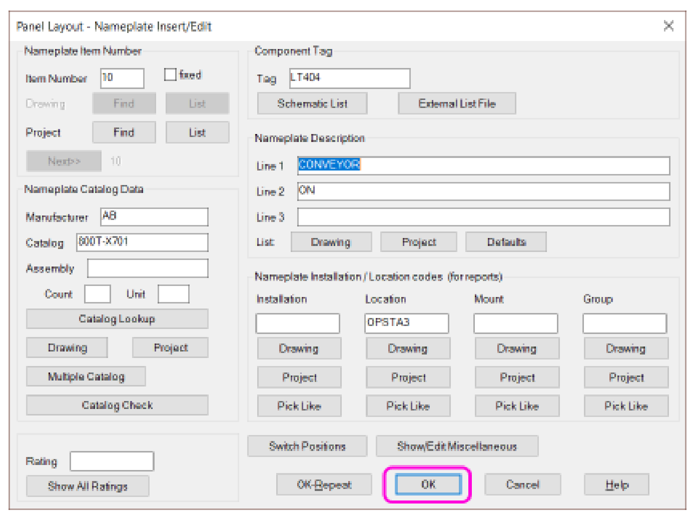
Manufacturer: AB
Catalog: 800T-X701

Click OK.

9. Respond to the prompts as follows:

   Select objects: *Select PB403 (1), right-click to the place the nameplate*

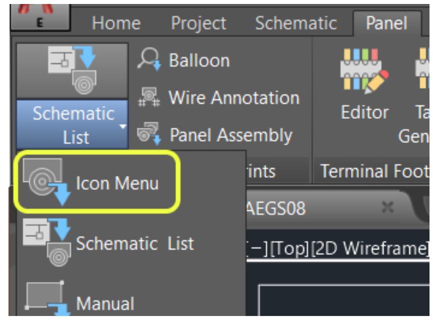
   As you select each footprint to insert, the nameplate block inserts. The Panel Layout - Nameplate Insert/Edit dialog box displays where you can annotate the nameplate and assign a BOM item number if needed.



10. In the Panel Layout - Nameplate Insert/Edit dialog box, click OK.

Note: A tag name links the data on the nameplate a tag name to the footprint and to the schematic component of the same name. Changing the tag name of any of these three representations triggers a prompt for permission to update the other related instances.

Insert a half round nameplate



1. Click Panel tab ➤ Insert Component Footprints panel ➤ Insert Footprints drop-down ➤ Icon Menu.
2. In the Insert Footprint: Panel Layout Symbols dialog box, click Nameplates.
3. In the Panel: Nameplates dialog box, click Nameplate, Catalog Lookup.
4. In the Nameplate dialog box, Choice A section, click Catalog Lookup.
5. On the Catalog Browser dialog box, enter the search string AB 800T.
6. Click search.
7. Change the catalog assignment to:
   800T-X59E Gray Custom Text Name Plate

   Click OK.

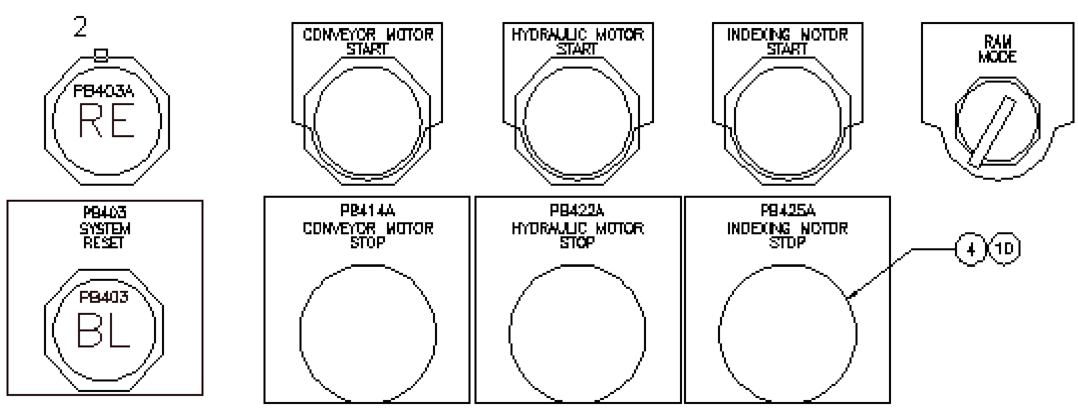
8. In the Nameplate dialog box, Choice A section, verify:

   Manufacturer: AB
   Catalog: 800T-X59E

   Click OK.

9. Respond to the prompts as follows:

Select objects: *Select PB403A (2), right-click to place the nameplate*



10. In the Panel Layout - Nameplate Insert/Edit dialog box, click OK.

    The nameplate is inserted.

Terminal Strip Editor

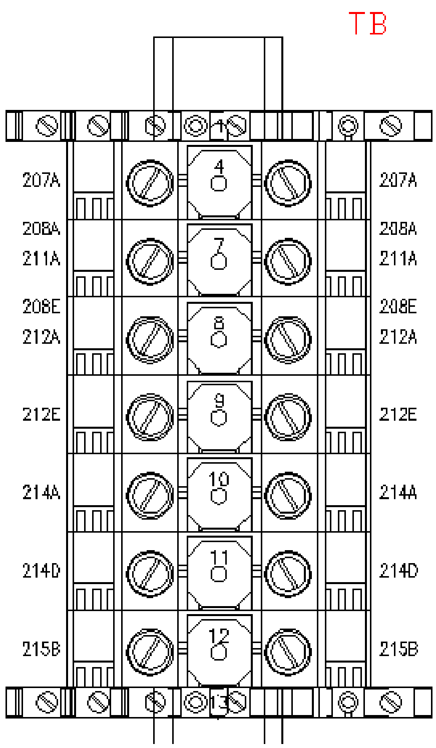
Use Terminal Strip Editor to manage terminals, edit terminal properties, associate terminals, and insert the graphical terminal strip.

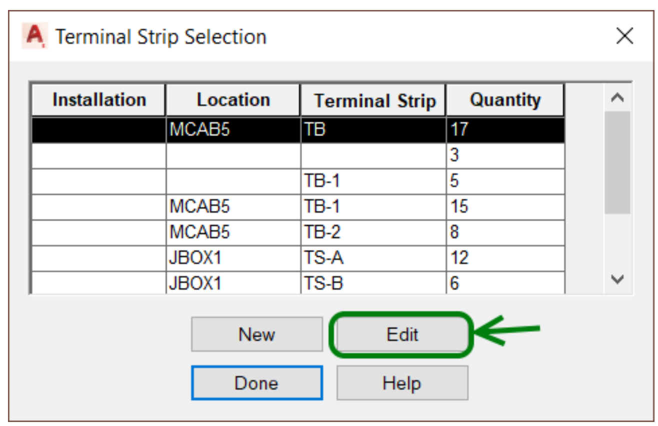
Terminal blocks connect devices that require quick disconnect or disassembly during product shipment. They can also be used to distribute power to other devices. The Terminal Strip Editor easily and quickly defines the locations for these connected devices during the system design process. Terminal strip editing is primarily used towards the end of the control system design cycle to expedite the labeling, numbering, and rearranging of terminals on a terminal strip.

Copy and paste terminal properties

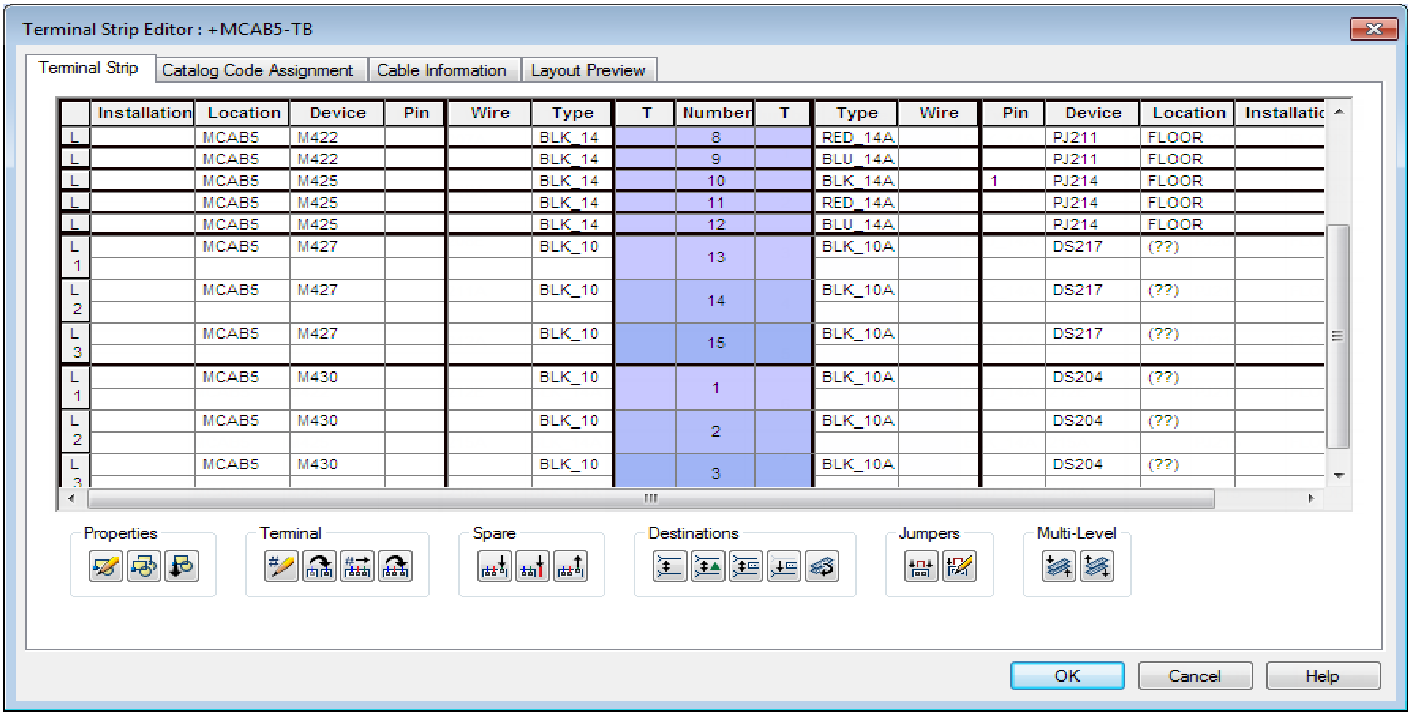
1. Open AEGS09.dwg.

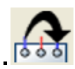
   The terminal strip to edit, “TB”, is already placed on the drawing. Zoom in on terminal strip “TB” to see what the terminal strip currently looks like.





2. Click Panel tab ➤ Terminal Footprints panel ➤ Editor.
3. On the Terminal Strip Selection dialog box, select Terminal Strip “TB” and click Edit.
4. On the Terminal Strip Editor dialog box, Terminal Strip tab, select terminal 1 in the grid.



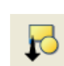
5. In the Terminal section, click the Move Terminal button. 
6. In the Move Terminal dialog box, click Pick Above. In the Terminal Strip Editor grid, select terminal 4.

   Note: You can also use the Move Up tool to move terminal 1 to the top of the grid.

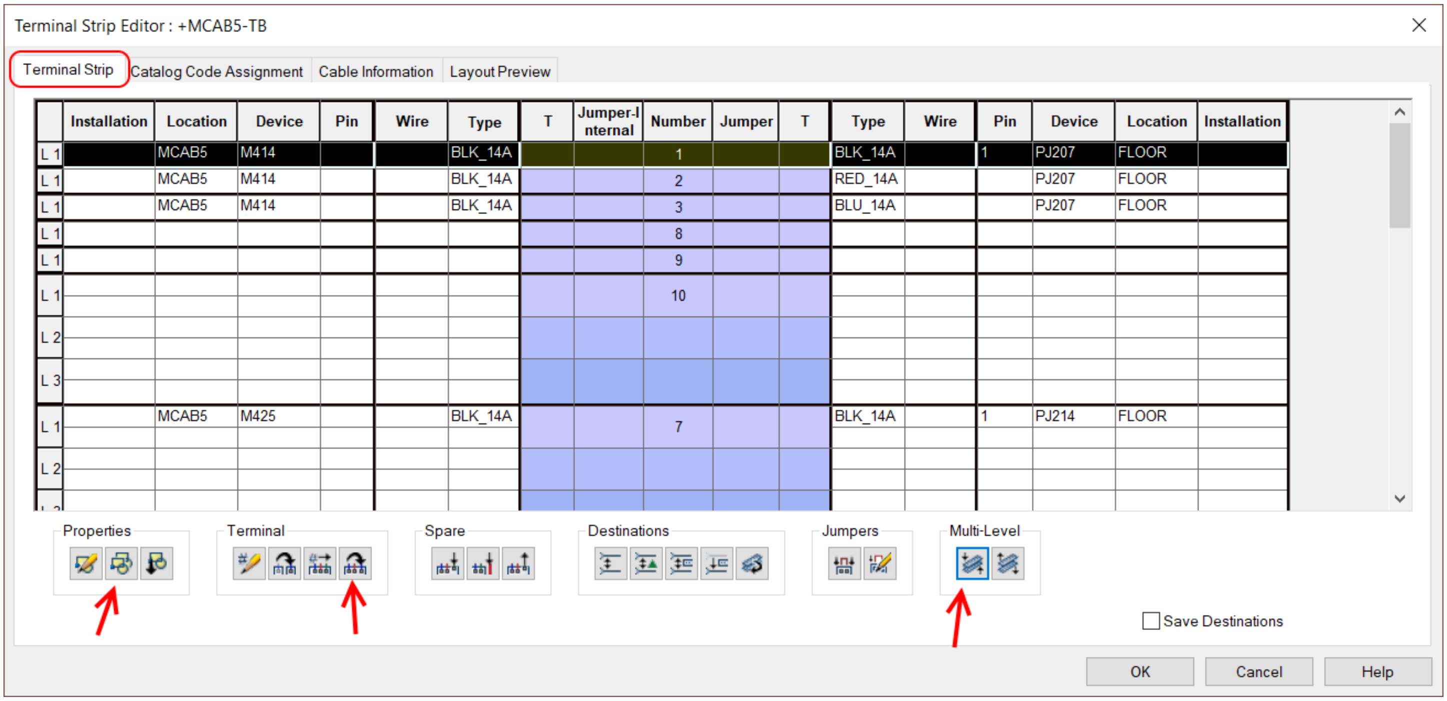
   Click Done.

7. Select terminal 4 in the grid.
8. In the Properties section, click the Copy Terminal Block Properties button. 

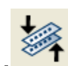
   Notice that when you click Copy Terminal Block Properties, terminals 5 and 6 also highlight. It is because terminals 4, 5, and 6 are **associated**. If you copy the properties from one of these terminals, you also copy the properties from the associated terminals. The Copy Terminal Block Properties tool then copies the properties from the terminals to one or many terminals within the same terminal strip.

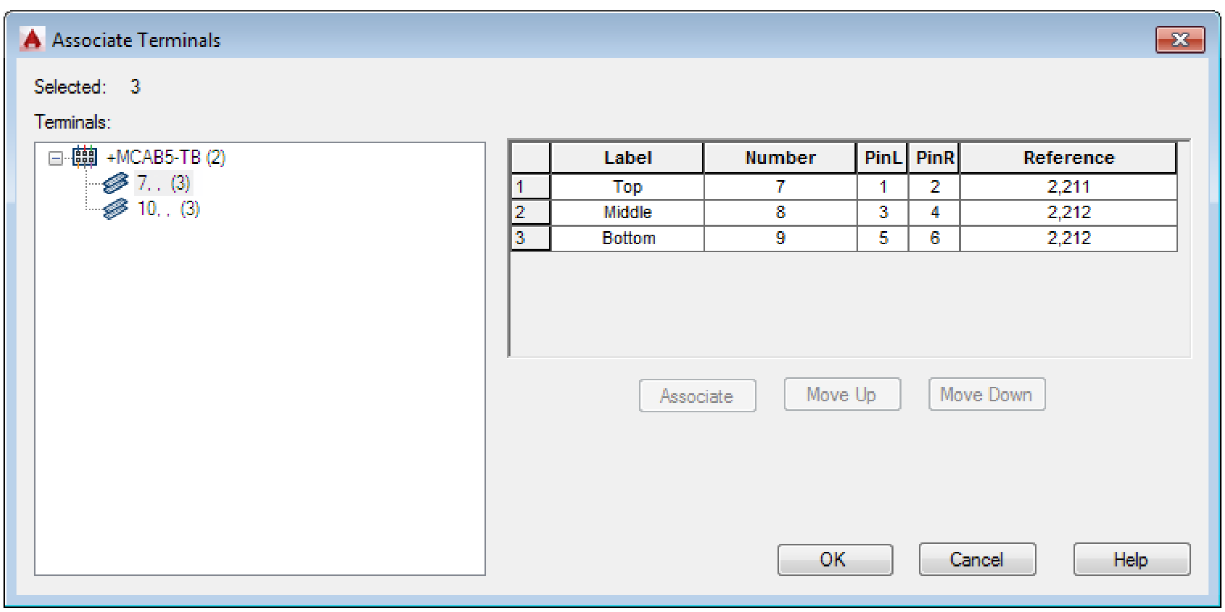
9. Select terminal 7 and 10 in the grid by holding down the CTRL key while you select the terminals.
10. In the Properties section, click the Paste Terminal Block Properties button. 

    The properties you copied from terminal 4 are pasted to terminals 7 and 10. Notice that both terminals are now 3-tiered terminals with level 1 assigned for both.

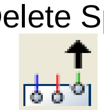


Associate terminals

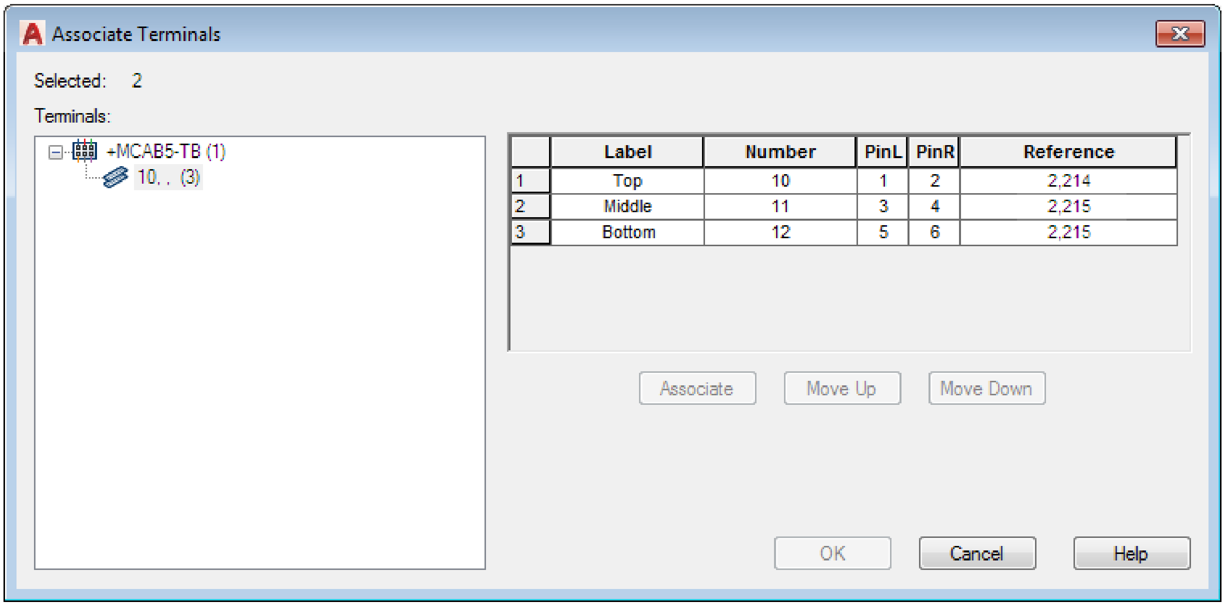
1. Select terminals 8 and 9 in the grid.
2. In the Multi-Level section, click the **Associate Terminals** button. 
3. On the Associate Terminals dialog box, select terminal 7, , (3) and click Associate.



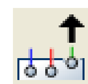
Click OK.

4. In the Spare section, click Delete Spare Terminals/Accessories to remove the blank terminals resulting from the Associate. 
5. On the Terminal Strip Editor dialog box, select terminals 11 and 12 in the grid.
6. In the Multi-Level section, click the Associate Terminals button. 

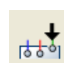
7. On the Associate Terminals dialog box, select terminal 10, , (3) and click Associate.



Click OK.

8. In the Spare section, click Delete Spare Terminals/Accessories to remove the blank terminals resulting from the Associate. 

Insert spare terminals and accessories

1. Select terminal 7 in the grid.
2. In the Spare section, click the Insert Spare Terminal button. 
3. On the Insert Spare Terminal dialog box, specify:

   Number: SPARE
   Quantity: 1

   Note: You can also assign catalog information for the spare terminal from the Insert Spare Terminal dialog box by clicking Catalog Lookup. If needed, you can then select the part from the Catalog Browser dialog box.

   Click Insert Above.

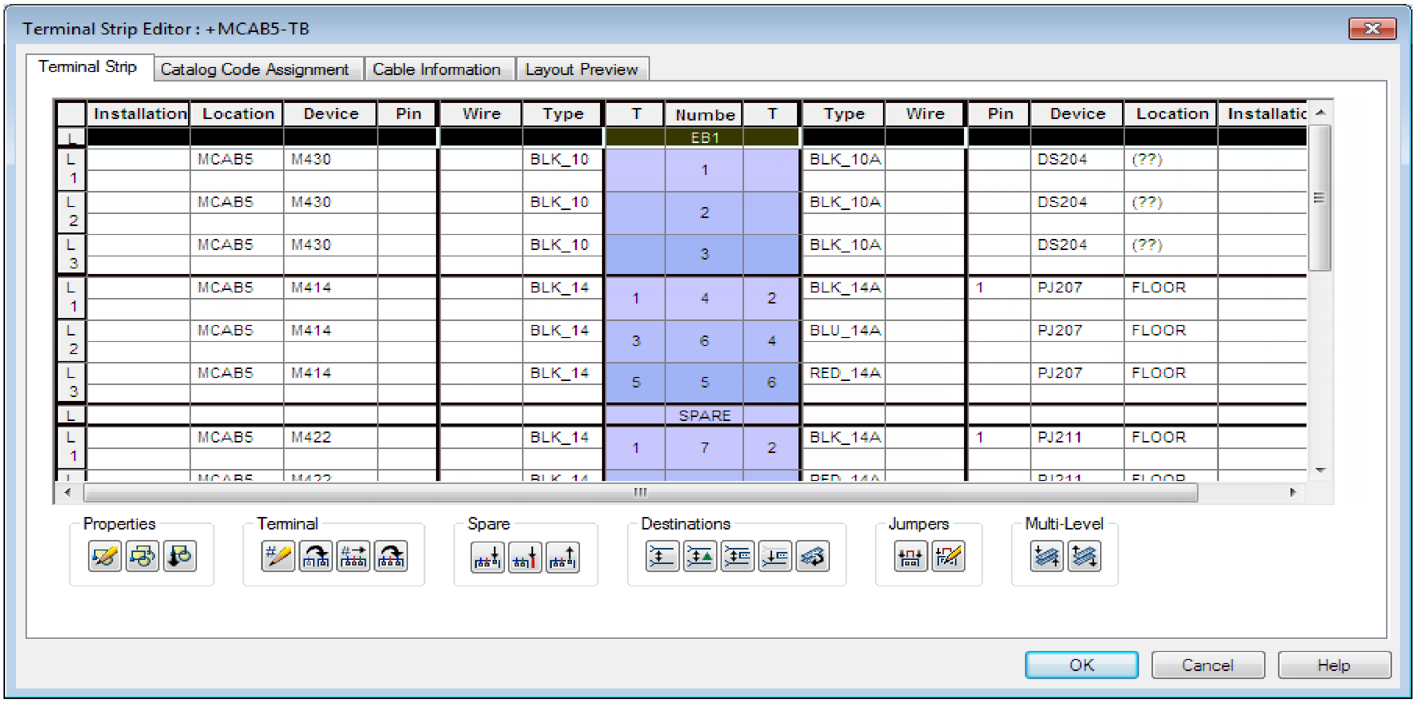
   Now you insert accessories (end barriers) into the terminal strip - one at the top and one at the bottom of the terminal strip.

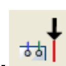
4. Select terminal 1 in the grid.
5. In the Spare section, click the Insert Accessory button. 
6. On the Insert Accessory dialog box, specify:

   Number: EB1
   Quantity: 1

Note: You can also assign catalog information for the accessory from the Insert Accessory dialog box by clicking Catalog Lookup. You can then select the part from the Catalog Browser dialog box.

Click Insert Above.



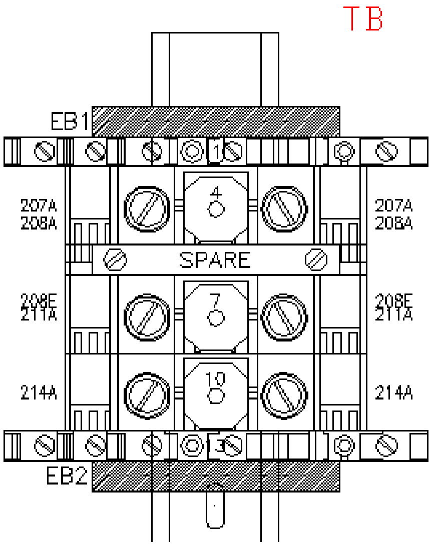
7. Select terminal 15 in the grid.
8. In the Spare section, click the Insert Accessory button. 
9. On the Insert Accessory dialog box, specify:

Number: EB2
Quantity: 1

Click Insert Below.

Insert the terminal strip into the drawing

1. On the Terminal Strip Editor dialog box, click the Layout Preview tab.
2. Select Graphical Terminal Strip as the terminal type to insert into the drawing.
3. Enter 2.0 in Scale on Insert.
4. Click Rebuild.



5. On the Terminal Strip Editor dialog box, click OK.
6. On the Terminal Strip Selection dialog box, click OK.

Generating Reports Tutorial

Generate and work with reports.

Time required	30 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Generating reports to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Generate a report
- Insert a report on to a drawing
- Change the format of a report
- Export the report to a spreadsheet

Project Bill Of Material for all locations (6 records)

TAGS	QTY	SUB	CATALOG	MFG	DESCRIPTION
CB322 CB324 CB326 CB328 CB330	5		EGH3015FFG	EATON	CIRCUIT BREAKER - E125 FRAME 3-POLE CIRCUIT BREAKER 15AMPS TYPE E125H, FIXED THERMAL & MAGNETI 690VAC, 250VDC, 15AMPS
DS304	1		194E-A25-1753	AB	IEC LOAD SWITCH 3 POLE 194E - LOAD SWITCH 25AMPS ON-OFF BASE MOUNTED SWITCH (INCLUDE 480VAC, 25AMPS
FU309	1		FRS-R-15	BUSSMANN	DUAL ELEMENT FUSE - CLASS RK5 TIME DELAY, CURRENT LIMITING 600VAC

Generating Bill of Material Reports

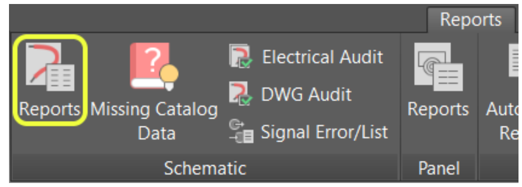
Extract catalog information from the project to create a Bill of Materials report.

Using AutoCAD Electrical toolset, you can perform a project-wide extract of all BOM data found on your project drawing set. The data is extracted from the project database, matched with standard entries in the catalog database, and then additional fields are pulled from the catalog files. You can:

- Format this data into various report configurations
- Output to report files
- Export to a spreadsheet or database program
- Place in an AutoCAD Electrical toolset drawing

Generate a Bill of Material (BOM) report

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. Open AEGS11.dwg.
4. Click Reports tab ➤ Schematic panel ➤ Reports.
5. In the Schematic Reports dialog box, select:

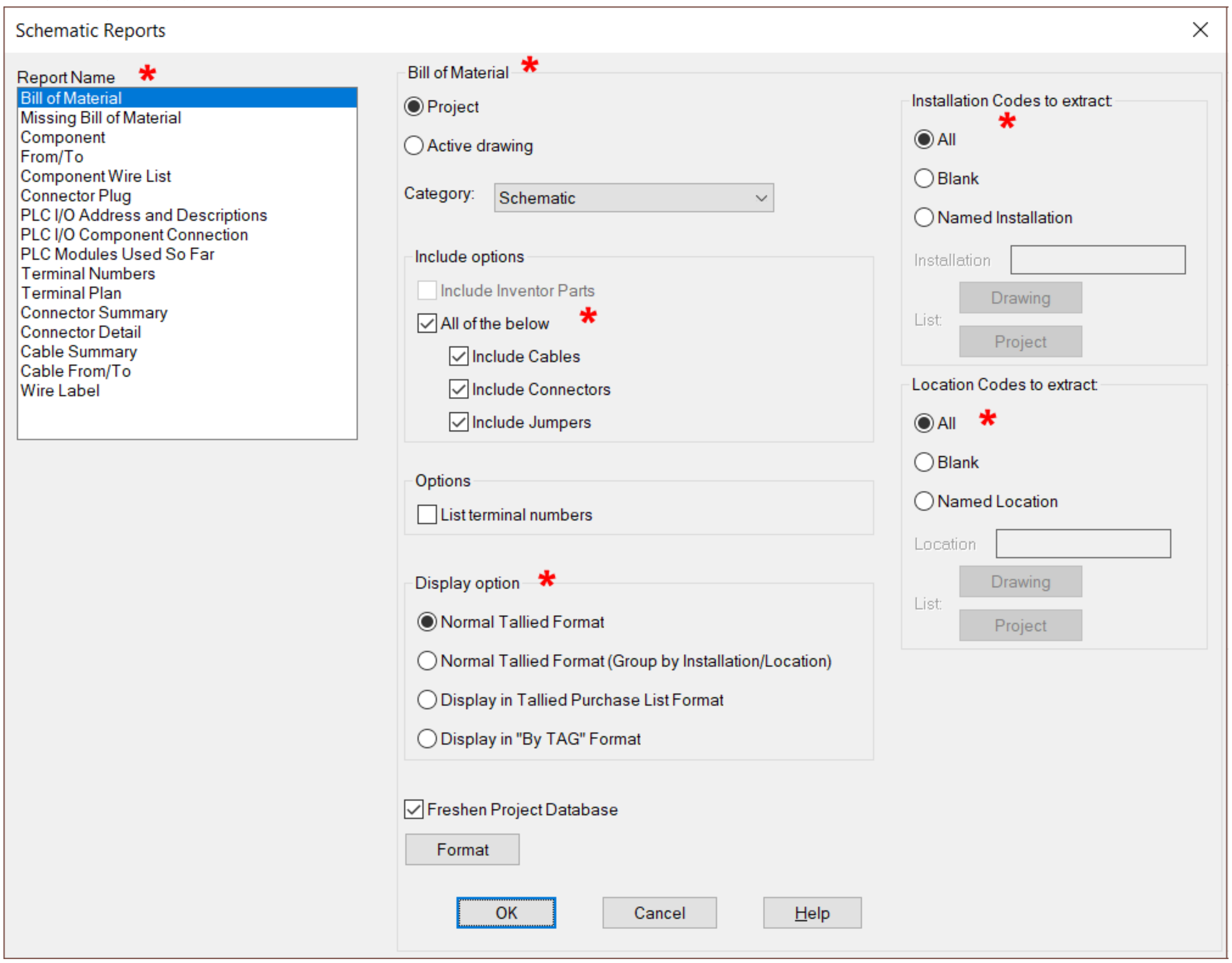


Report Name: Bill of Material
Bill of Material: Project

Verify that the following options are specified:

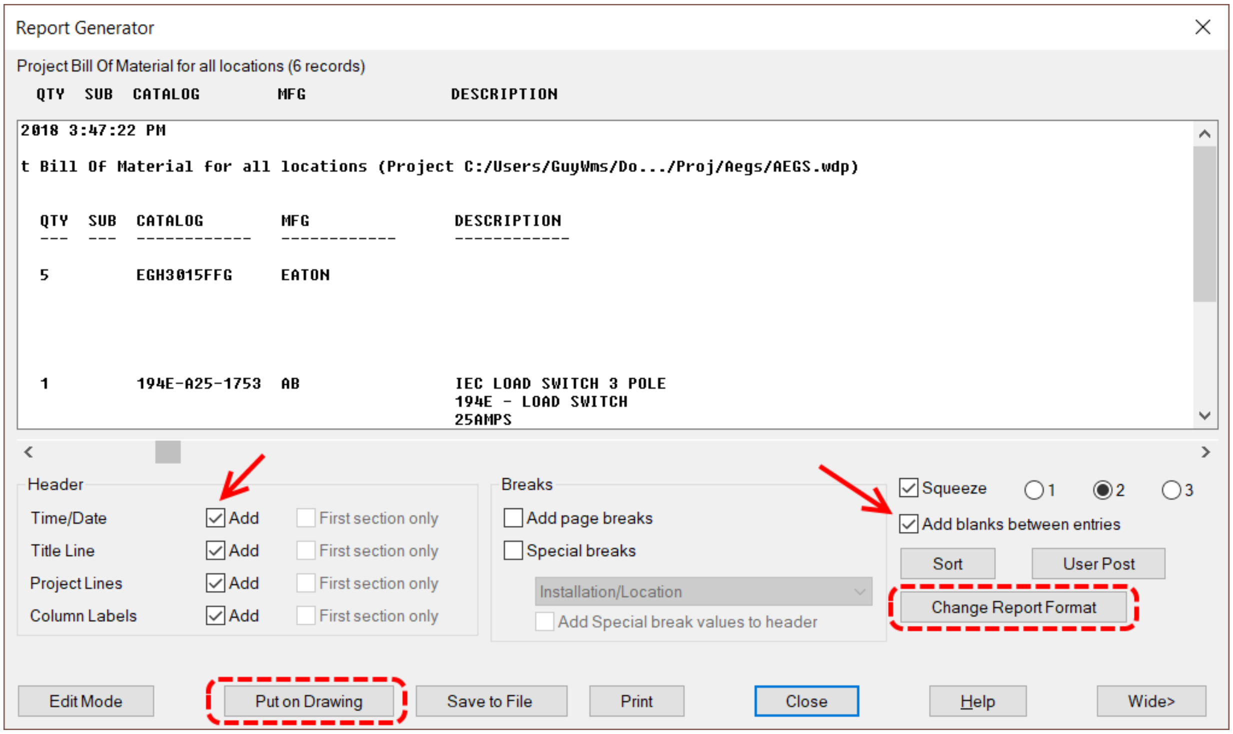
Include options: All the above (below?)
Display option: Normal Tallied Format
Installation Codes to extract: All
Location Codes to extract: All

Click OK.



