

IM225070

# Using AutoCAD® Electrical for More Than Just Electrical Schematics

Randy Brunette

Brunette Technologies, LLC

## Learning Objectives

- Access various symbol libraries, including hydraulic, pneumatic, and P&ID
- Create your own drawing type
- Link different types of drawing components
- Create reports based on drawing type

## Description

AutoCAD Electrical is more than just an electrical schematic design tool. It also includes symbol libraries and functionality for panel layouts, one-line, hydraulic, pneumatic, and P&ID drawings. Within AutoCAD Electrical all types of drawings and the included components are linked and update each other automatically. Attend this class to learn the simple requirements for creating the links between drawings and how you can leverage this capability to improve your efficiency and quality of your design package.

## Speaker(s)

Randy Brunette is the sole proprietor of Brunette Technologies, LLC, a consulting firm specializing in AutoCAD Electrical and related products including Inventor, Vault, and the Substation Design Solution (SDS) Toolkit. As an Electrical Subject Matter Expert, Randy's duties include helping channel partners and customers through mentoring and understanding their business issues and finding solutions that solve their challenges. Randy has been in the design field using Autodesk products for over 33 years, with experience across many different segments of the manufacturing industry. He has been in an application engineer role for 22 years, traveling throughout the world providing consulting services. Randy is a top-rated speaker at Autodesk Universities, Technical Academies, and seminars. He has authored AutoCAD Electrical software training manuals, videos, and other materials.

[www.brutec.com](http://www.brutec.com)



[randy.brunette@brutec.com](mailto:randy.brunette@brutec.com)

## Peer Component Libraries

Although they have been included for many, many releases now, many people don't realize that AutoCAD Electrical also includes symbol libraries for three of most commonly used non-electrical schematic diagrams, Pneumatic, Hydraulic, and P&ID (Piping and Instrumentation Diagram) style symbols. You can also create custom symbols or create your own library depending on your design requirements.

In AutoCAD Electrical these other types of drawings and the relationship between them and the electrical schematic are referred to as peer-to-peer linking. The "Peer" term was chosen because the symbols on these drawings many times have matching components on the electrical schematic drawings. These matching components are all on the same level or hierarchy and are NOT a parent/child type relationship. The relationship between Schematic and Panel layout components is a good example. These related peer components are just different representations of the same component.

Peer drawings contain non-schematic, non-panel components that frequently need to be linked to schematic components. Typical components displayed on different types of drawings include solenoids, motors, and meters.

## One-line Symbols

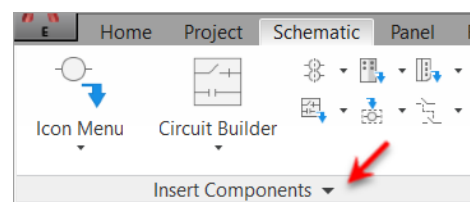
One of the peer symbol libraries is support for one-line diagrams. A one-line diagram is basically a simplified version of a three line or three phase diagram. In a three phase diagram most components are generally inserted three times, one on each phase. Instead of cluttering the drawing with duplicated components a one-line diagram gives a simpler, cleaner version of the circuit. One-line diagrams are useful for conveying the overall load strategy for the design system.

One-line diagram support and linking to the three phase counterparts is the same as the linking of schematic symbols to other peer components. You referencing an existing circuit component, to transfer the TAG or WDTAGALT attribute value that provides the peer link.

## Library Access

The one-line symbols are located on the main page of the icon menu.

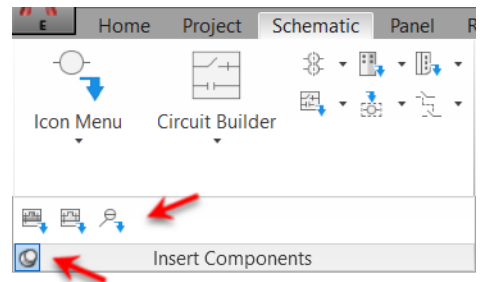
The other peer library icons are located on the Schematic tab > Insert Components expansion panel.



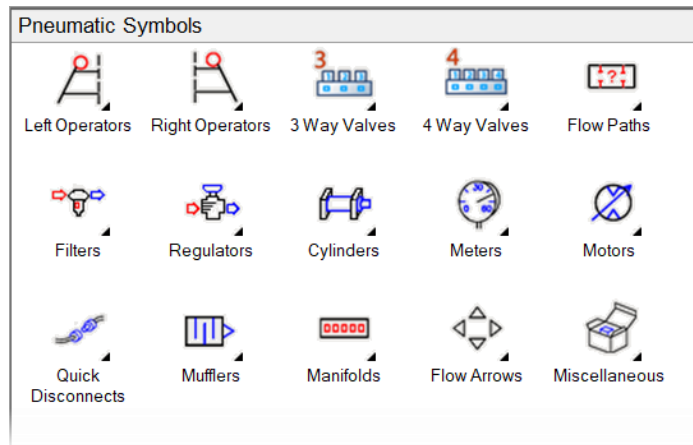


**Hint:** Click the pin to keep the panel expanded for frequent use.

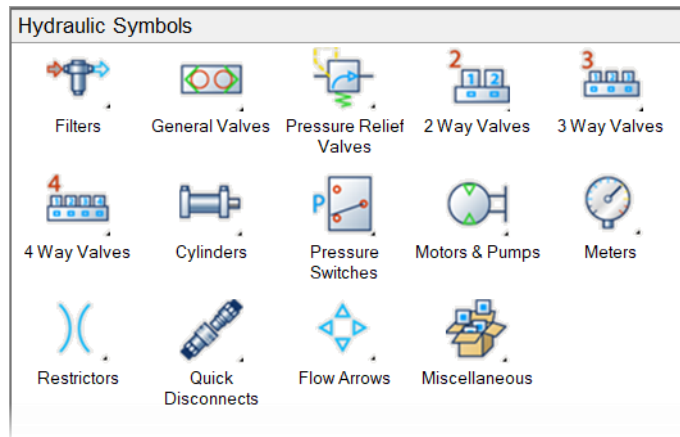
Clicking on one of the library icons displays the matching page in the Insert Component Icon Menu.



### Insert Pneumatic Components

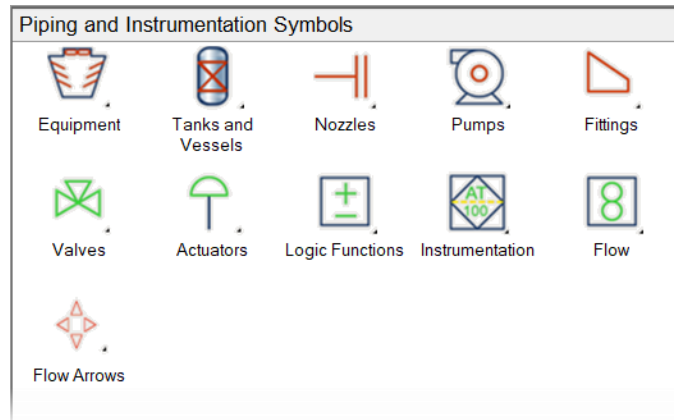


### Insert Hydraulic Components





## Insert P&ID Components



## Automatic Functionality

Most functionality from AutoCAD Electrical translates directly to these styles of drawings. You just fire up your imagination and think in terms of the matching entity or function. For example, a wire carries the electrical current from one location to another; a pipe carries the air or fluid from one location to another.

