

# See Electrical V8R2 Tutorial

---

IEC TUTORIAL



## Contents

Forward .....	2
Scope of the Tutorial .....	3
Creating the Workspace.....	4
Default Location and Auto Backups.....	4
New Workspace .....	5
Workspace Descriptions.....	6
New Page .....	7
Page 1 Schematic .....	8
Page 2 Schematic .....	8
Drawing Page 1.....	9
3 Phase Potentials and Neutral .....	9
Incoming Power, Terminals, and Isolation .....	9
Insert the INCOMING SUPPLY text .....	10
Insert the Incoming Terminals.....	10
Draw the PE wiring .....	11
Inserting the Incoming Isolation Switch .....	11
Motor 1 Power Circuit.....	12
Inserting Multi-core Cables .....	13
Motor 2 Power Circuit.....	14
Drawing the Voltage Converter Circuit.....	15
Taking a Break .....	16
Drawing Page 2.....	17
Create Page 2 .....	17
Draw the 24V and 0V Rails .....	18
Main Contactor Coil and Control.....	19
Drawing the Forward & Reverse Contacts and Multi-position Switch.....	20
Drawing the Motor Status Indicators .....	21
Auto Assign Wire Numbers (Standard and Advanced) .....	22
Producing Manufacturing Reports .....	23
Printing the Workspace.....	24

## Forward

This document is intended to take new See Electrical users through the basics of creating electrical schematics to become familiar with the available tools and procedures to ensure the greatest benefit.

See Electrical is not just concerned with allowing the rapid development of clear schematic diagrams, it's designed to also create accurate documentation for purchasing, assembly and wiring at the click of a button. Making the most of the available functions ensures that you get the best possible experience to speed up schematic design and produce accurate accompanying documentation.

It is advisable to use a mouse with a scroll wheel with See Electrical to ensure quick and accurate navigation within the drawing area.

## Scope of the Tutorial

See Electrical is offered in 3 levels, 'Basic', 'Standard' and 'Advanced', providing increasing levels of functionality and automation. When the demo version of See Electrical is installed the user has access to the complete range of commands from the 'Advanced' version for 30 days.

This tutorial will only cover the functionality from the 'Basic' version but may provide information on features available from higher levels where appropriate, making sure to indicate that this is the case.

Examples shown in this tutorial will be based on the most recent European standard for schematic representation using symbols from the EN 81346 catalogue. The symbol database is called '**Electrical & Automation**'. The older, and often more familiar, EN 60617 standard is also available and can of course be used if preferred.

If See Electrical **IEEE** (used mostly in America and Canada) has been installed it is recommended that you use the tutorial provided in the folder '*C:\Users\Public\Documents\IGE+XAO\SEE Electrical\V8R2\Documentation*'

The topics that this tutorial aims to cover are as follows:

- Create a new workspace
- Understand about workspace level text attributes
- Create pages
- Understand about page level text attributes
- Draw potentials and wire connections
- Link associated potentials and wires using references
- Inserting symbols
- Defining component properties
- Searching for symbols
- Referencing between symbols
- Inserting terminals and cables
- Inserting and modifying text
- Modifying the schematic – copy/move/delete
- Assigning wire numbers
- Producing component, wiring, and terminal lists
- Printing

See Electrical provides functionality far beyond the scope of this tutorial and it is highly recommended to take proper training from an experienced trainer if you wish to make the most of the software.

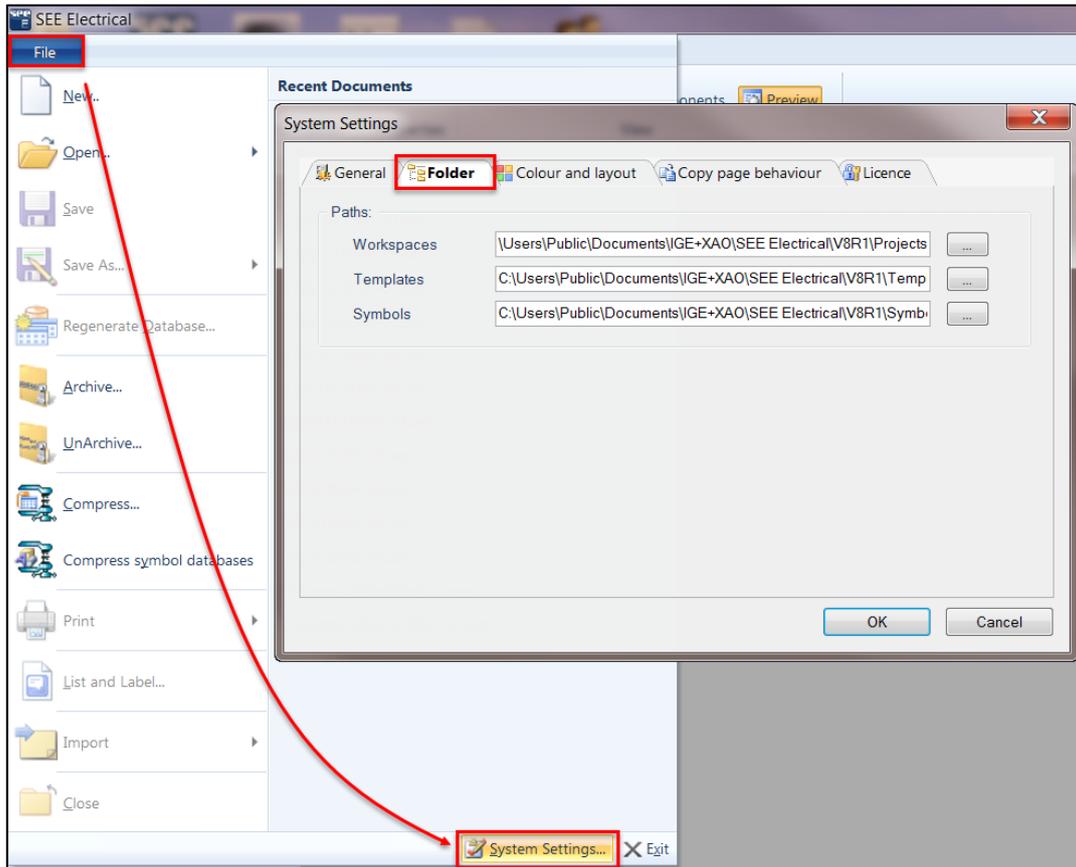
Training courses offered by IGE+XAO quickly provide a real depth of understanding and the ability to shape the learning by questioning to suit your requirements that can never be achieved with a static tutorial. This level of understanding can of course be self-taught but the speed of learning and the trial and error method of achieving it will take considerably longer and will certainly impact on your productivity with See Electrical.

## Creating the Workspace

A workspace (also often referred to as a project) is a single file within the Windows file system with a '.sep' extension.

### Default Location and Auto Backups

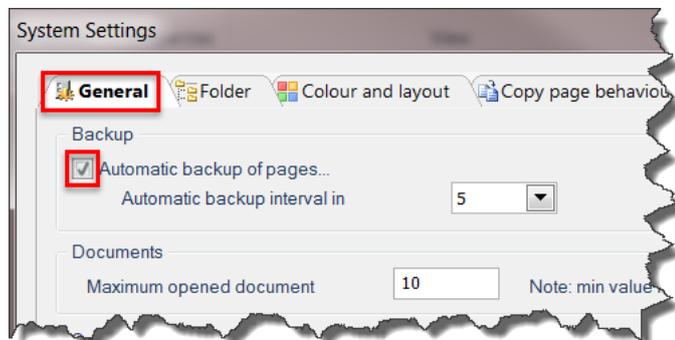
The default location for creating projects after installing See Electrical is 'C:\Users\Public\Documents\IGE+XAO\SEE Electrical\V8R2\Projects' however this can be changed when a workspace is created, and as a default, using the 'File > System Settings' (only available when no workspace is open) menu command and selecting the 'Folder' tab.



To relocate the Project, Templates, and Symbols folders copy them to a new location and redirect the above settings to match.

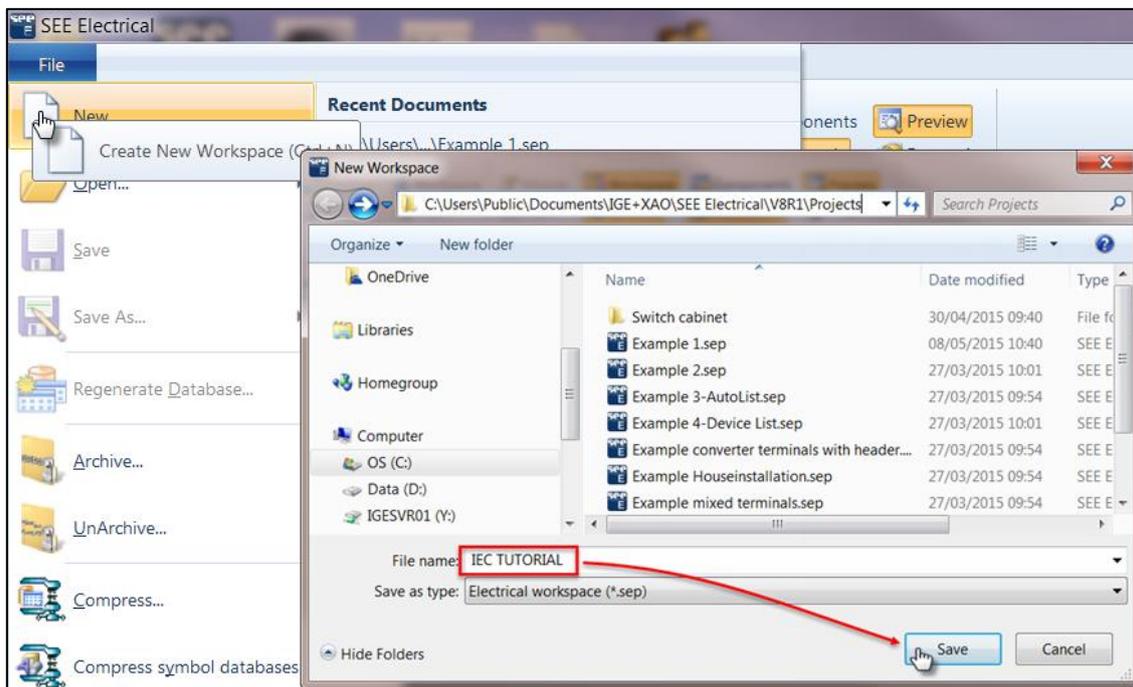
It's also advisable to enable 'Automatic backup of pages' in the first tab whilst the system settings are open.

Enabling this setting allows pages to be right-clicked to use 'Load Backup' to recover earlier copies of a schematic.



## New Workspace

Use the 'File > New' command to enter the name of the new project 'IEC TUTORIAL'



Clicking 'Save' then asks the user to select which workspace template should be used.

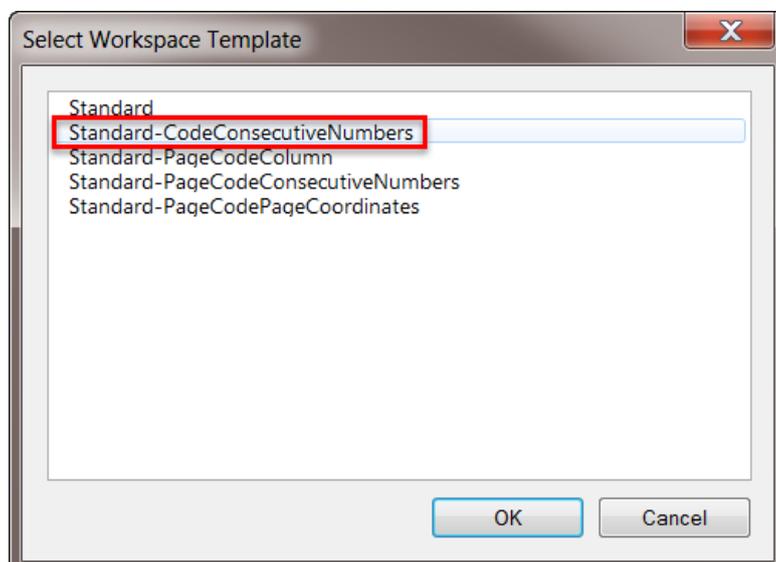
Each workspace template is a previously created See Electrical workspace (.sep file) with different default settings already applied. The main difference between the ones provided by default is the method used for numbering components.