6. In the Select Drawings to Process dialog box, select AEGS03.DWG, and click Process.
7. Verify that AEGS03.DWG displays in the Drawings to Process section of the dialog box and click OK.

The generated report displays in the Report Generator dialog box.



8. In the Report Generator dialog box, select:

   Header: Time/Date
   Header: Column Labels

   Add blanks between entries

Inserting Bill of Material Tables into Drawings

Insert a report as a table selecting from various table output options.

Insert a BOM into the drawing in tabular format

1. With the BOM report displayed in the dialog box, click Put on Drawing.

2. In the Table Generation Setup dialog box, select:

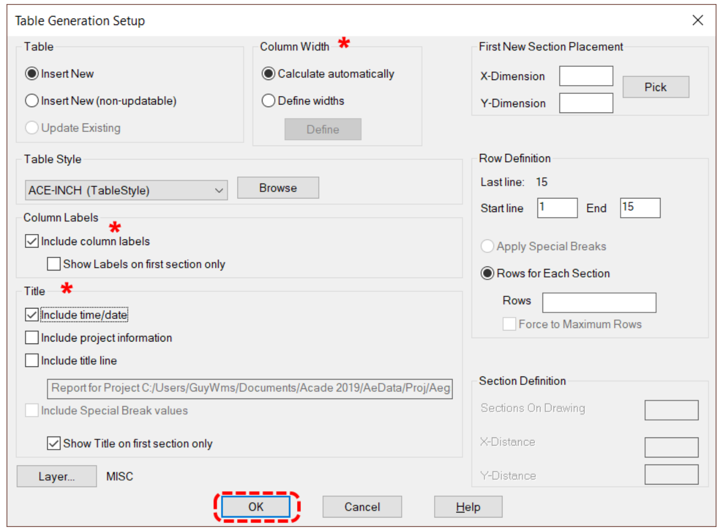
   Column Labels: Include column labels

   Title: Include time/date

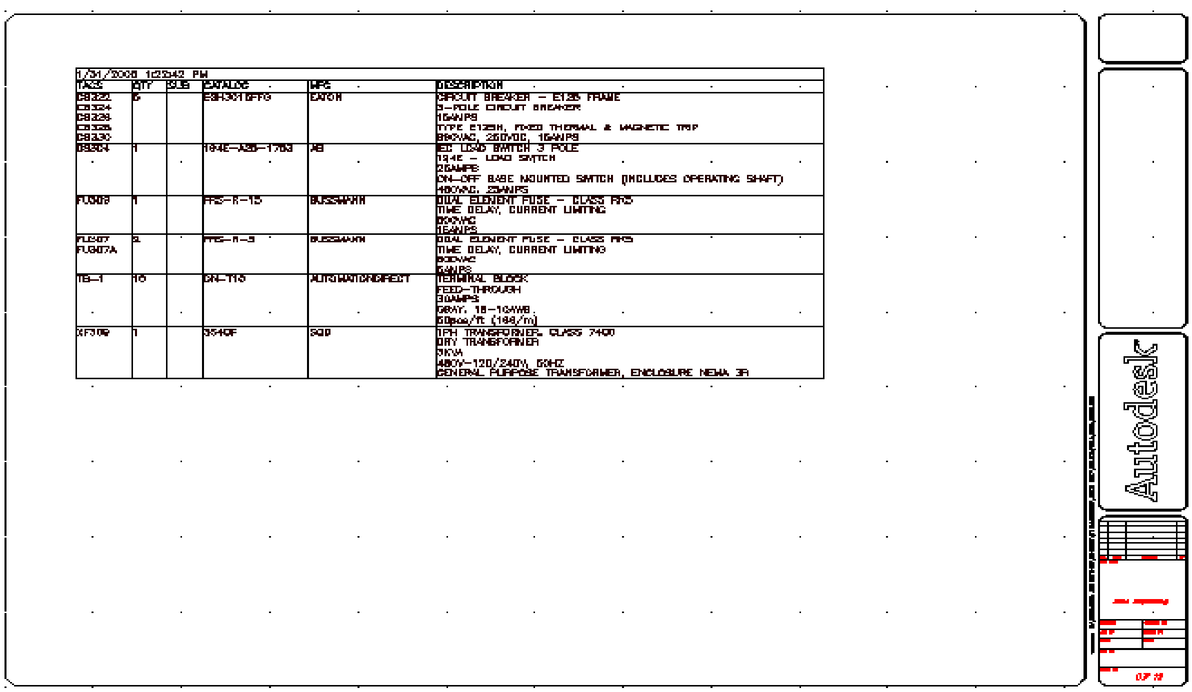
   Column Width: Calculate automatically

   Click OK.

   Note: The extents of the BOM table are displayed in temporary graphics. Press Z to zoom down, or R to flip into real-time pan and zoom mode, if necessary.



3. The table outline moves with your cursor. Position the table, and then click to place the table. The BOM table is built where you placed it.



4. In the Report Generator, click Close.

Changing Format of Bill of Material Report

Each AutoCAD Electrical toolset report is customizable:

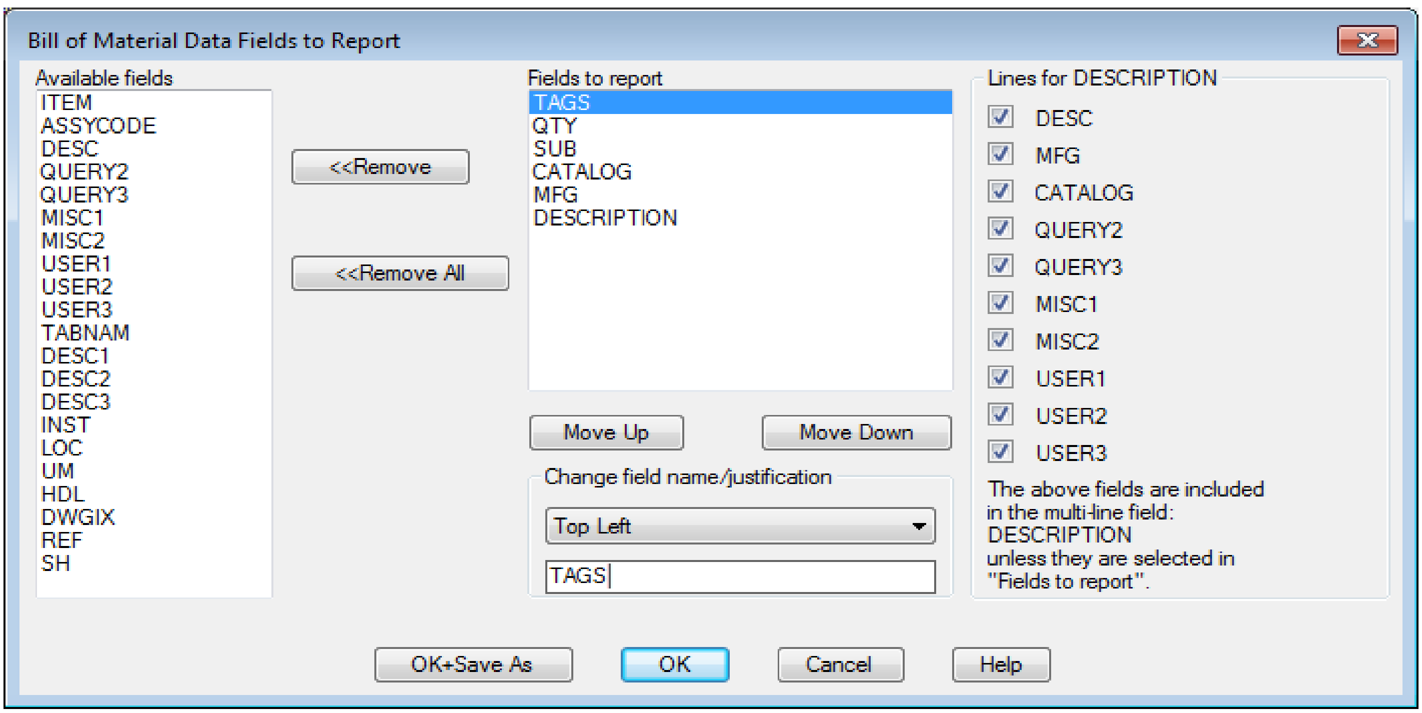
- Define which data fields are reported
- Define the order in which they appear
- Define the justification of any column
- Define the column labels

Remove the TAGS columns from the BOM

1. Erase the table, or UNDO, and rerun the BOM extract for AEGS03.DWG.
2. In the Report Generator dialog box, click **Change Report Format**. (graphic above)

   In the **Bill of Materials Data Fields to Report** dialog box, **Fields to Report** section, the fields that format the BOM are displayed.

3. Select TAGS in the Fields to report list.
4. Click <<Remove. The TAGS field is moved out of Fields to report and into Available fields.



   Note: You can also select a field in the Available fields list to add it to the report. You can rearrange columns using the Move Up and Move Down buttons. Clicking Ok-Save As saves these settings to a file for later use.

5. Click OK.

   Note: This new format becomes the default the next time you extract a BOM report. The BOM data in the Report Generator dialog box is reformatted and displayed.

6. Scroll down the report to verify that the component tags column is removed.
7. Insert the new version of the BOM table into the drawing.

Exporting Bill of Material Report to Spreadsheet

Export the report data to a spreadsheet.

You can move your BOM to a spreadsheet, database, or any other application that can read data in a comma-delimited or Microsoft Access format.

Export the BOM to an Excel® spreadsheet

1. In the Report Generator dialog box, click Save to File.
2. In the Save Report to File dialog box, select Excel spreadsheet format (.xls) and click OK.
3. In the Select file for report dialog box, enter an output file name or click OK to accept the default name BOM.xls. Click Save.
4. In the Optional Script File dialog box, click Close - No Script.
5. In Excel, click File ➤ Open.
6. Browse to the location where you saved the spreadsheet and select it. The default location is C:\Users\{username}\My Documents .
7. Click Open.

	A	B	C	D	E
1	QTY	SUB	CATALOG	MFG	DESC
2	5	1	EGH3015FFG	EATON	CIRCUIT BREAKER - E125 FRAME
3	1	1	194E-A25-1753	AB	IEC LOAD SWITCH 3 POLE
4	1	1	FRS-R-15	BUSSMANN	DUAL ELEMENT FUSE - CLASS RK5
5	2	1	FRS-R-5	BUSSMANN	DUAL ELEMENT FUSE - CLASS RK5
6	10	1	DN-T10	AUTOMATIONDIRECT	TERMINAL BLOCK
7	1	1	3S40F	SQD	1PH TRANSFORMER, CLASS 7400

Your BOM data displays in spreadsheet format. You can slide the column borders to expose the full column of text for each field. The first six columns of the spreadsheet are shown in the previous image. The first column is the tallied quantity, followed by subassembly quantity, catalog number, and manufacturer code. The remaining fields are the fields extracted from the mfg/cat combo query on the external catalog look-up file.

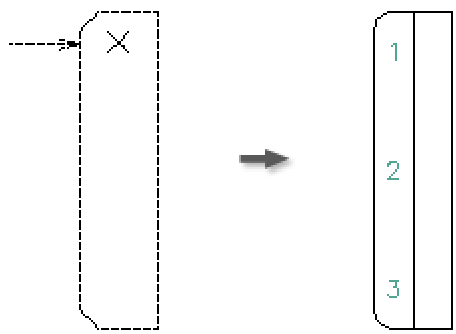
Connector Diagrams Tutorial

Insert, modify, and wire connectors.

Time required	45 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Connector diagrams to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Insert a connector
- Wire connectors
- Insert in-line connectors
- Stretch a connector
- Add a connector pin
- Move a connector pin
- Add connector descriptors



About Connector Diagrams

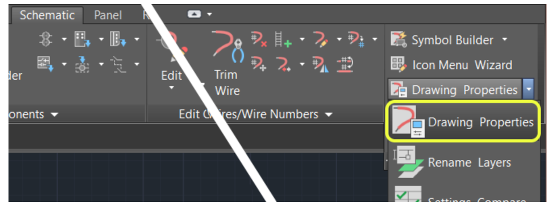
Understand connectors and point-to-point connector diagrams.

The connector wiring tools help you more easily create and work with point-to-point style wiring schematics (as opposed to ladder-style schematics). Although some of these tools are useful for ladder-style schematics, they are tuned to work well with drawings that are heavy on point-to-point connector diagrams. Instead of creating and maintaining a large library of schematic connector symbols, each symbol is generated parametrically. It is generated on the fly, per user input and at user-defined orientation. A connector toolbar contains tools for creating and editing connectors.

Inserting Connectors

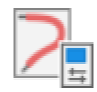
Dynamically build connectors by defining the connector parameters such as the number of pins, pin spacing, and pin values.

The Insert Connector tool generates a connector symbol from user-defined parameters. The symbol is created on the fly and inserted as a block insert into your active drawing file. Since they are created on an as needed basis, it eliminates the need for you to create and maintain a library of connector symbols.

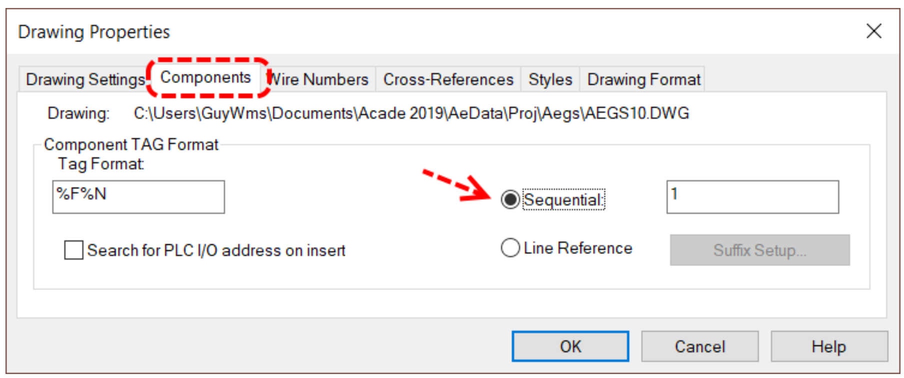


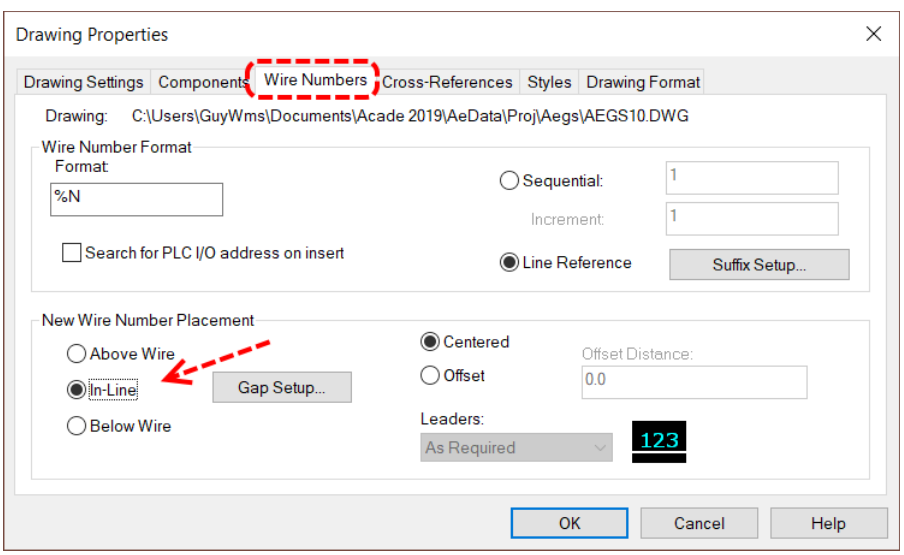
Change drawing properties

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. Open *AEGS10.dwg*.



4. Click Schematic tab ➤ Other Tools panel ➤ Drawing Properties drop-down ➤ Drawing Properties.
5. On the Drawing Properties ➤ Components tab dialog box, select Sequential.
6. On the Drawing Properties ➤ Wire Numbers dialog box, New Wire Number Placement section, select In-Line.
7. Click OK.

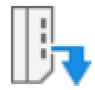


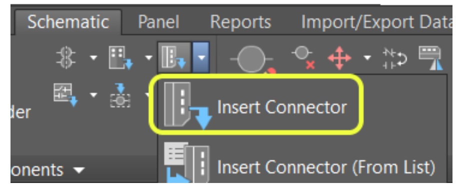


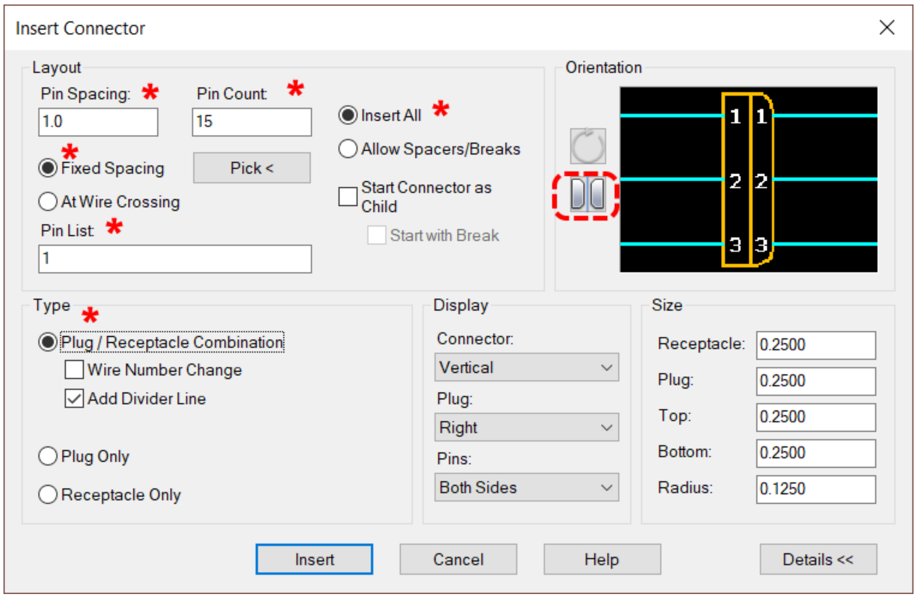
Add connectors to the drawing

1. Click Schematic ➤ Insert Components panel ➤ Insert Connector drop-down ➤ Insert Connector.
2. On the Insert Connector dialog box, specify:

   Pin Spacing: 1.0
   Pin Count: 15
   Fixed Spacing
   Pin List: 1
   Insert All

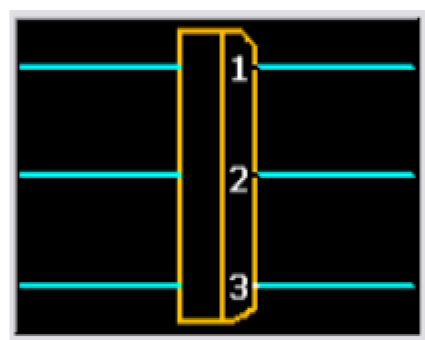






3. Click the Flip button to flip the connector about its long axis.

   The preview looks like the following image.



4. Click Insert.

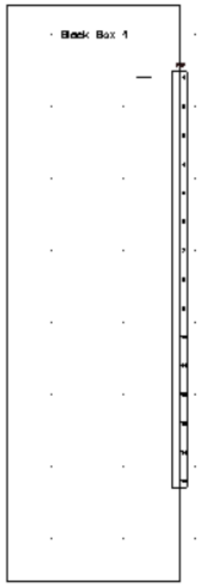
   A preview outline of the connector displays for placement on the drawing. It shows rounded corners for the plug side of the connector. An “x” indicates the insertion point of the connector. An arrow indicates the plug side wire connection direction for plug/receptacle or plug-only connector inserts, or shows the wire connection direction for a receptacle-only connector insert.

   Note: Before committing the connector outline to the drawing, press TAB to flip the connector through four different orientations. Or, press the V key to switch between vertical and horizontal orientations.

5. Respond to the prompts as follows:

   Specify insertion point or [Z=zoom, P=pan, X=wire crossing, V=horizontal/vertical, TAB=flip]:

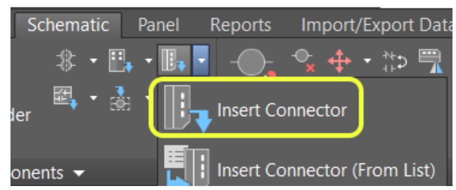
   Select to place the connector in the middle of the right-hand border of Black Box 1



The connector was automatically assigned a component tag of PJ1.

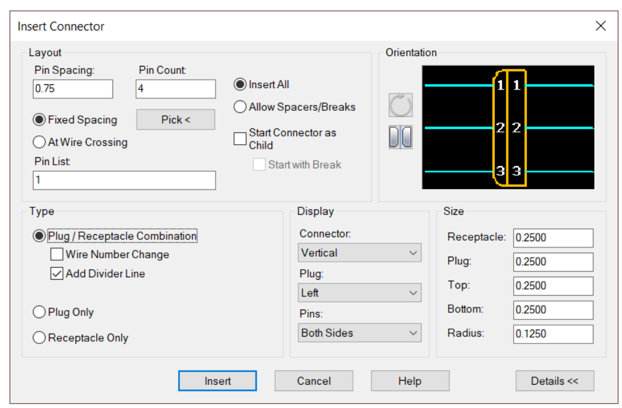
6. Click Schematic ➤ Insert Components panel ➤ Insert Connector drop-down ➤ Insert Connector.
7. On the Insert Connector dialog box, specify:

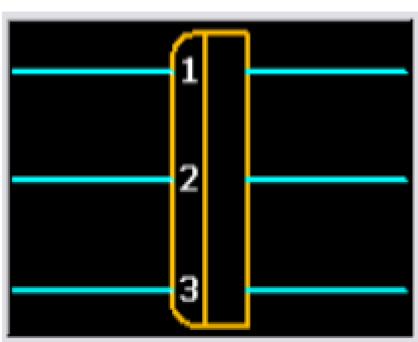
Pin Spacing: 0.75
Pin Count: 4
Fixed Spacing
Pin List: A
Insert All



8. Click the Flip button to flip the connector. 

The preview looks like the following image.

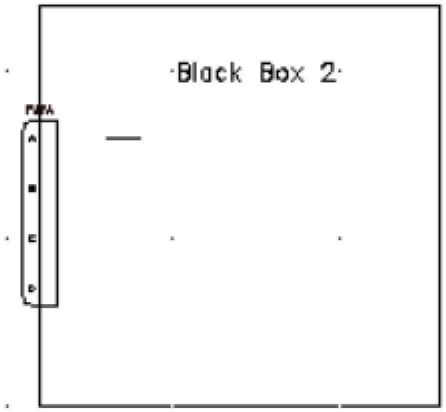




9. Click Insert.
10. Respond to the prompts as follows:

Specify insertion point or [Z=zoom, P=pan, X=wire crossing, V=horizontal/vertical, TAB=flip]:

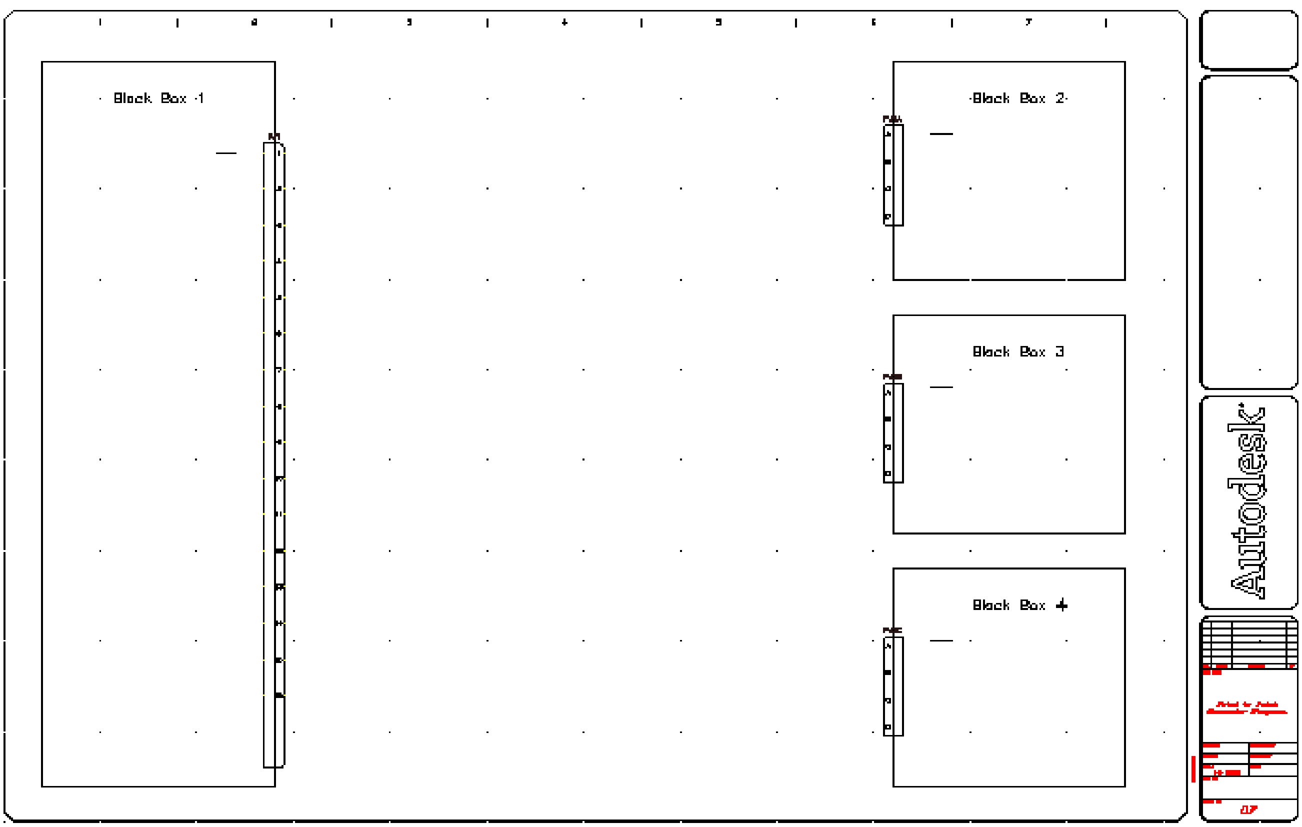
Select to place the connector in the middle of the left-hand border of Black Box 2



The connector was automatically assigned a component tag of PJ2.

11. Repeat steps 6 - 10 to place connectors on Black Box 3 and Black Box 4.

The connectors are assigned tags PJ3 and PJ4 respectively.



Wiring Connectors

Use the Insert Wire and Multiple Bus tools to add wires between connectors.

Black Box 1 is associated to a larger component such as a power box. Black Box 2 - Black Box 4 are smaller components that are part of the power box. The components must be wired together. The easiest way to do it is to use the Insert Wire and Multiple Wire Bus tools.

Wire the connectors together

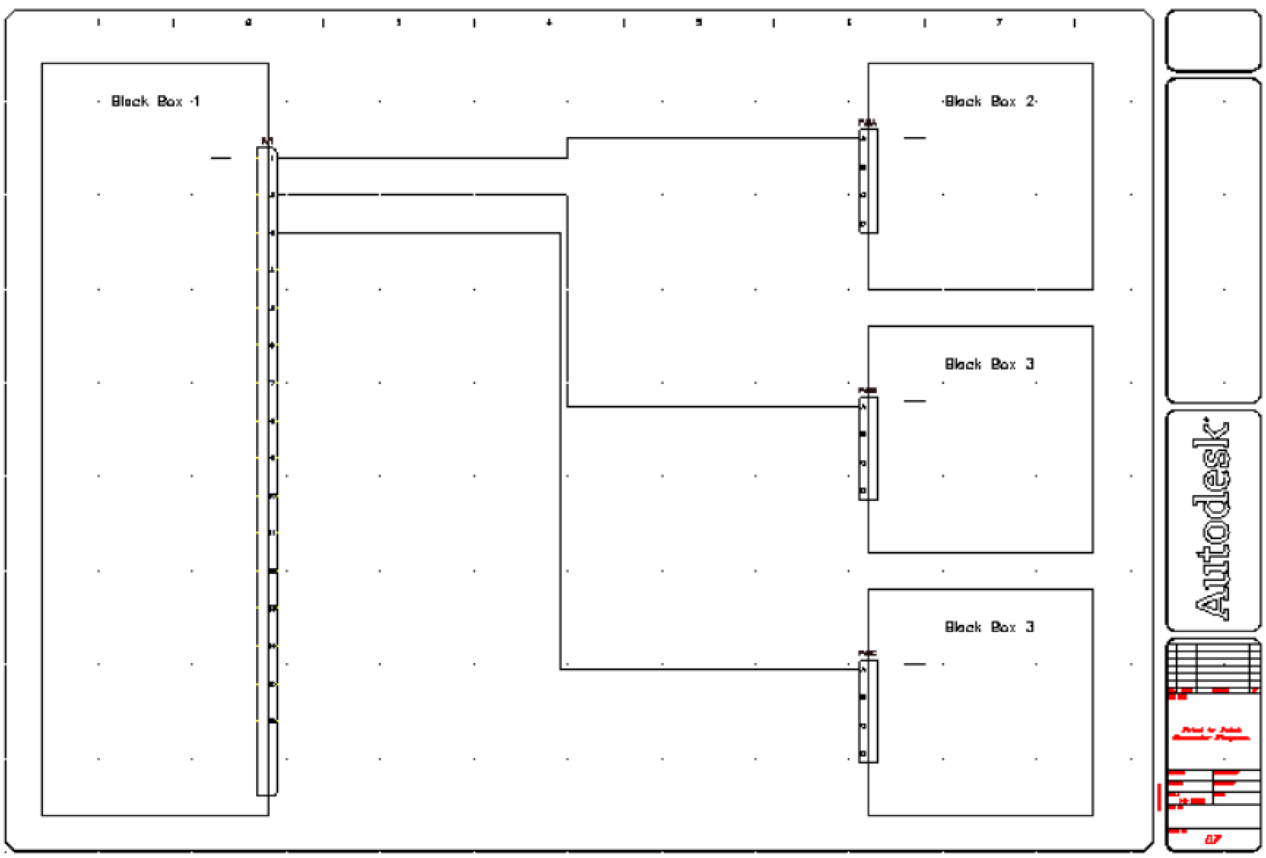
1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wires drop-down ➤ Wire.
2. Respond to the prompts as follows:

   Specify wire start or [wireType/X=show connections]:

   Click PJ1 at pin 1 on Black Box 1

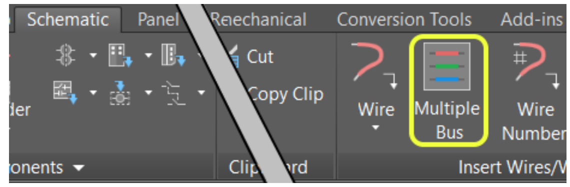
   Specify wire end or [Continue]: *Click PJ2 at pin A on Black Box 2*

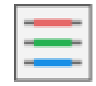
3. Repeat to connect PJ1 (Pin 2) to PJ3 (Pin A) and PJ1 (Pin 3) to PJ4 (Pin A). Right-click to exit the command.



