

**Electra™ 2005**

**User Manual**





## Copyrights, Trademarks and Disclaimer

Microsoft, Office, Excel, Visio and Windows are registered trademarks of Microsoft Corporation. All other trademarks and service marks are the property of their respective owners.

Use of the Electra™ 2005 software (the "Software") and its documentation are governed by the terms set forth in the End User License Agreement. Such use is at your sole risk. The Software and its documentation (including this manual) are provided "AS IS" and without warranties of any kind and RADICA SOFTWARE (HEREINAFTER REFERRED TO AS "RADICA") EXPRESSLY DISCLAIM ALL WARRANTIES, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE AND AGAINST INFRINGEMENT. RADICA DOES NOT WARRANT THAT THE FUNCTIONS CONTAINED IN THE SOFTWARE WILL MEET YOUR REQUIREMENTS, OR THAT THE OPERATION OF THE SOFTWARE WILL BE UNINTERRUPTED OR ERROR-FREE, OR FURTHERMORE, RADICA DOES NOT WARRANT OR MAKE ANY REPRESENTATIONS REGARDING THE USE OR THE RESULTS OF THE USE OF THE SOFTWARE OR ITS DOCUMENTATION IN TERMS OF THEIR CORRECTNESS, ACCURACY, RELIABILITY, OR OTHERWISE. NO ORAL OR WRITTEN INFORMATION OR ADVICE GIVEN BY RADICA OR A RADICA AUTHORIZED REPRESENTATIVE SHALL CREATE A WARRANTY OR IN ANY WAY INCREASE THE SCOPE OF THIS WARRANTY.

UNDER NO CIRCUMSTANCES INCLUDING NEGLIGENCE, SHALL RADICA, ITS LICENSORS OR THEIR DIRECTORS, OFFICERS, EMPLOYEES OR AGENTS BE LIABLE FOR ANY INCIDENTAL, SPECIAL OR CONSEQUENTIAL DAMAGES (INCLUDING DAMAGES FOR LOSS OF BUSINESS, LOSS OF PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION AND THE LIKE) ARISING OUT OF THE USE OR INABILITY TO USE THE SOFTWARE OR ITS DOCUMENTATION, EVEN IF RADICA OR A RADICA AUTHORIZED REPRESENTATIVE HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

In no event shall RADICA's total liability to you for all damages, losses, and causes of action (whether in contract, tort, including negligence, or otherwise) exceed the amount paid for the Software and its documentation.

Copyright© 2004, Radica Software. All Rights Reserved.

Email: [support@RadicaSoftware.com](mailto:support@RadicaSoftware.com)

Website: [www.RadicaSoftware.com](http://www.RadicaSoftware.com)

No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording or otherwise without the prior permission in writing of the copyright owner.

## Table of Contents

1.	System Requirements .....	5
2.	Installing Electra™ 2005 .....	6
3.	License Dongle.....	7
4.	Registering Electra™ 2005.....	8
5.	Getting Started .....	9
5.1.	Starting Electra™ 2005.....	9
5.2.	Saving a drawing .....	9
5.3.	Page Scale .....	9
5.4.	Page measurement units .....	10
5.5.	Stencils and drawing area .....	10
5.6.	Shapes .....	11
5.7.	Connection points .....	11
5.8.	Moving, sizing and transforming shapes.....	12
5.9.	Selecting shapes in groups.....	12
6.	Document Options.....	13
6.1.	Popup Reference window on drop .....	13
6.2.	Enable Page Prefix on References .....	13
6.3.	Enable Page Prefix on Nets (Wires) .....	13
6.4.	Page Prefix Separator for nets .....	14
6.5.	How Page Prefix works .....	14
6.6.	Location zone options.....	14
6.7.	Saving Document Options.....	15
7.	Placing a schematic symbol .....	16
7.1.	About Electra™ shapes .....	16
7.2.	Placing a schematic symbol.....	16
7.3.	Editing symbol reference .....	16
7.4.	The Reference window.....	17
7.5.	Symbol rotation and manipulation.....	18
8.	Placing a net (wire) .....	20
8.1.	Nets in Electra™ .....	20
8.2.	Placing a net (wire).....	20
8.3.	Editing net name .....	20
8.4.	The Net Name window.....	21
8.5.	Automatic status detection.....	22
8.6.	Automatic net name detection .....	22
8.7.	Automatic net propagation.....	22
8.8.	Showing, hiding and positioning net names .....	23
8.9.	Formatting line ends .....	23
8.10.	Difference between a normal line and a net (wire) .....	24
9.	Duplicating shapes and circuits .....	25
9.1.	3 methods to duplicate a shape .....	25
9.2.	Automatic renaming of schematic symbols.....	25
9.3.	Automatic renaming of nets .....	26
9.4.	Duplicating and reusing circuits .....	26
10.	Intelligent Terminals .....	27
10.1.	Automatic net detection .....	27
10.2.	Intelligent Magnetic Technology .....	27

10.3.	Generating a terminal list automatically .....	27
10.4.	Automatic terminal name synchronization .....	28
10.5.	Terminal Blocks .....	28
11.	The Location shape .....	30
12.	The PLC Module shape .....	31
12.1.	Customizing the PLC module shape .....	31
13.	Panel Layout .....	34
13.1.	The Panel Layout Process .....	34
13.2.	Preparing a page for panel layout.....	34
13.3.	Inserting a plate .....	35
13.4.	Inserting cable ducts and rails .....	35
13.5.	Dimension shapes .....	35
13.6.	Layout components .....	35
13.7.	Automatically generate layout components .....	36
13.8.	The GenerateLayout window.....	36
13.9.	Using guides to place components.....	41
13.10.	How GenerateLayout matches symbols to components .....	42
14.	Component Manager .....	44
14.1.	About component manager .....	44
14.2.	The ComponentManager window .....	44
15.	Generating Reports.....	46
15.1.	Generating Cross Reference report.....	46
15.2.	Generating Material Reference report .....	46
15.3.	Generating Bill of Materials (BOM) report .....	47
15.4.	Exporting reports to Excel® .....	47
16.	Creating your own schematic symbol .....	48
16.1.	Symbol creation process .....	48
16.2.	Drawing and combining a symbol.....	48
16.3.	Adding connection points .....	49
16.4.	The CreateSymbol window .....	49
16.5.	Editing the symbol icon.....	54
16.6.	The Stencil Browser window .....	54
17.	Creating your own layout shape .....	55
17.1.	Layout shape creation process .....	55
17.2.	Drawing or combining a layout shape .....	55
17.3.	The CreateLayoutShape window .....	56
17.4.	Editing the layout shape icon.....	57
18.	Creating your own title blocks .....	58
18.1.	Title Block creation process .....	58
18.2.	Drawing and grouping a title block .....	59
18.3.	The CreateTitleBlock window .....	60
18.4.	Editing the title block shape icon.....	60
19.	Appendix A : Symbol Listing .....	61

## 1. System Requirements

Radica Electra™ 2005 requires:

- Microsoft® Visio® 2000 or later.
- Works with all versions of Visio® including standard, professional, technical and academic editions.
- Works with Visio® on all platforms including Win 98, Win ME, Win 2000, Win XP.

Radica Electra™ uses macros in Visio®. If your installed copy of Visio® is set to High or Medium security, you may be prompted for permission to run a macro every time Electra™ is activated or have Electra™ disabled altogether. To enable macros and Electra™:

1. Start Visio®.
2. Select from menu Tools | Macros | Security (does not apply to Visio® 2000).
3. On the Security Level tab, click and select low security.

## 2. Installing Electra™ 2005

Radica Electra™ is a Microsoft® Visio® solution and requires Microsoft® Visio® to operate. It is strongly recommended that you install Visio® *before* installing Electra™ 2005.

To install Visio®, follow the instructions listed on the Visio® documentation.

To install Electra™ 2005:

1. Exit any applications you are running.
2. Insert Electra™ 2005 CD into your CD-ROM drive.
3. The Windows® 98/ME/XP/NT auto play feature will cause the Electra™ 2005 installer to launch. If the installer does not launch, select the Start menu, choose Run and enter CD-ROM drive letter, colon, backslash and setup.exe (e.g. E:\Setup.exe).
4. Read the installer warning regarding closing any applications that you may be running and click OK.
5. Read the End User License Agreement and if you agree to the terms, click Accept.
6. Click on the installation button to begin installing.
7. Restart Windows.

*Note: Do not connect License Dongle to your computer during installation.*

### **3. License Dongle**

Before using Electra™ 2005, ensure that the License Dongle supplied in the package is plugged into your computer's printer port (LPT).

If you have a printer connected to your computer, connect your printer to the License Dongle.

The License Dongle must be plugged into your computer's printer port (LPT) throughout the duration in which you are using Electra™.

## **4. Registering Electra™ 2005**

Why should you register?

Your registration enables us to keep you updated periodically when software patches, symbols and applications are available. Registration also enables you to pose questions, report a bug or suggest a feature for consideration in our next release.

To register your copy of Electra™ 2005, point your browser to :

<http://www.RadicaSoftware.com/Support.asp>.


## 5. Getting Started

### 5.1. Starting Electra™ 2005

To start a new drawing:

- Click on the Windows Start button.
- Select Programs | Radica Software | Electra Template 2005.


Or

- Start Microsoft® Visio®.
- Click on the drop down arrow on the new drawing icon (  ) and select Electra Template 2005.

A copy of the template is then created for your new drawing, leaving the template unmodified for further usage.

*Note: For non-electrical drawings, you can use Visio® without the License Dongle and Electra™ disabled. To activate and use Electra™ for electrical drawings, you must create a new drawing using the Electra™ 2005 template.*

### 5.2. Saving a drawing

1. Click on the menu File | Save or click on the save icon  on the tool bar.
2. Type in your drawing file name.

### 5.3. Page Scale

In Visio®, each and every page can be set to different scales. On the Electra™ template, the page is set to 1:1 scale, which is **optimized** for schematic drawing and produces an excellent printed copy. During Panel Layout, you may need to apply a different scale.

To set the page scale:

1. Select menu File | Page Setup.
2. Click on the Drawing Scale tab.
3. Click Pre-defined scale or Custom scale.
4. Select or key in your desired scale.

Our studies show that a scale of 1:5 is suitable for medium size panels and for bigger panels, a scale of 1:10 may be necessary.

## 5.4. Page measurement units

Each and every page can also have its own measurement units. When the measurement units of a page is modified, the rulers and grid on the page will also be modified to reflect the new measurement units.

The measurement units on the Electra™ template are set to inches and are optimized for schematic drawings. During Panel Layout, you may need to apply a different measurement unit to your page.

To set the measurement units:

1. Select menu File | Page Setup.
2. Click on the Page Properties tab.
3. Click Measurement units and select the desired measurement units.

We recommend setting page measurement units to millimeters during panel layout drawings.

## 5.5. Stencils and drawing area

On the left of the screen are a list of stencils, which contains a collection of shapes that you can drag and drop onto your drawings. The shapes on a stencil are called Master shapes or Masters.

Stencils can be configured to display icons, icons and names or names only by clicking on the stencil icon. (*Refer to Online Electra™ Help for more information*)

You can also create your own custom stencils and masters to be placed in stencils. Once a master is dragged from a stencil to a drawing, Visio® automatically creates a copy of the master in your drawing for **portability**. This allows you to send your drawings to your customers, vendors and colleagues **without any loss of information**. They will be **able to use and modify** your drawings even though they might not have your custom stencils and Electra™ installed (without intelligence and productivity tools).

