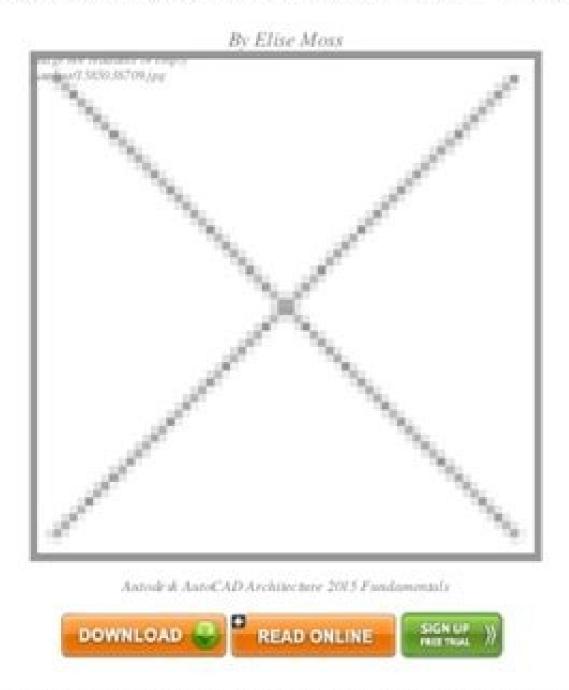
Autocad 2015 training manual free pdf



[Pub470] Autodesk AutoCAD Architecture 2015 Fundamentals PDF



|#1489347 in Books | 2014-10-01 | 2014-10-29 | Original language: English | 10.75 x 8.25 x .751, 1.55 | Binding: Perfect Paperback | 292 pages | File size: 58.Mb

I think that Autodesk AutoCAD Architecture 2015 Fundamentals are great because they are so attention holding, I mean you know how people describe Autodesk AutoCAD Architecture 2015 Fundamentals By Elise Moss good books by saying they cant stop reading them, well, I really could not stop reading. It is yet again another different look at an authors view.

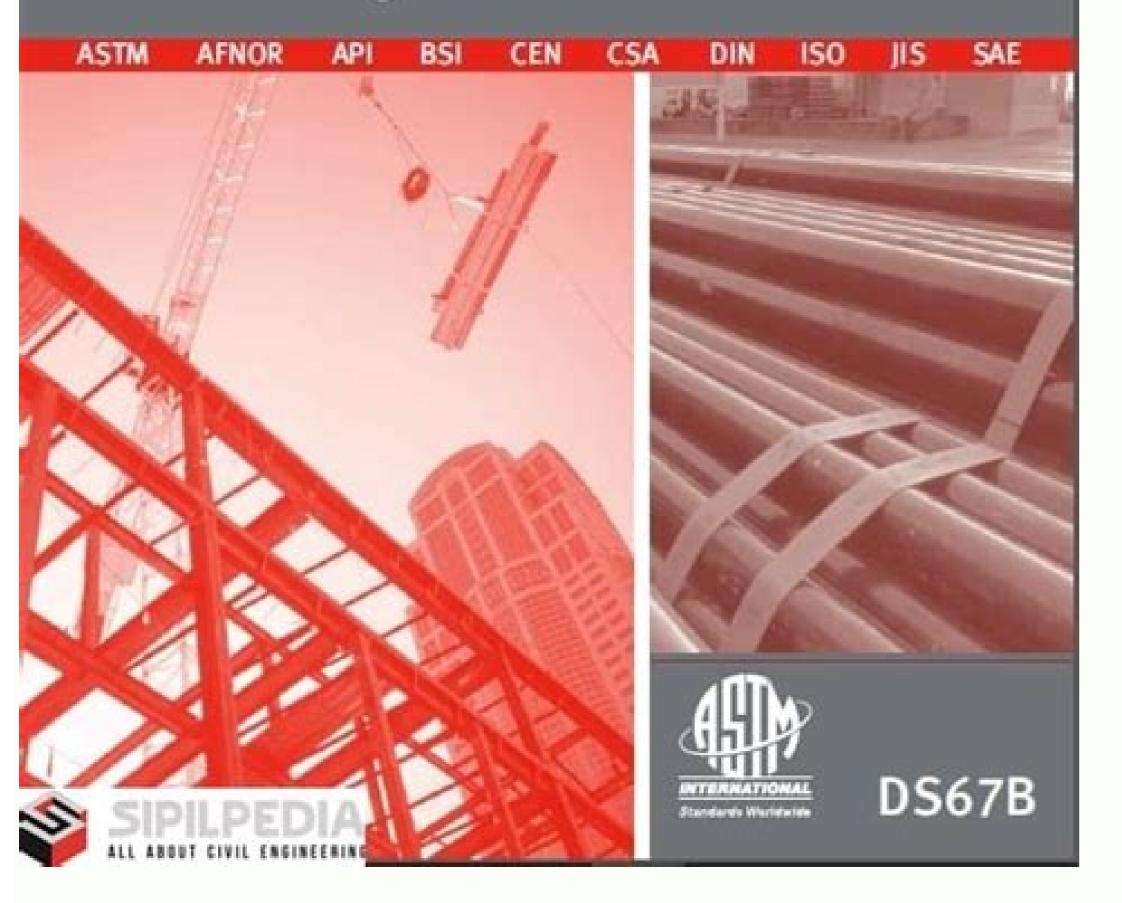
The many reviews about Autodesk AutoCAD Architecture 2015 Fundamentals before purchasing it in order to gage whether or not it would be worth my time, and all praised Autodesk AutoCAD Architecture 2015 Fundamentals: This fundamentals text introduces you to Autodesk's AutoCAD Architecture 2015 software The book covers the Layer Manager Design Center Structural Members Doors Windows and Wall's Step by step lessons take the reader from creation of a site plan floor plan and space planning all the way through to the finished building a standard three bedroom two bathroom residence By the end of the text you should feel comfortable enough to create a standard model and

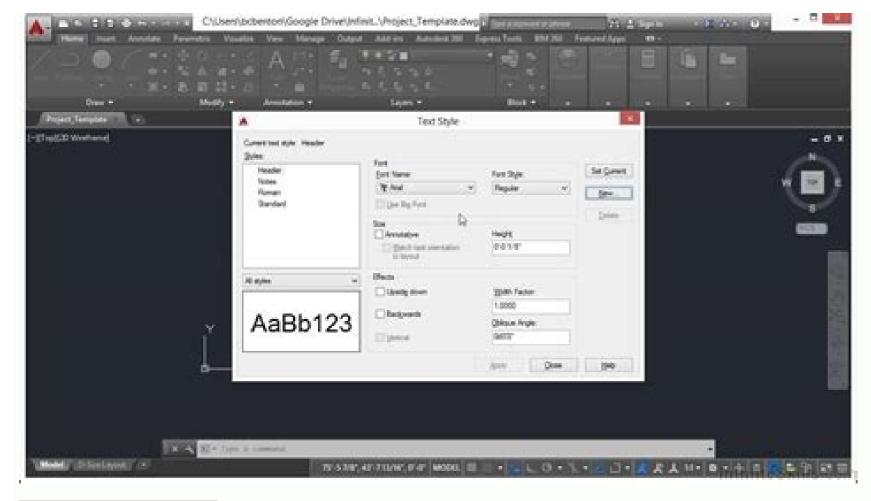
You can download in the form of an ebook: Autodesk AutoCAD Architecture 2015 Fundamentals, this is a great books that I think are not only fun to read but also very educational.

Click button and Read Online Now:

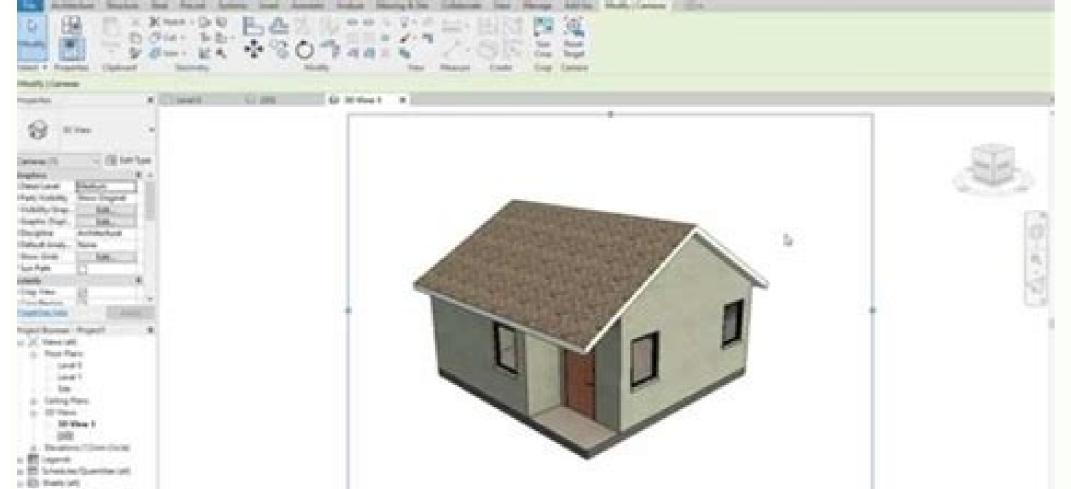
Handbook of Comparative World Steel Standards Third Edition

John E. Bringas, Editor









A DECK OF A

AVE Dealed		
at 22 Aprelias		
at the finance		
- me fact come		
	the second se	

建在标准计算机推进;