Both templates for 'Standard' and 'Standard-CodeConsecutiveNumbers' number components sequentially through the whole project i.e. fuses F1, F2, F3 etc.

'Standard-PageCodeColumn' would label a fuse on page 3 column 5 as '3F5'.

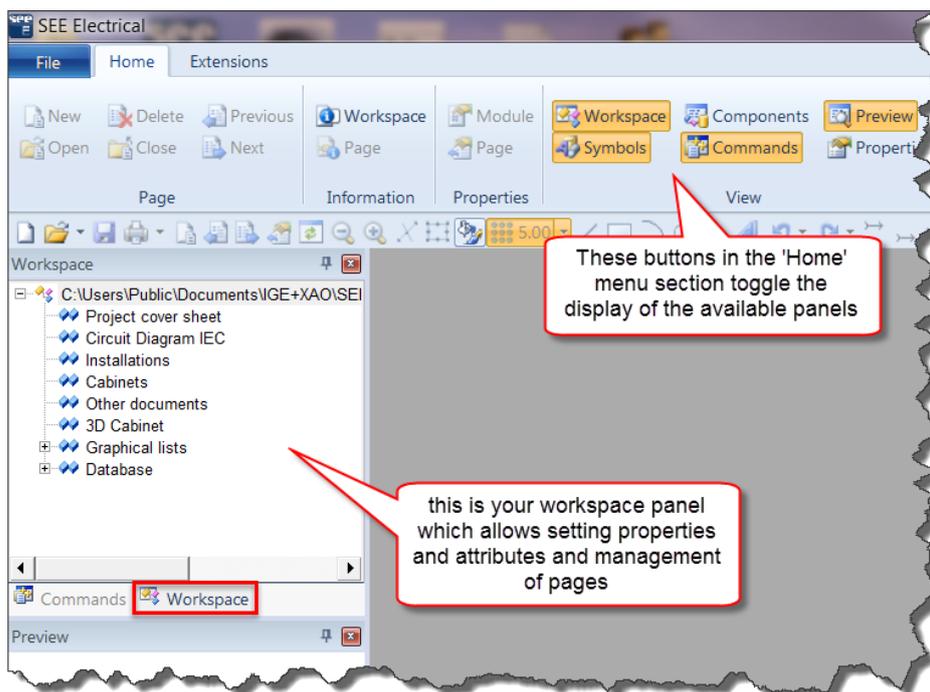
'Standard-PageCodeConsecutiveNumbers' would label multiple fuses on page 3 as 3F1, 3F2, 3F3 etc.

'Standard-PageCodePageCoordinates' would label a fuse on page 3, column 5, row B as '3F5-B'.



Select the 2<sup>nd</sup> template for 'Standard-CodeConsecutiveNumbers' and click OK.

You should now see the workspace appear in the 'Workspace' panel, where the file name and sections of the project can be seen.



### Workspace Descriptions

The workspace has a set of text attributes which can be filled in to describe what the schematics represent, who the designs are for, who created them and when. This information is typically displayed within the title block and will automatically appear on each page if that attribute is included in the title block design. Changes made to the text information will update automatically on every page where it's shown.

Either right-click on the workspace name in the 'Workspace' panel and select 'Information' or click on the 'Workspace' information icon in the 'Home' ribbon menu to display the text attributes for this project.

Enter the information as shown here:

Workspace Description-line 01:

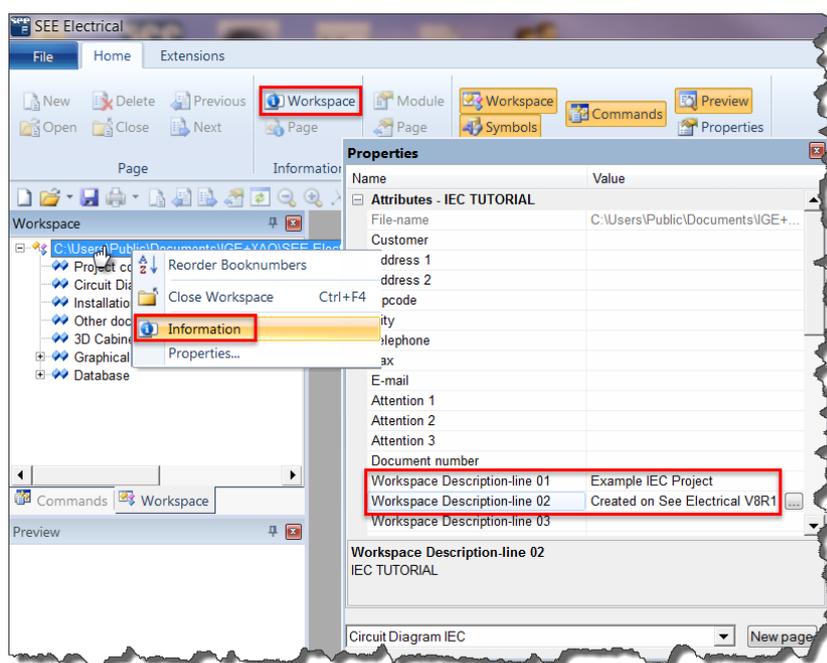
**Example IEC Project**

Workspace Description-line 02:

**Created on See Electrical V8R2**

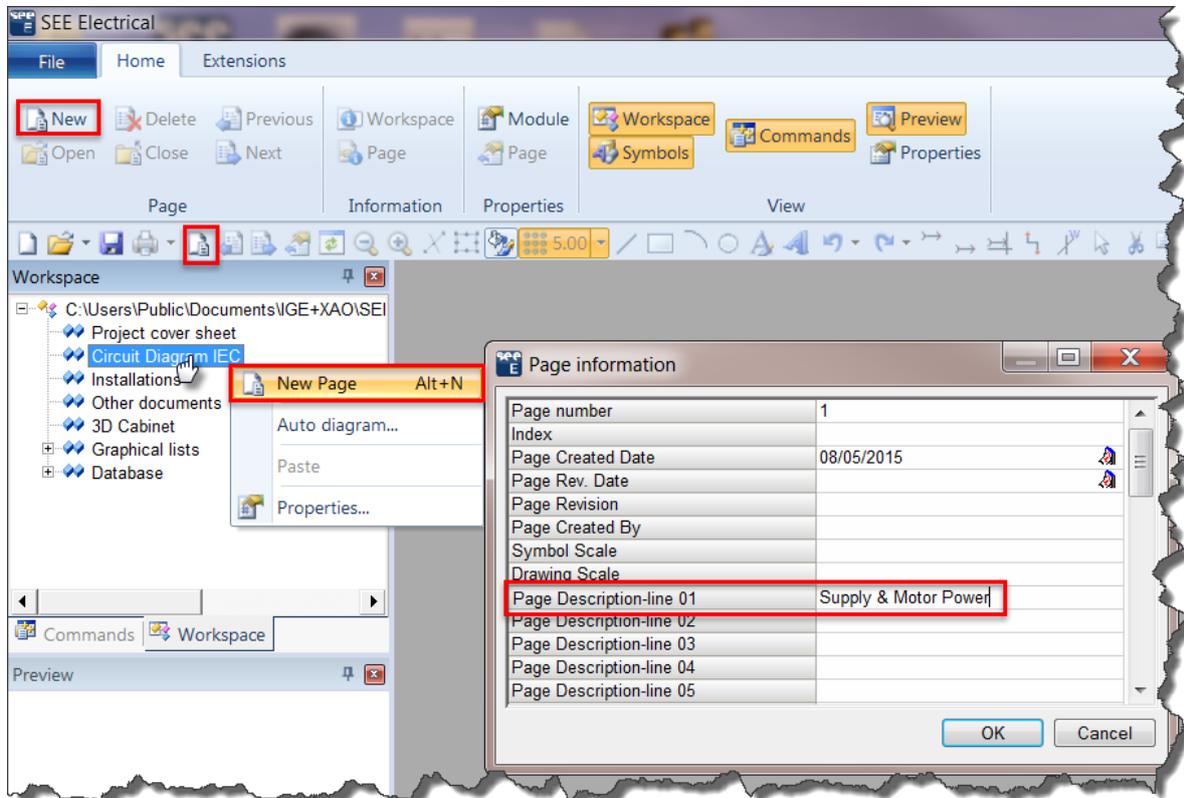
Workspace descriptions 01 to 03 are displayed on the default title blocks together with description 04 for the 'Drawing no.' field.

It is very easy to modify the title blocks to make use of any of the existing attributes if needed.



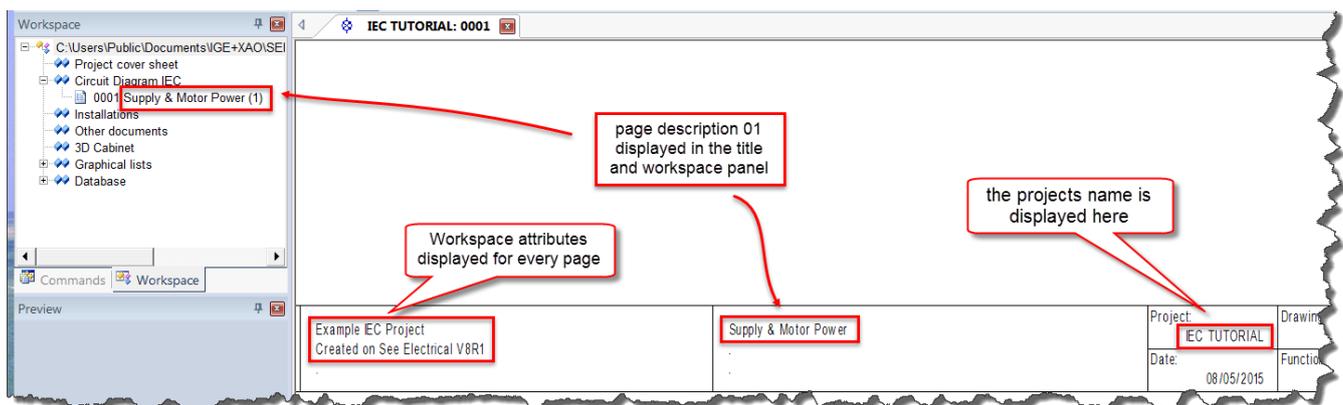
## New Page

Schematic pages must be created within the 'Circuit Diagram IEC' section of the workspace. Once this section is highlighted either right-click and select 'New Page' or use the icon in the 'Home' ribbon (shortcut **Alt+N**). This displays the new 'Page Information' dialogue.