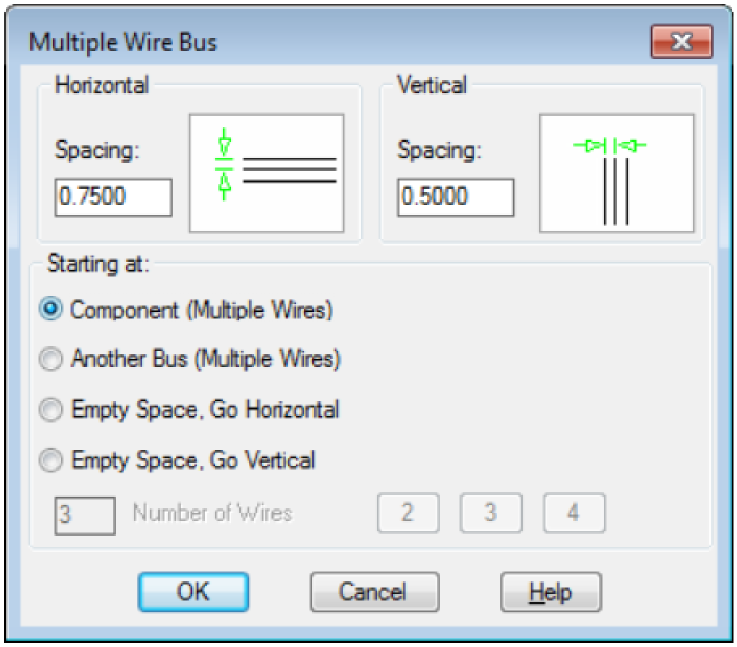
Notice that the Insert Wire tool drew the wire between the connectors while avoiding any existing geometry on the screen.

4. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Multiple Bus.
5. On the Multiple Wire Bus dialog box, specify:





Horizontal Spacing: 0.75
Vertical Spacing: 0.50
Starting at: Component (Multiple Wires)



6. Click OK.
7. Respond to the prompts as follows:

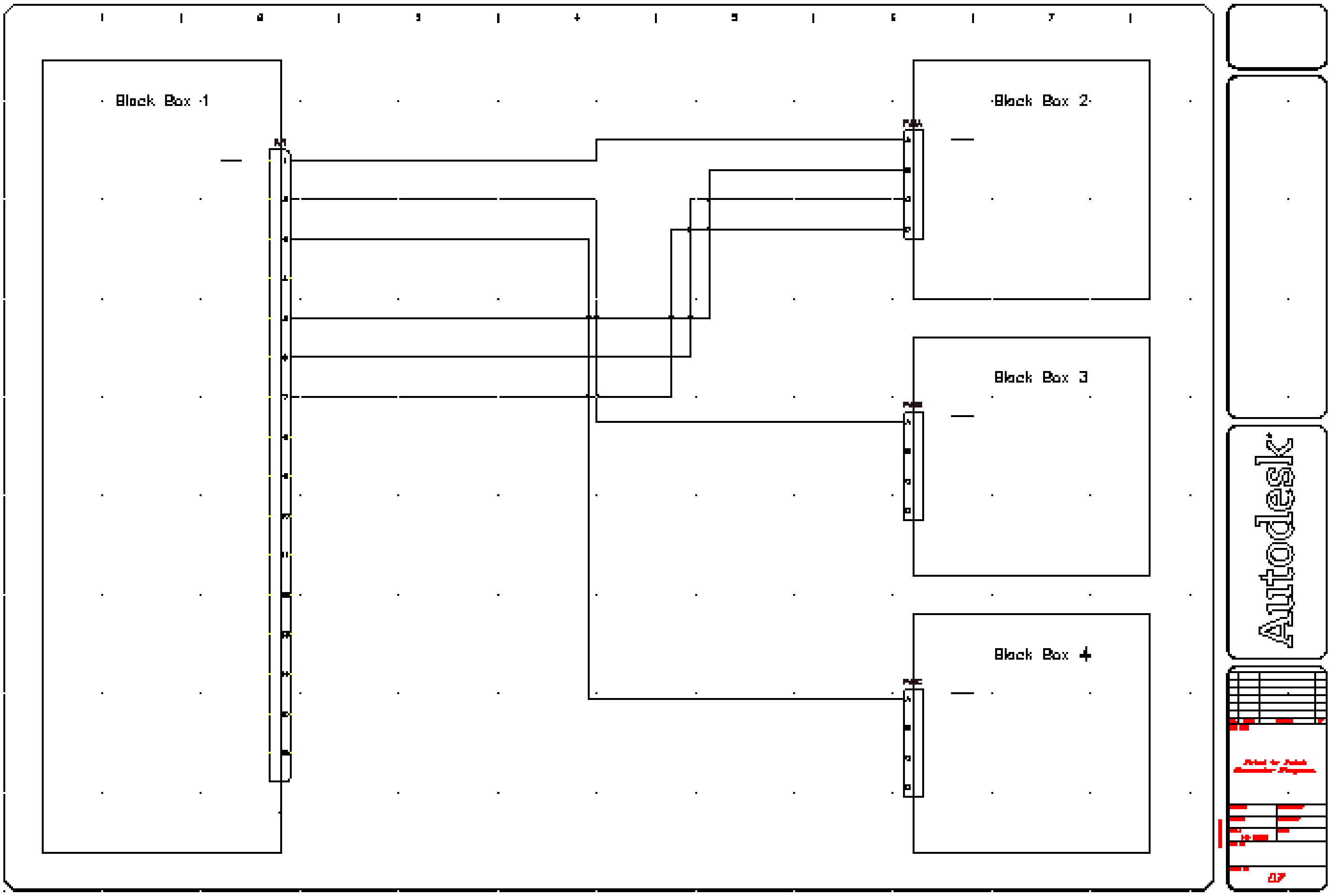
Window select starting wire connection points

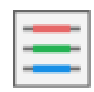
Select pins 5-7 on Black Box 1 (1) and right-click

to (T= wiretype): *Drag the wires to the right past the three wires you inserted,*

to Point (Continue/Flip): *Drag up the wires towards PJ2 on Black Box 2, enter C and press* ENTER *(to continue and lock the drag)*

to (Continue/Flip): *Drag the wires to the right and connect to pins B-D on PJ2 (2)*





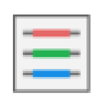
8. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Multiple Bus.
9. On the Multiple Wire Bus dialog box, click OK to use the previous settings.
10. Respond to the prompts as follows:

Window select starting wire connection points: *Select pins 9-11 on Black Box 1 and right-click*

to (T= wiretype): *Drag the wires to the right,*

to Point (Continue/Flip): *Drag up the wires towards PJ3 on Black Box 3, enter*C, *and press* ENTER*(to continue and lock the drag)*

to (Continue/Flip): *Drag the wires to the right and connect to pins B-D on PJ3*

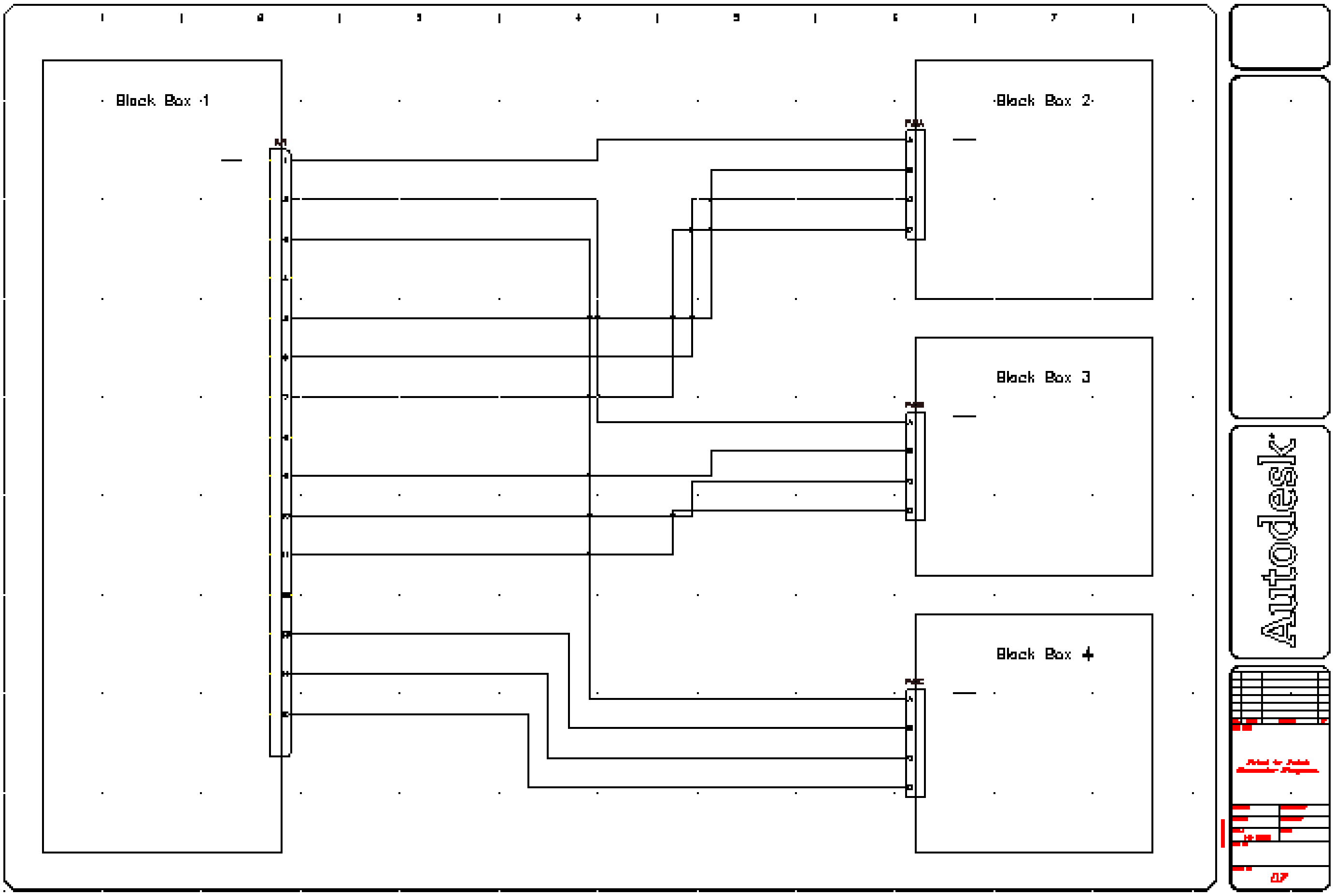


11. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Multiple Bus.
12. On the Multiple Wire Bus dialog box, click OK to use the previous settings.
13. Respond to the prompts as follows:

Window select starting wire connection points: *Select pins 13-15 on Black Box 1 and press* ENTER

to (T= wiretype): *Drag the wires to the right,*

to Point (Continue/Flip): *Drag the wires down towards PJ4 on Black Box 4, press*C,*and press*ENTER*(to continue and lock the drag)* to (Continue/Flip): *Drag the wires to the right and connect to pins B-D on PJ4*



Grouping Wires

Insert a connector across existing wires.

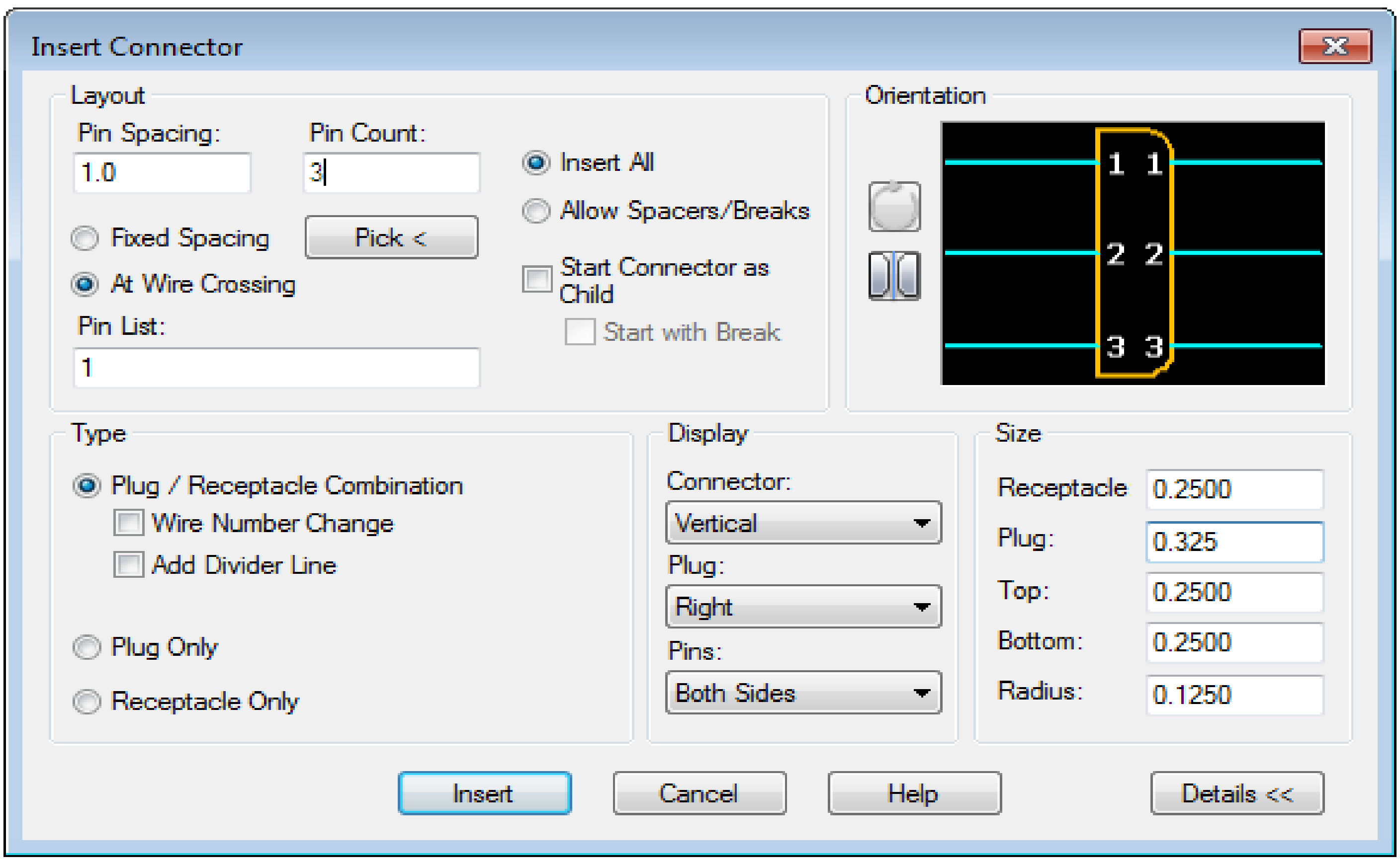
Now that you wired the connectors together, you insert [illegible] connectors to group the wires.

Insert in-line connectors

1. Click Schematic ➤ Inse[illegible]ert Connector.
2. On the Insert Connector di[illegible]

   Pin Spacing: 1.0
   Pin Count: 3
   At Wire Crossing
   Pin List: 1
   Insert All

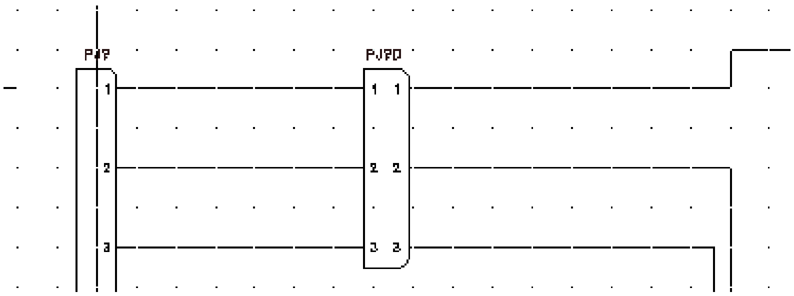
3. Click Details.
4. On the Type section, clear the Add [illegible] Line box.
5. On the Display section, set Plug to [illegible]ht and Pins to Both Sides.
6. On the Size section, set the Plug [illegible] 0.325.



7. Click Insert.
8. Respond to the prompts as follows:

Specify insertion point or [Z=zoom, P=pan, X=wire crossing, V=horizontal/vertical, TAB=flip]:

Select to place the connector on the wires connected to PJ1, Pins 1-3



9. Click Schematic ➤ Insert Components panel ➤ Insert Connector drop-down ➤ Insert Connector.
10. On the Insert Connector dialog box, specify:

    Pin Spacing: 1.0
    Pin Count: 9
    At Wire Crossing
    Pin List: 1
    Allow Spacers/Breaks

11. Click Insert.
12. Respond to the prompts as follows:

    Specify insertion point or [Z=zoom, P=pan, X=wire crossing, V=horizontal/vertical, TAB=flip]:

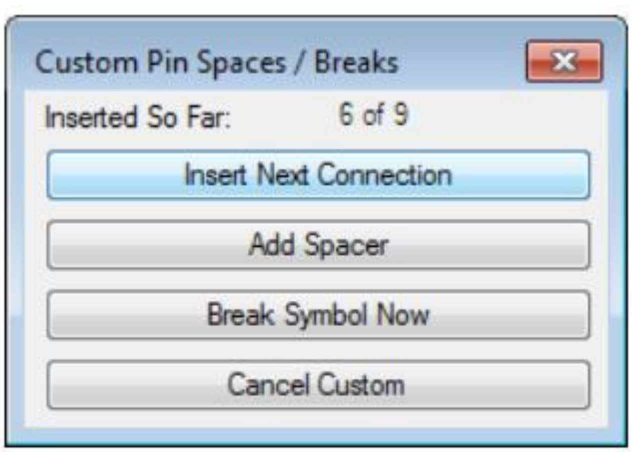
    Select to place the connector starting on the line at PJ1, Pin 5

    Notice how the connector expands when you cross the wires.

13. On the Custom Pin Spaces/Breaks dialog box, click Insert Next Connection.

    The dialog box displays which connector pin has been inserted so far. Keep clicking Insert Next Connection until you place six of the nine connections.

14. When the Custom Pin Spaces/Breaks dialog box says "Inserted So Far: 6 of 9," click Break Symbol Now.

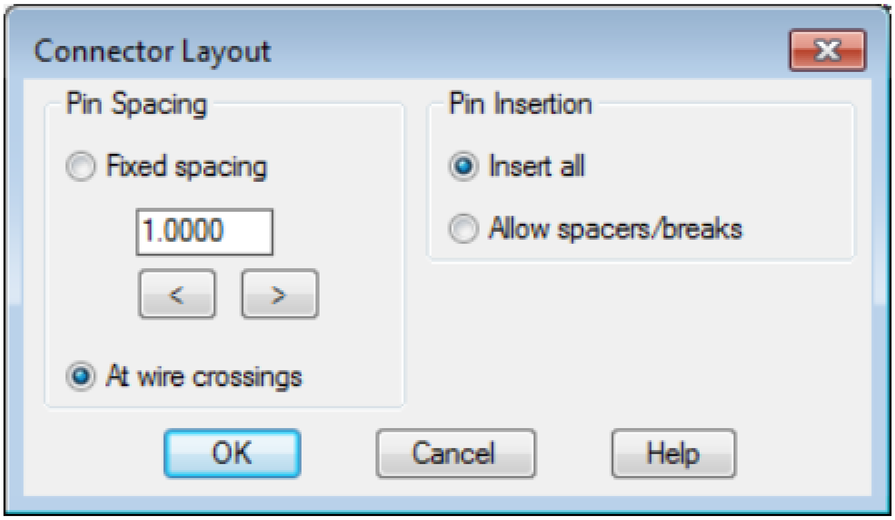


15. Respond to the prompts as follows:

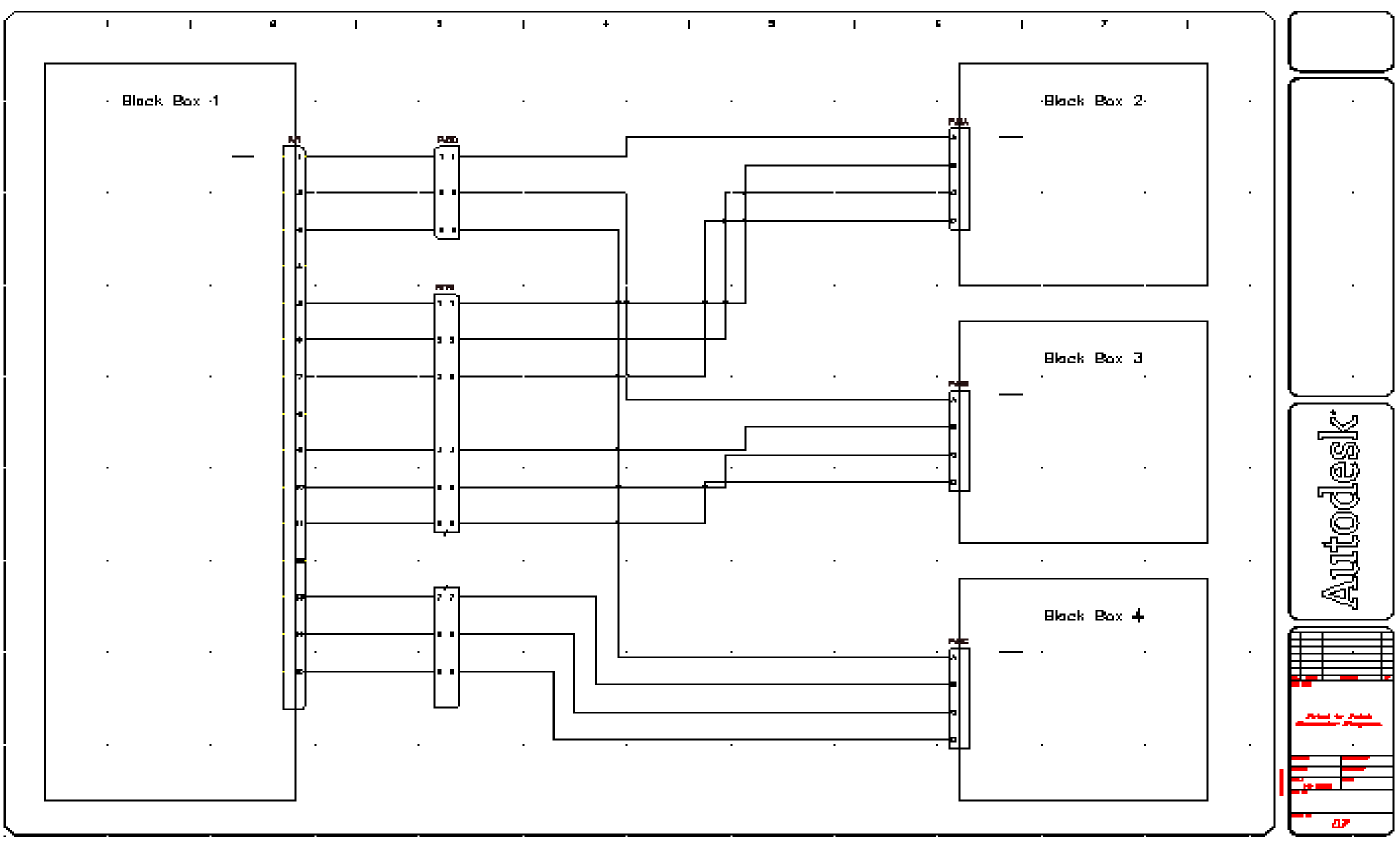
Specify insertion point or [Z=zoom, P=pan, X=wire crossing, V=horizontal/vertical, TAB=flip]:

Select to place the connector starting on the line at PJ1, Pin 13

16. On the Connector Layout dialog box, select Insert All.



17. Click OK.



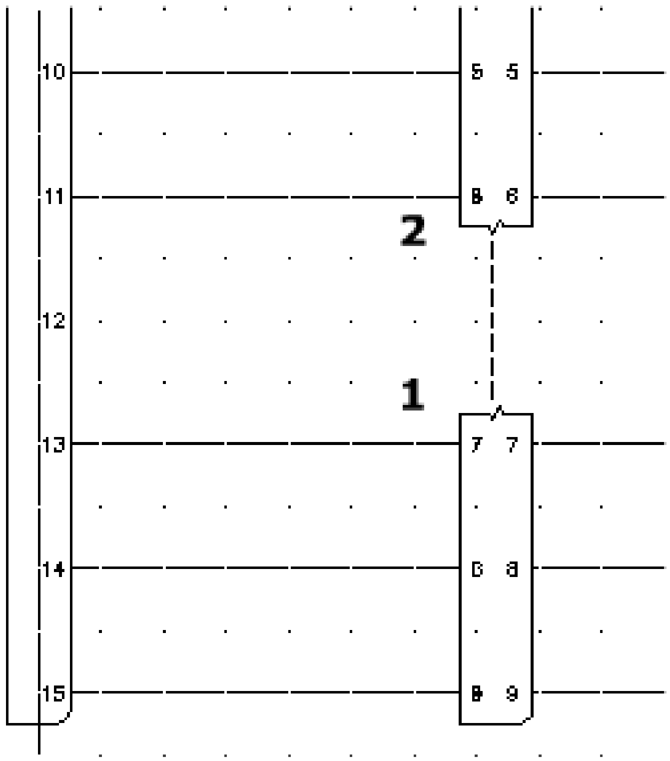
Note: Another method is to insert the entire connector and then use the Split Connector tool to break the existing connector.

18. Click Schematic tab ➤ Insert Components panel ➤ Dashed Link Line drop-down ➤ Link Components with Dashed Line.

19. Respond to the prompts as follows:

Component to link from: *Select the bottom portion of PJ6 (1)*

component to link to: *Select the top portion of PJ6 (2), right-click*

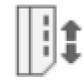


Modifying Connectors

Stretch existing connectors and add, move, and swap connector pins.

The Insert Connector toolbar has tools for modifying connectors and connector pins. You can also add, remove, or move the pins found inside of the connector.

Stretch existing connectors

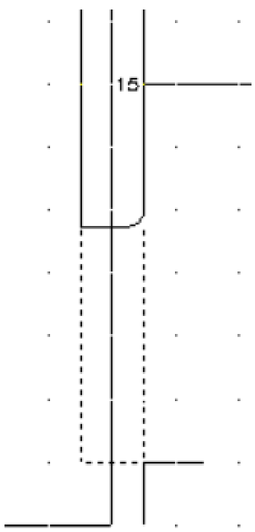


1. Click Schematic ➤ Edit Components panel ➤ Modify Connectors drop-down ➤ Stretch Connector.
2. Respond to the prompts as follows:

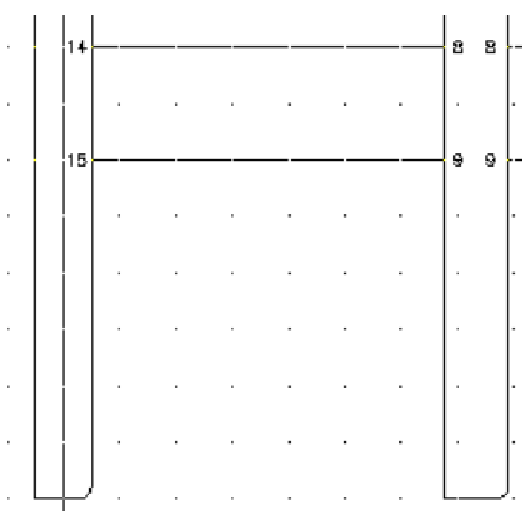
   Specify which end of connector to stretch: *Select the bottom of PJ1*

   Specify second point of displacement:

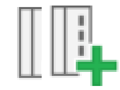
   Pull the connector down towards the bottom of Black Box 1



3. Repeat for PJ6, pulling the bottom of the connector down so that it is even with PJ1.



Add connector pins



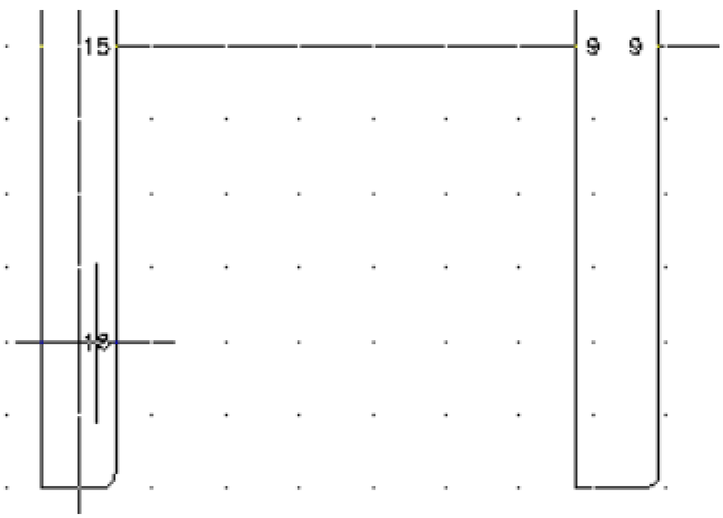
1. Click Schematic ➤ Edit Components panel ➤ Modify Connectors drop-down ➤ Add Connector Pins.
2. Respond to the prompts as follows:

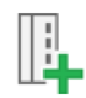
   Select connector: *Select PJ1*

   Specify where to insert new pin or [Reset]<16>:

   Select 4 spaces down from pin 15 on PJ1, right-click, and select Enter

   The next available pin number (16) inserts at the selected point.



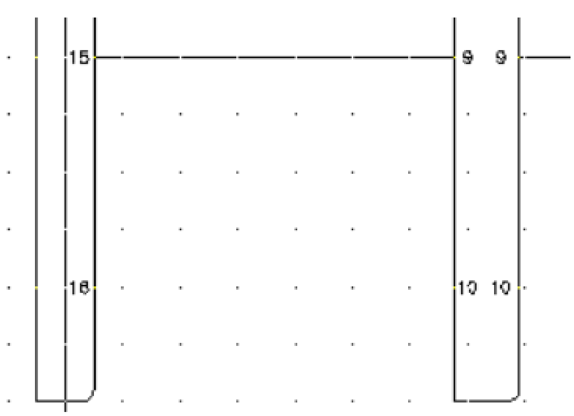


3. Click Schematic ➤ Edit Components panel ➤ Modify Connectors drop-down ➤ Add Connector Pins.
4. Respond to the prompts as follows:

   Select connector: *Select PJ6*

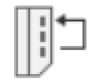
   Specify where to insert new pin or [Reset]<10>:

   Select the new pin 16 on PJ1 to insert pin 10 in-line with it, right-click and select Enter



Note: You can delete pins using the Delete Connector Pins tool. Select the pin you want to delete and it is automatically removed from the connector.