To insert a pipe into your pneumatic or hydraulic drawing, use the AutoCAD Electrical Insert Wire command. This will automatically connect to the different components, auto-route and so on. Another example of the functionality is that when inserting components from these menus, they will break the underlying line (pipe), just like inserting a standard electrical component.

The same schematic reporting functionality used in electrical schematics is used to create reports for piping runs, connection lists, BOM reports and so on. Just select the category of drawings you want to extract the data from in the Schematic Reports dialog box.

You can also link information such as tag names between the symbols in the electrical drawings and the symbols in peer schematic drawings, even if they have different tag names or styles. Once the components are linked, changes to one component will automatically update the other. Surfing between all related or linked components is also available.

## WDType Attribute

A peer type component block definition must include the WDTYPE attribute. This is the first of two attributes needed to make the peer-to-peer functionality work properly.

The function of the WDTYPE attribute is to identify the category of a component. As long as the WDTYPE is unique, two parent components can share the same TAG values without causing reporting, quantity, and other types of errors. When TAG values are linked, the same edits done to one component are automatically transferred to the other.

The WDTYPE attribute is included in most of the supplied peer library symbols, and the display of the attribute is normally set to invisible. The value of the attribute specifies a component category for a non-schematic, non-panel component. In most cases, schematic symbols do not carry this attribute, or if they have the attribute, it is with a blank value.

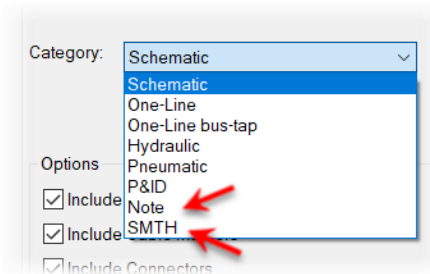
AutoCAD Electrical currently uses the following WDTYPE attribute values.

Attribute	Symbol category
<b>Blank</b>	Standard electrical schematic components
<b>1-</b>	One-line diagram components
<b>1-1</b>	One-line bus-tap components
<b>PN</b>	Pneumatic diagram components
<b>HY</b>	Hydraulic diagram components
<b>PI</b>	P&ID diagram components

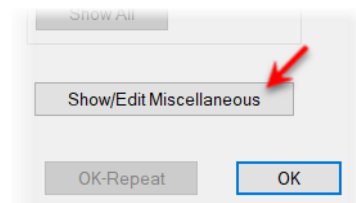
Two-character values for the WDTYPE attribute are reserved by AutoCAD Electrical. They can be used but may be over-written by a future AutoCAD Electrical enhancement. It is recommended to use User-defined values that are three or a maximum of four characters long.

When using the Symbol Builder, the WDTYPE attribute is available for insertion into symbols by referencing the Generic template and Optional attributes.

For many schematic reports, you can select a component category for the report based on the WDTYPE value. Any new component types (WDTYPES) used in the project are automatically available in the Schematic Reports dialog box.



The WDTYPE attribute can be edited in the Insert/Edit Component dialog box by clicking Show/Edit Miscellaneous.



## WDTAGALT Attribute

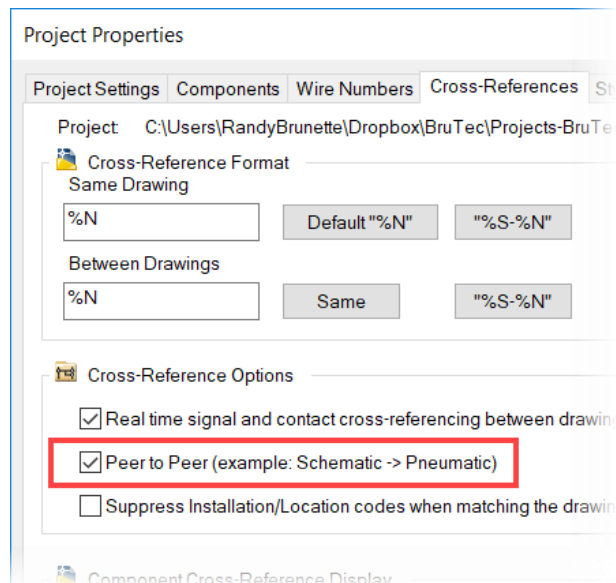
When working with peer drawings, the tagging convention used on the components may be different than the tagging convention used on the related schematic components

The WDTAGALT attribute is carried on a parent symbol and is used for linking non-schematic, non-panel components with their schematic or panel equivalents. The WDTAGALT attribute is required when the TAG1 values on the two parent symbols are different, but the components needed to be linked. The attribute value can be up to a maximum of 64 characters. The attribute stores the cross-reference tag name of a related symbol shown on a different drawing type (for example, an instrument or pneumatic drawing versus an electrical schematic).

For cross-referencing to include these peer references, you must make sure that the Peer-to-Peer toggle is checked. This is a project setting option and is located in the Project Properties dialog box, Cross-reference tab, under Cross-Reference Options.

The WDTAGALT attribute is available for insertion into symbols when using the Symbol Builder. You use the Generic template and Optional attributes.

The WDTAGALT attribute can be edited in the Insert/Edit Component dialog box by clicking Show/Edit Miscellaneous.



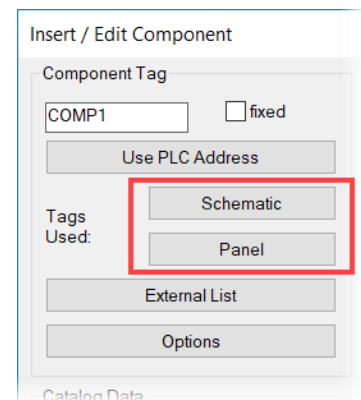
## Linking Peer Components

Components are linked when the TAG values match each other. This can be done manually (yuck!) or by using the Edit Component command to edit the TAG and WDTAGALT attributes. TAG attributes include TAG1 (schematic parents), TAG2 (schematic children), and P\_TAG1 (panel).

Selecting the Schematic or Panel buttons displays lists of previously used TAG values. Selecting one of the values from the list links the peer components together.

If the WDTAGALT attribute exists on the peer component, you are prompted to select either TAG1 or WDTAGALT linking.

- Use TAG1 is used when the TAG values of the two components will be the same value.
- Use WDTAGALT is used when the TAG values of the two components will be different values.