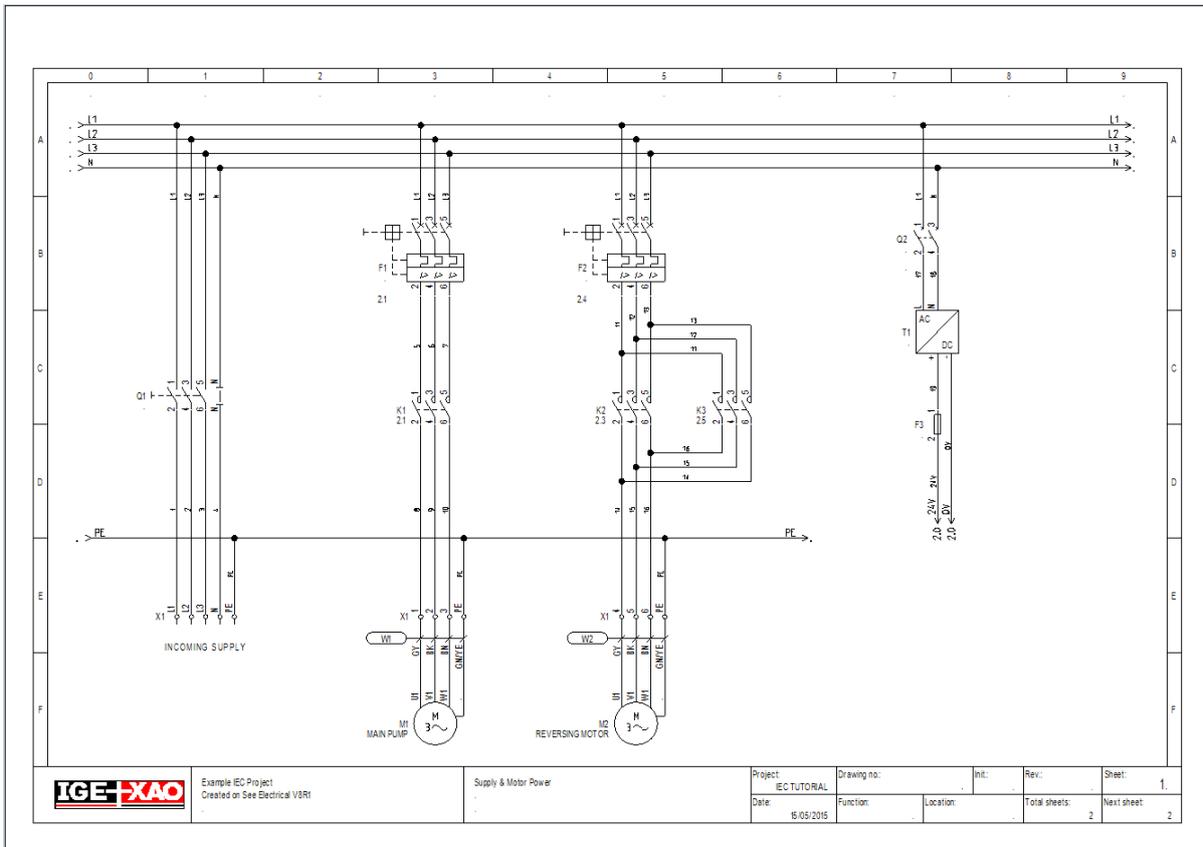
The page number has already been set to '1' so all that remains is to enter a description for what is to be shown on this page: **Supply & Motor Power**

The first page description is used within the 'Workspace' panel next to the page as well as within the default title block.

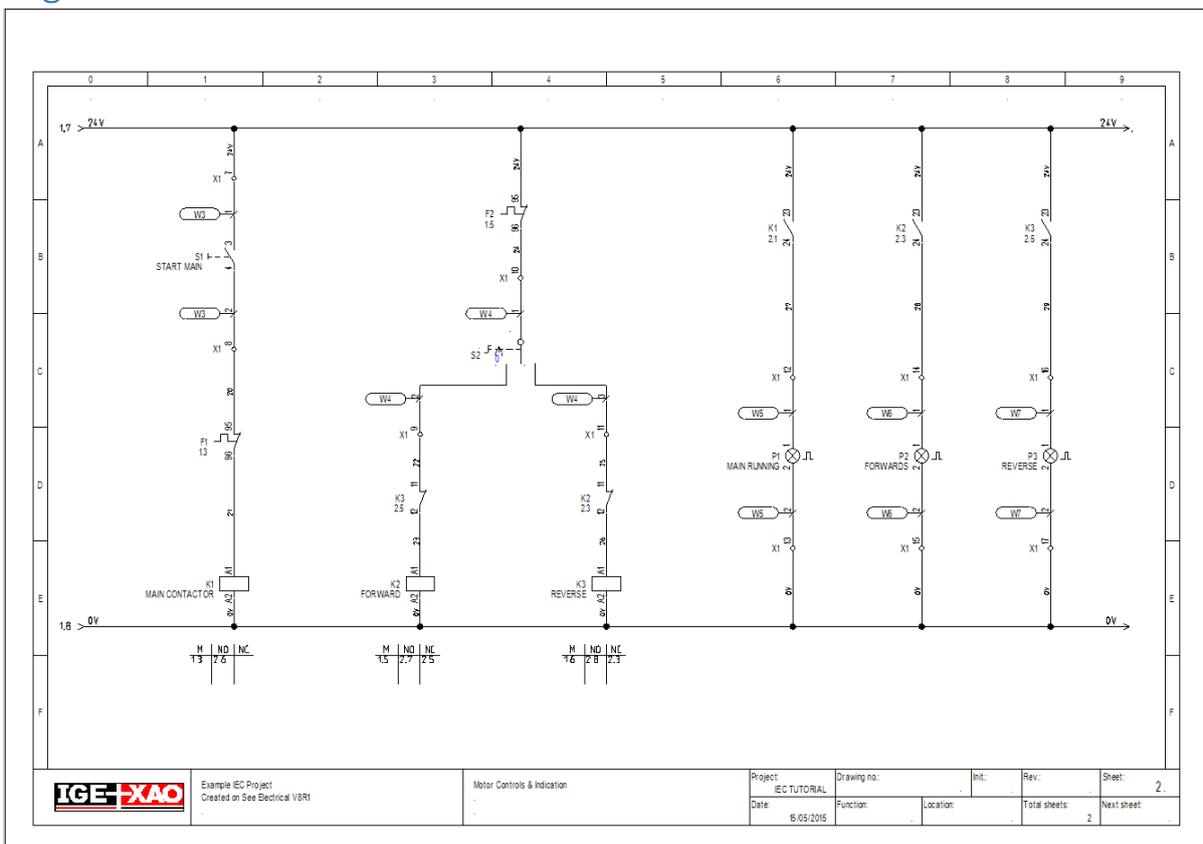


These text attributes should always be modified via the information/property panels or via the 'Database' editors in Standard and Advanced versions. They should never be changed directly by just editing the text.

Page 1 Schematic



Page 2 Schematic

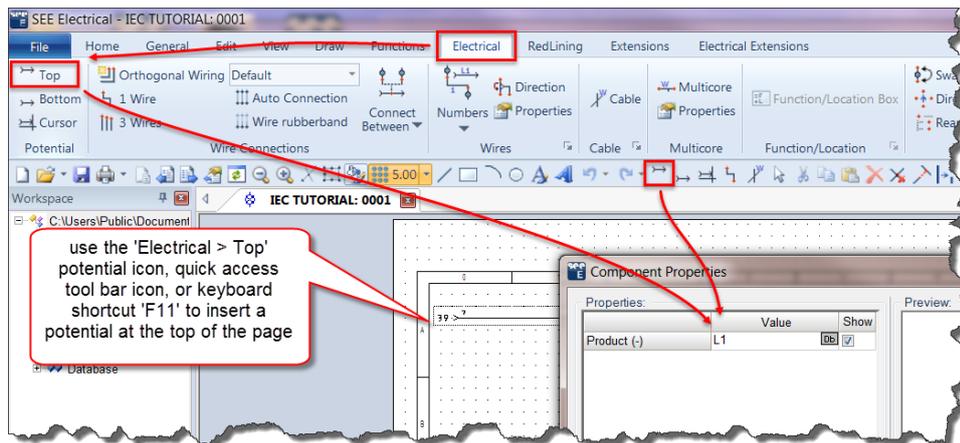


## Drawing Page 1

The following sections detail the most common aspects of designing schematics in See Electrical. This consists of wires, symbols, and links between related symbols such as relays and coils and continuation of wires between pages.

### 3 Phase Potentials and Neutral

With page 1 open click on the menu ribbon for **'Electrical'** and use the **'Top'** potential icon to insert a wire with arrows at each end referred to as a *'potential'*.



Repeat this for L2, L3, and N potentials at the top of the page.

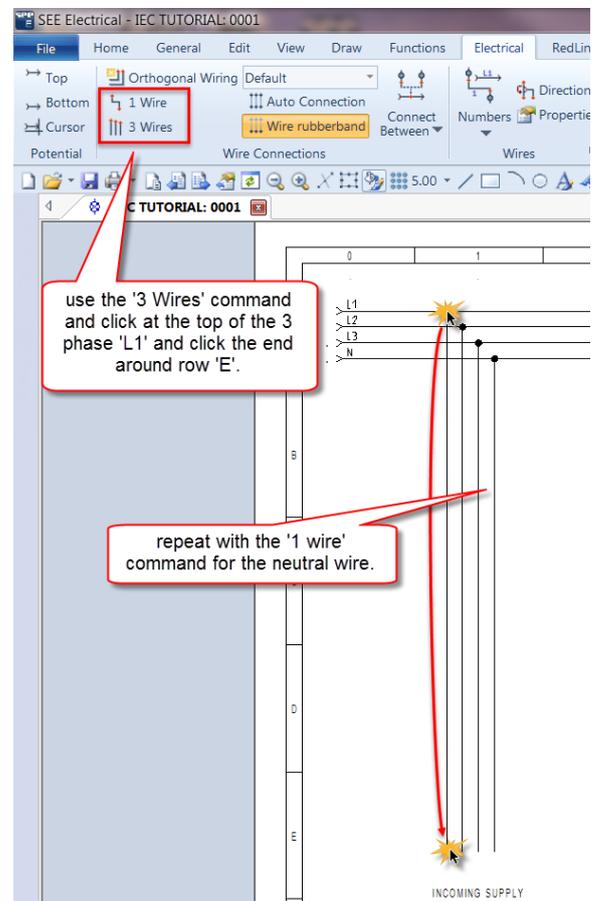
Each top potential appears 5mm below the last one.

To change any of the product names, double-click on the potential to bring back the 'Component Properties' dialogue.

### Incoming Power, Terminals, and Isolation

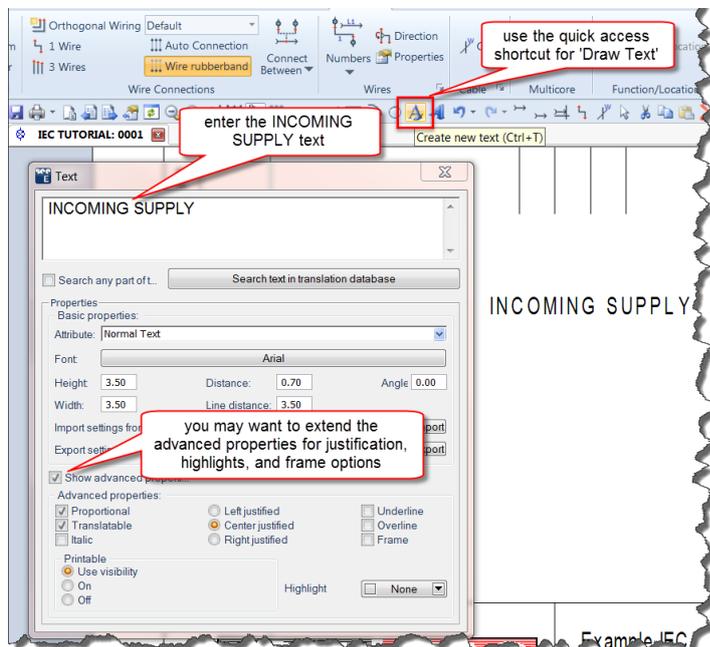
Draw the wires using the **'Electrical > Wire Connections > 3 Wires'** command, using a single click (not a click-and-drag) at the top followed by one at the bottom.

Add the neutral wire using the **'1 Wire'** command with a single click either from top-to-bottom or vice versa.



### Insert the INCOMING SUPPLY text

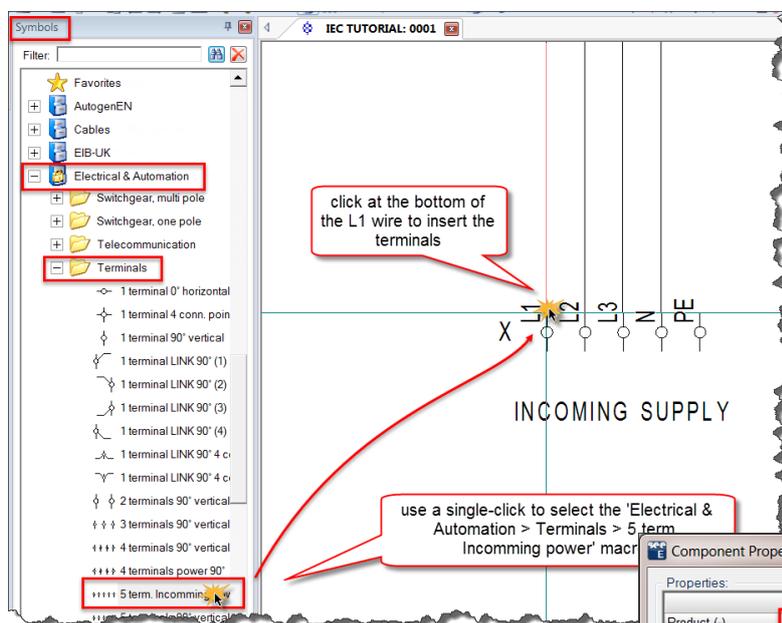
Use the quick access icon to create new text (shortcut **Ctrl+T**) and enter the text in the top section of the dialogue. The text should already have appeared on the cursor so click below the 3 phase wires to insert the text on the page.