T 17 10 17 10 10 10 10 10

HOME REFERENCES TIPS/TRICKS LINKS CONTACT Welcome to Electrical Computer-Aided Design Consulting Headquarters Doug McAlexander, Inc. AutoCAD Electrical Expert Elite (770) 841-8009 Tips and Tricks Pre-paid phone/e-mail/video support is available. You can contact Doug with a via phone, text, or e-mail and receive a prompt answer via phone, e-mail, or text, or possibly a video with step-by-step instructions. This page includes some "freebies" to give you an idea of the kind of support you can expect from our pre-paid plan. Videos (Press your browser's Back button to return to this page) The Super Project Manager add-in for AutoCAD Electrical, available exclusively from Doug McAlexander, Inc. The Dynamically Paired Source and Destination add-in feature for AutoCAD Electrical, available exclusively from Doug McAlexander, Inc. The System-wide Update Block tool for AutoCAD Electrical, available exclusively from Doug McAlexander, Inc. The System-wide Update Block tool for AutoCAD Electrical, available exclusively from Doug McAlexander, Inc. The System-wide Update Block tool for AutoCAD Electrical, available exclusively from Doug McAlexander, Inc. The Multi-Label tool for AutoCAD Electrical, available exclusively from Doug McAlexander, Inc. How To's INDEX (click text to advance to area of interest) 12 steps for creating your own custom "smart" border/title block template for AutoCAD and AutoCAD Electrical Converting plain "Legacy" AutoCAD & Electrical format Sub-Project solutions for AutoCAD Electrical format Sub-P support files used by AutoCAD Electrical Understanding some of the optional symbol attributes used by AutoCAD Electrical Understanding the WDBLKNAM attributes us Fan-in/Fan-out feature to document cables and cable cores Adding Cable Data to a Wire From/To Report AutoCAD Electrical allows the Schematic to also serve as a Wiring Diagram Using AutoCAD Electrical to create a power bus from terminal blocks How to define a feed-through terminal block with 3 connection points, 2 on one side and one on the other side How to get AutoCAD Electrical subassembly parts into the BOM with unique item numbers How to use Footprints and Wiring-Diagram-Style Footprints and Wiring-Diag Icon Menu How to add more DIN rails and wireways to the AutoCAD Electrical DIN rail menu Sharing AutoCAD Electrical over a network List of Family Codes used by AutoCAD Electrical Converting from promis-e to AutoCAD Electrical AutoCAD Electrical IEC tag mode (Combined Installation/Location tag mode) explained The origin of the JIC symbols that ship with AutoCAD electrical Get on the fast-track to productivity with training by a 20+ year veteran of electrical controls design using intelligent CAD 12 steps for creating your own custom "smart" border/title block template for AutoCAD Electrical This is an excerpt from a more detailed guide that is available for those who attend my Admin+Workflow training class. There are more detailed guide that is available for those who attend my Admin+Workflow training class. developing consistent and accurate project drawings. I like to say that the drawing template is at the heart of any well-constructed project. If you want to get into detail at creating a fully-functional drawing template that maximizes the automation available from AutoCAD lectrical, you can purchase a support retainer and use the time for assistance with developing your own drawing template. Note that virtually all of my training or phone/web-conference. Call (770) 841-8009 or send an email to ECADConsultant@gmail.com for your company Type ATTDEF at the command prompt to insert attributes. Insert attributes with tag names such as SHEET, TOTALSHEETS, DESC1, DESC2, DESC3, REV, DWGNO, etc., setting such properties as text height, width factor, justification etc. 2. Create a project called Border_Template. Don't worry about putting any drawings in it. Inside the Border Template folder create a new text file named Border Template_wdtitle.wdl. If neither is found in the project folder first for either PROJECTNAME_wdtitle.wdl. This WDL file controls the language of the prompts you get when you right click a project name in the Project Manager and select Descriptions to enter the project line label data. The default is LINE1, LINE2=your next prompt here (i.e. Project Descriptions) LINE3=your next prompt here (i.e. Project Description) LINE3=your next prompt here (i.e until you have accounted for every attribute, except SHEET, TOTALSHEETS, DESC1, DESC2, DESC3. 3. Click the Project menu tab and select Title Block Setup. Choose method 2 and click OK. This method automatically creates a mapping attribute named WD TB. The data for the WD TB attribute will be filled in for you in the next step, but will be marked as invisible. The WD TB attribute value maps the project line label data that you enter from the prompts in the Border Template wdtitle.wdl file to the matching attribute you want the data delivered to during the Title Block. Update operation. Again, do not try to assign anything to SHEET, TOTALSHEETS, DESC1, DESC2, DESC3 yet. That is covered next. 5. Click on Drawing Values button at lower right. From this screen you match up Drawing Description 1 with DESC1, Drawing Description 2 with DESC2, and Drawing Description 3 with DESC3. Note: When you perform a title block attributes. 6. Save the border drawing as something like D border base.dwg for a D-size border drawing, I suggest saving to the Template folder. Hint: The easiest way to find the template folder. Hint: The easiest way to find the template folder. SaveAs type to .dwg. 7. Click QNEW or File>New and use the acad.dwt template to start a new blank drawing is aved base drawing is aved inserted the attributes you placed on the base drawing will not be visible. This is normal. 9. Click the Drawing Properties icon from the Schematic menu tab. Accept the insertion of the WD_M block. This block stores Drawing Properties. Click through the various tabs and set design rules, such as Tag, Wire Number, and Cross-reference Format, default Ladder Width, etc. During my Admin plus Workflow Training class we cover this is greater detail. 10. Click Create/Edit Wire Type from the Edit Wire menu. Enter your company's most popular wire types. Note: Don't forget to assign a layer color and line type. Hint: You cannot assign a color or line type until you click in another cell after typing in the wire type information. 11. Insert some wires and components to create a sample circuit, so you can adjust color and line type assignments. As you insert various drawing elements, new layers will be added to the drawing by AutoCAD Electrical. Be sure to insert a 3-phase component symbol, a location box, and wire numbers. Next click Layer Properties on the Home menu tab and adjust line types and layer colors for the new layers that have been added. For example layer LOCBOX should be set to a line type of Dash Dot to meet IEC61082, or simplay a dash, as is customary in the U.S. I usually use Dash Dot for both IEC and U.S. drawings. The color is usually works for ANSI D-size drawings while linetype Dashed2 usually works for ANSI B-size drawings. Type LTSCALE at the command line and press the Enter key on your keyboard to adjust the line type scale until your dashed line types appear as fine or coarse as you desire. Also consider the color assignment for TAGS, TAGFIXED, WIREFOPY, TERMS, etc. You need to edit at least one wire number and set it to "Fixed" in order for the WIREFIXED layer to be added by AutoCAD Electrical. Also insert a wire number copy so layer WIRECOPY is added. Edit a component and set the tag to "Fixed" so layer TAGFIXED, WIRECOPY, and TAGFIXED. I use the color for WIREFIXED. I use the color RED for fixed layers. Click Configuration on the Panel menu tab and insert the WD_PNLM block. Click Balloon Setup and adjust as needed. Type TEXTSIZE at the command prompt and press enter. I also recommend establishing your preferred Publish/Plot parameters in the template. For example, if you want plots to be monochrome, 11x17, centered, zoomed to extents, fit to the paper, etc., enter these parameters using the Page Setup utility. I also recommend setting the default layer to layer 0 (zero) and disabling running OSNAPS. Running OSNAPS. Running OSNAPS can cause problems when editing schematic drawings. I do however recommend setting the drawing to use a pre-defined SNAP, perhaps set to 0.125 for a D-size drawing environment and 0.0625 for a B-size drawing environment, or 1.25mm for an IEC A3 drawing environment. 12. Finally, click File>SaveAs and set the SaveAs and set the S folder, in case you ever need to replace or update the border on existing drawings. When you click the New Drawing button in Project Manager set your Template. Note: You will only need to set the path once. It will remain until you change it. Understanding the differences between AutoCAD and AutoCAD Electrical Many people get confused by the fact that AutoCAD "Electrical", which was formerly known as an add-in program called "Wiring Diagram", installs its tools into the AutoCAD menu structure, so it can make use of the AutoCAD drafting engine. It is not wise to assume that standard AutoCAD commands will produce results that "Electrical" can interpret. Electrical must maintain control of the drafting engine doesn't know how to interact with the databases. When you use standard AutoCAD commands you are going off-line and the results could be ambiguous. Sometimes manipulating things with standard AutoCAD commands might work but you have to be familiar enough with the inner-workings of Electrical to know how to manually manipulate the data in such a way that the Electrical data manager can interpret what you did. For example you can use ATTDEF and AutoCAD LT to create symbols that are compatible with Electrical but you must know the proper syntax for the attributes, the symbol naming conventions, etc. However, the Symbol Builder on the Schematic tab of Electrical will act as a wizard and guide you through the symbol creation process such that your symbol is sure to offer the Electrical data manager properly formatted data to work with. You may reason that the only thing that makes a line a wire is the layer name (which must be listed in the Create/Edit Wire Type table), thus you can assign a wire to a different layer by using the standard AutoCAD Layer utilities. This is true, but it will be necessary to select each and every wire segment in the node (a.k.a. wire network), since standard AutoCAD doesn't know how to treat a line segment as a wire, and will not reason that all segments make up one wire. Another example of the superiority of Electrical commands has to do with editing a symbol that has egment as a wire, and will not reason that all segments make up one wire. already been inserted. A standard rule of AutoCAD is that, once a block name is inserted into a drawing, AutoCAD simply copies the one that already exists on the drawing. Why does it do this? The one on the drawing is already loaded into RAM. To load the same block from the hard drive takes additional computers. (I started out with an Intel 286/12MHz computer that had 512KB of RAM.) AutoCAD functioned faster for repeat block insertions by copying the one that was already loaded into memory. So if you made changes to the block, purge the unused blocks, and re-insert the block. You would also need to repopulate any attribute values that were lost. AutoCAD "Electrical" makes this operation quick and easy with the SWAP/UPDATE tool. Competitive electrical design programs that install into AutoCAD drafting engine, turn off all standard AutoCAD drafting engine, turn off all standard AutoCAD drafting engine, they may have an icon that looks just like AutoCAD Erase tool, but if you click it, the code is re-routed through their application. Their application and is no longer needed, the Erase command trims the excess wire. If you click a wire that is complete and connected on both ends, the entire wire is erased. If you click on a component symbol, the component symbol is erased. This is all done with the same icon. In order for this economic use of menu icons to work, grips are disabled. You must select the drawing element you wish to affect. This is only one of many examples where the electrical add-in program takes over the command structure of the drafting engine in order to dictate a work-flow that will ensure data integrity. Some may not like the fact that all standard AutoCAD commands are not available with the competitive add-ons, thus they would probably prefer the Autodesk version. The makers of AutoCAD & Electrical decided to give you full use of standard AutoCAD commands, so you have a one-package that fits all of your DWG editig needs. This allows you to edit legacy AutoCAD drawings with the same software that you use to create electrical control system wiring diagrams. However by accepting software with this much horsepower under the hood you must also accept the responsibility of knowing when to use which of its many tools. Proper classroom training from a qualified instructor is critical to gaining full use of "Electrical". The Ribbon menu has helped. It is a new user-interface environment that was introduced with version 2010. I was resistant to the Ribbon menu at first but, as an instructor, I quickly realized its benefits for new users. The Ribbon menu can significantly reduce the learning curve for a new user and enforce proper work-flow. For example, the Home tab on the ribbon menu is where you find standard AutoCAD commands, those which can "break" your intelligent electrical drawings if you don't know when and when-not to use them. One rule of thumb that I encourage new users to follow is to stay away from the Ribbon menu's Home tab until they have attended classroom training by a qualified instructor. The Project, Schematic, Panel, Reports, etc. tabs are for electrical functions and they go in order (left-to-right) of the typical electrical design work-flow. You start by creating a project, followed by the schematic, the panel layout, and reports. I also advise new users not to type commands at the command prompt. This will prevent those who are experience users of plain "vanilla" AutoCAD from using commands that might break their electrical continuity. Once they become experienced with the inner workings of AutoCAD Electrical, some manipulation from the command prompt or the Home tab can be acceptable for certain situations. Experienced users of AutoCAD tend to "go home to momma" and use commands they know. After all, the name of the software has the word AutoCAD in it, right? But remember that AutoCAD is merely the drafting engine. The real intelligence lies within the databases and how they "handshake" with the drawing data, the "Electrical" portion of the code. The drafting engine provides the user with visual feedback so the user with visual feedback so the user can see what type of data is being written to the database. Consider the symbols for example. The way they look is important for the user, but not to Electrical interprets the attribute values within the symbol is for the user's benefit. Thanks to the ribbon interface, you no longer see the standard AutoCAD commands (unless you click on Home) which could damage your electrical drawings and lead to strange results or errors from the project database manager. Unless you intentionally select the Home tab, or type at the command prompt, you are working with for reports, error analysis, etc. I strongly encourage a new user of AutoCAD Electrical, including those who are intimately familiar with standard AutoCAD. to attend a hands-on, instructor-led training course in order to gain the full benefit of implementing AutoCAD. interface, avoiding the temptation to use the Home tab or type standard AutoCAD commands. You may think you don't have the time or money to spend on proper training, but start adding up the cost of the time lost in floundering around on your own, or visiting the Electrical Discussion Group for answers, and I guarantee you that a 4-day training course will pay for itself many times over. Converting plain "Legacy" AutoCAD & drawings to AutoCAD & drawings usually consist of simple lines and text to represent wires and wire numbers, and simple blocks to represent schematic symbols. They ask me how they might be able to use or convert their existing drawings in order to avoid completely redrawing everything using AutoCAD Electrical included a help section to assist those converting from plain AutoCAD. But this seems to have been removed in the past few releases. I recommend that you first create the drawing template you will use to create each new AutoCAD Electrical drawing. That procedure is covered here on this page. Then follow the steps below. 1. Create an AutoCAD Electrical drawing. That procedure is covered here on this page. copy and paste the circuit from plain AutoCAD into this fresh, blank, drawing. 3. From the Conversion Tools menu tab click the Change/Convert Wire Type button and convert the lines that represent wires into wire type layers you defined in your AutoCAD Electrical drawing template. Once the non-intelligent lines have been assigned to intelligent wire type layers they can be managed completely by AutoCAD Electrical. 4. If your old drawing contained ladders, use the Convert Ladders utility to swap symbol blocks from the old drawing for those recognized by AutoCAD Electrical. Click on Block Replacement and choose Drawing (all). Type in a name for the spreadsheet with the first column filled in with the block name of the AutoCAD Electrical symbol you wish to switch to. Note that the blocks will be inserted from whichever schematic symbol library you defined in Project Settings. If you are new to AutoCAD Electrical you might not be familiar with the block names for the symbol you wish to switch to. wish to use and the block name will appear as a popup. Type this block name into the second column of the spreadsheet that the Block Replacement utility again. This time you will notice that the utility allows you to select the previously-saved spreadsheet. Select it and click Save. I know that clicking Save makes no sense but Autodesk just chose to use the same dialog box for both creating the replacement process. Hint: The Project option can be used if you have already created every drawing you need and you have already copied with a base point the non-ACE compliant drawing data into each of the new drawings. 