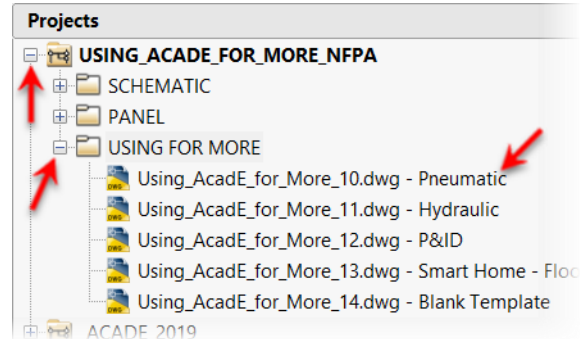


# Exercises

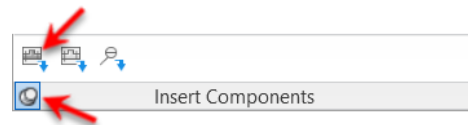
## \* Insert Peer Components

In this exercise, you open pneumatic, hydraulic, and P&ID style peer drawings and insert components from the matching symbol libraries.

1. In AutoCAD Electrical, open the Using\_AcadE\_for\_More\_NFPA project.
2. Expand the project.
3. Expand the Using For More sub-folder.
4. Right-click *Using\_AcadE\_for\_More\_10.dwg*. Click Open.



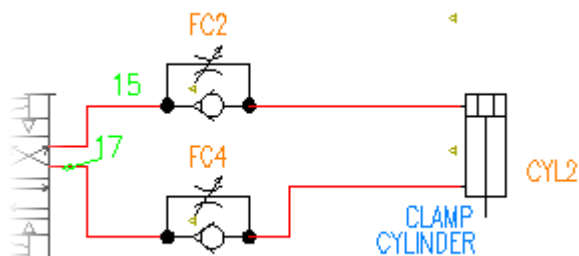
5. On the Schematic tab, Insert Components panel, Insert Components expansion panel, click Pin.
6. Click Insert Pneumatic Components.

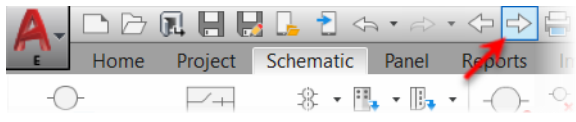
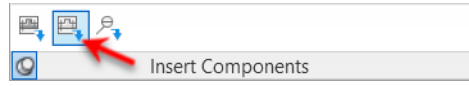
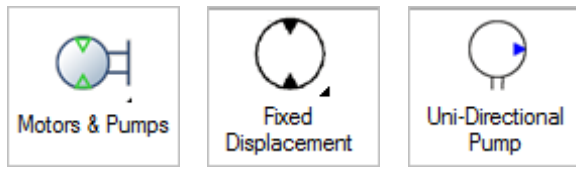
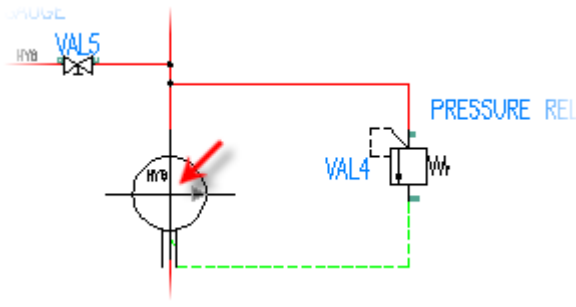
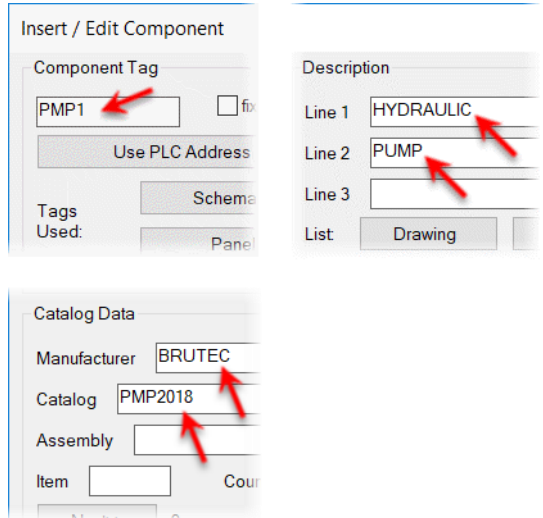


7. In the Insert Component menu, click Cylinders > Cushion Cylinder.

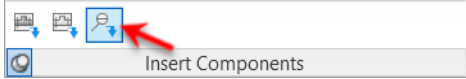
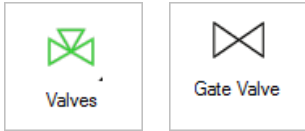
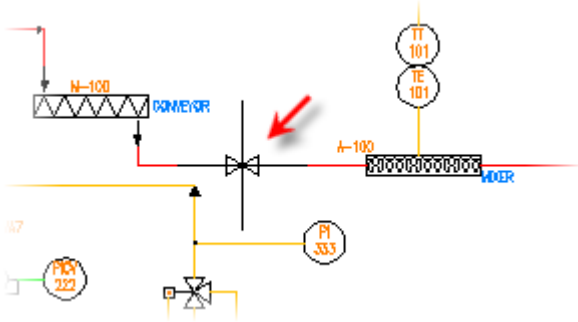


- In the drawing canvas, specify the insertion point at the end of the pipes to the right of FC2.
- In the Insert/Edit Component dialog box, do the following:
- Under Description, for Line 1, enter **CLAMP**.
- For Line 2, enter **CYLINDER**.
- Under Catalog Data, for Manufacturer, enter **BruTec**.
- For Catalog, enter **CYL2018**.
- Click OK.



<p>8. On the Quick Access toolbar, click Next Project Drawing.</p>	
<p>9. On the Schematic tab, Insert Components expansion panel, click Insert Hydraulic Component.</p>	
<p>10. In the Insert Component menu, click Motors &amp; Pumps &gt; Fixed Displacement &gt; Uni-Directional Pump.</p>	
<p>11. In the drawing canvas, specify the insertion point on the vertical line, just to the left of VAL4, just above the green bypass line.</p>	
<p>12. In the Insert/Edit Component dialog box, do the following:</p> <ul style="list-style-type: none"> <li>• Under Description, for Line 1, enter <b>HYDRAULIC</b>.</li> <li>• For Line 2, enter <b>PUMP</b>.</li> <li>• Under Catalog Data, for Manufacturer, enter <b>BruTec</b>.</li> <li>• For Catalog, enter <b>PMP2018</b>.</li> <li>• Click OK.</li> </ul>	

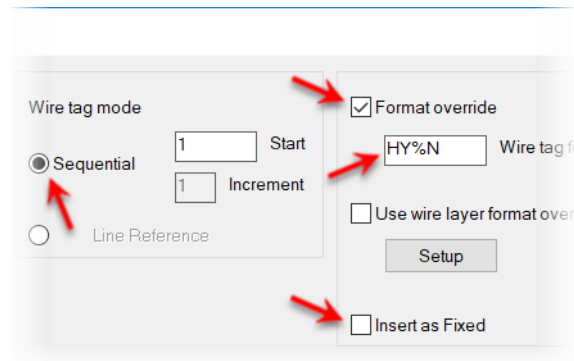


<p>13. On the Quick Access toolbar, click Next Project Drawing.</p> <p>14. On the Schematic tab, Insert Components expansion panel, click Insert P&amp;ID Component.</p>	
<p>15. In the Insert Component menu, click Valves &gt; Gate Valve.</p>	
<p>16. In the drawing canvas, specify the insertion point on the horizontal line between the M-100 Conveyor and A-100 Mixer.</p> <p>17. In the Insert/Edit Component dialog box, do the following:</p> <ul style="list-style-type: none"> <li>• Under Description, for Line 1, enter <b>MIXER</b>.</li> <li>• For Line 2, enter <b>VALVE</b>.</li> <li>• Under Catalog Data, for Manufacturer, enter <b>BruTec</b>.</li> <li>• For Catalog, enter <b>GVL2018</b>.</li> <li>• Click OK.</li> </ul> <p><i>This completes the exercise.</i></p>	