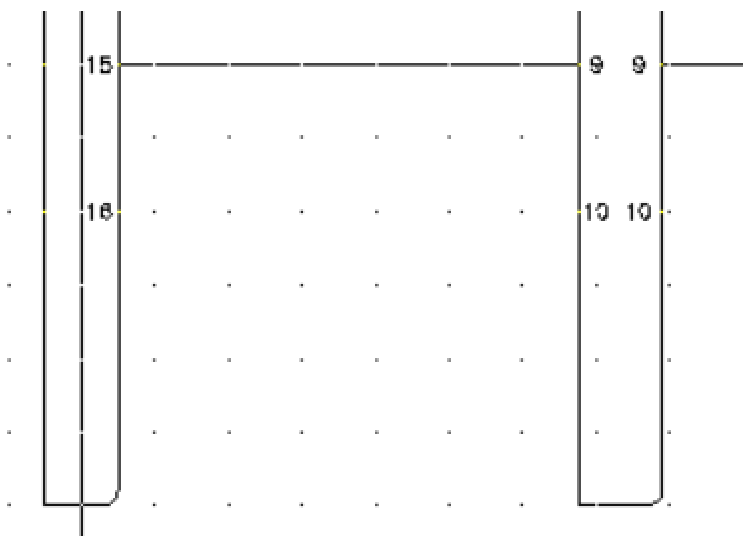
Modify connector pins



1. Click Schematic tab ➤ Edit Components panel ➤ Modify Connectors drop-down ➤ Move Connector Pins.
2. Respond to the prompts as follows:

    Select connector pin to move: *Select pin 16 on PJ1*
    Specify new location for pin 16: *Select 2 spaces up on PJ1*
    Select connector pin to move: *Select pin 10 on PJ6*
    Specify new location for pin 10:

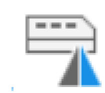
Select pin 16 on PJ1 to move pin 10 in-line with it, right-click



3. Click Schematic tab ➤ Edit Components panel ➤ Modify Connectors drop-down ➤ Swap Connector Pins.
4. Respond to the prompts as follows:

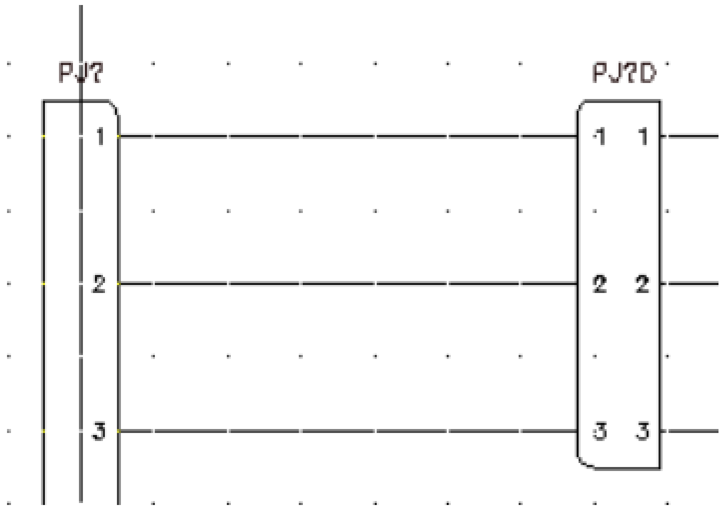
    Select connector pin: *Select pin 16 on PJ1*

    Select connector pin: swap with: *Select pin 12 on PJ1, right-click*



5. Click Schematic tab ➤ Edit Components panel ➤ Modify Connectors drop-down ➤ Reverse Connector.
6. Respond to the prompts as follows:

    Select connector to Reverse: *Select the top in-line connector, right-click*



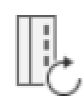
7. Click Schematic ➤ Insert Components panel ➤ Insert Connector drop-down ➤ Insert Connector.
8. On the Insert Connector dialog box, specify:

    Pin Spacing: 1.0
    Pin Count: 2
    Fixed Spacing
    Pin List: 1
    Insert All

9. Click Details.
10. On the Type section, select Add Divider Line.
11. On the Display section, set Pins to Plug Side.
12. Click Insert.
13. Respond to the prompts as follows:

    Specify insertion point or [Z=zoom, P=pan, X=wire crossing, V=horizontal/vertical, TAB=flip]:

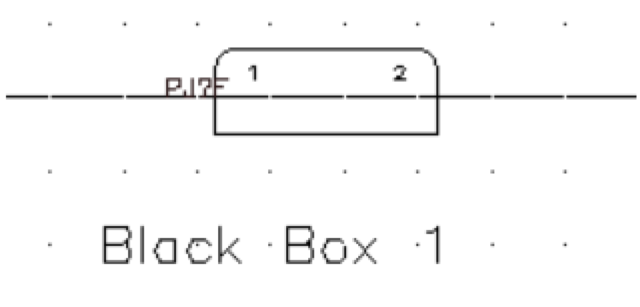
    Select to place the connector on the top of Black Box 1



14. Click Schematic ➤ Edit Components panel ➤ Modify Connectors drop-down ➤ Rotate Connector.
15. Respond to the prompts as follows:

    Select connector to Rotate or [Hold]:

    Select the new connector, right-click, and select Enter



Adding Wire Numbers

Add wire numbers to the connector diagram.

Wire numbers are blocks or attributes inserted on a line wire entity. AutoCAD Electrical toolset assigns each wire number type to its own layer. You can assign a different color to each of these layers so you can easily tell them apart. The wire number placement is set to in-line as defined on the Drawing Properties ➤ Wire Numbers dialog box.

Insert wire numbers

1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Insert Wire Numbers drop-down ➤ Wire Numbers.
2. On the Wire Tagging dialog box, specify:

   Wire Tag Mode: Sequential

   Start: 100

3. Click Drawing-Wide.

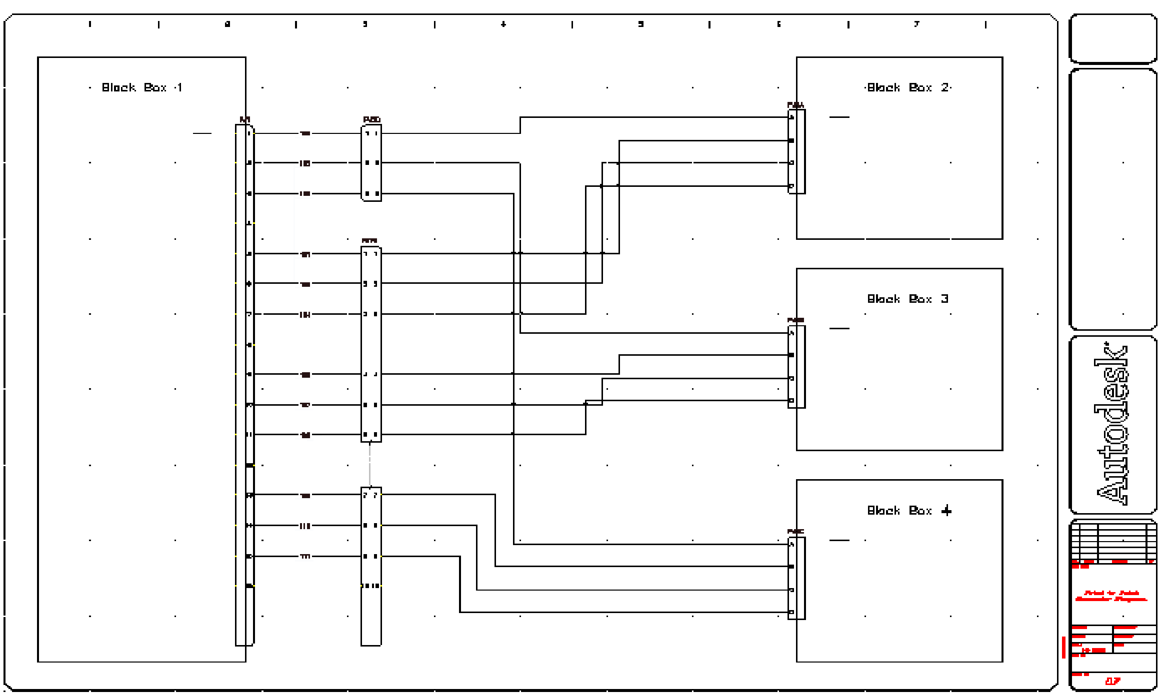
The wire numbers are automatically inserted into the drawing starting with number 100.

4. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Move Wire Number.
5. Respond to the prompts as follows:

   Specify new Wire Number location (select on wire):

   Select each wire closest to Black Box 1, right-click

   Note: You can align the wire numbers using the Align tool.



Adding Connector Descriptors

Add description values to the plug and receptacle sides of a connector.

AutoCAD Electrical toolset supports two lines of description text on each connector: one for the plug and one for the receptacle side of the connector.

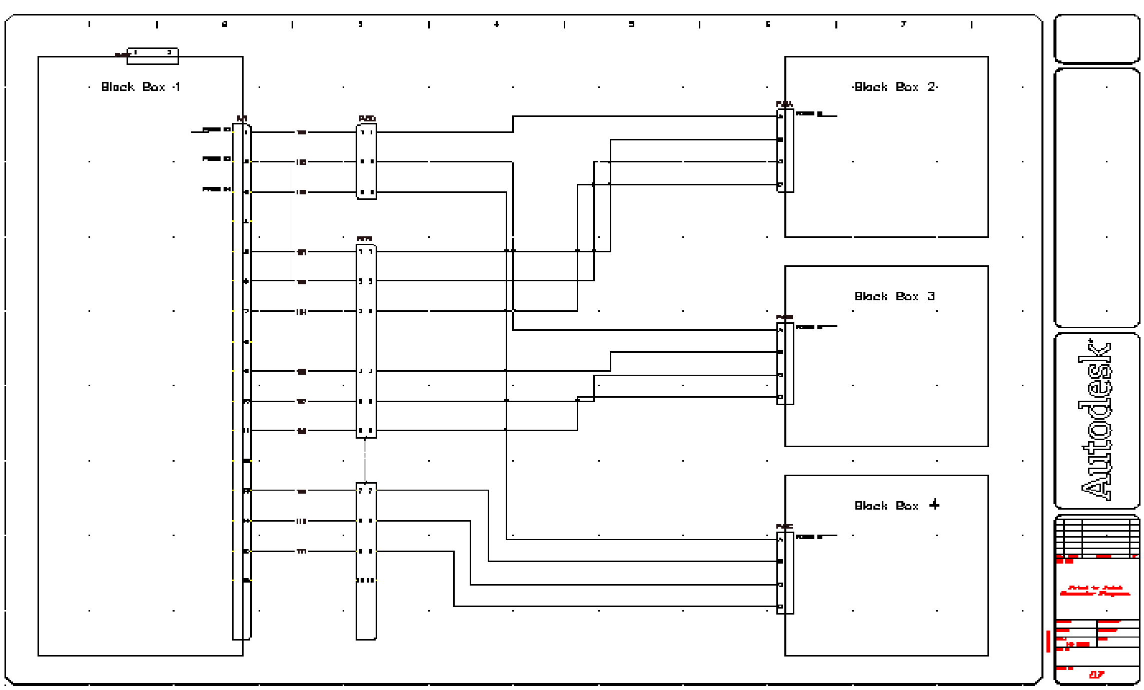
Add descriptions

1. Right-click connector PJ1 and select Edit Component.
2. On the Insert/Edit Component dialog box, Pins section, click List.
3. On the Connector Pin Numbers In Use dialog box, connector pin grid, click in the Description column for Pin 1.
4. On the Pin Descriptions section, enter POWER B2 for the Receptacle.
5. On the connector pin grid, click in the Description column for Pin 2.
6. On the Pin Descriptions section, enter POWER B3 for the Receptacle.
7. On the connector pin grid, click in the Description column for Pin 3.
8. On the Pin Descriptions section, enter POWER B4 for the Receptacle.

Sheet,Reference	Plug	Description	Rece...	Description	Wire Numbers
01, ?	1		1	POWERB2	100,
01, ?	2		2	POWERB3	102,
01, ?	3		3	POWERB4	105,

9. Click OK.
10. On the Insert/Edit Component dialog box, click OK.
11. Repeat to add the description POWER IN for Pins A on Black Box 2, Black Box 3 and Black Box 4.

    Your finished point-to-point diagram looks like the following image.



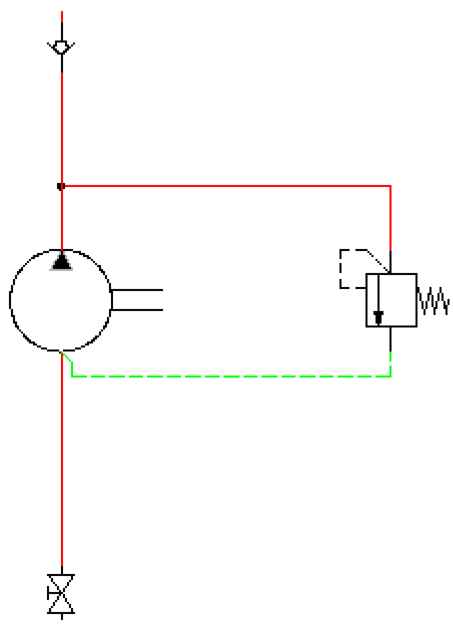
P&ID and Hydraulic Diagrams Tutorial

Create Piping & Instrumentation (P&ID) and Hydraulic drawings. The same workflow can be applied for Pneumatics.

Time required	65 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\P&ID to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics below to accomplish these tasks:

- Set up hydraulic and P&ID drawings
- Insert hydraulic and P&ID symbols
- Create pipes

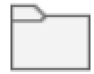


Setting Up Hydraulic Drawings

Use the Project Manager to manage your hydraulic drawings.

From here, you can create a drawing and modify any drawing properties.

Create a new drawing

1. Click Project tab ➤ Project Tools panel ➤ Manager. 
2. If AEGS is not the active project, activate the AEGS project.

If AEGS is in the list of open projects:

- Select AEGS and right-click.
- Click Activate.

If AEGS is not in the list of open projects:

- Select the project list drop-down.
- Click Open Project.
- On the Select Project File dialog box, navigate to and select the AEGS.WDP file.
- Click Open.

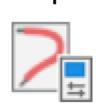
3. In the Project Manager, right-click the project name, and select Properties.
4. In the Project Properties ➤ Project Settings dialog box, click Default to switch on all paths for pneumatic, hydraulic, and P&ID schematic libraries.
5. Click OK.
6. In the Project Manager, click the New Drawing tool. 
7. In the Create New Drawing dialog box, specify:

    Name: AEGS12
    Template: *Mouse over the edit box to verify ACAD_Electrical.dwt is specified*

    If ACAD_Electrical.dwt is not specified, click Browse. Select it from the list of available templates.

    Description 1: Hydraulic Example
    Click OK.

    Note: If you want to set the component, wire number, cross-reference, style, and drawing format settings, click OK-Properties to proceed to Drawing Properties dialog box.

8. Enter DSETTINGS at the command prompt.
9. In the Drafting Settings dialog box ➤ Snap and Grid tab, turn on Snap and Grid and set the size of both to 0.125.
10. Click OK.
11. Click Schematic tab ➤ Other Tools panel ➤ Drawing Properties. 
12. In the Drawing Properties dialog box ➤ Drawing Format tab, Scale section, make sure that the feature scale multiplier is set to 1.0 inch.

13. Click OK.

    Note: For metric unit, the following settings are recommended so that the wire connection points are placed on the grids for easier drafting. Grid and Snap Size = 2.5 mm; Feature scale multiplier =20 (scale factor = 20).

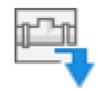
Inserting Hydraulic Schematic Symbols

Insert hydraulic components from the icon menu.

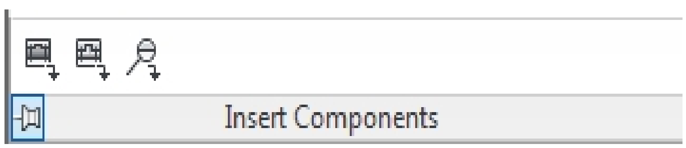
The hydraulic symbol library in AutoCAD Electrical toolset includes filters, valves, cylinders, pressure switches, motors, pumps, meters, restrictors, quick disconnects, flow arrows and more. The hydraulic symbol library consists of all the hydraulic symbols. It is found at \Users\Public\Documents\Autodesk\Acade {version}\Libs\hyd_iso125.

Insert hydraulic symbols

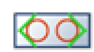
1. Click Schematic ➤ Insert Components panel ➤ ▾ ➤ Insert Hydraulic Components.

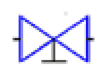


Note: By default, an expanded panel closes automatically when you click another panel. To keep a panel expanded, click the push pin icon in the bottom-left corner of the expanded panel.



2. In the Insert Component: Hydraulic Symbol dialog box, select the check box for Vertical.
3. In the Insert Component: Hydraulic Symbol dialog box, click the General Valves icon.
4. In the Hydraulic: General Valves dialog box, click Shut Off Valve Open.
5. Respond to the prompts as follows:





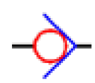
    Specify insertion point:

    Select to place the valve in the upper left corner of your drawing

6. In the Insert/Edit Component dialog box, specify:

    Component Tag: VAL2
    Click OK.

7. Repeat steps 1 - 3.
8. In the Hydraulic: General Valves dialog box, click Checkvalve Flow Left.
9. Respond to the prompts as follows:



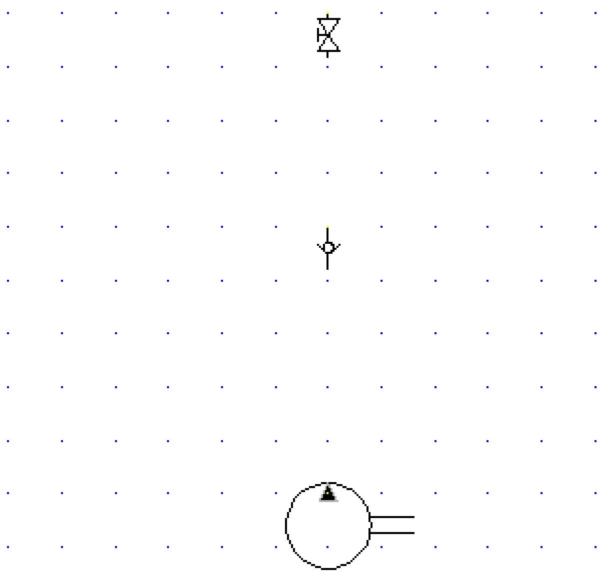
    Specify insertion point:

    Select to place the check valve below the shut off valve

10. In the Insert/Edit Component dialog box, click OK.

11. Click Schematic tab ➤ Insert Components panel ➤ ➤ Insert Hydraulic Components.
12. In the Insert Component: Hydraulic Symbol dialog box, click Motors & Pumps.
13. In the Hydraulic: Motors and Pumps dialog box, click Fixed Displacement.
14. In the Hydraulic: Fixed Displacement dialog box, click Uni-Directional Pump.
15. Respond to the prompts as follows:

Specify insertion point: *Select to place the pump below the check valve*

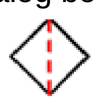


16. In the Insert/Edit Component dialog box, specify:

    Description: Line 1: Hydraulic Oil Pump
    Click OK.

17. Insert another Shut Off Valve Open below the Hydraulic Oil Pump.
18. Click Schematic tab ➤ Insert Components panel ➤ ➤ Insert Hydraulic Components.

19. In the Insert Component: Hydraulic Symbol dialog box, click Filters.



20. In the Hydraulic: Filters dialog box, click Filter.
21. Respond to the prompts as follows:

Specify insertion point: *Select to place the filter below the shut off valve*

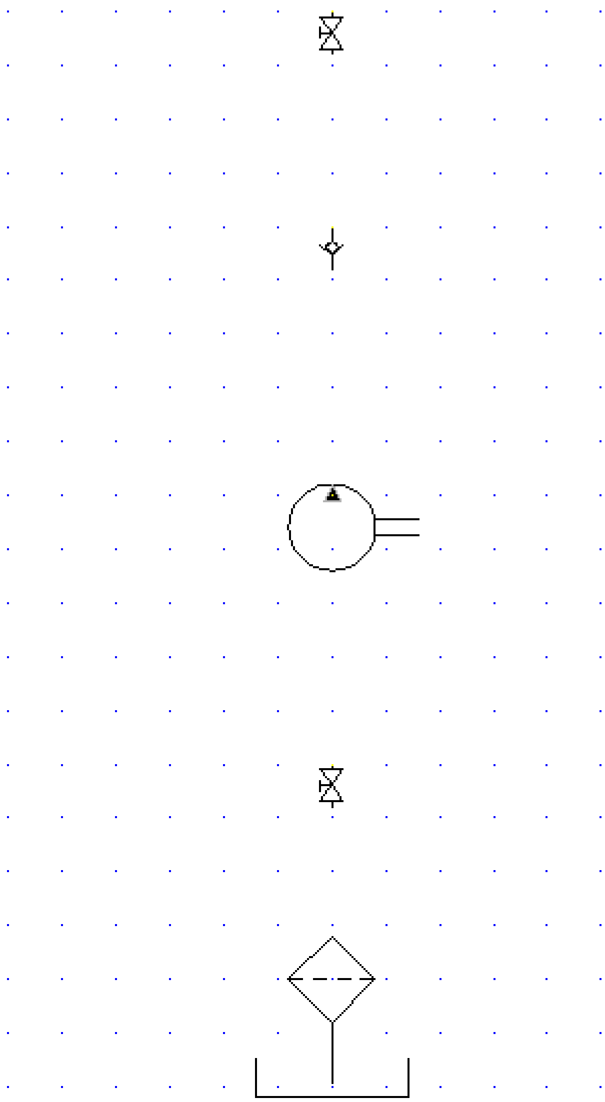
22. In the Insert/Edit Component dialog box, specify:

    Component Tag: FI2
    Description: Line 1: Filter
    Click OK.

23. Click Schematic tab ➤ Insert Components panel ➤ ➤ Insert Hydraulic Components.

24. In the Insert Component: Hydraulic Symbol dialog box, click Miscellaneous.
25. In the Hydraulic: Miscellaneous dialog box, click Reservoir.
26. Respond to the prompts as follows:

Specify insertion point: *Select to place the reservoir below the filter*



29. In the Insert/Edit Component dialog box, click OK.

Creating Pipes

Use the Insert Wire tool to insert lines that represent pipes on a hydraulic drawing.

In AutoCAD Electrical toolset, different types of wires represent the type of running pipes that allow water or oil flows from one instrument to another. Start by setting up the type of wires for pipe runs.

Insert wires as pipes

1. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Create/Edit Wire Type.
2. In the Create/Edit Wire Type dialog box, specify:

   Wire Color: RED
   Size: 20

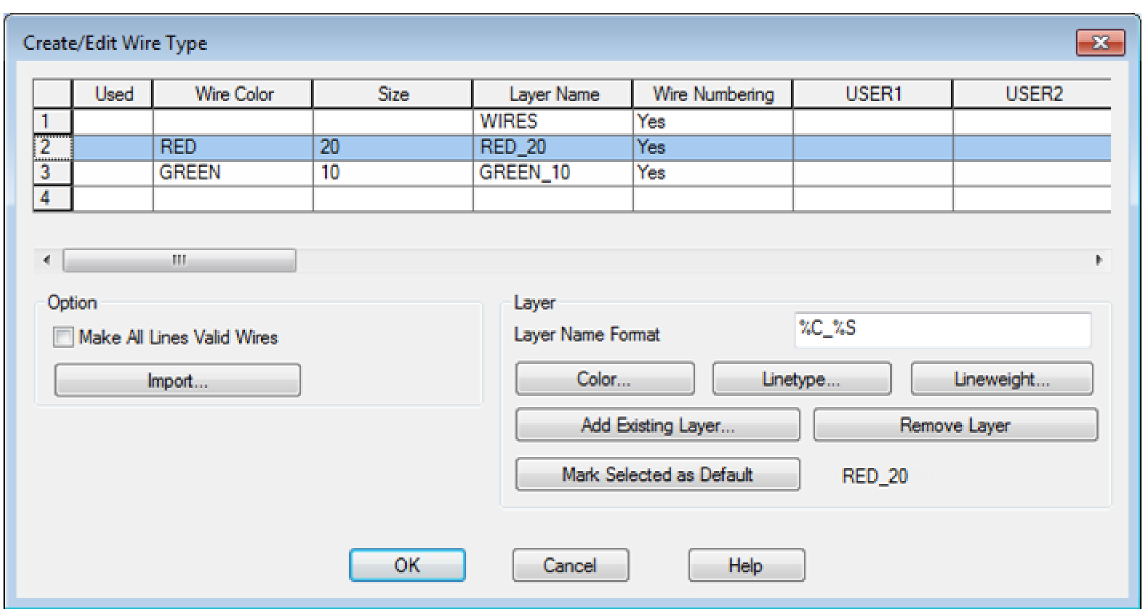
   The Layer Name is automatically created. The name RED_20 is assigned to the wire layer you are creating.

3. Click Color.
4. In the Select Color dialog box, select red and click OK.
5. Click Linetype.
6. In the Select Linetype dialog box, select Continuous and click OK.
7. In the Create/Edit Wire Type dialog box, specify:

   Wire Color: GREEN
   Size: 10
   Color: Green
   Linetype: Hidden2

   Note: If HIDDEN2 is not available, click Load. Select it from the list of line types on the Load or Reload Linetypes dialog box.

8. Select RED_20 in the grid and click Mark Selected as Default.

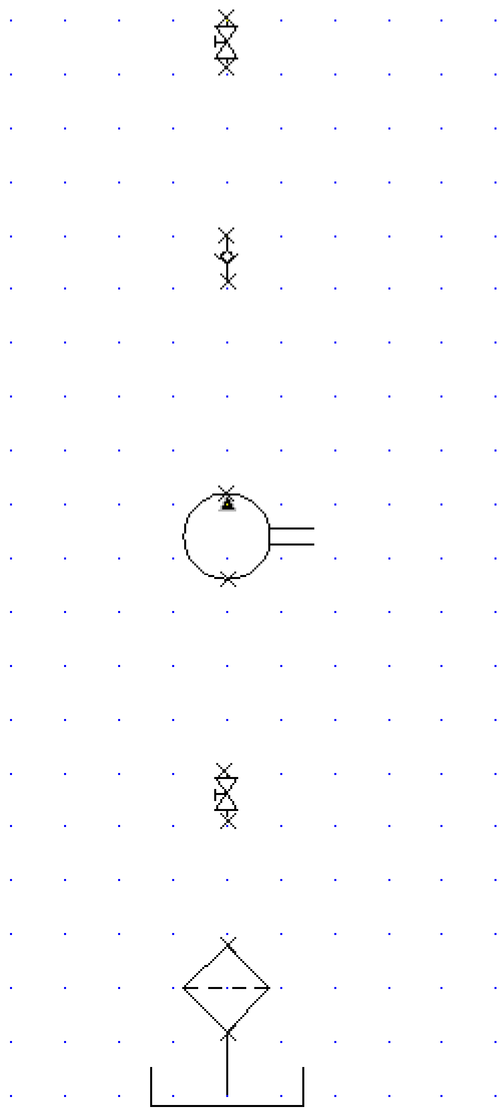


9. Click OK.

10. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Wire.
11. Respond to the prompts as follows:

Specify wire start or [wireType/X=show connections]:

Enter X *and press* ENTER

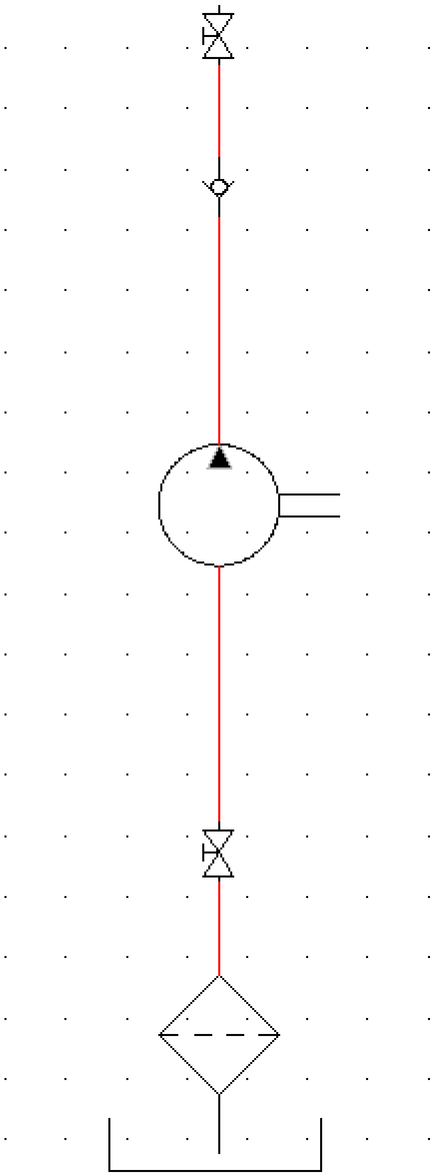


Specify wire start or [wireType/X=show connections]: *Select the bottom of the shut off valve*

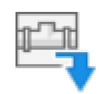
Specify wire end or [Scoot/T=wiretype, X=show connections]: *Select the top of the check valve*

12. Continue inserting wires connecting the components together. Right-click to exit the command.

Your drawing should look like the following:

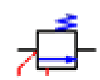


Note: You can also insert the vertical or horizontal pipes first and then insert the components onto the pipe, one at a time.



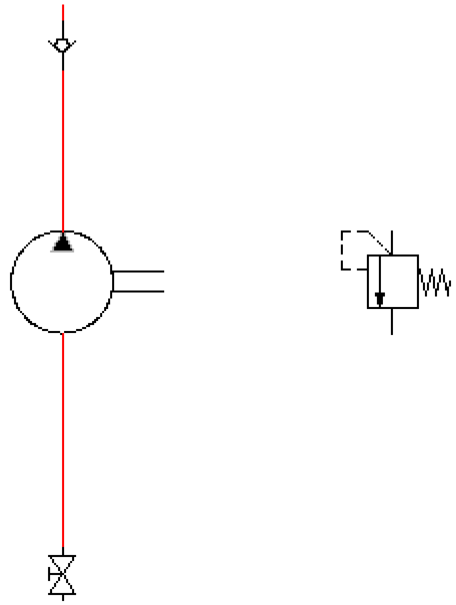
13. Click Schematic tab ➤ Insert Components panel ➤ ➤ Insert Hydraulic Components.
14. In the Insert Component: Hydraulic Symbol dialog box, select the check box for Vertical.

15. In the Insert Component: Hydraulic Symbol dialog box, click Pressure Relief Valves.
16. In the Hydraulic: Pressure Relief Valves dialog box, click N.C. Pressure Relief Valve with Preset -1.



17. Respond to the prompts as follows:

Specify insertion point: *Select to place the valve to the right of the pump*



18. In the Insert/Edit Component dialog box, specify:

    Component Tag: VAL4
    Description: Line 1: Pressure Relief
    Click OK.

19. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Wire. 
20. Respond to the prompts as follows:

    Specify wire start or [wireType/X=show connections]:

    Enter X, *press*ENTER

    Specify wire start or [wireType/X=show connections]: *Press* SHIFT + *right-click and select Midpoint from the menu, then select the midpoint on the pipe between the pump and the shut off valve above it*

    Specify wire end or [V=start Vertical/H =start Horizontal/Continue): *Drag the pipe to the right so that it is directly above the pressure relief valve. Drag the pipe down and click the top connection point on the pressure relief valve*

    You now insert a pipe that connects the end of the valve back to the pump.