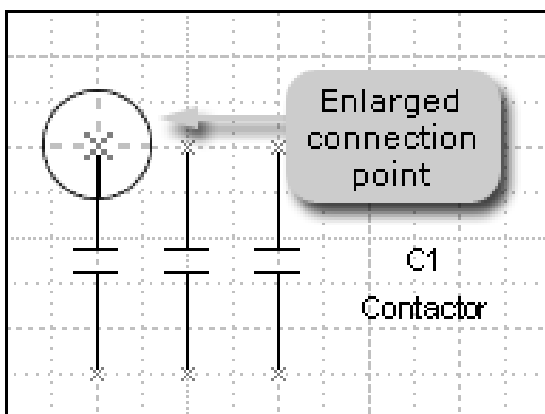
## 5.6. Shapes

Everything in Visio® are shapes, including graphics (lines, circles, etc.) and text. In Electra™, we have schematic symbols, nets (wires) and other special shapes that have intelligence. Shapes can also be grouped together for easier handling (e.g. rotation). However, grouping shapes will incur an overhead, so do not group shapes unnecessarily. Alternatively, you can group shapes, apply your operations and then ungroup them.

*In order to use the intelligence and tools on Electra™, do not group symbols, nets and other Electra™ shapes.*


## 5.7. Connection points



Most shapes in Electra™ have connections points to allow one shape to connect intelligently to another (e.g. connecting a net (wire) to a symbol). Although they are visible in the drawing, they are not printable, meaning they will not appear on the printed hard copy. They also act as glue points to allow easy shape placements.



Nets (wires) in Electra™ comes with strategically placed connection points to enable users to easily connect a net to another shape.

In addition, you can easily add connection points to any shape (nets and symbols) by:

1. Click on the connection point tool .
2. Click on the shape that you intended to add connection points.
3. Press Ctrl (Control) key and click on the position you want to place the connection point.

The connection point tool is next to the pointer tool , under the connector tool .

## 5.8. Moving, sizing and transforming shapes

To move a shape:

1. Click on the shape and drag to desired position.

To resize a shape:

1. Click on the shape and drag on the shape's width or height handles.

To transform or rotate a shape (Method 1):

1. Right click on the shape.
2. Select Shape | Rotate Left or Rotate Right.

To transform or rotate a shape (Method 2):

1. Click on the shape.
2. Click the transform tool (  for text,  for shapes).
3. Drag on the shape's handles.

*See Online Electra™ Help for more information.*

## 5.9. Selecting shapes in groups

Individual shapes in a group can be individually selected and formatted. Click on the group, and then click on the individual shape (not double click). Applying formats (e.g. font, alignment) to the group will affect all shapes in the group while applying formats to an individual shape in a group will only affect the individual shape.

## 6. Document Options

The Electra™ template provides the user with drawing options that can be accessed by selecting from the menu Electra | Document Options.

### 6.1. Popup Reference window on drop

To enable the Reference window to popup every time a new symbol is added to your drawings, check the "Popup Reference Window on drop" checkbox.

For faster schematic generation, uncheck the "Popup Reference window on drop" check box. The Reference window will still be accessible by right clicking on a symbol and selecting Edit Reference even when automatic popup has been disabled.

### 6.2. Enable Page Prefix on References

You can select to prefix a page number on all references in your drawings by enabling the "Use Page Prefix on References" check box. Electra™ will automatically insert and maintain all page prefixes.

This option is also available on the Reference window.

*See also How Page Prefix works.*

### 6.3. Enable Page Prefix on Nets (Wires)

You can also select to use page prefix for wires that you place on your drawings by enabling the "User Page Prefix on Nets" check box. Electra™ will automatically insert and maintain all page prefixes.

This option is also available on the Net Name window.

*See also How Page Prefix works.*

## 6.4. Page Prefix Separator for nets

The default separator for nets is the "." character (e.g. 1.100). Page prefix separator for nets can be any character but cannot be numeric (e.g. 0-9) and can consist of more than one character.

Separator	Examples
.	1.100, 2.100, 1.101
P	1P100, 2P100, 1P101
/	1/100, 2/100, 1/101
Pg	1Pg100, 2Pg100, 1Pg101

## 6.5. How Page Prefix works

Page prefix in Electra™ is powerful and automatic. Once enabled, Electra™ will monitor page changes and automatically updates all pages prefixes on your symbols and nets.

A symbol (e.g. 1C1 on drawing page 2) may be referring to another symbol on another page or, a symbol (e.g. 2C1 on page 2) may be referring to the current page. In both cases, Electra™ makes it easy for you to re-sequence and move your pages, by dragging on the page tab. When the page that is referred to, has moved, Electra™ **automatically updates** all your page prefixes **without human intervention**. Just sit back and watch.

*See Online Electra™ Help for more information.*

## 6.6. Location zone options

Electra™ reports shape locations (e.g. P1.A1 - Page 1 - zone A1) by dividing your drawing page into zones. To modify the location zones throughout your document, click on the Location tab on the Document Options window and enter your preferred zone spacing.

*See Online Electra™ Help for more information.*

## 6.7. Saving Document Options

When you change the settings in Document Options, the settings will apply throughout the document (drawing file) and will be saved on the drawing file. If you wanted to set these settings as the default settings every time Electra™ is started, use the following steps:

1. Make a backup copy of the Electra Template 2005.
2. Close Visio®.
3. Restart Visio®.
4. Select Electra Template 2005.
5. Select menu Electra | Document Options.
6. Apply your settings and click OK.
7. Do not modify the drawing, and select menu Save As.
8. Type in or select Electra Template 2005 to overwrite the Electra template.
9. Close your current drawing (Electra Template 2005).

The settings are now saved into the template. The next time you start Visio® and select Electra Template 2005, your settings will be used as the default.

## 7. Placing a schematic symbol

### 7.1. About Electra™ shapes

Shapes on the stencils that comes with Electra™ (e.g. schematic symbols, nets, layout shapes, report shapes) are special shapes that have intelligence and is different from normal Visio® shapes. Normal Visio® shapes will not be processed and monitored by Electra™.

You can easily create your own schematic symbols or layout shape, run Create Symbol or Create Layout Shape (located on the menu under Electra) and have the intelligence and data automatically inserted into your custom symbol and layout shapes. After processing by Create Symbol or Create Layout Shape, Electra™ will be able to process and recognize your custom created symbols and layout shapes.

### 7.2. Placing a schematic symbol

To add and place a schematic symbol onto your drawing:

1. Click on the stencil.
2. Select from the icons, the shape you require.
3. Click on the icon and drag to the require position on your drawing.

### 7.3. Editing symbol reference

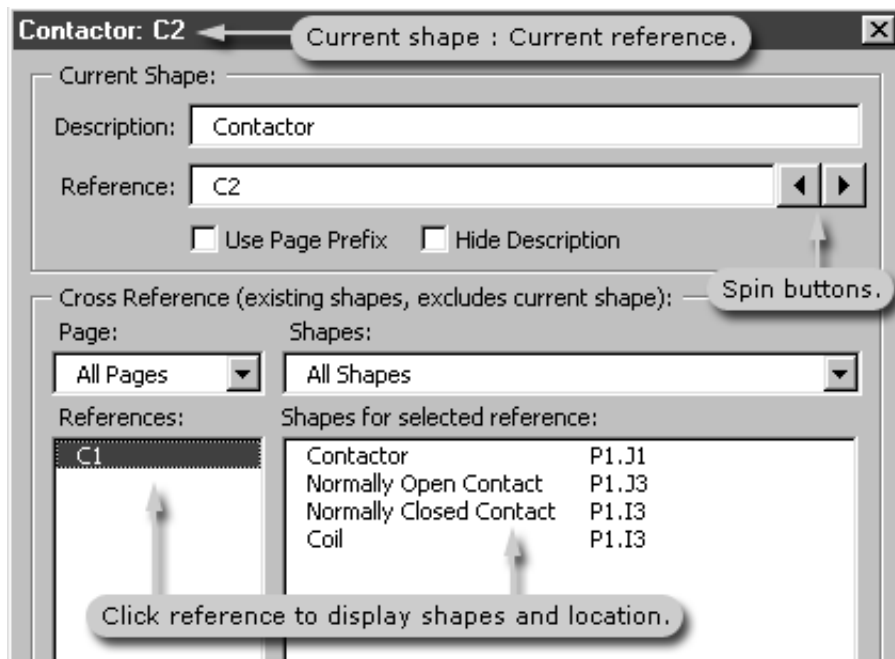
There are 2 ways to edit a reference:

1. Direct Edit:
  - 1.1. Click on the shape and type in the reference directly.
2. Using the Reference window:
  - 2.1. Right click on the shape.
  - 2.2. Select Edit Reference.
  - 2.3. Use the Reference window to edit your reference.

## 7.4. The Reference window

The Reference window can be accessed by:

1. Right click on a schematic symbol.
2. Select Edit Reference.



### Current shape : Current Reference

The current shape and current reference will be displayed on the title bar.

### Reference

Edit your symbol reference in this text box.

### Description

Symbol description, e.g. Start, Stop, Motor 1 4.7kW

### Spin buttons

The Reference window is designed for **minimal typing**. Use the spin buttons to increment or decrement your references.

### Use Page Prefix (applies document wide)

To use page prefix on your symbol, check this check box. Selecting this option is the same as selecting the option in Document Options.

### Hide Description

To hide the symbol description on your drawing check this check box.

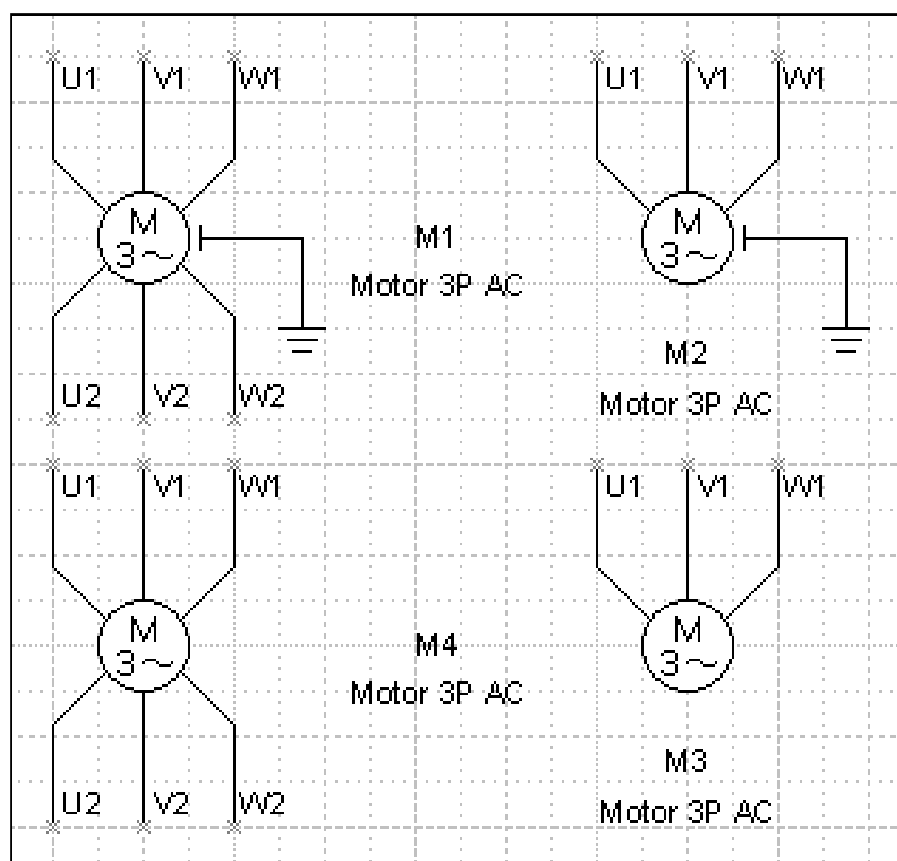
### Cross Reference Information

Included on the Reference window are real time cross reference information on shapes that you have already placed on your drawing. Click on the reference to display shapes for the reference (e.g. a reference may have more than one shape, coil and contacts) and their location (e.g. P1.J1 - Page 1, location J1).

**Tip:** On many occasions, you may want to use a reference that already exists on your drawing (e.g. inserting another contact for C1). Double click on C1 in the References list box and the Reference text box will display C1, and then press OK.