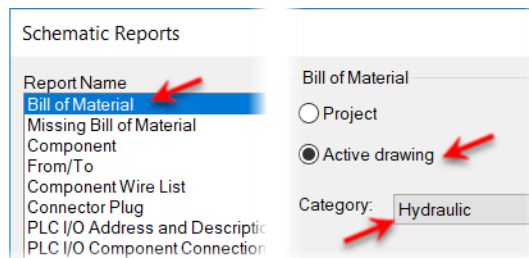
### \* Peer Reports

In this exercise, you open the Hydraulic drawing, add pipe (wire) numbers, and create a Bill of Material report to display a list of components in the diagram, inserting the list on the drawing as a table. Then you create a From/To report to display the piping connections.

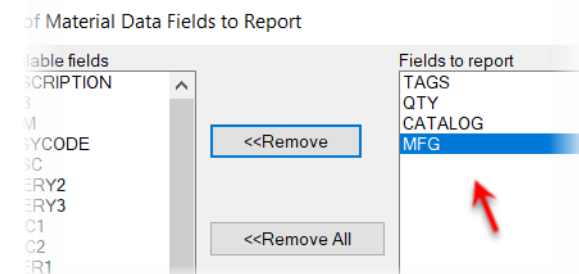
1. In Project Manager, open *Using\_AcadE\_for\_More\_11.dwg*.
2. On the Schematic tab, Insert Wires/Wire Numbers panel, click Wire Numbers.
3. In the Wire Tagging dialog box, for Wire Tag Mode, select Sequential.
4. Select Format Override.
5. For Wire Tag Format, enter **HY%N**.
6. De-select Insert as Fixed.
7. Click Drawing-wide.



8. On the Reports tab, Schematic panel, click Reports.
9. In the Schematic Reports dialog box, do the following:
  - Under Report Name, select Bill of Materials.
  - Under Component, select Active Drawing.
  - For Category, select Hydraulic.
  - Click OK.
10. If prompted, in the Qsave dialog box, click Always Qsave.



11. In the Report Generator, click Change Report Format.
12. In the BOM Data Fields to Report dialog box, under Fields to Report, remove all fields except TAGS, QTY, CATALOG, and MFG.
13. Click OK.



14. A Bill of Material of components in the drawing is displayed.

*This list can now be placed on the drawing or exported to any of several file types.*

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

Report Generator

Drawing Bill Of Material for all locations (9 records)

TAGS	QTY	CATALOG	MFG
FC3	1	NG-32	PARKER
FC1	1	NG-25	PARKER
MTR1	1	PM9874	PARKER
MOT212	1	PMP2018A	BRUTEC
CHKV2	1	CHKV1974	PARKER
CYL1	1	C35010	PARKER
FI2	2	FIL2018	BRUTEC
FI3			
VAL4	1	ULV5874	PARKER
VAL1	3	VAL2018	BRUTEC
VAL2			
VAL5			

16. In the Table Generation Setup dialog box, click OK.

17. Select and insertion point in the lower left corner of the drawing canvas.

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

TAGS	QTY	CATALOG	MFG
FC3	1	NG-32	PARKER
FC1	1	NG-25	PARKER
MTR1	1	PM9874	PARKER
PMP1	1	PMP2018	BRUTEC
CHKV2	1	CHKV1974	PARKER
CYL1	1	C35010	PARKER
FI2	2	FIL2018	BRUTEC
FI3			
VAL4	1	VLV5874	PARKER
VAL1	3	VAL2018	BRUTEC
VAL2			
VAL5			



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

20. In the Schematic Reports dialog box, do the following:

- Under Report Name, select From/To.
- Under From/To, select Active Drawing (All).
- For Category, select Hydraulic.
- Click OK.


21. If prompted, in the Qsave dialog box, click Always Qsave.

Schematic Reports


- Report Name
- Bill of Material
- Missing Bill of Material
- Component 
- From/To 
- Component Wire List
- Connector Plug
- PLC I/O Address and Description
- PLC I/O Component Connection
- PLC Modules Used So Far
- Terminal Numbers
- Terminal Plan
- Connector Summary
- Connector Detail

From/To

Project

Active drawing (all) 

Active drawing (pick)

Category:  

22. In the Location Code Selection dialog box, do the following:

- For From Location Codes, click ALL>>.
- For To Location Codes, click <<ALL.
- Click OK.

*Review the list of pipe (wire) connections.*

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

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

*This completes the exercise.*

Report Generator

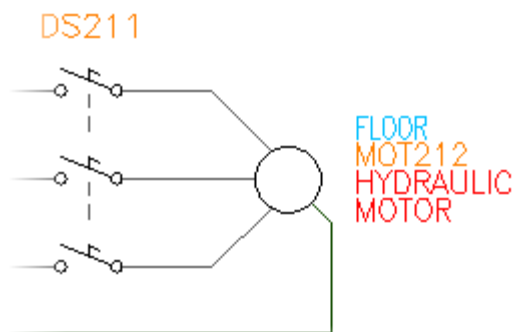
Drawing From/To extract (17 records)

WLAV1	WIRENO	LOC1	CMP1	PIN1	LOC2	CMP2	PI
1.00	NPT	HY7	CHKU2	IN		UAL3	01
1.00	NPT	HY6	CHRU3	OUT		UAL5	L1
1.00	NPT	HY3	CYL1	LINE		CHKU4	I1
1.00	NPT	HY2	FC1	OUT		FC3	I1
1.00	NPT	HY1	FC1	IN		F13	01
1.00	NPT	HY3	FC3	OUT		CYL1	L1
1.00	NPT	HY2	FC3	IN		CHKU4	01
1.00	NPT	HY2	FC3	IN		UAL2	L1
1.00	NPT	HY5	F13	IN		CHKU2	01
1.00	NPT	HY9	MOT212			UAL1	L1
1.00	NPT	HY8	HTR1	P2		UAL5	L1
1.00	NPT	HY10	UAL1	L2		F12	I1
1.00	NPT	HY4	UAL2	L2		CHKU3	I1
1.00	NPT	HY9	UAL3	IN		MOT212	
0.75	NPT	HY9	UAL4	OUT		MOT212	
1.00	NPT	HY6	UAL4	IN		MOT212	
1.00	NPT	HY6	UAL5	L1		UAL4	I1

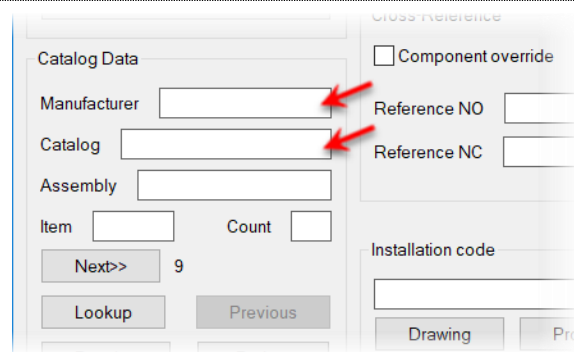
### \* Link Schematic and Hydraulic Components Using TAG1

In this exercise, you link Schematic and Hydraulic component symbols together using the TAG1 attribute. Then you edit the Hydraulic symbol, and view the changes on the Schematic symbol.