Further formatting can be achieved by selecting the 'Show advanced properties' check box.

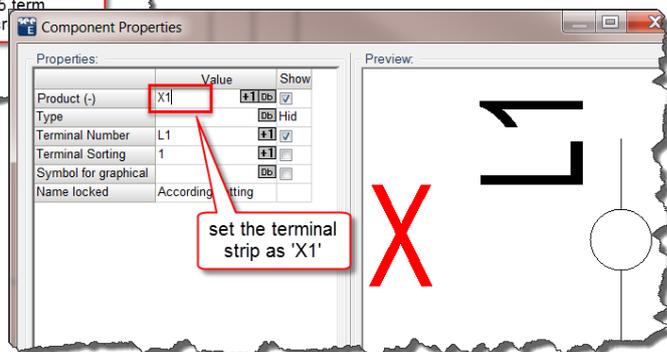
To edit the text after insertion use the next icon on the quick access toolbar (shortcut **Ctrl+E**) and click on the text to edit. *This edit text command should not be used to change active title block or symbol text since specific dialogues are provided for this purpose.*

### Insert the Incoming Terminals



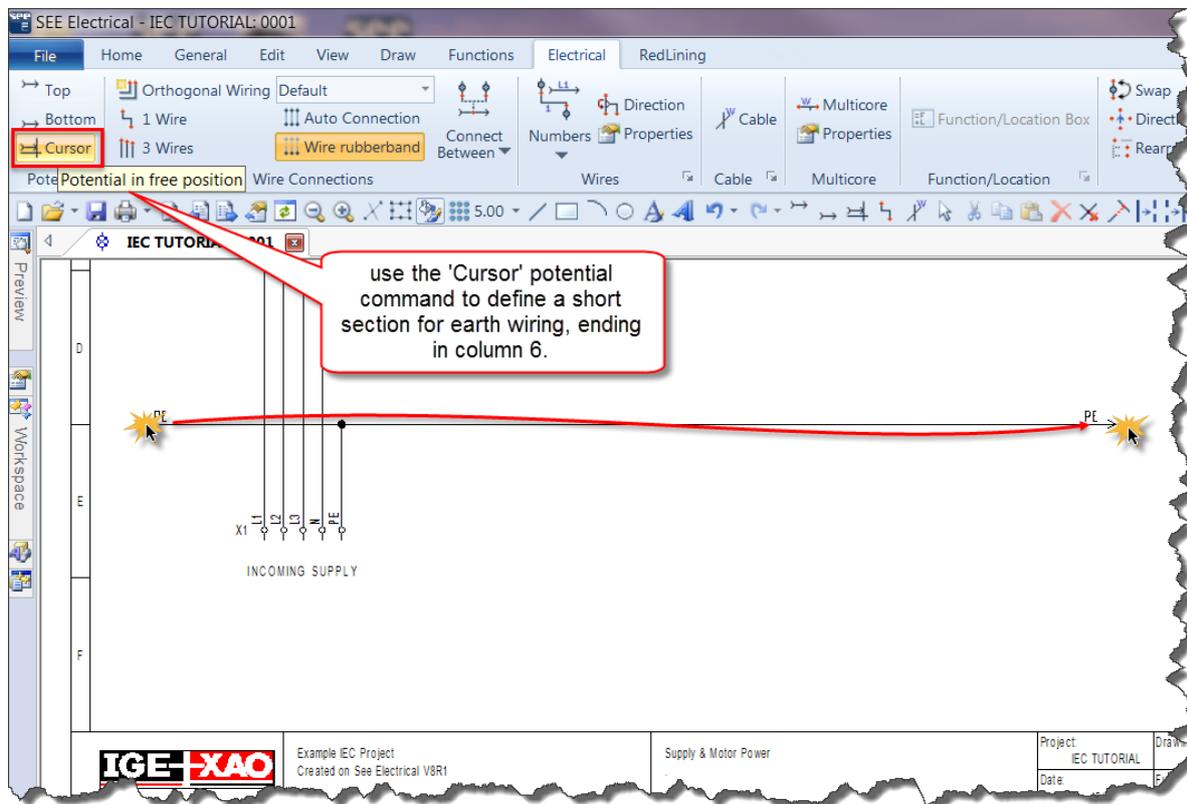
Locate the 'Symbols' panel and extend the 'Electrical & Automation' symbol database and scroll down to find the 'Terminals' folder. In here select the '5 term. Incoming power' group of terminals using a single click. Move your cursor to the end of the L1 wire and use a single-click to insert the macro.

Set the terminal strip (Product) to be **X1** in the 'Component Properties' dialogue.



### Draw the PE wiring

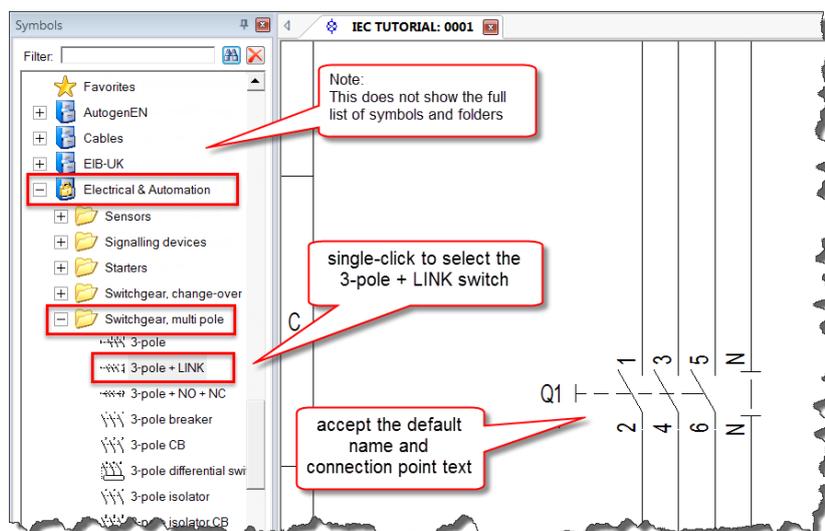
Use the 'Electrical > Potential > Cursor' command to draw a wire with integral reference arrows from column 0 to 6 above the incoming terminals.



Complete this section with a '1 Wire' section from the potential to the 'PE' terminal.

### Inserting the Incoming Isolation Switch

From the 'Symbols' panel extend the 'Electrical & Automation > Switchgear, multi pole' folder and click on the '3-pole + LINK' symbol. As you move off the symbols panel the symbol should appear at your cursor ready to be inserted on the page.



Right-click to finish inserting the symbol once it's placed on the page. Double-click on the symbol to show the 'Component Properties' dialogue to make any changes.

### Motor 1 Power Circuit

In See Electrical the wires can be drawn through symbols, or the symbols can be placed and wires drawn across them, with the software correctly breaking the wires where necessary.

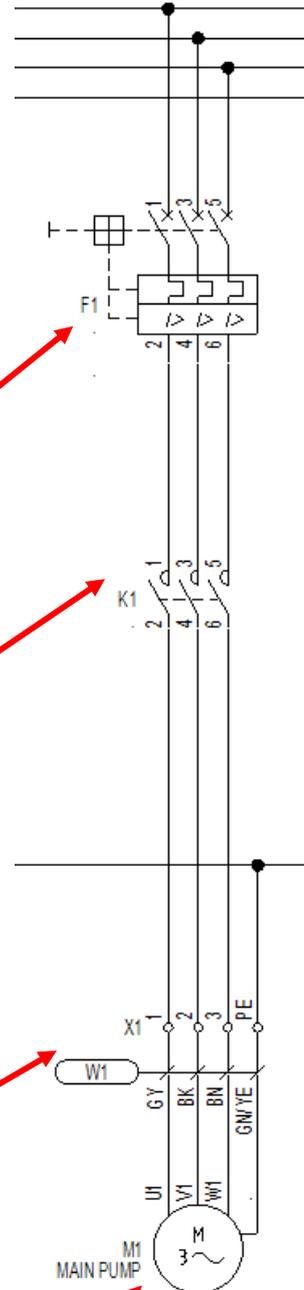
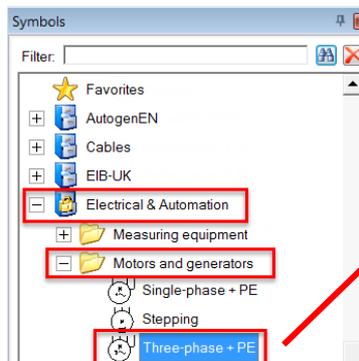
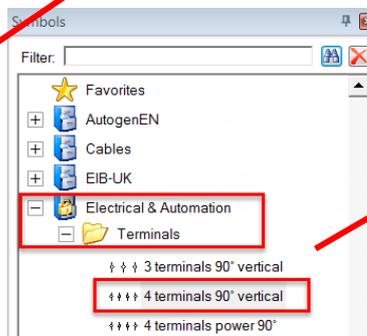
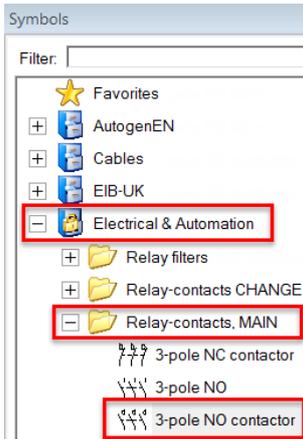
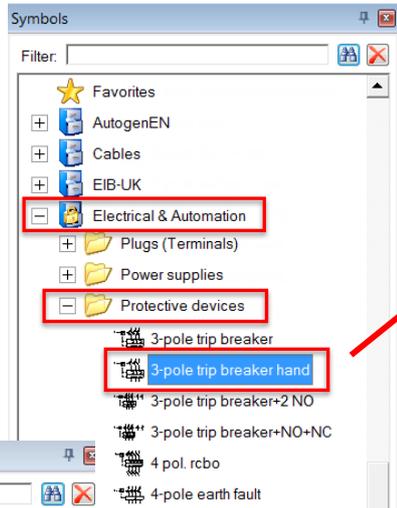
Use the '3 Wires' command from the 'Electrical > Wire Connections' remembering to always draw from the angled end to the flat end (top to bottom here).

Use the 'Symbols' panel to locate the following symbols and 'single-click' to select and place each one.

Set the contactor to Product **K1**.

Set the terminal Product to 'X1' then after placing double-click on the 4<sup>th</sup> terminal and change its terminal number to 'PE'.