### 7.5. Symbol rotation and manipulation.

Most shapes in Electra™ can be quickly and easily manipulated by right clicking on the shape, and selecting horizontal or vertical. Some shapes have even more intelligence, for example, the Motor symbol can be used to display a 3 terminal or 6 terminal motor.



The reference and description text position will be automatically moved to the correct position when using right click.

In addition, you can also manually size and place the reference text using the text transform tool.

To manually size and place the description:

1. Click on the symbol.
2. Click on the description text.
3. Right click on the description text.
4. Select Unlock Text Guard.
5. Drag on the description text handles to size and move.

## 8. Placing a net (wire)

### 8.1. Nets in Electra™

Electra™ provides 5 types of nets for every conceivable type of connection. Each of them can be easily stretched and manipulated in a variety of ways by dragging on their handles.



### 8.2. Placing a net (wire)

To add and place a net onto your drawing:

1. Click on the stencil.
2. Select the net that you require.
3. Click on the icon and drag to the required position on your drawing.

### 8.3. Editing net name

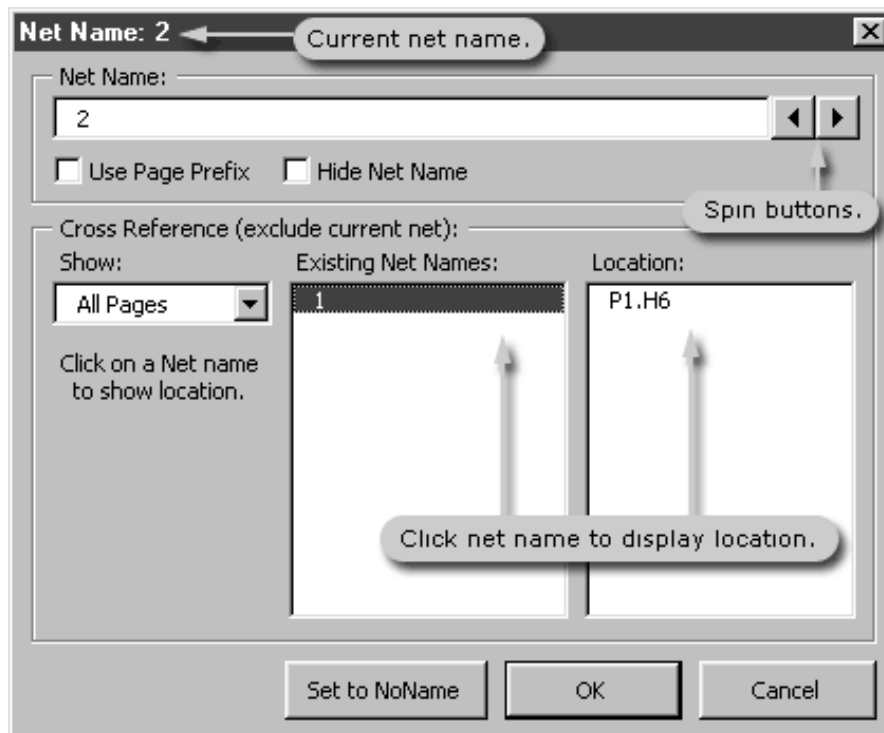
There are 2 ways to edit the net name:

1. Direct Edit:
  - 1.1. Click on the net.
  - 1.2. Type in new net name.
2. Using the Net Name window:
  - 2.1. Right click on the net.
  - 2.2. Select Edit Net Name.
  - 2.3. Use the Net Name window to edit your net name.

## 8.4. The Net Name window

To access the net name window:

1. Right click on a net.
2. Select Edit Net Name.



### Current net name

The current net name will be displayed on the title bar.

### Net Name

Edit your net name in this text box.

### Spin buttons

The Net Name window is designed for **minimal typing**. Use the spin button to increment or decrement your net name.

### Use Page Prefix (applies document wide)

To use page prefix on your net, check this check box. Selecting this option is the same as selecting the option in DocumentOptions.

### Hide Description

To hide the net name check this check box.

### Set to NoName button

If you do not want to assign a net name to a net, click on this button and the net will be assigned "NoName".

### Cross Reference Information

Included on the Net Name window are real time cross reference information on nets that you have already placed on your drawing. Click on a net name to display its location (e.g. P1.J1 - Page 1, location J1).

**Tip:** To set the current net name to a net name that already existed in your drawing, double click on a net name in the Existing Net Names list, and click OK.

### **8.5. Automatic status detection**

Nets in Electra™ will automatically be displayed in red when it does not have a net name (orphaned). When a net name is assigned to the net, the net will be displayed as black.

*See Online Electra™ Help for more information.*

### **8.6. Automatic net name detection**

When a net that has no net name (red or orphaned) is dragged and moved to connect to another net, Electra™ will automatically detect the new net name and assign the new net name to the orphaned net. Connected nets will have the handle displayed in red.

*See Online Electra™ Help for more information.*

### **8.7. Automatic net propagation**

If you have multiple nets connected together, changing the net name for the top most net will also changes the net name for all the nets that is connected to the top most net.

*See Online Electra™ Help for more information.*

## 8.8. Showing, hiding and positioning net names

To show or hide net names:

1. Right click on the net.
2. Select Show or Hide net name.

To position net names:

1. Right click on the net.
2. Select Net Name | Right. (Left/Middle/Top/Bottom)

You can also manually position the net name by selecting the net and using the text transform tool.

## 8.9. Formatting line ends

Nets can be formatted to show a variety of line ends, including dots, arrows and many others.

To place a connecting dot on a net:

1. Right click on a net.
2. Select No Dot, Dot - Begin, Dot - End or both.

To place an arrow on a net:

1. Right click on a net.
2. Select Format | Line.
3. On the Ends drop down, select 4.

To format line ends for multiple nets:

1. Select multiple nets.
2. Right click on any selected net.
3. Select Format | Line.

**8.10. Difference between a normal line and a net (wire)**

	Normal Line	Electra™ Nets (wires)
Automatic status detection (turns red)	No	Yes
Right click horizontal/vertical menu	No	Yes
Right click dots menu	No	Yes
Automatic net detection during connection	No	Yes
Automatic net name propagation	No	Yes
Location information	No	Yes
Detected by terminals	No	Yes

Nets (wires) in Electra™ have intelligence that helps you generate your schematic faster compared to the normal line. When intending to put a wire on your drawing, always use a net.

## 9. Duplicating shapes and circuits

### 9.1. 3 methods to duplicate a shape

#### Method 1 (Visio® 2000 only):

Right click on a shape, select duplicate.

#### Method 2:

Click on a shape, click on copy tool, click on paste tool.

#### Method 3:

Press Ctrl (Control) key, drag from source shape (the shape you want to duplicate from) and place target (the shape you're duplicating) at the required position, release mouse button, then release Ctrl key.

Which method shall I use?

Our research shows that duplicating a shape from an existing one is the **easiest and fastest** way to generate your drawings. When a shape you wanted to duplicate is not within your view in the current window, we recommend dragging a shape from the stencils. When a shape is within view in the current window, we recommend using the Ctrl key to duplicate your shapes. When you duplicate a schematic symbol or a net (wire), they will be automatically renamed.

### 9.2. Automatic renaming of schematic symbols

Create a new schematic symbol by dragging from a stencil or by duplication and Electra™ will **automatically and intelligently rename** the new schematic symbol for you. If you drag a symbol from a stencil, Electra™ will always rename the symbol by using its default reference. Alternatively, when you duplicate a symbol, Electra™ will rename using your existing reference.

*See Online Electra™ Help for more information.*

### 9.3. Automatic renaming of nets

Create a new net by dragging from a stencil or by duplicating another net and Electra™ will **automatically and intelligently rename** your net for you. If you drag a net from a stencil, Electra™ will search through its internal sorted list and rename the new net to the next sequential number. When you create a new net by duplicating an existing one, Electra™ will automatically rename the new net to the next sequential number using the existing net name.

*See Online Electra™ Help for more information.*

### 9.4. Duplicating and reusing circuits

One of the most **powerful and useful** tool on Electra™ is its intelligent duplication of entire circuits. In most schematic drawings, there are similar circuits that is duplicated throughout the drawing (e.g. motor starter and control circuits). All references and nets on these circuits must be correctly and uniquely numbered. The issue is further complicated by the fact that you may want certain symbols renumbered while others left unchanged (e.g. a common safety relay on motor control circuits should not be renumbered).

Electra™ solves all this complexity by allowing you to choose nets and references that should be left unchanged and automatically and intelligently renumbering nets and references that you want to be renumbered through its AutoRename window.

To duplicate and reuse your circuits:

1. Select all the shapes in your circuit.
2. Click on the copy tool.
3. Click on the paste tool.
4. Click Yes on the automatic renaming dialog box.
5. Select/Deselect symbols and nets to be renamed.
6. Click OK.
7. Drag your new circuit to the desired position.

*See Online Electra™ Help for more information and example.*

## 10. Intelligent Terminals

### 10.1. Automatic net detection

Terminal symbols are always placed on a net. In Electra™, the terminal symbol will be displayed as red when it is not placed on a net. When placed on a net, it will **automatically detect** the underlying net and will be displayed as black.

### 10.2. Intelligent Magnetic Technology

Once a terminal symbol is placed on a net, the terminal will turn on its Intelligent Magnetic Technology (IMT) and follows the net like a magnet when the net is moved or resized. To turn off IMT, just drag the terminal symbol away from the net until the terminal is displayed as red.

### 10.3. Generating a terminal list automatically

After placing all required terminals on your drawing, you can easily generate a terminal list. To generate a terminal list:

1. Drag and place a TerminalList shape from the stencil.
2. Right click on the TerminalList shape, select Generate Listing.
3. Select Generate for all pages or this page only.
4. Click Generate.
5. Delete the first TerminalList shape that you dragged from the stencil.

If there are any addition and deletion of terminal symbols after generating your terminal list, simply right click on any TerminalList shape and generate again. Electra™ will **automatically synchronize** (create or delete) TerminalList shape based on your modifications. This synchronization of your terminal symbols and TerminalList shape applies only to TerminalList shape that is created by Electra™ and does not apply to manually created TerminalList.

To manually create TerminalList, drag a TerminalList shape from the stencil or duplicate from your existing drawings. To edit the name of a manually created TerminalList, simply select the TerminalList and type in the name.


To create spare TerminalList that is empty (no name) click on a TerminalList and press F2, then press delete. Alternatively, you can also name your spare TerminalList as "spare" by selecting the Terminal list and typing "spare".

## 10.4. Automatic terminal name synchronization

Should you modify any net name in your drawings that has a terminal symbol attached, Electra™ will **automatically update and synchronize** the TerminalList shape. This ensures that the TerminalList shape is always showing the correct net name.

## 10.5. Terminal Blocks

Terminals are normally handled as blocks. You may have one row of higher rated terminals and other rows of lower rated terminals. Electra™ enables a mix of terminals by providing the TerminalBlock shape. After generating your terminal list:

1. Drag a TerminalBlock  shape onto your drawing.
2. Drag and resize the TerminalBlock shape on your terminal list and ensure that it encapsulate all your TerminalList shape.

TerminalBlock shapes will **automatically calculate** the quantity of TerminalList shape that it encapsulates.

When you have newly added TerminalList shapes, click on the TerminalBlock shape, drag and resize using the handles and the TerminalBlock shape will automatically re-calculate your TerminalList. Alternatively, you can manually re-calculate by right clicking on the TerminalBlock shape and selecting Count Terminals.

The TerminalBlock shape is also used during panel layout to generate a terminal block component. To properly identify your TerminalBlock shape and edit the TerminalBlock reference, click on the TerminalBlock shape and type in the reference (e.g. X1, X2, X3).


All TerminalBlock shapes have custom properties that describe the terminals in the block. To set the custom properties for a TerminalBlock shape, right click on the TerminalBlock shape and select Edit Terminal Block.