    Specify wire start or [wireType/X=show connections]:

    Enter T, *press* ENTER

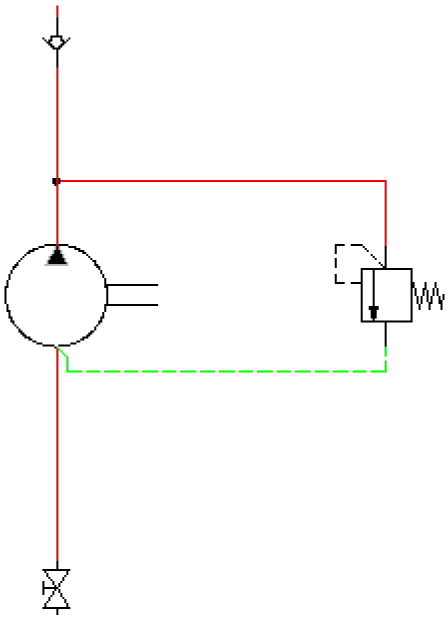
    Select the wire layer GREEN_10. Click OK.

    Tip: Make sure that Snap is turned off and that the Wire Layer is set to GREEN_10.

    Select the bottom connection point on the pump

    Specify wire end or [V=start Vertical/H =start Horizontal/Continue):

Drag the pipe down and to the right, click the connection point at the bottom of the pressure relief valve, right-click



Completing the Hydraulic Drawing

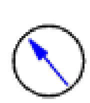
Continue to insert hydraulic components to finish the hydraulic diagram.

The rest of the hydraulic drawing consists of inserting a Pressure Gauge and Check Valve at the left side of the pump and then inserting devices (Cylinder; Restrictors; Filter; Check valve and 2-ways valve) along the top of the drawing.

Note: During insertion, clear the Vertical option in the Insert Component: Hydraulic Symbols dialog box.

Insert components

1. Click Schematic tab ➤ Insert Components panel ➤ ➤ Insert Hydraulic Components.
2. In the Insert Component: Hydraulic Symbol dialog box, click Meters.
3. In the Hydraulic: Meters dialog box, click Pressure Gauge.
4. Respond to the prompts as follows:

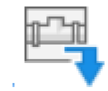
 

   Specify insertion point: *Select to place the pressure gauge to the far left (and slightly above) of the pump*

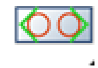
5. In the Insert/Edit Component dialog box, specify:

      Component Tag: MTR1
      Description: Line 1: Pressure Gauge

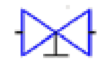
   Click OK.



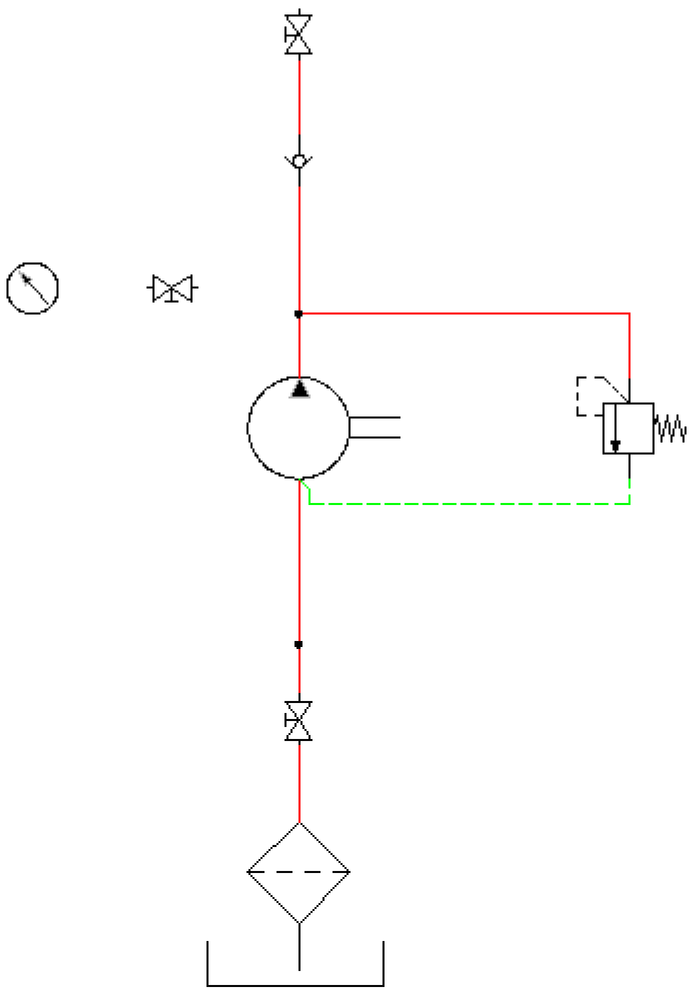
6. Click Schematic tab ➤ Insert Components panel ➤ ▾ ➤ Insert Hydraulic Components.
7. In the Insert Component: Hydraulic Symbol dialog box, deselect the Vertical check box.



8. In the Insert Component: Hydraulic Symbol dialog box, click General Valves.
9. In the Hydraulic: General Valves dialog box, click Shut Off Valve Open.
10. Respond to the prompts as follows:



Specify insertion point: *Select to place the valve to the right of the pressure gauge*



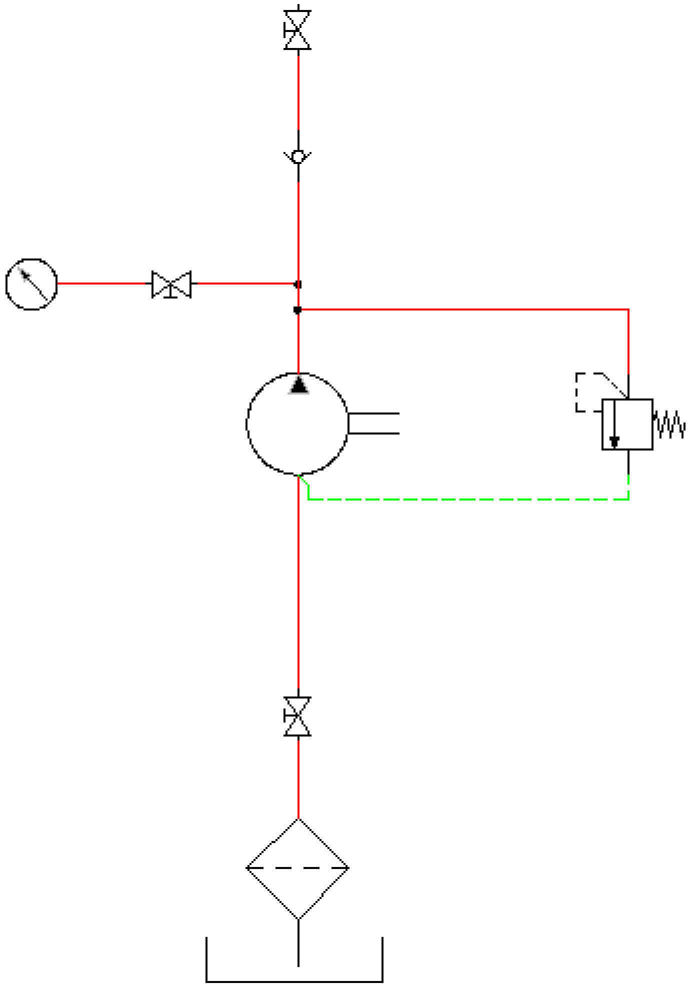
11. In the Insert/Edit Component dialog box, click OK.
12. Set the wire layer to RED_20.
13. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Wire.
14. Respond to the prompts as follows:

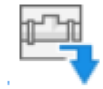
Specify wire start or [wireType/X=show connections]: *Select the right connection point on the pressure gauge*

Specify wire end or [Continue]: *Drag the pipe to the right and click the left connection point on the valve*

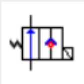
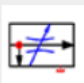
Specify wire start or [Scoot/wireType/X=show connections]: *Select the right connection point on the valve*

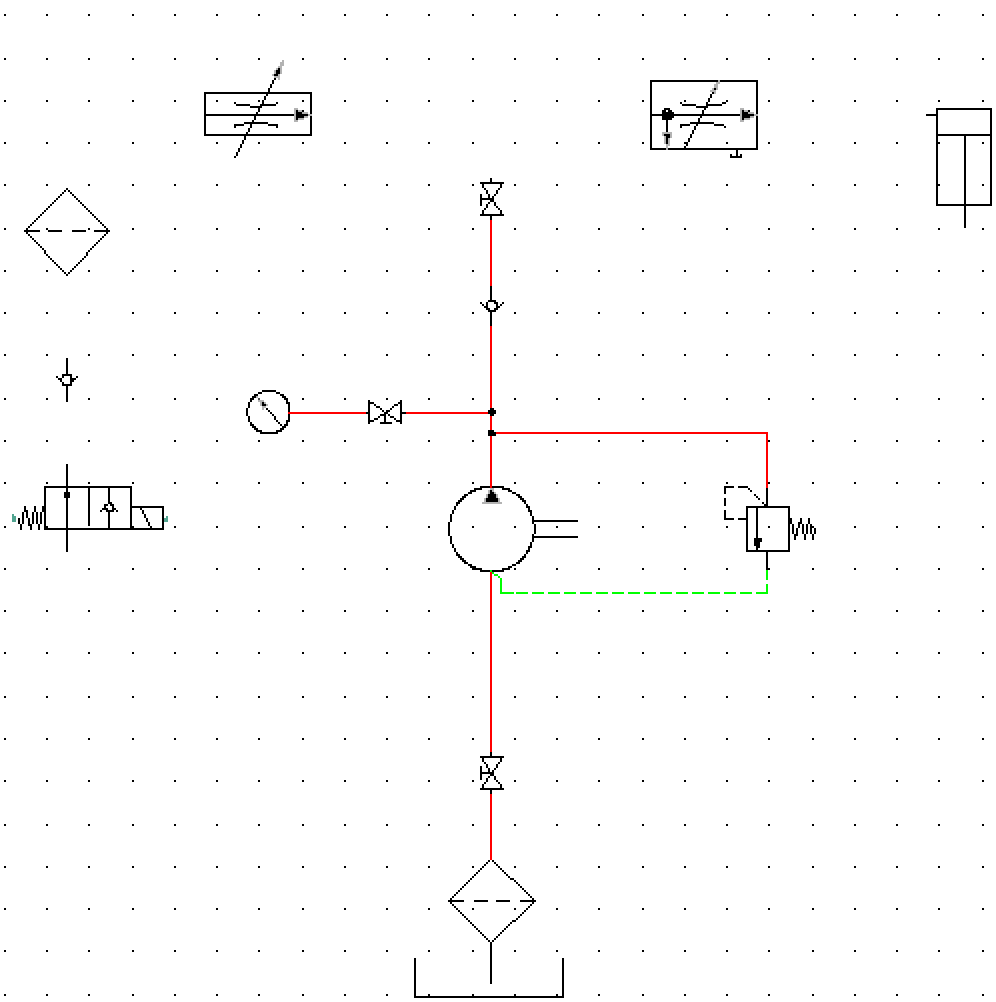
Specify wire end or [Continue]: *Drag the pipe to the right and click the vertical pipe, right-click*



15. Click Schematic tab ➤ Insert Components panel ➤ ➤ Insert Hydraulic Components. 
16. Insert and place the devices listed as shown in the following illustration. In the Insert/Edit Component dialog box, click OK after each insertion.

    Note: You can also insert the vertical or horizontal pipes first and then insert the components onto the pipe, one by one.

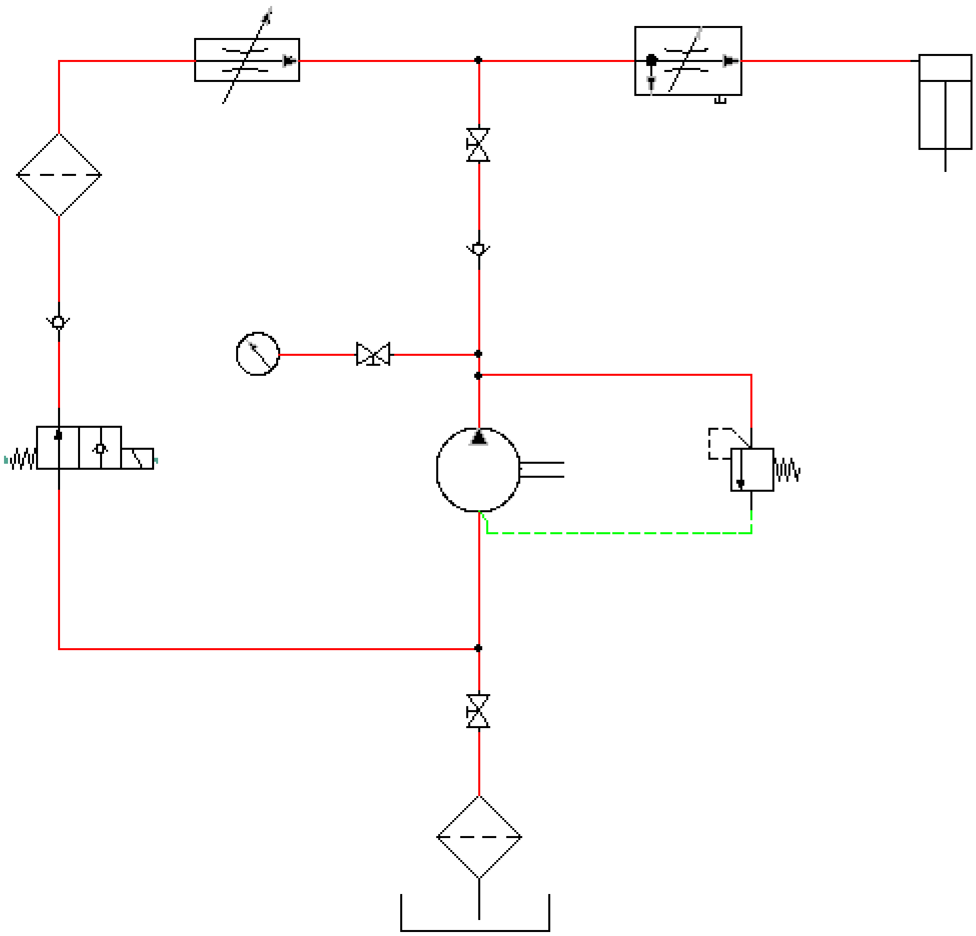
Icon	Symbol to Insert
	2 Way Valves ➤ Solenoid Spring Return -1 (insert as Vertical symbol)
	General Valves ➤ Checkvalve Flow Left (insert as a Vertical symbol)
	Filters ➤ Filter (insert as a Vertical symbol)
	Restrictors ➤ Restrictor with Variable Output Flow
	Restrictors ➤ By-Pass Flow Regulator with Variable Output Flow
	Cylinders ➤ Single Acting Single Ended Piston Rod



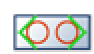
Tip: Align the components horizontally and vertically using the Align tool to make inserting the pipes easier.

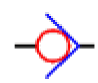
17. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Wire.

18. Connect the pipes from one control device to another as illustrated.

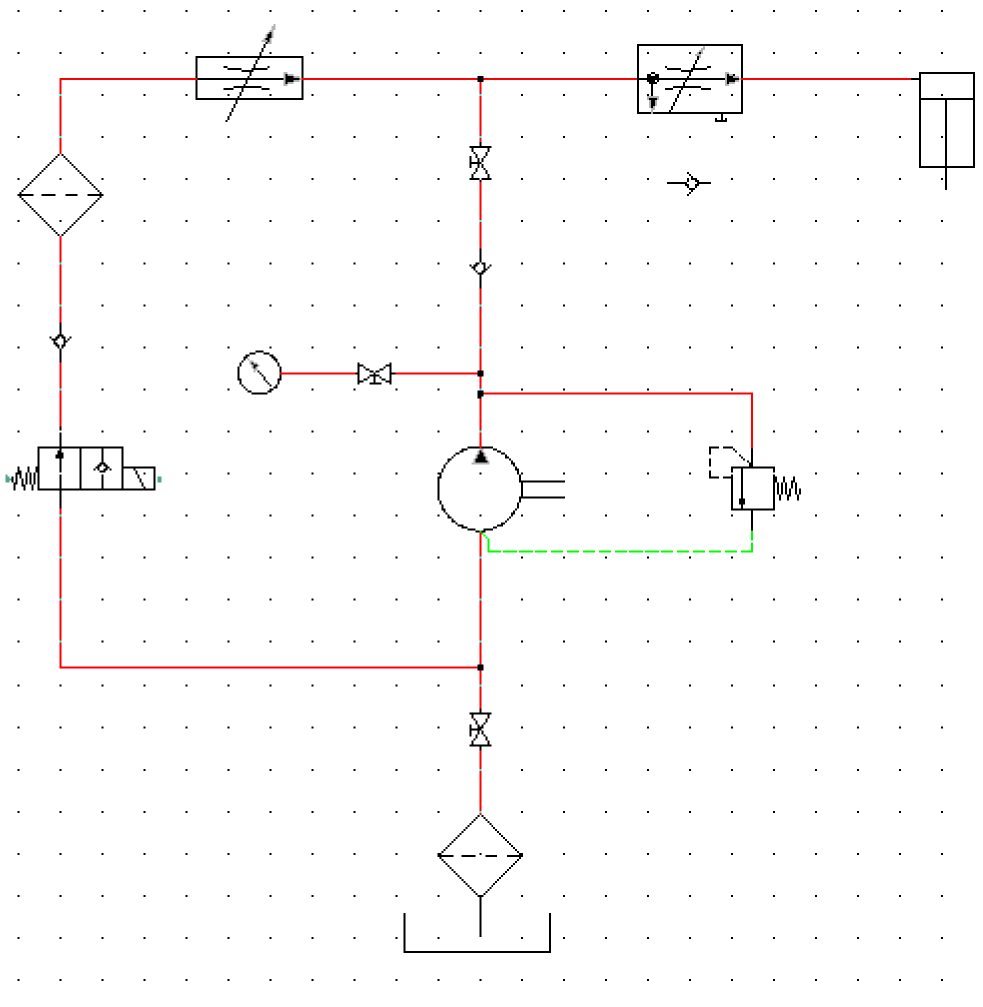


19. Click Schematic ➤ Insert Components panel ➤ ➤ Insert Hydraulic Components.
20. In the Insert Component: Hydraulic Symbol dialog box, click General Valves.
21. In the Hydraulic: General Valves dialog box, click Checkvalve Flow Left.
22. Respond to the prompts as follows:

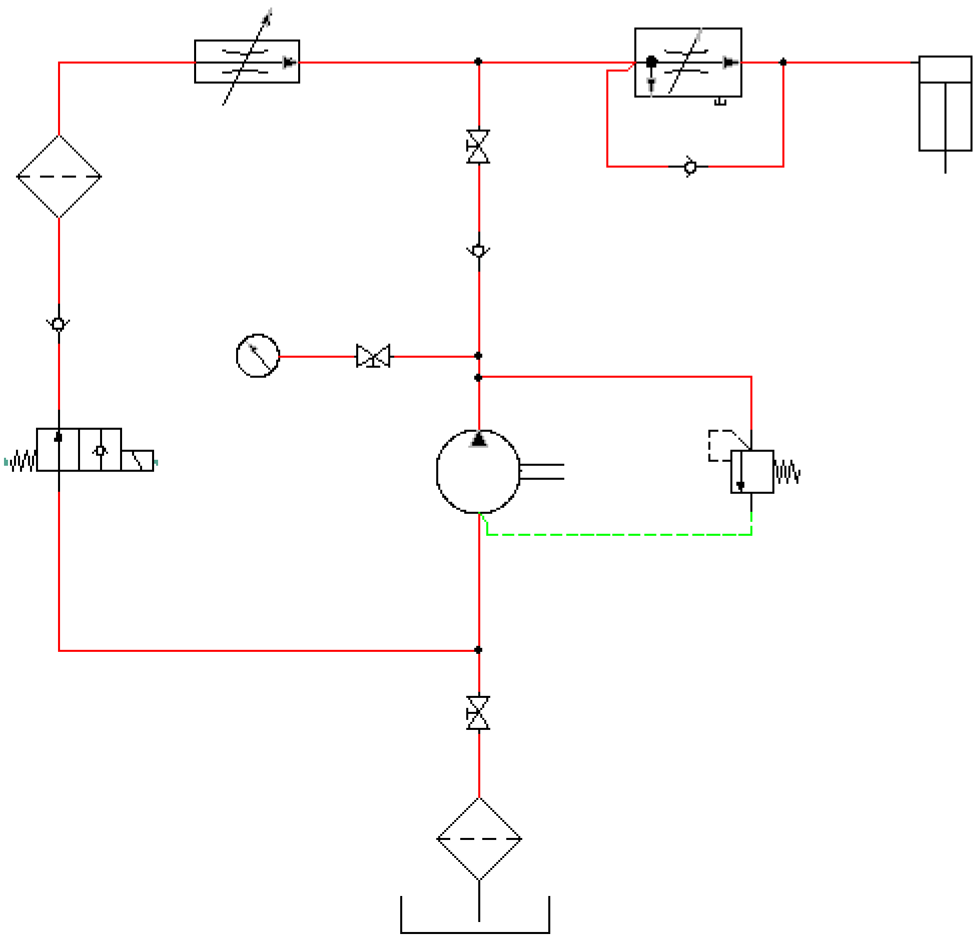




Specify insertion point: *Select to place the valve below the restrictor*



23. In the Insert/Edit Component dialog box, click OK.
24. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Wire.
25. Connect the pipes as shown.



The hydraulic schematic diagram is complete.

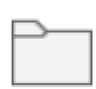
If you want to create a pneumatic drawing, use the Insert Pneumatic Components tool on the Schematic ➤ Insert Components panel. Refer to the pneumatic demo drawing file (Demo03.dwg) in the Extra Library Demo project.

Setting Up P&ID Drawings

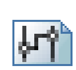
Use the Project Manager to manage your P&ID drawings.

From here, you can create a drawing and modify any drawing properties.

Create a new drawing



1. Click Project tab ➤ Project Tools panel ➤ Manager.
2. In the Project Manager, click the New Drawing tool.
3. In the Create New Drawing dialog box, specify:



Name: AEGS13
Template: *Mouse over the edit box to verify* ACAD_Electrical.dwt *is specified*
If ACAD_Electrical.dwt is not specified, click Browse. Select it from the list of available templates.

Description 1: P&ID Example

Click OK.

Note: If you want to set the component, wire number, cross-reference, style, and drawing format settings, click OK-Properties to proceed to Drawing Properties dialog box.

4. Enter DSETTINGS at the command prompt.
5. In the Drafting Settings dialog box ➤ Snap and Grid tab, turn on Snap and Grid and set the size of both to 0.125.
6. Click OK.
7. Click Schematic tab ➤ Other Tools panel ➤ Drawing Properties.
8. In the Drawing Properties dialog box ➤ Drawing Format tab, Scale section, make sure that the feature scale multiplier is set to 1.0 inch.
9. Click OK.

Note: For metric unit, the following settings are recommended so that the wire connection points are placed on the grids for easier drafting. Grid and Snap Size = 2.5 mm; Feature scale multiplier =20 (scale factor = 20).

Set up wire layers

1. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Create/Edit Wire Type.
2. In the Create/Edit Wire Type dialog box, click in the Wire Type #2 row and specify:

   Wire Color: RED
   Size: 25

   The Layer Name is automatically created. The name RED_25 is assigned to the wire layer you are creating.

3. Click Color.
4. In the Select Color dialog box, select red and click OK.
5. Click Linetype.
6. In the Select Linetype dialog box, select Continuous and click OK.
7. Click Lineweight.
8. In the Select Lineweight dialog box, select 0.30 and click OK.

   For this example, create three more wire types using the Create/Edit Wire Type dialog box.

9. In the Create/Edit Wire Type dialog box, specify:

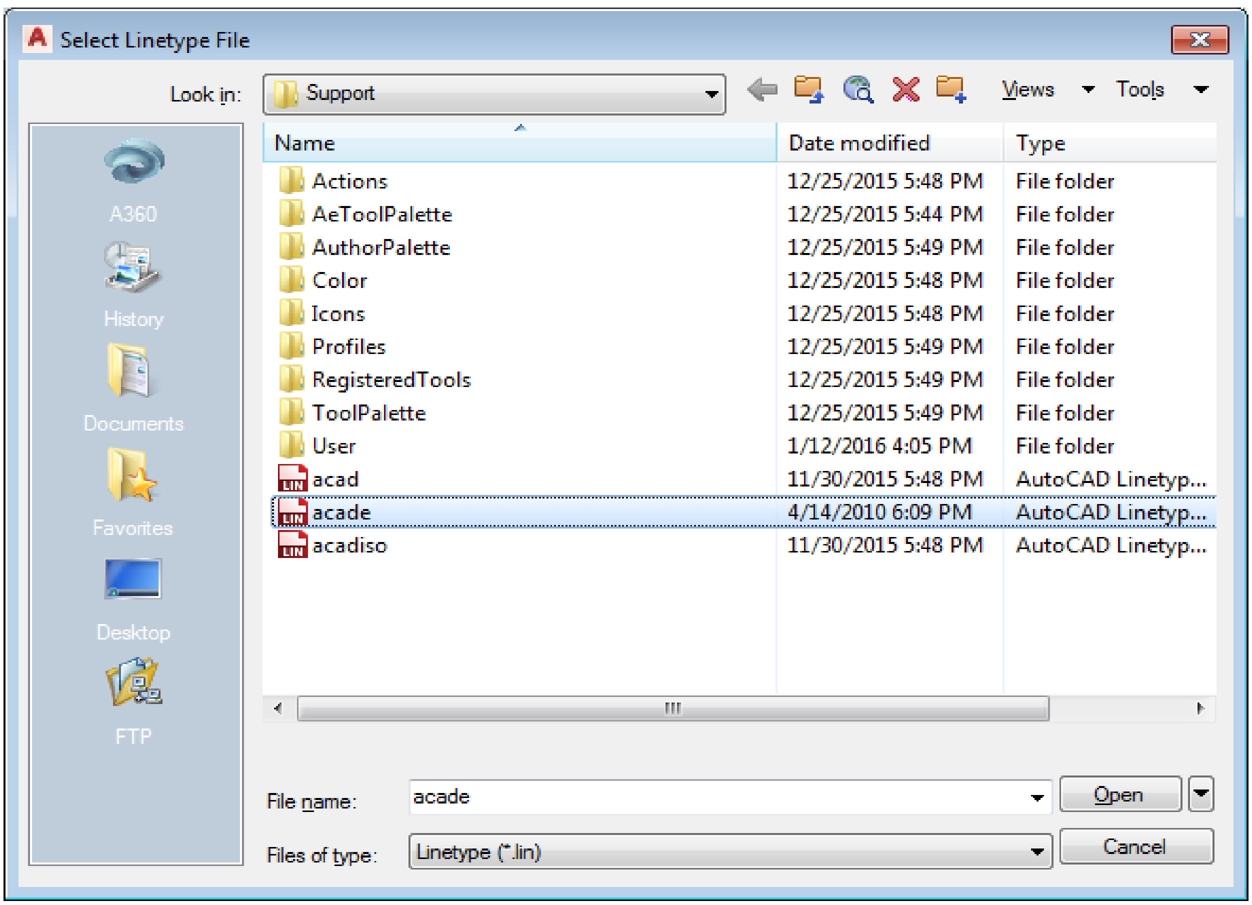
   Wire Type #3
   Wire Color: RED
   Size: 10
   Color: Red
   Linetype: Hidden2
   Lineweight: default
   Wire Type #4
   Wire Color: GREEN
   Size: 10
   Color: Green

	Used	Wire Color	Size	Layer Name	Wire Numbering	USER1	USER2
1				WIRES	Yes		
2	X	RED	25	RED_25	Yes		
3	X	RED	10	RED_10	Yes		
4		GREEN	10	GREEN_10	Yes		
5							

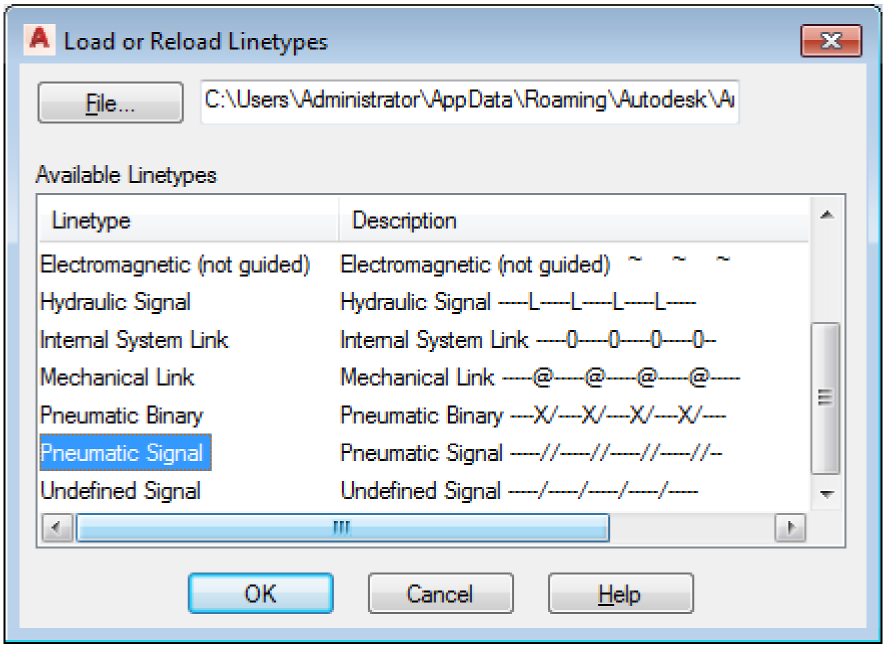
Note: For pipe runs in P&ID drawings, include the different linetypes from the acade.lin file. You can set up the wire types for pipes at the beginning of the drawing or before creating the pipes.

10. To set the Linetype for the GREEN_10 wire layer, click Linetype.
11. In the Select Linetype dialog box, click Load.
12. In the Load or Reload Linetypes dialog box, click File.
13. In the Select Linetype File dialog box, select acade.lin and click Open.

Note: The default location for the *acade.lin* file is \Users\{username}\AppData\Roaming\Autodesk\AutoCAD Electrical {version}\{release number}\{country code}\Support.



14. In the Load or Reload Linetypes dialog box, select Pneumatic Signal and click OK.



15. In the Select Linetype dialog box, select Pneumatic Signal and click OK.
16. In the Create/Edit Wire Type dialog box, click OK.

Inserting P&ID Schematic Symbols

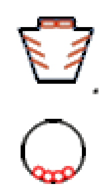
Insert P&ID components from the icon menu.

The P&ID symbol library in AutoCAD Electrical toolset includes equipment, tanks, nozzles, pumps, fittings, valves, actuators, logic functions, instrumentation, flow, and flow arrows. The P&ID symbol library consists of all the piping and instrumentation symbols. It is found at \Users\Public\Documents\Autodesk\Acade {version}\Libs\Pid.

Insert P&ID Symbols

1. Click Schematic tab ➤ Insert Components panel ➤ ➤ Insert P&ID Components.
2. In the Insert Component: Piping and Instrumentation Symbols dialog box, click Equipment.
3. In the PID: Equipment dialog box, click Ball Mill.
4. Respond to the prompts as follows:

   Specify insertion point: *Select to place the ball mill in the upper left corner of your drawing*

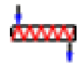
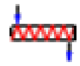


5. In the Insert/Edit Component dialog box, specify:

Component Tag: C-100

Description: Line 1: BALL MILL

Click OK.