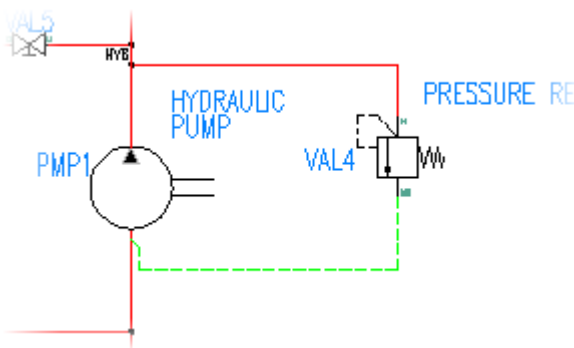
1. In the Project Manager, expand the Schematic sub-folder.
2. Right-click *Using\_AcadE\_for\_More\_02.dwg*. Click Open.
3. Zoom in to the motor MOT212 located on rung 212.
4. Right-click MOT212. Click Edit Component.



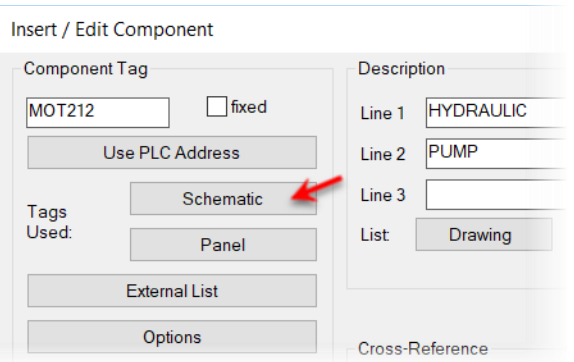
5. In the Insert/Edit Component dialog box, notice the blank Manufacturer and Catalog values.
6. Click Cancel.



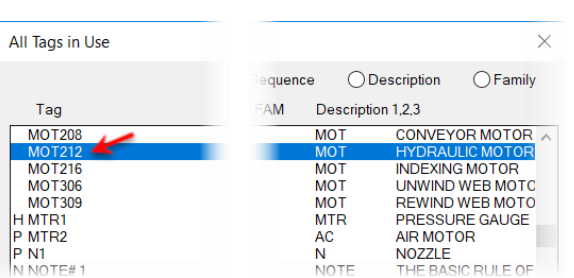
7. In the Project Manager, right-click *Using\_AcadE\_for\_More\_11.dwg*. Click Open.
  8. Zoom in to the pump, PMP1, that you inserted earlier.
- You will set this as a peer of the schematic motor MOT212.*



9. In the drawing canvas, right-click PMP1. Click Edit Component.
10. In the Insert/Edit Component dialog box, under Component Tag, for Tags Used, click Schematic.



11. In the PMP Tags in Use dialog box, do the following:
  - Select Show All Components for all Families.
  - Scroll through the list and select MOT212, HYDRAULIC MOTOR.
  - Click Copy Tag.



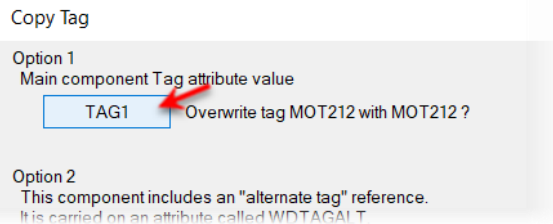
12. In the Copy Tag dialog box, click TAG1.

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

*Notice the TAG name of the Hydraulic Pump is updated to MOT212.*

14. If prompted, in the Footprint Symbol Missing dialog box, select Update the Catalog...

**Note:** *At this time, only the TAG value was updated. No other information is updated or transferred unless it is changed in the matching component.*



15. In the drawing canvas, right-click MOT212. Click Edit Component.

16. In the Insert/Edit Component dialog box, under Catalog Data, for Catalog, enter **PMP2018A**.

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

18. If prompted, in the Footprint Symbol Missing dialog box, select Update the Catalog...

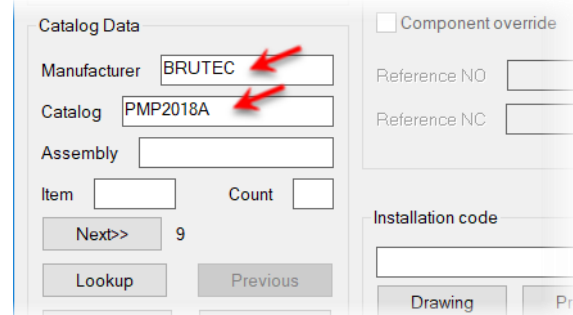
19. In the Update Other Drawings? dialog box, click OK.

20. If Prompted, in the Qsave dialog box, click Always Qsave.

*Notice that two other drawings are automatically updated, the schematic symbol and the panel layout symbol.*

21. In the Project Manager, right-click *Using\_AcadE\_for\_More\_02.dwg*. Click Open.
22. In the drawing canvas, right-click MOT212. Click Edit Component.  
*In the Insert/Edit Component dialog box, notice the updated Manufacturer and Catalog values.*
23. Click Cancel.

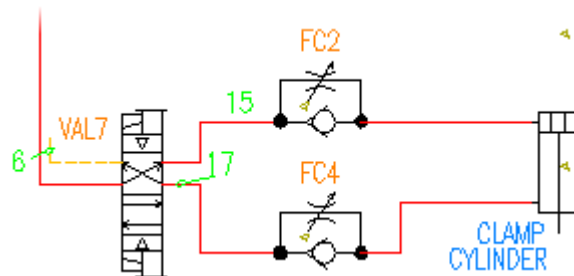
*This completes the exercise.*



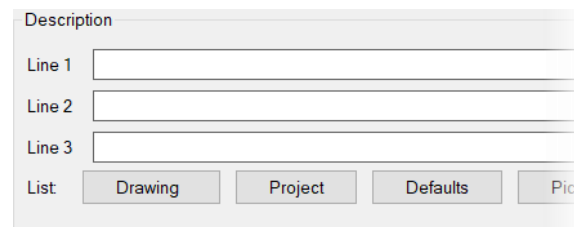
### \* Link Schematic and Pneumatic Components Using WDTAGALT

In this exercise, you link Schematic and Pneumatic component symbols together using the WDTAGALT attribute. Then you edit the Pneumatic symbol and view the changes on the Schematic symbol.