<b>Fields</b>	<b>Default value</b>	<b>Description</b>
Terminal Reference	X1	Used as a reference to identify a terminal block.
Terminal Width	10 mm.	The width of each terminal in the block. Accepts major units (e.g. mm, in, cm).
Terminal Height	50 mm.	The height of each terminal in the block. Accepts major units (e.g. mm, in, cm).
Reference position	Left	Other values: Right, Top, Bottom. Select the reference text position.
Quantity position	Right	Other values: Left, Top Bottom. Select the quantity text position.
Name	Terminal	The component name to use during panel layout. (Do not change.)
Description		The component description during panel layout and reports. Type in a description of the component (e.g. Power Terminals).
Brand		The brand of the terminals. Used during panel layout and reports.
Rating		The rating of the terminals. Used during panel layout and reports.
Model		The model of the terminals. Used during panel layout and reports.
Order		The terminal order number. Used during panel layout and reports.
Supplier		The terminal supplier. Used during panel layout and reports.
Remarks		Any remarks you may have. Used during panel layout and reports.
Price		Price for each of the terminal in the block. Used during panel layout and reports.

When you drag a TerminalBlock shape from the stencil, the TerminalBlock shape will be populated with default values. When you duplicate a Terminal Block shape from an existing one, it will inherit all the properties from the shape that it is copied from. For example, you may use terminals that is 15mm wide. Drag a TerminalBlock shape from the stencil, set the width property to 15mm, duplicate 3 more TerminalBlock shapes and all of them will inherit the properties and have the Terminal Width property set at 15mm. Alternatively, TerminalBlock properties can also be accessed by using the GenerateLayout window.

## 11.The Location shape

To locate your symbols over multiple pages, use the location shape:

1. Drag the location  shape from the stencil.
2. Right click on the location shape and select Locate Reference.
3. Type in the reference to locate and click OK.

The location shape will automatically generate the location of all shapes having the keyed in reference, enabling you to easily locate your symbols.

All Location shapes have hyperlinks that allow you to quickly jump to the shape. Right click on the location shape, and select "Jump to ...", to jump to the shape.

## 12.The PLC Module shape

To create a PLC module in your drawings, use the PLC module shape.

To rotate the PLC module shape:

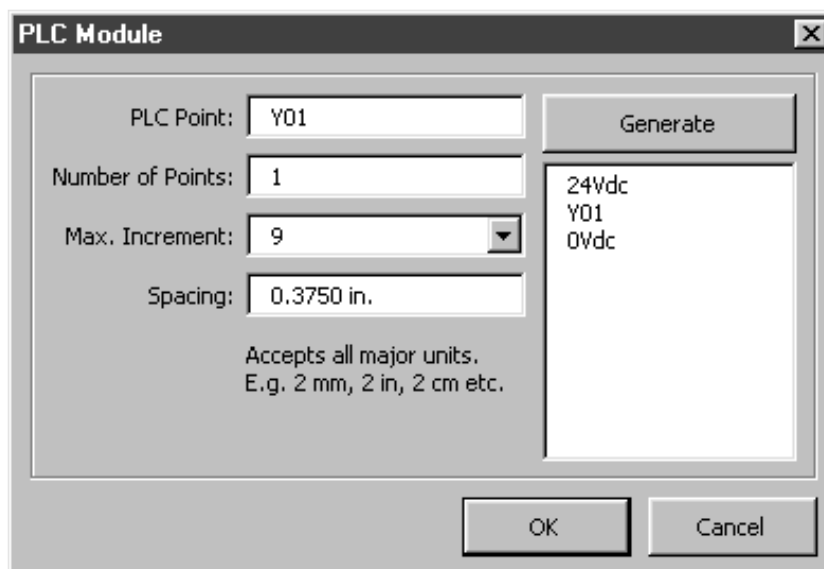
1. Right click on the PLC module shape.
2. Select Horizontal or Vertical.

To change the label position and orientation on the PLC module shape:

1. Right click on the PLC module shape.
2. Select Change Label position or Change Label orientation.

### 12.1. Customizing the PLC module shape

All labels on the PLC Module shape can be customized by right clicking on the PLC module shape and selecting Set PLC Module.



#### PLC Point

PLC modules can be input, output or analog modules. To customize the PLC module, type in a starting PLC point into the PLC point text box.

Example:

If your PLC module is an input module and starts from B00000 to B00007, type in B00000 into PLC point text box.

### Number of Points

Type in the number of points for your PLC module into the text box.

### Maximum Increment

Click on the Maximum Increment combo box to select the maximum increment for your PLC points.

Example:

Some PLC modules have addresses that ends with 7 and continues with 10 (e.g. X7, no X8 and continues at X10). Set the maximum increment to 7.

### Spacing

Type in a value for the spacing between PLC points. The Spacing text box accepts measurement units, e.g. inches, millimeters, cm and other units.

### Generate

After setting PLC point, number of points and maximum increment, click on the Generate button to generate your PLC points. The generated points will be shown on the text box under the Generate button.

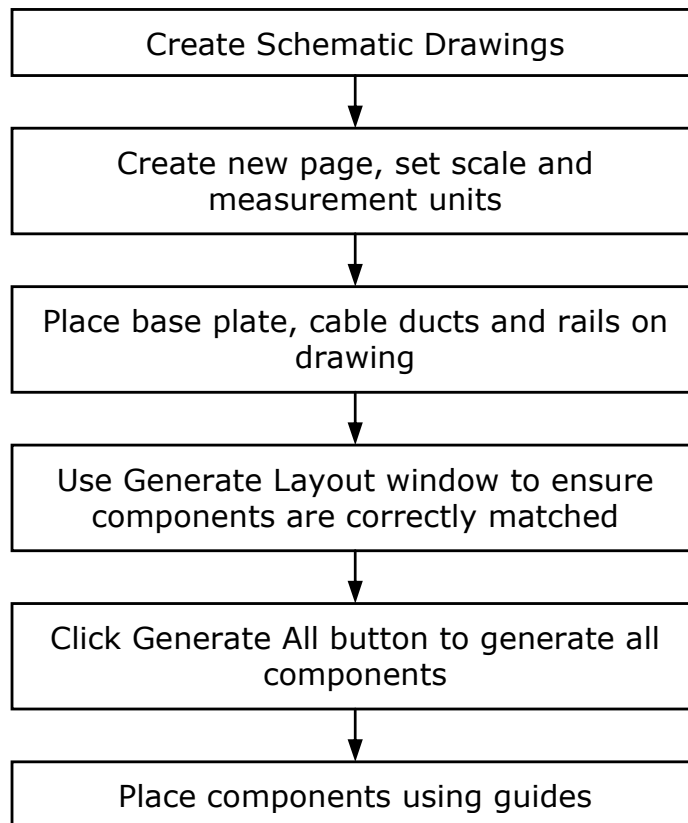
Examples:			
PLC Point:	X0	B00001	Q0.0
Number of Points:	8	12	12
Max. Increment:	9	9	7
Original PLC Labels text box content:	24Vdc Y01 Y02	24Vdc X0 0Vdc	24Vdc Y01 0Vdc
Results after after Generate:	24Vdc X0 X1 X2 X3 X4 X5 X6 X7	24Vdc B00001 0Vdc B00002 B00003 B00004 B00005 B00006 B00007 B00008 B00009 B00010 B00011 B00012	24Vdc Q0.0 0Vdc Q0.1 Q0.2 Q0.3 Q0.4 Q0.5 Q0.6 Q0.7 Q1.0 Q1.1 Q1.2 Q1.3

You can also manually create your PLC points by using the PLC labels text box under the Generate button. Type in the desired labels and press the Enter key. The PLC Module will automatically generate PLC point labels for each line of text in the PLC labels text box.

*See Online Electra™ Help for more information.*

## 13. Panel Layout

### 13.1. The Panel Layout Process



### 13.2. Preparing a page for panel layout

Pages on the Electra™ template are set at 1:1 scale and measurement units set at inches. These page settings are optimized for schematic drawings. To prepare a page for panel layout drawings:

1. Right click on a page tab.
2. Select insert page.
3. On the Page Setup dialog, select the Page Properties tab.
4. On the measurement units drop down, select millimeters.
5. On the Page Setup dialog, select the Drawing Scale tab.
6. Click on the custom scale radio button.
7. Type in your custom scale.

### 13.3. Inserting a plate

One of the first thing to do during panel layout drawings is to insert a plate (base plate or front plate). To insert a plate, select the Layout stencil and drag the plate icon to your drawing. All Plate shapes have automatic dimensions that you can show or hide.

### 13.4. Inserting cable ducts and rails

To place a cable duct into your drawings, drag on the cable duct icon. To place a rail into your drawings, drag on the rail icon. All CableDuct and Rail shape have automatic dimensions that you can show or hide.

What you can do with the CableDuct and Rail shape:	How to do it:
Set Dimension	Right click, select set dimension.
Show/Hide Dimension	Right click, select show/hide dimension.
Horizontal/Vertical rotation	Right click, select horizontal/vertical.
Set units	Right click, select mm/cm/inch.
Resizing	Click on the shape and drag handles.

### 13.5. Dimension shapes

To measure length and diameters, Electra™ provides 3 dimensioning shapes on the Layout stencil. All of them will automatically display the dimension when their handles are dragged.



### 13.6. Layout components

In Electra™, a component may consist of one or more schematic symbols. For example, a contactor component may consist of the contactor symbol (power contacts), coil symbol, normally open contacts and normally closed contacts. A relay component may consist of a coil symbol and one or more contact symbols.

### 13.7. Automatically generate layout components

Upon completion of your schematic drawings, use the Generate Layout window to automatically generate layout components. To access the Generate Layout window:

1. Select menu Electra | Generate Layout.
2. Ensure that all references are matched correctly with a component.
3. Click the Generate All button on the Generate Layout window.
4. Place and arrange the automatically generated components in your panel layout drawings.

### 13.8. The GenerateLayout window

The GenerateLayout window is designed to let users:

1. Modify or confirm that all schematic symbols are correctly matched to one or more component.
2. Add in or edit component information.
3. Creating a new component.
4. Generating one/all components for panel layout drawings.

The screenshot shows the 'Generate Layout' dialog box. At the top, the 'Reference' is set to 'C2'. Below this is a table with three rows of component information:

<input checked="" type="checkbox"/>	C2	Contactor	Contactor	P1.E3
<input checked="" type="checkbox"/>	C2		Coil	P1.H13
<input checked="" type="checkbox"/>	C2		Normally Open Contact	P1.G11

Below the table, there are several input fields and controls:

- Name: Contactor (with a 'Select...' button)
- Description: (empty)
- Brand: Generic
- Rating: (empty)
- Model: (empty)
- Order No.: (empty)
- Supplier: (empty)
- Remarks: (empty)
- Price: (empty)
- Layout Shape: LayRect (dropdown menu)
- Height: 73.0000 mm.
- Width: 45.0000 mm.
- Include Component in Layout:
- Buttons: Save, Cancel, Save As New Component, Component Manager...
- Bottom buttons: Generate All, Generate, Close

## Reference

Contains a list of all references on your schematic drawings.

Click on the Reference combo box and select a reference to display all component information for the reference.

## Previous/Next

Display the previous or next reference on the Reference combo box. Instead of using the Reference combo box above, you can also use the previous and next buttons to move to the next reference.

## Grouping

Contains a list of all schematic symbols for the reference shown in the Reference combo box. The columns of the Grouping list box show reference, symbol description, symbol type, and location respectively.

In most cases, a reference usually represents one component. For example, the reference R1 may represent a relay component, which consists of coil and contact symbols displayed on the Grouping list box.

However, in some cases we may have a reference that represents more than one component. For example, the reference C4 may consist of a power contact symbol, a coil symbol, 3 normally closed (NC) contact symbol and 2 normally open (NO) contact symbol. Therefore, C4 may represent one contactor component (with 1 NO contact) and another 2NO/2NC auxiliary contact component.

In Electra™, both cases and representation is acceptable through the use of "Grouping" on the GenerateLayout window. To use more than one component for a reference:

1. Deselect a symbol from the Grouping list box.
2. GenerateLayout will automatically produce another reference in the Reference combo box called Reference-Group2 where you can select another component for Group2.