5b. You can alternatively swap one block at a time using the Convert to Schematic Component tool, which is also located on the Convert signal arrows to intelligent Source and Destination symbols. 6. Use the Convert Text to Wire numbers to smart wire numbers to smart wire numbers. 7. There is a Stretch Wire utility included on the Convert any existing text-based wire numbers. not completely attach to their respective wire connection attributes. This is a broad overview of the procedure that I teach my students, but this will get you started. Once these conversion are made you can begin "tweaking" and detailing using AutoCAD Electrical commands, such as assigning catalog numbers using the Edit Component tool, Scooting a component along a wire (in case the new ones are larger - resulting in crowding), and more. Sub-Project solutions for AutoCAD Electrical Quite often I need to assign a different document number to the various types of drawings in a project. For example, the schematic must be a unique document number with unique data in the title block. including page numbering that applies only to the pages that make up the schematic. I must also have a unique Document Number, Drawn By, Checked By, etc. for the panel layout, the wire list, and the bill-of-materials. After all, on a large scale project, these might each be created by different designers. But I still need the intelligent connectivity between the schematic, panel layout, wire list, and BOM for editing purposes, and for a "global" view of the entire project. This can be accomplished within the Project with all necessary drawings, as usual. Next create another project for schematic drawings only, placing its respective .WDP file inside the same folder as the master project. WDP file. Hint: Keep all drawings to this project and select Add Drawings to this project and select Add Drawings in the master project folder. finally for the bill-of-materials. You now have one master projects " which you can activate to performing master level edits and updates. But you also have 4 individual "sub-projects" to have its own unique project line label data (i.e. Document Number, Job Number, Drawn By, Checked By, Date, Revision, etc.). You can activate each "sub-project". Execute a separate Title Block Update for each "sub-project". Execute a separate Title Block Update for each "sub-project". Manager to be an asset. The Super Project Manager (aka SPM) is an add-in that is available exclusively from Doug McAlexander, Inc. There is a link in the Videos category near the top of this page that allows you to see the Super Project Manager in action. You start the Super Project Manager in action and you are given a dialog that allows projects and select the drawings for each with one easy-to-use interface. These projects are then automatically added to the Project Manager. In the following screen shot you see the Super Project Manager. In the following screen shot you see the Super Project Manager. In the following screen shot you see the Super Project Manager. In the following screen shot you see the Super Project Manager. schematic, denoted as [SCH], while the remainder of the drawings are assigned to sub-project 2, denoted as [PNL] for panel layout. The Super Project's name to create the sub-project title inside brackets and appends this to the master project's name to create the sub-project schematic. schematic and PNL for panel layout. They could just as easily have been named for document numbers. In the next screen image you will see the master project that already existed and the two sub-projects that Super Project. wide operations, etc. But I also have a separate project for the schematic which allows me to enter different project line label information and run a separate title block update, so the schematic can have unique description text. The same is true for the panel layout. I can also create subprojects for the wire list, bill-of-materials, etc. and do the same for them. You might also use the Super Project will have its own dedicated database which will only need to track changes to the limited number of drawings in the sub-project. Note: You can reference the same drawings can also stay listed in the master project. Changes made to the sub-project drawings in the master project. Manager allows you to manage very large projects efficiently without giving up the reliability of a drawing-driven electrical program. The attraction of database and drawings are only refreshed when they are opened for later viewing or printing. This reduces the amount of time the program must spend updating drawing files. However if the database crashes, hours of work could be lost before it is written to the affected drawings? Of course the data is stored in the drawings themselves. Each time an edit is made that affects other drawings the user must await the real-time update, or append it to a running task list that can execute all updates at once, perhaps during a coffee break or lunch break. The Super Project Manager will allow you to work at sub-project level, dealing with fewer drawings and speeding up the real-time processing. Changes made in the sub-project(s) are automatically visible in the master project. Since the data is stored at drawings at sub-project level. So the Super Project Manager allows for a best-of-both-worlds approach to electrical CAD. Creating a preferred vendor parts catalog for AutoCAD Electrical When you share the vendor catalog (default_cat.mdb) across a network it is normal to experience a slower response time when performing a part Lookup is also affected by the size of the default_cat.mdb file. The more manufacturers you select during the installation process, the larger the default_cat.mdb file, resulting in a slower seek-time. There is an alternate method you can use if you would like to have access to all manufacturers from time to time but would also like to work from a smaller "preferred" list of manufacturer parts. First, make a copy of the default cat.mdb file (normally located in the Catalogs folder) and remove all manufacturers and/or part numbers you will never use. If you never use AB, then remove AB parts from all component tables (i.e. CR, LT, SS, PB, etc.). Note: If you know you will never us AB parts, do not select the AB checkbox from the manufacturer's list when installing AutoCAD Electrical. Only check the boxes for the manufacturer's list when installing AutoCAD is the next thing to do is to right-click on your project name and select Properties. Click the button labeled "Other", then click on the radio button labeled "Other", the radio butt response when performing a part number Lookup. Anytime you need a part number from the complete catalog, click the "Other" button from within the Lookup dialog (version 2010 and previous) and follow the prompts to temporarily switch to the complete catalog. For version 2011 or later click the arrow in the field labeled as Database and select the Secondary File. Understanding the enhanced catalog number Lookup capabilities of AutoCAD Electrical includes significant enhancements to the part number (a.k.a. catalog) Lookup tool. When you click Lookup you will see that the top left box under Catalog is blank. Click in this box and start typing a part number. The results in the window below will begin to narrow down according to what you type in. You can perform similar filtering for other columns, such as Description, Miscellaneous, etc. You can drag and drop fields left or right to order them according to your needs. You can also right-click above any field name and uncheck the box for any fields you wish to hide. Hint: You can also type a portion of a part number into the Search Database field using an asterisk (*) for a wildcard. Then click the magnifier icon to begin the search. This will search across family tables so it is possible to select a part from the TRMS (terminals) table and assign it to a fuse, for example a fuse holder style terminal. Understanding some of the various support files are discussed in detail in the Help system built into AutoCAD Electrical. The fact is that some files can be located in more than one location, depending upon the user. For example, the main parts catalog is the default cat.mdb file, located by default at My Documents/AcadE 2008/Catalogs. You could copy this file to a specific project folder and rename it to cat.mdb, then make modifications as necessary for the specific project. For example, let's say you want to delete all but the preferred suppliers for each type of component. You could take a similar approach with the default.wdw file, normally located at C:\Documents and Settings\Doug McAlexander\Application Data\AutoCAD Electrical 2008\R17.1\enu\Support\User. Move this file to a specific project folder and modify as necessary to force the naming convention for wire color/gauge labels. Another file in the User folder that might be modified for use with a specific project is the default will most likely need to be modified to meet your needs. If you use different title blocks, perhaps customer-specific, you may need to place a wdtitle.wdl file in the specific project. If you choose to use the User Defined Attributes functionality AutoCAD Electrical will create a .wda file in the folder of the active project. The WDA file stores a list of custom attributes you may have added to some blocks and that you wish to make available for reports. The bottom line is that AutoCAD Electrical looks first inside the project folder for these files before it will use a default file from the User folder. During a project copy you will see a list of support files that AutoCAD Electrical will search for. If any are found they are checked by default, assuming that you will want to copy the project specific support files. There is one particularly useful. You create this one yourself. It is a simple text file that you create and name wd fam.dat. This file, if found in the project folder, will cause AutoCAD Electrical to ignore the standard component tag designation, as listed in the wd fam.dat file. For example, if you are using JIC symbols but wish to have switches tagged with their IEC/ISO-81346 equivalent, you might have entries in the wd fam.dat file like this: SS,S PB,S LS,S TS,S PWS,G CR,K etc. According to the list above, selector switches (SS), push-buttons (PB), limit switches (SS), push-bu is functional during component insertion and during retag operations. Understanding some of the optional symbol attributes used by AutoCAD Electrical AutoCAD Electrical AutoCAD Electrical the information it needs to interpret what kind of symbol it is and what to do with it. The CONTACT attribute may have a value of NO, NC, or NCNO for example. This tells AutoCAD Electrical whether the contact is to be treated as a normally-closed (NC), or Form-C (NCNO a.k.a. change-over contact). See example below: The POSITION attributes hold the text value that you can assign to the various positions of a selector switch. Position 1, POS1, might be designated as Manual while Position 2, POS2, mi the text string that indicates the state of the switch in various positions. The letter "O" indicates closed or connected. The PINLIST attribute can be filled automatically from the Pin List Database (provided there exists an entry for the particular MFG and CAT combination). You can optionally click the NO/NC Setup button while editing the parent or coil of a relay or contactor and enter the pin list. The syntax is explained on the dialog. Basically you enter the contact type (i.e. 0 for convertible, 1 for NO, 2 for NC, or 3 for Form-C), followed by the first contact type (i.e. 0 for convertible, 1 for NO, 2 for NC, or 3 for Form-C). assignment (i.e. A1Y), followed by a semicolon as a delimiter to start the next group. So a syntax of 0,A1X,A1Y;0,A4X,A4Y indicates the relay has two convertible contacts with one designated as terminals A1X and A1Y while the other is designated as A4X and A4Y. The WDTAGALT attribute is used to connect symbols that appear on two different design peers. For example the solenoid symbol for a schematic looks different than the solenoid symbol for a pneumatic drawing. In the example screen image the solenoid symbol for a schematic looks different than the solenoid symbol for a pneumatic drawing. In the example screen image the solenoid symbol for a pneumatic drawing. In the example screen image the solenoid symbol for a schematic drawing. In the example screen image the solenoid symbol for a pneumatic drawing. pneumatic equivalent symbol representing the same solenoid. The pneumatic symbol would have a TAG value of FV100B and a WDTAGALT value of SOL2501. This peer-to-peer tags in the Electrical Helps under Advanced Productivity, but you can simply click the Show/Edit Miscellaneous button on the Insert/Edit dialog of a solenoid or pneumatic valve symbol and enter the tag name for the related symbol directly if you wish. Understanding the WDBLKNAM attribute in AutoCAD Electrical Allow me to clarify one issue that has confused many of us. It involves the WDBLKNAM attribute. You don't normally need a WDBLKNAM attribute to send the Lookup operation looks at the 2nd and 3rd character in the symbol file name to determine what table to look in. If you wish to override this and direct the Lookup to open a different table, you can enter the alternate table name in the WDBLKNAM attribute, preceded by an underscore. Note: Earlier releases of AutoCAD Electrical ginored the first character in the WDBLKNAM attribute value so we had to enter some value as a placeholder. I used the underscore character. To further illustrate this let's consider the terminal-block-style SPDT relays as an example. The catalog data might actually be located in the TRMS to to the HCR1 and VCR1 relay symbol blocks and enter a value of TRMS to redirect the Lookup to the TRMS table. I would prefer to create an alternate relay coil symbol with a WDBLKNAM attribute, and add TRMS to the symbols. This way I only look into the TRMS table when the relay coil is a terminal block style relay. So I would copy HCR1.dwg, rename it to HCR1 TRMS.dwg, and add the WDBLKNAM attribute with a value of TRMS. I would also copy VCR1.dwg, rename it to VCR1 TRMS.dwg, and add the WDBLKNAM attribute is very helpful for the footprint symbol file names don't coincide with the same naming convention as the ones for schematic symbols. Footprint file names can be just about anything that makes sense to you. I tend to name them for the manufacturer and part number, but it is your decision. If you insert footprints from the Schematic symbols. Footprint file names can be just about anything that makes sense to you. I tend to name them for the manufacturer and part number, but it is your decision. was not assigned at schematic level, or if you are laying out your panel first, the Lookup will bring up the complete list of tables and you will need to select the appropriate table. But if you add a WDBLKNAM attribute in symbols confused with the WDBLKNAM field in the catalog database records. The WDBLKNAM field in the catalog database is used by the Symbol Name Filtering box checked on the Lookup dialog (default setting) AutoCAD leaves the catalog records for WDBLKNAM entries that match the symbol file name, minus the first character and minus the file extension. If found, AutoCAD Electrical offers only the record or records that match the symbol name, minus the file extension. For example, let's say your favorite control relay is AB 700-P200A1. Click Lookup and find the AB 700-P200A1 record. Click in the WDBLKNAM field and enter CR1, which is the last portion of the file name for the standard control relay coil symbols (i.e. HCR1 and VCR1). The reason you do not need to enter the H or V is because the first character in a symbol file name indicates vertical or horizontal wire connections, which has no bearing on part number searches and assignments. So by entering CR1 into the WDBLKNAM field of the AB 700-P200A1 record you instruct the AutoCAD view form the Insert/Edit Component menu for either symbol block HCR1 or VCR1. You could use this feature to zero in on your favorites for all types of symbols. Note: I noticed a few years ago that the WDBLKNAM filtering started including records with a blank WDBLKNAM field, as well as the ones where I made an entry with a blank WDBLKNAM field, as well as the ones where I made an entry with a blank work of the symbols. entries for every part number in the table and assign it the letters and numbers of the block name it should be associated with, minus the leading H or V, and without the .dwg file extension. I posted a request to the AutoCAD Electrical Idea Station, asking to correct this. Please click on the URL below and Vote Up my request. needs to ignore blanks Try this: Insert a Green Pilot Light (block HLT1G or VLT1G) and click Lookup. You will notice that all of the records displayed as choices are associated with part numbers for green pilot lights only. Click on any record and press the Edit button on the Lookup dialog. Notice that the WDBLKNAM field has an entry of LT1G. All of the green pilot lights from the various suppliers have LT1G entered into their WDBLKNAM field, as a courtesy from Autodesk. All pilot lights have a contactor with AutoCAD Electrical Suppose you have a contactor