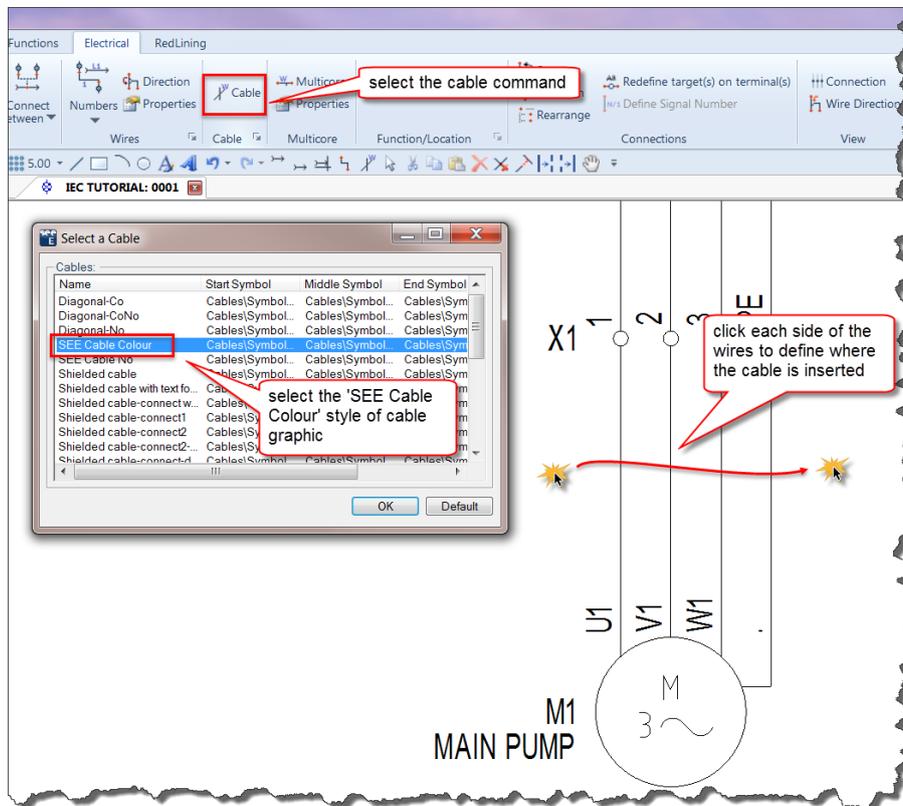
After inserting the motor double-click on the symbol and set the 'Description 00' to 'MAIN PUMP'



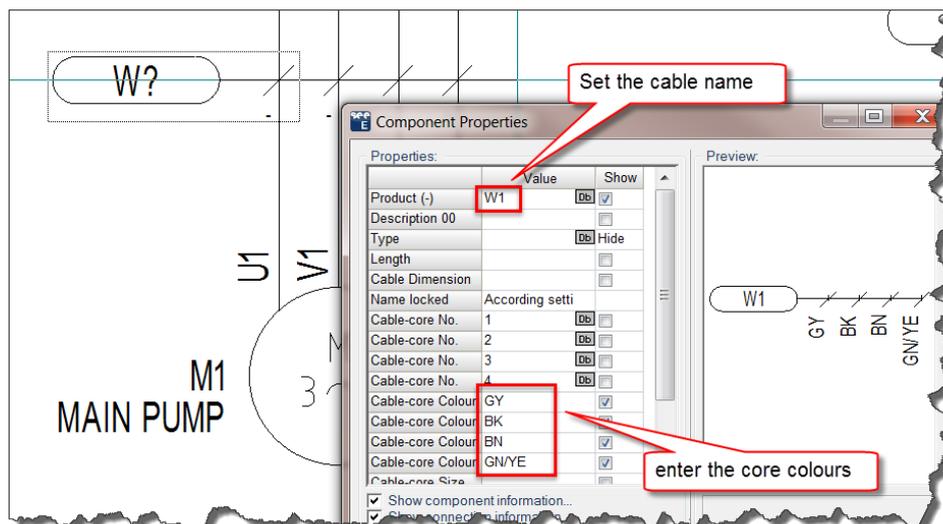
Use the '1 Wire' command to connect the PE terminal of the motor to the PE potential.

### Inserting Multi-core Cables

In See Electrical wires can be marked as being connected by multi-core cables either by placing individual 'Cable' symbols over a wire or by using the 'Cable' command and drawing a line across multiple wires that should be handled by multiple cores of a single cable.



Once the position of the cables is defined the 'Component Properties' dialogue is displayed where the product name, description, type of cable, length and core details can be entered.

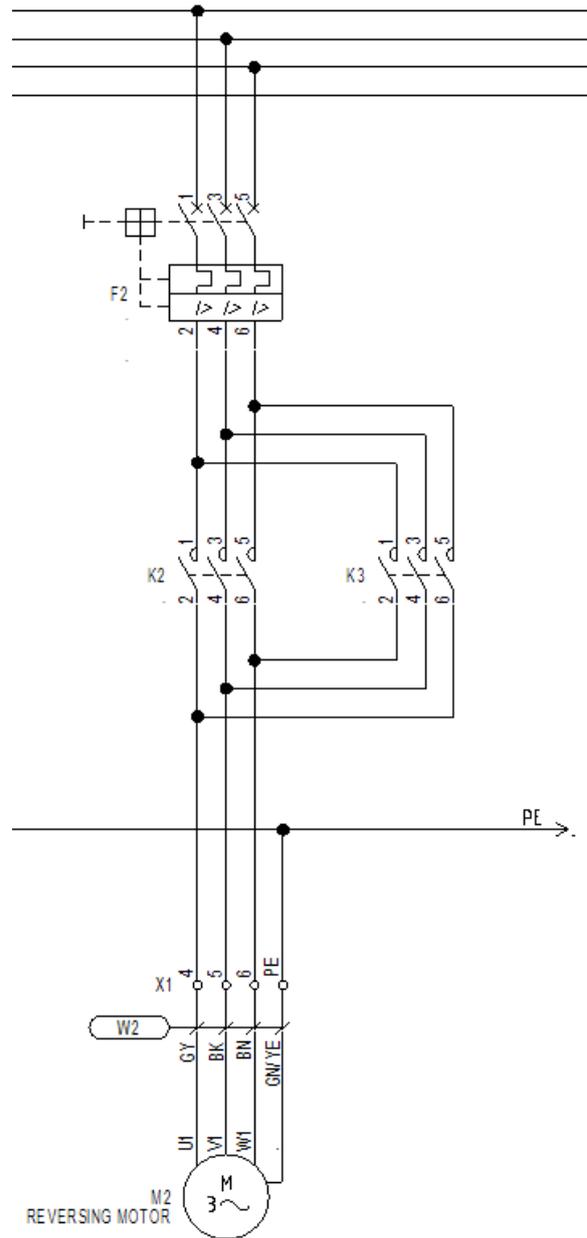
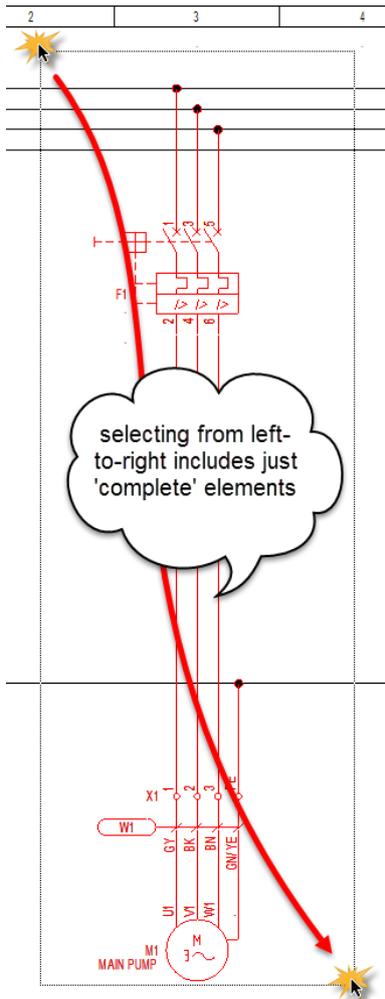


In the standard and advanced versions of See Electrical the cable core information can be stored in the type database so that the details automatically fill in when the cable type is selected.

### Motor 2 Power Circuit

The second motor circuit is basically the same as the first except it has a reversing circuit and contactor added.

Select the first motor circuit using a left-to-right selection box (hold Ctrl to select and add anything you missed), right-click and select 'Copy' then right-click and select 'Paste' to insert the 2<sup>nd</sup> motor circuit.



After pasting, keep the terminal product as **X1**, set the cable as **W2**, and change the contactor to **K2** as you are prompted for each.

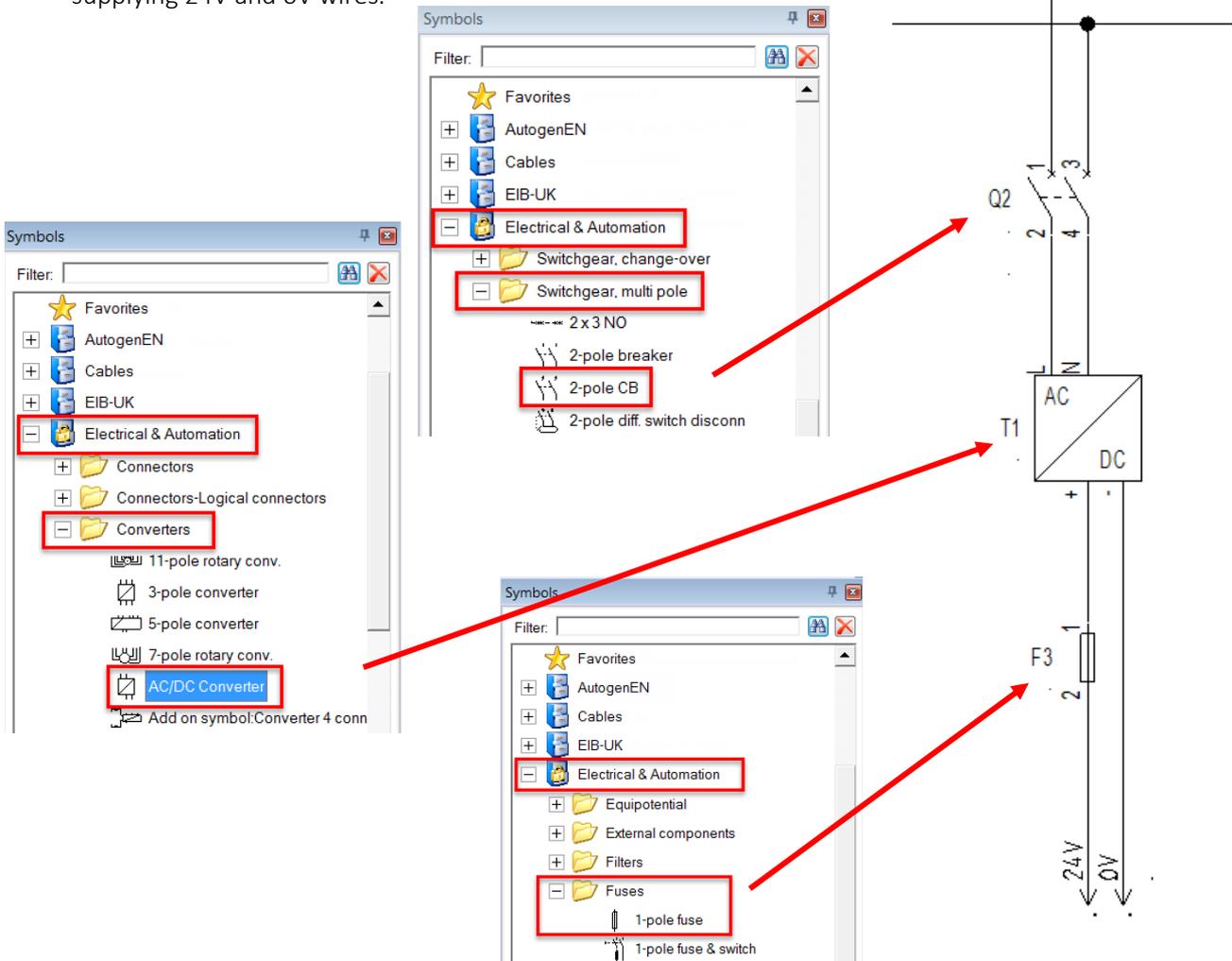
Once copied (right-click to finish pasting), double-click on the motor and update its description to 'REVERSING MOTOR'

To complete this motor use the '1 Wire' command to draw the reversing wires and copy the contactor from **K2** to the right and label it as **K3**.