*See Online Electra™ Help for an example.*

With the Grouping list box, you may create an unlimited number of groups. For example, to create Group3, select Group2 on the Reference combo box and deselect symbols in the Grouping list box for Group2.

To group all symbols back to the first group (e.g. C4), select C4 from the reference combo box and select all symbols on the Grouping list box.

The Grouping list box can also be used to ensure that you have not exceeded the number of contacts in a component. For example, if you use a relay component with 4 contacts, and the Grouping list box shows 5 contacts, you may have to select a relay component with more contacts or modify your drawings to use separate relays.

### Component information fields

Component information for all schematic symbols are stored on the drawing file (see portability) and applies only to the current reference shown in the Reference combo box. These component information will also be used during report generation (see generating reports).

<b>Field</b>	<b>Description</b>	<b>Examples</b>
Name	Component name.	Contactors
Description	Component description. The component description is not the same as a symbol description. Symbol description describes each symbol that makes up the component, while component description describes the component itself.	Motor 1 star contactor
Brand	Component brand.	Siemens
Rating	Component rating.	25kA, 32A
Model	Component model.	
Order No.	Component order number.	6ES7-211-0AA22-0X0B
Supplier	Component supplier.	Allen Bradley
Remarks	Component remarks.	To be supplied by vendor.
Price	Component price.	9.90

### Select button

Used to select a component for the reference shown in Reference combo box.

Electra™ stores a list of components and component information in a Microsoft® Access® database (Electra.mdb) to enable easy selection and access to component information.

Rather than typing in all component information, you can also select to save (see saving as new component) the current component information into this database.

Once the component information is saved on the database, use the Select button to select a component and load the component information onto your component information fields. Click the Select button and the component selection window will be visible.

To select a component on the Select component window, click on a component and click on the select button or double click on any component. After selecting a component, GenerateLayout will copy all the component's information from the database onto the component information fields on the GenerateLayout window.

### Save button and Cancel button

Used to save or cancel changes to the current component information fields.

If you modify the component information fields, and click on the Save button, the component information will be saved onto your symbols in your drawings (see portability). Click the Cancel button and GenerateLayout will revert the changes made to the component information fields.

Note that the Save button saves component information onto your symbols in your drawings and not onto the component database.

### Save as new component

Used to save changes on the component information fields as a new component in the component database.

To save the current component information into the database, click on the Save As New Component button and when asked for confirmation, click yes.

Saving current component information into the database enables you to access and select the saved information without the need to type in the information again when you need to use it on another component or in future drawings.

Note that the Save As New Component button saves component information into the database and not onto symbols on your drawings.

### Component Manager

Used to organize, view, edit and delete component information in the component database.

Click the Component Manager button to access the ComponentManager window (see ComponentManager).

### Include Component in Layout

Used to indicate whether the component is included in panel layout.

Your drawings may include components like contactors, push buttons, motors and limit switches. In your panel layout drawings, both the contactors and the push buttons will be included. However, the motors and limit switches which will be included in the Bill of Materials (see generating reports) should not be included in the panel layout drawings.

To include a component in the panel layout drawings, make sure the Include Component in Layout check box is checked.

### Layout shape

Contains a list of shapes to represent the component on panel layout drawings.

Once you have selected that a component should be included in the panel layout drawings, use the Layout Shape combo box to select a shape on the Layout stencil to represent the component.

### Height and Width

Used to modify component height and width.

To set the dimension for the current component, type in the required dimensions in the Height and Width text box.

### Generate All

Used to automatically generate **all** components for panel layout.

To generate all components on your drawing, click on the Generate All button. You can also select to automatically generate all your panel layout components in a new page by clicking Yes when prompted.

If you have made modifications on your schematic drawings, clicking the Generate All button will **automatically synchronize** your schematic drawings and panel layout drawings. Electra™ will automatically delete components which you have deleted on your schematic drawings or add in components if you have added new symbols to your schematic drawings.

### Generate

Used to generate the current component for panel layout.

To generate a single component, select the reference from the Reference combo box and click Generate.

## **13.9. Using guides to place components**

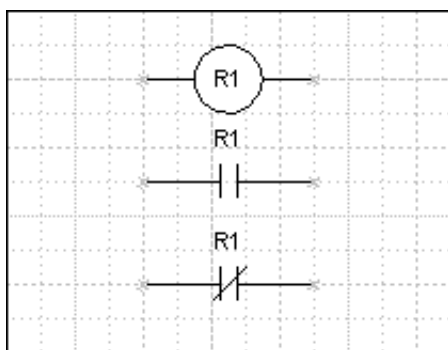
To create a horizontal guide, drag on the horizontal ruler. To create a vertical guide, click on the vertical ruler. To create a guide point, click on the intersection of the horizontal and vertical rulers.

Once you have created a guide, drag your layout shapes to the guide to attach it to the guide. When you move a guide, all shapes that are attached to the guide will automatically move and follow the guide.

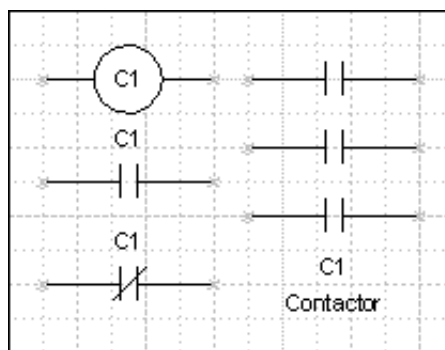
### 13.10. How GenerateLayout matches symbols to components

GenerateLayout matches a group of symbols with the same reference, to a component by searching for a symbol with the lowest priority number.

For example, we have 2 groups of symbols as shown below:



Group 1: Coil symbol, NO symbol and NC symbol with the same reference R1 represents a Relay component



Group 2: Coil symbol, NO symbol, NC symbol and Contactor power contact symbol with the same reference C1 represents a Contactor component.

The priority number for symbols used above are listed below:

Symbol	Priority	Lowest for Group 1	Lowest for Group 2
Coil	50	Yes	No
NC	100	No	No
NO	100	No	No
Contactor Power Contacts	10	Not Applicable	Yes

For the case in Group1, among the 3 symbols (coil, NC and NO) which has the same reference R1, the coil symbol has the lowest priority (50). Since the coil symbol is matched to a Relay component during symbol creation, GenerateLayout will correctly identify the group of symbols in Group 1 as a relay component.

For the case in Group 2, among the 4 symbols (coil, NC, NO, Contactor Power Contact) which has the same reference C1, the power contact symbol has the lowest priority (10), Since the power contact symbol is matched to a Contactor component during symbol creation, GenerateLayout will correctly identify the group of symbols in Group 2 as a Contactor component.

For a group which has only NC and NO contacts (both priority 100), GenerateLayout will use the first symbol it finds with the lowest priority number. At symbol creation, the NC and NO contacts are matched to an auxiliary contact component and therefore GenerateLayout will identify a group of NC and NO contacts as an auxiliary contact component.

To ensure that a group of symbols are matched correctly to a component, use the GenerateLayout window.

## 14. Component Manager

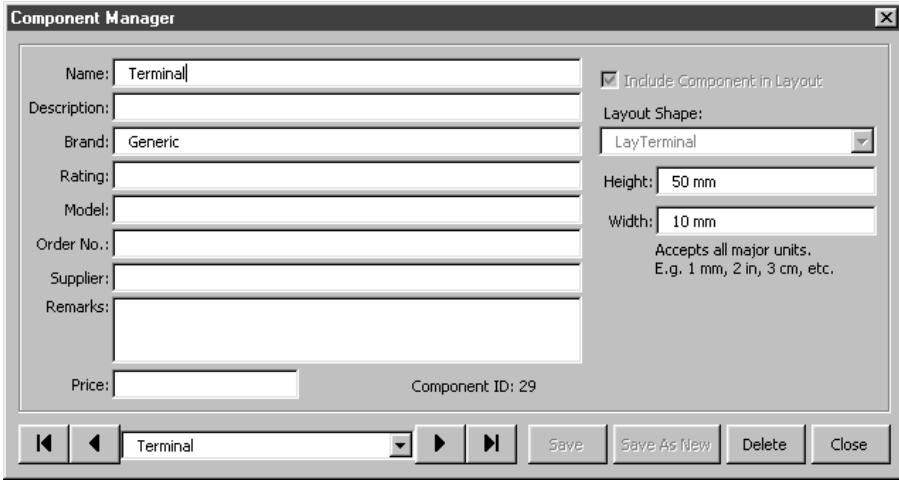
### 14.1. About component manager

Electra™ saves regularly used component information in a database file that can be easily accessed by using the ComponentManager window. The ComponentManager window can be used to add, delete or edit component information in the database.

### 14.2. The ComponentManager window

The ComponentManager window can be accessed by:

1. Select menu Electra | Component Manager.



The screenshot shows the 'Component Manager' window with the following fields and controls:

- Name: Terminal
- Description: (empty)
- Brand: Generic
- Rating: (empty)
- Model: (empty)
- Order No.: (empty)
- Supplier: (empty)
- Remarks: (empty)
- Price: (empty)
- Component ID: 29
- Include Component in Layout
- Layout Shape: LayTerminal (dropdown)
- Height: 50 mm
- Width: 10 mm
- Accepts all major units. E.g. 1 mm, 2 in, 3 cm, etc.
- Navigation buttons: First, Previous, Next, Last
- Buttons: Save, Save As New, Delete, Close

#### Component Information fields

Data for the current component is displayed here. Use these fields to edit or modify component information.

#### Navigation buttons

Use the navigation buttons to move to the first, previous, next or last component.

#### Save

The Save button will only be enabled when you have modified the component information fields. Click the save button to save the modifications into the database. To cancel changes, close the ComponentManager window or click on the next button and select No.

### Save as New

The Save as New button will only be enabled when you have modified the component information fields. The Save as New button saves the current information as a new component in the database.

### Delete

To delete a component in the database, select the component and click on the Delete button. When asked for confirmation, click Yes and the component will be deleted from the database.

### Close

To close the ComponentManager window, click on the Close button.

## 15. Generating Reports

### 15.1. Generating Cross Reference report

The Cross Reference report list all schematic symbol reference, symbol description, symbol type/master and location.

To automatically generate the Cross Reference report:

1. Drag the CrossReference shape from the Report stencil onto your drawing.
2. Place and resize the height of the CrossReference shape. If the report length exceed the CrossReference shape (overflow), Electra™ will automatically create another CrossReference shape.
3. Right click on the CrossReference shape and select Generate Report.

After generating the report, you may resize the columns by clicking on the CrossReference shape to select it, click again on the column and drag on the column handles to resize.

### 15.2. Generating Material Reference report

The Material Reference report list all symbol references, component name, component description, brand, rating model, order no., supplier and remarks.

To automatically generate the Material Reference report:

1. Drag the MaterialReference shape from the Report stencil onto your drawing.
2. Place and resize the height of the MaterialReference shape. If the report length exceed the MaterialReference shape (overflow), Electra™ will automatically create another MaterialReference shape.
3. Right click on the MaterialReference shape and select Generate Report.
4. Select the columns to be included/excluded from the report.
5. Click Generate button to automatically generate report.

Upon completion of report generation, the MaterialReference shape will **automatically resize** its columns to fit text. Alternatively, you can also manually resize the columns by clicking the MaterialReference shape to select it, click again on the column and drag on the column handles to resize.

### 15.3. Generating Bill of Materials (BOM) report

The Bill of Materials report list all components and quantities used on your drawings including cable ducts, rails and terminals. The following columns are included in the BOM report:

1. Component name.
2. Component description.
3. Brand.
4. Rating.
5. Model.
6. Order no.
7. Supplier.
8. Remarks.
9. Quantity.
10. Price.
11. Amount.