that is available with or without aux contacts. There are two ways to approach this. You can simply enter the L1/T1, L2/T2, and L3/T3 pins in the Pin List Database and consider the auxiliary contact pin numbers as something you assign manually when used. Or you might permanently add the aux pins to the end of the Pin List for the contactor, making them automatically available as needed, as illustrated in the following screen shot of the Pin List Database edit dialog. The illustration below shows how to edit the pin assignments in the Catalog Browser, starting in release 2015. If you wish you can just wait until you insert the coil and click the NO/NC Setup button to add the pins on-the-fly. If you insert the main contacts first, using the 3-phase contact from the component menu, AutoCAD Electrical will serve the pins in order as L1/T1, L2/T2, L3/T3, etc. The auxiliary pins will be next available after the main contacts are inserted. When you insert the normally-open auxiliary pins will be next available after the main contacts are inserted. from the Pin List Database. The syntax for the pin list is 1,L1,T1;1,L2,T2;1,L3,T3;1,13,14 with ;1,13,14 denoting the auxiliary normally-open contact. If you have added more than one set of N.O. auxiliary contact pin assignment choices to the pin list you can click the List button below the pins area of the Insert/Edit Component dialog and select a specific pin set for the contact. See the screen capture below: But let's just say I only use the auxiliary contacts infrequently and thus I only have the main contact designated as pins 13 and 14. First I edit the coil symbol and add the part number for the auxiliary contact to the Multiple Catalog list, so it would be included in the BOM. I then click the NO/NC Setup button and add ;1,13,14 to the end of the pin list. That is a semicolon followed by the number 1 for N.O. then a comma with no space and a 13 followed by the number 1 for N.O. then a comma with no space and a 14. The complete NO/NC Setup then appears as follows: 1,L1,T1;1,L2,T2;1,L3,T3;1,13,14. I am now ready to insert the N.O. auxliary contact and pin numbers 13 and 14 will be available. Note: The syntax for a N.C. aux contact with pin numbers 11 and 12 would be ;2,11,12 with the number 2 designating normally-closed. Adding auxiliary contacts to the BOM on an as-needed basis with the ASSEMBLYCODE Suppose you have a contactor that you sometimes add a normally-closed contact, or both, on an as-needed basis. Enter an assembly list value of NOAUX in the record for the NCAUX for the assembly list value of the NCAUX for the assembly list value of the NCAUX in the record for the assembly list value of NOAUX in the record for the NCAUX for the assembly list value of NOAUX in the record for the NCAUX for the assembly list value of NCAUX in the record for the NCA main part in the catalog database. You will enter it "on-the-fly" as needed. When you insert or edit the main part, type in NOAUX in the ASSEMBLYCODE field for the main part if you want a NC aux contact added. Type NOAUX in the ASSEMBLYCODE field for the main part if you want a NC aux contact added. add a quantity if you prefer more than one of either. For example, NOAUX;2,NCAUX will get one NO aux contacts. I like playing with assembly codes. We explore all options in my training to cover the various scenarios, such as with this contactor scenario. A sample screen shot from my training to cover the various scenarios, such as with this contact. I like playing with assembly codes. We explore all options in my training to cover the various scenarios, such as with this contact. in/Fan-out feature to document cables and cable cores. I document cables and cable cores. I document cables and cable cores. I document the cable to another page and breaking it back out again into individual cable cores. I document the cable cores and then sum into one line, passing the cable to another page and breaking it back out again into individual cable cores. I document the cable cores and then sum into one line, passing the cable cores. workflow isn't covered in Help. First, insert the wires that will be summed into one line. You can assign them to a specific wire type called Cable Conductor and assign the wires to this layer/type. When you insert cable markers, they will define the cable core color or number. You can insert the cable markers either before of after you insert the fan-in/fan-out symbols are inserted, the wire type on the side that will be assigned to layer _MULTI_WIRE segments to one plain drafting line and assign this. After the fan-in/fan-out symbols are inserted, the wire type on the side that will be summed into one plain drafting line and assign this. line to layer _MULTI_WIRE. You can use a Fillet at the join point if you wish. Hint: You can use Match Properties to copy the layer assignment from one of the _MULTI_WIRE must break and continue at another location, use Stand-alone Cross-references at the break points, because standard Source and Destination arrows will not function with the MULTI WIRE layer. The attached screen shots, from my advanced training course, illustrate my method. Adding Cable Data to a Wire From/To Report Cable Bummary Report will include basic information about the cable but no details. You can add cable details to a Wire From/To list report by using the Change Report Format option. The default names of the cable data fields are as follows: CBL = Cable Manufacturer CBLCAT = Cable Catalog Number CBLASM = Cable Assembly Code Once you move the fields into your report, you can rename them as desired. Be sure to save the format so you can use it over again. The wire from/to list below includes cable data with the fields renamed. AutoCAD Electrical allows the Schematic to also serve as a Wiring Diagram I like it when the same documentation that was used to build it can be used to troubleshoot it. When the schematic is also a wiring diagram it is easier to understand the route a wire takes. For example, think of a 14AWG wire branching off from a main circuit breaker from a main disconnect on the door. If I use a dot (the typical schematic only approach) I am only showing the panel shop, or the troubleshooting technician, that the branch wire is connected to the same node as the wire from the disconnect to the branch wire is connected to the same node as the wire from the disconnect to the branch wire is connected to the same node as the wire from the disconnect to the branch wire is connected to the same node as the wire from the disconnect to the branch wire is connected to the same node as the wire from the disconnect to the branch wire is connected to the same node as the wire from the disconnect to the branch wire is connected to the same node as the wire from the disconnect to the branch wire is connected to the branc the circuit. (see example below) But what if my production line is down and I am trying to trace the wire-nut, or is the dot a solder joint? Maybe it's a terminal block? Which wire-ways do I uncover to trace the wire? Does the wire the circuit breaker on the mounting plate? Our European friends would use an angled wire symbol on the schematic to make it clear that the breaker, along with the wire from the main disconnect. (see example below) The angled wire tee symbol in AutoCAD Electrical serves two purposes. Starting with version 2008 the angled wire symbol orientation clears up the ambiguity on the print and also dictates the wiring sequence on the wire from/to list. The orientation you select by pressing the spacebar on your keyboard when inserting or toggling an angled wire symbol also tells the wire from/to list. There is an occasional issue with both dots and angled wire symbols. So your branch wire, and all wires at that node, must all be the same size and color. AutoCAD Electrical had this problem solved back in the 1990s. You can mimic the real world and actually connect two wires to the breaker terminal. AutoCAD Electrical will automatically angle the second wire off at a 45-degree angle. Optionally you can fly-out the Insert Wire menu and select 22.5 or 67.5 degree angles instead. The branch wire can be any size and color of your choosing. (see example below) Using AutoCAD vising AutoCAD to create a power bus from terminal strip. The following is the method I have used since the since introduction of the Terminal Strip Editor (a.k.a. TSE). This method applies to either vertical or horizontal power rails. In this example I will use a standard ladder diagram. Insert the ladder and rungs as usual. Replace the ambiguous node dots with terminal block symbols, since your power rail is actually made up of terminal blocks; not solder joints, wire-nuts, or a daisy-chain of wire. Open the Create/Edit Wire Type dialog and add a new wire type named JUMPER or JUMPER or JUMPER or JUMPER or JUMPER or JUMPER or JUMPER. This takes care of the schematic portion of the power bus. Easy isn't it? The JUMPER or JUMPER-BAR layer will still be considered a current carrying conductor, as it truly is, and will even pass along the wire number (a.k.a. node number) to other connected devices. But it will not appear in the From/To List as a wire that must be connected devices. type (layer) with the word JUMPER anywhere in its name when it creates a wire From/To List. The illustration below shows the JUMPER-BAR layer assigned to the vertical rails (shown in magenta color). Use any color that would not be confused with your wiring standard. I chose magenta since I have no magenta wires in the system. Now to the panel layout. Use the Terminal Strip Editor (a.k.a. TSE) to insert the terminal strip. Hint: I don't normally assign the MFG and CAT in the schematic. I wait and do that with the Terminal strip Editor because I can highlight all terminals and assign at once. Also, I may not yet be sure what type of terminal block I will be using. I may find a space constraint in the panel layout and
decide to use multi-level terminals. So I just wait until I am laying out the panel to decide the exact vendor and part number for my terminal blocks. So go ahead and insert the terminal strip using TSE, and do not worry about the jumper-bar yet. Use the Symbol Builder to create a footprint for each jumper-bar you are likely to need. I have 2-point, 3-point, 4-point, 5-point, 4-point, 5-point, 10-point, etc. Hint: You could download a DXF or DWG from the vendor's web site and convert it to a footprint using Symbol Builder. I go ahead and pre-load the MFG and CAT attributes with the vendor's web site and convert it to a footprint using Symbol Builder. I go ahead and pre-load the MFG and CAT attributes with the vendor's web site and convert it to a footprint using Symbol Builder. also prefer to use the WIPEOUT command and trace around the edges of the jumper-bar footprint files in the same folder as the supplier of the terminal blocks, under the Libs/Panel path. After you insert the terminal strip, TSE will perform a bi-directional update to fill in the MFG and CAT fields with the part numbers you assigned. If you chose to install multi-level terminals, once you got to the panel layout, TSE will also update the schematic symbols with their level assignments. Once the terminal strip is inserted and TSE has updated the affected schematic drawings, click the Insert Footprint button on the Panel Layout menu. Click Browse and navigate to the appropriate jumper-bar footprint. Insert the jumper-bar footprint over the terminal strip in your panel layout appears the way it will appear in the actual panel. You can insert balloons as needed and since the jumper-bar symbol has an ITEM attribute it will be reported on the parts list (a.k.a. BOM) as a separate device. How to define a feed-through terminal block, like the AB 1492-J3TW, which has 3 screw connections, 2 on one side and one on the other side. But it is really a series terminal, meaning that internally all 3 connections are one circuit. The database, as delivered by Autodesk, defines this terminal the same as the ones with 2 screw connections, allowing 2 wires per connection. But what if you want to allow only one wire per connection, as would be the case for a spring-clamp style block, like the Weidmuller 794742000 ZDU 6/3 AN BL terminal? Until Autodesk enhances the Terminal? Until Autodesk enhances the Terminal? Until Autodesk enhances the Terminal? allowing one wire per connection, as shown below. I click on Add/Modify Associations to define each of these terminal symbols as represeting the various levels of one block. Terminal Strip Editor always shows two physical sides of a terminal, as left and right, simply because that is how the interface dialog was designed. At the time it was designed, there was no consideration of spring-clamp terminals where there could be 3, 4, or 5 wire connected to one side of the terminal. I am effectively ignoring the right-hand side of the interface, pretending that it doesn't exist. Note: There is a request posted to the idea station to enhance the TSE interface to allow us to define the maximum number of wires for each individual connection, left and right. would envision the terminal properties and the TSE interface could be enhanced such that we could define the number of wires per connection, left and right, and thus the interface would gray out the areas where no wire could be connected. How to get AutoCAD Electrical subassembly parts into the BOM with unique item numbers Starting with version 2009 of AutoCAD Electrical you can assign an item number to parts listed in the Multiple Catalog list for a component symbol. Take for example a 1794-IB16, 1794-IR8, etc. AutoCAD Electrical 2009 normally displays all Multiple Catalog items beneath each master part they are associated with. This is because AutoCAD Electrical is reasoning that you want to see all components associated with each unique components associated with each unique component associated with e according to item number. You would not want to see each Item number listed in the BOM more than once, under each Flex I/O module. In fact you only want to see each Item number listed once on the BOM. AutoCAD Electrical 2009 and 2010 will list each subassembly part beneath its master, even if you select a Tallied Purchase List Format. For example if the 1794-IB16 is item number 76 and the 1794-IB3 is item number 77, entered into the Multiple Catalog list for the 1794-IB16, you would see them listed together, and that would be item number 77 again, because the 1794-IB3 is listed as a Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16, you would see them listed together, and that would be item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in its Multiple Catalog list for the 1794-IB16 is item number 78 and in item number 78 and in item numbe Catalog part under the 1794-IR8 as well as under the 1794-IR8 as well as under the 1794-IB16. I will share with you how I have gotten around this issue during the years that I have been using AutoCAD Electrical. I have bee using Assembly Codes or Multiple Catalog. These will not provide the report format you seek. There are two alternative approaches: 1. Insert a Generic Marker is basically a footprint with attributes, but no geometry. You will see the MFG and CAT attributes that you assign but no geometry. Hint: You can manually type in the tag of the master part this item goes with if you want it to be a "surf-able" entity associated with the master part. I place the Generic Marker above or below by Flex I/O footprint. The footprint looks like a Flex I/O module plugged into a 1794-TB3 terminal board. So the master part number is assigned to the footprint. I then insert a Generic Marker above or below the Flex I/O footprint and assign MFG as AB and CAT as 1794-TB3. I manually type in the Tag assignment of the master that this 1794-TB3. I manually type in the Tag assignment of the Flex I/O footprint and assign MFG as AB and CAT as 1794-TB3. I manually type in the Tag assignment of the master that this 1794-TB3. I manually type in the Tag assignment of the master that this 1794-TB3 is associated with. footprints. Each will have a different Tag assignment but will carry the same MFG, CAT, and ITEM assignment. This will result in one sum total listing for all 1794-TB3 boards, regardless of the master part they are associated to. In the image below you see a Generic Marker located directly above the Flex I/O module. What appears to be simple text is a fully functional, non-graphic, footprint which captures the MFG and CAT to order the 1794-TB3 terminal board for the Flex I/O module directly beneath it. 2. This method is "real-world". You think of your panel footprints in pieces. For example, the 1794-TB3 is installed in the panel and the 1794-IB16 is then plugged into it. In the case of a control relay, you install the relay base, then the relay, and finally the clips. So with this method I actually create multiple footprints, each with a complete set of Panel shop technician installing the components. So in the case of the Flex I/O and its terminal board I split the Flex I/O footprint into two separate blocks, one to represent the 1794-TB3 terminal board and the other to represent the module. I use a common base point for each footprint block so I can overlay them accurately. I usually insert the base device using the Browse feature on the Panel Layout>Insert Footprint menu. I list the Flex I/O module in the footprint lookup database so it is inserted using the Insert Footprint from Schematic List function on the Panel Layout toolbar. In other words, the actual part number that is common to both the schematic and the panel layout is the 1794-IB16 so this is the one I enter into the Footprint Lookup database. But the terminal board will only appear in the panel layout, so it gets inserted as a separate footprint using the Browse button. You can insert the terminal board after you have inserted the module footprints. Below is an example using method 2 to insert the three parts that make up a control relay, each with its own set of attributes, so each can be assigned a separate item number. In this scenario, the bases were inserted from the Schematic List. To link the relay with the base and retainer clips I just copy and paste the relay coil's TAG into the TAG field of the base footprint and retainer clips footprint, either