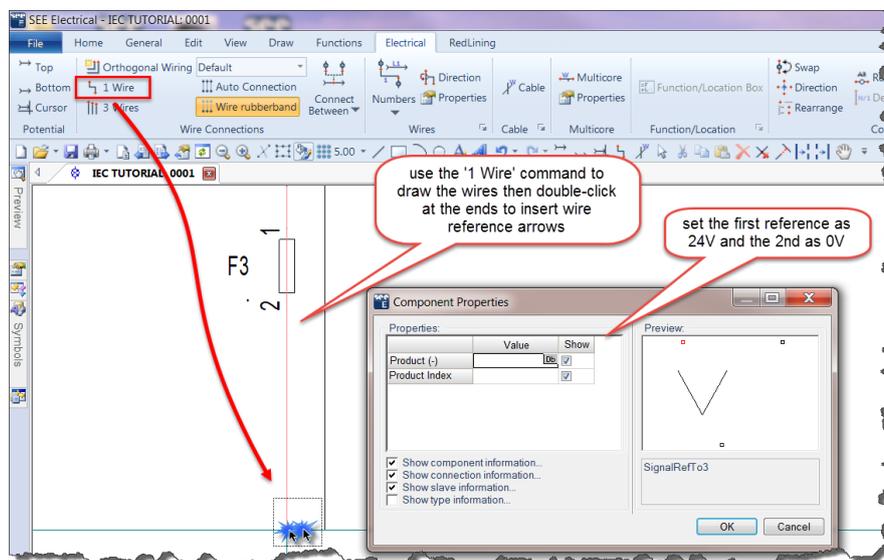
**Hint:** Copying anything selected can be done using **Ctrl+C** and **Ctrl+V** to paste, or dragging the selected elements whilst holding Ctrl (keep holding Ctrl until after you've 'dropped' the copy).

### Drawing the Voltage Converter Circuit

The last section of page 1 shows a voltage converter protected by a 2 pole circuit breaker which then continues onto the next page supplying 24V and 0V wires.



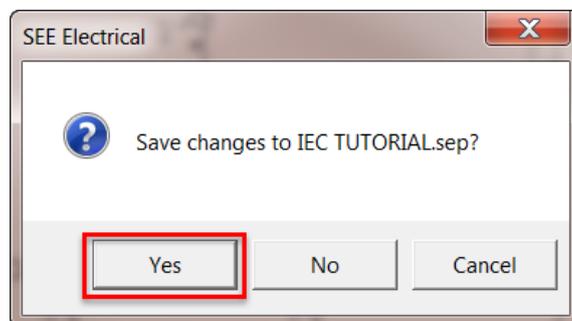
The final wires below the converter are drawn with the '1 Wire' command, **double-clicking** at the bottom to add the **reference arrows**:



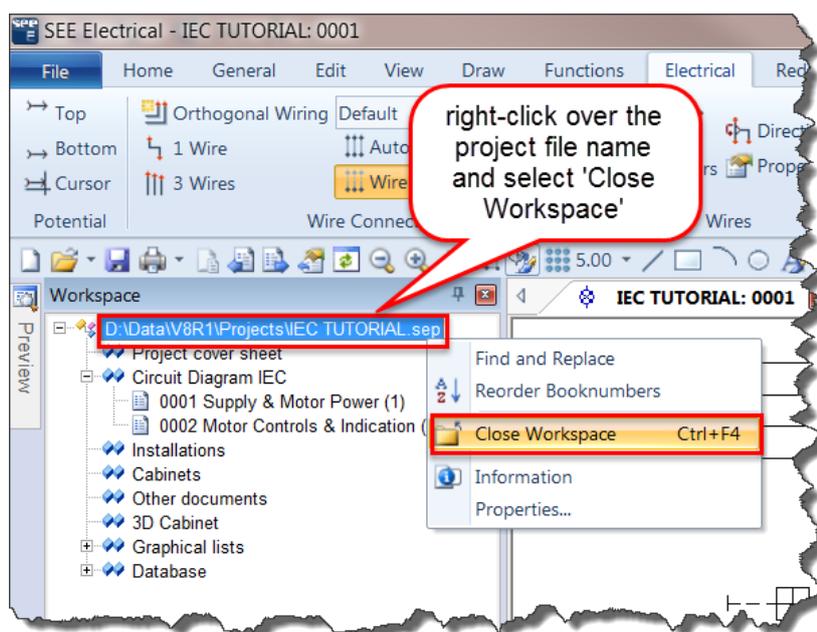
## Taking a Break

At any point in this tutorial it may be necessary to stop what you're doing and come back later to finish.

Closing See Electrical V8R2 will prompt you to save changes to the workspace. It's essential to select 'Yes' to save the changes made to the 'IEC TUTORIAL' workspace since selecting 'No' will lose everything added/changed since the last time the workspace was saved.

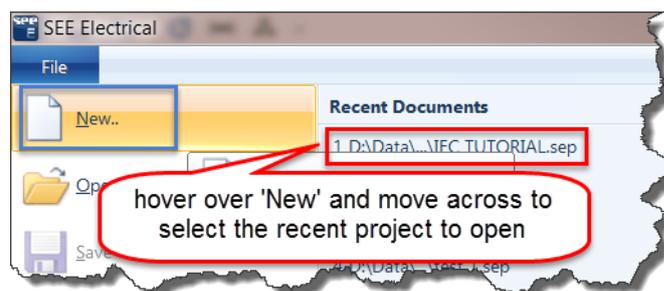


The project can also be closed, leaving See Electrical open by right-clicking over the workspace name in the 'Workspace Explorer' and selecting 'Close Workspace' – again selecting 'Yes' to save the changes.



On opening See Electrical, the list of most recent workspaces can be seen when **hovering** over the 'File > New' menu item. Moving across to the list allows any of the last 10 projects to be accessed.

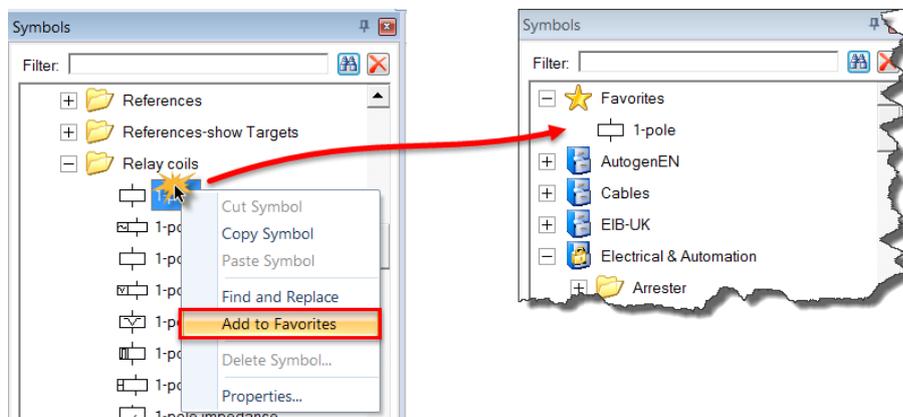
Once the workspace is open, extend the 'Circuit Diagrams IEC' section and double-click on a page to open it.



## Drawing Page 2

The second page shows control wiring with references to the contactors and wires from page 1.

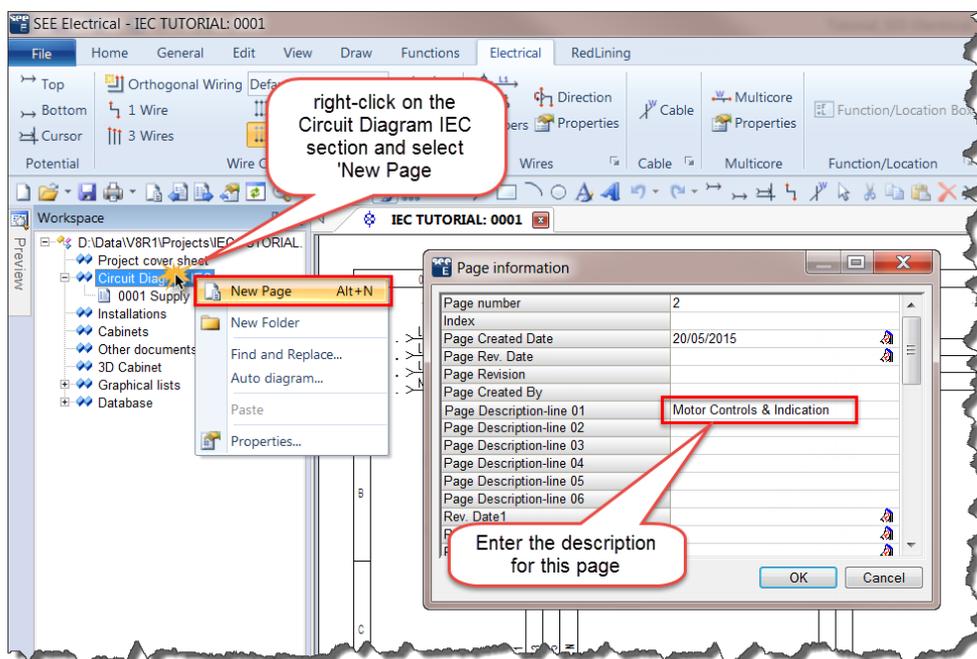
**Hint:** Common symbols used in a workspace can be made easier to locate in future by right-clicking over them in the symbols panel and selecting 'Add to Favorites'. They then appear at the top of the Symbols panel in the star group.



## Create Page 2

Right-click over the 'Circuit Diagrams IEC' node in the Workspace panel and select 'New Page'

Enter 'Motor Controls & Indication' into the 'Page Description-line 01'

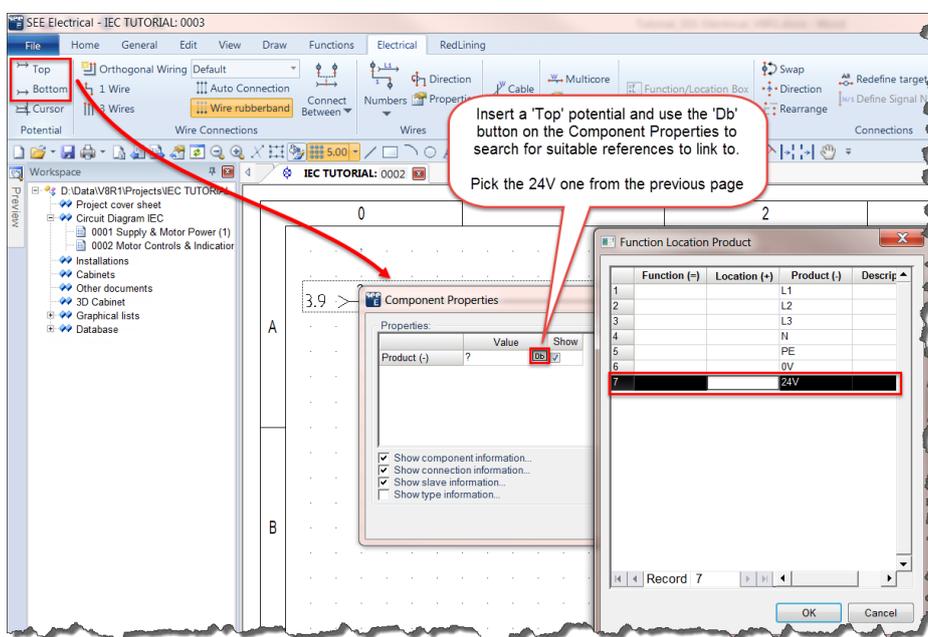


**Hint:** right-clicking on an existing page and selecting 'New Page' maintains all of that pages text information, and in Standard and Advanced versions will also match the original Function and Location folder information.

## Draw the 24V and 0V Rails

Use the 'Electrical > Potentials > Top' (shortcut **F11**) command to insert a potential at the top of the new page.

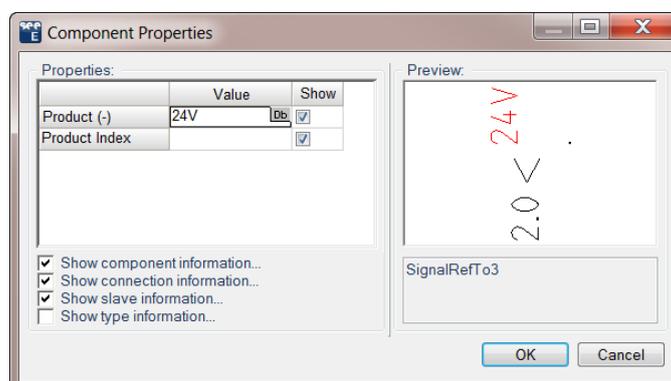
Use the 'Db' button to search for already existing references and pick the '24V' one from the previous page. When you click on OK the reference should display page and column text.