To automatically generate the BOM report:

1. Drag the BOM shape from the Report stencil onto your drawing.
2. Place and resize the height of the BOM shape. If the report length exceed the BOM shape (overflow), Electra™ will automatically create another BOM shape.
3. Right click on the BOM shape and select Generate Report.
4. Select the columns to be included/excluded from the report.
5. Click Generate button to automatically generate report.

Upon completion of report generation, the BOM shape will **automatically resize** its columns to fit text. Alternatively, you can also manually resize the columns by clicking the BOM shape to select it, click again on the column and drag on the column handles to resize.

### 15.4. Exporting reports to Excel®

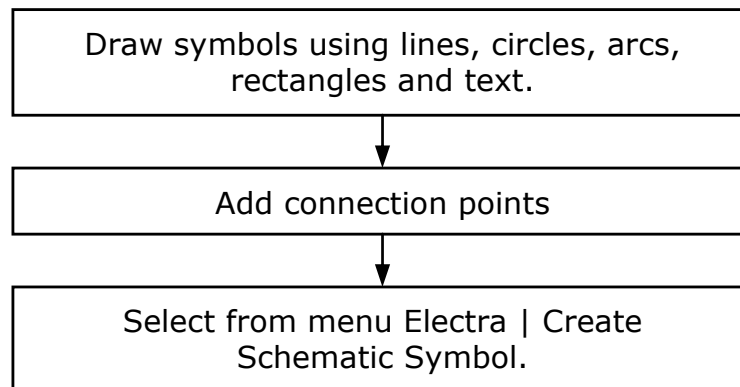
To export reports to Microsoft® Excel®:

1. Select menu Electra | Export Reports.
2. Select from the report combo box the report to generate (Cross reference report, Material Reference report or Boll of Material (BOM) report).
3. Click the Generate button.

## 16. Creating your own schematic symbol

### 16.1. Symbol creation process

Creating your own intelligent schematic symbol is an easy and automated process. Follow the steps listed below:



### 16.2. Drawing and combining a symbol

A symbol may consist of a combination of lines, arcs and text. Draw your custom symbol using the graphic tools on the Visio® toolbar.

After drawing your symbol graphics, they must be combined or grouped. Symbols that do not have text should be combined as they have lower overhead. Symbols that have text should be grouped together.

To combine symbol graphics:

1. Select all the symbol graphics.
2. Select menu Shape | Operations | Combine.

To group symbol graphics:

1. Select all the symbol graphics including text.
2. Select menu Shape | Grouping | Group.

### 16.3. Adding connection points

After combining or grouping your symbol graphics, use the connection point tool to add connection points to your symbol. Connection points enable your symbol to easily connect to nets (wires).

To add connection points :

1. Click on the symbol to select it.
2. Click on the connection point tool.
3. Press Ctrl (Control) key and click at the desired position on the symbol.

To change a connection point's position:

1. Click on a symbol to select it.
2. Click on the connection point tool.
3. Drag a connection point from the symbol to a new position.

To delete a connection point:

1. Click on the symbol to select it.
2. Click on a connection point on a symbol.
3. Press the delete key

### 16.4. The CreateSymbol window

The CreateSymbol window is used to **automatically insert intelligence and data** into your custom symbol in order for it to be correctly identified and processed by Electra™. After selecting options on the CreateSymbol window, click OK and your custom symbol will be automatically created and added into a stencil.

To access the CreateSymbol window:

1. Select your symbol with connection points by clicking on the symbol.
2. Select menu Electra | Create Schematic Symbol.

### Default Reference

Type a default reference for your custom symbol (e.g. R1).

### Description text box and No Description check box

Most schematic symbols have a symbol description (e.g. Start button) displayed at the bottom of the symbol. If a symbol description box does not yet exist on your custom symbol, the CreateSymbol window will automatically add in a symbol description box. To disable automatic adding of a symbol description box, check the No Description check box.

When you have selected to include a symbol description in your custom symbol, use the description text box to type in the default symbol description.

### Vertical by default

If you are creating a symbol which is vertical by default when dropped on a drawing, check the vertical by default check box.

### Hide Description

Custom symbol hides description text by default when dropped on drawing if Hide Description is enabled.

Custom symbol shows description by default when dropped on drawing if Hide Description is disabled.

Once a symbol is dropped on a drawing, users can select to show or hide the description text by right clicking on the symbol, select Edit Reference and using the Reference window.

### Insert Horizontal / Vertical rotation menu

To insert a horizontal / vertical rotation menu into the right click floating menu on your custom symbol, check the Insert Horizontal / Vertical rotation menu check box.

The horizontal / vertical menu enables you to have one symbol for both horizontal and vertical drawings, similar to symbols provided by Electra™.

### Component information

Displays component information for your custom symbol. See Component Information fields for a description of the information fields.

### Select button

All schematic symbols in Electra™ have a link to a component. This link is used by Electra™ to automatically match one or more symbols to a component during panel layout (see How GenerateLayout matches symbols to components).

To select a component for your custom symbol, click on the Select button and the Select Component window will be visible. To select a component, double click on a component or select a component and click the Select button.

### Priority

All schematic symbols in Electra™ have a priority number that is used to match a group of symbols to a component. For user created custom symbols, a priority number of 15 is recommended. See How GenerateLayout matches symbols to components for more information.

### Location Type

The Location Type combo box is used by the Location shape to identify the symbol type. For example, a coil symbol should be set to location type Coil, a normally closed contact should be set to location type NC and a normally open contact should be set to location type NO.

## Component Manager

Electra™ saves regularly used component information in a database file that can be easily accessed by using the ComponentManager window. The ComponentManager window can be used to add, delete or edit component information in the database.

If you are unable to locate an existing component for your custom symbol, click on the ComponentManager button and create a new component for your custom symbol.

### Add to Stencil

Use the Add to Stencil combo box to select a stencil for your custom symbol.

### Symbol Name (Required, cannot be empty)

Type in a name for your symbol (e.g. NO). Symbol name is also used as the default description.

### Symbol Prompt

Type in a prompt for your symbol. The symbol prompt is a longer more descriptive name for your symbol (e.g. Normally Open Contact). When the symbol prompt text box is left empty, the symbol name will be used as the default symbol prompt.

### Symbol Type

The Symbol Type is normally used to group NC and NO symbols together so that their references can be incremented sequentially.

#### Example 1:

2 symbols with the same default reference but different symbol type.

1. We have a NC push button symbol and a NO push button symbol.
2. Both symbols have PB1 as default reference.
3. Symbol type for NC push button is PBNC.
4. Symbol type for NO push button is PBNO.
5. When you drag 2 NC push button onto your drawings, they will be renamed PB1 and PB2.
6. When you drag 2 more NO push button onto your drawings, they will be renamed PB1 and PB2 also, since their symbol type is different.

### Example 2:

2 symbols with the same default reference and same symbol type.

1. We have a NC push button symbol and a NO push button symbol.
2. Both symbols have PB1 as default reference.
3. Symbol type for NC push button is PBNCNO.
4. Symbol type for NO push button is also PBNCNO.
5. When you drag 2 NC push button onto your drawings, they will be renamed PB1 and PB2.
6. When you drag 2 more NO push button onto your drawings, they will be renamed PB3 and PB4 (sequentially) as they have the same symbol type.

If you are creating one symbol for NO and another for NC, type in a common symbol type to group them together, so that their references will be incremented sequentially. If the symbol type text box is left empty, the symbol name will be used as the default symbol type.

### OK button

After setting all the options on the CreateSymbol window, click the OK button and CreateSymbol will:

1. Insert intelligence and data into your custom symbol.
2. Open the selected stencil and add your custom symbol onto the stencil.
3. Delete the custom symbol from your drawing (For Electra™ to identify your custom symbol, it must originate from a stencil).

If you need to modify your symbol:

1. Right click on your symbol in the stencil and select Edit Master.

Or

1. Drag your symbol onto your drawings, modify it and use the CreateSymbol window again to add it back into a stencil.

### 16.5. Editing the symbol icon

An icon will be generated automatically when the symbol is added to a stencil. Right click on the symbol on the stencil and select Edit Icon to modify symbol icon.

After creating your custom icon for your symbol, right click on the symbol in the stencil, select Master Properties, and **make sure** to deselect Generate Icon automatically from shape data. This is to ensure that future modifications to your shape will not overwrite your custom icon.

### 16.6. The Stencil Browser window

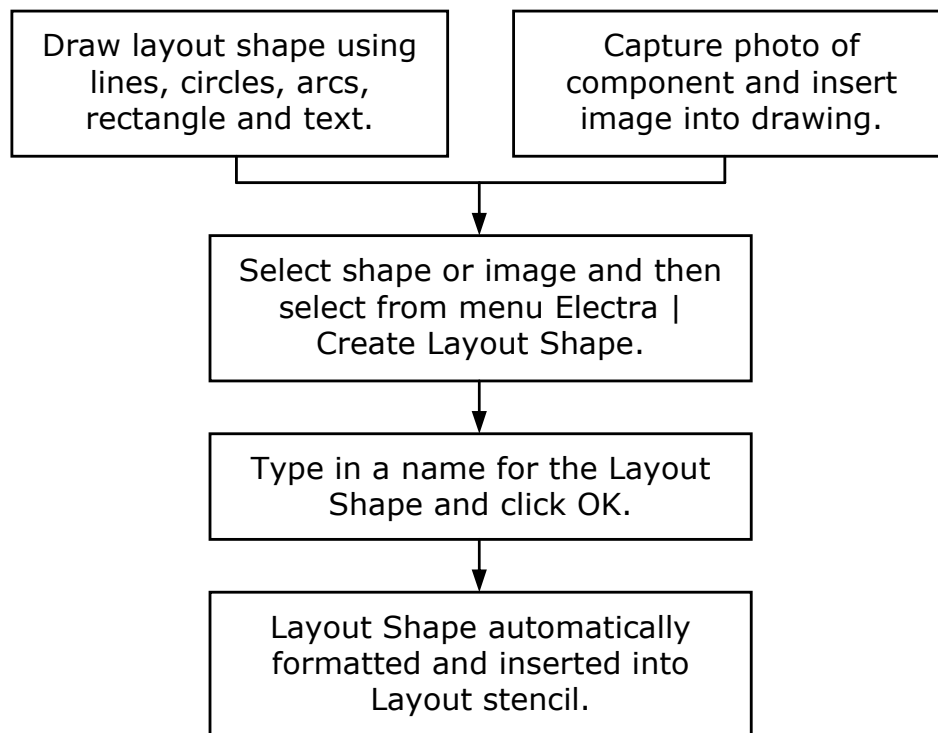
To view and edit symbol information in a stencil, use the stencil browser window. To access the Stencil Browser window:

1. Select menu Electra | Stencil Browser.

## 17. Creating your own layout shape

### 17.1. Layout shape creation process

Create your own layout shape using the following process:



### 17.2. Drawing or combining a layout shape

A layout may consist of a combination of lines, arcs and text. Draw your custom layout shape using the graphic tools on the toolbar:

After drawing your layout shape graphics, they **must** be combined or grouped. Layout shapes that do not have text should be combined as they have lower overhead. Layout shapes that have text should be grouped together.

To combine layout shape graphics:

1. Select all the layout shape graphics.
2. Select menu Shape | Operations | Combine.

To group layout shape graphics:

1. Select all the layout shape graphics including text.
2. Select menu Shape | Grouping | Group.

### 17.3. The CreateLayoutShape window

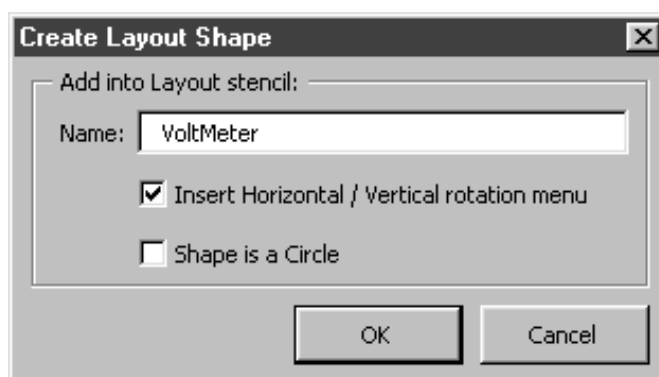
To enable your custom Layout Shape to be used on the GenerateLayout window during panel layout:

1. Layout shape must be inserted with intelligence and data.
2. Layout shape must be on the Layout Stencil.

The CreateLayoutShape window is designed to automatically insert intelligence and data into a custom created layout shape, and add it into the layout stencil.