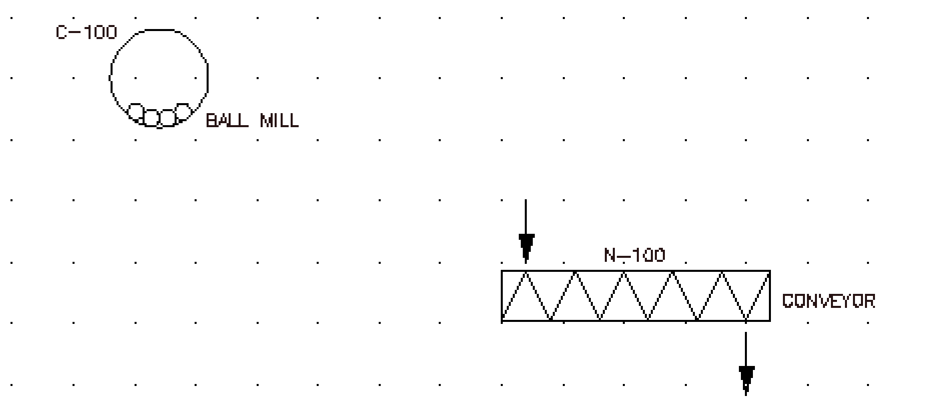
6. Repeat steps 1-2.
7. In the PID: Equipment dialog box, click Conveyors. 
8. In the PID: Conveyors dialog box, click Conveyor 1. 
9. Respond to the prompts as follows:

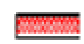
   Specify insertion point: *Select to place the conveyor to the right and diagonally below the ball mill*

10. In the Insert/Edit Component dialog box, specify:

    Component Tag: N-100
    Description: Line 1: CONVEYOR

    Click OK.



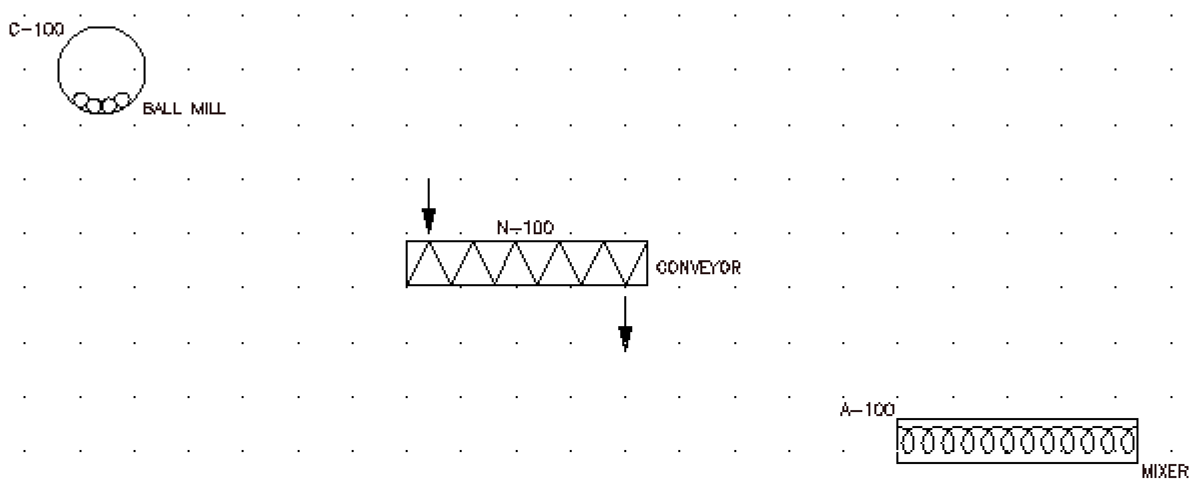
11. Repeat steps 1-2.
12. In the PID: Equipment dialog box, click Mixer 2. 
13. Respond to the prompts as follows:

Specify insertion point: *Select to place the mixer to the right and diagonally below the conveyor*

14. In the Insert/Edit Component dialog box, specify:

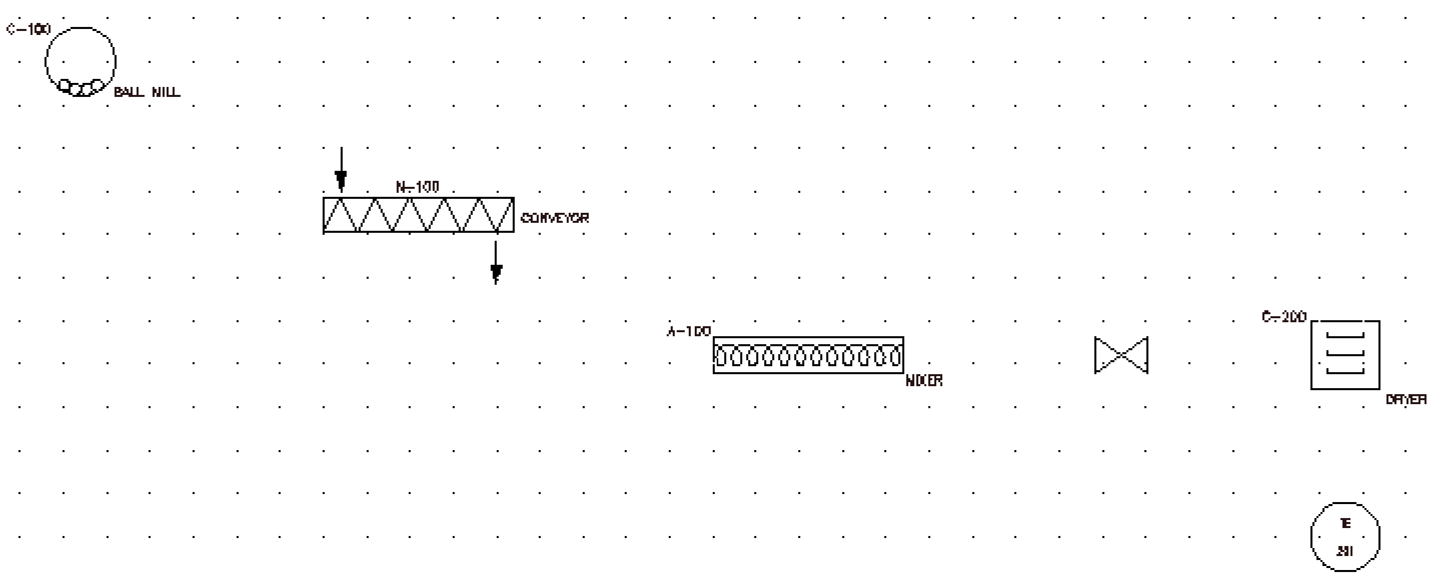
    Component Tag: A-100
    Description: Line 1: MIXER

    Click OK.



15. Click Schematic ➤ Insert Components panel ➤ ➤ Insert P&ID Components.
16. Insert and place the devices listed as shown in the following illustration. In the Insert/Edit Component dialog box, click OK after each insertion.

Icon	Symbol to Insert
	Valves ➤ Gate Valve In the Insert/Edit Component dialog box, clear the Component Tag
	Equipment ➤ Dryer Component Tag = C-200; Description Line 1 = DRYER
	Instrumentation ➤ Discrete Instruments ➤ Field Mounted Component Tag = TE 201



Tip: Align the components horizontally and vertically using the Align tool to make inserting the pipes easier.

Creating Pipes

Use the Insert Wire tool to insert lines that represent pipes on a P&ID diagram.

In AutoCAD Electrical toolset, different types of wires represent the type of running pipes that allow water or oil flows from one instrument to another.

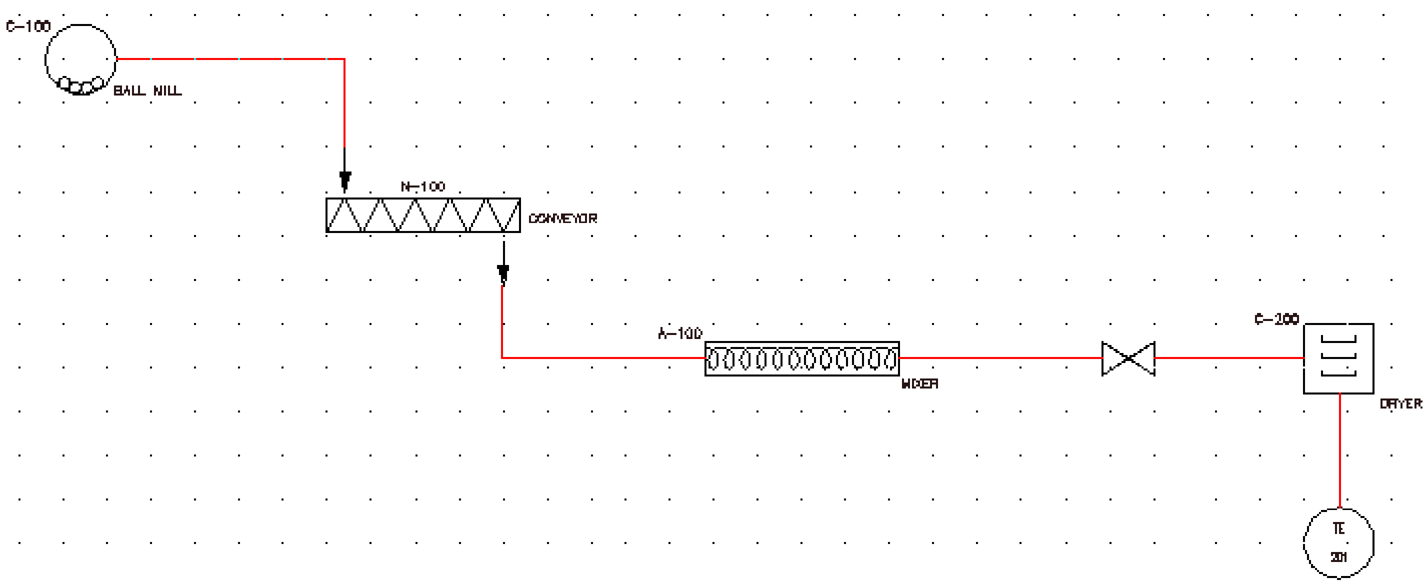
Insert wires as pipes

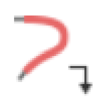
1. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Wire.
2. Change the wire type to RED_25:

   Specify wire start or [wireType/X=show connections]: *Enter* T, *press*ENTER

   Select the wire layer RED_25. Click OK.

3. Connect the pipes as shown. Right-click to exit the command.





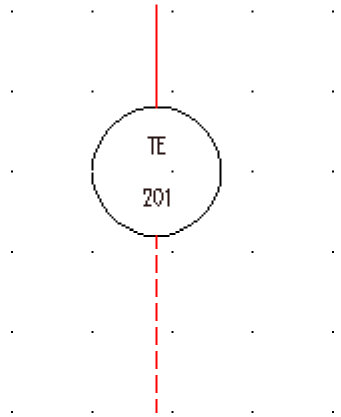
4. Click Schematic tab ➤ Insert Wires/Wire Numbers panel ➤ Wire.
5. Respond to the prompts as follows:

   Specify wire start or [wireType/X=show connections]: *Enter* T, *press*ENTER

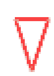
   Select the wire layer RED_10. Click OK.

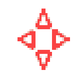
   Select the bottom of the discrete instrument

   Specify wire end or [Continue]: *Drag the wire down a few spaces, press* ENTER



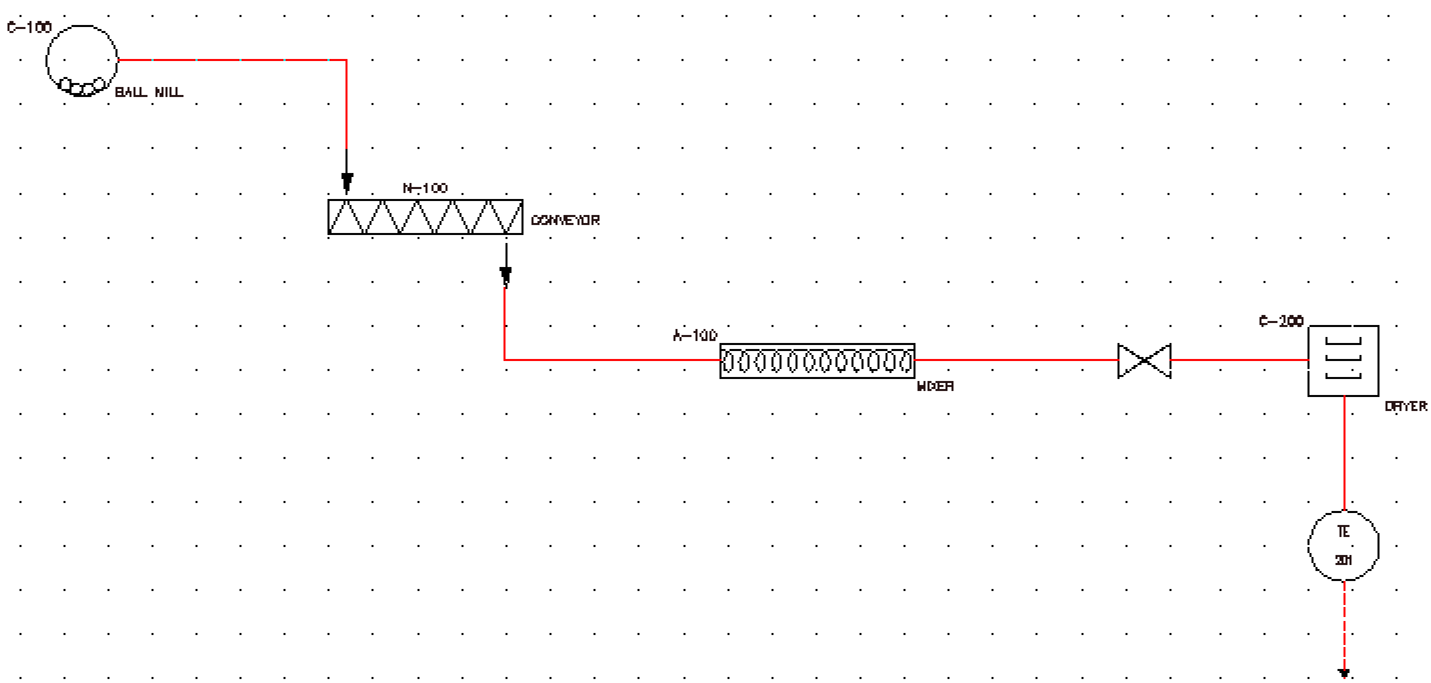
6. Click Schematic tab ➤ Insert Components panel ➤  ➤ Insert P&ID Components.
7. In the Insert Component: Piping and Instrumentation Symbols dialog box, click Flow Arrows.
8. In the PID: Equipment dialog box, click Flow Arrow Down.
9. Respond to the prompts as follows:





   Specify insertion point:

   Select to place the flow arrow at the bottom of the new wire



The P&ID diagram is complete.

If you want to see how to expand the P&ID drawing, refer to the P&ID demo drawing file (Demo01.dwg) in the Extra Library Demo project.

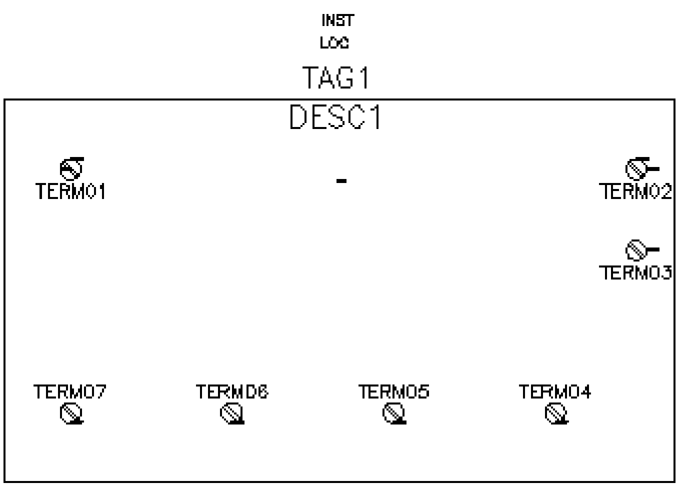
Symbol Builder Tutorial

Create custom symbols with Symbol Builder.

Time required	30 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Symbol Builder to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to accomplish these tasks:

- Create a schematic parent
- Add attributes
- Add wire connections
- Save the symbol



Creating Custom Symbols

Use the Symbol Builder to create an AutoCAD Electrical toolset symbol.

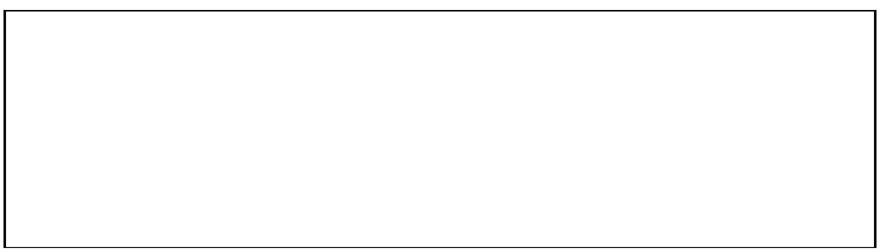
This utility builds a smart schematic symbol by either adding AutoCAD Electrical toolset attributes to the geometry of the symbol, or by converting text entities to AutoCAD Electrical toolset attributes. You can also use AutoCAD attribute definition and editing commands to do the same thing. This tool makes the task easier because you quickly pick and place attributes. It tracks what attributes are present and checks your work to make sure that any required attributes are not omitted.

Note: If you exit out of the Symbol Builder, restart it. On the Select Symbol/Objects dialog box, click Select objects and select any graphics and attributes you added so far. You can then start from where you left off.

Create a parent schematic symbol

1. If AEGS is not the active project, in the Project Manager, right-click AEGS and select Activate.
2. In the Project Manager, double-click AEGS to expand the drawing list.
3. Open AEGS03.dwg.
4. Draw a rectangle anywhere on the drawing.

   Tip: It is easiest to draw it in the white space on the left-hand side of the drawing.



5. Click Schematic tab ➤ Other Tools panel ➤ Symbol Builder drop-down ➤ Symbol Builder.
6. In the Select Symbol/Objects dialog box, Attribute template section, browse to the Library path C:\Users\Public\Documents\Autodesk\Acade {version}\Libs\jic125.
7. In the Attribute template section, select Symbol: Horizontal Parent, Type: Generic.
8. In the Select from drawing section, click Select objects, select the rectangle, and press ENTER.
9. Select OK.

Adding Attributes

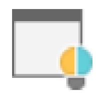
Add the attributes TAG1, DESC1, LOC, INST, FAMILY, MFG, CAT, and ASSYCODE to the custom symbol.

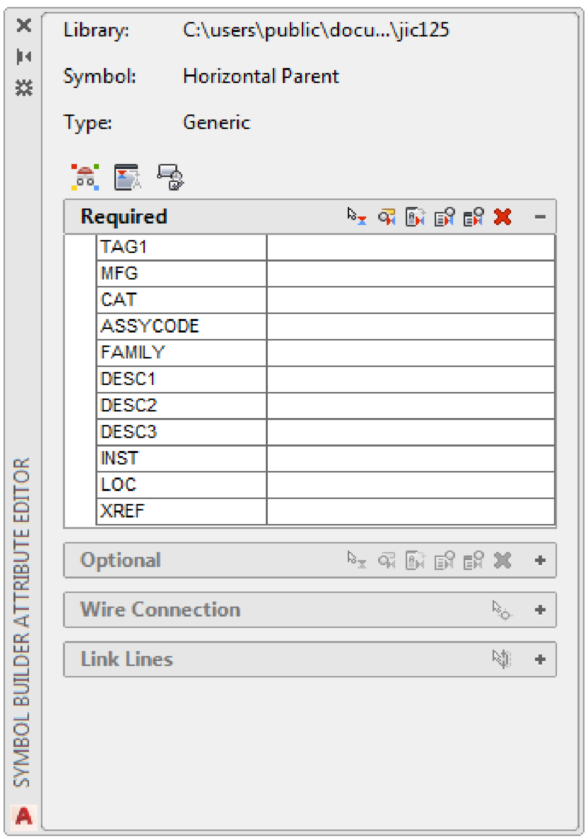
You are not limited to these attributes and you can include your own user-defined attributes on the AutoCAD Electrical toolset block files.

Note: The TAG1 attribute is the only one required for a parent schematic symbol. The other attributes in the Required section are expected on a parent schematic symbol, however the symbol is recognized as a parent symbol without them.

Add attributes

1. If the Symbol Builder Attribute Editor is not visible,

   Click Symbol Builder tab ➤ Edit panel ➤ Palette Visibility Toggle. 



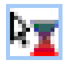
Use this palette to assign attributes to the rectangle as well as set the height and justification for each attribute. The palette displays the AutoCAD Electrical toolset attributes that you can insert and define as part of the symbol. Once an attribute is inserted on the symbol a check mark displays next to it and you cannot insert it again. AutoCAD Electrical toolset allows only one insertion of each attribute.

2. In the Symbol Builder Attribute Editor, select TAG1 and click the Properties tool. 

   Enter:

Value: PS
It is the default code used as the %F value of the tag format (such as “CR” , “PB”, “LT”)
Height: 0.125
Justify: Center

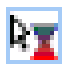
Click OK.

3. Click the Insert Attribute tool. 

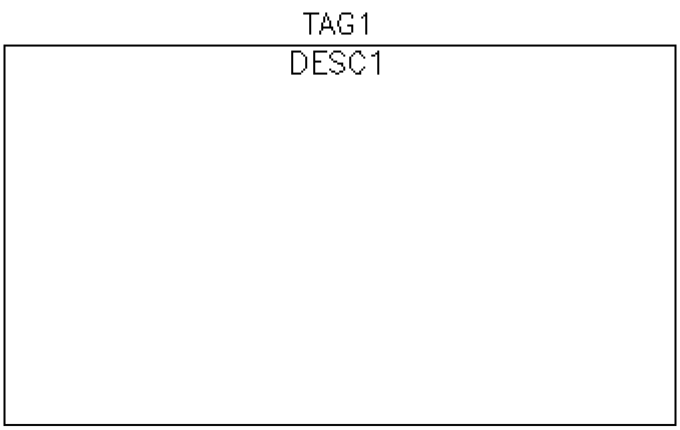
   Insert the attribute above the rectangle.

   In the Symbol Builder Attribute Editor, notice the check mark next to the TAG1 attribute. Continue placing the rest of the attributes.

4. In the Symbol Builder Attribute Editor select DESC1.

   Click the Insert Attribute tool. 

5. Insert the attribute below TAG1.

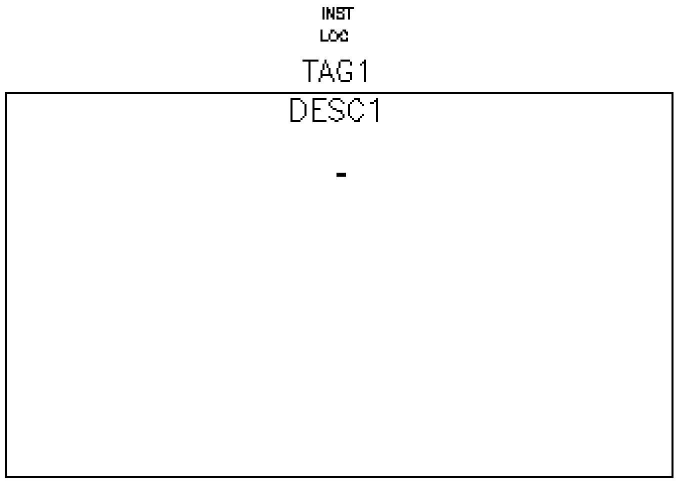


6. Insert the LOC and INST attributes as indicated.
7. Insert the FAMILY attribute near the center of the rectangle.
8. With FAMILY still highlighted in the Symbol Builder Attribute Editor, select the Properties tool. 

   Enter:
   Value: PS

   Click OK.

   This assigns the %F value to the FAMILY attribute inserted.



9. Select MFG and insert near the center of the rectangle. Repeat for CAT and ASSYCODE.

Adding Wire Connections

Insert wire connection attributes and related pin attributes.

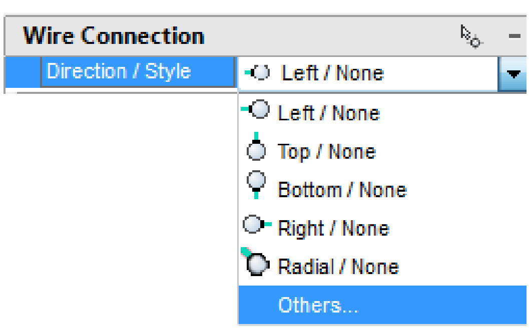
If an X?TERMxx of the component (for example, "X2TERM01") wire connection-point attribute lies within the small trap distance of the end of a wire, then AutoCAD Electrical toolset interprets the component connected to the wire. The only time the trap distance changes is when you change the Feature Scale Multiplier in the Drawing (or Project) Properties ➤ Drawing Format dialog box.

Note: Components with closely spaced wire connection points may not be processed properly if the connection points fall within the AutoCAD Electrical toolset trap distance of one another.

A wire connection attribute can have a related terminal text attribute, TERMxx, and terminal description attribute, TERMDESCxx. The "xx" is a two-digit number (starting at 01) that is used to match up with the corresponding X?TERMxx wire connection attribute.

Insert connection points

1. In the Symbol Builder Attribute Editor, expand the Wire Connection section.
2. In the Direction / Style list, select Others.



3. On the Insert Wire Connection dialog box select Terminal Style: Screw.

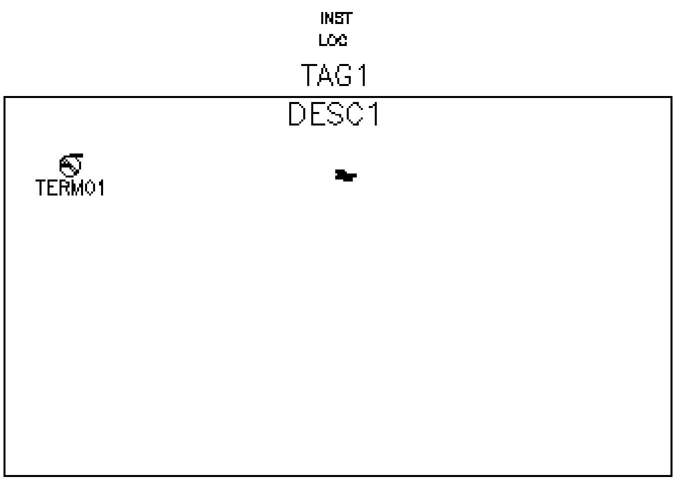
This terminal style inserts both the graphic to represent the screw and the wire connection points.

4. Check Use this configuration as default. It directs Symbol Builder to use the current Terminal Style and Scale as the default in the Symbol Builder Attribute Editor.
5. Select Connection direction: Left & Top.

   It determines the direction the wire attaches to the component.

6. Enter “L” as the value for TERM01 in Pin Information.
7. Select X2TERMDESC01 in Pin Information and click Delete.
8. Click Insert.
9. Select the Insert Wire Connection tool and insert the terminal in the upper left-hand corner as shown.

   Note: Always use AutoCAD Snap to insert the wire connection point.



10. Back on the Symbol Builder Attribute Editor, expand the Wire Connection Direction / Style list and select Right & Top / Screw.
11. Select the Insert Wire Connection tool and insert the terminal in the upper right-hand corner. 

    You can continue to insert wire connections until you press ENTER by entering the characters indicated in the command line prompt followed by a space. You can also select from the Direction / Style list.

12. Insert the rest of the terminals as follows:

    TERM03: Right
    Insertion Point: below TERM02
    TERM04: Bottom
    Insertion Point: in the lower right-hand corner
    TERM05: Bottom
    Insertion Point: to the left of TERM04
    TERM06: Bottom
    Insertion Point: to the left of TERM05
    TERM07: Bottom

Insertion Point: to the left of TERM06

13. Press Enter if necessary to return to the command prompt.
14. On the Symbol Builder Attribute Editor, expand the Pins section. Enter the Pin values as follows:

    TERM02 : **N**

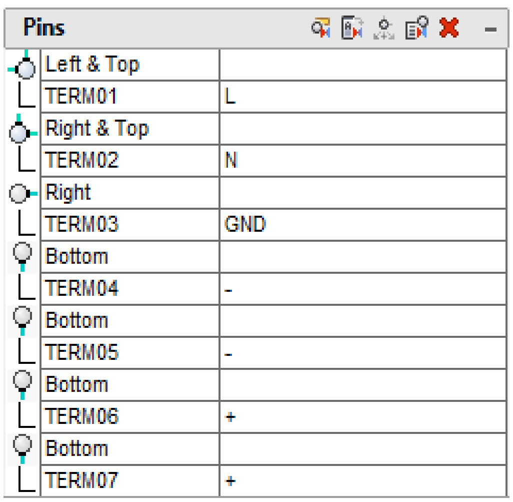
    TERM03 : **GND**

    TERM04 : **-**

    TERM05 : **-**

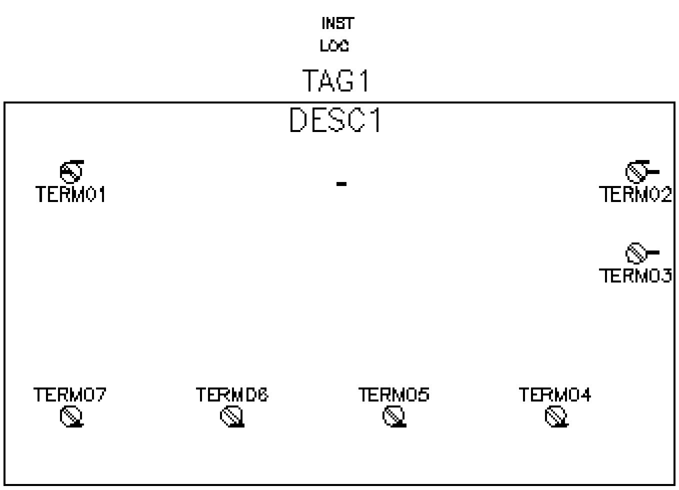
    TERM06 : **+**

    TERM07 : **+**

Pins	
Left & Top	
TERM01	L
Right & Top	
TERM02	N
Right	
TERM03	GND
Bottom	
TERM04	-
Bottom	
TERM05	-
Bottom	
TERM06	+
Bottom	
TERM07	+

    Your drawing looks like the following image:

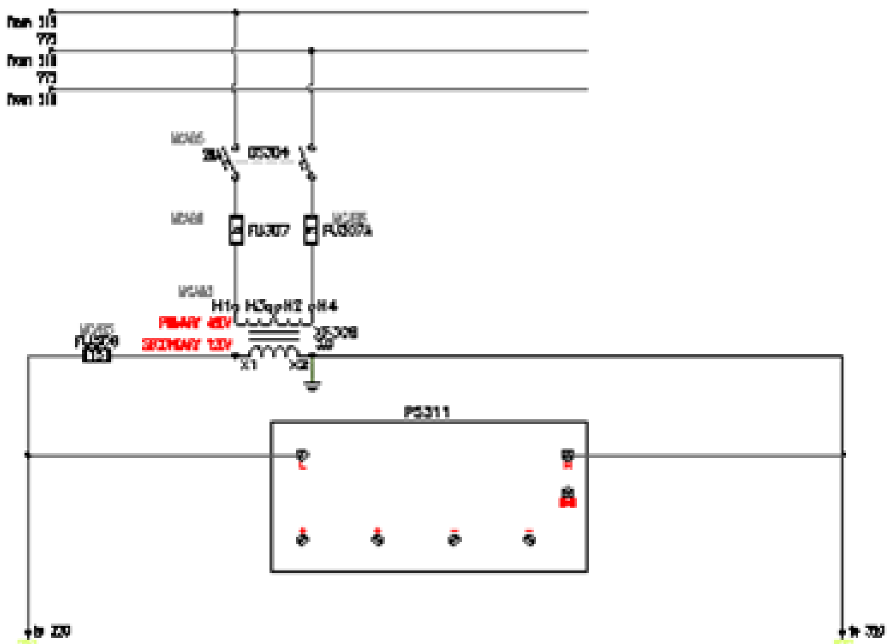
    

Saving the Symbol

Save and insert the symbol onto a drawing.

You have two options for saving the symbol: WBlock or Block. WBlock creates the symbol .*dwg* file while Block creates the symbol for this drawing file only.

1. Click Symbol Builder tab ➤ Edit panel ➤ Done.
2. On the Close Block Editor: Save Symbol dialog box, in the Base point section, click Pick point. Select a point in-line with the top terminals so that it is easy to place on a wire later.
3. Select WBlock.
4. Enter a file name or accept the default.
5. Click OK.
6. When asked to insert the symbol, click Yes.
7. Place the symbol on the empty wire on the left-hand side of the drawing.