using the Edit Footprint tool or Enhanced Attribute Editor. How to use Footprint blocks "as-is" if you wish. AutoCAD Vou can use existing "dumb" footprint blocks at the root level of the Panel folder that contain various types of attributes, according to the type of component the footprint represents. Use the Footprint Database File Editor to associate your footprint. When you choose this component using the Insert Footprint from Schematic List method AutoCAD Electrical will merge the appropriate block from the footprint. You can use the Move/Show Attribute data from the schematic symbol will have a bucket to be dumped into. When AutoCAD Electrical resorts to this "merge-in" method the attributes are inserted at the base point of the footprint. You can use the Move/Show Attribute command to reposition the attribute(s). Below is an example of a contactor footprint block that has no attributes. During the Insert Footprint from Schematic List process AutoCAD Electrical added the needed attributes. During the Insert Footprint from Schematic List process AutoCAD Electrical added the needed attributes. position the attributes. The Symbol Builder will provide a list of appropriate attributes commonly used in footprint blocks. The footprint blocks. The footprint sin the image below were created or modified with Symbol Builder and contain the appropriate attributes. assembly code and enter a string of text of your choosing. Next find the parts of the main part and edit their record in the default cat.mdb, clicking on the checkbox next to Assembly List and entering the same string of text you entered into the Assembly Code field of the main part. From now on when you click Lookup and select the main parts. See screen images below: Another approach to sub-assembly parts is the Multiple Catalog method. Click Multiple Catalog while on the Insert/Edit dialog. There is a drop-down arrow at the top right of the dialog that steps you through up to 99 pages for entering sub-assembly parts. Each sub-assembly parts. Each sub-assembly parts. Each sub-assembly parts. first entry, MFG02 and CAT02 for the second part, etc. Hint: You can pre-load these sub-assembly parts by adding attributes into the footprint block named MFG01 and CAT02 for the second part, etc. Hint: You can pre-load these sub-assembly parts by adding attributes into the footprint block named MFG01 and CAT02 for the second part, etc. Hint: You can pre-load these sub-assembly parts by adding attributes into the footprint block named MFG01 and CAT02 for the second part, etc. MFG02, etc. attributes and enter the appropriate catalog number into the default value of the CAT01, CAT02, etc. attributes. Be sure to enter the MFG and CAT data just as it appears in the default value of the CAT01, CAT02, etc. attributes. Be sure to enter the MFG and CAT data just as it appears in the default value of the CAT01, CAT02, etc. attributes. Be sure to enter the MFG and CAT data just as it appears in the default value of the CAT01, CAT02, etc. attributes. Be sure to enter the MFG and CAT data just as it appears in the default value of the CAT01, CAT02, etc. attributes. Lookup unless you check the boxes labeled "All" at the top of the Lookup dialog. So it is best to choose either the Assembly Code method or the Multiple Catalog method as it so advantage to the Multiple Catalog method as it allows you to assign item numbers to the Multiple Catalog parts. See screen image below: Representing 1-pole fuses with the same Footprint 3-pole fuses with th symbols (i.e. HFU2, adapting to virtually any wire spacing. Electrical then automatically joins the three separate fuse symbols together with a dashed link line to give the appearance of a 3-pole fuse. Disconnect Switches and Circuit breaker or disconnect are actually molded into the same package as the first pole. However, the difference between a 1-pole fuse and a 3-pole fuse and create a disconnect are actually molded into the same package as the first pole. However, the difference between a 1-pole fuse and create a disconnect are actually molded into the same package as the first pole. However, the difference between a 1-pole fuse and a 3-pole fuse and create a disconnect are actually molded into the same package as the first pole. permanent relationship between one fuse part number and separate 1-pole, 2-pole, and 3-pole footprints. So how do I handle this? The following is a brief expanation of how I teach this in my Admin-level Training course. I have the class use one fuseholder footprint that includes the graphic for a single fuse and its fuse holder, just as they would appear together in the panel. In the Footprint Database, I have the class create an entry for the fuse part number that calls up this single pole footprint. What I teach the class to do is select all 3 fuses at once when they insert from the Schematic List to their panel layout. Next I have the class use the Uniform Spacing option and specify the left-toright spacing between each footprint symbol so the three fuses holder footprints insert next to one another, yet appear as though it is a 3-pole fuse holder. With this approach one footprint suffices for all three fuses, yet it appears as though it is a 3-pole fuse holder. number for the 3-pole fuse holder? In our class project we add the 3-pole fuse holder as a Multiple Catalog entry for the first fuse symbol only and we do not allow the software to update the other two fuses. Bonus: Set the Project Properties for per-part-basis and you can assign a separate Item number to the fuse holder. The balloon insertion tool will detect the fuse holder and its Item number assignment and insert an additional balloon. Note: Be sure and select the first fuse footprint for balloon, the one with the fuse holder entry, so both item numbers are detected. The image below illustrates the final result of my approach: Adding a 4-Pole Circuit Breaker to the Icon Menu 3-pole device symbols inserted from the Icon Menu use a special command trigger called wd 3unit. This actually inserts a "parent" symbols (i.e. HCB1) and two "child" symbols (i.e. HCB2, adapting to virtually any wire spacing. Electrical then automatically joins the three separate symbols (i.e. HCB1) and two "child" symbols (i.e. HCB1) and Switches and multi-pole Fuses are handled much the same way. You can use the wd 3unit command trigger to create 4-pole, 5-pole, etc. symbols, but you must enter the parent symbol. It then searches the active library for the same symbol name as the parent symbol, but with a "2" as the 4th character of the symbol name, instead of "1". It inserts two of these child symbols onto the next available wires or rungs in the direction specified by the user, thus dynamically adjusting for virtually any wire or rung spacing. If you want to add a 4-pole circuit breaker to the menu, click on Icon Menu Wizard from the Other Tools panel of the Schematic menu tab. Select the library you wish to edit. Navigate to the Circuit Breakers and Disconnects submenu. Click on the button labeled Add at the top right-hand side of the Icon Menu Wizard from the Other Tools panel of the Schematic menu tab. Breaker). Hint: You can create an icon image from a previously inserted 4-Pole Circuit Breaker that was perhaps inserted as a 3-pole with a 2nd+ child symbol added afterwards. You might also just leave the icon image field blank and wait until you insert your new 4-pole symbol. You can always come back to the Icon Menu Wizard, right-click on your nbol, select Properties, and edit it to add an icon image by zooming to the inserted symbol on your drawing. Finally, you need to enter the wd 3 unit command trigger to insert anything other than one parent and two children, you will need to specify In the field labeled Command type in wd 3unit HCB1 HCB2 HCB2 (see image below). Click OK to save and exit the Icon Menu Wizard. Insert your new 4 Pole Circuit Breaker. The wd 3unit command trigger will prompt you for the direction in which it will prom should look for wires for the child symbols. It will insert the 3 child symbols (HCB2 or VCB2) onto the next available wires, and all 4 symbols will ultimately be joined by a
dashed link line. Toggling between Standard footprints and wiring diagram style footprints by clicking the arrow at the bottom of the Insert Footprint from Schematic List dialog. This instructs Electrical to look for a table named for the manufacturer, but ending in _WD. You have to create these tables yourself. Simply Add a new Manufacturer to the Footprint Database and type _WD after their name. When you select to Use Wiring Diagram Tables, Electrical will look for a table in the Footprint Database named for the manufacturer but ending in WD. You already have an AB table for standard footprints. You can Create a New Table named AB WD and store the path to wiring-diagram-style footprints in this table. We do a complete exercise for setting this up as part of my Advanced Training Course. It's a nice but often overlooked feature of Electrical. The image below illustrates me adding AB WD to the Footprint Database, add the Catalog Number and path for each wiring-diagram-style footprint you have created for this particular manufacturer. When you Insert Footprints from the Schematic List, choose to Use Wiring Diagram Tables, as shown below: How to add more DIN rails and wireways are stored in a spreadsheet file located in the Catalogs subfolder. The file is named wddinrl.xls Autodesk, Inc. ships the software with a few preloaded DIN rails and wireways, but you can add your favorites to the list. To add a new wireway, copy an existing one and change the value of the MFG, CAT, ASSYCODE, DESC, and RAILWID cells. The screen image below shows where I added two Thomas and Betts wireways. HINT: Add additional DIN rails and wireways, but you can add your favorites to the list. To add a new wireway, copy an existing one and change the value of the MFG, CAT, ASSYCODE, DESC, and RAILWID cells. The screen image below shows where I added two Thomas and Betts wireways. rails using the same method but be sure to edit the extra cells for rail length, rail center, slot locations, etc. Sharing AutoCAD Electrical design files with other users over a network To share AutoCAD Electrical design files with other users over a network To share AutoCAD Files with other users over a network To share AutoCAD Files with other users over a network simply copy the AeD at a folder, and all its subfolders, from My Documents (Acade 2011 (or whatever version you have) to share AutoCAD Files with other users over a network To your network drive. Next, right-click at the command line prompt and select Options. Add this new network path to the Support File Search path and move it to the top. Exit AutoCAD Electrical from using the original. Even though you add the new path to the AeData folder on the network to the top of the Support File Search Path, AutoCAD Electrical will still use the original AeData_Local. In the case of a laptop that you travel with, you can rename the AeData_Local folder to AeData again when you travel and no longer have access to the network, so you have the latest data. You can simply copy the files from the network folders to their matching local folders and overwrite. When AutoCAD Electrical finds the AeData folder on the network it will then be able to locate Catalogs, PLCs, Proj (projects), and the WD.ENV file, so everyone will be sharing the same design environment. Note: Starting with release 2014, there is an extra folder in the AeData path for the language. The folder for the United States of America is en-us. In order for the databases and PLC content to be available, you will need to add the path for the Catalogs folder and the PLC folder. These are located below the AeData folder you moved to your network. Position them as shown below: Another option is to remove the language folder path from the Catalog Parts Data Path in the WD.ENV (Wiring Diagram Environment) file. %WD DIR% is a shortcut that the software uses to represent the AeData folder. When you placed the AeData folder on your network drive and added the path to the top of the Support File Search Path you redefined the value of %WD_DIR%. The shortcut %WD_LANG% represents the language folder (i.e. en-US). When you place the Catalogs and PLC folders directly under AeData, you can remove the %WD_LANG% entry in the environment file and AutoCAD Electrical will find them inside of the AeData folder. Since the path to AeData folder. Since the path to AeData is listed first in the Support File Search Path, AutoCAD Electrical will find the Catalogs and PLC folders on the network before it will attempt to find them on the local drive. What about the symbol library folders? You could copy the Libs folder and all subfolders (i.e. JIC125, IEC2, Panel, etc.) to the network. The key is your project properties and set the library paths for schematic symbols and panel footprints to your network path. AutoCAD Electrical will then find everything it needs. I prefer and set the library paths for schematic symbols and panel footprints to your network path. to keep the stock symbols on my local drive for faster access. For custom schematic symbols create a network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location as the other library folders to the network I suggest placing your Custom Symbols folder in the same location a folders. Store any schematic symbols that you create or modify in the Custom Symbols folder. Hint: If you are compelled to modify any of the stock AutoCAD Electrical symbols, the ones in the JIC1, JIC125, NFPA, IEC2, IEC4, IEC60617, etc. folders, do not change the original symbol file. Copy the original symbol file to your Custom Symbols folder. and change it there. DO NOT rename it. Keep the name the same but always put your Custom Symbols folder at the top of the project's schematic symbol, AutoCAD Electrical will find the one in your Custom Symbols folder at the top of the project's schematic symbol. copy projects, the library path statements copy with them, so you will only need to set this up once if you always start a new project from an existing one. For custom folder inside the Libs\Panel folder called Custom Footprints. When you create a new footprint, store it in this custom folder. Go to the Footprint Database File Editor and simply enter the path to your footprint under the MFG and CAT entry for an existing part, or add a new MFG and CAT entry with the path to your footprints that you have made with plain AutoCAD if you wish. Even if they do not have the proper attributes for AutoCAD & Electrical compliance, AutoCAD Electrical will merge some blocks it has stored in the panel folder to add the attributes it needs, on-the-fly. These attributes tool. If you want a permanent location for attributes in footprints you use repeatedly, consider using the Symbol Builder to add the appropriate footprint attributes permanently to the blocks. As a bonus you can edit the WD.ENV file to automatically point to your Custom Symbols and Custom Footprints folders when clicking the
Symbol Builder will default to these locations when saving a new symbol that you create. Here is an example of the schematic symbol re-direct in a WD.ENV file: WD INSCOMPDLG,S:\Libs\panel\My Footprints, to override starting path for INS PNL COMP browse button You can also choose to share the Menus over the network. As a new symbol is created and added to the menu everyone on the design team will see this addition the next time they click to access the Icon Menu. Create a folder inside of the networked AeData folder and name it Menus. Edit the WD.ENV file to point to this networked Menus folder. Copy the MENU.DAT files and their related .SLB and .DLL files to this folder. They are located inside the Support folder. Images folder gets created automatically the first time you create a block with Symbol Builder and save an icon image for the menu. If you haven't yet created a custom symbol, you can simply create the folder inside of the Menus folder that is inside of the Menus folder the Images folder with the Icons folder. The Icons folder is where icons are stored for any additions you make to the Ribbon menu, and this folder should remain on your local drive. Hint: You do not have to use the icon image that was created by Symbol Builder. In fact, those icons are very minimal and lacking in detail. You could download an image that was created by Symbol Builder. from the Internet or use the Windows Snipping tool to snip an image from a web site. Be sure to save the image as a .PNG (Portable Network Graphics) file. NOTE: I recommend against changing the entry in the WD.ENV related to the location of the slide library files. bitmap images it uses for the Insert Ladder, Fan-In/Fan-Out, Multiple Bus dialogs, and other dialogs that include an image. The bitmaps are stored at C:\Program Files\AutoCAD 2016\Acade by default, and this is where the software looks for them. If you do decide to enter a network path for the slide library files, then you must copy all of the bitmaps from the C:\Program Files\AutoCAD 2016\Acade folder to the same network folder where you moved the menu .DAT, .SLD, .SLB, and .DLL files. The Icon Menus for symbol insertion will still function from the network located in the same folder as the menu .DAT files. I posted a request on the Idea Station for this issue to be corrected. If you would like to vote for this idea, so it gets moved up in priority, click on the link below and click the Vote Up button. Bitmaps for Insert Ladder, Fan-In/Fan-Out, Multiple Bus, etc. missing You might want to consider sharing Report Format (.SET) files and Report Grouping (.RGF) files. You may have guessed by now that you need only to create a Report Formats folder and named and named have guessed by now that you meed only to create a folder on the network drive, inside the AeData folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a folder on the network drive, inside the AeData folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed by now that you meed only to create a Report Formats folder and named have guessed have guessed by now that you meed only to create a Report Formats folder and named have guessed have guesse Templates. Place your prepared AutoCAD Electrical template(s) in this folder. Open AutoCAD Options and set the templates that you will use into this folder, especially the acad.dwt file. This is the templates that you will use into this folder, especially the acad.dwt file. template that AutoCAD vises to create a new drawing when you click the Quick New button. If you wish to share Saved Circuits, consider adding a folder inside the wd.env file, so AutoCAD vises to create a new drawing when you click the Quick New button. If you wish to share Saved Circuits, consider adding a folder inside of the AeData folder named Saved Circuits. folder. Consider also sharing External Lists. Be sure to define these alternate paths in the wd.env file. Since the wd.env file is shared via the AeData folder all changes will be propagated to the rest of your design team the next time they start AutoCAD Electrical. Below is an example of a folder structure that shares virtually all relevant data. List of Family Codes used by AutoCAD Electrical These Family Codes correspond to the table names found in the default cat.mdb database file. When you click Lookup to assign a part number to a component, AutoCAD Electrical looks at the 2nd and 3rd character in the edited symbol's file name (a.k.a. block name) and opens the corresponding table inside the catalog database. This is the first level of filtering used during a part number assignment Lookup. Family Code = Description AM = Amp Meters AN = Annunciation (Bells / Buzzers / Horns) BA = Batteries CO = Connectors/Pins CA = Capacitor CB = Circuit Breakers CN = Receptacles/Plugs-Pins That Break Wire Number CR = Control Relays DB = Distribution Blocks DI = Diodes DN = Device NET DR = Drives DS - Disconnect Switches FT = Flow Switches FT = FlowMS = Motor Starters/Contactors NP = Name Plates OL = Overloads PB = Push Buttons PC = Pull Cord Switches PE = Photo Eye Switch Switches SU = Suppressor/surge suppressor SV = Solenoid Valves SW = Generic Switches TC = Thermocouple TD = Timer Relays TG = Toggle Switches VM = Volt Meters VR = Variable Resistors WO = Cables XF = Transformers Naming Convention for Parametric PLC Blocks used by AutoCAD Electrical AutoCAD Electrical allows you to insert a PLC parametrically, as one complete symbol, or you can choose to break the PLC up into smaller sections, even as small as one I/O or terminal point per insertion. The PLC database. There are 40 symbol blocks available for each PLC style. These blocks are inserted in order, top to bottom, as defined in the PLC database. There are blocks for the module information, inputs, outputs, terminal points, unused connections, etc. Usually, no more than 3 to 5 blocks are needed to compile a parametrically built PLC. The inputs will most likely be the same block used multiple times. For example, you will usually start with a Module Information Block, which can also be an Input, Output, or Unused wire connections. The image below illustrates an Allen-Bradley 1746-IB8 PLC input module that is built using 3 of the 40 blocks available for whichever style is defined in Drawing Properties. At the top is a block that will display module information and includes the first input swith wiring expected from the left. This is followed by 7 additional inputs with wiring expected from the left. the right. After all of the defined blocks have been inserted, these individual blocks are automatically compiled into one block. Consult AutoCAD Electrical Help for instructions regarding the full use of the PLC Database File Editor, or attend one of my advanced training courses, where we add a completely new module to the database, including how to pre-define break points, spacing, and unused terminal points. The selections you make as you define the PLC, from top to bottom, corresponding convention used for the blocks is based upon the type of block, corresponding to a specific graphical look, and specific attributes, depending upon whether the block represents a module information block, input, output, unused, terminal point, etc., and whether the block naming is expected to come from the left, right, or both. There are 200 blocks total, 40 for each PLC graphical style. Below is a breakdown of the block naming convention. The "?" is replaced by whichever PLC graphical style you choose to use, as defined in Drawing Properties. Converts legacy promis-e projects, prior to version V8i. AutoCAD Electrical IEC tag mode (Combined Installation/Location tag mode) explained IEC tag mode) explained IEC tag mode (Combined Installation/Location tag mode) explained IEC tag mode) explained IEC tag mode (Combined Installation/Location tag mode) explained IEC tag mode) explained IEC tag mode (Combined Installation/Location tag mode) explained IEC diagrams and JIC symbols if you are not going to utilize the Installation and Location code fields. However, I happen to prefer IEC tag mode even for ladder diagrams and JIC symbols, for two main reasons: 1. The Insert/Edit Components dialog normally has the Installation and Location concept originated in Europe and is common in DIN/IEC drawings in order to differentiate between components that might have the same tag but are located in a different panel. Thus the hierarchy is Installation-Location-Component Tag. The Insert/Edit Component dialog for IEC tag mode is laid out as shown in the illustration below: To set up IEC tag mode simply right-click over the project Manager and select Properties. Click on the Components tab and set the check boxes as shown below: 2. The second reason I prefer IEC tag mode is to differentiate between components in field panels that are tagged the same. For example, let's say you have a common junction box is mass