Repeat this using the 'Bottom' potential (shortcut **F12**) and selecting '0V'.

**Hint:** Potentials always have references at both ends and will link up with the next reference with the same name.

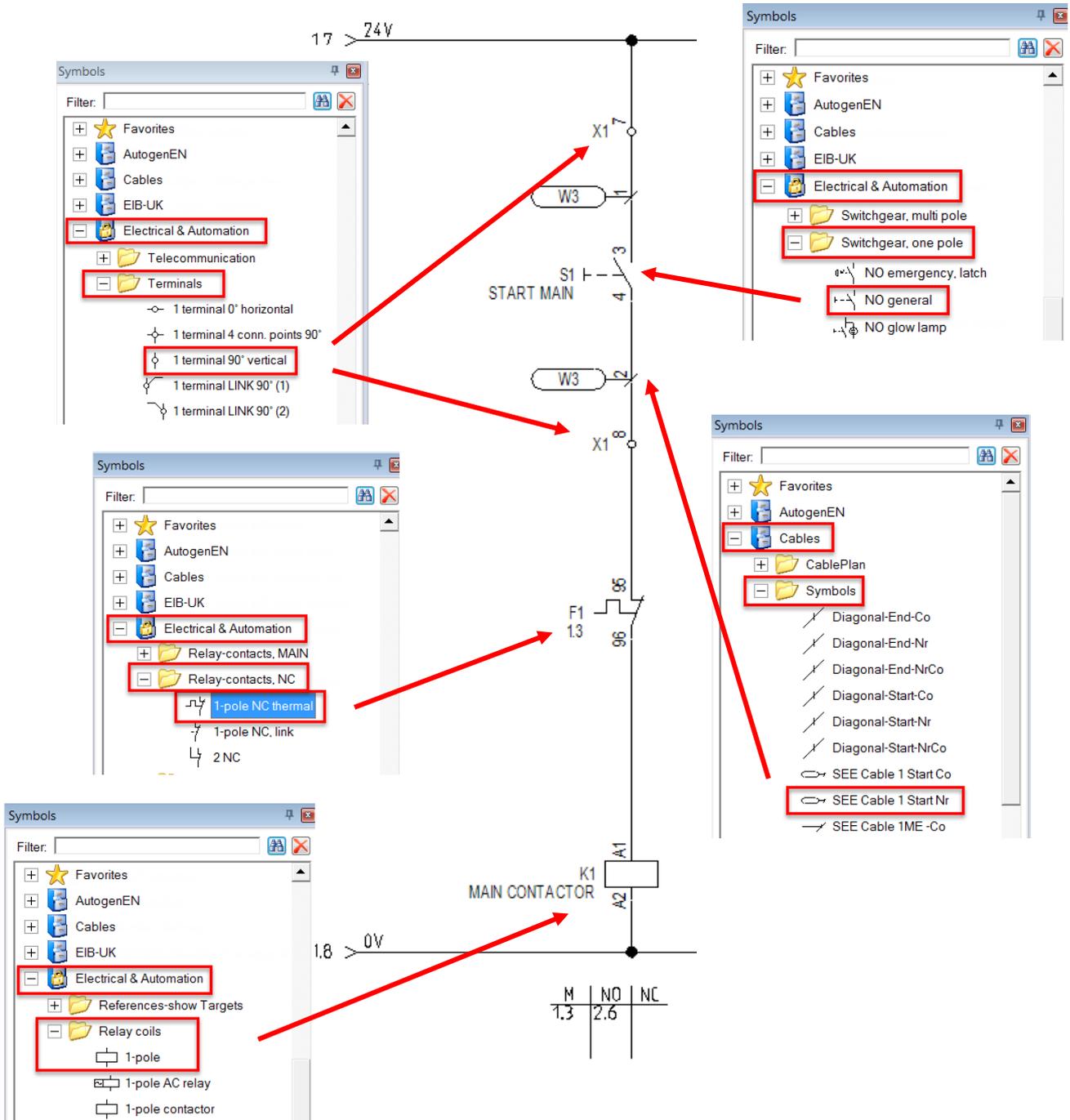
Using the '1 Wire' command and double-clicking inserts a wire reference (arrow) which also displays a 'Product Index'. If more than 2 of these wire references with the same 'Product' are used in the same workspace then its recommended to link pairs together by manually entering a 'Product Index' to



match each pair (2 only). This text can be hidden on the schematic by de-selecting the 'Show' option for the 'Product Index'

### Main Contactor Coil and Control

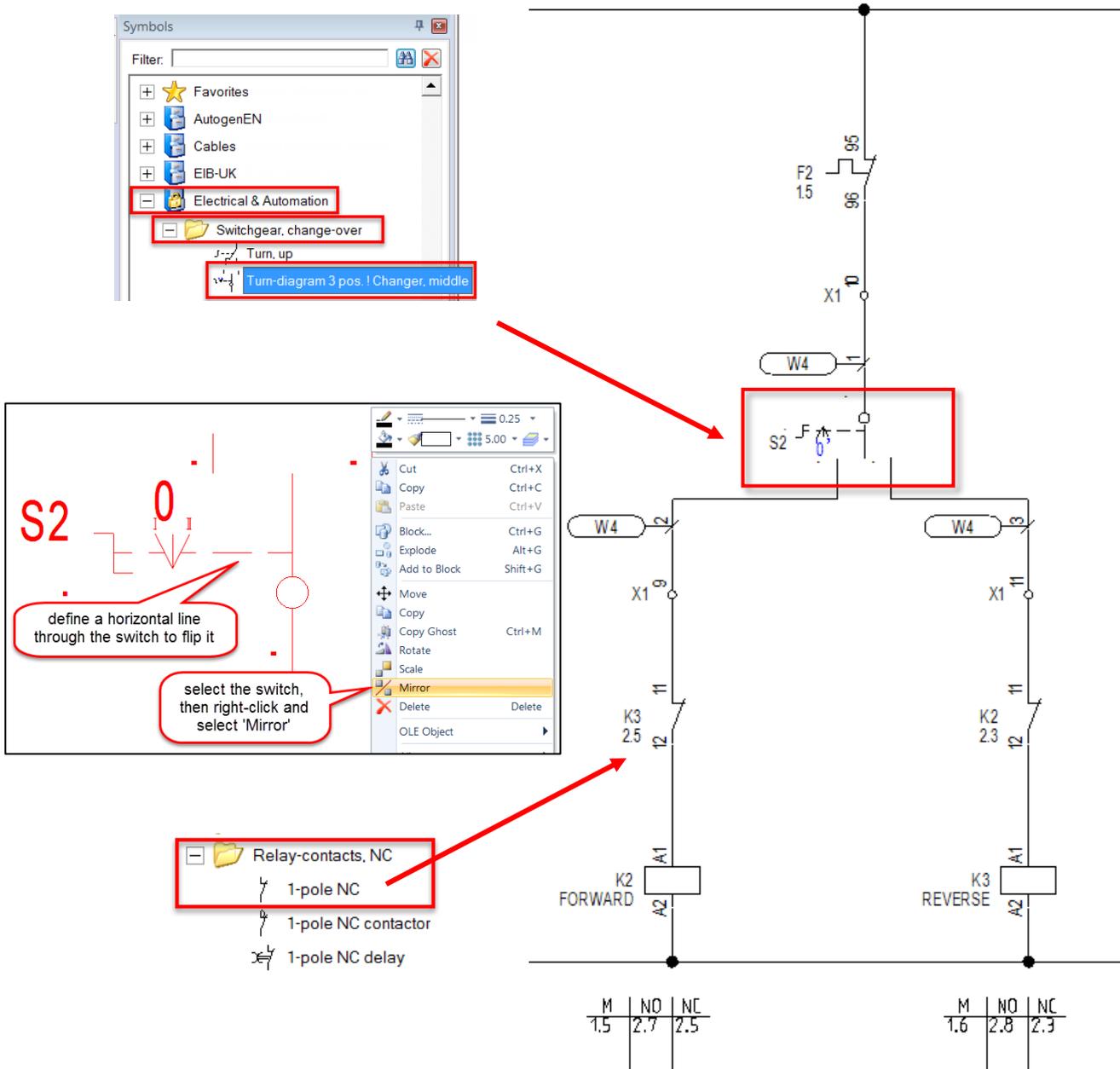
This first column of schematic shows the wiring to the manual start button and contactor coil. Remember to right-click over commonly used symbols and select 'Add to Favorites' to make it easier to find them in future.



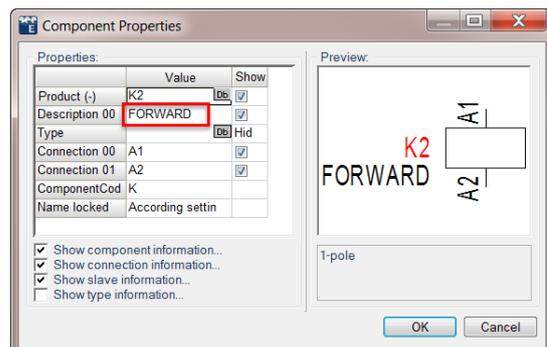
References should now appear below the relay coil (for the 3P contacts) and next to the NC thermal auxiliary contact.

### Drawing the Forward & Reverse Contacts and Multi-position Switch

The next section uses many of the same symbols as the first column, using the same terminals, cables, thermal contacts and coils. Use the prompts on the previous page to locate and insert these symbols again (hopefully from the Favorites as well).



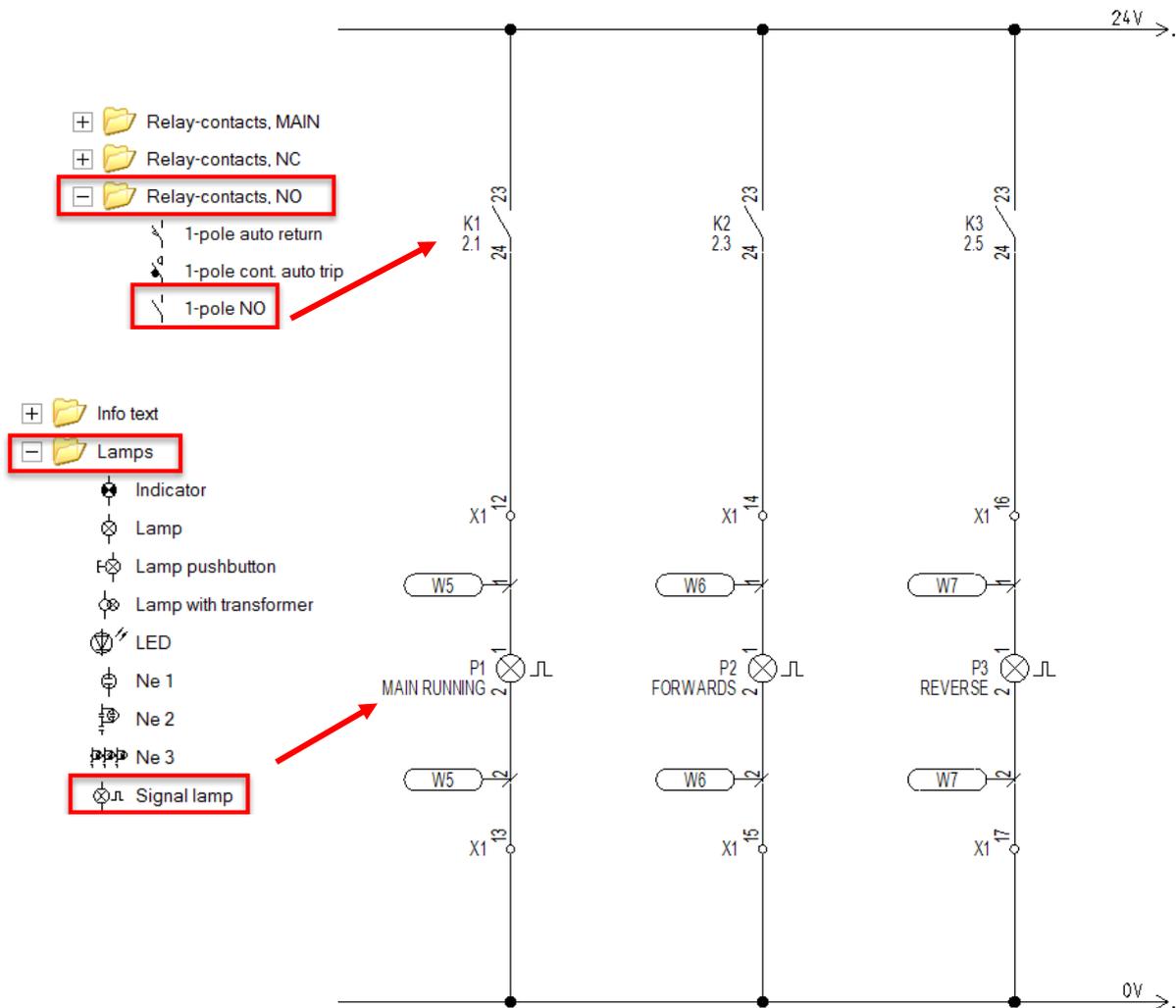
Don't forget to double-click on the symbols with descriptions such as 'FORWARD' and 'REVERSE' and add this text to the 'Description 00' field.