The wire breaks, the component tag inserts, and the wires connect to the symbol.

Note: New symbols you create can also be inserted with the AutoCAD Electrical toolset Insert Component command. You can add your new symbol to the icon menu. Or, you can select it from the Type it or Browse dialog box file selection options in the icon menu.

8. In the Insert/Edit Component dialog box, click OK.

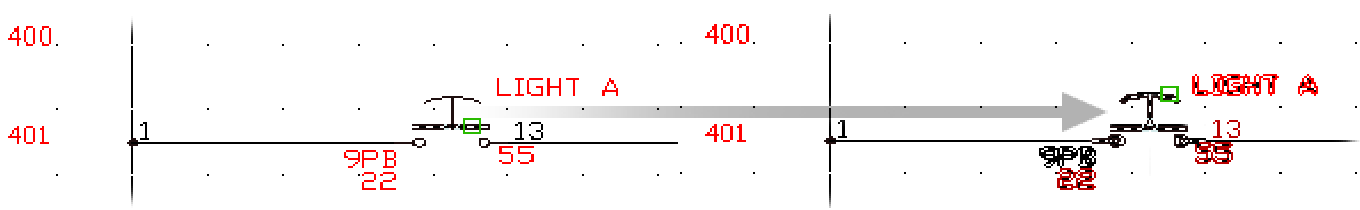
Migration of AutoCAD Data Tutorial

Convert non blocked geometry and text to a fully functional AutoCAD Electrical toolset block insert.

Time required	45 minutes
Prerequisites:	Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Convert to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics below to accomplish these tasks:

- Convert non blocked geometry and text to a schematic component
- Add wire connections
- Add geometry to the block
- Convert non blocked geometry and text to a panel footprint



About Tagging and Linking Tools

Understand the tagging and linking tools in AutoCAD Electrical toolset used to convert non blocked geometry and text to a fully functional AutoCAD Electrical toolset-aware block insert.

AutoCAD Electrical toolset has tagging and linking tools that enable non blocked geometry to be made aware of AutoCAD Electrical toolset. The existing geometry stays in place and is unblocked. Key text entities are converted to attributes with user picks and are linked into a generic, non graphical block insert. Wire connection attributes can also be merged into this generic block insert. The process to convert it from dumb text, circle, and line entities takes only moments to complete and the result appears as a fully functional AutoCAD Electrical toolset-aware block insert.

Exploding Block and Attributes

Use the Special Explode tool in AutoCAD Electrical toolset to explode blocks while maintaining the value previously defined in the attributes.

You can take advantage of the tagging tools to modify the text entities to attributes and the linking tools to make various blocks.

Explode AutoCAD® blocks

1. If AEGS is not the active project, activate the AEGS project.

   If AEGS is in the list of open projects:

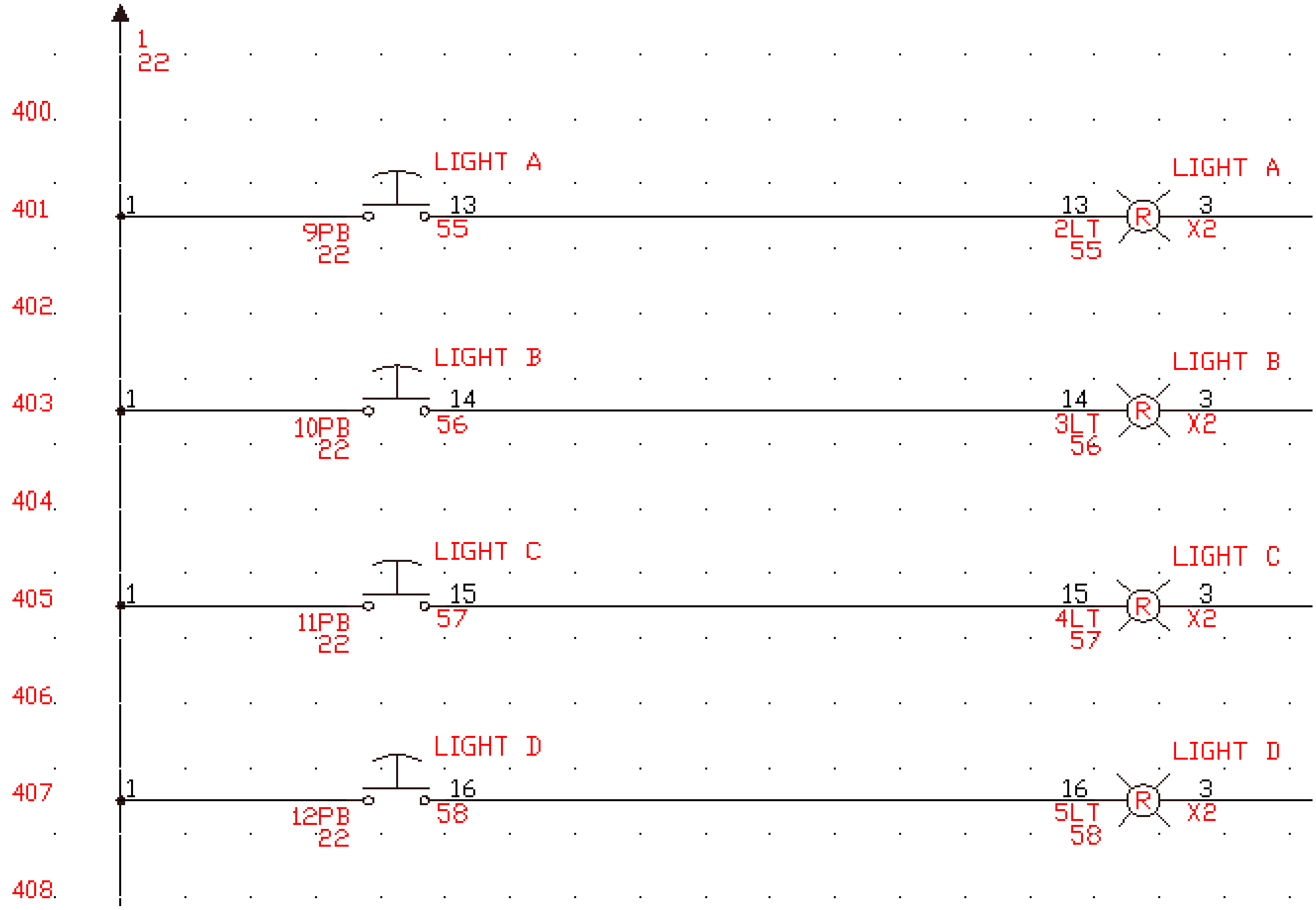
   - Select AEGS and right-click.
   - Click Activate.

   If AEGS is not in the list of open projects:

   - Select the project list drop-down.
   - Click Open Project.
   - On the Select Project File dialog box, navigate to and select the AEGS.WDP file.
   - Click Open.
2. In the Project Manager, double-click AEGS to expand the drawing list.

   There are four drawings in the project, Convert-01.dwg through Convert-04.dwg.

3. Open Convert-03.dwg.
4. Zoom in on the components in the upper left-hand corner of the drawing.

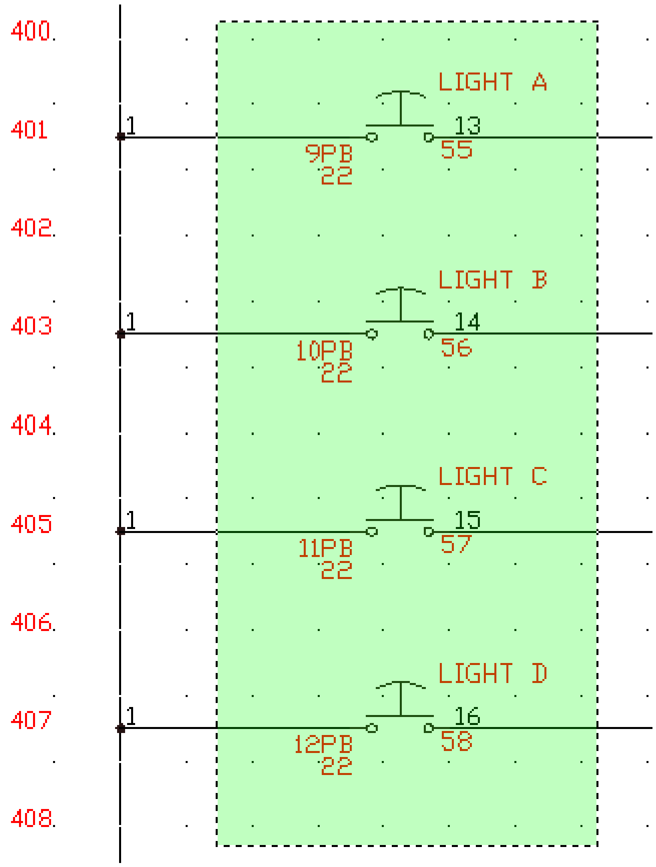


5. Click Conversion Tools tab ➤ Tools panel ➤ Special Explode.

   Use the Special Explode tool to explode attributes and blocks to geometry and text entities while maintaining the value previously defined in the attributes. You can take advantage of the tagging tools to modify the text entities to attributes and the linking tools to make various blocks.

6. Respond to the prompts as follows:

Select objects: *Select push button lights A - D (including all graphics and text) on lines 401 - 407 (use either single picks or window-select), right-click*



The blocks explode into separate text entities and geometry.

Tagging Schematic Components

Use the AutoCAD Electrical toolset Tagging tools to convert text entities into an attributed block.

Through the insertion of a template block, you have control over which attributes are inserted and visible. All necessary attribute definitions are automatically placed. During the tagging process, the text entity is removed and replaced with a template block file that contains multiple attributes used in AutoCAD Electrical toolset.

Tagging Results:

- The selected text entities are replaced with a template block file.
- The TAG attribute takes on the value of the converted text.
- The TAG attribute is set to fixed.
- The TAG attribute takes on the same ACAD properties as the tagged text.

Tag schematic components

1. Click Conversion Tools tab ➤ Schematic panel ➤ Tag Component.
2. Respond to the prompts as follows:

   Select objects: *Select 9PB, 10PB, 11PB, and 12 PB, right-click*

   Note: You may have to right-click several times to exit the command.

The text changes color to indicate that it has been tagged. The color of the TAG attribute is by layer. The attribute is the same layer as defined on the WD_M block. You can now link the descriptions and wire numbers.

3. Click Reports tab ➤ Schematic panel ➤ Reports.
4. In the Schematic Report dialog box, specify:

    Report Name: Component
    Active Drawing

    Click OK.

5. If asked to save the drawing, click Yes.

    In the Report Generator dialog box, notice that 9PB-12PB are listed in the TAGNAME column of the report

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

Linking Schematic Attributes

Use the AutoCAD Electrical toolset Linking tools to associate non blocked text to previously placed template blocks.

Through the modification of a template block, you have control over which attributes are inserted and visible. All necessary attribute definitions are placed using the properties of the existing text entities, such as justification, height, and location. If multiple template block files are selected, the value of the text is added to the previously defined template block attributes as hidden attributes and the text is not removed.

Linking Results:

- The selected text entities are replaced with an AutoCAD Electrical toolset attribute.
- Colors change to distinguish what has been already converted as defined in the WD_M block.
- Temporary lines display the link.

    The Link Descriptions tool links simple text as Description 1-3 attributes on an AutoCAD Electrical toolset block file. You can link them as description attributes to one or more existing template block definitions. During the conversion process, the text entity is removed and replaced with the next available description attribute, up to 3.

Link descriptions

1. Click Conversion Tools tab ➤ Attributes panel ➤ Link Descriptions.
2. Respond to the prompts as follows:

    Select objects: *Select 9PB, right-click*

    Select text to fill in next available DESC attribute: *Select LIGHT A, right-click*

Select objects: *Select 10PB, right-click*

Select text to fill in next available DESC attribute: *Select LIGHT B, right-click*

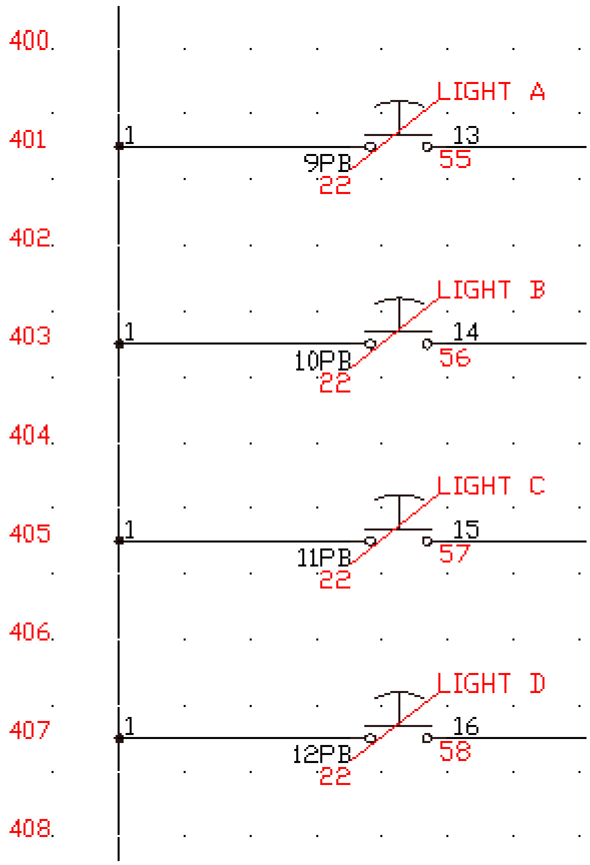
Select objects: *Select 11PB, right-click*

Select text to fill in next available DESC attribute: *Select LIGHT C, right-click*

Select objects: *Select 12PB, right-click*

Select text to fill in next available DESC attribute: *Select LIGHT D, right-click*

Note: You may have to right-click several times to exit the command.



Colors change to distinguish what has been converted and temporary lines display the link.

3. Click Reports tab ➤ Schematic panel ➤ Reports.
4. In the Schematic Report dialog box, specify:

Report Name: Component
Active Drawing

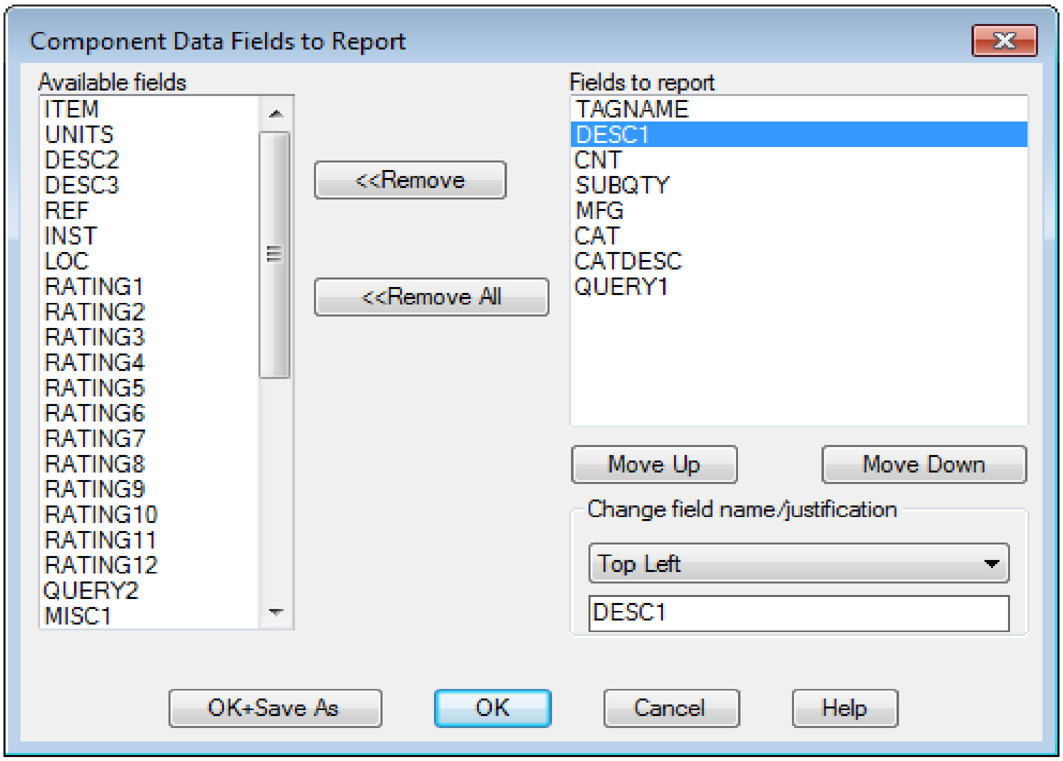
Click OK.

5. If asked to QSave the drawing, click Yes.

In the Report Generator dialog box, notice that 9PB-12PB are still listed in the TAGNAME column of the report.

6. In the Report Generator dialog box, click Change Report Format.
7. In the Component Data Fields to Report dialog box, select Desc1 from the Available Fields list.

   Desc1 moves into the Fields to report list. These are the fields to display in the Component report.



8. Click OK.

   The Report Generator dialog box now lists the TAGNAME and DESC1 values from the active drawing.

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

Adding Wire Connections

Use the Add Wire Connections tool in AutoCAD Electrical toolset to add wire connection attributes to the existing tagged block file.

Select line endpoints or geometry to add the appropriate wire connection attributes to. A new block definition is created with the newly added wire connections. You can later create a block file if the block is exploded.

Wire Connection Results:

- Visual indicators (x) appear where the wire connection attributes have already been applied.
- Wire connection attributes, terminal attributes, and terminal description attributes are added.
- The block definition is automatically modified during the attribute addition process.
- Terminal attribute colors change to distinguish what has been already converted as defined in the WD_M block.

Convert device pins to wire connection attributes

1. Click Conversion Tools tab ➤ Tools panel ➤ Add Wire Connections.
2. Respond to the prompts as follows:

    Select block TAG or PLC Address: *Select 9PB*
    Select end of wire (P=Pick Location): *Enter* P *and press* ENTER
    Select location (W=Wire):

    Press SHIFT + *right-click to select Endpoint from the Snap options, select the end point of the first wire on line 401*

    In the Wire Direction dialog box, select from left.

    Select TERM01 text object: *Select 22 (underneath 9PB TAG)*

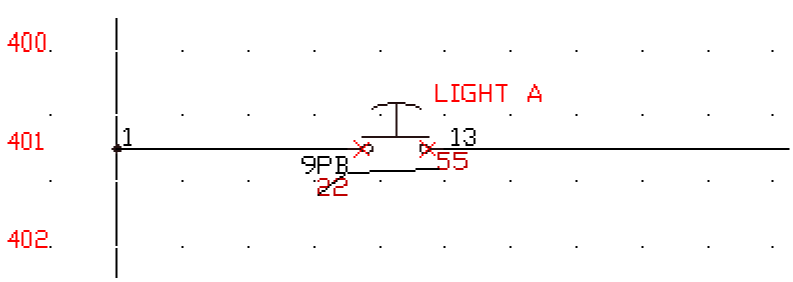
    Note: Visual indicators (x) appear where the wire connection attributes have been applied.

    Select location (W=Wire):

    Press SHIFT + *right-click to select Endpoint from the Snap options, select the end point of the second wire on line 401*

    In the Wire Direction dialog box, select from right.

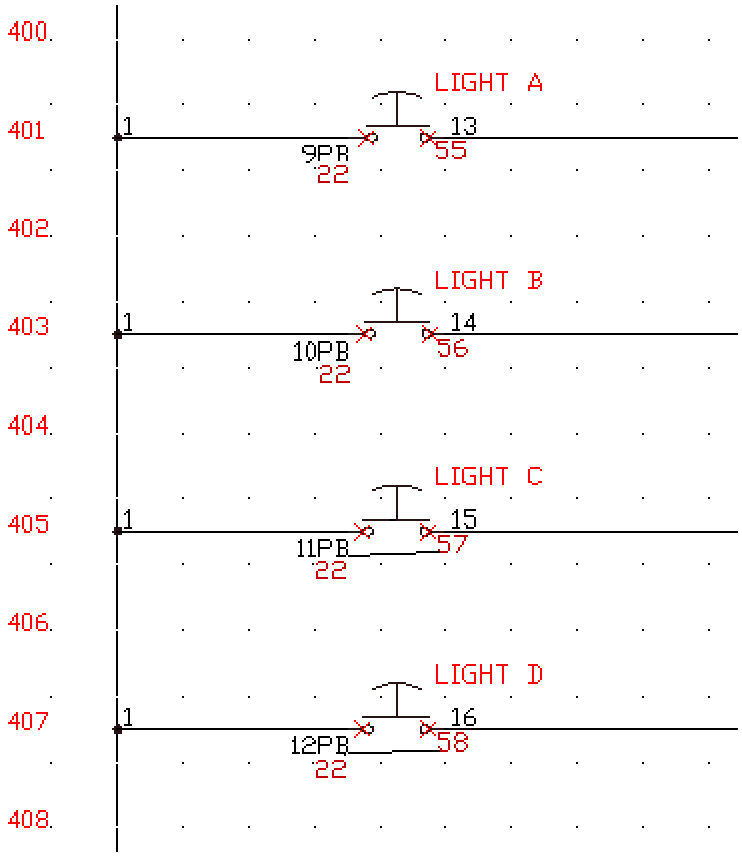
    Select TERM02 text object: *Select 55 (underneath line 401), right-click*



You are back at the prompt to Select block TAG or PLC Address.

3. Repeat for 10PB - 12PB.

Note: You may have to right-click several times to exit the command.



Pause the mouse over 9PB - 12 PB. The text, wire connection attributes, and description text all highlight. We still must convert the wire number text and add the geometry to our block.

4. Click Schematic tab ➤ Edit Wires/Wire Numbers panel ➤ Create/Edit Wire Type.
5. In the Create/Edit Wire Type dialog box, select Make all Lines Valid Wires and click OK.

Note: If the OK button is disabled, click one of the wire types to enable it.

6. Click Conversion Tools tab ➤ Tools panel ➤ Convert Text to Wire Number.
7. Respond to the prompts as follows:

Select LINE near wire number text: *Select the left endpoint of the wire with the text 13 above it (line 401)*

Select existing wire number text to convert: *Select text 13*

8. While you are still in the command, repeat for text 14 - 16 on lines 403 - 407.
9. Right-click to exit the command.

Adding Geometry

Use the Add Geometry tool in AutoCAD Electrical toolset to add the graphics to the block definition containing the previously added attributes.

Add Geometry Results:

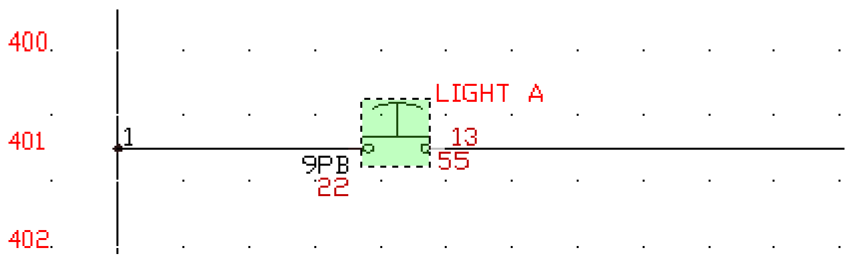
- TAG1, TAG2, PLC TAG, and TAGSTRIP attributes are defined and selected first.
- The block definition is automatically modified.
- The color of the geometry changes by layer to distinguish what has been already converted as defined in the WD_M block.

Add geometry to the block

1. Click Conversion Tools tab ➤ Tools panel ➤ Add Geometry.
2. Respond to the prompts as follows:

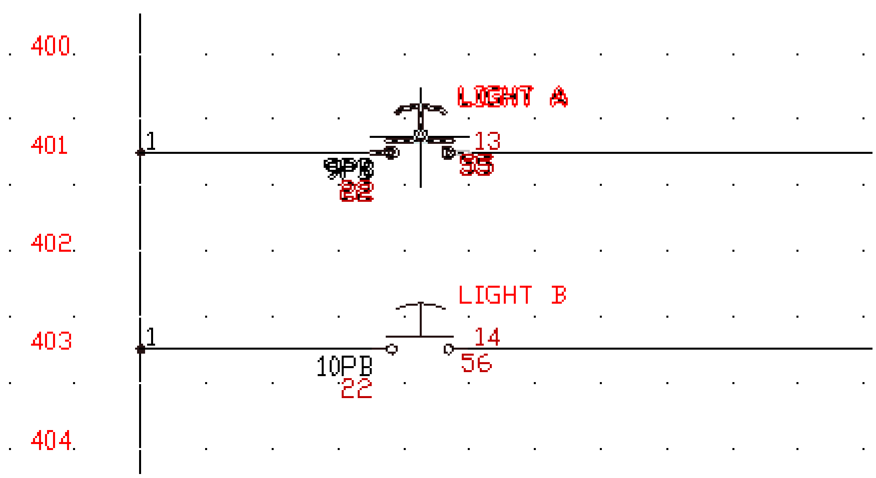
   Select block for additional geometry: *Select 9PB*

   Select objects: *Select the graphics for the push button, right-click*



   Specify insertion point: *Select the middle of the push button*

   The geometry is associated to the template block files. Check that everything has been tied to the block by mousing-over 9PB. The text, wire connection attributes, description text, and geometry highlights.



3. Repeat steps 1 -2 for 10PB, 11PB, and 12 PB.

Your blocks are now AutoCAD Electrical toolset-smart.

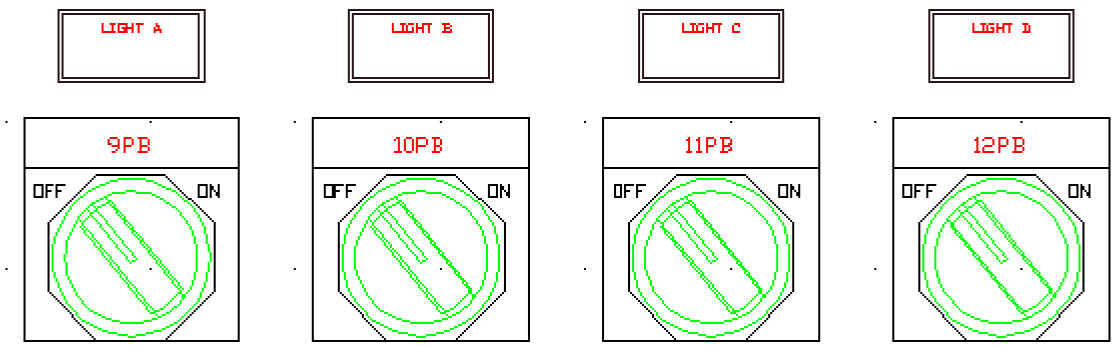
Tagging and Linking Panel Components

Use the AutoCAD Electrical toolset Tagging and Linking tools to associate non blocked text to panel components.

The AutoCAD Electrical toolset Tagging and Linking tools work on panel components the same way they work on schematic components.

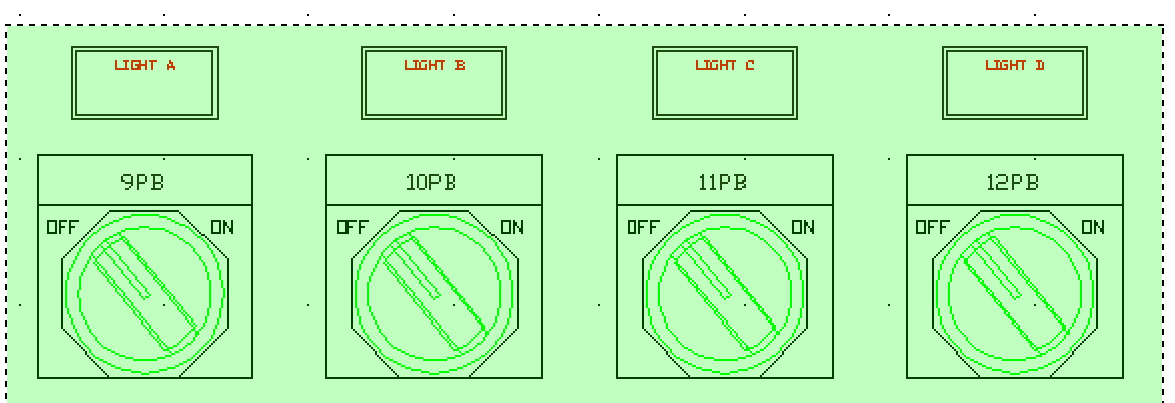
Tag and link panel components

1. Open Convert-04.dwg.
2. Zoom in on the components in the middle of the drawing.



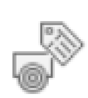
3. Click Conversion Tools tab ➤ Tools panel ➤ Special Explode. 
4. Respond to the prompts as follows:

   Select objects: *Select push button lights A - D (including all graphics and text) (use either single picks or window-select), right-click*



   The blocks explode into separate text entities and geometry.

   The Tag Panel Component tool makes selected text entities an attributed block file with the P_TAG1 attribute visible. The template block file (ACE_P_TAG1_CONVERT.DWG) contains attributes for a panel component.

5. Click Conversion Tools tab ➤ Panel panel ➤ Tag Footprint. 
6. Respond to the prompts as follows:

Select objects: *Select 9PB, 10PB, 11PB, and 12 PB, right-click*

Note: You may have to right-click several times to exit the command.

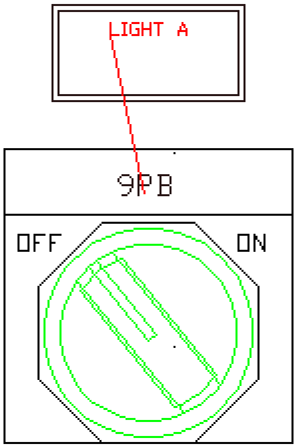
The text changes color to indicate that it has been tagged. The color of the PTAG attribute is by layer. The attribute is the same layer as defined on the WD_M block.

7. Click Conversion Tools tab ➤ Attributes panel ➤ Link Descriptions.
8. Respond to the prompts as follows:

    Select objects: *Select 9PB, right-click*

    Select text to fill in next available DESC attribute: *Select LIGHT A, right-click*

Note: You may have to right-click several times to exit the command.



Updating Panel or Schematic Components

Edit a schematic component and update the related panel footprint.

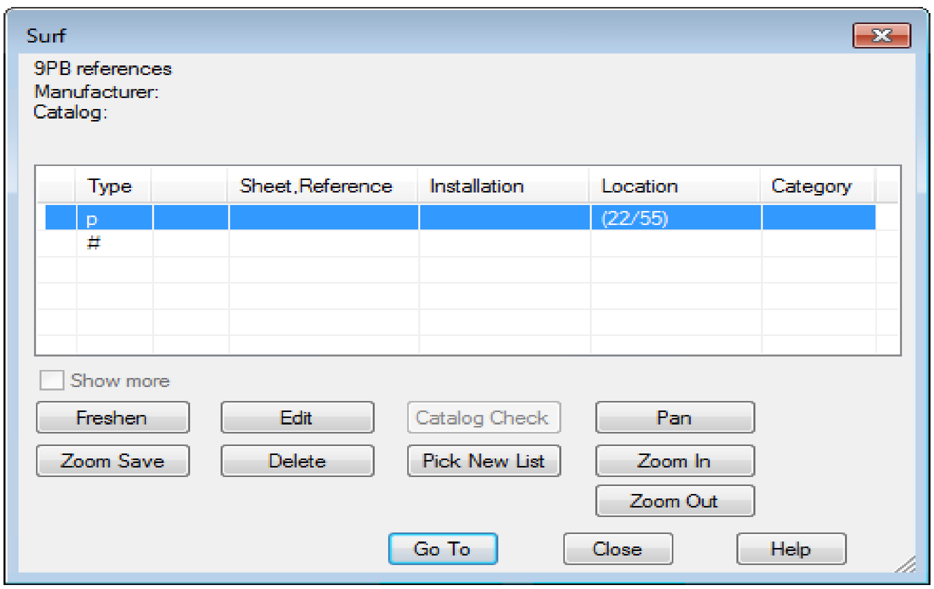
Once a panel component has a component tag assigned, it is automatically linked to the schematic component with the same tag. Updates to either the schematic or panel component prompt an update to the related component.