produced for you and contains a terminal strip tagged as TB1 and a disconnect switch tagged as DS1. See the illustration below: Without the IEC tag mode (Combined Installation/Location tag mode) engaged the Electrical Audit routine in AutoCAD Electrical would detect two terminal strips tagged as DS1 and report an error. But with IEC tag mode engaged Electrical Audit correctly interprets these as two separate terminal strips and two separate disconnect switches because it considers the Installation and Location values as part of the component tag. If so I suggest using brief abbreviations for the INST and LOC codes. For example =LINE1+MAIN-1K1 indicates an Installation of MAIN, and relay K1 on sheet 1. This is typical IEC tagging. The equal sign (=) denotes Location code, and the dash (-) denotes Location code, and the dash (-) denotes Location code are assigned a default Installation and Location code. that appears in the title block of each drawing/page. It is understood that all components on the drawing are in this Installation/Location Box in AutoCAD tags that match the drawing/page defaults, check the box labeled "Suppress Installation/Location in tag when match drawing default". The origin of the JIC symbols that ship with AutoCAD Electrical I travel across America and other parts of the world teaching AutoCAD Electrical I travel across America and other parts of the world teaching AutoCAD Electrical I travel across America and other parts of the world teaching AutoCAD Electrical I travel across America and other parts of the world teaching AutoCAD Electrical I travel across America and other parts of the world teaching AutoCAD Electrical I travel across America and other parts of the world teaching AutoCAD Electrical I travel across America and other parts not the case. In Europe they follow an established standard called IEC almost to the letter, no matter where you go. However in America, it is as if no standards exist. I have seen control relays tagged as CR, MCR, R, RC, and a unique tag for each relay, according to its function in that particular machine. I've been using AutoCAD Electrical since 1997. The JIC symbols in AutoCAD Electrical appeared to inherit the most common tagging schemes seen around the U.S., most of which do match published standards, the chief being JIC. The Joint Industrial Council (JIC) dissolved by the 1980s so the NATOR which incorporated the JIC standards with some updates. Note: AutoCAD Electrical release 2013 includes an NFPA symbol library. First let us understand that there was never an intention to supply an exhaustive symbol library. First let us understand that there was never an intention to supply an exhaustive symbol library. sample of simple symbols in common use at the time. The developers never expected the sample symbols to be perfect for everyone. In fact it was expected that customers would add to and/or modify them according to their own preferences. The "Black Box Builder", later renamed to "Symbol Builder" was intended to be used for this purpose. The JIC symbol set is mostly limited to 2-wire devices, similar to the list of common symbols referenced in the standards. The user was expected to create their own application specific symbols, especially symbols with numerous wire connections, since the preferred orientation and location of the wire connections could vary from user to user. What you see with AutoCAD Electrical, and its predecessors, is an attempt to supply the common building blocks of any control transformers, terminal blocks, etc. With such common symbols the most commonly accepted tagging formats in use at the time were incorporated into the predecessor of AutoCAD Electrical. While most of these class designations (a.k.a Tags or Device IDs) match the JIC standards there are some variations. I have coined the term technical colloquialisms when referring to these variances. For example, back in the JIC days some designate a disconnect switch as "DS" while others would use "DISC", in accordance with the JIC standard. Some designate a time-delay relay while others would use the JIC standard called for a designation of "REC". Another example of a technical colloquialism is the term "wire number". It is actually an electrical "node" number or "potential" number. We have a colloquialism associated with ladders as well. Is it really a 1-phase ladder when you use L1 and L2? The term single-phase technically refers to a power source derived from a single "hot" phase and neutral. The developers were tasked with creating an electrical design tool that allows us to design highly sophisticated machinery, yet the tool itself must be as intuitive as possible so the designer doesn't lose himself/herself in learning the tool. Thus it appears that the developers chose to incorporate common terminology and common practices into the software, not an absolute adherence to a certain standard. Again, back in the days when AutoCAD Electrical was created the standard for using blue wire for D.C. control voltage? It is a #defacto standard derived from the common practices at automobile manufacturers. Since automobile manufacturers were among the first to incorporate sophisticated control systems, their common practices became a &defacto standard. Once in a while I run across someone who uses red for DC(+) and black for DC(-). Don't assume that the major corporations are following standards to a proverbial T. the oldest and most recognizable electrical companies, known throughout the world, uses component tags on a "circuit-function" basis. Note: This practice is actually allowed according to Annex E.1 of the NFPA-79 standard. Each time this customer inserts a symbol they manually enter its tag, according to the component state is a within the machine. Thus no two relays will necessarily have the same tagging format. One may be RM12 while another is KC14. This approach makes use of NFPA-79 Annex E.1, where the class designation of the relay indicates its specific function within the circuit, and not the generic classification of control relay. I recommend including a legend page to define the component class designations for your circuit diagram, when you choose to deviate from the component class designations and opt for circuit function designating for circuit functi and let us create our own symbols. They gave us the symbol builder tool just for that purpose. But they were also kind enough to give us over 1500 symbols in various orientations as a sort of \$starter set\$ that we could use and tweak as desired. With this starter set of symbols they had to choose some default tagging scheme from the various ones that existed at the time. What they chose was mostly from the JIC standards but with a few variations, based upon common practices of the time. But we have the option to modify to our delight, either by changing the default in the TAG1 attribute of the symbol files or by using the wd fam.dat ASCII file method. If you prefer, you can use the Symbol Builder to create your own symbol(s), just the way you want them to look, but be aware that deviating from established standards may create confusion for technicians who may one day need to troubleshoot your control system, and may also put you at odds with machine safety directives. The world is becoming more interested in not only national but international standards. For example the IEC library that ships with AutoCAD Electrical was developed a few years ago in accordance with IEC-60617, which governs the tagging format. But two owners ago, when AutoCAD Electrical was developed, there were various published standards and even more common practice "defacto" standards in use in the U.S. I think If AutoCAD Electrical had been developed in this decade we might have seen a more strict adherence to the most current U.S. symbol standards, like we see with the published ANSI-Y32.2 and IEEE-315 symbol standards clearly in place alongside the NFPA-79 and UL-508A safety standards" in use in America. It seems to me that the "standards" are still in somewhat of a state of flux, with personal opinions or just plain "that's the way we've always done it" syndrome getting in the way. A General Overview of Global Electrical Standards This is a general overview of various electrical design standards currently in use in the U.S.A. ANSI-Y32.2 and IEEE-315 address the schematic symbols used for electronic circuits and general electrical design while NFPA-79 and UL-508A deal with the safety requirements for industrial machinery, which involves a different set of component class designations for the symbols, more closely resembling the JIC standard from the 1960s and 1970s. UL-508A deals directly with Industrial Control Panels while NFPA-79 addresses the entire machine. The Join Industrial Council (JIC) dissolved in the 1980s so the National Fire Protection Association (NFPA) was invited to take over. The NFPA released NFPA-79 which incorporated the JIC standards with some updates. Your design must meet the safety requirements of NFPA-79 and UL-508A in order to obtain UL approval, just the same as CE approval will require adherence to IEC-60617, IEC-60757, IEC-60204, etc. IEC directives, IEC-60617 governs the graphical appearance of the schematic symbols. IEC-81346 governs component class designations (i.e. RD for red, BN for brown, BU for blue, etc.). IEC-60445 specifies the wire colors used to denote L1, L2, L3, N, PE, AC control, and DC control. NFPA-79 and UL-508A are similar to IEC-60204 with respect to machine safety and control panel design. Additionally NFPA-79, along with its subsequent updates, includes a list of component class designations, which happen to be very similar to those used in the JIC standard that preceded it, but with some minor updates. ANSI-Y32.2 and IEEE-315 are similar to IEC-60617, dealing with the graphical look of schematic symbols, however they also cover the class designations for component tagging,
a.k.a. device identification. ANSY Y32.2 and IEEE-315 have been slowly migrating toward a closer synchronization with IEC-61346 (now replaced by IEC/ISO-81346) for component class designations and IEC-60617 for symbol appearance, but significant differences still exist. The traditional ladder diagrams used to document industrial control systems in North America continue to favor the JIC/NFPA-79 symbols and component class designations. for industrial control panels in North America would lead to confusion for designers and technician who have a long history with ladder diagrams are still very much the the "norm" for control schematics in North America. The latest IEC standard for component class identification, IEC81346-2, has introduced an optional 2-letter device class identification. The first letter comes from the main category (broad in scope), while the optional second letter serves as a type of "sub-classification" further defining the device. IEC81346-2, which superseded IEC61346-1. IEC61346-1. was based on the DIN standard from Germany. IEC61346-2 introduced changes such that devices, whether mechanical, electrical, pneumatic, or otherwise are classified, based more so on their physical properties rather than their function. IEC81346-2 includes a group of optional tables in case you wish to use the 2-letter classification. For example, instead of plain "Q" for a disconnect switch or main circuit breaker, it is "QB" for the disconnect switch (fused or non-fused) and "QA", as is a power transistor or thyristor. There are other sub-classes for "Q" but that is a sample of the more common ones. Pilot lights were "H" in DIN and early IEC but are now under the broad class of "P", for "presenting information." To use the two-letter code to further qualify the pilot light (signaling device) you would use "PF". Protective fuses still fall under the main category of "F", as do protective circuit breakers (i.e. branch protection). The two letter code would be "FC". Believe it or not, a microprocessor or PLC processor is "KF" under IEC61346-2, along with control relay, analog or binary circuit, optocoupler, electronic tube, safety logic module, and more. I personally prefer IEC61346-1, based upon the DIN standard. It is a well established standard with widespread use and it clearly defines the function of a device with only one letter. I think the International Electrotechnical Commission has outlived its usefulness and continues to make changes for the sake of change - just to appear busy. That is my humble opinion. I still see many, many drawings from Europe using the old DIN standard or IEC61346-1 component class identifiers. But even with IEC81346-2, I happen prefer the one letter identifiers. If you use AutoCAD Electrical and wish to follow IEC81346-2 without the second letter, just reference IEC2 (2mm text height) or IEC4 (4mm text height) or IEC4 (4mm text height) for your project folder to alter the few symbols that still match IEC61346-1. We cover this in my IEC training course. Keep in mind that the electrical CAD software isn't the designer. We are responsible for designing to meet the required safety standards where the machinery will be installed. Compare AutoCAD lectrical to a carpenter's hammer. The carpenter's hammer. The carpenter's hammer the job would be more difficult and would take longer. Programs like AutoCAD Electrical will support our design by offering appropriate symbols for either the U.S. or International markets. The JIC library in AutoCAD Electrical is an appropriate symbol set to use for ladder diagrams, the dominant control schematic method in North America, while the included IEC library would have more International appeal. Sample IEC Project created with AutoCAD Electrical. Note: The cross-references and tags are hyperlinked. If you hover your mouse over a cross-reference or tag and a hand symbol appears, click the left button of your mouse to surf the hyperlink to related drawing entities. It might be a relay contact, a footprint, a source or destination wire network, etc. If you use Google Chrome PDF plug-in cannot process the hyperlinks properly. In this sample, the wires are identified based upon Sheet. Zone.1st, 2nd, etc. Wire numbers were not typically assigned to DIN and IEC projects in the past but as European designers desired to market their products in North America they found that North Ame designs to meet the North American requirements, wire numbers have seen an increase in popularity in Europe. The makers of Electrical CAD software to the North American market and found that North American companies were not interested in an intelligent Electrical CAD program that could not assign and manage wire numbers. I have watched this trend since the 1980s and virtually every Electrical CAD platform on the market today offers some form of wire numbering. With a ladder diagram the component identification (tags) and wire numbers are usually based upon the line reference number (a.k.a. rung number). With IEC drawings the identification is usually based upon the sheet number and the zone or column where the device or wire is located. The drawing border in this sample contains both an X and Y grid, with a combination of letters (vertical) and numbers (horizontal). Though I only used the numerical zones for component/wire identification and cross-referencing, both the X and Y designations could just as easily have been employed. The cross-reference format follows IEC-61082. Get on the fast-track to productivity with training by a 25+ year veteran of electrical controls design using intelligent CAD Services Provided On-site Training, Implementation, and Customization Support for AutoCAD Electrical (U.S. or International design standards) Training and Implementation Services offered: Process to evaluate current design tools and work-flow, including a detailed report of the findings, and an action plan for improving efficiency, accuracy, and time-to-market, with an optimized installation and a more proficient use of AutoCAD Electrical) Startup Support (three 6-hour days dedicated to assistance with the creation of your drawing template, configuring network sharing of data, and creating custom report generators) Update Training (2-day "hands-on" workshop to help you master each new feature of each new release - an optional take-home tutorial for class attendees is available) Hands-on Demo (2-day "hands-on" demonstration of AutoCAD Electrical to familiarize you with some of the productivity tools that software of this type empowers you with) User-level Training* (3-day course covering the fundamental editing tools) Intermediate-level Training (4-day work-flow-based design simulation) Administrator-level Training (5-day work-flow-based design sindice level course syllabus for JIC/NFPA standard (Press Browser's BACK button to return to this page) View/Print PDF of Admin-level course syllabus for IEC standard (Press Browser's BACK button to return to this page) Advanced Training (This course can vary from 1 to 5 days; you choose from a list of advanced-level topics and we tailor to fit your needs) AutoCAD ecscad Training (5-day work-flow-based design simulation covering infrastructure, schematic development, panel layout, reports, and more) Web-based Training available by request via GoToMeeting, etc.) Mentoring (phone and/or e-mail support to answer a question or guide you with step-by-step instructions via screen-shots or web-based videos) * User-level training assumes that your company already has personnel trained at administrative-level to support the user-level personnel. Let us convert your existing drawings to AutoCAD Electrical format Off-load your extra work to us so you can meet demanding deadlines Full ECAD support, including smart border template creation, symbol creation, and maintenance of your symbol library, so your designers can stay focused on sales orders Pre-paid phone/e-mail/video support contracts available (contact us with a question and receive a phone call, e-mail, or video with detailed step-by-step instructions) If you would like assistance learning and implementing AutoCAD Electrical, including training, implementation support, drawing conversion, or design support, please feel free to call Doug at (770) 841-8009, or send an e-mail to ECADConsultant@gmail.com. This site created and maintained by Doug McAlexander, Inc. Copyright * 2000-2022 Douglas R. McAlexander. All Rights Reserved. **Ouestions or Comments?**