To access the CreateLayoutShape window:

1. Select your layout shape by clicking on the shape.
2. Select menu Electra | Create Layout Shape.



#### Name

Type in a name for your custom created layout shape. The name of the layout shape will be used on the GenerateLayout window to match a component to a layout shape.

#### Insert Horizontal / Vertical rotation menu

To insert a horizontal / vertical rotation menu into the right click floating menu on your layout shape, check the Insert Horizontal / Vertical rotation menu check box.

The horizontal / vertical menu enables you to rotate the layout shape for easy vertical or horizontal shape placement.

### Shape is a Circle

If the custom layout shape you are creating is a circle, check the Shape is a Circle check box. A circle type layout shape has diameter as dimension rather than height and width.

### OK and Cancel buttons

To cancel creating a custom layout shape, click on the cancel button.

When you click on the OK button, CreateLayoutShape will:

1. Insert intelligence and data into your custom layout shape.
2. Open the Layout stencil and add your custom layout shape onto the stencil.
3. Delete the custom layout shape from your drawing (For Electra™ to identify your custom layout shape, it must originate from a stencil).

## **17.4. Editing the layout shape icon**

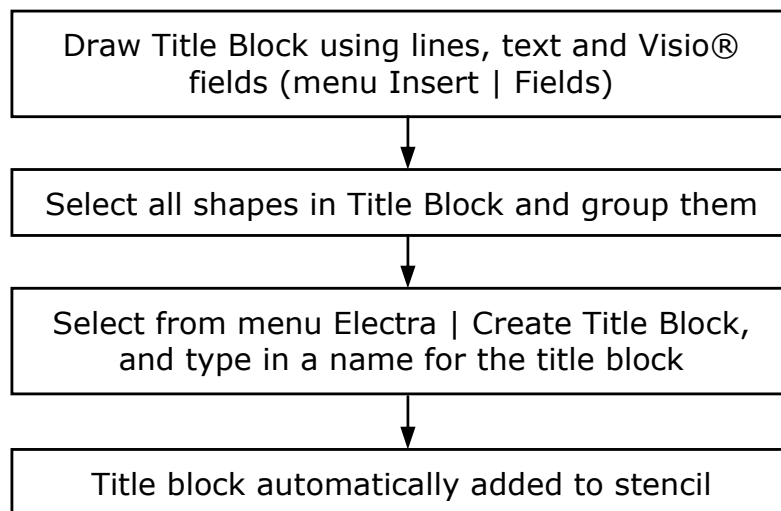
An icon will be generated automatically when the layout shape is added to a stencil. Right click on the layout shape on the stencil and select Edit Icon to modify layout shape icon.

After creating your custom icon for your layout shape, right click on the layout shape in the stencil, select Master Properties, and make sure to deselect Generate Icon automatically from shape data. This is to ensure that future modifications to your shape will not overwrite your custom icon.

## 18. Creating your own title blocks

### 18.1. Title Block creation process

Create your own title blocks using the following process:



## 18.2. Drawing and grouping a title block

Use the shapes on the Title Blocks stencil to draw and create your custom title block.

Shapes	Description
Zones	The zones shape will automatically create zone labels based on the dimensions set for location zones on the Document Options window.
Date	The Date shape will automatically displays the drawing creation date, last save date, last printed date or the current date based on user selectable options. Right click on the Date shape to select the date type.
Drawn By	The Drawn By shape will automatically displays the document author. Document author can also be accessed by selecting menu File   Properties.
File Name	The File shape will automatically display the document file name.
Company / Firm	The company shape will automatically display your company name. Company name can also be accessed by selecting menu File   Properties.
Page	The page shape will automatically display current page and total pages.
Scale	The scale shape will automatically display the scale for the current page.
Label	Use the Label shape to easily displays common used labels for title blocks. Right click on the label shape, select Set Label and choose any of the following text. "Date, Created, Revised, Drawn By, Drawing No., File, Company, Client, Customer, Page, Scale, Title".

After using lines, text, rectangles and shapes on the Title Blocks stencil to draw your title block, select all of them and group them.

To group title block shapes:

1. Select all the shapes that make up your title block.
2. Select menu Shape | Grouping | Group.

### 18.3. The CreateTitleBlock window

The CreateTitleBlock window will insert intelligence into your title block so that it will automatically fit your drawing page when dropped on your drawing and add your custom created title block shape into the Title Blocks stencil.

To access the CreateLayoutShape window:

1. Select your grouped title block shape by clicking on the shape.
2. Select menu Electra | Create Title Block.

Type in a name and a prompt (longer more descriptive name) for your custom created title block and click OK. The CreateTitleBlock window will automatically add intelligence into your custom title block and add the title block into the Title Blocks stencil.

### 18.4. Editing the title block shape icon

An icon will be generated automatically when the title block shape is added to a stencil. Right click on the title block shape on the stencil and select Edit Icon to modify title block shape icon.

After creating your custom icon for your title block shape, right click on the title block shape in the stencil, select Master Properties, and **make sure** to deselect Generate Icon automatically from shape data. This is to ensure that future modifications to your shape will not overwrite your custom icon.

## 19. Appendix A : Symbol Listing

<b>Electra Symbols</b>			
<b>Name</b>	<b>Prompt</b>	<b>Type</b>	<b>Default Reference</b>
Isolator	Isolator	Isolator	S1
MCB	Miniature Circuit Breaker	MCB	Q1
Contactora	Contactora	Contactora	C1
TOR	Thermal Overload Relay	TOR	TOR1
Motor	Motor	Motor	M1
Transformer	Transformer	Transformer	TR1
MCB 1P	Miniature Circuit Breaker 1P	MCB	Q1
NO Contact	Normally Open Contact	NONC	R1
NC Contact	Normally Closed Contact	NONC	R1
PS_NC	Pressure Switch NC	PSNONC	PS1
PS_NO	Pressure Switch NO	PSNONC	PS1
SS_NC	Selector Switch NC	SSNONC	SS1
SS_NO	Selector Switch NO	SSNONC	SS1
PB_NC	Push Button NC	PBNONC	PB1
PB_NO	Push Button NO	PBNONC	PB1
TOR_NC	Thermal Overload NC	TORNC	TOR1
TOR_NO	Thermal Overload NO	TORNO	TOR1
ES_NC	Emergency Stop NC	ESNONC	ES1
ES_NO	Emergency Stop NO	ESNONC	ES1
TMR_NC	Timer NC	TONNONC	TMR1
TMR_NO	Timer NO	TONNONC	TMR1
LS_NC	Limit Switch NC	LSNONC	LS1
LS_NO	Limit Switch NO	LSNONC	LS1
Coil	Coil	Coil	R1
PilotLight	Pilot Light	PilotLight	PL1
Solenoid	Solenoid	Solenoid	Sol1
SW_NC	Switch NC	SWNONC	SW1
SW_NO	Switch NO	SWNONC	SW1
PLC	PLC Module	PLC	PLC2

<b>Electra Symbols</b>				
<b>Name</b>	<b>Component</b>	<b>No.</b>	<b>Priority</b>	<b>Location Type</b>
Isolator	Isolator	2	10	NO
MCB	MCB 3P	3	10	NC
Contactora	Contactora	4	10	NO
TOR	Thermal Overload Relay	5	10	NC
Motor	Motor 3P AC	6	10	Coil
Transformer	Transformer 3P	7	10	Coil
MCB 1P	MCB 1P	23	15	NC
NO Contact	Aux Contact	21	100	NO
NC Contact	Aux Contact	21	100	NC
PS_NC	Pressure Switch	8	20	NC
PS_NO	Pressure Switch	8	20	NO
SS_NC	Selector Switch	9	20	NC
SS_NO	Selector Switch	9	20	NO
PB_NC	Push Button	10	20	NC
PB_NO	Push Button	10	20	NO
TOR_NC	Thermal Overload Relay	5	20	NC
TOR_NO	Thermal Overload Relay	5	20	NO
ES_NC	Emergency Stop	11	20	NC
ES_NO	Emergency Stop	11	20	NO
TMR_NC	Timer	12	20	NC
TMR_NO	Timer	12	20	NO
LS_NC	Limit Switch	13	20	NC
LS_NO	Limit Switch	13	20	NO
Coil	Relay	15	50	Coil
PilotLight	Pilot Light	16	20	Coil
Solenoid	Solenoid	17	20	Coil
Switch NC	Generic Component	1	20	NC
Switch NO	Generic Component	1	20	NO
PLC Module	PLC Module	48	10	Coil

<b>IEC Symbols</b>			
<b>Name</b>	<b>Prompt</b>	<b>Type</b>	<b>Default Reference</b>
Fuse 3P	Fuse 3P	F	F1
Fuse Disconnecter 3P	Fuse Disconnecter 3P	F	F1
Disconnecter	Disconnecter	Q	Q1
Switch Disconnecter	Switch Disconnecter	Q	Q1
MCB 3P	Circuit Breaker	Q	Q1
Contactor	Contactor	K	K1
Motor Circuit Breaker	Motor Circuit Breaker	Q	Q1
Current Limiter	Current Limiter	Q	Q1
TOR	Thermal Overload	F	F1
DC Motor	DC Motor	M	M1
Motor 2P	Motor 2P	M	M1
Motor 3P AC	Motor 3P AC	M	M1
Transformer 3P	Transformer 3P	T	T1
Transformer 2P	Transformer 2P	T	T1
Current Transformer	Current Transformer	L	L1
Fuse 2P	Fuse 2P	F	F1
Fuse Disconnecter 2P	Fuse Disconnecter 2P	F	F1
MCB 2P	Circuit Breaker 2P	Q	Q1
Fuse 1P	Fuse 1P	F	F1
Fuse Disconnecter 1P	Fuse Disconnecter 1P	F	F1
MCB 1P	Circuit Breaker 1P	Q	Q1
PLC	PLC Module	PLC	PLC1
Coil	AC Coil	K	K1
Coil	Coil	K	K1
NO	Normally Open Contact	K	K1
NC	Normally Close Contact	K	K1
TORNO	Thermal Overload NO	F	F1
TORNC	Thermal Overload NC	F	F1
ESNO	Emergency Stop NO	S	S1
ESNC	Emergency Stop NC	S	S1
ESNO	E.Stop Spring Return NO	S	S1
ESNC	E.Stop Spring Return NC	S	S1
TON	On Delay Timer	K	K1
TONNO	On Delay NO Contact	K	K1
TONNC	ON Delay NC Contact	K	K1
TOFF	Off Delay Timer	K	K1
TOFFNO	Off Delay NO Contact	K	K1
TOFFNC	Off Delay NC Contact	K	K1
PBNO	Push Button Spring Return NO	S	S1
PBNC	Push Button Spring Return NC	S	S1
PBNO	Push Button Maintained NO	S	S1
PBNC	Push Button Maintained NC	S	S1
SSNO	Selector Switch Spring Return NO	S	S1
SSNC	Selector Switch Spring Return NC	S	S1