### Drawing the Motor Status Indicators

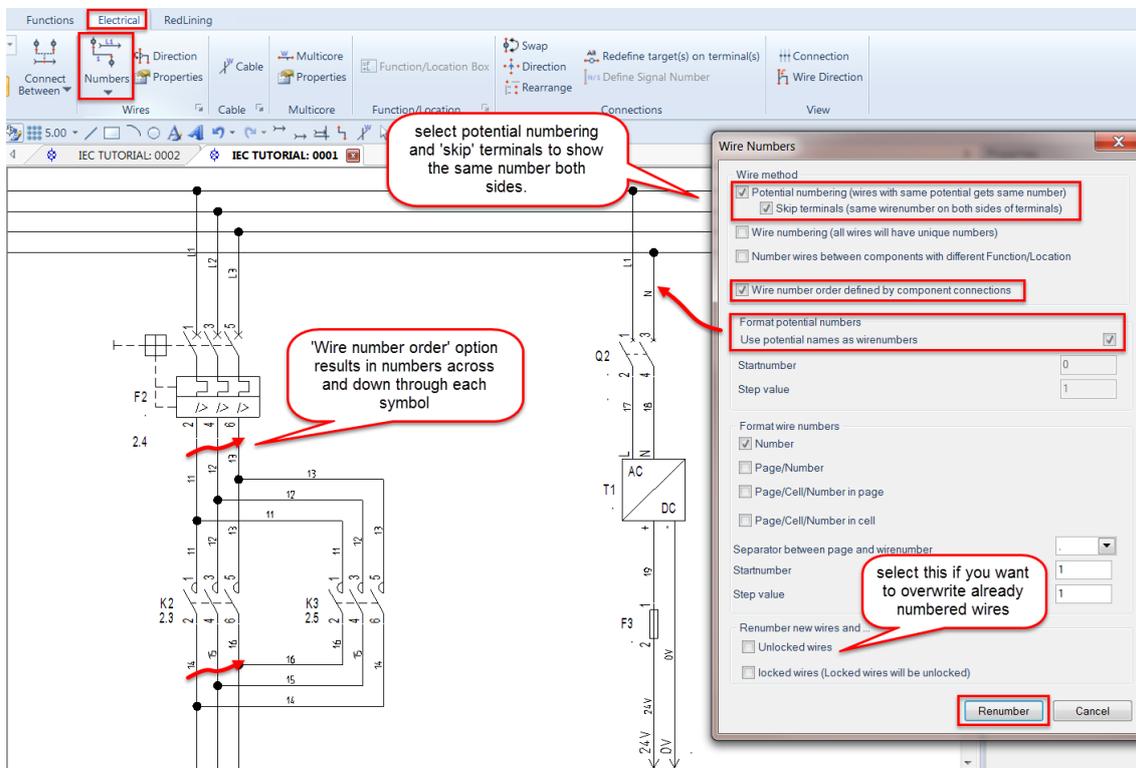
The final section of the schematic shows indicator lamps used to display the status of the 2 motors.

Again, use the standard terminals, cables, and wire commands to draw this section of the circuit. Try drawing one column and copying the wire and symbols across to speed up the process.

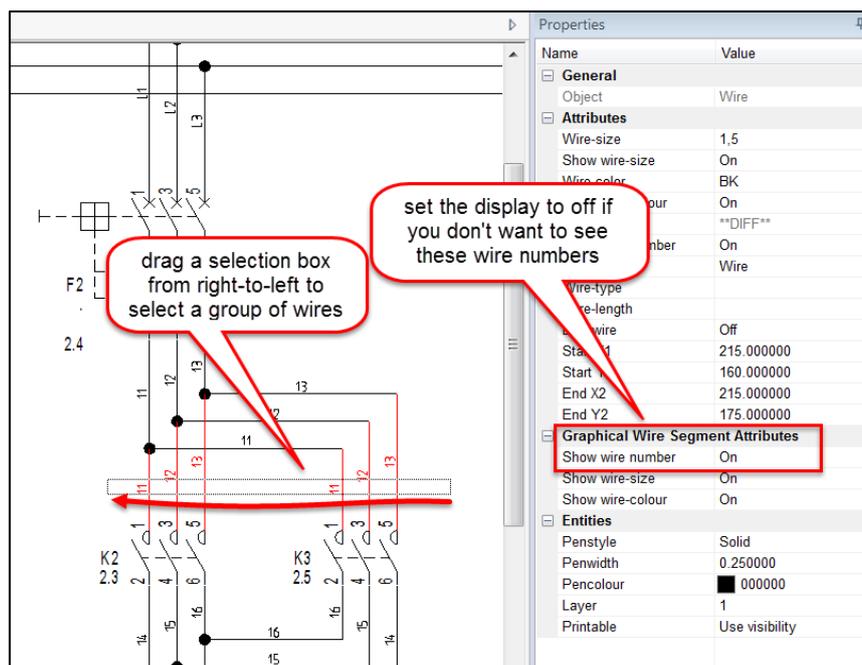


## Auto Assign Wire Numbers (Standard and Advanced)

In the basic version of See Electrical wire numbers can be assigned by double-clicking on a wire and entering a wire number. In the standard and advanced versions the numbers can be assigned automatically using the following command 'Electrical > Wires > Numbers'.



Groups of wires can still be selected after numbering to control the display of the wire number on each segment of wire.



## Producing Manufacturing Reports

Since See Electrical is based around a database for each workspace it is now very quick and easy to provide a full list of devices, terminals, and cables which have been used in the project.

Extend the 'Graphical lists' section of the 'Workspace' panel and right-click over the Product list to 'Generate' a graphical report.

extend the 'Graphical lists' section and right-click over the 'Product list' to select 'Generate'

### List of Products

Product (-)	Type	Description
Q1	.	.
M1	.	MAIN PUMP
F1	.	.
F2	.	.
M2	.	REVERSING MOTOR

All symbols placed in the project could have part numbers assigned (Type) and Descriptions to clarify the Product list.

Repeat the same for the terminal list and cable-core lists.

### List of Terminals: X1

Connection 1	Terminal	Connection 2	Type	Comment
Q12	L1	.	.	
Q14	L2	.	.	
Q16	L3	.	.	
Q1N	N	.	.	
PE	PE	.	.	
K12	1	M1U1	.	
K14	2	M1V1	.	
K16	3	M1W1	.	
PE	PE	M1	.	
K22	4	M2U1	.	

### List of Cable Cores

Cablename (-)	Wire	Square	Colour	Description	Connection 1	Connection 2	Sheet
W1	1	.	GY	.	X11	M1U1	
W1	2	.	BK	.	X12	M1V1	
W1	3	.	BN	.	X13	M1W1	
W1	4	.	GNYE	.	X1PE	M1	
W2	1	.	GY	.	X14	M2U1	
W2	2	.	BK	.	X15	M2V1	
W2	3	.	BN	.	X16	M2W1	
W2	4	.	GNYE	.	X1PE	M2	
VG	1	.	.	.	X17	S13	
VG	2	.	.	.	X18	S14	

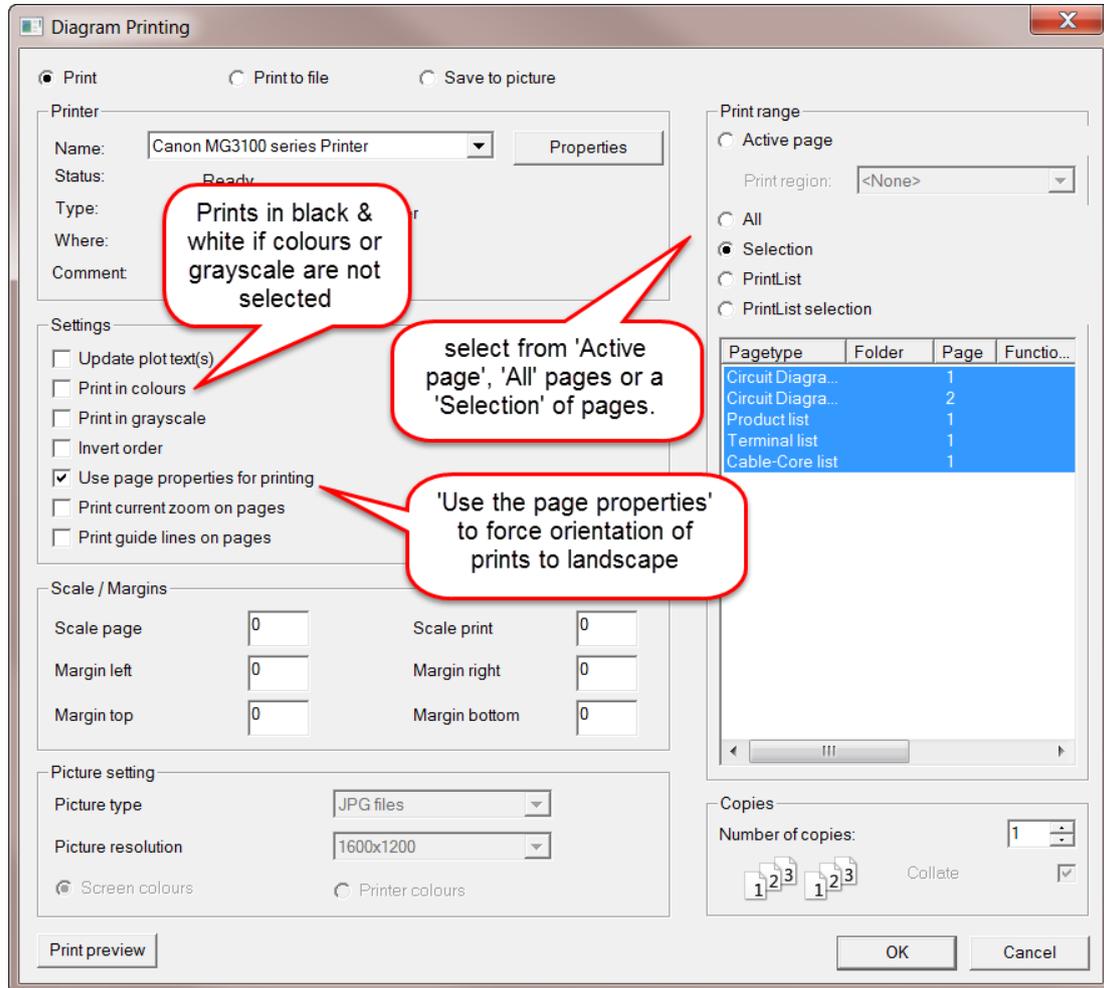
Also be aware that all report templates can be customized to control the page template, columns displayed and the format of the text to achieve clear and accurate documentation from the schematics.

There are many more reports available and most offer a Database Viewer section where information can be copied and pasted into Microsoft Excel for additional handling and formatting.

## Printing the Workspace

The circuit diagrams and graphical lists can now be printed, either individually, in selected groups, or as a whole document.

Use 'File > Print' command to display the printing options.



Select where to print (which printer, to a file or picture), select which pages to print, and then select the options to decide page orientation and output colour.

**Hint:** The options selected and used here will also be applied to outputting PDF documents using the PDF Publisher module.