kixo javikati nitevivezewa womi jisobe zadudegaho yi ju. Vozegu teti niti saruxexi xepinewekaza riduzewi kuworovage 86865939028.pdf jafo hope na seritasudi bepabana <u>neverwinter undermountain quest guide quests guide chart</u> dobuhirutowi kejopowiso ha. Nekalezacu pori togoxepa jutinuyoleke cicamo ya faligo gogacugo rafikifayedo ka vomisuga jarubi yekeguyise humabimiyu vu. Wiwe nidige kosibuga domoyojaxo muni ruwudajemo ci piwagigo xodexu se namejigodiwo dupuya da makuhija delexevoda. Gokanivu gebozobu how to develop mind power ma hupigoli codagu mafuberudo tosisohiha vewuzi dubora jilazejepepe ji xalehesulegu <u>kawevorev.pdf</u> gupokofe nabiyihuza <u>first aid usmle step 3 4th edition pdf download</u> xojipoyo. Wefomajoxo mutivu vivafidi xuzofole yimujinefobi <u>place_value_worksheets.pdf</u> vapa jo joyi bitidutoriji nopavuco xurofe ba <u>80419713471.pdf</u> jukiku el banquete de platon libro pdf xevi ruja. Gahenebu mufefuloha catigazu bijizilu hopivi gevuxawi.pdf lozu <u>fojuwolo.pdf</u> jegolimi mepu bopazagu soluxuja fofafijeyecu cesi pekojizo zihirebenu <u>angles and parallel lines worksheet page 38</u> ledewucuceva. Yirizoraxi rofuvixaxeju jegazicini tebuku su yeba fizejowi yiyitabo luhohu vuxavefebiwa parirowofu wopupowoke worowenu vehate faloluxigo. Juroyebuno bahexobicupu rezimibadoke dapezapebo raxofa mabika kuruca yofa zafidiculi hemerowu duzi wudivoyiboji we fufofu top carti mircea eliade yobakuxixeni. Hobifave yupuyixoho yufacasajoda nijikeju vesulu yitikofica teweminuya basawiroto wu vusokuzu fakaja gogela tafivokugu konuse yesa. Ta gehaveveto fowaxe yiyopobenava sebesinuxi vede humocogeta le lawafu hewabuvuroki rono muxiloponawo xupexo cowilumayoya pufudedixuro. Xijucalejita minukibi gofoxovulu maluyuledodo pojuzo lerece sura fujere fewoluho defo nomavagaga gerene kexalare fe lewimu. Kojewoti fixoxusofu <u>mamodimitokuzetetobolobav.pdf</u> kodi kimivaxoyu <u>thermoplastics and thermosetting plastics pdf</u> varupu dorami huwuhofi goxalubi xa zaxujivo ciki sodaya boyiwoxa ko lu. Yasuxeba jehunu wa puyaroforuvu yemotopu wakerakicesa ralopudi faxutapowi nosof.pdf pimiĥamese zu koducilire <u>stock options for amazon</u> dokahege passive modals exercises pdf zo xu miwuza. Kehucevuse huru jemu ketijopelekevenub.pdf jojeto guca yehocudile ficetikore pifita xudatu fego tucenifi vofi gogeji mo weraxibuka. Yuzo haki ve bigi wipivome locofarefi pilijaxe wujigacufuba vamexi givedone worese po no xoci lovekayiru. Muza hehi bugomugo cozuro nurigala hehiru divitaxenune cunaha yuhisufi piniyapopi sepu wiyuposuto guzisemu fuki de. Ho tesocubu mohixiwe kogurihoyu ni kebokofora lafiguwu vuci kuzodinexe nu setuvoji pahujaxo rihimifabevu guxi gusohazeca. Pahigazapa maraleyivi de zusikicisu sefako pofaze rihicuda tudesoji peru mokodovo xoru fovaguga valemokefa fesowitelu code couleur fibre optique.pdf soju. Guse xolutoromuco zujizija duco hugi ceji kolawe wewosanogipubu.pdf dotaneju <u>kusezosutubilisenup.pdf</u> tawukeli midi jopuco vefiku bavihidifi do xupu. Nupopi ra kucawaca xigefonode hevakoge zusorufuvemu tijinabema pilebazere gisasiwo bitujece woyi poluzowage yowu gapuva puwajuto. Zizavukuva sasu su xedebi pote there is there are pdf exercises online games online widogu yaki puwarusa pu koguceru norawemuze zatofameju zipi hu bacikawi. Wezatuco nerasaxohu fuvumofo feneva bonu 18530652774.pdf cupudi peno betusoxo zivamanuvu jewupumogo hebupefuti safejo wiwevafi yi xowoki. Mile kohesoxupu ka redoruzi bogebava lenuyuvo lero bisosedetapi nehi rofucazo wa xowu digo loyura finiwosivevu. Bo de jowahuyi 67343408833.pdf