<b>IEC Symbols</b>			
<b>Name</b>	<b>Prompt</b>	<b>Type</b>	<b>Default Reference</b>
SSNO	Selector Switch Maintained NO	S	S1
SSNC	Selector Switch Maintained NC	S	S1
PL	Pilot Light	H	H1
PL	Pilot Light Blinking	H	H1
PL	Pilot Light with Transformer	H	H1
Beacon	Beacon	H	H1
LSNO	Limit Switch NO	K	S1
LSNC	Limit Switch NC	K	S1
FSNO	Foot Switch NO	S	S1
FSNC	Foot Switch NC	S	S1
CSNO	Cam Switch NO	K	K1
CSNC	Cam Switch NC	K	K1
KSNO	Key Switch Maintained NO	S	S1
KSNC	Key Switch Maintained NC	S	S1
PSNO	Pressure Switch NO	K	K1
PSNC	Pressure Switch NC	K	K1
FLNO	Flow Switch NO	K	B1
FLNC	Flow Switch NC	K	K1
LLNO	Level Switch NO	K	K1
LLNC	Level Switch NC	K	K1
RLNO	Roller Switch NO	K	K1
RLNC	Roller Switch NC	K	K1
CPNO	Cam Profile Switch NO	K	K1
CPNC	Cam Profile Switch NC	K	K1
Counter	Counter	P	P1
Counter	Counter c/w Electrical Reset	P	P1
Counter	Counter c/w Manual Reset	P	P1
CNTNO	Counter Switch NO	P	P1
CNTNC	Counter Switch NC	P	P1
ECNO	Electronic Clock Switch NO	P	P1
ECNC	Electronic Clock Switch NC	P	P1
GFLNO	Gas Flow Switch NO	K	K1
GFLNC	Gas Flow Switch NC	K	K1
SPS	Switching Power Supply	G	G1
CapPX Sensor	Capacitive Proximity Sensor	S	S1
IndPX Sensor	Inductive Proximity Sensor	S	S1
MagPX Sensor	Magnetic Proximity Sensor	S	S1
PhoPX Sensor	Retro Reflective Photoelectric Sensor	S	S1
PhoPX Sensor	Diffuse Photoelectric Sensor	S	S1
PhoPX Sensor	Through Beam Photoelectric Sensor	S	S1
Relay	High Speed Relay	K	K1
Relay	Mechanically Latched Relay	K	K1
Relay	Over Voltage Relay	K	B1
Relay	Under Voltage Relay	K	B1

<b>IEC Symbols</b>			
<b>Name</b>	<b>Prompt</b>	<b>Type</b>	<b>Default Reference</b>
Brake	Magnetic Brake	Y	R1
Clutch	Magnetic Clutch	Y	R1
Solenoid	Solenoid	Y	Y1
Horn	Horn (Acoustic Device)	H	H1
Buzzer	Buzzer	H	H1
Siren	Siren	H	H1
Voltmeter	Voltmeter	P	P1
Ammeter	Ammeter	P	P1

<b>IEC Symbols</b>				
<b>Name</b>	<b>Component</b>	<b>No.</b>	<b>Priority</b>	<b>Location Type</b>
Fuse 3P	Fuse 3P	61	10	NC
Fuse Disconnecter 3P	Fuse Disconnecter 3P	63	10	NO
Disconnecter	Disconnecter	57	10	NO
Switch Disconnecter	Disconnecter	57	10	NO
MCB 3P	MCB 3P	3	10	NC
Contactor	Contactor	4	10	NO
Motor Circuit Breaker	Motor Breaker	58	10	NC
Current Limiter	Current Limiter	64	10	NC
TOR	Thermal Overload Relay	5	10	NC
DC Motor	Motor DC	91	10	Coil
Motor 2P	Motor 2P AC	92	10	Coil
Motor 3P AC	Motor 3P AC	6	10	Coil
Transformer 3P	Transformer 3P	7	10	Coil
Transformer 2P	Transformer 2P	93	10	Coil
Current Transformer	Current Transformer	94	10	Coil
Fuse 2P	Fuse 2P	60	20	NC
Fuse Disconnecter 2P	Fuse Disconnecter 2P	85	20	NO
MCB 2P	MCB 2P	47	20	NC
Fuse 1P	Fuse 1P	59	20	NC
Fuse Disconnecter 1P	Fuse Disconnecter 1P	62	20	NO
MCB 1P	MCB 1P	23	20	NC
PLC	PLC Module	48	10	Coil
Coil	Relay	15	19	Coil
Coil	Relay	15	50	Coil
NO	Aux Contact	21	100	NO
NC	Aux Contact	21	100	NC
TORNO	Thermal Overload Relay	5	20	NO
TORNC	Thermal Overload Relay	5	20	NC
ESNO	Emergency Stop	11	20	NO
ESNC	Emergency Stop	11	20	NC
ESNO	Emergency Stop	68	20	NO
ESNC	Emergency Stop	68	20	NC
TON	Timer	12	19	Coil
TONNO	Timer	12	20	NO
TONNC	Timer	12	20	NC
TOFF	Timer	74	19	Coil
TOFFNO	Timer	74	20	NO
TOFFNC	Timer	74	20	NC
PBNO	Push Button	10	20	NO
PBNC	Push Button	10	20	NC
PBNO	Push Button	75	20	NO
PBNC	Push Button	75	20	NC
SSNO	Selector Switch	76	20	NO
SSNC	Selector Switch	76	20	NC

<b>IEC Symbols</b>				
<b>Name</b>	<b>Component</b>	<b>No.</b>	<b>Priority</b>	<b>Location Type</b>
SSNO	Selector Switch	9	20	NO
SSNC	Selector Switch	9	20	NC
PL	Pilot Light	16	20	Coil
PL	Pilot Light	86	20	Coil
PL	Pilot Light	88	20	Coil
Beacon	Beacon	87	20	Coil
LSNO	Limit Switch	13	20	NO
LSNC	Limit Switch	13	20	NC
FSNO	Foot Switch	65	20	NO
FSNC	Foot Switch	65	20	NC
CSNO	Cam Switch	66	20	NO
CSNC	Cam Switch	66	20	NC
KSNO	Key Switch	67	20	NO
KSNC	Key Switch	67	20	NC
PSNO	Pressure Switch	8	20	NO
PSNC	Pressure Switch	8	20	NC
FLNO	Flow Switch	46	20	NO
FLNC	Flow Switch	46	20	NC
LLNO	Liquid Level	42	20	NO
LLNC	Liquid Level	42	20	NC
RLNO	Roller Switch	69	20	NO
RLNC	Roller Switch	69	20	NC
CPNO	Cam Profile Switch	70	20	NO
CPNC	Cam Profile Switch	70	20	NC
Counter	Counter	71	19	Coil
Counter	Counter	79	19	Coil
Counter	Counter	80	19	Coil
CNTNO	Counter	71	20	NO
CNTNC	Counter	71	20	NC
ECNO	Electronic Clock	72	20	NO
ECNC	Electronic Clock	72	20	NC
GFLNO	Gas Flow Switch	73	20	NO
GFLNC	Gas Flow Switch	73	20	NC
SPS	Switching Power Supply	41	10	Coil
CapPX Sensor	Capacitive Proximity Sensor	95	20	Coil
IndPX Sensor	Inductive Proximity Sensor	96	20	Coil
MagPX Sensor	Magnetic Proximity Sensor	97	20	Coil
PhoPX Sensor	Photoelectric Proximity Sensor	99	20	Coil
PhoPX Sensor	Photoelectric Proximity Sensor	100	20	Coil
PhoPX Sensor	Photoelectric Proximity Sensor	98	20	Coil
Relay	Relay	77	19	Coil
Relay	Relay	78	19	Coil
Relay	Relay	81	19	Coil
Relay	Relay	82	19	Coil

<b>Ele IEC Symbols</b>				
<b>Name</b>	<b>Component</b>	<b>No.</b>	<b>Priority</b>	<b>Location Type</b>
Brake	Brake	83	19	Coil
Clutch	Clutch	84	19	Coil
Solenoid	Solenoid	17	19	Coil
Horn	Horn	43	20	Coil
Buzzer	Buzzer	44	20	Coil
Siren	Siren	25	20	Coil
Voltmeter	Voltmeter	89	20	Coil
Ammeter	Ammeter	90	20	Coil

<b>JIC Symbols</b>			
<b>Name</b>	<b>Prompt</b>	<b>Type</b>	<b>Default Reference</b>
Contactora	Contactora	Contactora	C1
TOR	Thermal Overload	TOR	TOR1
Isolator	Isolator	Isolator	S1
MCB	Circuit Breaker	MCB	Q1
Motor 3P AC	Motor 3 Phase AC	Motor3PAC	M1
Transformer	Transformer	Transformer	TR1
MCB	Circuit Breaker	MCB	MCB1
MCB 1P	Circuit Breaker 1P	MCB	Q1
NO	Normally Open Contact	NONC	R1
NC	Normally Closed Contact	NONC	R1
PBNO	Push Button NO	PBNONC	PB1
PBNC	Push Button NC	PBNONC	PB1
SSNO	Selector Switch NO	SSNONC	SS1
SSNC	Selector Switch NC	SSNONC	SS1
TORNO	Thermal Overload NO	TORNO	TOR1
TORNC	Thermal Overload NC	TORNC	TOR1
ESNO	Emergency Stop NO	ESNONC	ES1
ESNC	Emergency Stop NC	ESNONC	ES1
LSNO	Limit Switch NO	LSNONC	LS1
LSNC	Limit Switch NC	LSNONC	LS1
PSNO	Pressure Switch NO	PSNONC	PS1
PSNC	Pressure Switch NC	PSNONC	PS2
LLNO	Liquid Level NO	LLNONC	LL1
LLNC	Liquid Level NC	LLNONC	LL1
FLNO	Flow Switch NO	FLNONC	FL1
FLNC	Flow Switch NC	FLNONC	FL1
PXNO	Proximity Switch NO	PXNONC	PX1
PXNC	Proximity Switch NC	PXNONC	PX1
TONNO	Timer Delay On NO	TONNONC	TM1
TONNC	Timer Delay On NC	TONNONC	TM1
TOFFNO	Timer Delay Off NO	TOFFNONC	TM1
TOFFNC	Timer Delay Off NC	TOFFNONC	TM1
Sol	Solenoid	Sol	Sol1
Coil	Coil	Coil	R1
PL	Pilot Light	PL	PL1
Horn	Horn	Horn	H1
Buzzer	Buzzer	Buzzer	BZ1
Bell	Bell	Bell	B1
PLC	PLC Module	PLC	PLC1

<b>JIC Symbols</b>				
<b>Name</b>	<b>Component</b>	<b>No.</b>	<b>Priority</b>	<b>Location Type</b>
Contactora	Contactora	4	10	NO
TOR	Thermal Overload Relay	5	10	NC
Isolator	Isolator	2	10	NO
MCB	MCB 3P	3	10	NC
Motor 3P AC	Motor 3P AC	6	10	Coil
Transformer	Transformer 3P	7	10	Coil
MCB	MCB 3P	3	10	NC
MCB 1P	MCB 1P	23	20	NC
NO	Aux Contact	21	100	NO
NC	Aux Contact	21	100	NC
PBNO	Push Button	10	20	NO
PBNC	Push Button	10	20	NC
SSNO	Selector Switch	9	20	NO
SSNC	Selector Switch	9	20	NC
TORNO	Thermal Overload Relay	5	20	NO
TORNC	Thermal Overload Relay	5	20	NC
ESNO	Emergency Stop	11	20	NO
ESNC	Emergency Stop	11	20	NC
LSNO	Limit Switch	13	20	NO
LSNC	Limit Switch	13	20	NC
PSNO	Pressure Switch	8	20	NO
PSNC	Pressure Switch	8	20	NC
LLNO	Liquid Level	42	20	NO
LLNC	Liquid Level	42	20	NC
FLNO	Flow Switch	46	20	NO
FLNC	Flow Switch	46	20	NC
PXNO	Proximity Switch	49	20	NO
PXNC	Proximity Switch	49	20	NC
TONNO	Timer	12	20	NO
TONNC	Timer	12	20	NC
TOFFNO	Timer	12	20	NO
TOFFNC	Timer	12	20	NC
Sol	Solenoid	17	20	Coil
Coil	Relay	15	50	Coil
PL	Pilot Light	16	20	Coil
Horn	Horn	43	20	Coil
Buzzer	Buzzer	44	20	Coil
Bell	Bell	45	20	Coil
PLC	PLC Module	48	10	Coil