Surf to the related schematic component

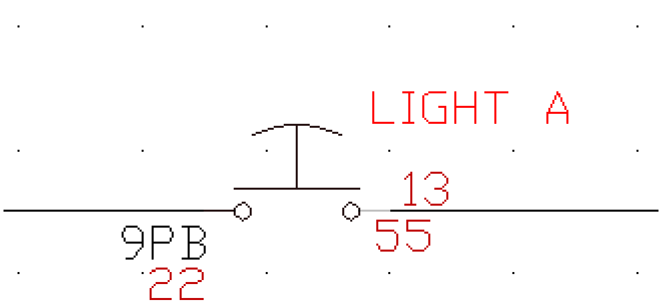
1. Click Project tab ➤ Other Tools panel ➤ Surfer.
2. Respond to the prompts as follows:

   Select tag for “Surfer” trace (or <Enter> to type it): *Select 9PB*

3. In the Surf dialog box, double-click the component marked with type “p.”

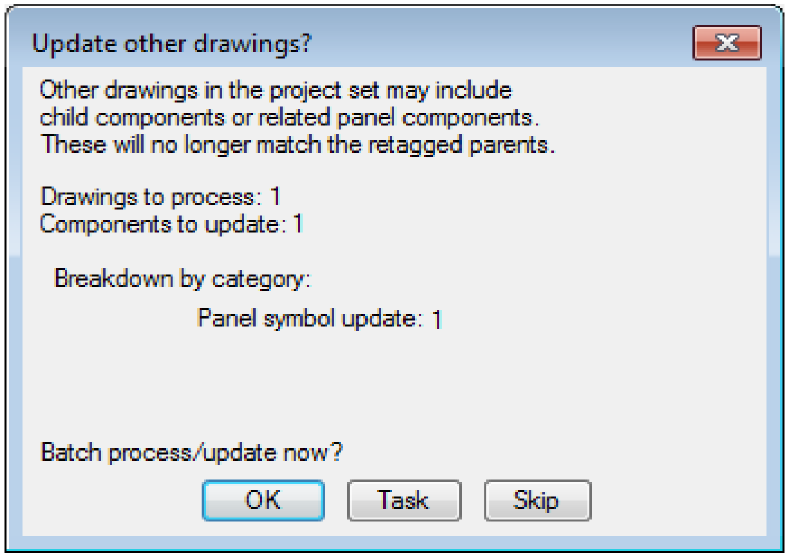


Surfer goes to the schematic drawing and zooms on the schematic component.



4. If asked to save the drawing, click Yes.
5. In the Surf dialog box, click Edit.
6. In the Component Insert/Edit dialog box, change the description to **LIGHT 1** and click OK.

The Update Other Drawings dialog box displays. This dialog alerts that other drawings in the project set include child components or related panel components.

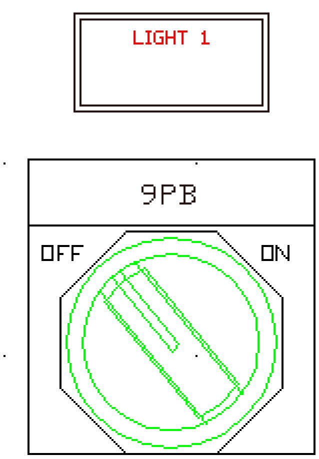


7. If asked to save the drawing, click Yes.
8. In the Update Other Drawings dialog box, click OK.
9. Click Project tab ➤ Other Tools panel ➤ Surfer.
10. Respond to the prompts as follows:

Select tag for “Surfer” trace (or <Enter> to type it): *Select 9PB*

11. In the Surf dialog box, double-click the component marked with type “#.”

Surfer goes to the panel layout drawing and zooms on the physical representation of the push button. Notice that the description for 9PB updated to reflect the change you made to the schematic component.



12. In the Surf dialog box, click Close.

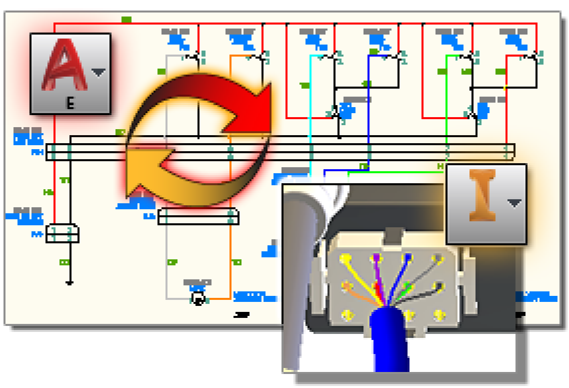
Interoperability: Inventor and AutoCAD Electrical Toolset Cable and Harness Tutorial

Learn to use AutoCAD Electrical toolset and Inventor cable and harness interoperability to digitally prototype and document your electrical designs.

Time required	35 minutes
Prerequisites:	• Familiarity with both AutoCAD Electrical toolset and Inventor is recommended but not necessary. This tutorial is designed to work whether you have only AutoCAD Electrical toolset or Inventor, or if you have both programs. • Know how to navigate the model space with the various view tools.
Tutorial file used	900501.dwg Copy all files located in
	Users\{username}\Documents\Acade {version}\Aedata\Tutorial\Aegs\Interoperability to Users\{username}\Documents\Acade {version}\Aedata\Proj\Aegs

Follow the workflow topics listed below to:

- Interchange cable and harness data between AutoCAD Electrical toolset and Inventor



Cable and Harness Tutorial Introduction (continued)

Understand the tutorial objectives and requirements.

In this tutorial, you learn how to interchange cable and harness data between AutoCAD Electrical toolset and Inventor. In the first half of the tutorial, the exchange direction is from AutoCAD Electrical toolset to Inventor.

In the second half, the exchange direction is from Inventor to AutoCAD Electrical toolset.

You do not need both programs to derive benefit from this exercise. The two XML files generated in this workflow are also included in the tutorial sample files. Therefore, if you have only one program, you can still perform the XML import operation.

If you have only AutoCAD Electrical toolset, you can review the Inventor portion of the tutorial. Then you can perform the import steps and subsequent steps on the Import the Inventor data page. You are directed to perform this import at the appropriate point.

Note: You need the Inventor Professional or Routed Systems versions for the Cable and Harness functionality.

Part 1: 2D to 3D

Learn how to export your data from AutoCAD Electrical toolset to Inventor.

The sample DWG file is a wiring diagram used for a seat assembly. The assembly uses electric motors to provide adjustments to the seat position.

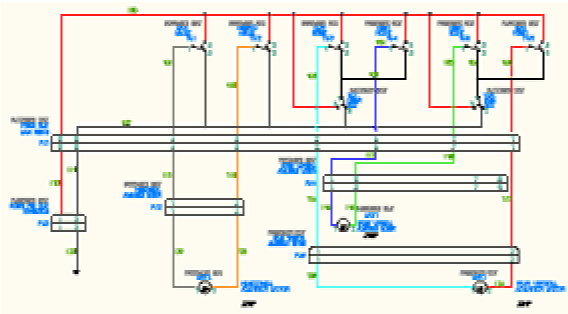
Open DWG

1. Start AutoCAD Electrical toolset.
2. Open the Project Manager. If this window is closed:

   Click Project tab ➤ Project Tools panel ➤ Manager.

3. Select **Open Project** from the project drop-down menu.
4. Select the project **ace_inv.wdp** and click **Open**. The project is in the location described on the introduction page.

5. Expand the *ACE_INV* project, and then double-click **900501.dwg**.



Ensure that the drawing is in Model Space.

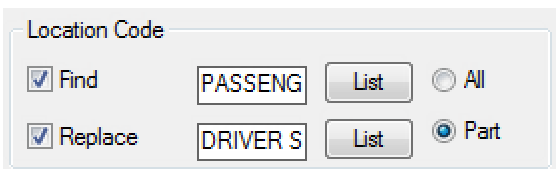
Rename Component Tags

Update the PASSENGER SEAT component tags to prepare to export the data to Inventor.

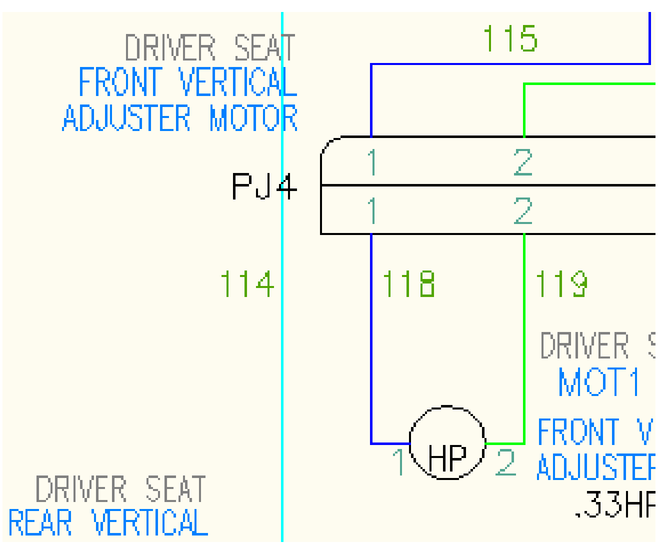
Assume that this drawing was copied forward and is now ready to modify and use for the driver seat.

1. Click Schematic tab ➤ Edit Components panel ➤ Retag Components drop-down ➤ Find/Edit/Replace Component Text.
2. Select **Active drawing (all)** in the *Find/Edit/Replace Electrical Component Text* dialog box, and click **OK**.
3. Set the following options and take the following actions in the *Find/Edit/Replace - this Drawing (all)* dialog box.

    - Select **Find** in the *Location Code* group.
    - Click **List**, then select **PASSENGER SEAT** in the *Loc values* dialog box, and click **OK**.
    - Select **Replace**, and then enter **DRIVER SEAT** in the text box.

    - Click **Start Search**, and review the results in the *Match 1 of 16* dialog box.
    - Click **Replace All**, and then click **Yes, Make Changes**. The component tags are changed to DRIVER SEAT.

    - Click **Cancel**.

4. Save the drawing.

Export to XML

Export the electrical data contained in your AutoCAD Electrical toolset digital prototype to an XML file.

You use this XML file later to import the data into Inventor.

1. Click Import/Export Data tab ➤ Export panel ➤ Inventor.
2. Ensure **Active Drawing** is selected in the *Autodesk Inventor Professional Export* dialog box, and click **OK**.
3. Save to the same directory you copied the tutorial files in the *Autodesk Inventor Professional XML File Export* dialog box. Use **driverseat_from_ace** for the file name.
4. Click **Save**.

Set the Project

Open the Inventor tutorial project.

1. Start Inventor.
2. Select **Get Started** ➤ **Launch** ➤ **Projects**.
3. Click **Browse**.
4. Browse to *Tutorial Files/Automotive* folder, and select **interop.ipj**.
5. Click **Open**.
6. Click **Done** in the *Projects* editor.

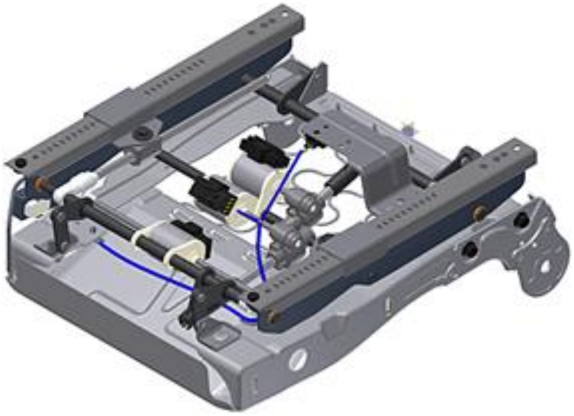
Open the Dataset

Open the model to prepare for importing the electrical data.

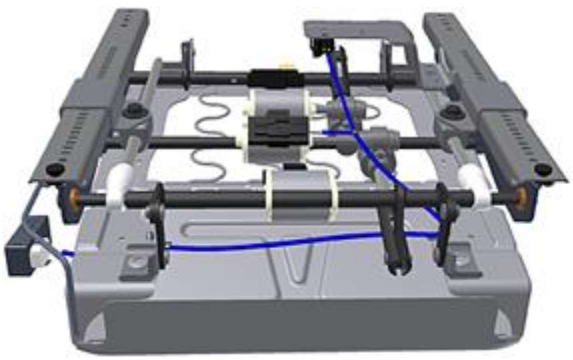
1. Open **100500.iam**. The file is contained in the 1000 folder. The model opens in the Default design view representation.

This sample has been stripped down to reduce data size. The complete seat looks like the following:

2. Switch to the **Electrical** design view representation.



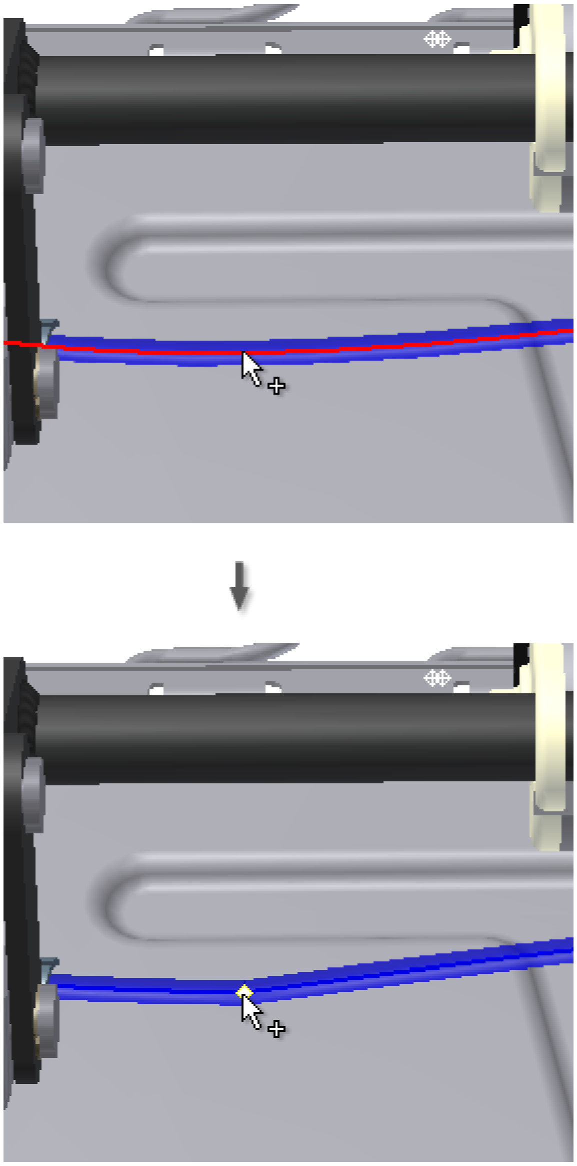
Orbit and zoom your view as you progress through the workflow, as needed. It can be helpful to approximate the following view as you get started with the workflow.



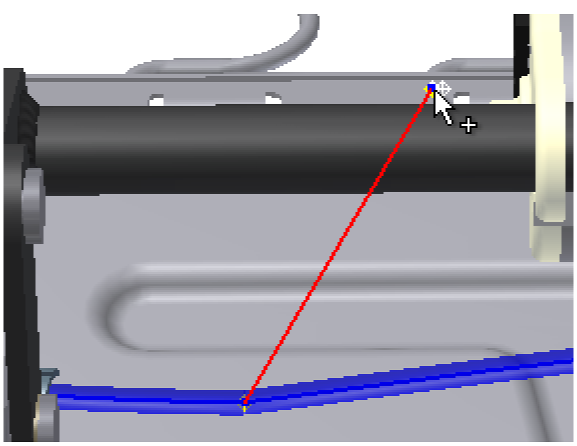
Add Harness Segments

Add the two harness segments to prepare for import.

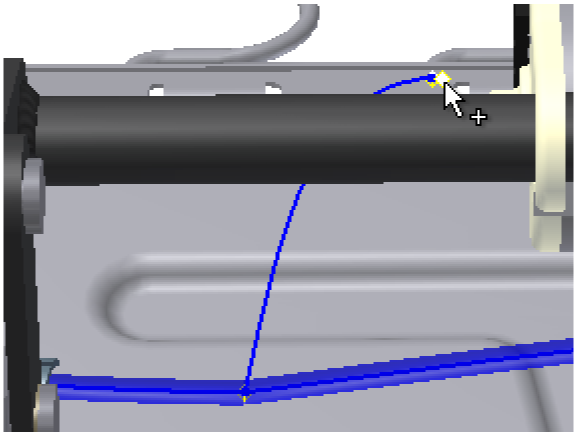
1. Double-click **Harness1** in the browser to edit the harness. Be careful to edit the harness assembly rather than the harness part.
2. Select **Cable and Harness** ➤ **Create** ➤ **Create Segment**.
3. Select the existing segment near the front of the seat to place the first point. The exact selection location is not critical.



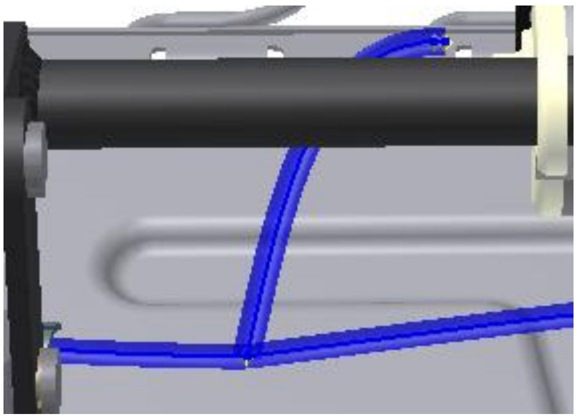
4. Select the existing work point to set the next segment point. The two work points are very close together. You may need to zoom in to see them clearly.



5. Select the other work point to set the final point.



6. Right-click, and select **Continue**. The first segment is created.

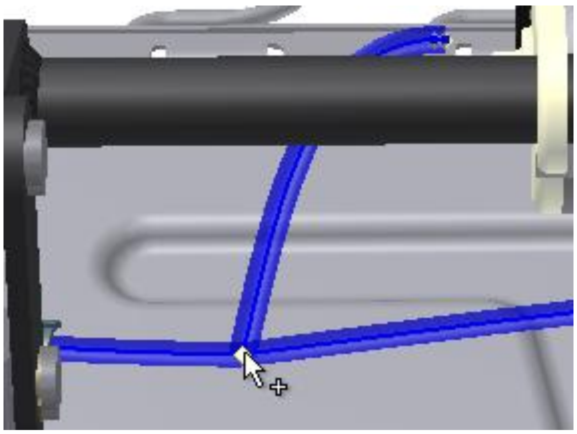


Add Harness Segments (continued)

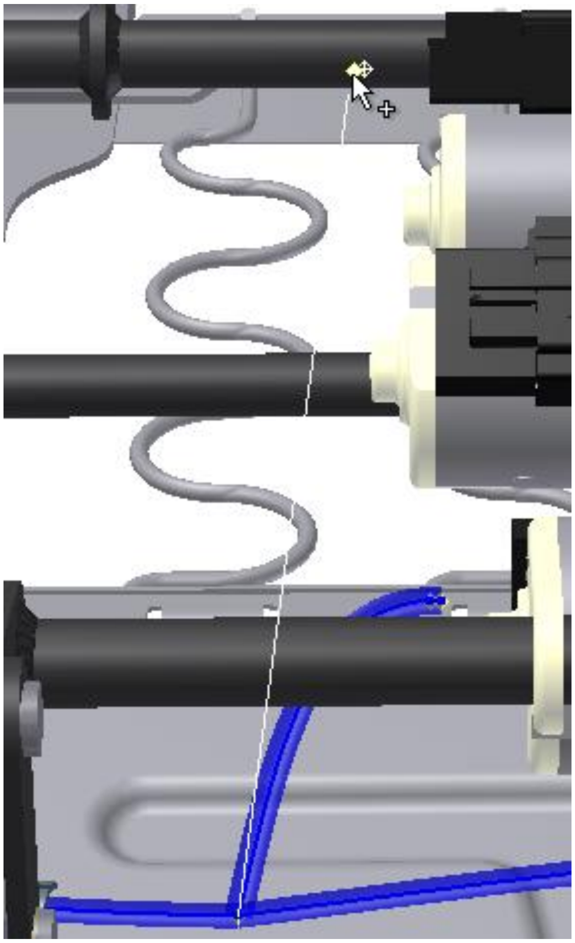
Finish adding the two harness segments.

Next, you add a second segment. This segment begins in the same location as the previous segment. Because you selected Continue from the context menu, Create Segment is still active and ready to create another segment.

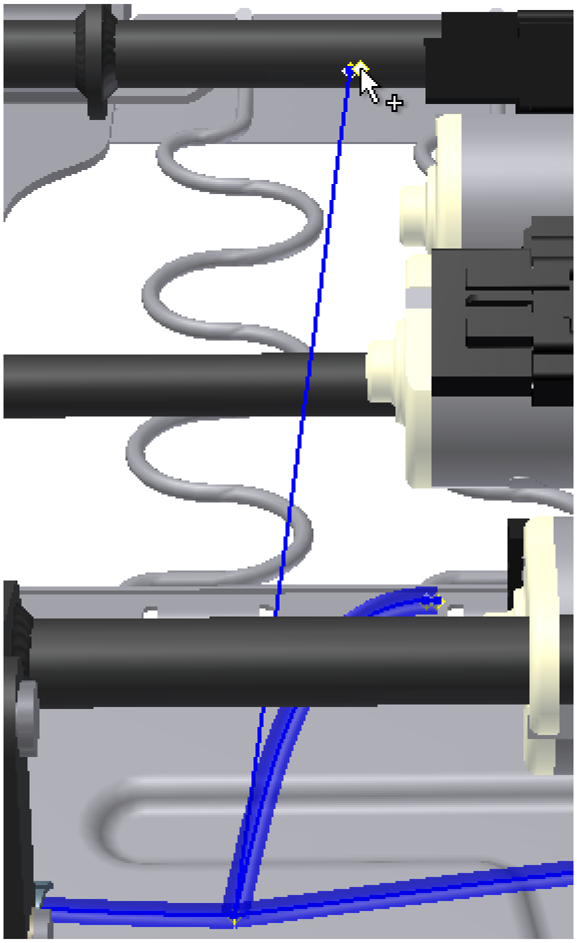
1. Select the segment point that you added previously.



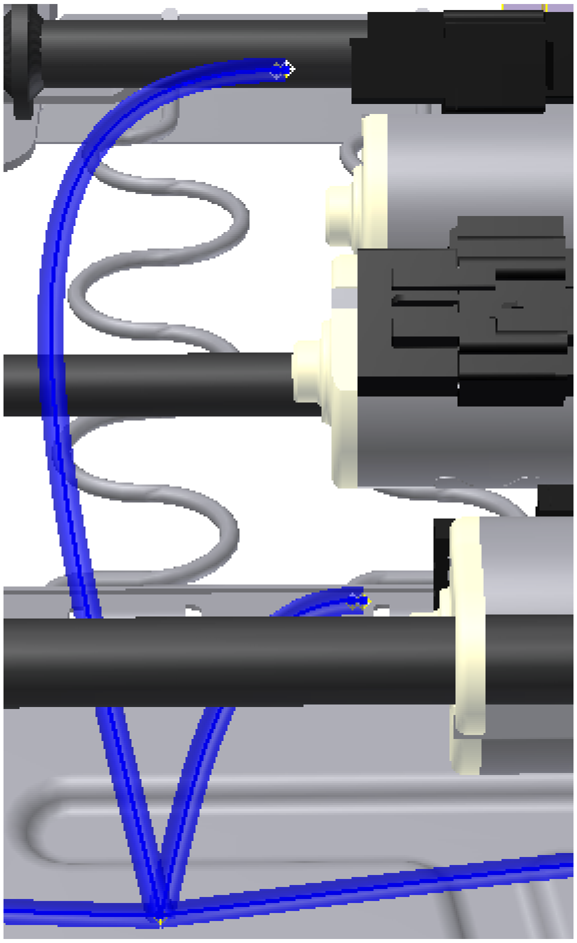
2. Select the existing work point.



3. Select the other work point.



4. Right-click, and select **Finish**. The segment is created.



Import the AutoCAD Electrical Toolset Data

Apply the AutoCAD Electrical toolset data to the Inventor 3D model.

1. Select **Cable and Harness** ➤ **Manage** ➤ **Import Harness Data**. 
2. Click the Browse button next to the Harness Data File field. Select **driverseat_from_ace.xml** you exported from AutoCAD Electrical toolset in the *Select Wire List Data File* dialog box. Click **Open**.

   Note: If you do not have AutoCAD Electrical toolset, you can now use **driverseat_from_ace.xml** provided in *Tutorial Files\Automotive\XML_delivered*.

3. Click **OK** in the *Import Harness Data* dialog box.

Issues

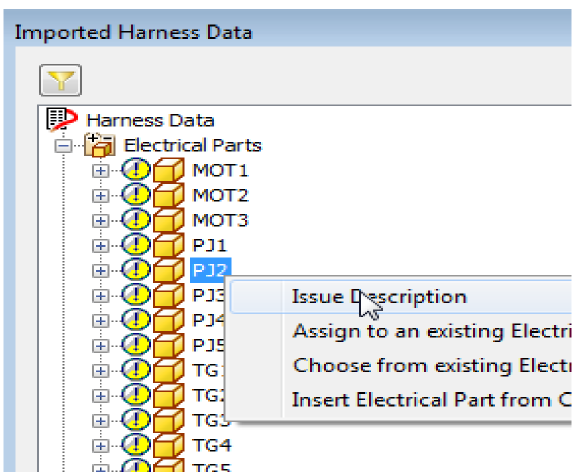
Review the issues related to missing or incomplete data.

The browser nodes in the *Imported Harness Data* dialog box contain the electrical components and wires imported from AutoCAD Electrical toolset.

1. Click Filter in the *Imported Harness Data* dialog box, and then select **Show Issues Only**.

Only items with issues display in the dialog box. The items are identified by the Issue icon. There are many issues, because the Inventor sample assembly does not contain many of the components contained in the AutoCAD Electrical toolset drawing as reflected in *driverseat_from_ace.xml*. Many of the RefDes in AutoCAD Electrical toolset do not have a matching RefDes in Inventor. The absence of various components and RefDes also means that connecting wires also have issues. The missing RefDes do not prevent you from successfully completing the exercise. This scenario is a reflection of a real-world design process in which data may be missing or incomplete but is acceptable for a given point in the workflow.

Scroll to the top of the item list, then right-click **PJ2**, and select **Issue Description**.



The issue description describes the problem and offers solutions. Review the information, and then close the issue description.

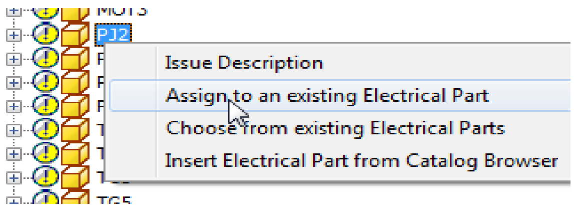
Tip: Click the Help button in the *Imported Harness Data* dialog box to open a reference topic that describes various elements and features in the dialog box.

Next, you use functionality on the same context menu to assign the missing RefDes.

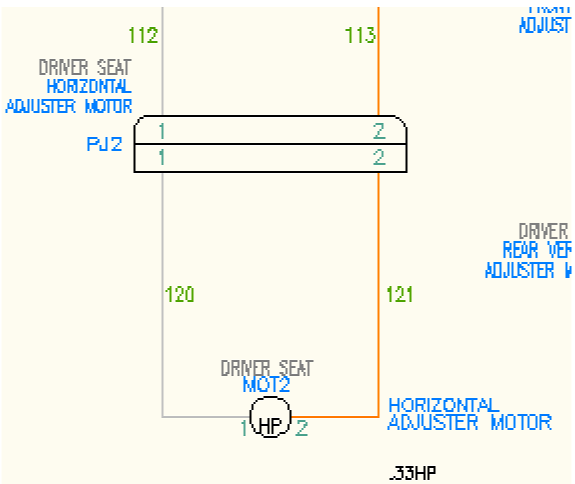
Assign missing RefDes

Add the RefDes value from AutoCAD Electrical toolset.

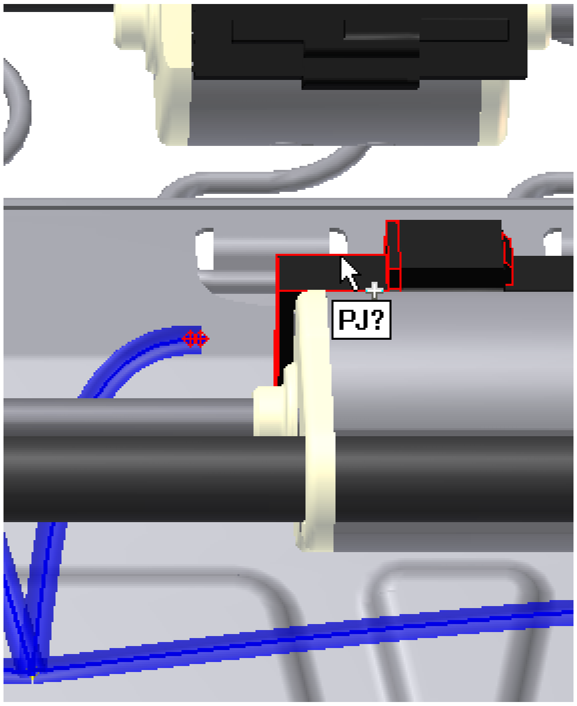
1. Right-click **PJ2** and select **Assign to an existing Electrical Part**.



With reference to the AutoCAD Electrical toolset drawing, *PJ2* is the RefDes specified for the connector that connects to the motor MOT2, the motor for horizontal adjustment.



2. Pause the cursor over the connector occurrence **900575:2** in the graphics window, and note the tooltip.



You can select the occurrence in the browser; however, when you use the graphics window, Inventor displays a tooltip. The tooltip shows the RefDes for that component. The question mark (?) indicates that the RefDes is not yet assigned.

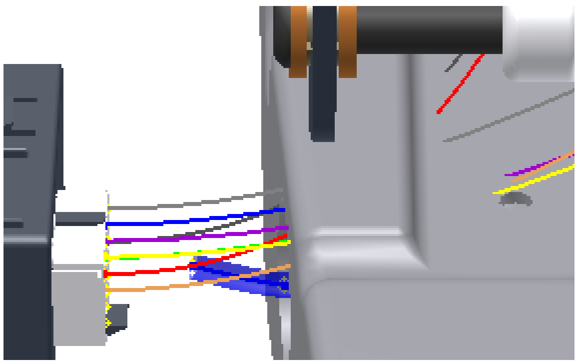
3. Select the connector.
4. Click **OK** in the *Select Electrical Part* dialog box.
5. The RefDes specified in AutoCAD Electrical toolset is assigned to the Inventor connector, and the issue associated with *PJ2* is removed. Because the dialog filter is set to show only items with issues, *PJ2* is not included in the list.

Finish the Import

Complete the import of the AutoCAD Electrical toolset data.

1. Click **OK** in the *Imported Harness Data* dialog box.
2. Close the message dialog box. For this exercise, accept the remaining issues without making further changes.

   The data from AutoCAD Electrical toolset is imported. You should see eight imported wires in the graphics window and in the browser.

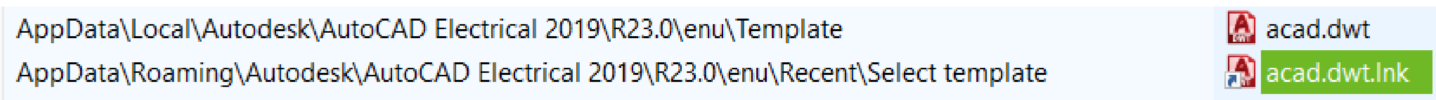
Key Notes

Details of commonly used commands.

SNAP: enter at command prompt (F9) to display current value and change. Can also be turned on / off. 0.125 is often a good value.

Define title block variables: ATTDEF @ command prompt.

Enter title block variables: Project|Other Tools|Title Block Setup.



C:\Users\GuyWms\AppData\Local\Autodesk\AutoCAD Electrical 2019\R23.0\enu\Template

Title Block Setup: **Project | Other Tools | Title Block Setup**