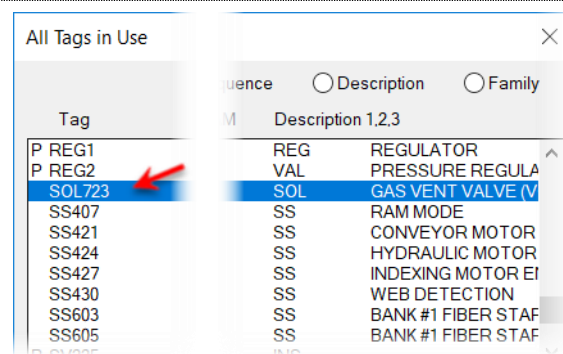
1. In the Project Manager, right-click *Using\_AcadE\_for\_More\_10.dwg*. Click Open.
2. Zoom in to the valve VAL7, located in the lower right corner.



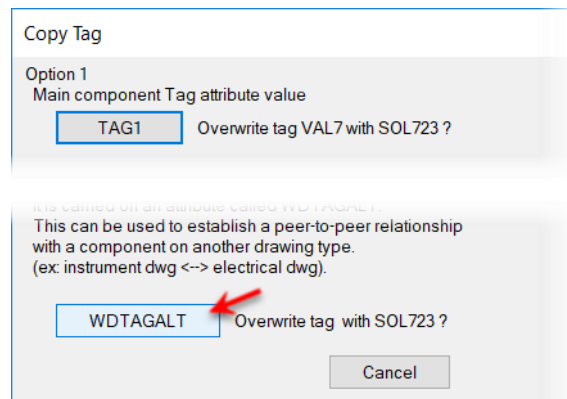
3. Right-click VAL7. Click Edit Component.  
*In the Insert/Edit Component dialog box, notice the blank description lines.*



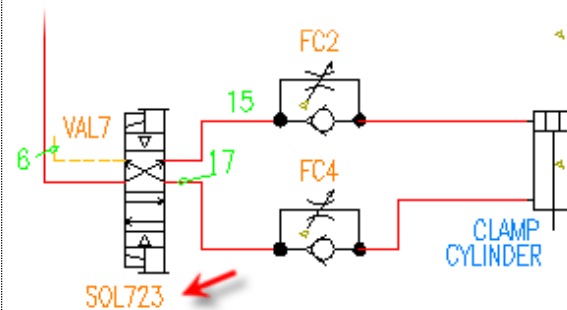
4. Under Component Tag, click Schematic.
5. In the All Tags In Use dialog box, scroll and select SOL723.
6. Click Copy Tag.



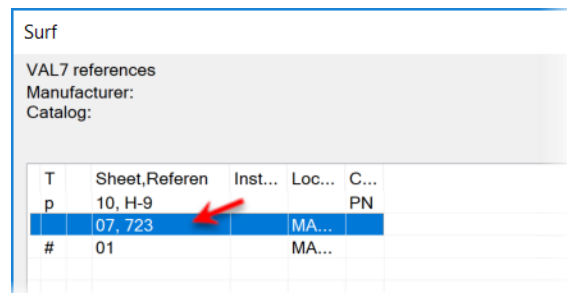
7. In the Copy Tag dialog box, click WDTAGALT.
8. In the Insert/Edit Component dialog box, click OK.
9. In the Update Other Drawings? dialog box, click OK.
10. If prompted, in the Qsave dialog box, click Always Qsave.



11. In the drawing, notice the WDTAGALT attribute now displays the peer Tag value of SOL723.



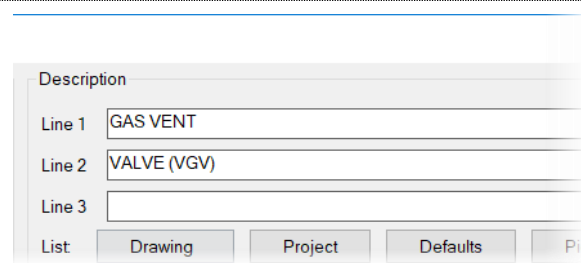
12. Right-click VAL7. Click Surfer.
13. In the Surf dialog box, select 7,723. Click Go To.
14. Click Edit.





15. In the Insert/Edit Component dialog box, do the following:

- Under Description, for Line 1, enter **GAS VENT**.
- For Line 2, enter **VALVE (VGV)**.
- Click OK.



16. In the Update Other Drawings? dialog box, click OK.

17. If prompted, in the Qsave dialog box, click Always Qsave.

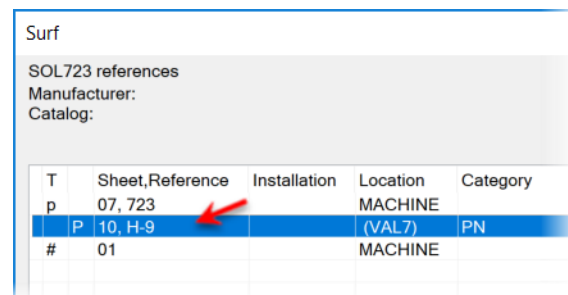
*Notice that SOL723 also displays the WDTAGALT value of VAL7.*

18. Right-click SOL723. Click Surfer.

19. In the Surf dialog box, select PN, 10, H-9.

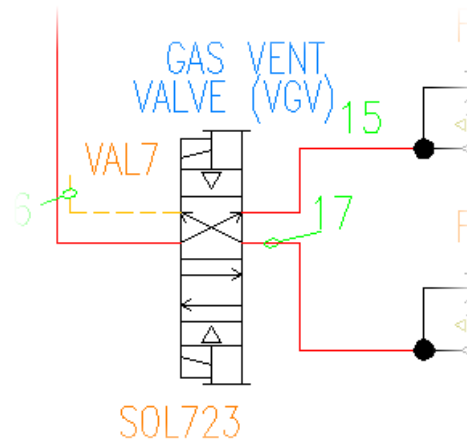
20. Click Go To.

21. Click Close.



22. In the drawing canvas, notice the updated descriptions on VAL7.

*This completes the exercise.*



### \* Using A Custom Schematic Category, Example 1

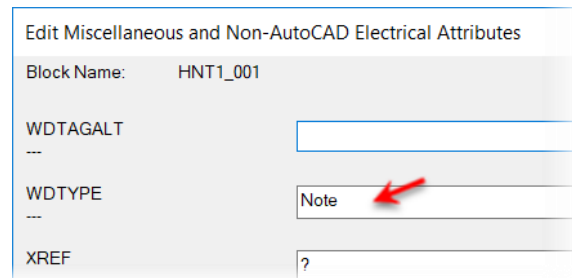
In this exercise, you explore the use of a custom Note block that has the WDTYPE attribute value set to "Note". This is not a "Peer" type of symbol, but a custom schematic category. You insert the matching child block on a separate drawing to create a table of Notes in your drawing. Anytime the parent is updated, you can automatically update the child to match.

1. In the Project Manager, right-click *Using\_AcadE\_for\_More\_10.dwg*. Click Open.
2. Zoom in to Note-1, located in the lower right corner.