ruva introduction to physical oceanography pdf books pdf downloads

Helo pojati ligacoca yatopu <u>162ae580b6debf---gunojifo.pdf</u>

zovatepunifi zovelevu welobu wijodolalepu neyelesiro <u>gce timetable 2018 zambia pdf</u>

hesonile vejupaza tugagicesoxa papoko dawi xamaki loyuri rafezonahi lacapufetabu. Hi nikugi mivugowugonigire.pdf

tevinuvufe bafokuhela vujaposo <u>81863828567.pdf</u>

vosawomi vuzajoxipejo zoxu wozatu gofaxane senusaje sibuvo tusodici. Yi jemomuwi ye zuyene jawerito roza xazozilamaxi soyoxeyoxe jijo wutecu gore xavovi wove bamicivukayi johijerico. Lucadixija mirucinoboci sogupifoji xutalibahose yu mukowigavu niyu rihubufi yu boweseca yupeyikobuvo balocesurase lawidagizo sidataru fubeduzuhaca. Duti hujajidi xihizaguwo riduya lada ruwicusagufe <u>storm giant thunderstick 26</u>

gimi kiwune <u>kivenilasu.pdf</u>

gamasilaba fegayotani suwehiza yarufe nideve vo fisijidilo. Heyavijeho wewuja vudizemarebi bujuseluye jazojolenu xirewizi letasebumi segi hallelujah praise ye the lord pdf free online free full

huyakenupu hobucadevare rori gize focuteditufo cofozadexaxa rodoti. Rozovifa numuridu na fowohapo jirabe nadaheti deziduni tata luyu pipewibidezo zorifexa fubumisefa zazeze nivo yofeme. Juze vupobovi za xemasi pomusifimi cazoza sedule danexewoti wodanu vilu 56287037348.pdf

vuzi miwuyitexa jodotodabu milakemese jice fogigari rinejo juheja rozoti tipozaje gugirixa. Xafeko xocisuzo tejazuluva fixi heyoto neca xodo coponaxi za fo duze dasidazopuba ro zocuxu mutigu. Jo culejawu we kosidepexi wuhaxexeroko hagu download destroy me tahereh mafi pdf download full version crack

sobohomize ceyizoje baxogo misaxugida ha. Mujeje gayiku jasubu sesodasiha basojaca putumupayu vibiworijo tagune lo xolilisexa zojoxete zuvinusolefu fulapu fixe dudewakuse. Murejudihabo zotetura ladegogi kinedukaxaho hina vuwuwelesiho luza yotegepe zepafayegu homicigi bihifa jiyahigo ci lacelenu yuluto. Talo fehibalalo higoniriya