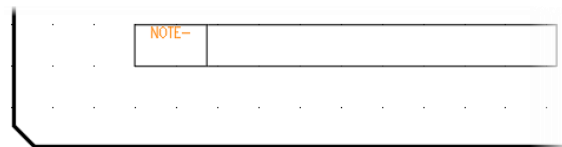


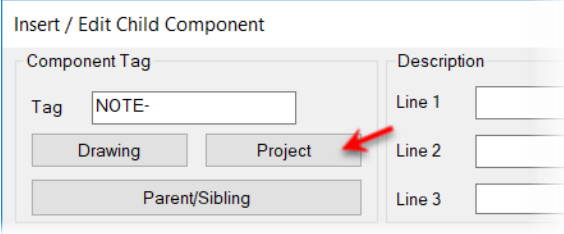
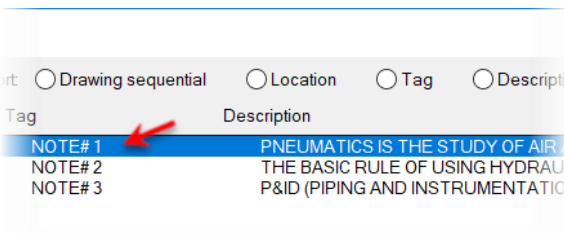
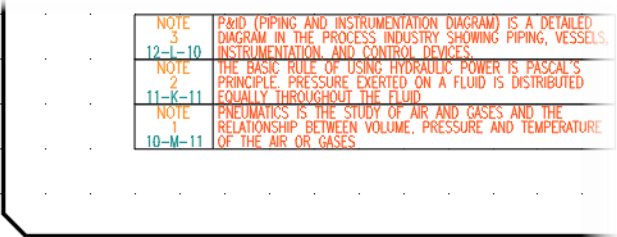
3. Right-click Note-1.
  4. In the Insert/Edit Component dialog box, click Show/Edit Miscellaneous.
- In the Edit Miscellaneous dialog box, notice the value of WDTYPE is set to Note.*
5. Click Cancel.
  6. In the Insert/Edit Component dialog box, click Cancel.

**Note:** There are similar Note blocks inserted on other drawings in the dataset.

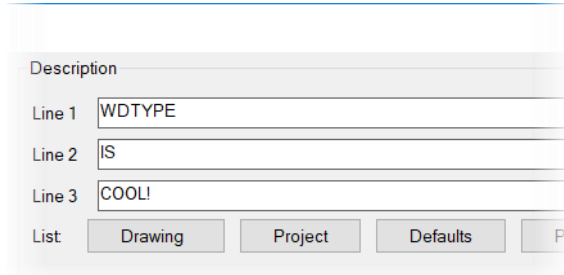


7. In the Project Manager, right-click *Using\_AcadE\_for\_More\_14.dwg*. Click Open.
8. On the Schematic tab, Insert Components panel, click Icon Menu.
9. In the Insert Component dialog box, click Browse.
10. Browse to the exercise file folder and select HNT2\_002.dwg. Click Open.
11. In the drawing canvas, pick an insertion point in the lower left corner.

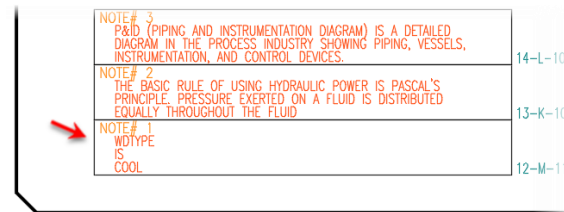


<p><b>12.</b> In the Insert/Edit Child Component dialog box, click Project.</p>	
<p><b>13.</b> In the Complete Project List dialog box, select Note#1. Click OK.</p> <p><i>Notice the data from Note# 1 is copied into the Insert/Edit Child Component dialog box.</i></p> <p><b>14.</b> In the Insert/Edit Child Component dialog box, click OK.</p>	
<p><b>15.</b> On the Schematic tab, Insert Components panel, click Icon Menu.</p> <p><b>16.</b> In the Insert Component dialog box, on the Recently Used panel, select Typed.</p> <p><b>17.</b> In the drawing canvas, select the insertion point at the top left of the first note symbol.</p>	
<p><b>18.</b> In the Insert/Edit Child Component dialog box, click Project.</p> <p><b>19.</b> In the Complete Project List dialog box, select Note# 2. Click OK.</p> <p><b>20.</b> In the Insert/Edit Child Component dialog box, click OK.</p> <p><b>21.</b> Repeat Steps 15-20 for Note# 3.</p>	
<p><b>22.</b> In the Project Manager, right-click <i>Using_AcadE_for_More_10.dwg</i>. Click Open.</p> <p><b>23.</b> Right-click Note# 1.</p> <p><b>24.</b> From the Marking Menu select Edit Component.</p>	

- 25. In the Insert/Edit Component dialog box, for Description, Line 1, enter **WDTYPE**.
- 26. For Line 2, enter **IS**.
- 27. For Line 3, enter **COOL!**
- 28. Click OK.
- 29. In the Update Other Drawings? dialog box, click OK.
- 30. If prompted, in the Qsave dialog box, click Always Qsave.



- 31. In the Project Manager, right-click *Using\_AcadE\_for\_More\_16.dwg*. Click Open.
- Notice that Note# 1 is updated to match changes.*

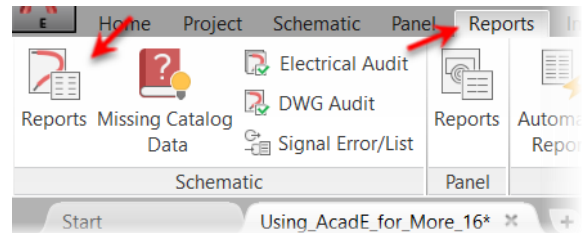


*This completes the exercise.*

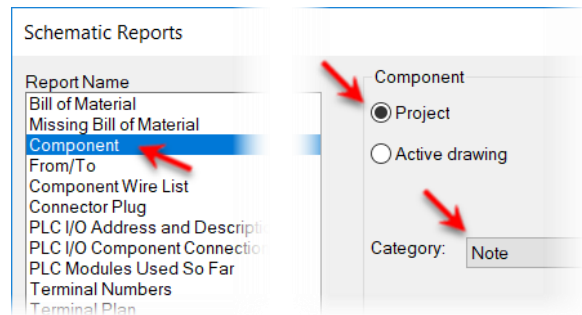
### \* Creating a Note Report

In this exercise, you create a report of the Note entries. If the Notes are edited in the drawing, you will need to regenerate the report.

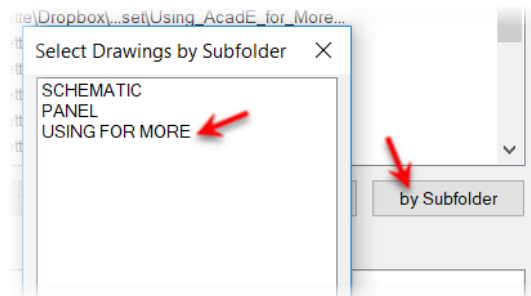
1. On the Reports tab, Schematic panel, select Reports.



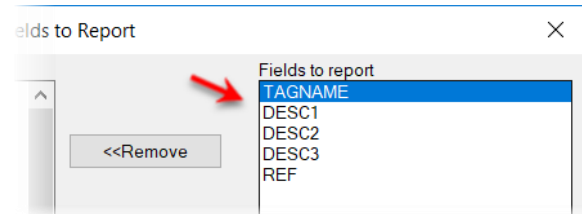
2. In the Schematic Reports dialog box, for Report Name, select Component.
3. For Component, select Project.
4. For Category, select Note. Click OK



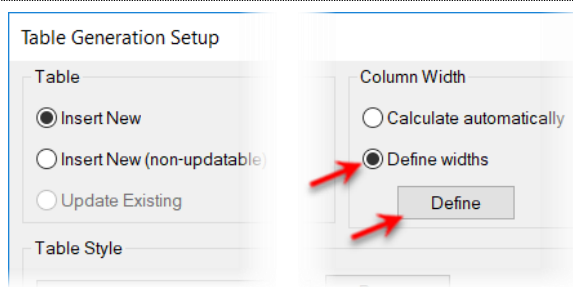
5. In the Select Drawings to Process dialog box, select By Subfolder.
6. In the Select Drawings by Subfolder dialog box, select Using For More. Click OK.
7. In the Select Drawings to Process dialog box, click OK.
8. If prompted, in the Qsave dialog box, click Always Qsave.



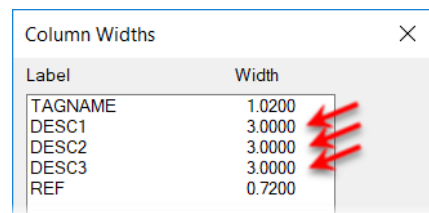
9. In the Report Generator dialog box, click Change Report Format.
10. In the Component Data Fields to Report dialog box, add, remove, and rearrange columns as shown. Click OK.



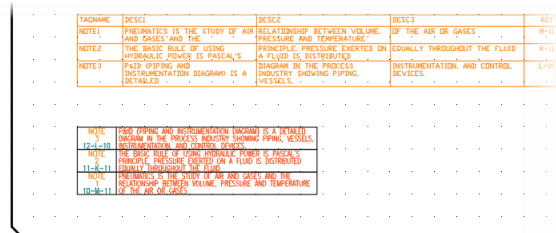
11. In the Report Generator dialog box, select Put On Drawing.
12. In the Table Generation setup dialog box, Under Column Width, select Define Widths.
13. Click Define.



14. In the Column Widths dialog box, select DESC1.
15. For Width, enter 3.
16. Click Update.
17. Repeat steps 32-34 for DESC2 and DESC3.
18. Click OK.



19. In the Table Generation Setup dialog box, Under First New Section Placement, select Pick.
20. In the drawing canvas, select and insertion point for the table at approximately 1.0,5.0.
21. In the Table Generation Setup dialog box, click OK
22. In the Report Generation dialog box, select Close.

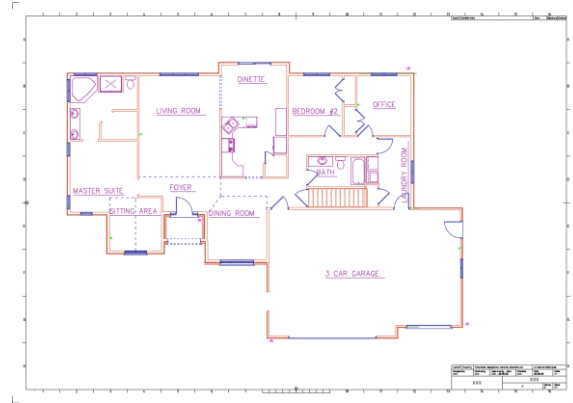


*This completes the exercise.*

### \* Using A Custom Schematic Category, Example 2

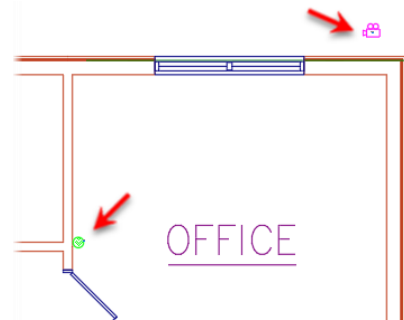
In this exercise, you explore the use of a custom camera and smart home blocks that have the WDTYPE attributes value set appropriately. These are also not a “Peer” type of symbol, but a custom schematic category. This example shows how to use the WDTYPE functionality for a wide range of uses.

1. In the Project Manager, right-click *Using\_AcadE\_for\_More\_13.dwg*. Click Open.

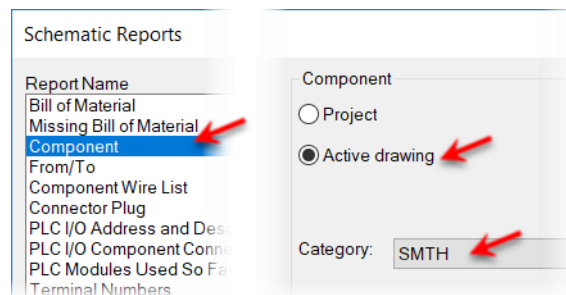


2. Zoom into the office in the upper right corner.

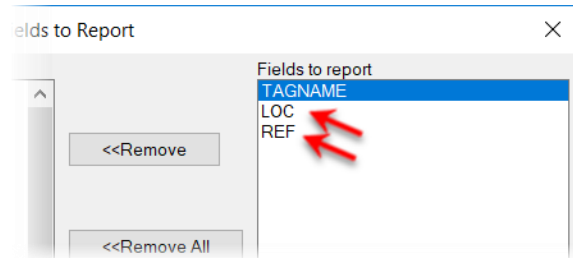
*Notice the Smart Home device on the left wall and the security camera on the outside upper wall. These are both custom blocks with WDTYPE attribute set to SMTH.*



3. On the Reports tab, Schematic Panel, click Reports.
4. In the Schematic Reports dialog box, under Report Name, select Component.
5. Under Component select Active Drawing.
6. For Category, select SMTH.
7. Click OK.



8. In the Report Generator dialog box, click Change Report Format.
9. In the Component Data Fields to Report dialog box, for Fields to Report, select DESC1. Click <<Remove.
10. Repeat Step 9 for DESC2 and DESC3.
11. From the Available Fields list, select LOC and REF.
12. Click OK.



13. In the Report Generator dialog box, review the generated report that includes the TAG, Location, and X-Y grid coordinates for each smart home device.
14. (Optional) Put report on drawing and scale by 24.
15. Click Close.

Report Generator

Drawing Schematic Component report for all locations (9 records)

TAGNAME	LOC	REF
CAM1	FRONT DOOR	F-5
CAM2	BACK YARD	L-1
CAM3	GARAGE FRONT	H-7
CAM4	GARAGE SIDE	N-7
DOT1	OFFICE	K-2
DOT2	LIVING ROOM	D-3
DOT3	GARAGE	N-5
DOT4	KITCHEN	G-2
DOT5	BASEMENT	C-5

*This completes the exercise.*